PAIZI

SailWind Layout Guide and Reference

Release SailWind 3.0
Document Revision 1.3

派兹

Contact Information

If you have any questions or suggestions about this installation or software, please contact:

Email: market@pzeda.com
Phone: 0755-86703052
Website: www.pzeda.com

Revision History

Revision	Changes	Date
1.0	Initial release, corresponding to SailWind V3.0	2024-03-25
1.1	• Updated descriptions on how to use Make Like Reuse in sections "Using Make Like Reuse in Object Mode" and "Using Make Like Reuse in Verb Mode". • Updated descriptions on automatic cluster placement in Section "Cluster Placement Dialog Box". • Added sections "Selecting Objects by Selected List" and "Dispersing Components by Schematic Sheet". • Deleted Chapter "Your License, and Software Features". • Updated descriptions on suite configuration in "Help > Installed Options" menu.	2024-09-27
1.2	• Added descriptions accordingly with new features Assign Color to Net and commands for toggling the display of all/selective unrouted connections, covering Sections "Assigning Colors to Nets", "Changing the Visibility of All Unrouted Connections", and "Changing the Visibility of Selective Unrouted Connections". • Added descriptions on RF Toolbar where function buttons Add Via Shield, Convert to Chamfered Paths, and Select Chamfered Paths can be accessed.	2024-12-10
1.3	• Added Chapter "AI-Powered Advanced Functionality".	2025-03-21

Table of Contents

Chapter 1
SailWind Layout QuickStart

Welcome to SailWind Layout, a powerful yet easy to use printed circuit board layout application.

PCB Layout Quickstart

SailWind Layout provides the tools you need to lay out your PCB design, from creating a decal library for your components to generating output for fabrication.

To get started quickly, click on any of the topics.

- Create a library for your decals (footprints)
    - Creating a New Library
- Add decals to your parts:
    - Create decals for your parts on page 160
    - Assign a decal to a part type (Netlist project) on page 235
- Import the schematic design:
    - Import a netlist on page 249
- "Set up the PCB layer structure" on page 315
- Create vias for routing
    - Creation of vias on page 330
- Create a board outline on page 261
- Set up design rules or constraints
    - Creation of Rules for Your Design
- Set grids on page 352
- Place parts on page 473
- Route traces on page 699

- Check for rules violations on page 883
- Annotate the design on page 916
- Generate reports on page 856
- Output the design on page 934

Chapter 2
User Interface

The SailWind Layout interface contains standard menus and buttons for accessing commands and settings.

User Interface Elements

The SailWind Layout interface elements are described below.

Note:
Pictures in this document are for reference only, to help users better understand the software operation. In the case of interface difference due to version changes, the interface of SailWind Layout in practice shall prevail.

Figure 1. Main Interface

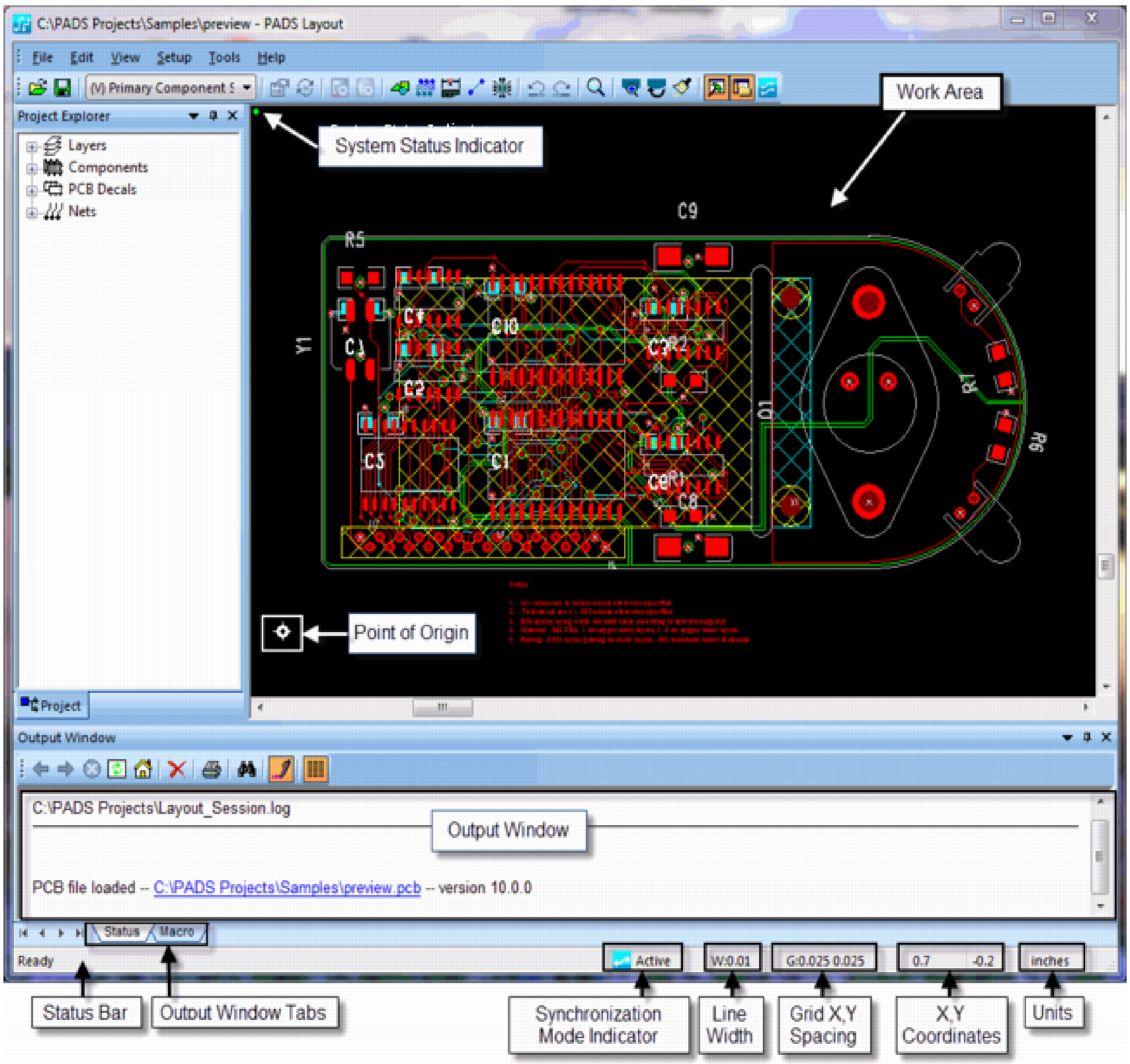


Table 1. GUI Elements

Elements	Description
Title Bar	The application icon, document name, and application name appear on the title bar. Click the application icon to open the Windows-standard control menu, which contains commands for working with the application window. The document name changes to reflect the state of the current document. For example, when no design file is loaded "Untitled - SailWind Layout" appears as the document name. When a design file is loaded, the path, the file name, and the file extension (*.pcb*) appear on the title bar.
Menu bar	The menu bar lists SailWind Layout commands. The menus also show the appropriate command icons, access keys, and

Table 1. GUI Elements (continued)

Elements	Description
	shortcuts. When a command ends with ellipses (...), additional information is needed to complete the command.
Standard Toolbar	The Standard toolbar contains commands that open and save designs, change the view, redraw, and access the Drafting, Design, Dimensioning, ECO, and BGA toolbars.
Work Area	The area in which you enter all design information is called the work area or the workspace. The work area contains two editors: the Layout editor and the PCB Decal editor. The Layout editor appears when you open SailWind Layout. You can place, route, and otherwise modify your board in this editor. The PCB Decal Editor is an editor that you start (through the Tools menu) where you can create or edit a decal. The maximum work area is a 56 by 56 inch square. You can measure it in inches, mils, or metric units. Set the units of measure in Global Options. The work area is divisible by an X,Y, or horizontal and vertical grid, which you can set to a minimum of .00001" between points. You can set the X and Y values separately. This is called the Design Grid on the **Grids** tab of the Options dialog box.
Status Bar	The status bar displays a command name or information on a selected connection, route, or component. The layer on which a selected trace exists appears on the status bar.
System Status Indicator	The system status indicator shows the processing state of the application in the upper left corner of the work area. It appears green when the system is idle or ready for operation, and red when the system is unavailable either because the system is processing or the work space cannot receive user input, such as when in CAM.
Point of Origin	The 0,0 coordinate location. X,Y coordinates are calculated from this point. When you create a new file, the default drawing format is centered at medium magnification in the work area with the origin, or the 0,0 point, in the lower left corner. The origin appears as a large white dot. As you move the cursor, its position relative to the origin appears in the X,Y Coordinates area. The numbers change in multiples of the design grid.
Layout-Router Synchronization Mode Indicator	Displays whether SailWind Layout and SailWind Router are in Synchronization mode. Also indicates which program is the active one versus the inactive one or whether the inactive program is out of sync.
Trace/Line Width	Displays the current line width setting.
Design Grid X,Y Spacing	Displays the setting of the Design grid. When you move an object or use a drafting command, the grid readout is replaced by a Delta X and Y reading, calculated from the cursor selection point when the command starts. Negative numbers indicate left and down.

Table 1. GUI Elements (continued)

Elements	Description
X,Y Coordinates	Displays the horizontal distance of the cursor from 0,0 as the x-coordinate. Also indicates the polar radius if you are using a polar grid. Displays the vertical distance of the cursor from 0,0 as the y-coordinate. Also indicates the polar angle if you are using a polar grid.
Units	Displays the current units used in the design (mils, millimeters, inches).

Panning Overview

The are many ways to pan around your design. Use the middle mouse button or the middle mouse wheel as the most efficient ways to pan.

- **Center the view based on the pointer location** — Point in the workspace to where you want to locate the new center of the view and middle-click or press the Insert key. The screen repaints the design with the point you chose at the center of the screen.
- **Dynamically pan in the workspace** — Hold down the Alt key and the middle mouse button and drag the pointer in the direction you want to move. When the design is in the spot you want, release the middle mouse button.
- **Move the view based on a miniature representation of the design area** — In the Status window on page 1733 click on the design representation to center the view and pan to that area.
- **Scroll vertically and horizontally** — Use the pointer and the scroll bars, or use the mouse wheel for vertical scrolling and press the Shift key while you use the wheel for horizontal scrolling.
- **Move the view one-half screen horizontally or vertically** — Turn on the NumLock and use the arrow keys on the number pad.
- **Move the view one grid unit horizontally or vertically** — Use the arrow keys with NumLock Off.

Zooming Overview

The most efficient way to zoom is using the middle mouse button. There are also shortcuts to zoom to the board outline, the extents of all design items, or a selected item.

There are many ways to zoom:

- **Zoom to the board outline** — Press Ctrl+B, or on the Standard Toolbar click the **Board** button, or on the View menu click **Board**, or press the Home key. This zooms to and centers the board outline in the workspace.

- **Zoom to the extents of all design items** — Press Ctrl+Alt+E, or on the Standard Toolbar click the **Extents** button, or on the View menu click **Extents**. This zooms to fit all objects, including those outside the board outline, into the workspace.

- **Zoom to the selection** — Select a design object and press Alt+Z, or on click the **View > Selection** menu item. Any object you select becomes centered in the view of the workspace.

  Alternatively, in the Project Explorer window, right-click and click the **Zoom to Selection** popup menu item. Then click a component, PCB decal, or net in the Project Explorer. In the Project Explorer, the right-click > **Allow Selection** popup menu item must be selected in order for the Zoom to Selection feature to work. If either of these options already have a check mark, and you click them again, you turn them off.

- **Zoom in to a defined area with the middle mouse button** — Move the pointer to the center of the desired zoom area. Hold down the middle mouse button and drag the pointer diagonally up across the design, indicating both the horizontal and vertical limits of the bounding box that defines your next view. When the area want to magnify is within the bounding box, release the mouse button. The zoom-in ratio also appears with the cursor.

  Alternatively, 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 pointer with the middle mouse button.

- **Zoom out of a defined area with the middle mouse button** — Move the pointer to the center of the desired zoom area. Hold down the middle mouse button and drag the pointer diagonally down across the design, indicating both the horizontal and vertical limits of the bounding box that defines your next view. The box in the center represents the current view size. The outer box that expands from the center box represents the new view size in proportion to the old. The zoom-out ratio also appears with the cursor.

  Alternatively, 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.

- **Zoom in and out with a two button mouse** — On the Standard Toolbar, click the **Zoom** button. This button is a toggle. The left mouse button now functions like a middle mouse button (see instructions earlier in this topic) allowing you to define an area for zooming. You can also place the pointer at the desired view center and click the left mouse button to zoom in, or click the right mouse button to zoom out. Click the **Zoom** button or press the Esc key to exit zoom mode to return the mouse to its normal functionality.

- **Zoom using the mouse wheel** — Press and hold the Ctrl key as you rotate the wheel button.

- **Zoom using the keyboard** — To zoom with a defined area, press the 5 key on the numeric keypad with NumLock on. This puts the pointer in zoom mode like holding the middle mouse button. To zoom in at the pointer, press the Page Up key on the numeric keypad. To zoom out at the pointer, press the Page Down key on the numeric keypad.

Design Origin and Workspace

The design space measures 56 inches X 56 inches. New designs begin with the design origin located at the -18”, -18” of the overall workspace. With the origin at this point in the workspace, it is helpful to keep the origin at the lower left of your board outline. Otherwise, you can change the location of the design origin within the workspace.

When you change the location of the origin, you receive a message that references the new location based on the old origin location and also provides coordinates in parenthesis based on the centerpoint of the workspace. By displaying your origin location based on the design space centerpoint, you can determine by the size requirements of your board whether you will reach the limits of the design space based on the placement of your design origin. From the workspace centerpoint, you have from -28” to +28” on both the x and y axis.

Setting the Origin by Click
Setting a Precise Origin Location

Setting the Origin by Click

Use the pointer to set the origin location for the design.

Procedure

1. **Setup > Set Origin**.
2. Click where you want the new origin to be. You are prompted to accept the new origin and are given the coordinates of the new origin location based on the grid of the original origin.

Tip
The message also displays coordinates in parentheses, of the new origin location based on the centerpoint of the workspace.

Related Topics

Design Origin and Workspace

Setting a 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. Click the **Setup > Set Origin** menu item. Alternatively, you can use the SO Modeless commands on page 83. The SO modeless command also allows you to set the origin to absolute coordinates of the design space.

Related Topics

Design Origin and Workspace

Redraw of the Display

Occasionally, fragments or an outline of an object might appear where the object no longer exists. If you see remnants of objects that you have moved, or you see lines where there are actually no design objects, you can force a complete full screen redraw from the video card. On the Standard Toolbar, click the **Redraw** button.

Cycling Through Design Views

Each time you change the view of your design workspace, SailWind Layout records the view. You cycle back and forth through views that you have created by panning and zooming the workspace.

Procedure

1. Change the view of the workspace.
2. To return to the previous view, click the **View > Previous View** menu item.
3. To restore the original view, click the **View > Next View** menu items.

Related Topics

Saving and Restoring a View

Saving and Restoring a View

If you are frequently viewing an area of your design, you can save a view of the design area and quickly restore it instead of panning and zooming to the view every time. You can save up to nine views. The ability to save views is not available in the PCB Decal Editor.

Procedure

1. Pan to and set the zoom level of the view you want to save.
2. Click **View > Save View**. The Save View Dialog Box appears.
3. Click **Capture**.
4. In the Capture a New View box, type a name and click **OK**.

Results

The view names appear at the bottom of the View menu for quick selection. You can also click **View > Save View** to revisit the views that you captured, and then manage the list of views. You can delete ones that are no longer useful. The Save View dialog box displays the current view as a blue square in the miniature representation of the design area. Any selected view names are displayed as red squares.

Software Launch Options

You can use software launch options, known as *command line switches*, to control the initial SailWind Layout configuration. These switches can open a file, start macros, and record a SailWind Layout session. You can use multiple command line options.

You add command line switches to the Start menu shortcut. See “Adding Start-up Options to a Shortcut”.

Table 2. SailWind Layout Command Line Options

Option	Description
full path to a file	Opens the specified design file when you start SailWind Layout. Type the full path. Do not use a forward slash (/) before the file name in the command line. Use double quotation marks for folders or file names with spaces. For example: `“SailWind_Folder\SailWind version\Samples\preview.pcb”` **Note:** SailWind_Folder is the root directory of SailWind; version indicates the software version.
/BMW[=initials]	([ ] represents optional text.) Opens the Basic Media Wizard. Note the capitalization. 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. For example, type /BMW or /BMW=xx, where xx is your initials, in the command line. To create session media files for the current SailWind Layout session, use the BMW modeless command on page 83. This option is associated with the BLT modeless command. BLT is the Log Test; it finds and runs the session media created by BMW to play back a recorded SailWind Layout session. For more information, see “Typing Modeless Commands”.
/dotest	Runs the integrity test on every file that you open.
/l	Opens the last file you had open when you start SailWind Layout.
-log:<full path>	Records your software session as a macro file in the specified location. Make sure to type a space between *..\SailWindPCB.exe* and the -log command. Use double quotation marks for folders or file names with spaces. For example: `-log:“C:\SailWind Projects\mymacro.log”`
/M<macro file>	Specifies the file to use as the default macro file. Note the required capitalization of M. Macros must be located in the *\SailWind Projects* folder. /m<macro name> — Runs the specified Sub procedure or Function procedure macro within the default macro file specified with the /M command. For example, to run the macro SubMacro contained in the file *user1.mcr*, type /Muser1.mcr /mSubMacro. If you need to launch a macro at software launch that is not located in the *\SailWind Projects* folder, see the /run command line option.
/nc	Starts SailWind Layout without displaying the splash screen that includes copyright information.

Table 2. SailWind Layout Command Line Options (continued)

Option	Description
/NTL	Disables true layer associations. Note the required capitalization of NTL. When you use Flip Side, the layer attributes will not move to the new layer with the component. TrueLayer is enabled by default and moves layer definitions with a part that is placed on the opposite side of the board. Parts are built specifying the generic “Mounted Side” and “Opposite Side” layers for electrical objects and more specific layers for documentation objects. For example, a surface mount decal’s pads are placed on the generic Mounted Side layer, while the silkscreen information is placed on the more specific Silkscreen Top layer. By default, this TrueLayer feature is enabled and when you move a surface mount component from the top layer to the bottom layer in your design, any documentation information you placed on the Silkscreen Top, Solder Mask Top, Paste Mask Top, and Assembly Drawing Top layers are automatically flipped and associated with the Silkscreen Bottom, Solder Mask Bottom, Paste Mask Bottom, and Assembly Drawing Bottom, without needing to redesign the part to be a bottom mounted part or needing to design both a top mount and bottom mount component. For more information, see “TrueLayer and Layer Association”.
/run=[full path]	Runs the specified macro when you launch the software. Make sure to type a space between "SailWindPCB.exe" and the "-run" command. When there are spaces in paths or file names you must use double quotes around the full path. For example: `/run=“C:\SailWind Projects\Samples\mymacro.mcr”`
/s<full path>	Starts a Basic script when you start SailWind Layout. Use double quotation marks for file names with spaces. For example: `/s“C:\SailWind Projects\Attributes to Excel.bas”`

Adding Start-up Options to a Shortcut

You can add various start-up options to the properties of a shortcut, such as starting up SailWind Layout with a specific project already opened.

Tip
If you create your own shortcuts, copy the Start menu shortcuts instead of generating ones from the executables in the install directory. Start menu shortcuts contain a “wrapper” that allows the proper environment variables to be defined as the program launches.

Procedure

1. In the shortcut properties, click in the box with the pathname.
2. Press the End key, press the Spacebar key, and then type the command line switch you want to use. Enclose with double quotes " " each string that contains a space. When specifying a file to

start, do not use a / before the file name. You can specify multiple command line switches. For example, to start the program with *preview.pcb*, the command line might read:

“\<install_folder>\<version>\Programs\SailWindPCB.exe” “C:\SailWind projects\Samples \preview.pcb”

Related Topics

Software Launch Options

Disabling Check for Updates

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

Procedure

1. Click the **Help > Check for updates** menu item.
2. In the Check for Updates Dialog Box, select the Disable “Check for Updates” functionality check box.

Typing Modeless Commands

You can set or change some settings and apply commands at any time using an abbreviated command in a pop-up window command line. Many of the modeless commands duplicate menu commands and Options settings. There are also many modeless commands that activate unique commands that cannot be accessed anywhere else.

Procedure

1. Type the modeless command on page 83 for the command you want. Use a space between the command and the argument. For example: N GND or GD 50.
2. Press the Enter key.

Related Topics

Modeless Command Dialog Box

Keyboard Shortcuts

Project Explorer

The Project Explorer shows a hierarchical structure for design layers, components, PCB decals, and nets. When you update your design, the hierarchical structure of the Project Explorer automatically reflects the changes. Open a design to access its hierarchical structure of components, PCB decals and nets. The Project Explorer is not available in the PCB Decal Editor.

You can use the Project Explorer to interact with objects in the design:

Selecting Design Objects Using the Project Explorer
Zooming to a Design Object Using the Project Explorer

Selecting Design Objects Using the Project Explorer

You can set up SailWind Layout so that selecting a design object in the Project Explorer automatically selects it in the workspace as well.

Procedure

1. If the Project Explorer window is not open, on the Standard Toolbar, click the **Project Explorer Window** button.
2. In the Project Explorer window, right-click.
3. Make sure the **Allow Selection** popup menu item is checked. If needed, click **Allow Selection** to check it.
4. In the Project Explorer hierarchy, select a design object (component, PCB decal, net). It also selects in the workspace.

Zooming to a Design Object Using the Project Explorer

You can set up SailWind Layout so that it zooms in on a design object in the workspace when you select the corresponding design object in the Project Explorer.

Procedure

1. If the Project Explorer window is not open, on the Standard Toolbar, click the **Project Explorer Window** button.
2. In the Project Explorer window, right-click.
3. Make sure the **Zoom to Selection** popup menu item is checked. If needed, click **Zoom to Selection** to check it.
4. In the Project Explorer hierarchy, select a design object (component, PCB decal, net). The entire workspace view displays the selected object.

Output Window

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

The Output Window is located in the bottom section of the display window. You can dock or float, and display or hide the Output Window.

See the following for more information on the tabs of the Output Window:

- The Status Tab
- The Macros Tab

The Status Tab

The Status tab displays information on the current session. You can use the Status tab to work with HTML documents. Although it is not a Web browser, you can use it to open Internet Web pages that any links reference.

If the Status tab is closed, and you get an error while performing tasks, the output window automatically opens with the Status tab active.

Use the Status tab to:

- Record, display, and print the session log.
- Print and display report files.
- Print and display web pages.
- Open program files.

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

The default path for the session log comes from the *.ini* file entry *FileDir=C:\SailWind Projects*.

The status log presents different types of information in different colors. Text color representations are shown in the following table:

Table 3. Status Log Text Color Representations

Color	Meaning
Red	Errors
Green	Warnings
Black	Messages
Blue	Links to files, Web pages, and database objects. These items are also underlined.

Page Navigation in the Status Tab
Filtering the Messages in the Status Tab
Searching Text in the Status Tab
Printing Status Log Messages
Reports Inside the Status Tab

Page Navigation in the Status Tab

Instead of reports opening in your default text editor, you can set up SailWind Layout so they can open and display them inside the **Status** tab like new pages in a web browser. Using the toolbar buttons, 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 status log display.

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

Table 4. Status Tab Navigation Toolbar Buttons

Button	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 status log.

Filtering the Messages in the Status Tab

You can choose which types of messages to display in the status log of the Output Window.

Procedure

1. Right-click in the log area of the **Status** tab, point to the **Filter** popup menu item, and on the popup submenu, click to place a check beside any one or combination of the following items:

Table 5. Filter Popup Submenu Items

Menu Item	Description
Message	Displays messages (black text)
Warning	Displays warning messages (green text)
Error	Displays error messages (red text)
Show all	Displays all messages types (error, warning, and message).

> **Tip**
> Check marks indicate which message types are turned on for viewing in the log.

2. Click a checked menu item to clear the check and filter the log removing the display of those messages. Those message types are hidden until they are displayed again. They are not deleted from the log.

Searching Text in the Status Tab

Similar to searching in a document, you can search text in the **Status** tab. For example, after a long software session, you might want to look for a report from a previous state of the design. You can search for the report name instead of visually searching by scrolling through the log.

Procedure

1. In the Output Window, in the **Status** tab, click the **Search** button.
2. In the Find dialog box, type the text you want to find and set any other options if required. The search begins as soon as you start typing a word and scrolls through the log to the first occurrence of the work, highlighting the word.
3. If the search does not immediately find the correct occurrence of your search, click the **Next** or **Previous** buttons to cycle through the occurrences.
4. When finished, close the Find dialog box.

Printing Status Log Messages

You can print a hard copy of the session's status log for review purposes.

Procedure

1. In the Output Window, in the **Status** tab, click the **Print** button.
2. In the Windows Print dialog box, set options as needed.
3. Click the **Print** button.

Reports Inside the Status Tab

The session status log contains links to reports that you can view and print. Links appear in blue, underlined lettering. While most reports open in your default text editor, you can design some reports to open within the **Status** tab.

If you click a link and a report opens in the **Status** tab instead of your default text editor, you must use the **Home** button to return to the status log. For more information, see "Page Navigation in the Status Tab".

To print a report that appears in the **Status** tab, follow the same instructions for printing the status log. For instructions, see "Printing Status Log Messages".

Changing the Default Text Editor

Reports open in the default text editor. You can change the default text editor used with SailWind Layout by changing the "Editor" entry in the *SailWindpcb.ini* file.

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

Procedure

1. Open the *SailWindpcb.ini* file in a text editor. By default it is located in the following folder —*C:\<install_folder>\<version>\Settings*.
2. In the [general] section, specify a new text editor executable name after "Editor=". 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.

Locked Files

The SailWind products help you avoid making changes to a file that another user has already opened. 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 locks out all other users. If you try to open a file that someone else has already opened, you 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 the **File > Save As** menu item.

SailWind Layout Default Settings

The default system and design settings (Options, Colors, Rules, Layer stackup, Vias, CAM, and Attributes) are preset by the *default.asc* and *decaledt.asc* files. The file *default.asc* contains design mode settings, and the file *decaledt.asc* contains decal editor mode settings.

You can create your own start-up file to customize the software launch defaults. For more information, see "Creating Start-up Files".

Software Interface Customization

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

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.

Toolbar Customization

Click the **Tools > Customize > Toolbars and Menus** tab to create custom toolbars and shortcut menus.

For related information, see "Moving or Copying Menu Commands".

Tip
To create a custom main menu, use the **Commands** tab on the Customize dialog box.

See the following subtopics for more information:

Creating a Custom Toolbar

Create a new empty toolbar and add items (commands) to it.

Procedure

1. On the Tools menu, click Customize. The Customize dialog box opens.
2. Create a new Toolbar:
   a. In the Toolbars and Menus tab on page 1220, in the Toolbars box, click the **New** button .
   b. Type the name for the toolbar and click **OK**.
      - The new (empty) toolbar appears on the software 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).
   c. Drag the toolbar to where you want it on the software interface.
3. Add items (commands) to your new toolbar:
   a. Click the **Commands** tab.
   b. In the Categories list, select a menu or toolbar name to display commands specific to that menu or toolbar, or select All Commands. If you are working in a special mode in SailWind Layout (for example, the Decal Editor), some categories of commands are not available for customization.
   c. In the Commands list, select the command you want and drag it to the toolbar.
4. When you have finished adding commands to the new toolbar, click **Close**.

Related Topics

Deleting a Custom Toolbar

Toolbars and Menu Item Removal

Renaming a Custom Toolbar

Showing or Hiding a Toolbar

Increase space in the software interface by displaying the toolbars you need to use and hiding others that you do not use. You can also remove toolbars that contain tools that you do not use in favor of displaying custom toolbars containing those you use.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. In the Toolbars and Menus tab on page 1220, 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. Clear the check box to hide the toolbar.
4. Click the **Close** button.

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

Deleting a Custom Toolbar

You can remove or delete any toolbar that you customize. You cannot, however, delete a system toolbar.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. In the Toolbars and Menus tab on page 1220, in the Toolbars list, select the toolbar.
3. Click the **Delete** button.

Related Topics

Adding Items to Toolbars and Menus

Toolbars and Menu Item Removal

Showing or Hiding a Toolbar

Renaming a Custom Toolbar

You can change the name of any toolbar that you create (customize). You cannot, however, rename a system toolbar.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. In the Toolbars and Menus tab on page 1220, in the Toolbars list, select a custom toolbar.
3. Click the **Edit** button .
4. In the Toolbar Name dialog box, type the new name and click the **OK** button.

Related Topics

Showing or Hiding a Toolbar

Deleting a Custom Toolbar

Resetting Toolbars to Defaults

You can reset one or all system toolbars so that they display their default buttons.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. Click the Toolbars and Menus tab on page 1220.
3. Reset one of the following:
    - A single toolbar:
        i. In the toolbars list, select the toolbar.
        ii. Click the **Reset** button.
    - All system toolbars: In the Toolbars area, click the **Reset All** button.

Command and Menu Customization

You can customize commands and then add the commands to menus or as buttons on toolbars. You can also customize Menus.

See the following subtopics for more information:

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, select an existing command on which to base your new command then define the new command properties.

You can also create a custom command from a macro command file. For instructions, see "Adding Macros to Toolbars and Menus" in the *SailWind Layout Command Reference*.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. In the Commands tab on page 1213, in the Categories list, click a menu or toolbar name to display items (commands) specific to that menu or toolbar, or click the "All Commands" list item.

   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 "Adding Macros to Toolbars and Menus" in the *SailWind Layout Command Reference*.
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 on page 1050 dialog box, specify the properties of your new command:
   a. In the Command name box, type the name of the command.
   b. In the Description box, type a description of the custom command.
   c. If an image was associated with the original command, click Use Default Image to use that same image with your custom command. Click Select User-Defined Image to use a different image, edit an image, or create a new image.
   d. Click the **OK** button to close the "Add command" dialog box and return to the Customize dialog box.
5. If you are finished with all customizations, click the **Close** button.

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

Adding Macros to Toolbars and Menus [SailWind Layout Command Reference Manual]

Resetting Toolbars to Defaults

Editing a Custom Command

SailWind Layout provides you with the ability to edit the commands you create (custom commands). You cannot edit system commands, however.

Procedure

1. On the **Tools** menu, click **Customize**. The Customize dialog box opens.
2. In the Commands tab on page 1213, 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 your custom command and click the **Edit** button .
4. In the Edit Command dialog box on page 1050, change the properties of your custom command:
    a. In the Command name box, type the name of the command.
    b. In the Description box, type a description of the custom command.
    c. If an image was associated with the original command, select Use Default Image to use that same image with your custom command. Click “Select User-defined Image” to use a different image, edit an image, or create a new image.
    d. 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**.

Deleting a Custom Command

You can remove or delete any command you create (custom commands). However, you cannot delete system commands.

Procedure

1. On the **Tools** menu, click **Customize**. The Customize dialog box opens.
2. In the Commands tab on page 1213, 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 your custom command and click the **Delete** button.
4. Click **Close**.

Creating a Custom Menu

You can create a new empty menu and then add items (commands) to it to create a custom menu.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. In the Commands tab on page 1213, in the Categories list, click 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. In the interface, right-click over your new menu and click the **Button Appearance** popup menu item.
5. In the Button Appearance Dialog Box, in the Button text field, type the name for the menu and click the **OK** button. 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.

   If you are working in a special mode in SailWind Layout or SailWind Logic (for example, the Decal Editor in SailWind 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 the **Close** button.

Toolbar and Menu Content

You can modify the items on toolbars and menus.

See the following subtopics for more information:

Adding Items to Toolbars and Menus

You can customize toolbars by adding new buttons and you can customize menus by adding new commands.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. In the Commands tab on page 1213, in the Categories list, select a toolbar or menu name to display commands specific to that menu or toolbar, or select All Commands.

   If you are working in a special mode in SailWind Layout or SailWind Logic (for example, the Decal Editor in SailWind 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.

   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 the **Close** button.
5. For instructions to customize the text or image of the toolbar or menu item, see “Customizing the Appearance of New Toolbar and Menu Items”.

Related Topics

Creating a Custom Command

Adding Macros to Toolbars and Menus [SailWind Layout Command Reference Manual]

Toolbars and Menu Item Removal

Moving or Copying Menu Commands

Customizing the Appearance of New Toolbar and Menu Items

You can customize the text and image of new toolbar and menu items.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. In the interface, right-click over your new menu item or toolbar button and click the **Button Appearance** popup menu item.
3. In the Button Appearance Dialog Box, make changes to the appearance of the item.
4. When you have finished customizations in the Button Appearance dialog box, click **OK**.
5. In the Customize dialog box, click **Close**.

Moving or Copying Buttons Using the Customize Dialog Box

Use the Customize dialog box to rearrange toolbar buttons to meet your preferences.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box must remain open to allow these changes.
2. Move or copy the toolbar button:
    - To move the button: Click the toolbar button and drag it to a new place on the same toolbar or to a different toolbar.
    - To copy the button: Click the toolbar button. Press and hold the Ctrl key while dragging the button to place the copy on the same toolbar or to a different toolbar.
3. Close the Customize dialog box.

Moving or Copying Buttons Without the Customize Dialog Box

If desired, you can move a toolbar button by pressing the ALT key and dragging it. You can also copy a toolbar button by pressing the ALT + CTRL keys while dragging the item you want to copy. Using these key combinations allows you to arrange custom toolbars without using the Customize dialog box.

Procedure

1. Press and hold the Alt key.
2. Move or copy the toolbar button.
    - To move the button: Drag the toolbar button to a new place on the same toolbar or to a different toolbar.
    - To copy the button: Drag the toolbar button and then press and hold the Ctrl key instead of the Alt key to create a copy of the button to place on the same toolbar or to a different toolbar.

Moving or Copying Menu Commands

You can customize the arrangement of command on a menu using the Customize dialog box.

Procedure

1. Click the **Tools > Customize** menu item.
2. In the main window, click the menu to expand it and display the contents. The Customize dialog box must remain open to allow these changes.
3. Move or copy the menu command:
    - To move the command: Drag the menu item to its new location on the same menu or to a different menu.
    - To copy the command: Press and hold the Ctrl key while dragging the item to create a copy to place on the same menu or on a different menu.
4. Close the Customize dialog box.

Related Topics

Command and Menu Customization

Adding Items to Toolbars and Menus

Resetting Toolbars to Defaults

Toolbars and Menu Item Removal

You can customize a toolbar or menu by removing items and buttons. The method to use depends on whether the Customize dialog box is open.

Use one of the following methods:

- 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 the Alt key. 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

Shortcut Key Customization

You can create and customize shortcut keys by using the **Keyboard and Mouse** tab of the Customize dialog box.

To assign a shortcut key to a macro, see "Assigning Shortcut Keys to Macros" in the *SailWind Layout Command Reference*.

See the following subtopics for more information:

Creating a Shortcut Key

You can create shortcuts that apply in a specific mode. The same shortcut key may have different functionality depending on the mode in which you are working.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. In the Keyboard and Mouse tab on page 1214, 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.

   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" on page 1106.
5. Create one of the following shortcut types:

   - **Shortcut key** — Click "Press new shortcut key(s)" and press the keyboard keys that you want to use. For detailed information about rules and restrictions for creating shortcut keys, see "Rules and Restrictions for Key Sequences". As you enter the new shortcut, similar shortcuts appear in the "Similar shortcuts assigned to other commands" box. This helps to avoid the creation of a new shortcut that conflicts with an existing shortcut.
   - **Mouse action** — Click "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. The new shortcut appears in the Current shortcuts box on the Customize dialog box.

Rules and Restrictions for Key Sequences

SailWind Layout has certain restrictions when using key sequences to make a shortcut.

The first character may consist of the following, including 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. Mouse pointer events cannot be combined with key sequences, although the Ctrl, Alt, and Shift modifiers are allowed.
- Subsequent characters may consist of alphanumeric characters (a-z0-9).

Tip
Some combinations, like Alt+Tab, are intercepted by Windows and are not available.

Related Topics

Creating a Shortcut Key

Command and Menu Customization

Resetting to Default Shortcut Keys

Listing Available Shortcut Keys

Listing Available Shortcut Keys

You can generate a report in an HTML format displaying a table of commands and the shortcuts assigned to them.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. In the Keyboard and Mouse tab on page 1214, click **Report**.
3. In the Save report dialog box, specify a file name and location, then click **Save**. A hyperlink to the file appears in the Output window, on the **Status** tab.

Related Topics

Command and Menu Customization

Creating a Shortcut Key

Resetting to Default Shortcut Keys

Deleting a Shortcut Key

Delete or remove shortcuts you no longer want to use. You can also delete them as the first step to changing an existing shortcut.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. In the Keyboard and Mouse tab on page 1214 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.
6. When finished, close the Customize dialog box.

Related Topics

Creating a Shortcut Key

Resetting to Default Shortcut Keys

Resetting to Default Shortcut Keys

You can restore all shortcut keys to their default settings.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. In the Keyboard and Mouse tab on page 1214, click **Reset All**.
3. On the confirmation dialog box, click **Yes**.
4. When finished, close the Customize dialog box.

Screen Appearance

You can customize the software interface by toggling the appearance of tooltips, changing the button icon size, enabling personalized menus (shortened menus based on usage), changing the menu display, changing the Microsoft visual style of the windows and dialog boxes, and changing the interface language.

For more information, see the Options tab of the Customize dialog box on page 1218.

Related Topics

Setting the Interface Language

Resizing the Layers List

Setting the Interface Language

You can change the language used in the text of the interface. SailWind Layout supports the following languages: English and Chinese Simplified.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. In the Options tab on page 1218, in the Visual Style area, in the Interface language list, click the desired language.
3. In the prompt, “You must restart SailWind Layout for the change of language to take effect”, click **OK**.
4. Close the Customize dialog box.
5. Restart SailWind Layout.

Resizing the Layers List

You can change the width of the Layers list on the Standard toolbar if your layer names are longer than what is displayed by default. Although you can press the Alt key to move buttons on toolbars, you cannot use the Alt key to resize the Layers list.

Procedure

1. Click the **Tools > Customize** menu item. The Customize dialog box opens.
2. On the Standard Toolbar of the main Interface, select the Layers list box.
3. Use the resizing bar that appears on the cursor as you hover over the right side of the Layers list box. Resize as needed.
4. Close the Customize dialog box.

Window Placement

You can customize the way windows (Output window, Project Explorer window, Markups window, Archive Navigator) appear in your workspace.

See the following subtopics for specific information:

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.

Procedure

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 title bar. 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. The window reappears, covering the application.
4. Move the pointer away from the window. 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 does not attach to the current view; instead, it hovers, blocking the view to anything behind it.

Procedure

1. If the window is currently set to hide automatically, first turn off the Auto Hide feature.
2. Double-click the window title bar. The window detaches and you can move it to any part of the screen. To undo 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 does not block the view to anything behind it.

You can dock a window in its last docked location, or dock a window to a different location.

Docking to the Last Location

You can dock a window to its previous docked location.

Procedure

1. Select the window to dock.
2. Double-click the window title bar. The window reattaches to the interface.

Docking to a New Location

Dock a window to a new location using the cursor.

Procedure

1. Using the title bar, drag the window. 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 2.

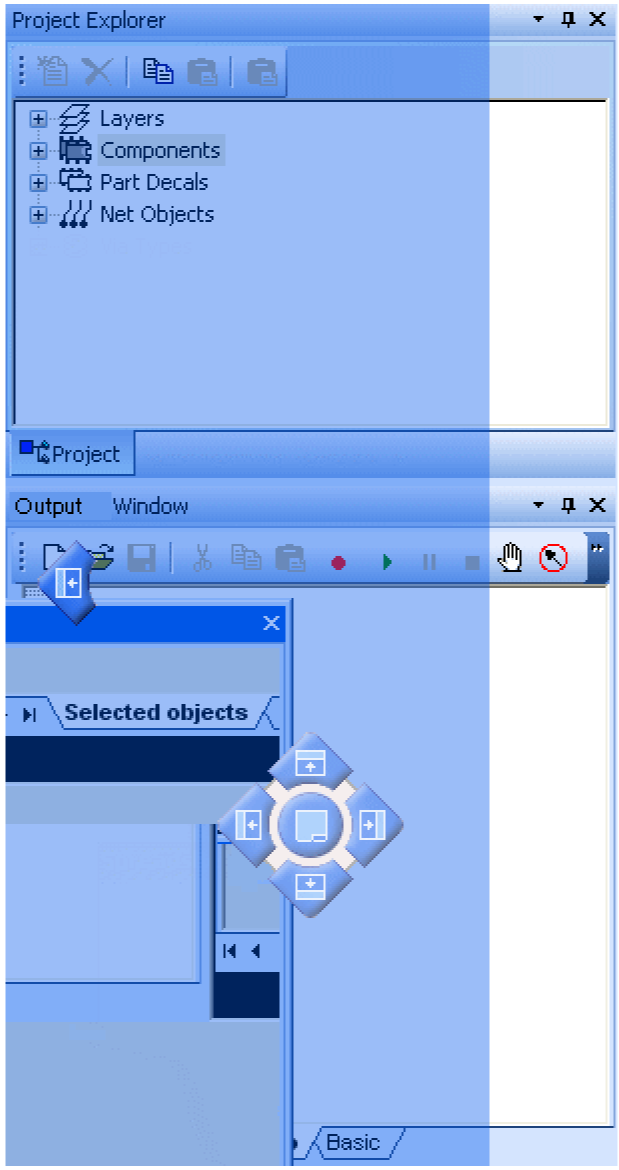
   Figure 2. Window Dragging Graphic

   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. 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.

Figure 3. Docking a Window



3. Release the mouse button while hovering over the arrow that indicates where you want to dock the window. The window docks to the user interface, and the other windows in the user interface resize.

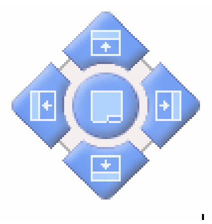
Embedding Two Windows Within One Window Space

If desired, you can embed a window within another window space to improve the efficiency of the work area.

Procedure

1. Using the title bar, drag a window into another window. When you start dragging a window, additional graphics appear in the user interface. A group of arrows containing graphics appears in the center of docked windows into which you are dragging, as shown below. Depending on the window in which you are dragging, the group of arrows may also have a tab graphic in the center.

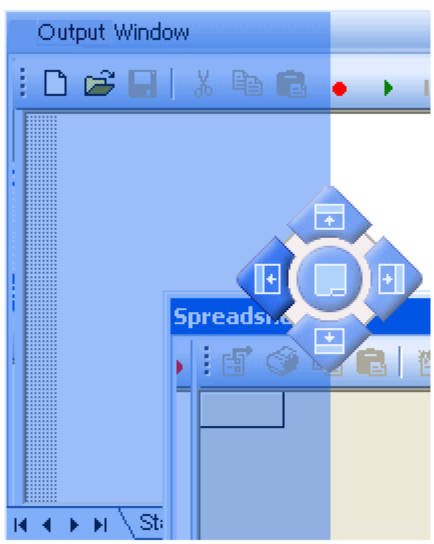
   Figure 4. Dragging a Window—Arrow Group

   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. A transparent colored block appears along the side of the window you are dragging, as shown below. 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 Output Window.

   Figure 5. Dragging and Docking a Window

3. Release the mouse button while hovering over the arrow that indicates where you want to dock the window. 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 title bars, and all of the windows within the original window frame hide automatically.

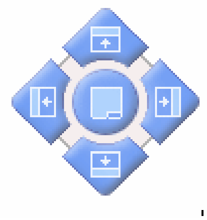
Creating Tabs Within Windows

Create a tab of your window within the space occupied by another window.

Procedure

1. Using the title bar, drag a window into another window. 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 into which you are dragging, as shown below. Depending on the window into which you are dragging, the group of arrows may also have a tab graphic in the center.

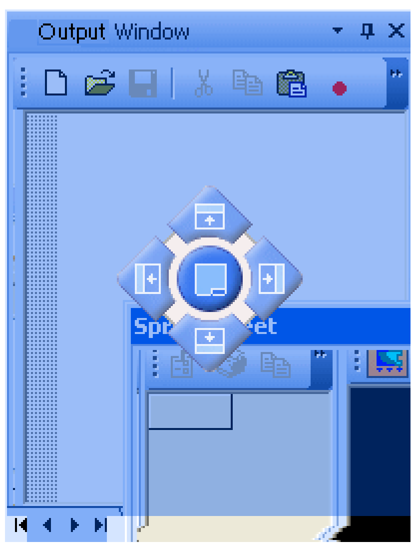
   Figure 6. Dragging and Docking a Window—Arrow Commands

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

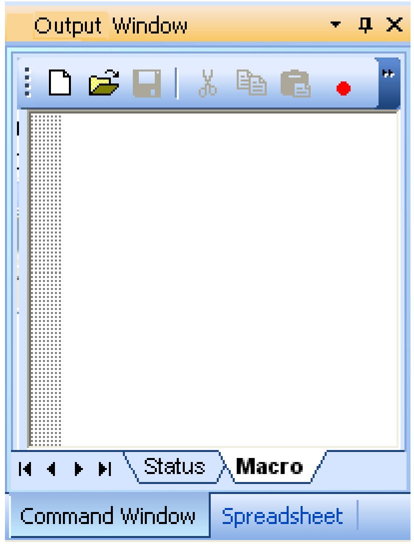
2. While dragging the window, hover over the tab graphic. A transparent colored block appears over the window you are dragging, as shown below. This block indicates where the window will be docked when you release the mouse button. For example, if you hover over the tab in the Project Explorer window, a block appears over the Project Explorer.

   Figure 7. Dragging a Window—Transparent Block

3. Release the mouse button while hovering over the tab. The window is embedded as a tab within a window, as shown below. You can click each tab to access each window.

Figure 8. Window Embedded as a Tab



4. To rearrange the tab, drag the tab to a new position within the row of tabs.

5. To move an embedded window, drag the tab out of the window in which it is embedded and the tab will convert back into a window that you can place where desired.

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

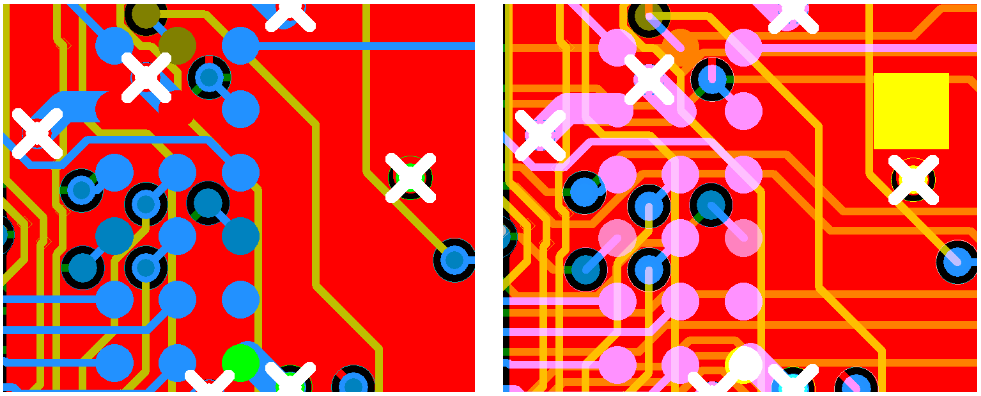
Transparent View Mode

Use Transparent Mode to view traces on several layers simultaneously. Transparent mode can show you hidden obstacles directly under the current active layer.

To toggle transparent view mode, use the Modeless Command on page 83 T.

With Transparent mode you can view traces on several layers at once. Transparent mode can show obstacles that may be hidden directly under the current active layer. When Transparent mode is on, you see trace-over-trace, trace-over-part outline, copper-over-trace overlaps drawn in a third, lighter color. The overlap color is determined by the colors you assign to your traces. With Transparent mode off, overlaps do not appear.

Figure 9. A Routed Board with Transparent Mode Off and On

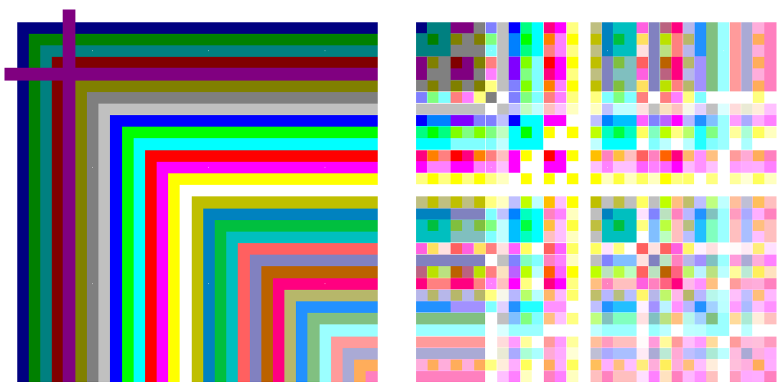


You should use transparent mode for inspecting dense route areas or when routing in areas with layer-specific obstructions.

Color Selection and Transparent Mode

The transparent effect is more visible if you use darker colors for traces. Bright colors like yellow and aqua wash out the effect.

Figure 10. A Thirty Two Color Grid with Transparent Mode Off and On



Tip
Items that you select may not clear normally and may remain highlighted until you click **Redraw** on the toolbar.

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 toggle outline view mode, use the Modeless Command on page 83 O.

Resolution Options

With resolution options, modeless commands ‘OL’ and ‘OH’ on page 83, you can change the display resolution for faster redraw of very large designs. High resolution, which is the default, displays objects in their true shape. Low resolution displays pads as square or rectangular objects. The last resolution setting you used is saved, for example, if you set low resolution, selecting outline mode toggles between normal view and low resolution mode.

Related Topics

Viewing Protected Routes with Outline Mode

Bottom View

You can change the view of the design space to view your PCB design from behind. Enable this bottom view to help visualize the placement of components or the placement of reference designators on the bottom side of the board.

Use one of the following to toggle Bottom View:

- **View > Bottom View** menu item
- Standard Toolbar > **Bottom View** button
- Use the B modeless command on page 83
- Press Alt-B

The entire design space is mirrored. Moving the cursor to the left results in a positive move on the x axis while in Bottom View.

If you have enabled the Active layer comes to front setting in the Global Options on page 1531, the active layer remains at the front when you switch to Bottom View. If the Active layer comes to front setting is disabled when you switch to Bottom View, you will see the bottom layer at the front regardless of which layer is active.

Fonts

Text strings and labels in your designs can use stroke font and/or system fonts that are installed on your system.

- You can set fonts for each text string and label you create in your design, choosing stroke font or system fonts for each selection. You can also have a combination of stroke font and system fonts within the same design.
- You can search for fonts used for text strings or labels, and can then open the item Properties to apply a different font name and style to all objects that you select for modification.
- Designs to be output to printers and plotters can also use both stroke fonts and system fonts. When making font selections, consider the following:

- System font text is supported in RS-274X Gerber format when Fill mode is on and is output as a set of filled polygons. For more information see the settings in the Photo Plotter Advanced Setup Dialog Box.

- System fonts are not supported in the RS-274D CAM output format. If you attempt to use this output format with system fonts, the program displays a warning message. If you proceed, system fonts will not be output. Instead, you should use the RS-274X format with system fonts.

- 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. For instructions, see "Replacing Missing Fonts".

- For systems using languages that do not include stroke font, English stroke font is used.

- Non-ASCII symbols, such as +/-, ohm, and degrees are available on your system for the installed fonts you select. If the character sets you choose are not available, a blank space or blank text box appears where the symbols should be. In this case, choose character sets that are available on your system to enable the symbols to display in your design.

When you open an existing design that was created on a system without system font supported, you must choose whether to use stroke or system fonts for every text string or label in the design. To convert the existing text or label fonts to another font, use the Text on page 1750 or Part Label Properties on page 1603 dialog box. To find fonts easily, use the Find Dialog Box.

Chapter 3
Modeless Commands and Shortcuts

Learning and using modeless commands and command shortcuts available in the function keys, keypad keys and keyboard key combination can make your use of the software more efficient.

Tip
Some commands are only accessible as a modeless command.

Modeless Commands

You can set or change some settings and functions at any time by typing a code letter for the command, entering the new value, and pressing the Enter key. This is called a *Modeless Command*.

Modeless Commands usually apply to values that you change frequently during design. Use the Modeless Command "G," for example, to change the grid setting. Type "G", the new setting, and press the Enter key.

To show this help topic at any time while SailWind Layout is running, type "?" and press the Enter key.

Modeless Command Formats

Tip
(X,Y) = coordinates; (s) = text; (n) = number

Table 6. Modeless Command and Shortcut Key Table Conventions

Convention	Description
<value>	A variable, or something you can type (modeless commands only).
{ }	An optional command argument.
<Click>	Click the left mouse button.
MButton+<Click>	Click the middle mouse button or wheel.
RButton+<Click>	Click the right mouse button.
<RotateBackward>	Rotate the wheel backward, where the top of the wheel rotates toward your palm.

Table 6. Modeless Command and Shortcut Key Table Conventions (continued)

Convention	Description
<RotateForward>	Rotate the wheel forward, where the top of the wheel rotates away from your palm.

Tip
Spaces have significance in modeless commands and shortcut keys. For example, SS W1 and S SW1 have different meanings. SS W1 means to search for and select W1, while S SW1 means to search for SW1.

Modeless Command List

The following is a complete list of all of the modeless commands:

Table 7. Modeless Commands for Setting the Design Units

Name	Command	Description
Set Design Units to Inches	UI	Sets the Design Units to inches in the **Global** category > **General** subcategory of the Options dialog.
Set Design Units to Mils	UM	Sets the Design Units to mils (thousands of an inch) in the **Global** category > **General** subcategory of the Options dialog.
Set Design Units to Millimeters (metric)	UMM	Sets the Design Units to metric (millimeters) in the **Global** category > **General** subcategory of the Options dialog.

Table 8. Modeless Commands for (Cartesian) Grid Settings

Name	Command	Description
Global Grid Setting	G<x> {<y>}	Sets the Design and Via grids simultaneously. The second parameter is optional. If you type one parameter, it applies the same value to both <x> and <y>. For example, G 25, G 8.3 or G 16-2/3, G 5 25.
Displayed (Dot) Grid Setting	GD <x> {<y>}	Sets the Displayed (Dot) grid. The second parameter is optional. If you type one parameter, it applies the same value to both <x> and <y>. For example, GD 8-1/3, GD 25 25, or GD 100.
Design Grid Settings	GR <x> {<y>}	Sets the Design grid (placement and routing). The second parameter is optional. If you type one parameter, it applies the same value to both <x> and <y>. For example, GR 8-1/3, GR 25 25, GR 25.

Table 8. Modeless Commands for (Cartesian) Grid Settings (continued)

Name	Command	Description
Via Grid Settings	GV <x> {<y>}	Sets the Via grid. The second parameter is optional. If you type one parameter, it uses the same value for both <x> and <y>. For example, GV 8-1/3, GV 25 25, or GV 100.

Table 9. Modeless Commands for (Polar) Grid Settings

Name	Command	Description
Polar Grid On/Off	GP	Polar grid on/off. Use the polar grid for Radial Move, circular component arrays, and to create radial drawings. **Note:** Routing on the polar grid requires you to set the Angle Mode to Any Angle. See the AA modeless command for more info.
Move to a Point Specified by Polar Coordinates (radius angle)	GP r a	Move to a point specified by polar coordinates (radius and angle).
Move to a Point Specified by the Angle, Using the Existing Radius	GPA a	Move to a point specified by the angle, using the existing radius.
Move to a Point Specified by the Radius, Using the Existing Angle	GPR r	Move to a point specified by the radius, using the existing angle.
Move to a Point Specified by the Current Angle (da) Value, Using the Existing Radius	GPRA da	Move to a point specified by the current angle (da) value, using the existing radius.
Move to a Point Specified by the Current Radius (dr), Using the Existing Angle	GPRR dr	Move to a point specified by the current radius (dr), using the existing angle
• - 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.		

Table 10. Modeless Commands for Setting Origins

Name	Command	Description
Set the Origin Using Design Coordinates	SO	Sets the origin using current design coordinates.

Table 10. Modeless Commands for Setting Origins (continued)

Name	Command	Description
		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 8.3 8.3 or SO 25 25. See also: “Design Origin and Workspace”.
Set the Origin Using Absolute Coordinates of the Design Space	SOA	Sets the origin using the absolute coordinates of the design space. Coordinates must be given. The design space measures 56 inches X 56 inches and with the design space origin at the center of this space, the coordinates measure from -28” to +28” on both the x and y axis. By displaying the design space coordinates, you can determine whether you will reach the limits of the design space based on the placement of your design origin and the size requirements of your board. For example, SOA 8.3 8.3 or SOA 25 25.

Table 11. Modeless Commands for Line/Trace Angle Settings

Name	Command	Description
Any Angle	AA	Any angle mode. Sets the Line/trace angle setting in the Design options to Any angle (no angle restrictions).
Diagonal Angle	AD	Diagonal angle mode. Sets the Line/trace angle setting in the Design options to Diagonal (45 degree angles only).
Orthogonal Angle	AO	Orthogonal angle mode. Sets the Line/trace angle setting in the Design options to Orthogonal (90 degree angles only - no dialog or 45-degree corners).

Table 12. Modeless Commands for Line/Trace Width Settings

Name	Command	Description
Reduce Display Width	R <n>	Sets the “Minimum display width” in the General options on page 1531 to <n>, for example, R 50. Any line widths that are less than this value are drawn as single pixel center lines. This feature is no longer needed to speed refresh times with the power of today’s video cards, but is still used to assist in the selection of very small line segments.
Change Current Width to <n>	W <n>	Changes the current trace or line width to the number <n> you enter, for example W 5.

Table 13. Modeless Commands for Line Style Settings

Name	Command	Description
Line Style (status)	LS	Shows the current line style setting in the status bar.
Line Style Solid	LSS	Sets the default line style for new 2D line objects to Solid lines.
Line Style Dashed	LSD	Sets the default line style for new 2D line objects to Dashed lines.
Line Style Dotted	LSO	Sets the default line style for new 2D line objects to Dotted lines.
Line Style Dash Dotted	LSA	Sets the default line style for new 2D line objects to Dash Dotted lines.
Line Style Dash Double-Dotted	LSB	Sets the default line style for new 2D line objects to Dash Double-dotted lines.

Table 14. Modeless Commands for DRC Settings

Name	Command	Description
Turn Off DRC Mode	DRO	Turn off on-line DRC mode. Sets the on-line DRC setting in the Design options to Off.
Prevent	DRP	Prevent errors mode. Sets the on-line DRC setting in the Design options to Prevent errors.
Warn	DRW	Warn errors mode. Sets the on-line DRC setting in the Design options to Warn errors.
Ignore Clearance	DRI	Ignore clearance mode Sets the on-line DRC setting in the Design options to Ignore clearance.

Table 15. Modeless Commands for Searching

Name	Command	Description
Search Absolute	S <x> <y>	Search absolute. Moves the pointer to the specified X and Y coordinates, for example S 1000 1000.
Search for Reference Designator/Pin	S <s>	Search for reference designator/pin <s>, for example S U1.1 or S U1.
Search Relative	SR <x> <y>	Search relative. Moves the pointer by the specified X and Y offset, for example SR -100 -50.

Table 15. Modeless Commands for Searching (continued)

Name	Command	Description
Search Relative X	SRX <x>	Search relative X at current Y. Moves pointer by the specified X offset, for example SRX 300.
Search Relative Y	SRY <y>	Search relative Y at current X. Moves pointer by the specified Y offset, for example SRY 400.
Search and Select	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 SailWind Layout to search and select W1, while S SW1 tells it to search for SW1.
Search and Select Using Asterisk *	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.
Absolute Move to <n>, Current Y	SX <x>	Search absolute X at current Y. Moves the pointer to the specified X coordinate and the current Y coordinate, for example SX 300.
Absolute Move to <n>, Current X	SY <y>	Search absolute Y at current X. Moves the pointer to the specified Y coordinate and the current X coordinate, for example SY 400.
Search for and Select Route Segments Using Pixels	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.

Table 16. Modeless Commands for Drafting Shape Control

Name	Command	Description
Circle shape draw mode.	HC	Sets drawing mode to circle shape.

Table 16. Modeless Commands for Drafting Shape Control (continued)

Name	Command	Description
Path shape draw mode.	HH	Sets drawing mode to path shape.
Polygon shape draw mode.	HP	Sets drawing mode to polygon shape.
Rectangle shape draw mode.	HR	Sets drawing mode to rectangle shape.

Table 17. Modeless Commands for Objects Snapping

Name	Command	Description
Object Snap Mode	OS	Toggles Snap to Objects mode on/off in the Object Snap options on page 1540.
Object Snap Radius	OSR <n>	Sets the snapping radius to the specified value (in current design units) in the Object Snap options on page 1540. If no value is given, OSR displays current value of snap radius in the status bar.
Object Snap Type	OS <n>	Sets the Snap to object types in the Object Snap options on page 1540 and clears those not selected. You can use one or more of the following values: • **C** — center of a circle or an arc. • **E** — corner of a drafting shape or line end. • **I** — intersections; the cross point of two lines or arcs. • **M** — midpoint of a line or arc. • **O** — component origin. • **P** — a pin or via origin. • **Q** — quadrants; four orthogonal points on a circle or arc. For example: If you want to only snap to midpoints, type OS M. Or if you want to snap to all objects, type OS CEIMOPQ.

Table 18. Modeless Commands for Object Visibility

Name	Command	Description
Bottom View	B	Mirrors the entire design space from below. Moving the cursor to the left results in a positive move on the x axis while in Bottom View.
Complementary Format	C	Type C and press the Enter key to change the display to a complementary format, which views thermals and antipads on plane layers. Type

Table 18. Modeless Commands for Object Visibility (continued)

Name	Command	Description
		C a second time and press the Enter key to restore the normal non complementary view.
Active Layer to Front On/Off	D	Toggles the active layer setting in the **Global** category > **General** subcategory of the Options dialog.
Drill Outline On/Off	DO	Toggles the display of the drill outline. The display of the drill is disabled when you give a net a specific highlight color in View > Nets.
Pin Number Display On/Off	PN	Toggles the display of the pin numbers. This command also toggles the Pin Num. column check box in the Display Colors Setup Dialog Box. Sizing is controlled by the settings in the Display Options on page 1525.
Transparent Mode On/Off	T	Toggles the display of objects as either transparent or opaque. See also: “Transparent View Mode”.
Text Outline On/Off	X	Toggles the display of text outlines.

Table 19. Modeless Commands for Net Highlighting

Name	Command	Description
Highlight Nets One by One	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. **Tip** Highlighted nets take on the Highlight color specified in the Display Colors dialog box on page 1318.
Unhighlight Each Highlighted Net in Reverse Order	N -	Unhighlights each highlighted net in reverse order in which they were added using the N command.
Remove All Highlighting from the Nets	N	Removes all highlighting from the nets.

Table 20. Modeless Commands for Net Name Visibility

Name	Command	Description
Toggle Visibility of Net Names	NN	Toggles the visibility of net names.

Table 20. Modeless Commands for Net Name Visibility (continued)

Name	Command	Description
		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 the net name placement is controlled by the settings in the Display Options on page 1525.
Toggle Display of Net Names on Pins	NNP	Toggles the display of net names on pins. This command toggles the Show net names on...Pins check box in the Display Colors Setup Dialog Box. Sizing is controlled by the settings in the Display Options on page 1525. The display of net names also depends on the Net Name check box and the presence of a non-background color tile on the layer in the Display Colors Setup Dialog Box.
Toggle Display of Net Names on Traces	NNT	Toggles the display of net names on traces. 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 on page 1525. The display of net names also depends on the Net Name check box and the presence of a non-background color tile on the layer in the Display Colors Setup Dialog Box.
Toggle Display of Net Names on Vias	NNV	Toggles the display of net names on vias. 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 on page 1525. The display of net names also depends on the Net Name check box and the presence of a non-background color tile on the layer in the Display Colors Setup Dialog Box.

Table 21. Modeless Commands for Layer Visibility (Quick Layer View)

Name	Command	Description
Quick Layer View	Z	Quick layer view. With no command arguments, Z displays the initial layer view.
Add or Remove Layer from Current Set of Displayed Layers	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.
View Only the Range of Layers You Type	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.
View Only the Layers You Type	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.
View All layers	Z *	View all layers. **Restriction:** Z supports only the asterisk * regular expression.
Hide All layers	Z -*	Hide all layers. **Restriction:** Z supports only the asterisk * regular expression.
View the Active Layer	Z A	View the active layer. If the active layer is changed, it has no effect on the display.
View the Assembly Drawing Bottom layer.	Z ADB	View the Assembly Drawing Bottom layer.
View the Assembly Drawing Top layer.	Z ADT	View the Assembly Drawing Top layer.
View Only the Bottom Layer	Z B	View only the bottom layer.
View Only the Current 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.
View All Documentation Layers	Z D	View all documentation layers.

Table 21. Modeless Commands for Layer Visibility (Quick Layer View) (continued)

Name	Command	Description
View All Electrical Layers	Z E	View all electrical layers.
View All Internal Layers	Z I	View all internal layers.
View Only the Outside Layers (Top and Bottom Layers)	Z O	View only the outside layers, that is, the top and bottom layers.
View the Paste Mask Bottom.	Z PMB	View the Paste Mask Bottom.
View Paste Mask Top	Z PMT	View the Paste Mask Top.
View Solder Mask Bottom	Z SMB	View the Solder Mask Bottom.
View Solder Mask Top	Z SMT	View the Solder Mask Top.
View Silkscreen Bottom	Z SSB	View the Silkscreen Bottom.
View Silkscreen Top	Z SST	View the Silkscreen Top.
View Only the Top Layer	Z T	View only the Top layer.
View Unrouted Connections (that are visible on all layers)	Z U	Toggles the view of unrouted connections that are visible on all layers.
View All Layers	Z Z	View all layers.
Restore a Quick Layer View Configuration	ZR <name>	Restore a quick layer view configuration. For example, ZR L23 restores the configuration stored as L23. **Tip** Type ZR and press the Enter key to see a list of all layer configurations created by ZS in this session. See also: ZS.
Save Current Set of Displayed Layers as a Quick Layer View Configuration	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.

Table 22. Modeless Commands for Layers

Name	Command	Description
Change the Current Layer to <n>	L <n>	Changes the current layer to <n>. <n> can be any number or name, for example (L 2) or (L top).

Table 22. Modeless Commands for Layers (continued)

Name	Command	Description
Layer Direction on the Current Layer	LD	Toggles between horizontal and vertical routing layer direction on the current layer. Changes the Routing Direction setting for the active layer in the design.
Paired Layer Command	PL <n1> <n2>	Paired Layer command, where <n> can be layer numbers or layer names. (for example, PL 1 2 or PL top bottom). Sets the Layer pair setting in the Routing options on page 1542.

Table 23. Modeless Commands for Managing the 3D View Point in SailWind 3D

Name	Command	Description
View from Top	T	View from top.
View from Bottom	B	View from bottom.
View from Left	L	View from left.
View from Right	R	View from right.

Table 24. Modeless Commands for Vias

Name	Command	Description
Toggle Between End Via Modes	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 **Tip** See the Status bar to know what mode you have switched to.
Open Via Dialog	V	Opens the Vias Dialog Box.
Automatic Via Selection	VA	Sets the Via Mode to Automatic in the Vias Dialog Box. See the dialog box topic for more information.
Use Partial Via	VP	Sets the Via Mode to Partial in the Vias Dialog Box. See the dialog box topic for more information.
Use Through Hole Via	VT <name>	Sets the Via Mode to Through in the Vias Dialog Box. If multiple vias exist, you can also

Table 24. Modeless Commands for Vias (continued)

Name	Command	Description
		type the name of the via to use. See the dialog box topic for more information.

Table 25. Modeless Command for Pour and Hatch Outlines

Name	Command	Description
Pour Outline On/Off	PO	Toggles the pour outline on/off and toggles the Display mode of copper planes between Pour outline and Hatch outline in the "Copper Planes / Hatch and Flood" on page 1511 options.

Table 26. Modeless Commands for Outline Mode Settings

Name	Command	Description
Outline Mode On/Off	O	Toggles outline mode on and off. The OH and OL modeless command determine the resolution mode of outline mode. 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 on page 1531. See also: "Outline View Mode".
Outline Mode in High Resolution	OH	Toggles outline mode with High resolution. Displays all objects with their true shape.
Outline Mode in Low Resolution	OL	Toggles outline mode with Low resolution. Displays rounded objects with squared corners. Oval and rectangular pads look the same. Trace segments do not have arcs at the ends. As an exception, round pads are drawn as circles.

Table 27. Modeless Commands for General Mode Settings

Name	Command	Description
Toggle Make Like Reuse Operations	RV	Toggles Make Like Reuse operations to compare or ignore value and tolerance attributes. See also: "Design Elements Made Like a Reuse Definition".
Toggles Shove Mode On/Off	SH	Toggles Shove mode on or off.

Table 28. Modeless Commands for Measurement

Name	Command	Description
Quick Measure	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 on page 1537is on. Measurements are gridless when Grid Snap is off. Dynamically reports delta x, delta y, and delta x,y in current design units on page 1531. You can also measure precise Euclidean distances between polar grid nodes using this command. Quick Measure mode also works with Object Snapping. But unless you are within the snap radius of an object type, you are not given the option to choose between snapping to the grid or an object for your starting point. Snapping to object types takes priority over snapping to the grid. You can enable Snapping to Objects on the popup menu during the measure operation, in the Options dialog box on page 1540, or by using the OS modeless commands. See also: “Measurement”.
Quick Length	QL	Quick Length. 1. Area-select the route items you want to measure; such as trace segments, nets, or pin pairs. 2. Type QL and press the Enter key. A report of the route item lengths for routed, unrouted, and total length appears in the default text editor. See also: “Measurement”.

Table 29. Modeless Commands for Multiple Undo and Redo Operations

Name	Command	Description
Multiple Undo Command (1-100)	UN [<n>]	Multiple undo command (1-100), <n> is optional. (Ex. UN 2) UN will undo 1 time.
Multiple Redo Command	RE [<n>]	Multiple redo command (1-100), <n> is optional. (Ex. RE 2) RE will redo 1 time.

Table 30. Miscellaneous Modeless Commands

Name	Command	Description
Enable Color by Net	Y	Toggles the application of custom color to net pads, pins, vias, and—optionally—traces, according to the settings in the View Nets Dialog Box.
Open File <name>	F <name>	Open file <name>, where <name> is the path and name of the file to open (for example, F demo.eco).
Show This Help Topic	?	Show this help topic.
Database Integrity Test	I	Runs Database Integrity Test.

Table 31. Modeless Commands for the Basic Media Wizard/Log Test

	Command	Description
Basic Log Test	BLT	Basic Log Test. Opens the Log Test Dialog Box. BLT finds and runs BMW session playback media. See also: BMW and BLT
Open Basic Media Wizard	BMW	Opens the Basic Media Wizard dialog box. • BMW records session playback media for a problematic SailWind Logic, SailWind Layout or SailWind Router session. It can create playback media based on your last SailWind 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
Start BMW Session Logging	BMW ON	Starts BMW session logging.
Stop BMW Session Logging	BMW OFF	Stops BMW session logging.

Table 32. Modeless Command for the Unrouted

Name	Command	Description
Enable Visibility of the Unrouted in the Same Net as the Selected Object	VUS	Show the unrouted in the same net as the selecetd object.
Disable Visibility of the Unrouted in the Same Net as the Selected Object	HUS	Hide the unrouted in the same net as the selecetd object.

Table 32. Modeless Command for the Unrouted (continued)

Name	Command	Description
Show All the Unrouted	VUA	Show all the unrouted in the design, except those in the locked net(s).
Hide All the Unrouted	HUA	Hide all the unrouted in the design, except those in the locked net(s).

Function Keys

Use function keys in SailWind Layout to perform commonly used tasks.

The following table defines the function key command assignments.

Table 33. Function Key Command Assignments

Function Key	Description
F1	Open Help (context sensitive)
F2	Add Route
F3	Dynamics Route
F4	Toggle Layer Pair
F5	Select Pin Pair
F6	Select Net
F7	Autoroute Selected
F8	Unassigned
F9	Unassigned
F10	Unassigned
F11	Unassigned
F12	Unassigned

Keypad Keys

Use the keypad to perform common layout tasks.

The following table defines the keypad key command assignments.

Table 34. Keypad Key Command Assignments

Keypad Keys	Description
(Number Keys) with NumLock On	
Keypad (0)	Center the view using the pointer location.
Keypad (1)	Redraw.
Keypad (2)	Pans the workspace down one-half the screen height.
Keypad (3)	Zooms out at the pointer.
Keypad (4)	Pans the workspace left one-half the screen width.
Keypad (5)	Starts Zoom from center (Num Lock only).
Keypad (6)	Pans the workspace right one-half the screen width.
Keypad (7)	Zoom to the Board.
Keypad (8)	Pans the workspace up one-half the screen height.
Keypad (9)	Zoom in at the pointer location.
Keypad (.)	Starts Zoom from corner (Zoom Mode only).
(Command Keys) with NumLock Off	
<Insert>	Centers the view using the pointer location.
<End>	Redraw.
<Down Arrow>	Moves the pointer down one design grid.
<Page Down>	Zooms out at the pointer.
<Left Arrow>	Moves the pointer left one design grid.
<Right Arrow>	Moves the pointer right one design grid.
<Home>	Zooms to the Board.
<Up Arrow>	Moves the pointer up one design grid.
<Page Up>	Zooms in at the pointer.
<Delete>	Delete the selected object.

Keyboard Shortcuts

SailWind Layout provides certain keyboard combinations (shortcuts) to access or perform some common layout tasks.

The following table provides a complete list of the keyboard shortcut commands.

Table 35. Keyboard Shortcuts for Panning, Zooming and Navigation

Name	Shortcut Keys	Description
Zoom to Board	<Home>	Zooms to the board. Fits the board outline into the workspace.
	Ctrl+B	
Zoom Extents	Ctrl+Alt+E	Zooms to extents. Fits all objects in the design into the workspace.
Zoom Area In/Out (Zoom by Drag)	MButton+ <Start Drag>	Zooms area in or out. Drag pointer up to zoom in. Drag pointer down to zoom out.
Start Zoom from Corner	Shift+MButton+ <Start Drag>	Starts Zoom from Corner.
Zoom to Selection	Alt+Z	Zooms to selection. Fits the selected objects into the workspace.
Zoom Mode On/Off	Ctrl+W	Toggles Zoom Mode On/Off.
Center View (Using Pointer Location)	MButton+<Click>	Centers the view at the pointer.
	<Insert>	
Zoom In at Pointer (Zoom Mode)	LButton+<Click>	Zooms in at the pointer (zoom mo de).
	<spacebar>	
Zoom Out at Pointer (Zoom Mode)	RButton+<Click>	Zooms out at the pointer (zoom mode).
Zoom In at Pointer	<Page Up>	Zooms in at the pointer.
	Ctrl+<Rotate Forward>	
Zoom Out at Pointer	<Page Down>	Zooms out at the pointer.
	Ctrl+<Rotate Backward>	
Move Pointer Down (One Design Grid)	<Down Arrow>	Pointer moves down one design grid.
Move Pointer Up (One Design Grid)	<Up Arrow>	Pointer moves up one design grid.

Table 35. Keyboard Shortcuts for Panning, Zooming and Navigation (continued)

Name	Shortcut Keys	Description
Move Pointer Left (One Design Grid)	<Left Arrow>	Pointer moves left one design grid.
Move Pointer Right (One Design Grid)	<Right Arrow>	Pointer moves right one design grid.
Dynamic Panning (Pan by Drag)	Alt+MButton+<Start Drag>	Pans the sheet area below the pointer to the center of the workspace.
Pan Workspace Down (One Line)	<Rotate Backward>	Pans workspace down one line.
Pan Workspace Up (One Line)	<Rotate Forward>	Pans workspace up one line.
Pan Workspace Right (One Line)	Shift+<Rotate Backward>	Pans workspace right one line.
Pan Workspace Left (One Line)	Shift+<Rotate Forward>	Pans workspace left one line.
Pan Workspace Down (One Pixel)	Ctrl+Alt+<Rotate Backward>	Pans workspace down one pixel.
Pan Workspace Up (One Pixel)	Ctrl+Alt+<Rotate Forward>	Pans workspace up one pixel.
Pan Workspace Right (One Pixel)	Alt+Shift+ <Rotate Backward>	Pans workspace right one pixel.
Pan Workspace Left (One Pixel)	Alt+Shift+<Rotate Forward>	Pans workspace left one pixel.

Table 36. Keyboard Shortcuts for Selection

Name	Shortcut Keys	Description
Selection Filter	Ctrl+Alt+F	Displays the Selection Filter.
Select	LButton+<Click>	Selects an object.
Select	<Spacebar>	Selects an object.
Select All Board Objects	Ctrl+A	Selects all objects in the design based upon the selection filter choices.
Area Select	LButton+<Start Drag>	Starts an area selection.
Area Complete	LButton+<End Drag>	Completes an area selection.

Table 36. Keyboard Shortcuts for Selection (continued)

Name	Shortcut Keys	Description
Cancel Area Selection	LButton+<Cancel Drag>	Cancels an area selection.
Cycle Selection	<Tab>	Cycles the selection of adjacent objects within the pick radius.
Toggle Selection	Ctrl+LButton+<Click>	Toggles object selection.
Select Target (BGA Function)	Ctrl+Shift+Z	Select target.

Table 37. Keyboard Shortcuts for File Operations

Name	Shortcut Keys	Description
Create New Design File (Blank)	Ctrl+N	Creates a new blank design file.
Open File (Design)	Ctrl+O	Opens a design file.
Save	Ctrl+S	Saves the design file

Table 38. Keyboard Shortcuts for Opening Menus and Dialog Boxes

Name	Shortcut Keys	Description
Open SailWind 3D window	3	Opens the SailWind 3D window if the cursor is active in the design area.
Open File Menu	Alt+F	Opens the File menu.
Open Edit Menu	Alt+E	Opens the Edit menu.
Open View Menu	Alt+V	Opens the View menu.
Open Setup Menu	Alt+S	Opens the Setup menu.
Open Tools Menu	Alt+T	Opens the Tools menu.
Open Help Menu	Alt+H	Opens the Help menu.
Open Shortcut Menu	RButton+<Click>	Opens the Shortcut menu.
Properties (Current Object)	LButton+<DoubleClick>	Displays Properties for the currently selected object.
Properties (for Selected)	Alt+<Enter>	Opens Properties dialog box for selected object.

Table 38. Keyboard Shortcuts for Opening Menus and Dialog Boxes (continued)

Name	Shortcut Keys	Description
	Ctrl+Q	
Options Global	Ctrl+<Enter>	Opens the Options dialog box.
Display Colors Setup	Ctrl+Alt+C	Opens the Display Colors Setup dialog box.
Options > **Design** Category	Ctrl+Alt+D	Opens the **Design** category of the Options dialog box.
Options > **Global** category > **General** subcategory	Ctrl+Alt+G	Opens the **Global** category > **General** subcategory of the Options dialog box.
Open Status Window	Ctrl+Alt+S	Opens the Status Window
Open View Nets Dialog	Ctrl+Alt+N	Opens the View Nets dialog box.

Table 39. Keyboard Shortcuts for Placement Operations

Name	Shortcut Keys	Description
Move Selected Object(s)	Ctrl+E	Moves the selected object(s).
Rotate Selected (90) (during move) (CCW)	<Tab>	Rotates a component (during component move mode) (CCW).
Rotate Selected (90) (during move) (CW)	Shift+<Tab>	Rotates the selected component (during component move mode) (CW).
Rotate Selected (90) (CCW)	Ctrl+R	Rotates the selected components(90 degrees) (CCW).
Spins Selected	Ctrl+I	Spins the selected object.
Flip Side (Selected)	Ctrl+F	Flips an object to the opposite side of the board.
Align Component	Ctrl+L	Align Component.

Table 40. Keyboard Shortcuts for Routing Operations

Name	Shortcut Keys	Description
Add Corner	LButton+<Click> <Spacebar>	Adds a corner (interactive routing).
Add Tack	LButton+<Click>	Add a tack.
Add Via	Shift+LButton+<Click>	Adds a via (interactive routing).
Add Jumper	Ctrl+Alt+J	Add a jumper (interactive routing).
Backup	<Backspace>	Removes the last routed corner on a trace or the last corner on a 2D line (in polygon or path drawing mode).
Stretch	Shift+S	Stretch (interactive routing mode).
Complete Route	<Enter>	Complete route.
Complete	LButton+<Click> LButton+<DoubleClick>	Complete (interactive routing) when pointer is near a valid completion point.
End (routing)	Ctrl+LButton+ <Click>	End (interactive routing).
Length Minimization	Ctrl+M	Runs length minimization.
Quick Measure/Reset DxDy	Crtl+<PageDown>	Quick Measure • Works the same as the Q modeless command. • Resets the Delta X,Y values to zero and begins measuring from the current location. This command works in any mode, with Delta X,Y on the status bar instead of the grid display.
Show Route Length (Design View)	Ctrl+<PageUp>	Shows the route length (Design View).
Create Union	Ctrl+G	Create a Union.
Route Loop	Ctrl+J	Route a Loop.
Create Cluster	Ctrl+K	Create a Cluster.

Table 40. Keyboard Shortcuts for Routing Operations (continued)

Name	Shortcut Keys	Description
Teardrop Properties	Ctrl+T	Displays the Teardrop Properties

Table 41. Keyboard Shortcuts for Bus Routing Operations

Name	Shortcut Keys	Description
Bus Route	Ctrl+Alt+B	Bus Route
Add Corner to Guide (Bus Routing Only)	Alt+LButton+<Click>	Add a corner to a guide (Bus Routing only).
Add Via to Guide (Bus Routing Only)	Alt+Shift+LButton+ < Click>	Add a via to a guide (Bus Routing only).
Cycle Via Pattern (Bus Route Only)	Ctrl+<Tab>	Cycle through the via patterns (Bus Routing only).
End Guide (Bus Routing Only)	Ctrl+Alt+LButton+<Click>	End Guide (Bus Routing only).
Next Guide (Bus Route Only)	<Tab>	Next Guide (Bus Route only)
Previous Guide (Bus Route Only)	Shift+<Tab>	Previous Guide (Bus Route only).

Table 42. Keyboard Shortcuts for Editing

Name	Shortcut Keys	Description
Cut	Ctrl+X	Cut
Copy	Ctrl+C	Copy.
Paste	Ctrl+V	Paste.
Redraw	<End>	Redraw.
	Ctrl+D	
Delete/Unroute	<Delete>	Unroute selected object.
Cancel Command	<Escape>	Cancels command.
Redo	Ctrl+Y	Redo.
Undo	Ctrl+Z	Undo.

Table 42. Keyboard Shortcuts for Editing (continued)

Name	Shortcut Keys	Description
Highlight Object	Ctrl+H	Highlights Object.
Unhighlight	Ctrl+U	Removes highlighting from current object.

Table 43. Keyboard Shortcuts for Viewing

Name	Shortcut Keys	Description
Bottom	Alt+B	Mirrors the view of the design in the workspace.
Next View	Alt+N	Displays the next view.
Previous View	Alt+P	Displays the previous view.

Table 44. Mouse Button Substitutions

Name	Shortcut Keys	Description
Activate Right Click Pop-up Menu (Right Mouse Button)	M	Activates the shortcut menu for the current mode. Same as right-click.
Left Mouse Click	<Spacebar>	Activates a left mouse button click (to add corners, select items, complete, etc.) at the current pointer location.

Table 45. Keyboard Shortcuts for Panning, Zooming and Navigation in SailWind 3D

Name	Shortcut Keys	Description
Zoom to Board	<Home>	Zooms to the board. Fits the board outline into the workspace.
	Ctrl+B	
Zoom Extents	Ctrl+Alt+E	Zooms to extents. Fits all objects in the design into the workspace.
Zoom Area In/Out (Zoom by Drag)	MButton+<Start Drag>	Zooms area in or out. Drag pointer up to zoom in. Drag pointer down to zoom out.

Table 45. Keyboard Shortcuts for Panning, Zooming and Navigation in SailWind 3D (continued)

Name	Shortcut Keys	Description
Zoom to Selection	Alt+Z	Zooms to selection. Fits the selected objects into the workspace.
Center View (Using Pointer Location)	MButton+<Click>	Centers the view at the pointer.
	<Insert>	
Zoom Out	<Page Down>	Zoom out at the pointer.
Zoom In	<Page Up>	Zoom in at the pointer.
Zoom Out at Pointer	Ctrl+<Rotate Backward>	Zoom out at the pointer.
Zoom In at Pointer	Ctrl+<Rotate Forward>	Zoom in at the pointer.
Pan by Drag	Alt+MButton+<StartDrag>	Board moves in direction opposite to dragging direction.

Table 46. Keyboard Shortcuts for Managing the 3D View Point in SailWind 3D

Name	Shortcut Keys	Description
View from Top	Shift+T	View from top.
View from Bottom	Shift+B	View from bottom.
View from Left	Shift+L	View from left.
View from Right	Shift+R	View from right.
Rotate View Around Y-axis	Shift+MButton+<Drag>	Rotate view around Y-axis.
Rotate View Around Z-axis	Ctrl+MButton+<Drag>	Rotate view around Z-axis.

Table 47. Keyboard Shortcuts for File Operations in SailWind 3D

Name	Shortcut Keys	Description
Create New Design File (Blank)	Ctrl+N	Creates a new design file.
Open File (Design)	Ctrl+O	Opens a design file.
Save	Ctrl+S	Saves a design file.

Table 48. Keyboard Shortcuts for Opening Menus and Dialog Boxes in SailWind 3D

Name	Shortcut Keys	Description
Properties (for Selected)	Ctrl+Q	Displays Properties for the currently selected object.
Options > Design Category	Ctrl+Alt+D	Opens the **Design** category of the Options dialog box.

Table 49. Keyboard Shortcuts for Placement Operations in SailWind 3D

Name	Shortcut Keys	Description
Move	Ctrl+E	Moves the selected object(s).
Rotate Selected (90) (during move) (CCW)	<Tab>	Rotates a component (during component move mode) (90 degrees) (CCW).
Rotate Selected (90) (CCW)	Ctrl+R	Rotates the selected components (90 degrees) (CCW).
Rotate Selected (90) (during move) (CW)	Shift+<Tab>	Rotates the selected components (during component move mode (90 degrees) (CW).
Spin Selected	Ctrl+I	Spins the selected object.
Flip Side (Selected)	Ctrl+F	Flips an object to the opposite side of the board.
Align Component	Ctrl+L	Opens Align dialog box.
Delete Component	<Delete>	Delete component(s). (ECO mode must be turned ON.)
Complete Move	LButton+<Click>	Complete move.
	<Space>	
Cancel Move	<Esc>	Cancel move.

Table 50. Keyboard Shortcuts for Editing in SailWind 3D

Name	Shortcut Keys	Description
Cut	Ctrl+X	Cut (ECO mode must be turned ON).
Copy	Ctrl+C	Copy.

Table 50. Keyboard Shortcuts for Editing in SailWind 3D (continued)

Name	Shortcut Keys	Description
Paste	Ctrl+V	Paste (ECO mode must be turned ON).
Redraw	<End>	Refreshes silkscreen, soldermask, plane data in SailWind 3D window.
	Ctrl+D	
Cancel Command	<Escape>	Cancels command.
Redo	Ctrl+Y	Redo.
Undo	Ctrl+Z	Undo.
Reload 3D	Ctrl+<F5>	Reload 3D data and refresh SailWind 3D window.

Chapter 4
Design and Editing Basics

Read the topics that follow to learn the basics of design and editing in SailWind Layout.

Database Limits

As of PowerPCB 4.0, the database limits increased.

The following table shows the old and new database limits:

Tip
If your design uses these new limits, you may not be able to export it into a PADS-format ASCII file compatible with a previous version of the program. For more information, see “ASCII Format” in the *SailWind Layout Command Reference*.

Table 51. Database Limits

Description	PowerPCB version 2 and 3	PowerPCB version 4 and SailWind Layout
Drawing items per design	16,777,216	Same
Drawing pieces per design	16,777,216	Same
Corners per design	16,777,216	Same
Arcs per design	16,777,216	Same
Text strings per design	32,768	Same

Table 51. Database Limits (continued)

Description	PowerPCB version 2 and 3	PowerPCB version 4 and SailWind Layout
Text length per design	16,777,216	Same
Pieces per drawing	32,768	Same
Corners per drawing	16,777,216	Same
Arcs per drawing	16,777,216	Same
Text strings per drawing	32,768	Same
Text length per drawing	16,777,216	Same
Corners per piece	32,768	Same
Arcs per piece	32,768	Same
Drawings per PCB decal	32,768	Same
Reference designator characters	15	Same
Components per design	32,768	16,777,216
Terminals, via types, and jumper types per design	32,768	16,777,216
Gates per part type	100	702
Gates per design	32,768	16,777,216
Pin pairs per design	32,768	16,777,216
Nets per design	32,768	16,777,216
Alphanumeric pin numbers per design	32,768	16,777,216
Alphanumeric pin number length	4	7
Number of layers	30	250
Number of electrical layers	30	64
Number of pins per component	2000	32,768
Decal name length	40	Same
Part type name length		Same
Net name length	47	Same
Layer name length	31	Same

Table 51. Database Limits (continued)

Description	PowerPCB version 2 and 3	PowerPCB version 4 and SailWind Layout
Component rotation precision	0.001×	Same
Pad rotation precision	0.01×	Same
Generic polylines angle precision	0.1×	Same
ASCII file line length	2307	Same
Attribute name length	256	Same
Attribute value length	2048	Same

Tip
Consider all limits of 32,768 and 16,777,216 as formal. Actual limits may be smaller due to memory limitations.

The Selection Filter

The layout of a printed circuit board displays overlapping layers of objects and dense areas of design objects on each layer, making it difficult to select the object you want. Use the Selection Filter to help solve this problem.

Use the Selection Filter to specify the type of object and even the layer on which to select objects. The Selection Filter has two tabs, the Object and the Layer. As a shortcut, you can also use preset Selection Filter setups available on the shortcut menu.

There are three ways to use the Selection Filter. For more information, see the topics in this section.

Filtering Selection Using the Selection Filter Shortcuts

Use the Selection Filter shortcuts to quickly set the Selection Filter to select items in the design.

Procedure

1. Ensure nothing is currently selected in the design. If something is already selected, the shortcut menu will not display the Selection Filter presets. To clear selection in the design, do one of the following.
    - Right-click and click the **Cancel** popup menu item.
    - Click in an empty area of the design space - outside the board area if possible, since you could accidentally select a transparent object within the board area.
2. Right-click and click one of the selection filter presets.

Results

In the Selection Filter dialog box, check boxes are selected according to the preset that you chose.

Filtering Selection by Object Type in the Selection Filter

Use the **Object** tab of the selection filter to specify which type of objects you would like to be able to select in the design.

Procedure

1. Click the **Edit > Filter** menu item.
2. In the Selection Filter dialog box, on the Object tab on page 1710, select the design and drafting items by selecting the appropriate object type check box(es).

Tip
You can use the **Anything** or **Nothing** buttons to quickly select all or clear all check boxes.

3. Click **Close**.
4. Select your items in the design.

Filtering Selection by Layer in the Selection Filter

Use the **Layer** tab of the selection filter to specify on which layer would like to be able to select objects in the design.

Procedure

1. Click the **Edit > Filter** menu item.
2. In the Selection Filter dialog box, on the Layer tab on page 1708, select the layers from the layer list by selecting the appropriate layer check box(es).

Tip
You can use the **Anything** or **Nothing** buttons to quickly select all or clear all check boxes.

3. Click **Close**.
4. Select your items in the design.

CAUTION:
Unlike the check boxes on the **Object** tab, those on the **Layer** tab are not altered by changing any of the Selection Filter presets on the shortcut menu. You must return to the **Layer** tab to re-enable selection of objects on the layer(s) you disabled. Disabling a layer for selection will prevent you from routing to the layer or placing objects on the layer.

Object Selection

Most design work involves adding, modifying, and deleting items. SailWind Layout provides many ways to select items and two ways to edit items.

You can first select the object on page 1842 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.

There are several ways to select objects:

- One at a time
- Several objects at once

- All objects within an area
- All objects of the same type

Single Object Selection

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 design space, all previously selected objects are no longer selected.

But since you may click on a transparent design object and select it, right-click and click the **Cancel** popup menu item to de-select any selected items.

Multiple Object Selection

To select multiple objects, press and hold the Ctrl key while you click over each item in sequence. Alternatively, you can drag a selection area around many objects and everything in the selection area will be selected according to your settings in the Selection Filter on page 114.

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. 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.

You can add additional objects to the selection or remove objects from the selection by pressing the Ctrl key while you click.

Selection Tips

If you try to select an object in a dense or crowded area, use the Selection Filter to disable other items for selection or use the Cycle command. For instructions, see "Cycle Picking". Alternatively, you can make area selection, and then right-click to choose the **Selected List** menu item. For details, see "Selecting Objects by Selected List".

You can click the **Edit > Find** menu item to search for and select objects in the design. For more information, see "Object Finding".

The Project Explorer also enables you to select design objects. For instructions, see "Selecting Design Objects Using the Project Explorer".

Tip
The Drag Moves area in the **Global > General** 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 setting to No Drag Moves.

The following table shows how to use Shift or function key combinations to automatically select multiple items.

Table 52. Selecting Using Shift and Function Keys

Item to select	Key combination
Pin Pair (traces, connection, and both pins)	Shift+click trace or connection.
Whole Net (traces, connections, and pins)	F6+click a trace, pin, or connection.

Table 52. Selecting Using Shift and Function Keys (continued)

Item to select	Key combination
Whole Net (pins only)	Shift+click a pin.
Multiple Trace Segments	Click the first segment, then Shift+click the last segment. All segments in between are selected.
Whole Drawing Shape	Shift+select an edge.

The following table shows how to extend the selection of currently selected objects.

Table 53. Extending the Selection of Selected Objects

Additional items to select	Command
Select Pin Pairs from selected pins	Right-click and click the **Select Pin Pairs**, popup menu item or press the F5 key.
Select Pin Pairs from selected traces	Right-click and click the **Select Pin Pairs** popup menu item, or press the F5 key.
Select Nets from selected pins	Right-click and click the **Select Nets** popup menu item, or press the F6 key.
Select Nets from selected pin pairs	Right-click and click the **Select Nets** popup menu item, or press the F6 key.
Select Nets from selected traces	Right-click and click the **Select Nets** popup menu item, or press the F6 key.
Select Electrical Nets from selected nets	Right-click and click the **Select Electrical Net** popup menu item.

For instructions to select items in the two modes, see the following topics:

- Selecting Using Select Mode
- Selecting Using Verb Mode

Selecting Using Select Mode
Selecting Using Verb Mode

Selecting Using Select Mode

You can apply various commands to an object when you select it first. This is also called *Object Mode*.

Procedure

1. Point to the object and click. The selected item highlights.
2. Use one of the following:

- Right-click to open a popup menu of commands that are appropriate for the selected item.
- Click a toolbar button to start a command.
- Click a menu command.

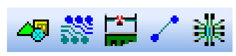
Selecting Using Verb Mode

When you select a command first, you are restricted to select only those design objects to which the command applies.

Procedure

1. Open one of the toolbars by clicking one of the following buttons on the Standard Toolbar. Each toolbar provides a specific set of additional buttons.

Figure 11. Toolbar Buttons



2. Press the Esc key 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 already selected item.
3. Click one of the buttons on the newly opened toolbar. The command attaches to the pointer and the Status Bar displays the command name. Now you can apply the command to objects that you select. When you move the pointer into the workspace, a small V appears on the pointer to show that the selected command is active - the pointer is in verb mode.
4. Select an object 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. Some commands work with area selection (multiple items). For example, the Move command or the Bus Route command.

Examples

On the Design Toolbar, click the **Spin** button. The small V attaches to the pointer within the workspace. Select a component. The component is in Spin Mode. Indicate a new orientation or press the Esc key to cancel the spin.

Cycle Picking

When you cannot easily select the object you want because several objects occupy the same location use cycle picking.

Procedure

1. Move the pointer over the object to select, and click.
2. If you did not select the correct object press the Tab key. Alternatively, you can click the **Cycle** button on the Standard Toolbar, or click **Cycle** on the **Edit** menu, or right-click and click the **Cycle**

popup menu item. Any of these actions deselects the currently selected object and selects a new object at the same location within the pick radius.

3. You can continue to click the **Cycle** button to cycle the current selection through all objects at the pointer location.

Selecting Objects by Selected List

Selected List displays what currently selected graphically, from which to choose the target object(s) you want. It is a more efficient choice for you to select object(s) in a dense or crowded area.

Procedure

1. Make area/multiple selection in dense areas of the design to include your target object(s).
2. Right-click, and click the **Selected List** popup menu item.
3. The Selected List Dialog Box opens and lists all the objects currently selected in the workspace.
4. Select target object(s) in the list by checking the entry check box, which will be highlighted in the workspace simultaneously. You can also:

    - Add additional design objects to the list: Press and hold the Ctrl key while you click.
    - Remove the selected item(s) from the list by clicking **Remove**.

    Note:
    To clear the list, you can also click in an empty area of the workspace.

    - Filter objects by name: Type a wildcard or expression in the Name box, and press the Enter key or click **Filter**. If you do not want to restrict the results with a filter, you can display all items by typing * (asterisk).
    - Fits the selected objects into the workspace by clicking **Fit to View**.
    - Exports what currently in the list to a local CSV file by clicking **Export**.

    For specific parameter description, see Table 200.
5. Close the dialog box.

Selecting Stitching Vias in a Shape

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 inside copper or a copper plane outline.

Procedure

1. Select the copper or copper plane outline.
2. Right-click and click the **Select Stitching Vias** popup menu item.

Isolated Stitching Via Selection

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.

The search for isolated stitching vias ignores connections to CAM plane layers.

- **To select all isolated stitching vias in the design** — With no object selected, right-click in the design area and click the **Select Isolated Stitching Vias** popup menu item.

- **To select isolated stitching vias associated with a net** — Select a net then right-click and click the **Select Isolated Stitching Vias** popup menu item.

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.

Restrictions and Limitations

The Selection Report is available only when at least two objects of different types are selected in your design.

Procedure

1. Select more than one type of object.

2. Right-click and click the **Selection Report** popup menu item, or click the **View > Selection Report** menu item.

Results

The *report.rep* file opens in your default text editor and displays everything you have selected in your design at this moment.

Object Finding

Use the Find dialog box to find and select single or multiple objects by reference designator, part type, line width, or other aspect.

Find works two ways, depending on your selection mode on page 115:

- **Select mode** — Find ignores the Selection Filter on page 114 settings and selects whatever you ask it to.
- **Verb mode** — Find only looks for items that are logical for verb mode.

A few of the Find By commands present more complex commands for finding.

Finding By Attribute

You can find objects by attribute name or attribute value.

Procedure

1. Click the **Edit > Find** menu item. The Find Dialog Box opens.
2. In the “Find By” list, click Attribute. 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**. When you click an attribute, attribute values appear in the list box on the right. You can select more than one attribute.
4. You can select one or more values from the Attribute Value box.
5. Click **OK** to apply the action as selected in the Action list.

Finding By Keepout

You can sort keepouts by restriction type when finding them.

Procedure

1. Click the **Edit > Find** menu item. The Find Dialog Box opens.
2. In the “Find By” list, click Keepouts. A list of restriction types in the design appears in the Restrictions box on the left.
3. To filter the keepouts, type a wildcard or expression in the Value box and click **Apply**. You can select one or more restrictions. If you select a Component Height restriction, a list of available heights appears in the list box on the right.
4. Click **OK** to apply the action as selected in the Action list. The action is applied to all keepouts that have the selected restriction.

Finding Fonts

Use the Find dialog box to find fonts in labels or text strings in your design. When you use it in conjunction with the Select action, you can then use the Properties dialog box to set up new fonts or font styles.

Procedure

1. Click the **Edit > Find** menu item. The Find Dialog Box opens.
2. In the “Find By” list, click Label Fonts or Text Fonts.
3. In the Text Fonts box, click one or more fonts to find. The list includes all fonts in the design. 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 one or more styles to find. The list includes all font styles in the design.
5. Click **OK** to apply the action as selected in the Action list.

Finding By Physical Design Reuse

You can use the Find dialog box to find reuse objects by reuse type or by reuse name.

Procedure

1. Click the **Edit > Find** menu item. The Find Dialog Box opens.
2. In the “Find By” list, click Reuse Type. 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** or click a reuse type. A list of reuse names (names for each instance of the type) appears in the list box on the right.
4. Click **OK** to apply the action as selected in the Action list.

Finding By Test Point Types

Use the Find dialog box to find the test point types in your design.

Procedure

1. Click the **Edit > Find** menu item. The Find Dialog Box opens.
2. In the “Find By” list, click Test Point Types.
3. In the Test Point Types box, click to search by Via, Component pin, or Net. The contents of the list box at the right of the dialog box changes depending on what you select. The table below shows what the List Box displays for each of the available selections

Table 54. 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**.
5. Click **OK** to apply the action as selected in the Action list.

Results

The search result differs depending on which box has the current selection, the Test Point Types box or the one to the right of it. If you have selected a category, all objects of that category are found, but if a specific item is selected in the box to the right of the category, only that item(s) is found.

Finding by Thermal Attributes

Find thermals sorted by their inner pad size.

Procedure

1. Click the **Edit > Find** menu item. The Find Dialog Box opens.
2. In the “Find By” list, click Thermal Attributes.
3. To filter the attributes, type a wildcard or expression in the Value box and click **Apply**.
4. In the Inner Size box, select an 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.
5. Click **OK** to apply the action as selected in the Action list.

Finding by Hatch Outline

Find small copper plane hatch outlines that you want to delete.

Procedure

1. Click the **Edit > Find** menu item. The Find Dialog Box opens.
2. In the “Find By” list, click Hatch outline.
3. Type the area size of the copper plane in squared current design units in the Value field. All copper planes of equal size or less are selected. For example, your current design units are mils. To select all copper plane areas smaller than two square mils, type 4 in the value box.
4. Click **OK**.
5. Press the Delete key.

Finding by Isolated Hatch Outline

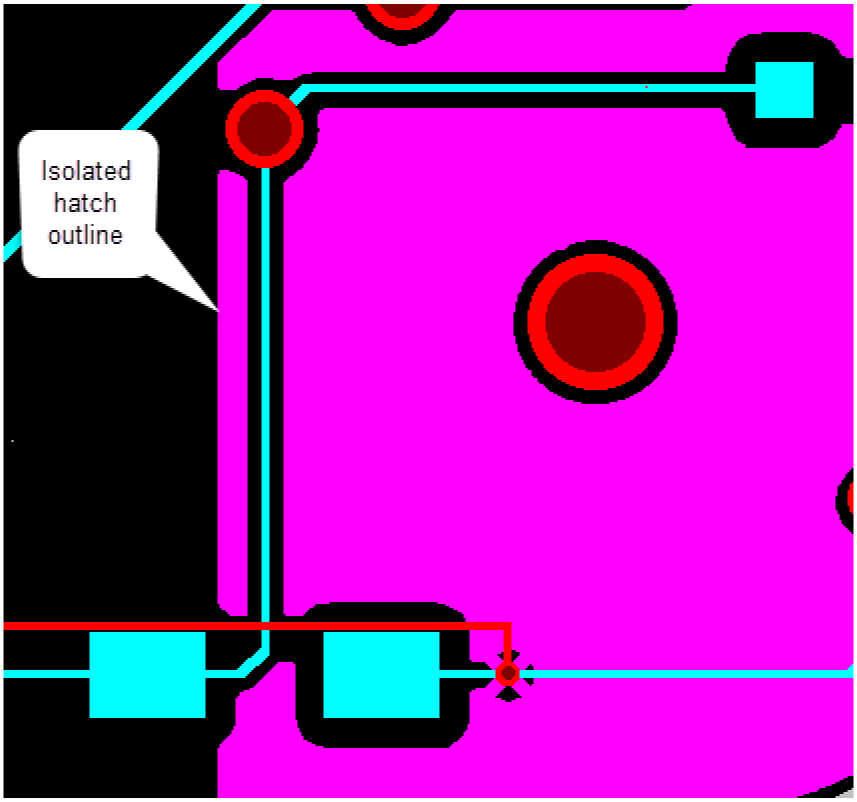
Locate areas of copper plane that have been isolated from the main plane area, but are still part of the same netname. Cleared obstacles that are too close to the edge of the copper plane outline usually cause isolated copper planes.

Procedure

1. Click the **Edit > Find** menu item. The Find Dialog Box opens.
2. In the “Find By” list, click Isolated hatch outline. Copper plane areas not assigned a netname are not considered isolated; they are excluded from this operation.
3. Click **OK**.

4. Press the Delete key.

Figure 12. Isolated Hatch Outline



Highlighting Objects

You can highlight an object without performing an operation on it.

Procedure

1. Select the objects to highlight.
2. Click the **Edit > Highlight** menu item to show the object as highlighted.

   The highlight color from the Display Colors Setup Dialog Box is applied to highlighted objects.

   If you selected the object afterwards, it loses its highlighting.
3. To remove the highlighting from the object, select the object and click the **Edit > Unhighlight** menu item.

Measurement

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.

Procedure

1. Locate your pointer at the starting location of your measurement. Without further movement of the pointer, type the Q Modeless Command on page 83 and then type Enter.
2. Move the pointer in any direction and a line is 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, press the Esc key to exit the Quick Measure mode.

Using Quick Length

Use the QL modeless command to measure routed objects you have selected in the design.

Procedure

1. Select a net, pin pair, or one or more routed segments in the design.
2. Type the QL Modeless Command on page 83 and then press the Enter key. The measurement of the selected route objects is given in a *Layout.err* report file which opens.

Cut, Copy, and Paste Commands

Use the Cut, Copy, and Paste commands just as you would in any other Windows software. You can paste the same copy repeatedly; it remains on the clipboard until you overwrite it with a new copy or cut action. You can also paste into a different design.

While the cut, copy, and paste commands function as you would expect in the Windows OS, these commands also perform uniquely in specific situations:

- **Cut, Copy, and Paste support attributes** — When you cut, copy, or paste objects their attributes are cut, copied, and pasted with them. Attribute labels are also cut, copied, and pasted with the objects to which they are assigned.
- **Cut, Copy, and Paste also support physical design reuses** — You can copy only one physical design reuse at a time. 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 net-name conflicts. This comparison will also detect other errors and warnings, as described in "Process of an Added Physical Design Reuse".
- **Cut, Copy, and Paste a group** — When you paste a group, an error may appear in one of the Trace Copy dialog boxes. For more information, see the "Trace Copy Dialog Box" topic. The Group editing options on page 1503 also affect this.
- **Pasting** — You can only paste items in DRC Off mode.
- **Copy Traces, Traces and Vias, or Routed Pin Pairs Only** — When you select and copy a trace or a trace and via combination, you start a specialized operation that immediately attaches a copy of the selected routing to the pointer and lets you repeatedly paste with a mouse click. This function bypasses the paste command and automatically adds the design object to the pointer and allows you to paste with design rule checking enabled.
- **Capture an Area as a Bitmap Image** — You can use the **Edit > Copy as Bitmap** menu item to define a rectangular area from which to copy graphics information to the Clipboard as a bitmap image, which you can use in other files and documents. For instructions, see "Copying a Bitmap".
- **Copying and Pasting in ECO mode** — Any addition to the design that would force updating the netlist or part list must take place in ECO mode. Use the ECO Options Dialog Box to target and direct the *.eco* file. If something is in the buffer that is not in the current part list or netlist, and you try to paste outside of ECO mode, SailWind Layout prompts you to enter ECO mode to complete pasting.
- **Copy Multiple Selections** — You can copy and paste multiple or mixed selections, or different item types on different layers using a selection rectangle. You can also build extended selections. For example, you can use a selection rectangle to copy an SMD part, its via fanouts, and an associated dimensioning item that was assigned to a documentation layer. You can only select items that appear on screen. You can copy the selected items to the Clipboard, retaining their layers and relative positions.
- **Paste Multiple Selections** — When the Clipboard holds multiple selections comprised of some ECO and some non-ECO registered items, Paste operates differently depending on whether you are in ECO mode. If you paste in ECO mode, all items can be pasted. Part reference designators are added sequentially. If you are not in ECO mode, you can paste only non-ECO registered items.

Delete Command

Like the Copy and Paste operations, the Delete command is sensitive to whether you are in ECO mode. When the ECO Toolbar is not open, you can press the Delete key to unroute selected segments, pin pairs, or nets, leaving the connections intact.

Restriction:
You cannot delete physical design reuses that contain glued components or protected routes.

When the ECO Toolbar is open, you can use Delete to completely remove parts or nets from the design. You can remove any non-ECO items like copper, lines, or text normally. You can also unroute routed traces normally. If you delete a group of ECO and non-ECO items, or combined ECO items (parts and pin pairs or nets), confirmation prompts, one for each item, will appear sequentially.

Copying a Bitmap

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.

Procedure

1. Click the **Edit > Copy as Bitmap** menu item.
2. Area select on page 118 the area to copy. All visible 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 paste to place the image.

Tip
You can also insert the picture with the **Paste Special** menu item in products that support this feature.

Object Snap

You can snap to specified types of objects instead of snapping to the grid when you use the Q (Quick Measure) modeless command, perform any drafting modification, or perform any component placement command.

This includes:

- Add new drafting shape (of any type)
- Modify existing drafting shape (moving a corner or edge)
- Add a new corner to an existing drafting shape
- Split the edge of an existing drafting shape
- Move an entire drafting shape or group of shapes
- Add a new component

- Add a new stitching via
- Move a component or via
- Add a terminal in the Decal Editor
- Move a terminal in the Decal Editor

During all these procedures, the **Snap to Objects** menu item and the **Snap to** sub-menu of object types can be found on the popup menu. You can also control object snapping using the OS Modeless Commands, or the Object Snap options on page 1540.

You can enable the display of markers to indicate the different types of snapping points in the Object Snap options on page 1540.

The snapping to objects takes precedence over snapping to the grid if Snap to grid is enabled for the Design grid when the pointer is within the Snap radius.

If there is more than one object type in the vicinity of the Snap radius, the object type with the highest precedence is selected. The order of precedence is the order in which the object types appear in the Object Snap options on page 1540. You can always filter out the types of objects to snap to if there are conflicts.

You can snap to points on the following design objects as long as they are visible in the design:

- drafting items (2D lines, coppers, keepouts, board outline, dimensioning lines)
- pins
- vias
- components

Chapter 5
Setting Colors

You can set and apply colors in SailWind Layout in one of four ways: to all design objects globally, to individual nets, to design objects set up in the Decal Editor, and to design objects in the CAM document output.

- **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. Once you bring the decal into a design, the decal assumes the design colors. 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.
- **CAM** — Use the Select Items Dialog Box to set colors to objects in CAM before printing the CAM document. You must have a color printer for the colors to be available in the Select Items dialog box.

The following topics cover the Design color settings.

Setting Colors for the Design

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. Click the **Setup > Display Colors** menu item.

 Tip
 You can clear the Display Colors Setup dialog box of the preset color assignments in the Layers/Object Types. Click **Assign All** and in the Assign Color to All Layers dialog box, click Assign Background Color to All Objects and then click **OK**.

2. In the Display Colors Setup Dialog Box, set the color(s) for design objects:

 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 color tile for the object type in the correct layer row.

 Tip
 To change the palette of colors from which you can select, see "Changing 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). For more information, see "Hiding Design Objects".

 Tip
 To automatically make an object type the same color on all layers, use the Assign Color to All Layers dialog box. on page 1094

3. Click **OK**. The display reflects your color settings.

Results

When you save the design, SailWind Layout stores the settings with design data.

You can save display color settings you commonly use as you work on a design. See "Saving 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.

Changing the Color Palette

You can change the color tiles that appear in the color palette. Changing the color tiles allows you to quickly choose custom colors.

Procedure

1. Click the **Setup > Display Colors** menu item.
2. In the Display Colors Setup dialog box on page 1318, 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. 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. You may need to click the selected color tile again to see the color change. The new color appears in the tile you selected.
6. Click **OK** to close the Display Colors Setup dialog box.

Results

You can bring back the original color palette by clicking **Default Palette**.

Making All Objects Visible

The Assign Colors to All Layers dialog box contains several shortcuts for quickly assigning colors to all layers, including the background color.

Procedure

1. Click the **Setup > Display Colors** menu item.
2. In the Display Colors Setup dialog box on page 1318, click **Assign All**.
3. In the Assign Color to All Layers Dialog Box, click Automatically Make Objects Visible.
4. In the Color Preference list, select one of the following:

    - **One Color Per Object Type** — Assigns one color for a certain object type on all layers. For example, assign the color 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** to return to the Display Colors Setup dialog box.
6. Click **OK** to close the Display Colors Setup dialog box.

Results

You can also use the same dialog box to make objects invisible. See “Hiding Design Objects”.

Hiding Design Objects

When you are placing or routing components in a layout design, it can be helpful to make 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.

You can also hide individual net connections or traces using the View Nets Dialog Box.

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".

Procedure

1. Click the **Setup > Display Colors** menu item.

   > **Tip**
   > Save your current color configuration on page 135 to revert to the design colors you were using.

2. In the Display Colors Setup Dialog Box, use the method that matches what you want to do:

   - **Make one object type invisible** — Set its tile to the background color.
   - **Make all objects on a layer invisible** — Clear the check box to the right of the layer name. For specific information on hiding layers including shortcuts, see "Hiding Layers".
   - **Make an object type invisible on all layers** — Clear the check box above the column for that object type.
   - **Make all objects invisible** — Change their colors on all layers to the background color. Click **Assign All**. In the Assign Color to All Layers Dialog Box, click Assign Background Color to All Objects. 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.

3. Click **OK**.

Hiding Layers

Hide layers in the design to reduce clutter in the display. The invisible layers are still listed in dialogs and the layer list, but are invisible in the design.

> **Tip**
> A wide array of shortcuts for hiding layers is available as modeless commands. See the "Modeless Commands for Layer Visibility (Quick Layer View) on page 83" table.

Procedure

1. Click the **Setup > Display Colors** menu item.
2. In the Display Colors Setup Dialog Box, clear the check box to the right of the layer name that you want to hide. As an alternative, you could use the Z modeless command to hide a specific layer. For example, type Z -2 to hide layer 2 from view.

Results

The layer is hidden without affecting the color tiles of the objects on that layer. You could achieve the same effect by assigning the background color to all the objects on the layer but then you would have to reassign colors to the layer objects to make them visible again.

Making Pin Numbers and Net names Visible or Invisible

While working in a layout design, you can use modeless commands to turn display of net names and pin numbers on or off.

- See the PN modeless command in the "Modeless Commands for Object Visibility table" on page 83.
- See NN, NNP, NNT, and NNV modeless commands in the "Modeless Commands for Net Name Visibility table on page 83".

Pin Number and Net Name Display Options

You can set the text size of displayed net names and pin numbers, and the maximum allowable gap between net names on traces.

Use the Display category of the Options dialog box on page 1525.

Saving 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. Click the **Setup > Display Colors** menu item.
2. In the Display Colors Setup Dialog Box, set the color assignments you want.
3. Click **Save** in the Configuration area.
4. In the Save Configuration dialog box that appears, type the configuration name.

- You are not prompted if you overwrite an existing color configuration.

- 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.

Results

The current color settings are saved under the new name. Configurations are saved in the *C:\<install_folder>\<version>\Settings* folder as the saved name with a *.ccf* extension. To change the default display colors setup, name the color configuration *default.ccf*.

To use a previously saved configuration, select the configuration name from the Configuration list.

To delete a configuration, select the configuration in the list and click **Delete**.

Make 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 “SailWind Layout Default Settings”.

For the PCB Decal Editor, see “Creating a New Default Decal Editing Environment”.

Chapter 6
Managing Libraries and Library Data

Read the following topics to learn more about the creation, modification, and management of libraries, including information on library search order, import/export of library data, and the generation of library reports.

SailWind Libraries

The parts, decals, and other items you use to lay out a design in SailWind reside in one or more SailWind libraries.

Libraries store decal and part type attributes and attribute labels, but they do not store attribute values. The following table shows the four kinds of items that can exist in a library.

Table 55. Library Items

Item	Description
Decal	The graphical representation of the part when it is drawn. This is often referred to as the footprint.
Part	Data about a part, including logic family, attributes, pins, and gates. For example, as a 74LS02.
Lines	Graphical data you can store in the library to use in any design file. For example, a company logo.
CAE Decal	The graphical representation of a schematic part, such as a NOR gate. These items are read-only. Use SailWind Logic to create and modify CAE decals.

A single SailWind library consists of 4 files, each containing items of a specific type identified by the file's extension, as follows:

Table 56. SailWind 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 *SailWind Logic Guide*. For information on creating line graphics, see “Adding Drafting Items from a Library” in this manual.

Conversion of Older PADS Libraries to the Current Format

To support new functionality, the library structure for SailWind Layout and SailWind Logic updates from time to time. When you convert earlier versions to the latest version libraries, they change so they become compliant with the latest software. The conversion changes the file name extension.

For steps detailing how to convert libraries, see the Library Converter Help.

Tip
If you have libraries from PowerPCB versions 1 or 2 you need to use the library converter in PowerPCB 3.x.

A report file is created listing which libraries converted along with their conversion status (fully converted, converted with n failures, or no conversion possible). For each library the report will list the items that converted with their status (converted OK or failed to convert).

For information on how to convert older PADS libraries to the current SailWind format, see the SailWind Library Converter User’s Guide.

Creating a New Library

When you create a new library, you actually create a blank container for library content and add it to the Library List. After you create this new library, you can then fill it with content.

Tip
You can also add an existing library - for more information, see “Adding Libraries to the Library List”.

Procedure

1. Click the **File > Library** menu item.
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**.

Results

Your library is added to the bottom of the Library list which is also the last library in the search order on page 145. To move your library up in the library list and search order, see "Setting the Library List Order".

Displaying Items in a Library

Use the Library Manager dialog box to display the items contained in a library.

Procedure

1. Click the **File > Library** menu item.

2. In the Library Manager Dialog Box, select a library in the Library list or select the (All Libraries) item at the top of the list.

3. Click one of these buttons to display categories of items in the library:

    - **Decals** — PCB decals (component footprints)
    - **Parts** — The electrical information that joins with the PCB decal
    - **Lines** — Drafting objects
    - **Logic** — CAE decals (schematic symbols) (Use SailWind Logic to edit CAE decals.)

4. An empty filter box yields no results. To filter the list, type wildcards or expressions on page 155 in the Filter box. If you do not want to restrict the results with a filter, you can display all items by typing * (asterisk) in the Filter box.

5. Click **Apply**.

Results

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 have selected.) The preview window displays a graphic of the library object.

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

Modification of Library Data

SailWind Layout provides a wide range of options for modifying your library data to meet your design requirements.

Adding Items to a Library

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

Procedure

1. Click the **File > Library** menu item.
2. In the Library Manager Dialog Box, select a library in the Library list. 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** — The electrical information that joins with the PCB decal
   - **Lines** — Drafting objects
   - **Logic** — CAE decals (schematic symbols) (Use SailWind Logic to edit CAE decals.)
4. Click **New**
   - **Decals** — Opens the PCB Decal Editor with the default configuration required to create a new decal. For more information, see "Creation of a New Decal".
   - **Parts** — Opens the Part Information Dialog Box with the default configuration required to create a Part Type. For more information, see "Creation of 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. For more information, see "Creating a Drafting Object" and "Saving a Drafting Item to a Library".
   - **Logic** — New and Edit are unavailable. Use SailWind Logic to create CAE decals. For more information, see ""Creating a New CAE Decal"".

Deleting Items from a Library

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

Procedure

1. Click the **File > Library** menu item.
2. In the Library Manager Dialog Box, select a library in the Library list. If you select (All Libraries), the **Delete** button is unavailable.
3. Click one of these buttons to display categories of items in the library:
    - **Decals** — PCB decals (component footprints)
    - **Parts** — The electrical information that joins with the PCB decal
    - **Lines** — Drafting objects
    - **Logic** — CAE decals (schematic symbols)
4. An empty filter box yields no results. To filter the list, type wildcards or expressions on page 155 in the Filter box. If you do not want to restrict the results with a filter, you can display all items by typing * (asterisk) in the Filter box.
5. Click **Apply**.
6. 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 have selected.)

   Use Ctrl+click to select multiple non-sequential items. Use Shift+click or drag the cursor to select a range of items.
7. Click **Delete**.

Copying a Library Item

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

Procedure

1. Click the **File > Library** menu item.
2. In the Library Manager Dialog Box, select a library in the Library list.

   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** — The electrical information that joins with the PCB decal

- **Lines** — Drafting objects
- **Logic** — CAE decals (schematic symbols)

4. An empty filter box yields no results. To filter the list, type wildcards or expressions on page 155 in the Filter box. If you do not want to restrict the results with a filter, you can display all items by typing * (asterisk) in the Filter box.
5. Click **Apply**.
6. Click **Copy**.
7. 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

Editing Items in a Library

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

Procedure

1. Click the **File > Library** menu item.
2. In the Library Manager Dialog Box, select a library in the Library list.

   If you select (All Libraries), the **Edit** button is unavailable.
3. Click one of these buttons to display categories of items in the library:

    - **Decals** — PCB decals (component footprints)
    - **Parts** — The electrical information that joins with the PCB decal
    - **Lines** — Drafting objects
    - **Logic** — CAE decals (schematic symbols)

4. An empty filter box yields no results. To filter the list, type wildcards or expressions on page 155 in the Filter box. If you do not want to restrict the results with a filter, you can display all items by typing * (asterisk) in the Filter box.
5. Click **Apply**.
6. Click **Edit**.

- **Decals** — Opens the PCB Decal Editor with the selected decal. For more information, see "Editing a Library Decal".

- **Parts** — Opens the Part Information dialog box on a selected part. For more information, 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. For more information, see Creating a Drafting Object and "Saving a Drafting Item to a Library".

- **Logic** — Unavailable. Use SailWind Logic to create CAE decals. For more information, see "Creating a New CAE Decal".

Deleting All Items in a Library

You can delete all library items by replacing an existing library with a new empty one of the same name. Doing so ensures that there are no items left inadvertently when you delete a library.

Restrictions and Limitations

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

Procedure

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

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

1. Follow the steps to export library data from the library on page 151.
2. Then follow the steps to import the library data to another library on page 151.

Library Availability and Search Options

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

Adding Libraries to the Library List

If you want to make a library's contents available, add the library to the library list. 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. Click the **File > Library** menu item, and then click **Manage Lib. List**.
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**.

Results

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

Removing Libraries from the Library List

If you want to prevent a library's contents from being used in a design, remove the library from the library list.

Procedure

1. Click the **File > Library** menu item, and then click **Manage Lib. List**.
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.

Library Content and the Search Order

Parts and decals you create do not have to be located in the same library together. When your part refers to a decal in a different library, SailWind Layout automatically picks up the decal when needed.

If you have multiple library items of one type and with the same name, the first occurrence of the item is chosen when the library is searched.

For example, during import of a schematic netlist, the libraries are searched for an 0603 decal. But there are two, each in a different library. The libraries are searched from the first library in the list, to the last library in the list. The first occurrence found will be selected for use to represent the part in the design.

When you have multiple libraries available, they are processed in their order in the library list whenever the libraries are searched. The libraries are searched during the following procedures:

- Searching for library items using various dialogs
- Importing a netlist from the schematic
- Updating your design from the library
- Annotating your design with new components from the schematic

You can change the search order of libraries using the Library List Dialog Box.

Setting the Library List Order

You can specify the order in which libraries are searched.

Procedure

1. Click the **File > Library** menu item, and then click the **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.

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

Sharing a Library Across a Network

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

Procedure

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

Results

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. Click the **File > Library** menu item, and then click the **Manage Lib. List** button.
2. In the Library List dialog box, in the Library list, select the library. 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 for editing. You can set library read-only status only in the Microsoft Windows File Manager. To ensure file protection, the system administrator who owns the files is the only one who can set or clear the read-only status.

Procedure

1. In Windows Explorer, locate your library files. By default, libraries installed with the software are located at *C:\<install_folder>\<version>\Libraries.*
2. Select all four library files, right-click and click the **Properties** popup menu item.
3. In the Properties dialog box, on the **General** tab, select the Read-only check box.
4. Click **OK**.

Results

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

Synchronizing with SailWind Logic

You can enable or disable the synchronizing of library settings between SailWind Logic and SailWind Layout. When you select the “Synchronize with SailWind Logic” check box in SailWind Layout, all library changes you make within SailWind Layout copy also to SailWind Logic.

Tip
To ensure a round-trip synchronization, select the same check box in SailWind Logic.

Procedure

1. Click the **File > Library** menu item, and then click the **Manage Lib. List** button.
2. In the Library List Dialog Box, select the Synchronize with SailWind Logic check box.

Library Attribute Management

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.

Tip
The Manage Library Attributes dialog box does not manage attributes in the design. Use the Attribute Dictionary to manage attributes in the design.

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.

Restrictions and Limitations

- 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. Click the **File > Library** menu item.
2. In the Library Manager dialog box, click **Attr Manager**.
3. In the Manage Library Attributes Dialog Box, in the Select Library list, select an individual library or the (All Libraries) item at the top of the list.
4. In the Item Types list, choose whether to apply the new attribute to All Items, or only either Part Types or PCB Decals.
5. Click **Add Attr**. The Add New Attribute to Library Dialog Box appears.
6. For the Attribute Name, do one of the following:
    - Type an attribute name into the box
    - Click **Browse Lib. Attr** to search all libraries for an existing attribute name
7. If desired, you can type a value in the Attribute Value box.
8. Click **OK**. The Attribute Name appears in the Attributes in Library list.
9. Click **Close**.

Results

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 on page 1584 (for Part Types).

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.

Restrictions and Limitations

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

Results

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 on page 1584 (for Part Types).

Renaming Attributes of Library Items

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

Restrictions and Limitations

Note:
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. Click the **File > Library** menu item.
2. In the Library Manager dialog box, click **Attr Manager**.
3. In the Manage Library Attributes Dialog Box, in the Select Library list, select an individual library or (All Libraries).
4. In the Item Types list, choose whether to rename the attribute(s) from All Items, or from either Part Types or PCB Decals.
5. In the Attributes in Library list, select one or more attributes to rename, and then click **Add>>**.
6. Double-click in the New Name cell. Type the new name that you want to specify for the attribute or to display all existing attributes in the library, click **Browse Lib. Attr**. If you would like to assign one of these existing names, select it and click **OK**. to close the Browse Library Attributes dialog box. This will load the name into the New Name cell.

   Tip
   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.

7. When you have finished with specifying the New Name, click **Rename Attrs**.
8. Click **Close**.

Results

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 on page 1584 (for Part Types).

Import and Export of Libraries

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

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 on page 1590. 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. Click the **File > Library** menu item.
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**.

   Note:
   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**.

Exporting Library Data

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

Procedure

1. Click the **File > Library** menu item.
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** — Exports PCB decals (component footprints)
    - **Parts** — Exports Components
    - **Lines** — Exports drafting objects
    - **Logic** — Exports CAE decals (schematic symbols)
4. An empty filter box yields no results. To filter the list, type wildcards or expressions on page 155 in the Filter box. If you do not want to restrict the results with a filter, you can display all items by typing * (asterisk) in the Filter box.
5. Click **Apply**.
6. Select one or more items in the list, and then click **Export**.
7. 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.

Library Report Creation

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

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

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.

Examples of parts reports are shown below.

Procedure

1. Click the **File > Library** menu item.
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, then 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 include in the report. 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 Parts Report in .lst Format below.)
   - Comma-separated values format (csv): format recognized by MS Excel. (See Parts Report in .csv Format below.)

9. Click **Save**.

10. In the Report Manager dialog box, click **Close**.

Results

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 13. Parts Report in .lst Format

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


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

Figure 14. 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","Z-16A",
"anlogdev","AD1321","AD1321KZ","HIGH SPEED PIN DRIVER WITH INHIBIT","ANALOG DEVICES","Z-16A",
"anlogdev","AD1322","AD1322KZ","ULTRAHIGH SPEED PIN DRIVER WITH INHIBIT","ANALOG DEVICES","Z-16A",
"anlogdev","AD1376","AD1376 (J,K)D","HIGH SPEED, 16-BIT A/D CONVERTER","ANALOG DEVICES","DH-32E",
"anlogdev","AD1377","AD1377 (J,K)D","HIGH SPEED, 16-BIT A/D CONVERTER","ANALOG DEVICES","DH-32E",
"anlogdev","AD1378","AD1378 (S,T)D","WIDE TEMPERATURE, 16-BIT A/D CONVERTER","ANALOG DEVICES","DH-32E",
"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-48A",
```

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 along with the date and time the item was modified.

For more information, see "Creating a Report of the Parts in a Library".

Procedure

1. Click the **File > Library** menu item.

2. In the Library Manager dialog box, select a library from the Library list. 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**.

Results

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.

Wildcards and Expressions

You can use wildcards and expressions to filter the information that displays.

Table 57. 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 58. 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.

Table 58. Usage Examples of Wildcards and Expressions (continued)

Expression:	Results in all items that:
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.
[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 7
Creating and Editing PCB Decals

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

The following topics present a detailed description of how to create and edit PCB Decals using the PCB Decal Editor. They also discuss the setup of the PCB Decal Editor as well as the process of updating a design from the library after you have edited the decals.

Setting Up the PCB Decal Editor

You use the PCB Decal Editor to create new decals or to modify existing ones. You can modify the Decal Editor's layers, colors, rules, editor options, and Decal Wizard options.

Procedure

1. Click the **Setup > Layer Definition** menu item to change the layer stackup in the Layers Setup on page 1441dialog box. If you need to add objects to specific layers, use the Layers Definition settings to configure the design to your requirements.

2. Click the **Setup > Display Colors** menu item to change the colors of objects or layers in the Display Colors Setup on page 1318 dialog box. As you work on a decal, you may find it useful to apply different colors to objects or layers to visually differentiate them.

   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. Click the **Tools > Options** menu item to change the options in the Options dialog box on page 1531.

4. Click the **Setup > Decal Rules** menu item to change the decal rules in the Decal Rules on page 1231dialog box.

> **Note:**
> Rules set at this level of the rules hierarchy are used only by SailWind Router.

Related Topics

Creating a New Default Decal Editing Environment

Creating a New Default Decal Editing Environment

SailWind Layout provides 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-the-box default environment.

If you have personal preferences or your company has specific design standards for decal creation, you can modify and save the default environment's settings to create a new default that will support those preferences. This can help you to improve productivity by loading your preferred setting each time the Decal Editor is opened.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item. The Decal Editor opens with the default configuration.
2. Set options and settings to reflect your personal preferences or company design standards. For more information, see "Setting Up the PCB Decal Editor".
3. Click the **File > Save as Start-up File** menu item.
4. In the Start-up File Output dialog box, select the appropriate check boxes in the Sections area, and click**OK**.

   When you immediately click the **File > New** menu item, 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. Click the **Tools > PCB Decal Editor** menu item to start the Decal Editor.
2. Click the **Setup > Display Colors** menu item.
3. In the Display Colors Setup Dialog Box on page 1322, 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. To change the palette of colors from which you can select, click **Palette**. For more information, see "Changing 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). For more information, see "Hiding Design Objects".

    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. on page 1094
4. Click **Apply**.

    If you modify the display color settings, you can save the new configuration to a file for reuse in future design sessions. For more information, see "Saving Color Assignments to a File".

Tip
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

Creation of a New Decal

In the PCB Decal Editor, you can use the Decal Wizard to create some footprints automatically, or use the Drafting Toolbar commands to manually create a decal.

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.

In order to create a decal, you need to begin with the package dimensions and enlarge the decal to the appropriate material condition on page 1838.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item to start the Decal Editor.
2. On the Standard Toolbar, click the **Drafting Toolbar** button to open the Drafting Toolbar, and then click the **Wizard** button.
3. Choose one of these methods to calculate the required dimensions and create a new PCB decal:
    - **Manually Calculate the Required Decal Dimensions** — You manually calculate the decal dimensions required for the package and enter the dimensions into the Decal Wizard to create the new decal.
    - **Use the Decal Calculator to Generate the Required Dimensions** — You enter the package dimensions into the Decal Calculator area of the dialog box (if the feature is available for the specific component family) and let the Decal Calculator generate the decal according to preset standards of material conditions.
4. 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. For more information, see “Decal Editing Tasks”.

Related Topics

The Decal Wizard Options Configuration File

The Decal Wizard Options Configuration File

You cannot modify or delete the Default configuration of the Decal Wizard Options but you can save a new configuration file.

This configuration file is saved in the *UserDir* folder. The *UserDir* environment variable is set in the *\Settings\SailWindpcb.ini* file, and by default is set to the *\Settings* folder. The file has an extension of *.dwc*. These files are in text format and can be opened with your default text editor.

Custom Package Types and Pitch Ranges

You can use your default text editor to further customize the configuration file you saved from the Decal Wizard Options dialog box. You can edit the existing package types or add new package types.

If you edit the text of your *.dwc* file, you must consider the following requirements:

- Only pre-defined component families can be used, and all the required parameters for the component family must exist (for example, Toe).
- The component_type section numbers must not skip a number; however they do not need to be in consecutive order in the file. For example, if numbers 4 and 6 exist, there must also be a number 5 somewhere in the file.
- Pitch ranges cannot overlap and, all together, must cover all values from 0 to infinity.
- The DecalName parameters are restricted to <Pitch>, <LeadSpan>, <LeadSpan1>, <LeadSpan2>, <Height>, <BodyWidth>, <BodyLength>, and <PinCount>.

If these requirements are not met and you attempt to use the configuration in the Decal Wizard Options dialog box, a warning appears to explain the problem and you cannot use the configuration file.

Creating a Decal Manually

If you cannot create a decal using the decal wizard, you can create it manually using the Decal Editor Drafting Toolbar buttons.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item to start the Decal Editor with the default decal editing environment.

   You do not 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 on page 170.
3. Add and place the terminals on page 172.
4. Create any copper shapes on page 179, such as copper associated with terminal pads.
5. Define the terminal pad stacks on page 185.
6. Create the Assembly on page 195, Silkscreen on page 196 and Placement on page 197 part outlines.
7. Add other items, such as labels, attributes, keepouts, solder mask relief, height values, and hard breakouts/fanouts. For more information, see “Decal Editing Tasks”.

- 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. For more information, see “Creating Placeholder Attribute Labels”.

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

Related Topics

Library Content and the Search Order

Library and Component Decal Edits

You can edit a library decal or the decal assigned to a component in a design.

Modifications to Components

When you add a component from the library to a design, the component is copied from the library into the design as a separate instance.

There are then two versions of the component - the library version and the design version. You can modify either or both of these; but which one you modify makes a difference:

- If you want the changes to apply only to the Library version of the component, use the PCB Decal Editor to make the changes. If you modify the Library version of a component, only that version is changed; instances of the component existing in the design are not changed. (If you want to change the design components to match the Library version, you can click the **Tools > Update from Library** menu item.)

- If you want the changes you make to apply only to the component(s) in the current design, use the Layout Editor to make the changes. If you modify the design version of a component, the change affects only the selected component(s) in the current design (or all instances of that component in the design); the Library definition of the component is not changed (unless you specifically change it by right-clicking and clicking the **Save to Library** popup menu item with the component selected in the design).

Most of the items that make up a component definition can be edited only in the Layout editor, or only in the PCB Decal Editor, not in both places. But the following items can be edited in either editor, and you must decide which editor to use:

- In the component part type:
    - Attributes
- In the component's decals:
    - Attributes
    - Labels
    - Silkscreen
    - Keepouts

- Copper
- Solder mask
- Paste mask
- Part outline width
- Decal rules
- Pad stack

Editing a Library Decal

You can use the Decal Editor to modify PCB Decals in a library. The changes you make apply at the library level but do not apply automatically to instances within the design.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item to start the 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 on page 155 in the Items box and click **Apply**.

   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 SailWind libraries, the Part Type List for Decal <name> dialog box opens. Review the information and close the dialog box. For more information, see "Checking for Errors Between Part Types and Assigned PCB Decals".

7. Edit the decal. For information on how to edit a PCB decal, see "Decal Editing Tasks".
8. Save the decal.

Checking for Errors Between Part Types and Assigned PCB Decals

If you open a decal for editing in the Decal Editor, and you have already assigned it to part types in the SailWind Libraries, the “Part Type List for Decal <name>” dialog box opens.

From this 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.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item to start the 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 on page 155 in the Items box and click **Apply**.

   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**.
7. If the decal is assigned to any part types in the SailWind libraries, the Part Type List for Decal <name> dialog box on page 1606 opens. To open the dialog manually, click the **Tools > Part Types** menu item. You can sort the table by a given column by clicking the column header.
8. On the Part Type List for Decal <name> dialog box, click **Edit Part**.
9. The Part Information for Part <name> dialog box opens. From this dialog box, you can check to ensure that the pins match the pins listed in the **Pins** tab table, or when pin mapping exists - on the **Pin Mapping** tab.
10. When you have finished reviewing the information, click **OK** to close the Part Information for Part <name> dialog box.
11. Click **Close** to close the Part Type List for Decal <name> dialog box.
12. Edit the decal. For information on how to edit a PCB decal, see “Decal Editing Tasks”.
13. Save the decal.
14. Click the **File > Exit Decal Editor** menu item to return to your design.

Fixing Logical Errors

You can correct any logical errors that appear in the Error Status column of the “Part Type List for Decal” dialog box.

Procedure

1. On the **Tools** menu, click **PCB Decal Editor** to open the 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 on page 155 in the Items box and click **Apply**.

   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**.
7. If the decal is assigned to any part types in the SailWind libraries, the Part Type List for Decal <name> dialog box on page 1606 opens. To open the dialog manually, click the **Tools > Part Types** menu item. You can sort the table by a given column by clicking the column header.
8. On the Part Type List for Decal <name> dialog box, select a line item with a logical error.
9. Click **Show Errors** to create and open a report with details of the errors.
10. Click **Edit Part** to open the Part Type.
11. In the part type, fix the errors and then click **OK**.
12. The Part Type List for Decal dialog box will update the Error Status.

Fixing Mismatched Pins Errors

You can correct any mismatched pins errors that appear in the Error Status column of the “Part Type List for Decal” dialog box.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item to start the Decal Editor.
2. Click **Open**. 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 on page 155 in the Items box and click **Apply**.

   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**.
7. If the decal is assigned to any part types in the SailWind libraries, the Part Type List for Decal <name> dialog box on page 1606 opens. To open the dialog manually, click the **Tools > Part types** menu item. You can sort the table by a given column by clicking the column header.

8. On the Part Type List for Decal <name> dialog box, select a line item with a mismatched pin error.
9. Click **Show Errors** to create and open a report with details of the errors.
10. In the PCB Decal Editor, correct the error.
11. In the Part Type List for Decal dialog box, click **Refresh**, to update the dialog box.

Tip
You can click Check Decal 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 Assigned to a Component in the Design

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

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

Procedure

1. Select the component(s). The selected components must all use the same decal.
2. Right-click and click the **Edit Decal** popup menu item to open the PCB Decal Editor.
3. Edit the decal. For information on how to edit a PCB decal, see "Decal Editing Tasks."

Restriction:
If your design is in default layer mode and you edit a design decal and attempt to change it 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 click **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. After you have completed the edits to the decal in the library, you can then switch the current design to increased layer mode and update the decal.

4. If you want to save changes to the library:

a. Click the **File > Save Decal As** menu item.

b. 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.

c. Click **OK**.

5. Click the **File > Exit Decal Editor** menu item.

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.

Decal Editing Tasks

You can use the Decal Editor to perform editing tasks, including editing properties, locating the decal origin, adding, deleting and modifying terminals, creating and associating copper shapes, creating and modifying custom pad stacks, working with labels and attributes, and a multitude of other editing tasks.

SailWind Layout combines decals and part types to create components. Use the PCB Decal Editor to create and edit the decal associated with a part type in the parts library. Many PCB Decal Editor drafting operations are identical to Layout Editor drafting operations.

When the PCB Decal Editor starts, your current design is stored and PCB Decal Editor takes over. The colors used in the PCB Decal Editor come from the Layout Editor. When you exit the PCB Decal Editor, use File commands to save information, and exit as you would a stand-alone program. You will return to the current design in SailWind Layout.

When editing decals, note the following:

- SailWind Layout supports 16 alternate decals per part type.
- You can use Dimensioning within the PCB Decal Editor; however, dimensions are converted to 2D lines and text when you save the decal. For more information, see the Dimensioning Process topic.

PCB Decal Editor drafting works with the objects that make up a decal:

- Decal name
- Terminals
- 2D lines
- Text
- Copper
- Copper voids

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 then right-click and click the **Properties** popup menu item. You can also Double-click on the item, or select the item and click the **Properties** button on the Decal Editor Toolbar.
2. In the displayed dialog box, edit the decal name properties.
3. Click **OK**.

Location of 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 are creating a new decal, you can define the origin whenever it is most convenient. For example, if you are creating a through-hole device decal:

- You could create Pin 1 on the default origin, and then build the remainder of the decal relative to that origin.
- You could build the entire decal anywhere within the workspace, and then move the origin to Pin 1.

Setting the Decal Origin with Set Origin

You can set a decal's origin while you are editing it in the Decal Editor.

Procedure

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

Setting the Decal Origin Using a Modeless Command

You can set a decal's origin by using a modeless command and specifying the x and y location coordinates.

Procedure

1. Select the decal.
2. Type the modeless command SO <x> <y>.

Tip
Type SO for relative coordinates, SOA for absolute coordinates.

Terminals

Terminals define the physical connection points on the PCB decals. The terminal Properties establish the connectivity relationship between the schematic symbol and the PCB decal.

Adding Terminals

Terminals are the pads or lands of a decal.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item to start the Decal Editor.
2. Click the **Drafting** button to open the Drafting Toolbar, and then click the **Terminal** button.
3. In the "Add Terminals Dialog Box" on page 1073, in the Start pin number area, choose a pin type and then 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.
    - 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 the Increment prefix or Increment suffix option.
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 Verify valid JEDEC pin numbering check box to ensure that legal alphanumeric values are used. This option only ensures that valid alpha and numeric combinations are used. To arrange rows and columns according to JEDEC, use the **Tools > Assign JEDEC Pinning** menu item.

7. Click to indicate a position for the new terminal. Repeat as necessary.

> **Tip**
> You can also add terminals by replicating one or more existing terminals. See "Using Step and Repeat to Add an Array of Terminals" on page 173.

Related Topics

Creating Copper in the Decal

Associating Copper with Terminals

Using Step and Repeat to Add an Array of Terminals

You can add an array of terminals using the step and repeat function.

Procedure

1. Select one or more terminals, right click and click the **Step and Repeat** menu item.
2. In the Step and Repeat dialog box on page 1734, 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.
4. Click **OK** to create the array.

    - 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. Click the **Tools > PCB Decal Editor** menu item to start the Decal Editor.
2. Click the **File > Open Decal** menu item.
3. To filter the decals, in the Filter area, type a wildcard or expression on page 155 in the Items box and click **Apply**.
4. Click to indicate the decal to which you want to assign JEDEC pinning and click **OK**.
5. Click the **Tools > Assign JEDEC Array Pinning** menu item. The JEDEC Array Pinning Dialog Box appears.
6. Choose the Decal Type by selecting either the Component or Substrate option.

7. Click **OK**.

8. Save the Decal.

Modifying Terminal Properties

You can modify the various properties assigned to terminals such as the x,y locations, pin numbers, pad stacks and copper associations.

Procedure

1. With a decal open in the PCB Decal Editor, select a terminal, right-click and then click the **Properties** popup menu item.

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 on page 1574. Use this dialog box to modify one or more selected pad stacks.

4. In the Terminal Properties dialog box, you can clear the Associated Copper check box to disassociate copper from the terminal.

5. Click **Apply** to apply your modifications or **Cancel** to cancel the changes.

6. 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

Aligning Objects

Customizing Pad Stacks of Decal Pins

Modifying Terminal Number Properties

You can modify the properties of an individual terminal to change its location or pin number.

Procedure

1. With a decal open in the PCB Decal Editor, select a terminal number, right-click, and then click the **Properties** popup menu item.

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

You can move one or more terminals to change their location within the decal.

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.

Related Topics

Moving Design Objects Radially

Moving a Terminal Number

You can move one or more terminal numbers to change their location within the decal.

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.
4. 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

You can swap numbers between two terminals to reorder them.

Procedure

1. In the PCB Decal Editor, select the terminals you want to swap.
2. Right-click and click the **Swap** popup menu item.

   The terminal numbers switch.

Terminal Renumbering

When constructing PCB decals, you may need to renumber the terminals. There are two methods to renumber terminals.

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. With a decal open in the PCB Decal Editor, select a starting terminal, right-click, and then click the **Renumber Terminals** popup menu item.

   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.

   - 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 the Increment prefix or Increment suffix option.

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.

   This option only ensures that legal alphanumeric combinations are used. To arrange rows and columns according to JEDEC, use the **Tools > Assign JEDEC Pinning** menu item.

6. Click **OK**.

   - 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.
   - 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 the **Back** popup menu item or press the Backspace key to restore the original number. If you selected a group of terminals to renumber, Back will undo the renumbering one terminal at a time. To undo completed terminal renumbering click Undo from the Edit menu.

9. To cancel the terminal renumbering process at any time, right-click and click the **Cancel** popup menu item. SailWind Layout restores the original terminal numbering.

Tip
To renumber multiple terminals, click and hold the left mouse button, then drag the pointer across the terminals while pressing the Shift key. When you release the mouse button, all of the selected terminal numbers update, excluding terminals that are already renumbered.

---

10. Right-click and click the **Complete** popup menu item when you finish reassigning terminal numbers.

Renumber 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.

The Pin Numbers dialog box can be accessed from the **Setup > Pin Numbers** menu item.

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

You can use the Renumber Pins dialog box to edit individual pin numbers on a decal.

Procedure

1. Click the **Setup > Pin Numbers** menu item.

2. Double click a Number cell, or select a number cell and click **Edit**.

3. Type a new pin number.

Copying and Pasting Pin Numbers

You can copy table data you select from the Pin Numbers dialog or from Microsoft Excel and paste it into the Numbers list. The cell you select in the table is the *paste origin*. Pasted data appears 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 **Copy** in the Pin Numbers dialog box.
2. Select the cells into which you want to paste the data. Data will only paste into selected cells.
3. Click **Paste** to paste the data into the table starting at the paste origin.

Deleting a Terminal

You can delete unwanted pins directly from within the PCB Decal Editor.

Procedure

1. In the PCB Decal Editor, select the terminal to delete. You can select more than one at a time.
2. Press the Delete key. The remaining terminals are automatically renumbered. The alphanumeric labels remain unchanged.

Moving a Decal Name

You can drag a decal name with the mouse to move it from one location to another.

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

Rotating 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.

Prerequisites

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, click 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. For more information, see “Set 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.

Results

Examine the resulting copper shape and look for any of these conditions:

- If the shape edges are not correct, see “Edge Precision of Drafting Shapes on page 615”.
- If the shape is on the wrong layer or needs to be modified, see “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

Surface Mount Device Pads

Surface Mount Device (SMD) pads are usually rectangular, although you can assign any shape. When defining an SMD pad, set the pad stack for all other layers to a round shape, size 0 pad, drill size 0, and clear Plating. The drill oversize parameter does not apply. If you use associated copper to define the SMD pad, set the pad stack for that layer to a square shape, size 0.

Use the Via under SMD routing command to place vias directly under SMD pads. For more information, see “Adding Vias in Pads”. You can undersize the SMD pads for the paste mask photoplot during the CAM process.

Plane Connections

Pins that are supposed to connect to the plane are usually plotted for manufacturing using spoked thermal relief pads.

Creation of Custom Pad Shapes

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. Use associated copper to create custom pad shapes or to create hard breakouts in decals.

CAM interprets pads and other objects differently when they are associated with copper in the PCB Decal Editor. 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.

Restrictions and Limitations

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".

   You can create multiple copper shapes.

2. With a decal open in the PCB Decal Editor, select a terminal, right-click and then click the **Associate** popup menu item.

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 on page 1854 and also the connection point for thermals if this pad will ever be connected to a copper plane. If associated copper surrounds the terminal pin by a large amount it could prevent thermal spokes from being generated.

Note:
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.

Results

CAM interprets terminals differently when associated with copper. For more information, see Associated Copper and CAM.

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 the **Unassociate** popup menu item.

Tip
As an alternative, in the Terminal Properties, you can clear the Associated Copper check box to unassociate copper from the terminal.

Pad Sizes and Pad Stacks

A *pad* is a small area of copper that acts as a conductor for component pins and vias. The pad ensures connectivity between the trace entering the drill hole and the copper plating that lines the inside of the hole.

Pads are classified as two types:

- **Through hole pads** — Are used for components that mount with pins that go through the board. When through hole pads are drilled and plated, a small ring of copper remains and ensures connectivity between the trace entering the drill hole, the copper plating that lines the inside of the hole, and the pad on the opposite side of the board. Vias are considered through hole pads, but may be created to go through only certain layers.

- **Surface mount pads** — Are used for components that have pins, which are glued to an outside layer of the board. Routing to vias provides connectivity to other layers.

Pad Stacks

On a two-layer board, a component pin or via drill hole has a separate pad on the top layer and the bottom layer. You can set different shape, size, and diameter values for each pad. If you add any inner routing layers to the design, you can define pads on those layers. The resulting two or more pads for the pin or via is called the *pad stack*.

Pad stacks are classified into two categories:

- **Component pad stacks** — Component pad stacks are used for component pins and are either surface mount, with no drill diameter, or through hole. Pad stack information for a component is stored with its part decal information.

- **Via pad stacks** — Via pad stacks are used for feed-throughs and can be through hole or partial. Partial vias begin or end on an outer or inner layer. Partial vias are used on multilayer boards and are created by drilling laminate layers separately for layer-dedicated vias, then pressing them together and drilling the through holes. If a via connects an outer layer and an inner layer, it is called a blind via. If a via connects two inner layers, it is called a buried via. The via type determines whether a via is through hole or partial. The via description is the combination of type, plating, and pad stack information.

You can edit pad stacks by layer, so you can set component or via pads to zero, turning them off, on layers where they are not needed to create more routing real estate around a drill hole. You can assign different shapes to them for different routing or photoplot applications. The resulting configuration of size, shape, diameter, and layer description for a pad stack is called its pad stack information. For more information, see the Editing Pad Stacks topic.

For information on installing vias, see the Via Setup on page 327 topics.

Pad Stack Default Layers

The default layer list features a special generic layer list. It lists Mounted Side, Inner Layers, and Opposite Side. Such a setup allows quick customization of the electrical layers and adds flexibility if you choose to mount the component on the bottom side or add extra inner layers.

Mounted/Opposite Side

If you customize the Mounted Side or Opposite Side, your customized pad will flip with the component if you choose to mount the component on the bottom side of the board. However, if you choose to customize the specific layer of your top or bottom side of the board, that customized pad will remain on that layer even though you flip the component.

Inner Layers

If you customize the Inner Layers, you can add layers to your design and your pad customizations will propagate to any added. However, if you choose to customize each specific layer, and you increase the layers of your design, your customized pads will remain only on those specific internal layers.

Pad Stacks and Antipad Definitions

Setting an antipad definition for the inner layer modifies the photoplot output for CAM planes and split/mixed planes. If you want a unique antipad on split/mixed plane layers, add a new layer for the split/mixed plane before defining the antipad.

Related Topics

Setting Up an Inner Layer

Pad Stacks and Associated Copper

When defining associated copper for a terminal, define a zero size, square-shaped pad in the terminal pad stack on the layer of the associated copper. Routing commands interpret the zero size, square-shaped pad to be the routing target for the associated pad copper on that layer.

Drill Size

During manufacturing, the interior surfaces of drill holes are coated with metal plating. For vias, the plating enables connectivity when the layers are pressed together. Plating reduces the diameter of drilled holes. The size difference does not affect vias as much as component holes, where a smaller diameter can hinder part insertion.

The drill size you define for a pad stack is assumed to be the finished hole size, after plating. Manufacturing should use a larger drill bit than the specified drill size specified so that once the plating is added, the resulting inner diameter is at, or close to, the original finished specification. So that you can use the actual drill bit size in clearance checking, there is a universal “Drill oversize” setting on the **Design** tab of the Options dialog box, which adds a fixed amount of diameter to all drill size definitions. The combination of the pad stack drill size and the drill oversize setting is the diameter used by the batch design rule checks. This is also the drill hole size that displays on a pin or via.

In most cases, manufacturers use drill sizes equal to the pad stack drill size plus twice the thickness of the plating. To determine what value to enter, know how your board manufacturer chooses drill diameters.

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.

Restrictions and Limitations

When customizing pad stacks of decal pins, the following restrictions apply

- If you are 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 are working in the Design Editor, the decal you want to edit must exist in the design.

Procedure

1. Click the **Setup > Pad Stacks** menu item.
2. In the "Pad Stacks Properties Dialog Box" on page 1566, ensure that your decal is selected in the Decal name list.
3. Under the "Pin: Plated:" list, select the pin to customize. If you do not want to customize all the pins simultaneously, you can add one or more specific pin numbers to the list using the Add Pin Dialog Box. on page 1066
4. Under the "Sh: Sz: Layer:" list, select one or more layers of the pad stack you want to customize.

   For more information, see:

   - Pad Stack Default Layers on page 184
   - 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.

   For more information, see:

   - "Design Rule Versus Pad Stack - Thermals and Antipads" on page 797
   - Creation of Thermals in the Pad Stacks
   - Creation of Antipads in the Pad Stacks
   - 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.

Results

Verify that your custom thermals are showing up in CAM plane plots. For custom thermals to be visible, 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 Copper Plane Thermal Attributes

Creation of Thermals in the Pad Stacks

You can set up thermals wherever you can create or modify pad stacks. Create thermals on split/mixed plane layers and CAM negative planes (for RS-274-X output).

Restrictions and Limitations

When creating thermals in pad stacks, the following restrictions apply:

- In the PCB Decal Editor, thermals cannot be created in the Pad Stack Properties for Pin dialog box. Instead, click the **Setup > Pad Stacks** menu item 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 plane and split/mixed plane layers. If you want a unique thermal on a particular layer, add the new layer and define the specific thermal setting.

Procedure

1. Select one or more layers.
2. In the Parameters area, click the **Thermal** tab.
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. No pad stack thermal will be used if the “Use design rules for thermals and antipads” check box is selected in the Options dialog box

> **Copper Planes** category > Thermals subcategory on page 1514. For more information, see "“Customizing Design Rule Thermals” on page 669”.

Note:
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 on page 1641 dialog box. Make sure you are also using RS-274-X output format in the Photo Plotter Advanced Setup on page 1619 dialog box.

Related Topics

Customizing Pad Stacks of Decal Pins

Creation of 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 and Limitations

In the PCB Decal Editor, antipads cannot be created in the Pad Stack Properties for Pin dialog box. Instead, click the **Setup > Pad Stacks** menu item to use the Pad Stacks Properties dialog box.

Procedure

1. Select one or more layers.

   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 plane and split/mixed plane layers. If you want a unique antipad on a particular layer, add the new layer, and define the specific antipad setting.

   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 Parameters area, click the **Antipad** tab.

3. Click the appropriate pad shape to enable the size and shape controls for antipads. 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. No pad stack antipad will be used if the “Use design rules for thermals and antipads” check box is selected in the Options dialog box > **Copper Planes** category > Thermals subcategory on page 1514. For more information, see “Customizing Design Rule Antipads” on page 670.

   Note:
   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 on page 1641 dialog box. Make sure you are also using RS-274-X output format in the Photo Plotter Advanced Setup on page 1619 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. This practice has some common exceptions. For example, a square pad usually designates pin 1 of through-hole components.

Procedure

1. Click the **Setup > Pad Stacks** menu item.
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 and then click **OK**.
5. When you are prompted to save changes to all or the selected parts click the appropriate button:
    - **All** — Use this choice 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** — Use this choice 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.

Related Topics

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.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item to open the Decal Editor.
2. Open a decal.
3. Select the pin or pins for which you want to change the pad stack. Alternatively, click the **Setup > Pad Stacks** menu item and select or add the specific pad stack that needs to be edited.
4. Right-click and click the **Pad Stacks** popup menu item. The Pad Stack Properties for Pin Dialog Box appears.
5. In the “Pin No. Plated”: area, click the pin you want to change, then in the “Sh: Sz: Layer:” area, click a layer on which you want to change the pad stack information. If you opened the Pad Stacks properties dialog box from the Setup menu, you can edit multiple pad stack layers at once.

6. Make the pad stack modifications.

7. Click the “Assign to all selected pins” check box 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. To save the changes, you must also save the decal.

> **Note:**
> 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.

Related Topics

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. You can save pad stack changes to the parts library.

Procedure

1. Select the part.

2. Right-click and click the **Save to Library** popup menu item. 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.

Pad Stack Report

You can create a text format, pad stack report file for a selected component or via or all components and vias.

A sample file output appears below.

```
PAD STACK LISTING
Finger Format — Size Shape Orientation Length Offset Drill
Pad Format —— Size Shape Drill
Shape=[OF]OvalFinger,[RF]RectFinger,[R]Round,[S]Square,[A]Annular,[O]Odd
Pad stack for Via: STANDARDVIA
START(1)  55 C 37
INNER  70 C 20 (inner layers not otherwise described)
```

```
END(2)   55 C
Pad stacks for Part Decal: DIP14
Used by Part Types: 7400
PIN 0 (All pins not otherwise described) (Plated)
TOP(1)   60 C  37
INNER  80 C  (inner layers not otherwise described)
BOTTOM(2)  60 C
PIN 1 (Plated)
TOP(1)   60 S 37
INNER  80 C (inner layers not otherwise described)
BOTTOM(2)  60 C
Pad stacks for Part Decal: DIP14\SO
Used by Part Types: 7400
PIN 0 (All pins not otherwise described) (Plated)
TOP(1)   24 R 90 60 0 0
INNER  0 C (inner layers not otherwise described)
BOTTOM(2)  0 C
```

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.

For custom antipads, the default antipad shape around the slotted hole is always oval. For custom thermals, the default pad shape around the slotted hole depends on the pad shape on the specific layer.

For custom thermals, settings on the Pad Stacks Properties Dialog Box control spoke angle and width. The Clearance Rules dialog box on page 1167 pad to copper clearance controls the calculation of the outer width.

Note:
A custom thermal or antipad for a slotted hole has the same orientation and offset as the slot.

Other information on slotted holes includes:

- Slotted holes are displayed in the same color as drills.
- The Drill Oversize option in the Options dialog box **Design** tab applies to plated slotted holes.
- Drill-to-drill clearance checking checks slotted holes.
- You can use slotted holes as test points.

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

Slotted Hole Geometry
Slotted Hole Offset Versus Pad Offset
CAM Output and Slotted Holes
Slotted Holes in CAM350
Creating Slotted Holes in Decals
Creating Slotted Holes in Pins

Slotted Hole Geometry

The valid range of values for slotted hole length, orientation, and offset parameters are the same as those for a pad. They are expressed in the design units selected in the associated Pad Stacks Properties dialog box.

Length Value Range: >= 0 and <= 1000 mils (25.4 mm)

Orientation Value Range: >= 0.000 and <= 179.999 degrees

Offset Value Range: >= -500 (-17.2 mm) and <= 500 mils 17.2 (mm)

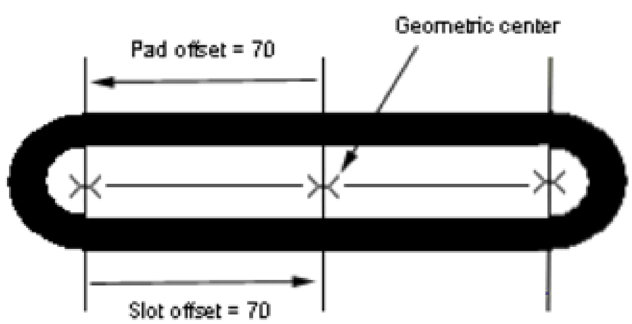
Slotted Hole Offset Versus Pad Offset

You can change the offset of oval pad shapes to move the electrical center of the pin (as well as the center of the drill). Since standard conventions consider slotted holes to be drills, such conventions also consider the electrical center to be the center of the slotted hole.

If you moved the pad offset to the far end of the pad, you would quickly move the slotted hole outside the pad boundary. Instead of moving a pad offset, you can use a slotted hole offset to move the slotted hole.

Slotted hole 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. For example, if you want to move the electrical center of a 200x60 mil pad 70 mils to the left, set a pad offset of 70. To center the slotted hole, set a slotted hole offset of 70. See the graphic below for more information. The maximum amount of offset you can set is one half the length.

Figure 15. Pad Offset Versus Slot Offset



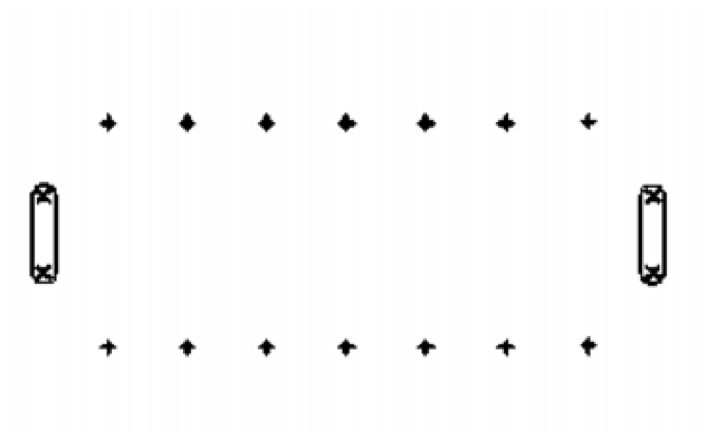
CAM Output and Slotted Holes

The CAM output type determines how slotted holes are defined and represented.

Drill Drawings

Slotted holes are shown on drill drawing as 2 drill symbols, one at each end of the centerline of the slotted hole. The true outline (edge-to-edge representation) of the slotted hole is then drawn around the two drill symbols.

Figure 16. Drill Drawing of a DIP14 with 30x120 Mil plated Slotted Holes at Each End



Slotted holes are shown in the drill chart as a value in the Size column. The size is the edge-to-edge size of the slotted hole, meaning that the width of the slotted hole is also the drill size. The Quantity column shows the number of slots.

The table below shows a sample Drill chart of a DIP14 with a 30x120 mil plated slotted holes at each end

Table 59. Sample Drill Chart

SIZE	QTY	SYM	PLTD
35	14	+	PLTD
30x132	2	X	PLTD

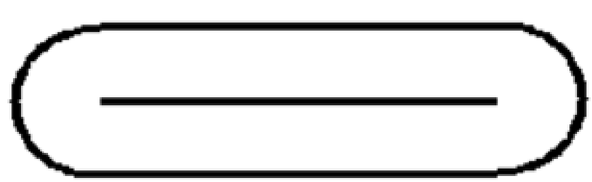
NC Drill

Two drill symbols for each end of a slotted hole are created in the NC drill data. Slotted holes are output according to pin type output (plated/nonplated). Slotted hole test points are output according to test point output. For example, if you output plated pins and test points, slotted holes that are plated and/or test points will also be output.

Gerber Output

Slotted holes are represented two ways in Gerber data. The first is as a centerline with endpoints that are one half the drill size from the ends of the slotted hole. The second ways is as a closed, unfilled oval, showing the outer edge of the slotted hole. The centerline of this oval is the outer edge of the slotted hole. Both are drawn with the smallest round aperture available.

Figure 17. Gerber Output of Slotted Holes



Slotted Holes in CAM350

The method of fabrication determines the hole representation you use in CAM350.

- To create slotted holes with a series of drills, use the centerline format and the Gerber to Drill feature in CAM350.
- To mill slotted holes, use the outer edge format and Gerber to Mill in CAM350.

Related Topics

Slotted Holes

Creating Slotted Holes in Decals

Creating Slotted Holes in Pins

Customizing Pad Stacks of Decal Pins

Pad Stack Properties for Pin Dialog Box

Drill Drawing Options

Creating Slotted Holes in Decals

You can add slotted holes to a decal in the Decal Editor.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item to open the Decal Editor.
2. Click the **Setup > Pad Stacks** menu item.
3. In the Pad Stacks Properties Dialog Box, set any pad stack options you want, such as plated, size, shape, and layer.
4. Select the Slotted check box to add a slotted hole to the decal.
5. Set the length, orientation, and offset of the slotted hole.
6. Click **OK**.
7. 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

You can select individual pins on a decal and add slotted holes.

Procedure

1. Select a pin, right-click, and then click the **Properties** popup menu item.
2. Click **Pad Stack**.

3. In the Pad Stacks Properties Dialog Box, 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

SailWind Layout uses the assembly outline and assembly refdes on the assembly drawing to define the exact outline and identify the body of the part. The assembly outline shows where the part is to be placed during the board assembly process.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item to open the Decal Editor.
2. Click the **Drafting Toolbar** button then click the **2D line** button and then on the Assembly Drawing Top layer, draw the outline. For more information on drawing 2D shapes, see Creating a Drafting Object.

   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 **Add New Label** .

   b. In the Add New Decal Label on page 1060 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 the **Move** popup menu item.

   f. Position the new label.

   Tip
   To create assembly layer reference designators globally in the design editor, see Generating a Second Set of Reference Designators for Assembly Drawings.

Results

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. on page 1318

Creating Silkscreen Decal Objects

The silkscreen outline prints 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 anyone 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. Click the **Tools > PCB Decal Editor** menu item to open the Decal Editor.
2. Click the **Drafting Toolbar** button then click the **2D line** button and then on the Silkscreen Top layer, draw the outline. For more information on drawing 2D shapes, see Creating a Drafting Object.

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”.
5. Silkscreens for large components with many pins often have miniature dots or pin numbers visible next to selected pins to make it easier locate a pin on the board during testing or troubleshooting sessions. 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 **Add New Label**.
   b. In the Add New Decal Label on page 1060 dialog box, in the Attribute box, type a name for the new attribute. 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 the **Move**. popup menu item.

f. Position the new placeholder.

Results

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. on page 1318

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

Some parts need additional spacing to allow for machine-placement or rework on the physical board. The placement (or nudge) outline defines these spacings.

The placement (nudge) outline (sometimes called a "courtyard") defines the boundary where nudging on page 499 of the part initiates when another part is moved against it.

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. Click the **Tools > PCB Decal Editor** menu item to open the Decal Editor.
2. Click the **Drafting Toolbar** button then click the **2D line** button and then on layer 20 (120 in max layers mode), draw the 2D line item(s) defining the nudge outline. For more information on drawing 2D shapes, see Creating a Drafting Object.

> **Tip**
> The outline does not have to be a contiguous shape enclosing the part. It can be made up of multiple 2D items—any object placed on Layer 20 (the Placement layer) is considered part of the "outline".

Results

The placement 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. on page 1318

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.

Restrictions and Limitations

DXF import only supports the following geometries: POINT, LINE, ARC, CIRCLE, ELLIPS, TRACE, SOLID, 3DFACE, POLYLINE, LWPOLYLINE (AutoCAD R14), and BLOCKS with hierarchy.

Procedure

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 on page 297.

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.

 Note:
 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**.

Results

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 Join and Close 2D Lines and Copper Shapes.

Related Topics

Importing DXF Files

DXF Format

Attributes in the PCB Decal Editor

You can add attributes 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 does not exist in the PCB Decal Editor; therefore, attributes in the PCB Decal Editor are text strings only. They do not 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 (to edit a design decal), 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.

Restriction:
When you enter PCB Decal Editor using the Tools menu item (to edit a library decal), 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.

Creating Attributes in the PCB Decal Editor

You can define attributes for a decal by typing in the desired name or selecting an attribute name from a pre-defined list.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item to open the Decal Editor.
2. Click the **Edit > Attribute Manager** menu item.
3. In the Decal Attributes Dialog Box, click **Add**.
4. Do one of the following:

    - Type an attribute name in the new Attribute cell, and press the Enter key. Attribute names are not checked in the PCB Decal Editor.
    - Select the new Value cell and click **Edit**.
    - Type an attribute value, and press the Enter key. You can also type units for the value. If you do not specify units, the decal units are used.

5. Click **Close**.

Tip
You can also add attributes using the **Browse Lib. Attr.** button.

Related Topics

Attributes in the PCB Decal Editor

Control of Solder Mask and Paste Mask

Modifying an Attribute

You can use the Attribute Manager to modify an attribute.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item to open the Decal Editor.
2. Click the **Edit > Attribute Manager** menu item.
3. In the Decal Attributes Dialog Box, Select the Attribute cell or the Value cell for the attribute.
4. Click **Edit**.
5. Type a new name or value, and press the Enter key.
6. Click **Close**.

Creating Attribute Labels in the PCB Decal Editor

Labels in the PCB Decal Editor offer you greater flexibility than labels in the Layout Editor; you can display either decal attributes or an attribute from an object that uses the decal.

Other than the enhanced flexibility, you use attribute labels in the PCB Decal Editor exactly as you use labels in the Layout Editor on page 581.

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. For more information, see “Creating Placeholder Attribute Labels” on page 201

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. Click the **Tools > PCB Decal Editor** menu item to open the Decal Editor.
2. With a decal open, click the **Drafting Toolbar** button to display the Drafting Toolbar.
3. Click the **Add New Label** button.
4. In the Add New Decal Label Dialog Box, in the Attribute list, select the attribute you want, or if the attribute you desire is not shown in the list, in the Attribute list click <Browse Lib. Attr.> to pick from the list of all library attributes. If you are creating labels for jumpers, Reference Designator is the

only available attribute. Also, 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.

5. 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.

6. 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. 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. Additionally, be aware that Value is also unavailable if the attribute is ECO-registered and SailWind Layout is not in ECO mode.

7. 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 on page 1862). 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.

8. Complete the label configuration by specifying the desired font, the layer for the label, position, X,Y location, rotation, size, stroke width, mirroring, justification and right reading selections. For more information on the details of these settings, see Add New Decal Label Dialog Box.

9. Click **OK**.

Related Topics

Creating Placeholder Attribute Labels

Creating Placeholder Attribute Labels

A *placeholder label* is a label for an attribute that does not exist.

Placeholder labels have two practical uses:

- 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 does not exist yet.

Procedure

1. When you create a label on page 200 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.

4. 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 that the label displays.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item to open the Decal Editor.
2. Select a decal label on page 589, right-click, and then click the **Properties** popup menu item. If you select multiple labels, settings in this dialog box apply to all selected labels.
3. 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.

   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.
4. 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. 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. Additionally, be aware that Value is also unavailable if the attribute is ECO-registered and SailWind Layout is not in ECO mode.
5. 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 on page 1862). 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.
6. Complete the label configuration by specifying the desired font, the layer for the label, position, X,Y location, rotation, size, stroke width, mirroring, justification and right reading selections. For more information on the details of these settings, see Decal Label Properties Dialog Box.
7. Click **OK**.

Related Topics

Creating Attribute Labels in the PCB Decal Editor

Non-Decal Attributes

A decal label always associates with an attribute name. At any time, that attribute may or may not be a decal attribute.

For example, you may create a decal attribute called My Attribute and then create a label for it. If you delete the decal attribute, the label remains, but it is now associated with a non-decal attribute with the name My Attribute.

The reverse is also true. You can create a label and associate it with an attribute that is not a decal attribute (by choosing an attribute from the Library Attribute Manager or by typing an attribute name in the Add New Decal Label dialog box). The label is now associated with a non-decal attribute.

Creating Keepout Areas in a Decal

You can create keepouts in the PCB Decal Editor. Decal-level keepouts appear similar to other keepouts. When you create a decal-level keepout, any part using the decal in the Layout Editor uses the specified keepout restrictions.

You can create a decal-level keepout that restricts Trace and Copper, Copper Plane, Via and Jumper, and Test Points. You cannot create keepouts for placement, component height, or component drills. You create a keepout in the PCB Decal Editor the same way you create one in the Layout Editor.

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 cannot move a decal-level keepout independently of the part to which it belongs. Once you create a decal-level keepout, you must enter the PCB Decal Editor to modify any properties of the keepout. You modify keepouts in the PCB Decal Editor exactly as you would in the Layout Editor. User interaction is the same for either; the only difference is the type of objects you can restrict.

For more information on creating and modifying keepouts, see the Keepouts on page 465topic.

Procedure

1. With a decal open in the Decal Editor, click the **Drafting Toolbar** button to open the Drafting Toolbar, and then click the **Keepout** button.
2. Right-click and click a draw mode from the popup menu for the type of shape you want 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. If you select the Placement restriction, it prevents placement of other components. Components bodies are repelled from one another on mounted layers using the clearance distance specified in the Body to Body design rule on page 1167. In addition, you can create a larger nudge outline on page 197 or courtyard that also applies only to the mounted layer. But a component keepout can be placed on any layer. For example, you create a placement keepout on the opposite side of the board in order to keep the area below free of components. Since the keepout is added to the library decal, it moves with the component in the design.
5. Click the layer on which to place the keepout. 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 the variable <Opposite Side> layer. You cannot do this in the Layout Editor - keepouts are assigned to a specific layer.
6. Click **OK**. The keepout is created. If you create other keepouts, they use the restrictions you set here as the default.

Results

If you export your design in ASCII format to an older version of the software where Placement keepouts are not valid, you will receive a warning message. If the Placement restriction is the only restriction, the keepout will not be exported. If the Placement restriction was not the only restriction, the keepout will be exported without the Placement restriction.

Modifying Decal-level Keepouts

After you create a decal-level keepout, you must use the PCB Decal Editor to modify any of its 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, and then click the **Select Shape** popup menu item.
2. Right-click and click the **Properties** popup menu item.
3. Turn restrictions on or off and modify the layer settings. For more information, see Drafting Object Properties.
4. Click **OK**.

Restriction:
You cannot 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.

Layers for Decal-level Keepouts

To create keepouts assigned to Inner Layers (any layer other than top or bottom), you must increase the number of layers to three.

Decal-level keepouts use the same layer assignments as keepouts created in the Layout Editor, plus an additional option for Opposite Layer. Opposite Layer assigns restrictions to the side opposite the one on which you place the component. For example, while editing a decal, create a Copper Plane keepout and choose Opposite Side from the Layer list. In the Layout Editor, and with the component mounted on the top side, the keepout prevents copper planes in that area on the bottom layer. If you click Flip Side to place the component on the bottom layer, the keepout prevents copper planes in that area on the top layer.

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 the **Generate Drafting Shape** popup menu item.
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.

Step and Repeat

The Step and Repeat tool found in the PCB Decal Editor defines complex, repetitive array patterns so that the fanout of traces from a component on a Device Under Test PCB is consistent. Such consistency ensures device simulation and testing takes place under exacting conditions.

Step and Repeat arranges terminal, drawing, copper, cutout, or text items in a planar or polar array pattern. You can replicate single or multiple items.

Step and Repeat also automatically increments repeated terminals and text suffixes.

Linear Step and Repeat

Use the Step and Repeat dialog box to create linear arrays.

Figure 18. Initial Setup for a Linear Step and Repeat

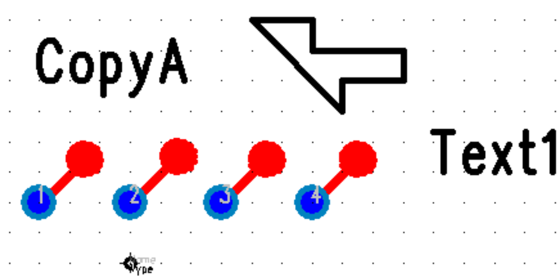
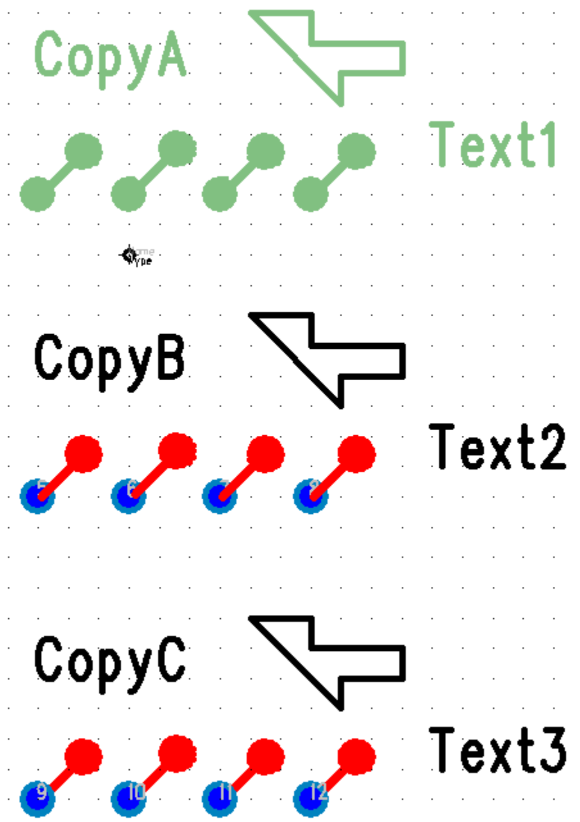


Figure 19. Results after Performing a Linear Step and Repeat



Polar Step and Repeat

Use the Polar tab in the Step and Repeat dialog box to create angular, or circular arrays.

Figure 20. Initial Setup for a Polar Step and Repeat

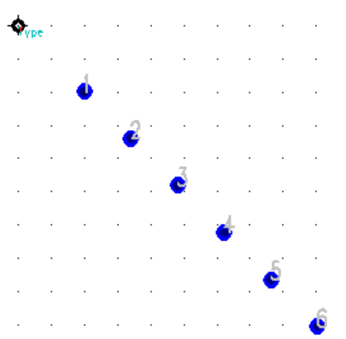
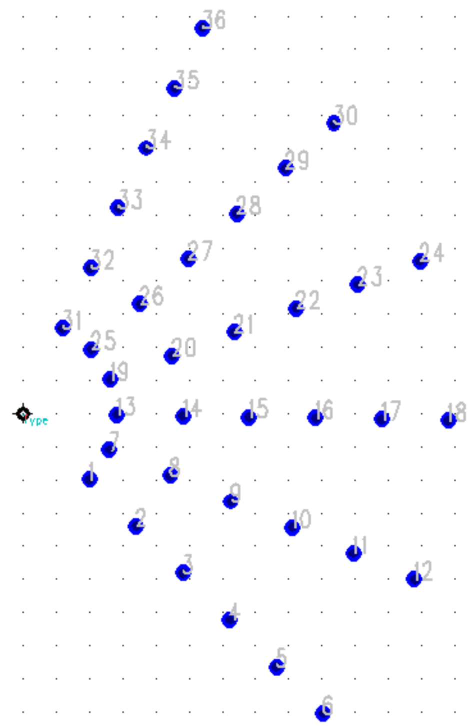


Figure 21. Results after Performing a Polar Step and Repeat

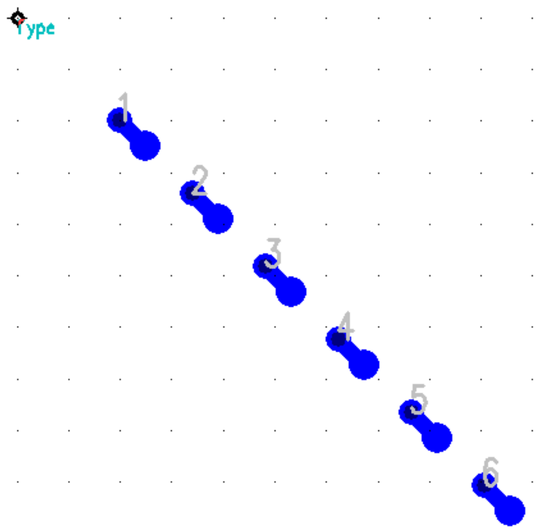


Radial Step and Repeat

Use the Radial tab in the Step and Repeat dialog box to create radial arrays.

Figure 22. Initial Step for a Radial Step and Repeat with Associated Copper

Figure 23. Results after Performing a Radial Step and Repeat with Associated Coppers



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.

Control of Solder Mask and Paste Mask

You can use several methods to control solder or paste mask openings in the gerber outputs. You can add physical shapes as openings on the mask layers and you can also generate the openings when creating the gerber outputs. Since different areas can have multiple values, the precedence of these values follow a hierarchical rule. You can use any combination of such methods.

Physical Shapes in the Work Area

When you create a physical shape (opening) on the mask layer using the following methods, you can view it in the Decal Editor and the Design Editor.

In the Pad Stacks

You can add the solder and/or paste mask layer to a pad stack and create the shape and size of the opening. And since you can create all of a component's pins at the same time, you can potentially add mask openings to all the pins of a component at once. This method limits you to the pad shapes that are available to pad stacks.

You can do this at the design level to make it design specific, or at the library level to make it available for each new design that uses the decal.

In the Decal Editor

When you edit the decal in the Decal Editor, you can add a copper shape to a solder mask or paste mask layer and that shape becomes the opening in the mask. There are two ways to create the shape:

- You can draw the shape freehand using copper. There is no limit to the shape like in the pad stacks.

> **Tip**
> Use this method to create gang solder mask relief by drawing a copper rectangle over a whole row of pads.

- You can use the Generate Drafting Shape command to generate a copper shape based on the shape of a terminal that you can add to the solder or paste mask layer. This has the same effect as adding the shape to the pad stack.

Generated Shapes in the CAM Output

When you generate these shapes in the CAM Output you cannot view them in either the Decal Editor or the Design Editor. They are only visible in the CAM Preview.

In the CAM Output

When you create the solder or paste mask output, you can add the pads (from the component layer) and the pad shapes become the mask openings. You can globally oversize or undersize the pad shapes with a setting in the Plot Options.

In an Attribute

You can apply a unique over or under size setting to special components using an attribute. You can assign an oversize or undersize by adding the “CAM.Solder mask.Adjust” or “CAM.Paste mask.Adjust” attribute to a decal. The attribute value applies to all pads of the component.

Mask Hierarchy

There is a hierarchical order to the solder mask and paste mask values when there are conflicts. When multiple values exist at different hierarchy levels, the highest priority value is used; the others are ignored.

The following table content is true only while the CAM.Apply Oversize To All Pads attribute on page 960 is set to Yes (the default setting on all new designs in PADS 2007.1 and later versions).

Table 60. Solder & Paste Mask Size Priorities

Hierarchy Level	Description
1 (Highest Level - overrides all others)	• Pad sizes on the top and bottom solder and paste mask layers in the pad stack. If a pad size is defined on both the component level and the decal level, the priority order is: a. Component instance level value b. Library decal level value For more information, see Customizing Pad Stacks of Decal Pins • Component level over/undersize values of the CAM.Solder mask.adjust, and CAM.Paste mask.adjust attributes. This applies to Components and Vias. **Tip** The CAM.Solder mask.adjust and CAM.Paste mask.adjust attributes are combined not only with pad shapes on the Solder mask or Paste mask layers, but in the absence of those pad shapes they can also be combined with top and bottom layer component pads when added to the configuration of the solder and paste mask layers during the creation of the CAM document. For more information, see Design Object Attributes **CAUTION:** Because these two items are of equal priority, when they are both defined, they are combined to generate the mask shape.

Table 60. Solder & Paste Mask Size Priorities (continued)

Hierarchy Level	Description
2	• Library decal level over/undersize values of the CAM.Solder mask.adjust, and CAM.Paste mask.adjust attributes. **Tip** The CAM.Solder mask.adjust and CAM.Paste mask.adjust attributes are combined with top and bottom layer component pads when added to the configuration of the solder and paste mask layers during the creation of the CAM document. For more information, see Creating Attributes in the PCB Decal Editor
3	• PCB level over/undersize values of the CAM.Solder mask.adjust, and CAM.Paste mask.adjust attributes. For more information, see Adding an Attribute to All Objects in the Design
4 (Lowest level - is overriden by all others)	• Over/undersize value of the global CAM, Plot Options > Over(Under)size Pads By value in combination with the CAM.Apply Oversize To All Pads attribute. For more information, see Plot Options Dialog Box

Related Topics

Creating a Solder Mask Gerber-format File

Creating a Paste Mask Gerber-format File

Tented Vias With Solder Mask

Solder Mask Openings in the Decal Editor

Using the Decal Editor, you can create custom solder mask openings for the solder mask layers.

Creating a Solder Mask Opening by Copying a Pad Shape

You can create a solder mask opening by copying a pad shape, over- or undersizing it, and saving it as copper on the Top or Bottom solder mask layer.

Procedure

1. Open the decal from the library or the design.
2. With nothing selected, right-click, and click the **Select Terminals** popup menu item.
3. Select the terminal you want to copy, right click, and click the **Generate Drafting Shape** popup menu item.
4. In the Generate Drafting Shape Dialog Box, from the Type list, select Copper.
5. From the Layer list, select Solder Mask Top or Solder Mask Bottom.
6. Set Width and “Oversize/Undersize value” as desired.
7. Click **Create**.
8. Click **Close**.

   To see the new opening shape(s), you will need to open the Display Colors Setup Dialog Box, assign a color to the Top or Bottom solder mask layer, and make the layer active by selecting it from the Layer List.

Creating a Solder Mask Opening by Drawing the Opening Shape

You can create a solder mask opening by drawing the mask opening shape in copper on the Top or Bottom solder mask layer.

Tip
You can also use this procedure to create gang relief of solder mask.

Procedure

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 on page 613.

   To see the new opening shape(s), you will need to open the Display Colors Setup Dialog Box, assign a color to the Top or Bottom solder mask layer, and 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, and then click the **Select Components** popup menu item.

2. Select the component that has the pin(s) for which you want to create solder mask openings.

3. Right-click, and then click the **Properties** menu item.

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, click the **Add** button under the Pin: Plated: list, and select the pin. If not, go to Step 6.

6. Click 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**.

   To see the new opening shape(s), you will need to open the Display Colors Setup Dialog Box, assign a color to the Top or Bottom solder mask layer, and make the layer active by selecting it from the Layer List.

Related Topics

Control of Solder Mask and Paste Mask

Paste Mask Openings in the Decal Editor

Read the sections that follow to learn how to use the Decal Editor to create custom paste mask openings for the paste mask layers.

Creating a Paste Mask Opening by Copying a Pad Shape
Creating a Paste Mask Opening by Drawing the Opening Shape

Creating a Paste Mask Opening by Copying a Pad Shape

You can create a paste mask opening by copying a pad shape, over- or undersizing it, and saving it as copper on the Top or Bottom paste mask layer.

Procedure

1. Open the decal from the library or the design.
2. With nothing selected, right-click, and then click the **Select Terminals** popup menu item.
3. Select the terminal you want to copy, right-click, and then click the **Generate Drafting Shape** popup menu item.
4. In the Generate Drafting Shape Dialog Box, in the Type list, click Copper.
5. In the Layer list, select Paste Mask Top or Paste Mask Bottom.
6. Set Width and Oversize as desired.
7. Click **Create**.
8. Click **Close**.

   To see the new opening shape(s), you will need to open the Display Colors Setup Dialog Box, assign a color to the Top or Bottom paste mask layer, and make the layer active by selecting it from the Layer List.

Creating a Paste Mask Opening by Drawing the Opening Shape

You can create a paste mask opening by drawing the mask opening shape in copper on the Top or Bottom paste mask layer.

Procedure

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 on page 613.

   To see the new opening shape(s), you will need to open the Display Colors Setup Dialog Box, assign a color to the Top or Bottom paste mask layer, and 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, and then click the **Select Components** menu item.
2. Select the component whose pin(s) you want to create paste mask openings for.
3. Right-click, and then click the **Properties** menu item.
4. In the Component Properties Dialog Box, click the **Pad Stack** button.
5. In the Pad Stacks Properties Dialog Box, if you want to create an opening for only a single pin, click the **Add** button under the Pin: Plated: list, and select the pin. If not, go to Step 6.
6. Click 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**.

   To see the new opening shape(s), you will need to open the Display Colors Setup Dialog Box, assign a color to the Top or Bottom paste mask layer, and 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

When you import a netlist into a design, SailWind Layout writes a copy of the library content for the parts into the design. Over time, however, as you make updates to the library, the parts in the design fall out of synchronization 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

Restrictions and Limitations

When updating a design from the Library, these restrictions will apply:

- 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.
- To find differences between the library and design in the comparison report, search for “different”.
- Design changes resulting from updating can cause existing electrical nets to be truncated, split, or deleted altogether. It can also cause existing electrical nets to be extended, or new electrical nets to be created if the refdes prefixes of updated components are specified in the Electrical Nets dialog box. For more information, see “Electrical Nets” on page 337.

Procedure

1. Verify that the library order is correct. Updating from the library searches libraries for part types using the library order shown in the “Library List dialog box” on page 1448.
2. If you want to compare/update only specific components, select them. Otherwise, the update from library process selects all components in the design by default.
3. Click the **Tools > Update from Library** menu item.
4. Modify the settings in the Update from Library Dialog Box.
5. Click **OK**.

   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.

Related Topics

Undo an Update From Library

Prevention of Update From Library Undo Buffer Overruns

Undo an Update From Library

If an update operation updates at least one item, the update process creates an undo checkpoint so that you can undo any changes made to the design. You can revert to the last-saved checkpoint using the **Edit > Undo** menu item.

Tip
If nothing updates, the update from library process does not create a new undo checkpoint and preserves undo data and undo checkpoints created by previous commands.

Prevention of Update From Library Undo Buffer Overruns

SailWind Layout stores the data required to undo an update 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:

“Too much data to fit into the Undo buffer. The Undo buffer will be cleared. Continue?”

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 by comparing the design content with the library content. If it finds the design content to be different from the library content, it updates the design content according to the options set in the Update From Library dialog box, and documents the results in the report file. The compare process is the same, except that the design is not updated.

General Rules

During Compare/Update, the system will enforce a number of general rules to assure proper execution of the process.

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. Libraries are searched in the order shown in the Library List dialog box (**File > 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. If no match is found, comparison of the item is skipped, and a “not found” entry is made in the report file.

- **Timestamps** — Each part type and decal in the design and library has a timestamp. Depending on the options chosen in the Update from Library dialog box, comparison of the timestamps can determine what is compared/updated. See Update from Library Dialog Box. Designs created with SailWind Layout versions prior to 9.1 do not have timestamps. See Older Designs for information on how to handle these designs.

What is Compared

During the Compare/Update process, a number of parameters related to the Part Types and the PCB Decal constructions are compared.

The following tables list the content compared and reported for each item selected for comparison or update.

Table 61. Compared and Reported Part Type Structures

Item or Element	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

.

Table 62. Compared and Reported PCB Decal Structures

Item or Element	What is Compared & Reported
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

Table 62. Compared and Reported PCB Decal Structures (continued)

Item or Element	What is Compared & Reported
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
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

Table 62. Compared and Reported PCB Decal Structures (continued)

Item or Element	What is Compared & Reported
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
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), text font, 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. This is important to consider when working with older designs.

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, SailWind Layout writes the results of the operation to a new *UpdateFromLibrary_Layout_Report.txt* file in *C:\SailWind 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
==============================================================================================================
                                                                    Design                 Compared   Update
 Part Type                                 Found in Library         Timestamp              Content    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
==============================================================================================================
                                                                    Design                 Compared   Update
 PCB Decal                                 Found in Library         Timestamp              Content    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
LCC205Q                                  | 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       0.00
    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       0.00
    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     150.00
    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      50.00
    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      50.00
       Point Location               /                 same /    2:(    -118.00      50.00 ) mils    /    2:(    -118.00      50.00
       Point Location               /                 same /    3:(    -118.00     -50.00 ) mils    /    3:(    -118.00     -50.00
       Point Location               /                 same /    4:(     -37.50     -50.00 ) mils    /    4:(     -37.50     -50.00
 ------------------------------------------------------------------------------------------------------------------------------
 .
 .
```

Update Report Messages

The following tables list and describe the messages returned for compared/updated items in the Update Report.

Table 63. Part Type and PCB Decal Summary Messages

Appears in Column:	Message	Explanation
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.
	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

Table 63. Part Type and PCB Decal Summary Messages (continued)

Appears in Column:	Message	Explanation
		• 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 do not qualify for Compare/Update (for example, items with identical timestamps), or • The item was not found in the library.

.

Table 64. Part Type and PCB Decals Comparison Details Messages

Appears in Column:	Message	Explanation
Comparison	same	Item content in the design and library are the same. (Order can be different.)
	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 8
Creating and Modifying Part Types

Read the topics that follow to learn more about the processes used to create, modify, and save part types to the libraries, as well as the structures involved in assigning pin numbers and gates to part types.

Part Types

You cannot add a part to a SailWind Layout design, or import the part as a component of a netlist, unless the library has an available part type and a PCB decal for the part.

- The PCB decal represents the "footprint" of the part (including pad and drill sizes) on the board, and is required both to place the part on the board and to route the nets connecting to the part's pins.
- The part type represents the electrical configuration of the part, including its logic family, pins, gates and gate pins, and mapping of gate pins to the part's pins. It also includes other information, such as part attributes and the list of PCB decals assigned to the part. When you add a part to a design, it is added as an instance of the part type.

You create a new SailWind part type on page 230, or modify an existing part type on page 235, by specifying the properties of the part in the tabs of the Part Information dialog box, as follows:

- General — on page 1590Part statistics, logic family, pin number mapping status, connector status, ECO-registered status, list of reference designator prefixes to define the set of parts to which these properties apply, define part as special purpose (connector, die part, or flip chip).
- PCB Decals — on page 1593List of assigned PCB Decals. You must assign a decal to the part before specifying gate information, signal pin names, and alphanumeric pin names.
- Gates — on page 1588Default and alternative gates, gate swapping eligibility, gate properties, gate pin-swapping eligibility.
- Pins — on page 1596Signal names for pins, such as power and ground pins.
- Attributes — on page 1584Attributes of the part.

- Connectors — on page 1586Default and alternative CAE decals used as connectors in SailWind Logic. 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.
- Pin Mapping — on page 1600Optional pin mapping of logical to physical pin numbers This tab is unavailable when you select the Define Mapping Of Part Type Pin Numbers To PCB Decal check box on the General tab, and assign one or more decals containing alphanumeric pin numbers. The decal determines the number of pins in the part.

Creation of a New Part Type

You can create new part types for a large selection of component types.

There are five different component types that require specific instructions to create the part type:

- The component is an electrical part, and the logical and physical pin numbering is the same--both numeric, or both alphanumeric. For detailed instructions, see “Creating a Part Type with Identical Schematic and Layout Pin Numbering”.
- The component is an electrical part, and the logical and physical pin numbering is different--the logical is alphanumeric and the physical is numeric. For detailed instructions, see “Creating a Part Type with Different Schematic and Layout Pin Numbering”.
- The component is a connector. For detailed instructions, see “Creating a Connector Part Type”.
- The component will not be ECO registered. For detailed instructions, see “Creating a Non-ECO-Registered Part Type”.
- The component is a non-electrical part, such as a mounting screw or locating post, that is not connected to any net. For detailed instructions, see “Creating a Non-Electrical Part Type”.

Creating a Part Type with Identical Schematic and Layout Pin Numbering

You can create a part type that uses the same logical and physical pin numbering. You can choose either alphanumeric-to-alphanumeric or numeric-to-numeric for the schematic-to-layout pin numbering.

Restrictions and Limitations

- Your PCB decal(s) must exist to use this procedure. For more information, see “Creation of a New Decal”.

Procedure

1. Click the **File > Library** menu item.
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 on page 1590 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 on page 1593, assign one or more decals to the part.

> **Note:**
> If your logic symbol uses alphanumeric pin numbering and the PCB decal uses numeric pin numbering, you must use the **Pin Mapping** tab to map the alphanumeric symbol pins to the numeric decal pins. For instructions, see Mapping Alphanumeric Pin Numbers to Numeric Decals.

7. On the **Gates** tab, add at least one gate. (Add multiple gates if you want to be able to swap them.) For instructions, see "Assigning CAE Decals to Gates".

8. On the Pins tab on page 1596, you must fill out the Pin Group and Name columns; the other columns are optional. For instructions, see "Editing Pins Table Data".

9. On the Attributes tab on page 1584, add attributes as necessary. For instructions, see "Adding Attributes to a Part Type" on page 243.

10. Click **OK**. For instructions, see "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

Creation of a New Part Type

Creating a Part Type with Different Schematic and Layout Pin Numbering

You can create a part type that uses alphanumeric numbering for the logical pins and numeric numbering for the physical pins. Use the **Pin Mapping** tab to map the alphanumeric symbol pins to the numeric decal pins.

Restrictions and Limitations

- Your PCB decal(s) must exist. For more information, see "Creation of a New Decal".

Procedure

1. Click the **File > Library** menu item.

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 on page 1590 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 on page 1593, assign one or more numeric decals to the part. For instructions, see “Mapping Alphanumeric Pin Numbers to Numeric Decals”.

8. On the Gates tab on page 1588, add at least one gate. (Add multiple gates if you want to be able to swap them.) For instructions, see “Assigning CAE Decals to Gates”.

9. On the Pins tab on page 1596:

   a. The pin numbering in the part type needs to match the pin numbering style used in the logical schematic symbol, so change the pin numbers to alphanumerics to match the schematic alphanumerics. For instructions, see “Renumbering Pins in the Pins Table”.

   b. Fill out the Pin Group and Name columns; the other columns are optional. For instructions, see “Editing Pins Table Data”.

10. On the Attributes tab on page 1584, add attributes as necessary. For instructions, see “Adding Attributes to a Part Type” on page 243.

11. On the Pin Mapping tab on page 1600, map the unmapped logical pins (alphanumeric) to the physical decal pins (numeric). For instructions, see “Mapping Alphanumeric Pin Numbers to Numeric Decals”.

12. Click **OK**. For more information, see “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

Creation of a New Part Type

Creating a Non-ECO-Registered Part Type

You can create a non-ECO-registered part type. A non-ECO-registered part is a part—such as a mounting-hole—that does not transfer during forward or backward annotation.

Procedure

1. Click the **File > Library** menu item.

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 on page 1590 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 on page 1593, assign one or more decals to the part. For instructions, see “Mapping Alphanumeric Pin Numbers to Numeric Decals”.

7. On the Gates tab on page 1588, add at least one gate. (Add multiple gates if you want to be able to swap them.) For instructions, see “Assigning CAE Decals to Gates”.

8. On the Pins tab on page 1596, you must fill out the Pin Group and Name columns; the other columns are optional. For instructions, see "Editing Pins Table Data".

9. On the Attributes tab on page 1584, add attributes as necessary. For instructions, see "Adding Attributes to a Part Type" on page 243.

10. Click **OK**. For more information, see "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

Creation of a New Part Type

Creating a Non-Electrical Part Type

You can create a non-electrical part type such as a mounting screw or locating post and add it to a parts list.

Restrictions and Limitations

- Your PCB decal(s) must exist to use this procedure. For more information, see "Creation of a New Decal".

Procedure

1. Click the **File > Library** menu item.

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 on page 1590 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 on page 1593, assign a non-electrical (no pin terminals) decal to the part. For instructions, see "Mapping Alphanumeric Pin Numbers to Numeric Decals".

7. On the Attributes tab on page 1584, add attributes as necessary. For instructions, see "Adding Attributes to a Part Type" on page 243.

8. Click **OK**. For more information, see "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

Creation of a New Part Type

Creating a Connector Part Type

You can create a connector part type that enables a feature in SailWind Logic to split a connector into multiple pin decals, making each pin its own gate.

Restrictions and Limitations

- Your PCB decal(s) must exist to use this procedure. For more information, see "Creation of a New Decal".

Procedure

1. Click the **File > Library** menu item.
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 on page 1590of 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 on page 1593, assign one or more decals to the part. For instructions, see "Mapping Alphanumeric Pin Numbers to Numeric Decals".
8. On the Pins tab on page 1596, you must fill out the Pin Group and Name columns; the other columns are optional. For instructions, see "Editing Pins Table Data".
9. On the Attributes tab on page 1584, add attributes as necessary. For instructions, see "Adding Attributes to a Part Type" on page 243.
10. On the Connector tab on page 1586, add pin decals to be used in the schematic. For instructions, see "Assigning Special Symbols to a Connector".
11. Click **OK**. For more information, see "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

Creation of a New Part Type

Modification of Part Types

Read the sections that follow to learn more about the processes used to modify part types and to check them for errors, as well as how to add and modify part type attributes.

Modifying a Part Type

You can modify part types to accurately represent the requirements of your designs.

Procedure

1. Click the **File > Library** menu item.
2. Select a Library, and then click the **Parts** button.
3. Select a part, and then click the **Edit** button.
4. When you have finished modifying the properties, click **OK** to save the changes.

Assigning PCB Decals to Part Types

Filter your library search for decals and assign a decal and then alternates if desired.

Procedure

1. Click the **File > Library** menu item.
2. In the Library Manager Dialog Box, select a Library, and then click the **Parts** button.
3. Select a part, and then click the **Edit** button.
4. On the PCB Decals tab on page 1593, 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 on page 155. Type asterisk (*) in the Filter box to display all decals.

c. Type a number in the Pin Count box, and then click **Apply**.

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.

5. 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 the Assign New PCB Decal dialog box, type the name and then click **OK**.

      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.

6. 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.

---

7. When you have finished modifying the properties, click **OK** to save the changes.

Related Topics

Creation of a New Part Type

Library Content and the Search Order

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. Click the **File > Library** menu item.
2. In the Library Manager Dialog Box, select a Library, and then click the **Parts** button.
3. Select a part, and then click the **Edit** button.
4. In the Part Information for Part dialog box, select the Gates tab on page 1588.
5. Select the cell under CAE Decal 1 and click the **Edit** button or double-click the cell under CAE Decal 1.
6. Type a decal name in the CAE Decal box or click the ... (browse) button to search for a decal from a library.

7. 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 on page 155. 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.

8. To assign one or more decals:

   - Select a decal in the Unassigned decals list and click **Assign**, or 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**.

   - You can assign up to four CAE decals to a part. Assigned decals must have the same number of pins.

9. 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.

10. Assign Gate pins on the Pins tab on page 1596.

11. When you have finished modifying the properties, click **OK** to save the changes.

Related Topics

Creation of a New Part Type

Modification of Gates in Parts in the Library

Modification of Gates in Parts in the Library

You can modify parts in the library using the Library Manager. Information you can modify includes location, decal, connector, gates, alphanumerics, signal pins, and attributes.

For each gate, you can type the CAE Decal name; the name of the logic symbol that is used to display the part in the schematic. Alternate decal assignments must have the same number of pins. You can define one primary and three alternate decals for each gate.

When at least one decal is assigned to a part, you can type or modify its gate information. This includes swap enabling or disabling for gates within a part or between similar parts. This information lets SailWind Layout know which gates it can substitute for connection length minimization after placement.

You can also swap pins within gates to uncross connections and facilitate routing. In both gate and pin swapping, you assign a number to the gate or pin in the Gates tab. Pins with like numbers can swap within a gate. Gates with like numbers can swap within the part, or to other similar parts. A one (1) indicates the gate is swappable with gates of the same part type in the PCB design database. If a part contains more than one type of swappable gate, then identify the second type with the number 2, the third type with 3, and so on. Zero (0) indicates that no swapping can occur.

For more information, see the Creating and Modifying Part Types on page 229topic.

Modifying the Pins Table Information

The Pins table contains the mapping of the pins data and allows you to add, edit, and renumber the pins.

Adding a Series of Pins to the Pins Table

You can use the Add Pins dialog box to add a series of pins to the Pins table instead of adding one pin at a time.

Procedure

1. Click the **File > Library** menu item.
2. In the Library Manager dialog box on page 1449, select a Library, and then click the **Parts** button.
3. Select a part, and then click the **Edit** button.
4. In the Part Information for Part dialog box, select the Pins tab on page 1596.
5. Click the **Add Pins** button.
6. In the Add Pins Dialog Box, type the number of pins to add in the Number of pins box. Total pins for the part can not exceed 32,767 pins.
7. 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. 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.

   Note:
   If you type alphanumerics and the decal uses numerics, you must use the **Pin Mapping** tab to map the alphanumerics onto the decal.

8. In the Increment options area, choose what to increment by clicking either the Increment prefix or Increment suffix option.
9. 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.
10. If using alphanumerics, you can select the Use JEDEC pin numbering check box to ensure that legal alphanumeric values are used.
11. When you have finished modifying the properties, click **OK** to save the changes.
12. Click **OK** to close the Part Information for Part dialog box.

Related Topics

Creation of 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 **File > Library** menu item.
2. In the Library Manager dialog box on page 1449, select a Library, and then click the **Parts** button.
3. Select a part, and then click the **Edit** button.
4. In the Part Information for Part dialog box, select the Pins tab on page 1596.
5. 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.
6. Click the Number cell and type a pin number for the pin. Typically, the pin number should match those used in the PCB decal. For example, alphanumeric to alphanumeric. Pin numbers can be either numeric or alphanumeric.

   Note:
   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. If your logical schematic symbol uses alphanumeric pin numbering, you can continue to keep numeric PCB decals and use the **Pin Mapping** tab to overlay different alphanumeric pin numbers onto the numeric PCB decal pin numbers.

7. 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.

   Note:
   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.

8. Click the Type cell and in the list, choose a pin type. The type column is only used with gate pins.
9. Click the Swap cell and type a swap number, or use the up/down arrows. 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.
10. 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.

Note:
When editing pin data for connectors, only the Pin Group and Number columns are relevant. Data entered in other columns is rejected. Connectors do not have gates, so the Pin Group column just indicates whether a pin is a connector pin or an unused pin.

11. When you have finished modifying the properties, click **OK** to save the changes.

Related Topics

Creation of a New Part Type

Renumbering Pins in the Pins Table

You can change the numbering of pins in the **Pin** tab table.

Procedure

1. Click the **File > Library** menu item.
2. In the Library Manager dialog box on page 1449, select a Library, and then click the **Parts** button.
3. Select a part, and then click the **Edit** button.
4. In the Part Information for Part dialog box, select the Pins tab on page 1596.
5. Select one or more cells in the Number column.
6. Click the **Renumber** button. In the Renumber Pins Dialog Box, the Number of pins box displays the number of pins selected for renumbering.
7. 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. 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.
8. In the Increment options area, choose what to increment by clicking either the Increment prefix or Increment suffix option.
9. 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.
10. If using alphanumerics, you can select the “Use JEDEC pin numbering” check box to ensure that legal alphanumeric values are used.
11. When you have finished modifying the properties, click **OK** to save the changes.

Related Topics

Creation of a New Part Type

Part Type Error Checking

Validation takes place when you click **Check Part**, **OK**, **Save As**, or when you click a different tab.

The validation process 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

Creation of a New Part Type

Adding and Modifying Part Type Attributes

You can add part type attributes to part types. You can also modify existing part type attributes to accommodate changes and additions to your design data.

Adding Attributes to a Part Type

You can add part information (attributes) to a part type such as manufacturer, part number, and so forth.

Procedure

1. Click the **File > Library** menu item.
2. In the Library Manager dialog box on page 1449, select a Library, and then click the **Parts** button.
3. Select a part, and then click the **Edit** button.
4. In the Part Information for Part dialog box, click the Attributes tab on page 1584.
5. Click **Add** to add a new attribute row.
6. Type the new attribute name and value. Alternatively, you can click the **Browse Lib Attr** button to select an existing attribute from a list.
7. Repeat Steps 2 and 3 to add additional attributes.
8. When you have finished modifying the properties, click **OK** to save the changes.

Adding Height Information to Library Parts

You can add height information to part types. SailWind Layout uses height information to prevent layout designers from placing components in height-constrained areas. SailWind Layout also uses the height information when exporting a design to a 3-dimensional modeling application.

Procedure

1. Click the **File > Library** menu item.
2. In the Library Manager dialog box on page 1449, select a Library, and then click the **Parts** button.
3. Select a part, and then click the **Edit** button.
4. In the Part Information for Part dialog box, click the Attributes tab on page 1584.
5. Click **Add** to add a new attribute row.
6. For the new attribute name type Geometry.Height. Alternatively, you can click the **Browse Lib Attr** button to select Geometry.Height from a list.

7. Type the value in current design units.

8. When you have finished modifying the properties, click **OK** to save the changes.

Related Topics

Adding Height Information to Design Components and Jumpers

Restricting Heights on Component Layers

Restricting Heights in Areas of Component Layers

Defining Default Attributes for New Parts

You can save a default set of attributes for use with each new part.

Note:
Attribute names but not values are saved in the default attribute list.

Procedure

1. Click the **File > Library** menu item.

2. In the Library Manager dialog box on page 1449, select a Library, and then click the **Parts** button.

3. Select a part, and then click the **Edit** button.

4. In the Part Information for Part dialog box, click the Attributes tab on page 1584.

5. Once the list of attributes is correct, click **Save As Default**.

6. When you have finished modifying the properties, click **OK** to save the changes.

Results

The default attribute list is saved to *C:\<install_folder>\<version>\Settings\defaultattribute.txt*, which is shared between SailWind Logic and SailWind Layout.

Related Topics

Library Attribute Management

Creation of 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. Doing so allows you to use any of the special symbols as alternate pins of the connector.

Instead of displaying it as a single symbol, SailWind Layout breaks up the connector 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. Click the **File > Library** menu item.
2. In the Library Manager dialog box on page 1449, select a Library, and then click the **Parts** button.
3. Select a part, and then click the **Edit** button.
4. In the Part Information for Part dialog box, click the Connector tab on page 1586.
5. To filter the Gate Decals list, select a library in the Library list, type wildcards on page 155 in the Filter box, and then click **Apply**. To display all items, type asterisk * and click **Apply**.
6. Select the item in the list.
7. Click **OK**.
8. When you have finished modifying the properties, click **OK** to save the changes.

Related Topics

Creating a Connector Part Type

Creation of a New Part Type

Mapping Alphanumeric Pin Numbers to Numeric Decals

You can complete the pin mapping in one of several ways.

Procedure

1. Click the **File > Library** menu item.
2. In the Library Manager dialog box on page 1449, select a Library, and then click the **Parts** button.
3. Select a part, and then click the **Edit** button.
4. In the Part Information for Part dialog box, select the Pin Mapping tab on page 1600.
5. In the decal list above the preview window, select the assigned decal to which you want to map alphanumeric pins.
6. 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**.

   CAUTION:
   If there are any errors in the pin numbering, correct them by editing the table on the **Pins** tab. Do not attempt to renumber pins in the table on the **Pin Mapping** tab.

- **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.

> **Note:**
> **Copy Map** and **Paste Map** only work with the whole pin mapping table and not selective rows.

7. Repeat as necessary.

8. When you have finished modifying the properties, click **OK** to save the changes.

Related Topics

Creating a Part Type with Different Schematic and Layout Pin Numbering

Creation of 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 the **Save to Library** popup menu item.

3. In the Save Part Types and Decals to Library Dialog Box, 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 New Library

Import and Export of Libraries

Chapter 9
Starting a New Design

Typically, you start a new design by creating a new design file, and then importing a netlist created by a schematic tool such as SailWind Logic. But if the design you are planning is simple and does not require a schematic, you can create the design all within SailWind Layout itself. Designs you create in this way are referred to as “layout-driven” designs.

Creating a New Design File

Use the New command to create a new design file.

Procedure

1. Open SailWind Layout and click **New**. (If you already have a design open in SailWind Layout, click the **File > New** menu item).

2. In the Set Start-up File Dialog Box, select the start-up file. 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 start-up for all new design files, select the Don't Display Again check box.

4. Click **OK**.

Related Topics

Start-up Files

Import of a Schematic Design Netlist

The process of laying out a design in SailWind 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 netlist file import process, all the parts are sourced from the library, and their decals are stacked at the origin, ready for placement.

CAUTION:
For a netlist import to succeed, all 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 in an existing library.

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 Design by Manually Importing the SailWind Logic Netlist" on page 271
- Creating a New PCB Design from an OrCAD Netlist

Cross-Probing

You can cross-probe with SailWind Logic to simplify importing the netlist and placing the parts.

The procedure you use to cross-probe on page 1819 depends on which schematic tool you are using. For product-specific instructions, see Cross-Probe Between SailWind Products in the *SailWind Logic Guide*.

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 includes the ability to import a netlist created in ORCAD.

Procedure

1. Change the filename extension of the OrCAD netlist from *.net* to *.asc*. SailWind Layout only imports netlist files with *.asc* extensions.
2. Open the netlist file in a text editor.
3. In SailWind Layout, click the **File > Reports** menu item.
4. In the Reports Dialog Box, select "PowerPCB V3.0 Format Netlist", and click **OK**.
5. In the report that is displayed, copy the header line, which will look something like this:

   !PADS-POWERPCB-V3.0-MILS! DESIGN DATABASE ASCII FILE 2.0
6. Paste the header line in the OrCAD netlist, replacing the original header, which will look something like this:

   PADS-PCB
7. Replace the software version number (V3.0) in the new header with the number of the software version you are running.
8. Save and close the netlist file.
9. In SailWind Layout, click the **File > Import** menu item.
10. In the File Import dialog box, browse for and select the OrCAD netlist *.asc* file, and then click **Open**.
11. 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 in an existing library. (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. Error messages of the type “*Bad *PART* ascii data line format” may indicate that an invalid character has been found in the part name. The characters invalid in part names are:

. , : ^ [ ] $ * { } ( ) @ ? = \ <space>

d. When all the errors have been resolved, repeat this procedure.

Results

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 > Disperse Components** menu item to spread out the components.

Layout-Driven Designs

Most PCB designs are schematic-driven, meaning designers create them in schematic software and the components, connections, and constraints then pass to SailWind Layout. After they are in SailWind Layout, according to those constraints, layout designers place components, making their connections permanent with copper traces.

But if the design is simple, and you are able to skip creating the schematic(s), you can add components, create connections, and create constraints from the ECO toolbar in SailWind Layout. The ECO toolbar contains all the tools that are design-altering.

If your design is schematic-driven, it must remain synchronized/in parallel between the schematic and layout tool. You can record design modifications made by tools on the ECO toolbar in an ECO file that you can use to synchronize the schematic design with the layout design.

But if you create a layout-driven design, and you have no schematic to synchronize with, there is no need to maintain a *.eco* file. When you open the ECO Toolbar and are prompted with the ECO Options dialog box, you can safely clear the “Write ECO file” check box and then ignore all the settings of the ECO Options dialog box.

Using the tools on the ECO Toolbar, you can:

- Add and delete components
- Add and name connections/nets
- Rename and delete nets
- Rename component reference designators
- Change components

- Swap pins and gates
- Assign constraints/design rules

For more information on the ECO Toolbar, see “ECO Mode Operations (Layout-Driven Design Tools)”.

Chapter 10
File Operations

Read the topics that follow to learn more about various file operations.

Start-up Files

Use start-up files to save color settings, the attribute dictionary, and other universal parameters and then apply them to every new *.pcb* file. To set and apply default settings use the System Default (*default.asc*) start-up file.

For information on creating start-up files, see the Specifying the Start-up File on page 255 and Creating Start-up Files topics.

The provided start-up files listed in the table below show typical values for different technologies, and do not represent any specific technology.

Table 65. Start-up Files

File Item	System Default Startup File	Chip on Board Startup File	Low Temp Co-fired Ceramic Startup File	MCM-L Startup File
Filename	default.asc	cob-startup.stp	ltcc-startup.stp	mcml-startup.stp
Units	mils	mils	mils	metric (mm)
Design grid	100x100	5x5	1x1	0.005
Via grid	25x25	5x5	1x1	0.005
Layers	2	2	13	2
Trace Width	12	3	4	0.05
Vias (top/inner/bottom)				
Standard	55/55/55	30/30/30	55/55/55	0.2/0.2/0.2
Micro	None	None	6/4/6	None

Table 65. Start-up Files (continued)

File Item	System Default Startup File	Chip on Board Startup File	Low Temp Co-fired Ceramic Startup File	MCM-L Startup File
Clearances				
Trace-Trace	6	3	5	0.35
Trace-Via	6	3	4	0.35
Via-Via	6	3	4	0.35

Unsupported Start-up Files

The start-up file is an ASCII file with an *.stp* file extension.

CAUTION:
This method of creating a start-up file is unsupported.

By design, items you can include in the start-up file are restricted by the Start-up File Output Dialog Box. Nevertheless, it is possible to export an unrestricted ASCII file using the ASCII Output dialog box. Then you could rename the extension from *.asc* to *.stp* and place the file in the *C:\<install_folder>\<version>\Settings* folder where it would be found and used as a start-up file. For example, if you repeatedly design products using the same board outline, you could use this method to export your board outline into the start-up file. Then you would start with your board outline when beginning a new design.

Related Topics

Creating Start-up Files

Specifying the Start-up File

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 saved in a start-up file.

Procedure

1. In the current design, specify the options and preferences you want to include in the start-up file.
2. Click the **File > Save As Start-up File** menu item.
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. If you want to select all the check boxes, click **Select All**.

- **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.

  For related information, see "Modifying the Default Attribute Dictionary".

5. In the Units list, select the units you want to use in the start-up file. 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**.

Results

The start-up file is written to the *C:\<install_folder>\<version>\Settings* folder. Start-up files have an *.stp* file extension.

Related Topics

Specifying the Start-up File

Start-up Files

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.

Use start-up files only when creating new designs; specifying a new start-up file does not affect existing designs.

Procedure

1. Click the **File > Set Start-up File** menu item.
2. In the Set Start-up File Dialog Box, select the start-up file and click **OK**. If you plan 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

Opening Files

You can open current or older design versions and reuse files. In some circumstances, when you open a file, SailWind Layout might convert the data to current formats. If another user has the file open, it is locked to any edits.

These file types can be opened:

- Design files, *.pcb — PADS Layout binary format
- Reuse files, *.reu — A physical design reuse type, known as the reuse definition on page 1854
- Old binary files, *.job — PADS Perform V6 and PADS Work V7 format *.job* files

When you open a file in SailWind Layout, one or more conversions may take place to update your data to current data format. For more information, see File Open Conversions.

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 or later. 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. Click the **File > Open** menu item.
2. In the File Open dialog box, select the file and click **Open**.

Related Topics

File Open Conversions

File Open Conversions

When you open a file, some data conversions may take place to update your data to current data format.

Copper Moved

In some previous versions of PADS Layout you could create copper that appeared on all layers (Layer 0). SailWind Layout now requires you to create copper on a specific layer. If you open a file containing copper created for all layers, the copper is placed on Layer 1 and a message appears indicating which items were moved.

Attributes Converted

When you open a design, the Attribute Dictionary is loaded. You can change the default attributes that are imported by changing the default attribute dictionary.

For more information, see Default and Other Attribute Properties and Usage.

If you open a design created with a version prior to PowerPCB version 3.0 any part type attributes found are converted to new attributes, which are added to the Attribute Dictionary (if necessary), and applied to the items on which they were found. For example, if you assigned value and tolerance in your existing design, they are converted to Value and Tolerance attributes and then added to the appropriate objects.

Table 66. Items Converted to Attributes in Version 3

Attribute Name	Created When
Value	A value is found appended to a part type name.
Tolerance	A tolerance is found appended to a part type name.
ASSEMBLY_OPTIONS	Assembly Variants exist in the design.
DFT.Nail Count Per Net	The nail count per net is greater than one, the default. The nail count may be greater than one if the Insert Multiple Test Points Per Net check box is selected in the Audit Rules tab of the DFT Audit dialog box.

Labels Converted

If you open a design created with a version prior to PowerPCB version 3.0, the reference designators and part types are converted to current labels. The location, orientation, and color of the labels are preserved if they were on the Top or Bottom layers. The default height and width are used, mirroring is turned off, and labels are placed on the Top layer. Also, visibility is set to Value.

For information on changing these settings see Modifying Part Label Properties.

Differential Pair Trace Width and Gap Converted

Width and gap for differential pair traces can be set for all layers or per layer. When you open a design, the width is set for all layers and is the larger of the recommended widths for each member of the differential pair. The gap assigned to the differential pair is set for all layers.

For information on setting the width and gap per layer, see Creating Differential Pair Design Rules.

Replacing Missing Fonts

Text and labels in your designs use stroke font or the system fonts installed on your system. When you open a design that uses fonts or characters that are not installed on your system, the Font Replacement dialog box—in which you can specify how the missing fonts should be replaced—opens automatically.

You can replace missing fonts automatically or manually, or you can skip the replacement process for fonts you identify. When you choose to replace fonts manually, you are asked to confirm your font selections before the replacement process initiates. If you choose to skip font replacement, you do not confirm your selections to start the replacement process, or if you cancel the replacement, the loading of the design for display on your system is cancelled, and the original design file is preserved.

Restrictions and Limitations

Type 1 fonts are not supported.

Procedure

1. In the Font Replacement Dialog Box, set the replacement mode and font for each missing font, as follows:
    a. To accept the replacement font named in the Replace with font field, select Automatic from the Mode drop-down list.
    b. To choose a different replacement font, select Manual from the Mode drop-down list, then select the replacement font from the Replace with font drop-down list.
    c. To keep the existing font setting, select Skip from the Mode drop-down list. Clearances for text strings will be preserved, and objects using the missing font will appear in the display as empty boxes.
2. When you have made the desired settings for all the missing fonts, click **OK**.

Related Topics

Finding Fonts

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 *.pcb* design file, a schematic file, libraries, and any additional files or folders you want. You choose what to archive; all fields are optional.

Procedure

1. Open the design you want to archive.
2. Click the **File > Archive** menu item.
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 have indicated is created.

If you chose to compress the files using the zip format, the archive folder contains only the *.zip* file, which has 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 (24-hour) time, MM is the minute, and SS is the second of the exact time you created the file.

If you chose to create a PDF, the file is created using the design name and placed in the archive folder.

Chapter 11
Setting up the Design Environment

Read the following topics to learn more about the procedures for setting up some initial non-electrical portions of the design.

Creating a Board Outline

Use the **Board Outline and Cut Out** button on the Drafting Toolbar to create a board outline.

Restrictions and Limitations

You can create only one board outline per design.

Procedure

1. On the Drafting Toolbar, click the **Board Outline and Cut Out** button.
2. Right-click and click the shape of your board outline.
3. Draw your outline. For specific shape instructions, see "Creating a Rectangle Drafting Object" on page 624, "Creating a Circle Drafting Object" on page 624, "Creating a Polygon or Path Drafting Object". on page 623
4. To change the color of the board outline, click the **Setup > Display Colors** menu item. In the Other area, set the color of the Board Outline tile. Any board cut outs you create will be the same color.

Related Topics

Importing a Board Outline and Cut Out from AutoCAD

Board Outline Reuse

Creating a Board Cut Out

Creating a Board Cut Out

Use the **Board Outline and Cut Out** button on the Drafting Toolbar to create a board cut out. You can create only one board outline per design, so use the same command and creation process for creating cut outs.

Restrictions and Limitations

You cannot create a notch using a board cut out. To create a notch, edit the board outline instead.

Prerequisites

The board outline on page 261 must exist before you can create a cut out.

Procedure

1. On the Drafting Toolbar, click the **Board Outline and Cut Out** button. Since 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. For specific shape instructions, see“Creating a Rectangle Drafting Object” on page 624, “Creating a Circle Drafting Object”, “Creating a Polygon or Path Drafting Object”. on page 623

4. To change the color of the board cut out, click the **Setup > Display Colors** menu item. In the Other area, set the color of the Board Outline tile.

Results

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 a DXF file created in AutoCAD provided you meet certain preliminary requirements in the AutoCAD setup.

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.

   Only one polyline object can be used for the board outline since SailWind 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 SailWind 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 SailWind Layout design file.

Results

- Did the board outline or cutout import as a 2D line? If so, it was not 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 is not closed? In AutoCAD, edit the polyline and close it.

Related Topics

Creating a Board Outline

Creating a Board Cut Out

Board Outline Reuse

Board Outline Reuse

If you need to create more than one board with the same outline, there are several ways to reuse the board outline in a new design depending on the situation.

Copying and Pasting the Board Outline

If desired, you can copy and paste the outline from one design into the next.

Procedure

1. Open the design with the Board Outline that you want to copy.
2. With nothing selected, right-click and click the **Select Board Outline** popup menu item.
3. Shift-click the board outline.
4. Click the **Edit > Copy** menu item.
5. Open the new design.
6. Click the **Edit > Paste** menu item. The board outline is attached to the pointer.
7. Position the Board Outline with the pointer and click to place it.

Related Topics

Board Outline Reuse

Adding the Board Outline to the Library

You can save the board outline to the library and later add it to other designs.

Procedure

1. Open the design with the Board Outline that you want to copy.
2. With nothing selected, right-click and click the **Select Board Outline** popup menu item.
3. Shift-click the board outline.
4. Right-click and click the **Save to Library** popup menu item.
5. In the Save Drafting Item to Library dialog box on page 1393, choose a library and then type the name of the item.
6. Click **OK**.

7. Reuse the board in a future design.

   a. Open the next design.

   b. On the Drafting Toolbar, click the **From Library** button.

   c. In the Get Drafting Item from Library dialog box on page 1393, locate the board outline you save to a library in a previous step and select it in the Drafting Items list.

   d. Click **OK**. The board outline is attached to the pointer.

   e. Position the Board Outline with the pointer and click to place it.

Related Topics

Board Outline Reuse

Exporting the Board Outline into an ASCII File

You can share your board outline with another user who does not share your libraries. You can export the board outline into an ASCII file for import by the other user.

Procedure

1. Create on page 261 a board outline. If you have a design with a board outline, you can use this design; remove any extra line items so they are not included in the export in the next step.

2. Export on page 291 to an ASCII file. In the ASCII Output Dialog Box, select the Lines check box only.

3. Import on page 290 the saved ASCII file into a new design.

Related Topics

Board Outline Reuse

Adding the Board Outline to an Unsupported Start-up File

If you are consistently using the same board outline for your designs, add the board outline to a startup file and when you start a new design, the board outline is there by default. Export the board outline in an ASCII file along with other startup features, rename the file extension to *.stp*, and place the file in the start-up file directory for use as a start-up file.

For more information, see Unsupported Start-up Files in “Start-up Files”.

Procedure

1. First, read the “Creating Start-up Files on page 254” topic.

2. Follow the steps in “Exporting the Board Outline into an ASCII File” to create an ASCII file with the board outline and any other line items, stackup, rules, or other defaults you might want in your start-up file.

3. Rename the *.asc* file extension to *.stp* and place the file in the *C:\<install_folder>\<version>\Settings* folder.

4. Follow the instructions in "Specifying the Start-up File" to use the start-up file for future designs.

Related Topics

Board Outline Reuse

Moving a Board Cut Out

When you move a board cut out, avoid using the "Move by Origin" move preference.

Procedure

1. Check the Move Preference:

   a. Click the **Tools > Options** menu item.

   b. In the Options dialog box, click the Design category on page 1503.

   c. In the Move preference area, select either "Move by cursor location" or "Move by midpoint". If the move preference is set to "Move by origin", the cut out may snap off screen since the pointer snaps to the origin of the board outline and not the cut out.

   d. Click **OK**.

2. With nothing selected, right-click and click the **Select Board Outline** popup menu item.

3. Shift-click the cut out to select the whole shape.

4. Right-click and click the **Move** popup menu item.

Results

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 Topics

Creating a Board Cut Out

Creating and Adding Board Fiducials

Board fiducials are pads on the outer layers of the board surrounded by adequate soldermask clearances to allow for better visual contrast and increased recognition. Various steps of the board fabrication and assembly processes use board fiducials for positioning and alignment, most importantly during the automated placement of components. Automated vision systems on specialized processing equipment use fiducials as visual targets to align and calibrate the machines.

Procedure

1. Create a new decal on page 160 of a single pad stack on page 185 for the fiducial:
    a. Define a pad on the top (mounted) layer and no pad on the inner or bottom (opposite) layers.
    b. Add to this a soldermask opening two or three times the size of the pad on the top soldermask layer.
    c. Next, add a pad the same size as the fiducial to the top assembly layer to help viewers of the assembly drawing orient the location of the components.
    d. You can create a keepout around the fiducial to assure no other design objects on adjacent layers will interfere with the visual recognition of the fiducial by the automated systems.
2. Create a non-ECO Registered Part on page 232 for this new decal.
3. Add the new part using the Add Component button on the ECO Toolbar on page 821.
4. Place the fiducials around the edge of the board inside the board outline.

Examples

Fiducial

This example will provide the information needed to create a fiducial.

1. Create a new decal of a single pad stack for the fiducial using these suggested dimensions:

Table 67. Usage Example - Fiducial

Layer	Shape	Size	Description
Top (Mounted)	Round	40 mils	Top pad
Solder Mask Top	Round	80 mils	Solder mask to void
Assembly Top	Round	40 mils	Assembly ref. image top
All Layers (Keepout)	Round	80 mils	Component/via/routing
Layer 20	Circle	80 mils	Courtyard outline

1. Create a non-ECO Registered Part for this decal and name it FIDUCIAL40 or some other descriptive name of your choice.
2. Add the new part using the **Add Component** button on the ECO Toolbar.
3. Place two fiducials at the extremities of the board in diagonally opposed corners 1/4 inch inside the board edges

Creating and Adding Board Mounting Holes

Board mounting holes typically provided places to secure and support the printed circuit board in the device enclosure. Mounting holes usually fit with ANSI and ISO standard mounting hardware. Depending upon the specific requirements of the design, mounting holes may be plated or non-plated.

Procedure

1. Create a new decal on page 232 of a single pad stack on page 185 for the mounting hole:
   a. Define a pad on the top (mounted) layer and the bottom (opposite) layer.
   b. Define a pad on the inner (signal) layer and add a plane anti-pad definition to the pad.
   c. Add soldermask openings for the pads on the top and bottom soldermask layers.
   d. Next, add pads to the top and bottom assembly layer to help viewers of the assembly drawing find the location of the mounting holes.
   e. You can create a keepout around the mounting hole to assure that there is adequate clearance for the fastener and any required tools.
   f. Specify the drill size of the through hole.
   g. If the mounting hole is to be plated, check the plated check box; if the mounting hole is to be unplated, leave the plated check box unchecked.
2. Create a non-ECO Registered Part on page 232 for this new decal.
3. Add the new part using the on page 821**Add Component** button on the ECO Toolbar.
4. Place the mounting holes at the required locations within the board outline.

Examples

Mounting Hole (Plated)

This example will provide the information needed to create a plated mounting hole for a #6-32 ANSI screw (.157” hole).

1. Create a new decal of a single pad stack for the mounting hole using these suggested dimensions:

Table 68. Usage Example - Mounting Hole (Plated)

Layer	Shape	Size	Description
Top (Mounted)	Round	280 mils	Top pad
Bottom (Opposite)	Round	280 mils	Bottom pad
Inner (Signal)	Round	208 mils	Inner pad
	Round	208 mils	Inner anti-pad (clearance)

Table 68. Usage Example - Mounting Hole (Plated) (continued)

Layer	Shape	Size	Description
Solder Mask Top	Round	280 mils	Solder mask top void
Solder Mask Bottom	Round	280 mils	Solder mask bottom void
Assembly Top	Round	280 mils	Assembly ref. image top
Assembly Bottom	Round	280 mils	Assembly ref. image bottom
Layer 20	Circle	300 mils	Courtyard outline
All Layers (Drill hole)	Round	157 mils	Plated hole
All Layers (Keepout)	Round	354 mils	Component/via/routing

1. Create a non-ECO Registered Part for this decal and name it MTGP-6 or some other descriptive name of your choice.

2. Add the new part using the **Add Component** button on the ECO Toolbar.

3. Place the mounting holes at the required locations within the board outline.

Mounting Hole (Non-Plated)

This example will provide the information needed to create a non-plated mounting hole for a #6-32 ANSI screw (.157” hole)

1. Create a new decal of a single pad stack for the mounting hole using these suggested dimensions:

Table 69. Usage Example - Mounting Hole (Non-Plated)

Layer	Shape	Size	Description
Top (Mounted)	Round	40 mils	Top pad (visual ref.)
Bottom (Opposite)	Round	40 mils	Bottom pad (visual ref.)
Inner (Signal)	Round	40 mils	Inner pad (visual ref.)
	Round	205 mils	Inner anti-pad (clearance)
Solder Mask Top	Round	157 mils	Solder mask top void
Solder Mask Bottom	Round	157 mils	Solder mask bottom void
Assembly Top	Round	157 mils	Assembly ref. image top
Assembly Bottom	Round	157 mils	Assembly ref. image bottom

Table 69. Usage Example - Mounting Hole (Non-Plated) (continued)

Layer	Shape	Size	Description
Layer 20	Circle	300 mils	Courtyard outline
All Layers (Drill hole)	Round	157 mils	Non-Plated hole
All Layers (Keepout)	Round	354 mils	Component/via/routing

1. Create a non-ECO Registered Part for this decal and name it MTGNP-6 or some other descriptive name of your choice.

2. Add the new part using the **Add Component** button on the ECO Toolbar.

3. Place the mounting holes at the required locations within the board outline.

Chapter 12
Working with SailWind Logic

Read the topics that follow to learn more about creating a PCB layout from a SailWind Logic schematic, cross-probing, and forward- and backward-annotation of design changes.

Creating a New PCB Design by Manually Importing the SailWind Logic Netlist

Use this method if you do not have both SailWind Logic and SailWind Layout on your computer. This manual method requires you to manually export the netlist from SailWind Logic and then import it into SailWind Layout.

Alternatively, if you have both SailWind Logic and SailWind Layout on your computer, use the SailWind Layout Link within SailWind Logic for a simpler method. For more information, see Creating a New PCB Design Using the Automated SailWind Layout Link in the *SailWind Logic Guide*.

CAUTION:
Forward annotation of changes to an existing design requires a different process. See "Forward-Annotation of Design Changes from SailWind Logic".

Prerequisites

This is a continuation of a process started in SailWind Logic. You must have the netlist (**.asc*) file. For more information, see "Manual Netlist Process Between SailWind Logic and SailWind Layout" in the *SailWind Logic Guide*.

Procedure

1. In SailWind Layout, click the **File > Import** menu item.
2. Select "ASCII Files (*.asc)" in the file type list.
3. Navigate to the location of the netlist file created from the SailWind Logic design, select it, and click Open.

Results

The import process sources all the part types and decals from the library and stacks the decals at the origin. You can then click the **Tools > Disperse Components** menu item to spread out the components. Alternatively, you can arrange parts based on the schematic sheet.

If an errors report file (*ascii.err*) is generated and errors are found in the netlist, see "Troubleshooting the Netlist Process" for more information.

Troubleshooting the Netlist Process

If SailWind Layout finds errors in the netlist import process, it generates an errors report file (*ascii.err*) and displays the error report file in a Notepad window. SailWind Layout does not generate an errors file if it does not find any errors.

The following are errors that are reported and steps to perform in fixing those errors:

- 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

Procedure

1. In SailWind Layout, click the **File > New** menu item.
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. (For instructions, see "Adding Libraries to the Library List" on page 144, "Creating and Modifying Part Types", and "Creating and Editing PCB Decals".)
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 SailWind Logic to SailWind Layout.

Related Topics

Creating a New PCB Design by Manually Importing the SailWind Logic Netlist

Dispersing Components by Schematic Sheet

After receiving netlist from SailWind Logic, you can use the "Disperse by Logic" feature to disperse components while retaining their relative positions almost as indicated in the schematic sheet. You can also specify what action to take and perform on the components to disperse.

Prerequisites

- You must have the netlist (**.asc*) file passed from SailWind Logic to SailWind Layout, and all the errors in the *ascii.err* file resolved.
- Netlists to use with this feature must be generated by SailWind Logic in versions later than V3.0.34.

Procedure

1. With nothing selected, right-click and click the **Disperse by Logic** popup menu item.
2. In the “Disperse by Logic” dialog box, specify the dispersing strategy, with details described below.

Field	Description
Min Space	Specifies the minimum allowable spacing between parts to disperse in mils.
Union/Cluster area	Specifies what action to take and perform on the parts to disperse by clicking **Disperse All**/**Disperse**, with options below: • Nothing (default): Nothing else • Create Union: Use the parts to create a union. • Create Cluster: Use the parts to create a cluster.
Sheets/Parts area	• Sheets list on the left: Lists all the schematic sheets. • Parts table on the right: Displays information of parts in the selected schematic sheet, including reference designator, PCB decal, and value. Changes depending on the selection made in the Sheets list.
Disperse All	Disperses parts in all the schematic sheets, except those glued or in unions/clusters.
Disperse	Disperses parts in the selected schematic sheet, except those glued or in unions/clusters. **Note:** If no space is available for placement, the parts attach to the pointer for manual placement in the design.

Cross Probe with SailWind Logic

You can control cross probing with SailWind Logic by using the SailWind Layout Link within SailWind Logic.

For instructions, see “Cross-Probe Between SailWind Products” in the *SailWind Logic Guide.*

Forward-Annotation of Design Changes from SailWind Logic

You can import design changes from SailWind Logic into SailWind Layout. Create the ECO file containing the design changes by comparing the schematic and the layout design. You can then bring forward design changes from the SailWind Logic design using one of three methods.

Tip
If you are creating a new PCB by importing a netlist for the first time, see “Creating a New PCB Design by Manually Importing the SailWind Logic Netlist” on page 271.

CAUTION:
Design changes resulting from forward-annotating can cause existing “electrical nets” to be truncated, split, or deleted altogether. It can also cause existing electrical nets to be extended or new electrical nets to be created if the refdes prefixes of updated components are specified in the Electrical Nets dialog box. For more information, see “Electrical Nets” on page 337.

If you have both SailWind Logic and SailWind Layout on your computer, use the “SailWind Layout Link” in SailWind Logic for the simplest and fastest automated process. For more information, see Forward Annotating Using the Automated SailWind Layout Link in the *SailWind Logic Guide*.

If you do not have SailWind Logic on your computer, you must compare the schematic and layout designs in either SailWind Logic or SailWind Layout to generate an *.eco* file to apply to the layout design. Using the Compare/ECO tool in SailWind Layout requires less steps since it can also automatically import the differences of the generated *.eco* file.

Forward Annotating Using an ECO File Generated by SailWind Layout

SailWind Logic provides you with updated netlist file that you can compare the current PCB layout to generate an *.eco* file. You can then import the file to update the PCB layout.

This procedure is a continuation of the procedure Method 2 — Generate the ECO File in SailWind Layout in the *SailWind Logic Guide*.

CAUTION:
Design changes resulting from forward-annotating can cause existing electrical nets to be truncated, split, or deleted altogether. It can also cause existing electrical nets to be extended or new electrical nets to be created if the refdes prefixes of updated components are specified in the Electrical Nets dialog box. For more information, see “Electrical Nets” on page 337.

Restrictions and Limitations

- 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 category of the SailWind Logic, Options dialog box.

Prerequisites

You must have an updated schematic netlist (*.asc*) file, and have the PCB design open in SailWind Layout.

Procedure

1. 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.

2. In SailWind Layout click the **Tools > Compare/ECO** menu item.

3. In the Compare/ECO dialog box, click the Documents tab on page 1185.

4. In the “Original Design to Compare and Update” area, select the “Use Current PCB Design” check box. If the check box is unavailable, clear the check box of the same name in the “New Design with Changes” area.

5. In the New Design with Changes area, browse for the updated schematic netlist (*.asc*) file. You will need to change the file type to “ASCII Files (*.asc)”.

6. Click the Comparison tab on page 1181, and select the options you want to use for design comparison.

   Tip
   To avoid unexpected changes during forward annotation, consider comparing data before you forward-annotate. 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 *\SailWind Projects* folder. To see the differences, click **Show Report** in the Process Status dialog box.

7. On the **Documents** tab, select the Generate ECO File check box, and verify the ECO Filename. Give the file a unique name to avoid overwriting any existing ECO files.

8. Click the Update tab on page 1187, and select the Update Original Design check box. This saves you from manually importing the generated ECO file.

9. Set the other options for updating.

10. Click **Run**. The Process Status Dialog Box opens. Output files are written to the *\SailWind Projects* folder. Messages or errors that occur during comparison are also written to *Layout_Session.log* and *Layout.err* in the *\SailWind 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 the Generate ECO File check box in the Compare/ECO Tools dialog box, **Documents** tab. For more information, see “ECO File Commands” in the *SailWind Layout Command Reference*.

- *Layout.rep* — The Differences Report file. Describes the differences between the “old” and the “new” compared files. Generated when you select the Generate Differences Report check box from the Compare/ECO Tools dialog box, **Documents** tab. For more information, see “Differences Report” on page 848.

- *ecogtmp0.asc* — Temporary copy of the “old” netlist

- *ecogtmp1.asc* — 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.

Forward Annotating by Importing an ECO File from SailWind Logic

You import the *.eco* file of design changes into SailWind Layout.

For more information, see Method 3 — Generate the ECO File in SailWind Logic in the *SailWind Logic Guide*.

CAUTION:
Design changes resulting from forward-annotating can cause existing electrical nets to be truncated, split, or deleted altogether. It can also cause existing electrical nets to be extended or new electrical nets to be created if the refdes prefixes of updated components are specified in the Electrical Nets dialog box. For more information, see “Electrical Nets” on page 337.

Restrictions and Limitations

- 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 category of the SailWind Logic, Options dialog box.

Tip
As long as the PCB design has not undergone engineering changes, the last *.asc* file from SailWind Logic can be compared to the current design in SailWind Logic to generate the *.eco* file of the engineering changes in the design. If the last exported *.asc* file is lost, you can export an on page 291 *.asc* file from SailWind Layout to compare against the current schematic to generate the *.eco* file that gets imported into SailWind Layout to update the PCB layout. The same effect is created using “Forward Annotating Using an ECO File Generated by SailWind Layout” but with that method, the process is semi-automated by automatically importing the *.eco* file.

Prerequisites

You must acquire an *.eco* file, and have the PCB design open in SailWind Layout.

Procedure

1. 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.
2. Click the **File > Import** menu item.
3. On the File Import dialog box, select the "ECO Files (*.eco)" file type.
4. Browse to the file to import and click **Open**.

Results

If there are errors during the import, an *eco.err* file opens in your default text editor. No changes 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 Annotation from SailWind Layout to SailWind Logic

Using the process called *backward annotation*, you can export your PCB layout changes "back" to the schematic. You can send part, gate, pin, net, and attribute changes.

CAUTION:
While there are three methods to backward annotate design changes, only one method is recommended — Back Annotating Using an ECO File from SailWind Layout. The other methods generate an *.eco* file by design comparison - a less accurate process. The recommended method records the *.eco* file in SailWind Layout while making ECO changes and manually imports it into SailWind Logic for the best results. For more information, see "Recording Versus Generating an ECO File" on page 814.

Back Annotating Using the Automated SailWind Layout Link

If SailWind Logic and SailWind Layout reside on the same computer, you can use the SailWind 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.

Restrictions and Limitations

Transferring non-ECO-registered parts and non-electrical parts is constrained. See Options Dialog Box, Design Category for details.

During design comparison, a reuse definition is ignored and actual elements in the physical design reuse are used in the comparison.

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 Back Annotating Using an ECO File from SailWind Layout for the best results. For more information, see "Recording Versus Generating an ECO File" on page 814.

Prerequisites

You must have the older schematic open in SailWind Logic, and the newer PCB design open in SailWind Layout.

Procedure

1. In SailWind Logic, click the **Tools > SailWind Layout** menu item.
2. If SailWind Layout is not already open with the design, the Connect to SailWind Layout dialog box appears.
   a. Click **Open** to open the PCB design you want to annotate from in SailWind Layout.
   b. In the File Open dialog box, select the *.pcb* file and click **Open**.
3. In the SailWind Layout Link dialog box, click the Design tab.

> **Tip**
> To avoid unexpected changes during backward annotation, consider comparing data before you back-annotate. 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 *\SailWind 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, select the “Compare Design Rules” and “Show Net List errors report” check boxes.
   b. Click the **ECO From PCB** button.

Results

If you receive a message that the schematic net list may have errors, you should not continue, but select the Show Net List errors report check box and run ECO From PCB again to investigate and fix the errors before you proceed.

Related Topics

Cross-Probe Between SailWind Products [SailWind Logic User's Guide]

Back Annotating Using an ECO File from SailWind Layout

Create the *.eco* file of design changes using SailWind Layout and then import it into the SailWind Logic design.

Recording engineering changes to your design is the preferred means of generating the *.eco* file since it creates a perfect “before-and-after” record of changes. You can generate an *.eco* file by comparing designs; however, 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” on page 814.

Restrictions and Limitations

- Transferring non-ECO-registered parts and non-electrical parts is constrained. see “Options Dialog Box, Design Category” for details.
- During design comparison, a reuse definition is ignored and actual elements in the physical design reuse are used in the comparison.

Prerequisites

If you neglected to record the *.eco* changes and you must generate the *.eco* file by comparing two designs, see “Comparing Two Versions of a Design”.

Procedure

1. Record the engineering/netlist changes in an *.eco* file. For more details, see“Recording ECO Changes on page 814”.
2. In SailWind Logic, click the **Tools > Options** menu item. In the Options Dialog box, click the **Design** category.
3. Set the “Allow overwriting of attribute values in design with blank values from library” check box appropriately to allow or prevent overwriting of non-blank attribute values with blank (“placeholder”) values from the library.
4. Click **OK**.
5. In SailWind Logic, with your design open, click the **File > Import** menu item.
6. In the File Import dialog box, in the file type list, select “ECO Files (*.eco)”.
7. Browse for and select the ECO file to import.
8. Click **Open**.

Results

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 (*eco.err*), are written to the Output window.

Back Annotating Using an ECO File Created in SailWind Logic

You can acquire the layout design in an *.asc* file format and compare it to the schematic design with the SailWind Logic Compare/ECO tools. You can then import *.eco* file of design changes into the SailWind Logic schematic design.

Restrictions and Limitations

- Transferring non-ECO-registered parts and non-electrical parts is constrained. see “Options Dialog Box, Design Category” for details.
- During design comparison, a reuse definition is ignored and actual elements in the physical design reuse are used in the comparison.

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 “Back Annotating Using an ECO File from SailWind Layout” for the best results. For more information, see “Recording Versus Generating an ECO File” on page 814.

Prerequisites

You must have an *.asc* file exported from SailWind Layout. For more details, see “Exporting an ASCII File on page 291”.

Procedure

1. In SailWind Logic, click the **Tools > Options** menu item. In the Options dialog box, click the **Design** category.
2. Set the “Allow overwriting of attribute values in design with blank values from library” check box appropriately to allow or prevent overwriting of non-blank attribute values with blank (“placeholder”) values from the library.
3. Click **OK**.
4. In SailWind Logic, click the **Tools > Compare/ECO** menu item.
5. In the Compare/ECO dialog box, click the Documents Tab.
6. In the Original Design to Compare and Update area, select the Use Current Schematic Design check box. If it is unavailable, you will first need to clear the check box of the same name in the New Schematic Design with Changes area.
7. In the New Schematic Design with Changes area, browse for the new *.asc* file exported from SailWind Layout. In the New Design File Name dialog box, change the file type to “ASCII Files (*.asc)” to locate your file.
8. Click the Comparison Tab, and select the options you want to use for design comparison.
9. Optionally, 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 *\SailWind Projects* folder. Click **Show Report** in the Process Status dialog box to view the differences.

10. Select the Generate ECO File check box, and verify the ECO Filename. Give the file a unique name to avoid overwriting any existing ECO files.
11. Click **Run**. Output files are written to the *\SailWind Projects* folder. Messages or errors that occur during comparison are also written to *Logic_Session.log* and *Logic.err* in the *\SailWind Projects* folder.
12. In SailWind Logic, click the **File > Import** menu item.
13. In the File Import dialog box, in the file type list, click “ECO Files (*.eco)”.
14. Browse for and select the ECO file to import.
15. Click *Open*.

Results

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 (*eco.err*), are written to the Output window. You must correct the errors before you can import the file.

Related Topics

Backward Annotation Results

Backward Annotation Results

The backward annotation process handles various layout changes differently while back-annotating to the schematic.

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.
- 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.

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.
- 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 multi-gate 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 multi-gate 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 multi-gate part, all gates are updated. An error message is generated if the old reference designator does not exist.

Gate Level Backward Annotation

You can backward annotate swapped gates.

- SailWind Logic creates an offpage symbol at each swapped gate. An error message is created if the gate does not exist.

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.

Pin Level Backward Annotation

You can backward annotate swapped pins, pins added to a net, and pins disconnected from a net.

Swapped Pins

- SailWind 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.

Chapter 13
Importing and Exporting

Read the topics that follow to learn more about the import and export of various file types.

File Import Formats

The **File > Import** menu item imports design data from another file into a *.pcb* design.

The supported file formats are listed in Table 70.

Table 70. Import Data Formats

Format	Extension	Description
ASCII on page 290	*.asc*	PADS-ASCII format text Exception: SailWind Layout no longer imports Perform 6 ASCII format files.
Data eXchange Format, or DXF on page 297	*.dxf*	A graphical information format used by AutoCAD and other CAD systems.
Engineering Change Order	*.eco*	Contains forward annotation or backward annotation information generated in a schematic capture package, concerned with logic changes to the design. For more information, see "Working with SailWind Logic".
Intermediate Data Format, or IDF on page 300	*.emn*	Data in IDF 2.0 or 3.0 format. You must have the IDF Interface option in SailWind Layout to import IDF files. For more information, see "Intermediate Data Format".
OLE on page 293	*.ole*	You can use the **Edit > Insert New Object** menu item to embed files from other applications as OLE objects in your design. If you then export all these objects to an OLE file, you can embed them as a set into another design using the **File > Import** menu item. For more information, see "Inserting OLE Objects in SailWind Layout".
Collaboration	*.clb*, *.cle*	Design markups in the collaboration file (*.clb*) and encrypted collaboration file (*.cle*) formats. For more information, see "Importing Markups Using the Markups Dialog Box", and "Importing Markups Using File > Import".
Protel 99SE design database on page 303	*.ddb*	Protel 99 design files in the binary format or ASCII format.
Protel 99SE design on page 304	*.pcb*	Protel 98 design files in the binary format or ASCII format.
Protel DXP / Altium Designer design on page 304	*.pcbdoc*	Altium DXP/2004/2006/Altium Designer design files in the binary format or ASCII format.
P-CAD design on page 305	*.pcb*	P-CAD design files in binary and ASCII formats generated in P-CAD 2001, 2002, 2004, or 2006.

Table 70. Import Data Formats (continued)

Format	Extension	Description
CADSTAR PCB design on page 305	*.pcb*	CADSTAR design files in binary format generated in CADSTAR 5.0, 6.0, 7.0, or 8.0.
CADSTAR PCB archives on page 306	*.cpa*	CADSTAR files in ASCII format generated in CADSTAR 5.0, 6.0, 7.0, or 8.0.
OrCAD Board on page 306	*.max*	OrCAD Layout design files in binary generated in OrCAD Layout 9.X, 10.X.
Eagle Board	*.brd*	Eagle design files
PADS Maker	*.pcb*	PADS Maker (PCB design software from Digi-Key®, based on Siemens Digital Industries Sofware)

File Export Formats

Exporting selectively extracts data from the current *.pcb* file and saves to a file in another format.

The supported formats are listed in the table below.

Table 71. SailWind Layout Export Data Formats

Format	Extension	Description
ASCII Export on page 291	*.asc*	PADS-ASCII format text **Restriction:** SailWind Layout does not export Perform 6 ASCII format files. If you are exporting to the format of a release earlier than PADS 9.5, virtual pins in the design are exported as vias.
Data eXchange Format, or DXF on page 298	*.dxf*	A graphical information format used by AutoCAD and other CAD systems.
Intermediate Data Format, or IDF on page 301	*.emn* and *.emp*	Export data to IDF 2.0 and 3.0 formats. You must have the IDF Interface option in SailWind Layout to export IDF files. Virtual pins in the design are exported as vias. For more infomation, see “Intermediate Data Format”.
OLE on page 293	*.ole*	You can use the **Edit > Insert New Object** menu item to embed files from other applications as OLE objects in your design. If you then export all these objects to an OLE file, you can embed them as a set into another design using the **File > Import** menu item.

Table 71. SailWind Layout Export Data Formats (continued)

Format	Extension	Description
		For more information, see "Inserting OLE Objects in SailWind Layout".
PADS HyperLynx HYP on page	*.hyp*	You can use the **File > Export** menu item to export the design to a *.hyp* file. Virtual pins in the design are exported as vias.
IPC-D-356 or IPC-D-356a on page 310	*.ipc*	Export your data to IPC-D-356 or IPC-D-356 revision A formats. Virtual pins in the design are exported as vias.
ODB++ on page 308	*.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. Virtual pins in the design are exported as vias.
Collaboration Files	*.cle*	Export markups in the encrypted collaboration file format for use in collaboration software. For more information, see "Exporting Markups Using the Markups Dialog Box", and "Exporting Markups Using File > Export".
CCE Files on page 309	*.cce*	Export the design in compressed (*.cce*) format for use in design review and collaboration software, such as CAMCAD, or visECAD. Virtual pins in the design are exported as vias.
CAM350 File on page 876	*.cam*	Launch the CAM350 link to export the CAM350 data and/or open the file in CAM350. Virtual pins in the design are exported as vias.
SPECCTRA on page 1001	*.do*	Launch the Specctra Translator to create the *.do* file for the Specctra autorouter. Virtual pins in the design are exported as vias.

Importing an ASCII File

You can import an ASCII file created with previous versions of SailWind Layout.

If your data requires format or other changes, such as the addition of an attribute dictionary, to update your data to the current SailWind release, SailWind Layout makes the updates at the time of import. If the ASCII file contains test points, SailWind Layout removes the test points that are not accessible and lists them in the report file. SailWind Layout preserves physical design reuses when you import an ASCII file.

Restrictions and Limitations

- You cannot import the ASCII file if any of the following conditions is 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 in the design to match the number in the imported ASCII file.
    - Perform 6 ASCII created the ASCII file.
- Improvements in thermal relief definition introduced in PADS 9.2 might 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 or later. 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 on page 1590. When you import an ASCII file created by a PADS version previous to 9.0, 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. Click the **File > Import** menu item.
2. In the File Import dialog box, select the ASCII Files (*.asc) file type.
3. Browse to the file to import and click **Open**.

Results

The file is imported, and import process information is written to *ascii.err* in the *...\SailWind Projects* folder.

Related Topics

Exporting an ASCII File

Exporting an ASCII File

You can use ASCII files to exchange design data between SailWind Layout and external translators or previous versions of SailWind Layout.

Restrictions and Limitations

- SailWind Layout does not export to PADS-Perform 6 ASCII format.
- Exporting to older formats will result in the loss of any newer software features not supported by older formats.
- Improvements in thermal relief definition introduced 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 or later 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 on page 1590. With this change, the following changes occur in a design when you export it to an ASCII file of a PADS version previous to 9.0:
    - 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.
- If you are exporting to the format of a release earlier than PADS 9.5, virtual pins in the design are exported as vias.

Procedure

1. Click the **File > Export** menu item.
2. On the File Export dialog box, select the ASCII Files (*.asc) file type.
3. Specify the filename to export to and click **Save**.
4. In the ASCII Output Dialog Box, check the appropriate check boxes to specify the design sections that you want to write to the ASCII file, or click **Select All**.
5. In the Format list, select the format version. (If you want to export to PowerBGA V3.0, use the PowerPCB V3.0 format.)
6. In the Units list, select the units to export. 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 appropriate.
8. Click **OK**.

Results

The *.asc* file is created. An export status log file— *ascii.err*—and an export report file—*Powerpcb.Rep*—are created in the *\SailWind Projects* folder. Existing physical design reuse information is retained.

Related Topics

Importing an ASCII File

Exporting OLE Files

You can export all the OLE objects in your design to an OLE file, which you can then import into another design.

Procedure

1. Click the **File > Export** menu item.
2. On the File Export dialog box, select the OLE Files (*.ole) file type.
3. Specify the filename to export to and click **Save**.

Results

The *.ole* file containing all OLE objects in your design is created.

Related Topics

Inserting OLE Objects in SailWind Layout

Importing an OLE File

Importing an OLE File

You can import an OLE file (such as a screenshot) into your design.

Restrictions and Limitations

- When you import an OLE file into your design, all OLE objects currently in the design are deleted.

Procedure

1. Click the **File > Import** menu item.
2. On the File Import dialog box, select the OLE Files (*.ole) file type.
3. Browse to the file to import and click **Open**.
4. Click **Yes** in the two prompts that appear.

Results

Any existing OLE objects in the design are removed, and the OLE objects in the selected OLE file are imported into the design.

Related Topics

Inserting OLE Objects in SailWind Layout

Exporting OLE Files

DXF Format

The DXF format handles database objects differently than ASCII files. DXF does not support physical design reuses. DXF also does not support route protection information; therefore, when you import or export a DXF file, the process removes all route protection information. DXF does, however, store slotted hole information for both import and export. Because jumpers, although not part of the part list, are considered vias, you can export them to DXF.

SailWind Layout does not import DXF files from PowerPCB 1.3 and earlier. Versions before PowerPCB 3.0 do not read the DXF files created by PowerPCB 4.0.

If you import a DXF file that contains an ellipse, or a scaled arc with a radius that is too big for the database, a dialog box appears asking you to enter an approximation error. The approximation error determines how the ellipse/arc will be broken into line segments.

For information on creating objects that properly translate to copper polygons and to avoid creating self-intersecting polygons, see Defining Copper Objects in AutoCAD.

DXF Messages

The following are possible DXF messages:

- Warning: Found a self-intersecting copper piece while importing hatched solid. File line: XXXX. Continue process?

  The DXF file contains a self-intersecting polygon. File line: XXXX refers to the line where the error occurs. Correct the polygons in the AutoCAD application. For information on creating objects that properly translate to copper polygons and avoiding creating self-intersecting polygons, see Defining Copper Objects in AutoCAD.

- Warning: Board cutout containing point <x,y> intersects existing board cutout containing <x,y>. New cutout not imported from DXF file.

  The Overlapping board cutouts are reported as warnings when importing a DXF file. Click **OK** to continue importing or click **Cancel** to cancel.

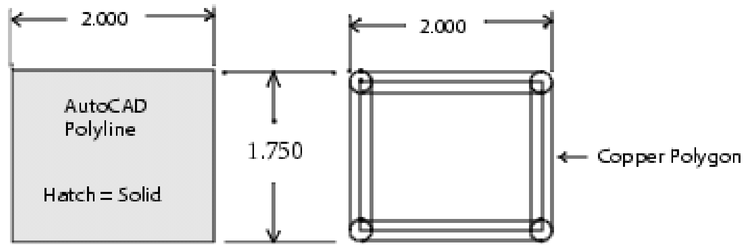
- Warning: Board cutout containing point <x,y> intersects the board outline. Cutout not imported from DXF file.

  The cutouts that overlap the board outline are reported as warnings. Click **OK** to continue importing or click **Cancel** to cancel.

Defining Copper Objects in AutoCAD

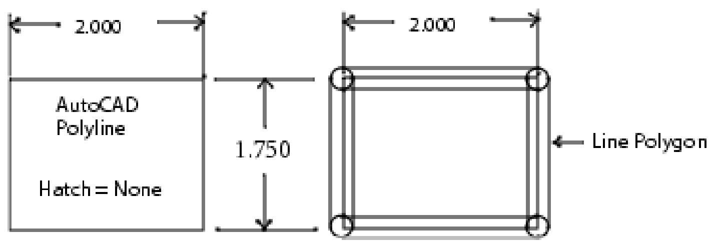
To define an object in AutoCAD that properly translates to a copper polygon, create a closed polyline that does not self-intersect and assign a solid hatch type. If you create a polygon in this way, it converts to a copper polygon in SailWind Layout (Figure 24). In addition, the copper polygon is created in SailWind Layout so that the outside edges of the polygon are aligned with the edges of the shape in AutoCAD, ensuring the entire shape is dimensioned and checked correctly. The following graphic demonstrates this conversion.

Figure 24. Copper Object with Closed Polygon (Solid Hatch Type)



If you create a polygon without hatching in AutoCAD, the polygon converts to a 2D line in SailWind Layout (Figure 25). In this case the centerlines of the 2D line are aligned with the edges of the shape in AutoCAD. The following graphic demonstrates this conversion.

Figure 25. Copper Object with Closed Polygon (No Hatching)



DXF Export of Filled Polygons

Filled copper polygons export as polylines with a solid hatch.

Legacy Layer Mapping

The following is a cross-reference of the DXF layer names used in the various releases of PADS products. We don't recommend you try to bring an old DXF file into a new release of SailWind Layout.

Table 72. PADS Legacy Layer Mapping

PADS Perform	PowerPCB v1.1 - v1.3	PowerPCB v1.5 - current
TXTnn	TEXTnn	TEXT_nn
LINnn	2D_LINEnn	2D_LINE_nn
COPnn	COPPERnn	COPPER_nn

Table 72. PADS Legacy Layer Mapping (continued)

PADS Perform	PowerPCB v1.1 - v1.3	PowerPCB v1.5 - current
CCOnn	COPPER_CUTOUTnn	COPPER_CUTOUT_nn
CPRnn	POUR_UNFLOOD	POUR_VOID_nn
CPOnn	POUR_FLOOD	POUR_HEADER_nn
HOLnn	HATCH_OUTLINEnn	POUR_OULINE_nn
HVOnn	HATCH_VOIDnn	POUR_VOID_nn
THVnn	VIA_THERMALnn	POUR_VIATHERM_nn
THRnn	PAD_THERMALnn	POUR_PADTHERM_nn
BRD00	BOARD_OUTLINE	BOARD_OUTLINE_00
CONnn	CONNECTIONnn	LINK_00
PADXnn	PADS_INNER	PADS_INNER
PADXBnn	PADS_BOTTOMnn	PADS_BOT
PADSTnn	PADS_TOPnn	PADS_TOP
PNM00	PART_NAME_TOP and PART_NAME_BOTTOM	PART_NAME_TOP and PART_NAME_BOT
CMB00	PART_TOP	PART_TYPE_TOP
COM01	PART_BOTTOM	PART_TYPE_BOT
BDYnn	BODYnn	
DRLTPnn	DRIL_PLTE_THRUnn	DRIL_PLTE_THRU_nn
DRLTN	DRIL_NO_PLTE_THRUnn	DRIL_NPLTE_THRU_nn
DRLNP	DRIL_PLTE_PRTLnn-nn	DRIL_PLTE_PRTL_nn
DRLNN	DRIL_NPLTE_PRTLnn-nn	DRIL_NPLTE_PRTL_nn
TRKnn	TRACKSnn	TRACE_nn
VIAi	VIAi	VIA
PKGnn	PARTINFO	PART_INFO
PCB30	PADS_PARAMETERS	PCB_PARAMS
KEY00	DRILL_SYMBOL	DRIL_SYMBOL

Table 72. PADS Legacy Layer Mapping (continued)

PADS Perform	PowerPCB v1.1 - v1.3	PowerPCB v1.5 - current
CTBnn	COMPONENT_TEXTnn	PART_TOP_TEXT_nn and PART_BOT_TEXT_nn

Importing DXF Files

You can import DXF files of the AutoCAD 2004 DXF format.

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 cannot be changed.

If you need to simply import RF shapes into the decal or into the design, use the procedure given in Importing RF Shapes in DXF Format.

Restrictions and Limitations

- 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. Click the **File > Import** menu item.
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.

   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, check the appropriate check boxes to specify the items that you want to import, or click **Select All**.
6. In the DXF-File Unit list, select the units to use in the DXF file. 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**.

Results

The *.dxf* file is imported and opened in SailWind Layout.

Line styles imported in the DXF file are matched with the five SailWind Layout line styles. When a line style does not resemble any of the SailWind Layout line styles, a solid line is created.

If your imported geometries have been imported as multiple line items, and you want to convert them to single objects, use the procedures listed in Join and Close 2D Lines and Copper Shapes to do this.

CAUTION:
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 be imported as regular copper shapes.

Related Topics

Importing a Board Outline and Cut Out from AutoCAD

DXF Format

Exporting DXF Files

You export DXF files to transfer design elements to AutoCAD. For example, you can export component height information with the design in order to create a 3D model.

Restrictions and Limitations

- The *.dxf* file version exported is in version 12 format.
- Nets bridged with copper are not supported in the *.dxf* file format and are exported as regular copper shapes.
- Although 2D lines are drawn with a round pen in SailWind Layout, they are squared (not rounded) in DXF.
- Also, many DXF software tools display patterns of styled lines as hollow and not filled.

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” on page 1338, in the Export Type area, choose between standard and flat output.

   If you select flat output, setup of DXF drill sizes and symbols is unavailable, and 2D Line styles (dash, dotted, dash-dotted, dash-double-dotted) are converted to solid lines.
5. In the Layer Selection area, move the layers you want to export information from to the Selected list.
6. In the Select Input Items area, check the items that you want to export, or click **All Items**.
7. In the DXF-File Unit list, select the units to use in the DXF file. The list is unavailable if it is not necessary to set the units.

8. If you are exporting a standard file, perform the following steps if you want to define DXF drill sizes and symbols:

    a. Click **Setup**.

    b. In the Setup DXF Drill Size and Symbols dialog box, 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.

       If you do not associate a drill size to a 2D line library item, the default drill symbol is exported for that item.

    c. 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**.

    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 in the Symbol Size box, and then type the plus sign line width in the Line Width box.

    f. Click **OK**.

9. Click **OK**.

Results

The *.dxf* file is created.

Related Topics

DXF Format

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. Click the **File > Export** menu item. Save as DXF type and in the Export dialog box click **Setup**.

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.

   If no drill size is associated 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 import board outlines, keepouts, components, and holes into your SailWind Layout design by importing an IDF *.emn* file exported from a mechanical design system.

Restrictions and Limitations

- SailWind 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.

Procedure

1. Click the **File > Import** menu item.

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. In the IDF Import Dialog Box, in the Select Import Items area, check the check boxes of the items that you want to import, or click **All Items** to import everything.

5. Click OK.

Results

The *.emn* board file is imported and opened in SailWind Layout. A status log file is written to *C:\SailWind Projects\idfimport.sts*.

Related Topics

Adding Components During Importing [SailWind Layout Command Reference Manual]

Importing Holes [SailWind Layout Command Reference Manual]

Exporting IDF Files

Exporting IDF Files

You can create IDF files to export board outlines, keepouts, components, and holes to a mechanical design system.

Tip
For a more accurate IDF export, set any IDF-specific part height, drilled hole, and part outline information before exporting. For more information, see"Setting IDF-Specific Part Height Information for IDF Export", "Adding IDF-Specific Drill Hole Information for IDF Export", and "Defining IDF-Specific Part Outlines for IDF Export".

Procedure

1. Click the **File > Export** menu item.
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 on page 302 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, check the items that you want to export, or 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. 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. If the Geometry.Height attribute for any exported part type and decal pairs does not exist or is set to zero, 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.

   If you do not specify the height (or specify it as zero), the mechanical design system may prompt you to enter a height when importing the IDF files.

   Tip
   If you want to apply the height for all part type and decal pairs, select the For All Parts check box.

9. Click **OK**.

Results

The *.emn* (board and placement) and *.emp* (part library) files are created. A status log file is written to *C:\SailWind Projects\idfexport.sts*.

Related Topics

Setting IDF-Specific Part Height Information for IDF Export

Missing Height Dialog Box

Importing IDF Files

Defining IDF-Specific Part Outlines for IDF Export

For a more accurate IDF export, create any IDF-specific part outlines before you export to IDF. The IDF library file requires part outline information for all parts.

Procedure

1. Click the **Tools > PCB Decal Editor** menu item.
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 [SailWind Layout Command Reference Manual]

Adding IDF-Specific Drill Hole Information for IDF Export

For a more accurate IDF export, use the HOLE part attribute to set any IDF-specific drill hole information for single-pin parts before you export to IDF. (Vias and multiple-pin parts, even though they appear as drilled holes in the mechanical design system, do not use the HOLE attribute.)

Mechanical design systems interpret drill holes as board drills.

Procedure

1. Click the **File > Library** menu item.
2. In the Library Manager dialog box, click the **Parts** button.
3. From the Part Types list, select the single-pin part for which you want to add drill hole information, and click **Edit**. (Even if you use the same part type with different decals for holes and nonholes, set the HOLE attribute in the part type nevertheless.)

4. In the **Attributes** tab of the Part Information dialog box, add the HOLE part attribute.

5. Click **OK**.

Setting IDF-Specific Part Height Information for IDF Export

For a more accurate IDF export, set any IDF-specific part height information before you export to IDF. You do this by assigning the Geometry.Height attribute at the Decal or Part Type hierarchy level.

For information on how to do this, see “Assigning Attributes to Design Objects With the Object Attributes Dialog”.

Related Topics

Design Object Attributes

Attribute Manager

Part Height Information [SailWind Layout Command Reference Manual]

Importing Protel 99SE Design Database Files

You can import a Protel *.ddb* file directly into SailWind Layout.

Tip
Run the standalone translator to control additional aspects of the translation, including file placement and attribute mapping. See the SailWind Layout Translator User’s Guide for more information. The SailWind Layout Translator can be found on the Start menu under the SailWind folder for the version you have installed and on Windows 7 within the Translators folder.

Procedure

1. Click the **File > Import** menu item.

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**.

Results

The *.ddb* file is imported and opened in SailWind Layout. The resulting SailWind 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 *\SailWind Projects* folder by default.

Importing Protel PCB98 Design Files

You can import a Protel *.pcb* file directly into SailWind Layout.

Tip
Run the standalone translator to control additional aspects of the translation, including file placement and attribute mapping. See the SailWind Layout Translator User's Guide for more information. The SailWind Layout Translator can be found on the Start menu under the SailWind folder for the version you have installed and on Windows 7 within the Translators folder.

Procedure

1. Click the **File > Import** menu item.
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**.

Results

The *.pcb* file is imported and opened in SailWind Layout. The resulting SailWind 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 *\SailWind Projects* folder by default.

Importing Protel DXP / Altium Designer Design Files

You can import a*.pcbdoc* file directly into SailWind Layout.

Tip
Run the standalone translator to control additional aspects of the translation, including file placement and attribute mapping. See the SailWind Layout Translator User's Guide for more information. The SailWind Layout Translator can be found on the Start menu under the SailWind folder for the version you have installed and on Windows 7 within the Translators folder.

Procedure

1. Click the **File > Import** menu item.
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**.

Results

The *.pcbdoc* file is imported and opened in SailWind Layout. The resulting SailWind 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 *\SailWind Projects* folder by default.

Importing P-CAD Design Files

You can import a *.pcb* file directly into SailWind Layout.

Tip
Run the standalone translator to control additional aspects of the translation, including file placement and attribute mapping. See the SailWind Layout Translator User's Guide for more information. The SailWind Layout Translator can be found on the Start menu under the SailWind folder for the version you have installed and on Windows 7 within the Translators folder.

Procedure

1. Click the **File > Import** menu item.
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**.

Results

The*.pcb* file is imported and opened in SailWind Layout. The resulting SailWind Layout file is saved in the same folder as the original file with the suffix *_pads*. For example, if the original filename 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 *\SailWind Projects* folder by default.

Importing CADSTAR PCB Design Files

You can import a *.pcb* file directly into SailWind Layout.

Tip
Run the standalone translator to control additional aspects of the translation, including file placement and attribute mapping. See the SailWind Layout Translator User's Guide for more information. The SailWind Layout Translator can be found on the Start menu under the SailWind folder for the version you have installed and on Windows 7 within the Translators folder.

Procedure

1. Click the **File > Import** menu item.
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**.

Results

The *.pcb* file is imported and opened in SailWind Layout. The resulting SailWind 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 *\SailWind Projects* folder by default.

Importing CADSTAR Archives

You can import a *.cpa* file directly into SailWind Layout.

Procedure

1. Click the **File > Import** menu item.
2. On the File Import dialog box, select the “CADSTAR archive (*.cpa)” file type.
3. Browse to the file to import and click **Open**.

Results

The *.cpa* file is imported and opened in SailWind Layout. The resulting SailWind 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 *\SailWind Projects* folder by default.

Importing OrCAD Board Files

You can import a *.max* file directly into SailWind Layout.

Tip
Run the standalone translator to control additional aspects of the translation, including file placement and attribute mapping. See the SailWind Layout Translator User’s Guide for more information. The SailWind Layout Translator can be found on the Start menu under the SailWind folder for the version you have installed and on Windows 7 within the Translators folder.

Procedure

1. Click the **File > Import** menu item.
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**.

Results

The *.max* file is imported and opened in SailWind Layout. The resulting SailWind 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 *\SailWind Projects* folder by default.

Importing Eagle Design Files

You can import a *.brd* file directly into SailWind Layout.

Tip
Run the standalone translator to control additional aspects of the translation, including file placement and attribute mapping. See the SailWind Layout Translator User's Guide for more information. The SailWind Layout Translator can be found on the Start menu under the SailWind folder for the version you have installed and on Windows 7 within the Translators folder.

Procedure

1. Click the **File > Import** menu item.
2. On the File Import dialog box, select the "Eagle Board files (*.brd)" file type.
3. Browse to the file to import and click **Open**.

Results

The *.brd* file is imported and opened in SailWind Layout. The resulting SailWind Layout file is saved in the same folder as the original file with the suffix *_pads*. For example, if the original filename is *EagleTest.pcb*, the new filename will be *EagleTest_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, *EagleTest_pads_1.pcb*, *EagleTest_pads_2.pcb*, and so on.

A log file, with the same name as the resulting *.pcb* file, is saved in the *\SailWind Projects* folder by default.

Importing PADS Maker Design Files

You can import a *.pcb* file directly into SailWind Layout.

Procedure

1. Click the **File > Import** menu item.
2. On the File Import dialog box, select the PADS Maker Layout files (*.pcb) file type.
3. Browse to the file to import and click **Open**.

Results

The *.pcb* file is imported and opened in SailWind Layout. The resulting SailWind Layout file is saved in the same folder as the original file with the suffix *_pads*. For example, if the original filename is *PADS MakerTest.pcb*, the new filename will be *PADSMakerTest_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, *PADSMakerTest_pads_1.pcb*, *PADSMakerTest_pads_2.pcb*, and so on.

A log file, with the same name as the resulting *.pcb* file, is saved in the *\SailWind Projects* folder by default.

Exporting ODB++ Files

You can export an entire design to an ODB++ job, which you transfer from computer to computer without any loss of data.

CAUTION:
ODB++ export is based on the CAM documents generated from your design.

- If you have not generated CAM documents and proceed without them, the software generates a set of CAM documents using default configurations and uses them for the ODB++ export. The resulting auto-generated CAM documents may be different than what you would create.
- A warning appears if you inadvertently duplicate one or more layers during the process of creating CAM documents.

Tip
Virtual pins in the design are exported as vias.

Procedure

1. Click the **File > Export** menu item.
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).

   If you have copper planes that are not flooded, they are flooded automatically when the CAM documents are being generated.

   If you have not 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 default settings.

   If you have not created any of the required CAM documents for your design you are prompted to generate the default documents.

4. Make appropriate setting changes in the ODB++ Export Dialog Box.

5. Click **Save**.

Results

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, and visECAD.

Procedure

1. Click the **File > Export** menu item.

2. In the File Export dialog box, in the Save as type list, select CCE Files (*.cce).

3. Browse to select a file to overwrite, or type a new file name. The file name will default to the existing design name.

4. Click **Save**.

5. In the CCE Export Dialog Box, specify the Solder Mask, Paste Mask, and Component Outline Layer options.

6. Click **OK**.

Related Topics

CCE Export Dialog Box

The IPC-D-356 Netlist

The IPC-D-356 netlist format describes the netlist in terms of X, Y locations, which is the format required for testing bare boards.

The IPC-D-356 format is an ASCII format in machine readable form which contains the X,Y locations of points of connectivity of nets, ensuring that the resulting printed circuit board is not only manufactured correctly, but is correct to the original design.

Prior to the IPC-D-356 netlist, ASCII file output was the only file output. This output described connectivity between pins by their reference designators. However, machines require coordinates and CAM outputs consisted of artwork files only. Connectivity could be discovered based on the layer stackup of the artwork, but it could never be checked against the original design.

The IPC-D-356 netlist includes net names, reference designators, and pin numbers. This is not required to test connectivity between coordinates, but it produces results which are easier to understand.

You can also use the updated IPC-D-356A format, which is an updated revision to the D-356 netlist format. The revision A format has more features and is an improvement on the original D-356 format.

Exporting an IPC-D-356 Netlist

You can export either an IPC-D-356 netlist or an IPC-D-356 Revision A netlist.

Warning:
Nets bridged with copper are combined when you export the design into the *.ipc* (IPC356) file format.

Procedure

1. Click the **File > Export** menu item.
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**. The *.ipc* file is created.

Related Topics

The IPC-D-356 Netlist

Chapter 14
Layers

Read the topics that follow to learn more about layer stackup and procedures for setting up the layers of your design.

Layer Modes

SailWind Layout supports two layer modes.

Supported layer modes are as follows:

- **Default layer mode (up to 30 layers)** — can consist of up to a maximum of 30 electrical layers or a combination of electrical and nonelectrical layers. In default layer mode, you can only import, add, or load other default layer mode items, such as files or library items, to your design. You cannot load increased layer mode objects into the default layer mode design.

- **Increased layer mode (up to 250 layers)** — can consist of up to a maximum 64 electrical layers and 186 nonelectrical layers. The total number of layers includes associated layers such as mask, silkscreen, drill drawing, and assembly layers

  In increased layer mode, you can load both default and increased layer mode objects into your design. Changing from default layer mode to increased layer mode increases all nonelectrical layer numbers by 100. In default layer mode, layer number 20 is used for placement outlines. In increased layer mode, layer 120 is used for placement outlines. Layer 25 in default layer mode, or 125 in increased layer mode, was historically used for oversizing thermals and antipads for CAM planes prior to the CAM Plot Options for CAM Plane Layers on page 1641.

  You do not have to convert existing default layer mode libraries, reuse files, or archived designs to increased layer mode. You can use existing library decals, drawings, and reuses that are saved in default layer mode for both default and increased layer mode designs. You can cut from a design in default layer mode and paste into a design in increased layer mode. If you make sure that you have consistent layer definition in libraries, reuses, and designs, no problems with layer matching will occur.

Restriction:
Once you change the design to increased layer mode, you cannot return to default layer mode.

Note:
You cannot export a design with more than 30 electrical layers to a SailWind Layout PADS-format ASCII file prior to version 4.0.

Either default or increased layer mode is specified in each design or design fragment such as *.pcb*, *.asc*, *.stp*, *.dxf*, *.reu*, and high speed *.edp* files, library decals and drawing items, copy/paste buffer, and external CAM documents.

Related Topics

Increasing the Maximum Number of Available Layers

Special Functionality of Layers 20 and 25

During the evolution of the SailWind design environment, certain layers within the standard layer setup have been adopted for specific uses. These uses include special functionality assigned to Layer_20 and Layer_25.

Layer 20 - Placement Outlines

When components are placed on a board, it is desired to keep them separated by a specific clearance in order to prevent them from overlapping and creating potential unwanted connections between the copper features of the components. Since some component leads extend beyond the body of the components, a specific outline that includes the desired clearance can be added on Layer 20 to represent the maximum boundary of the component

This placement outline is:

- Commonly referred to as the placement (or nudge) outline - sometimes called a component courtyard.
- Used to specify additional clearances around a component to accommodate specific requirements for specialized placement and rework equipment.
- Used by on-line DRC and Verify Design to check for placement outline spacing violations.This is in addition to the standard “Body to body” design rule clearance check.

Tip
To check the outlines, in the Verify Design, Clearance Checking Setup dialog box, enable the Placement outline check box.

Note:
Though only one layer is specified to contain the placement outlines, SailWind Layout uses the component placement layer assignment (top or bottom) of each component to determine how to interpret and check outline clearances during design verification.

For more information, see “Creating a Placement (Nudge) Decal Outline” on page 197.

Layer 25 - 3D Body Outlines

Historically, when using CAM planes (negative planes), positive images of anti-pads would be created in the pad stack on Layer 25. The plane outline would also be added as copper or a 2D line on Layer 25 around the board outline to create a gap from the plane to the board edge. This data would be merged with the CAM layer when the output files were generated for the photo plotter (Gerber files). Over time, the software evolved and “custom thermals and anti-pads” functionality was extended to CAM planes thereby making the use of anti-pads on Layer 25 obsolete.

Layer 25 is currently used for 3D body outlines. If you would like to view your design in the 3D Viewer, and you do not have a 3D model available for the component, the system will attempt to extrude a 3D representation of the component from the data available in the pcb decal. The 3D Viewer will automatically look for a 3D body outline on Layer 25 to use for extruding the shape.

Tip
If no 3D body outline is present on Layer 25, the system will attempt to use the silkscreen image of the component as the basis for the shape extrusion.

By default, the Land Pattern Creator uses Layer 25 for 3D body outlines when generating PCB decals.

For more information, see “Choosing Between Split/Mixed and CAM Layers”.

Choosing Between Split/Mixed and CAM Layers

Solid planes can be represented in two ways—positive and negative.

A positive plane is created by using solid polygons on a Split/Mixed layer to represent the area that you want filled with copper. Pads, vias, thermals, anti-pads and other design objects are subtracted from the solid area. This method most accurately represents how the actual plane will look at fabrication. When the image is sent to a photo plotter, a positive representation of the plane is created on the film.

In the past, the photo plotter would use a very small aperture to “paint” the solid areas. This typically required the photo plotter to create tens of thousands of line segments to represent the plane. The file size could become quite large and took a lot of time to process, both by SailWind Layout and the photo plotter.

A negative plane on a CAM layer is created by using the absence of data to represent the area that you want filled with copper and you then create positive images of the design elements (pads, vias, thermals, anti-pads) that you want to remove from the copper. When the image is sent to a photo plotter, only the positive images of the design elements to be removed are created on the film. The film is then photographically reversed to create the positive image used to fabricate the board. Using this process, the photo plotter only has to “paint” the areas where copper will be removed resulting in a much smaller file size and a substantial reduction in processing time.

Historically, when designs were sent to a photo plotter service bureau using very slow dial-up modems, file size was extremely important. Therefore, it was much more desirable to represent planes using the much smaller file sizes produced by negative (CAM) planes. In order to utilize this method, a CAM plane layer would be specified in SailWind to represent the plane, and then the positive images of anti-pads would be created as pads on Layer 25. This data would be merged with the CAM layer when the output files were generated for the photo plotter (Gerber files). The resulting files were compact, more economical to transfer and quicker to process. Unfortunately, these negative CAM planes had certain design restrictions that created potential risks in their use.

Though these negative image planes were quick to generate, they were designed for single net assignment and no routing. The framework for CAM planes was rudimentary and did not allow for custom thermals or anti-pads. Verification of CAM planes was severely limited and required a lot of manual checking to verify plane connectivity. Also, considering that the plane was represented by a negative image, the plane outline could not be drawn and a positive image object had to be added to layer 25 around the board outline to create the gap from the plane outline to the board edge.

Although “custom thermals and anti-pads” functionality has since been extended to CAM planes and the CAM plane functionality remains, it is outmoded by the advanced features available for planes on Split/Mixed plane layers. Split/Mixed planes are quick to generate and process using today's computing power and modern laser photo plotters. Split/Mixed planes allow you to split the layer into multiple planes. They allow a mixed use of planes and routing with custom thermals, automatic back-off from traces and control of the shape of planes.

For more information about Split/Mixed plane features, see the features attributed to the Plane area in “Differences Between Copper Shapes and Copper Planes”.

Association of Component and Documentation Layers

Layer association outputs text, line, or shape items on manufacturing plots for the component layers, top or bottom.

When you associate a documentation layer on page 1189dedicated to a manufacturing plot type with a component layer, the CAM output Document Type selections automatically include the associated documentation layer items with the plot type. You can associate the following layer types with a component layer: Paste Mask, Solder Mask, Silkscreen, and Assembly.

TrueLayer and Layer Association

When you define the pad stacks of a decal, the default layers for the pad stack are universal - Mounted, Inner, Opposite. They can be used universally on either side of the board. The mounted side can be the top or bottom of the board and the opposite side is always opposite the mounted side. If you were to instead create a pad stack that used specific layers, when you flipped the component to the other side of the board, the pad stacks would not flip.

Although you might add a specific Solder Mask, Paste Mask, Silkscreen or Assembly layer to the pad stacks of a decal, the TrueLayer feature flips the layer information when a component is flipped to the bottom side of the board. For example, if you have a surface mounted device with only a pad stack definition on the Solder Mask Top layer, when you flip the device to the bottom layer, the definition from the Solder Mask Top layer gets moved to the Solder Mask Bottom layer. You do not need to define a pad stack definition for the Solder Mask Bottom layer in case the component gets flipped.

To disable the TrueLayer feature, see the “Software Launch Options” topic for instructions on command line options.

Setting Up Layers

Set up your layers by first defining them in the Layer Setup dialog box.

Procedure

1. Click the **Setup > Layer Definition** menu item. The Layers Setup Dialog Box appears.
2. Set the number of layers for the design and if needed:
    - Increase the total number of layers on page 315available to the design database.
    - Designate that this is a single-sided board on page 316.
    - Increase the number of electrical layers on page 316.
3. Set up the outer on page 317 layers.
4. Set up the inner on page 318 layers.
5. Determine the visibility of non-electrical layers on page 319.
6. Definelayer and substrate thickness on page 320.

Increasing the Maximum Number of Available Layers

SailWind Layout features two layer modes—default layer mode and increased layer mode. Should you choose to switch to the increased layer mode, you can raise the number of available layers.

For more information regarding the two modes, see “Layer Modes”.

Restrictions and Limitations

When you switch a design to increased layer mode, you cannot return the design to default layer mode.

Procedure

1. Click the **Setup > Layer Definition** menu item. The Layers Setup Dialog Box appears.
2. Click **Max Layers**. The Increase Maximum Layer Number Dialog Box opens.
3. Click **OK** to switch to increased layer mode.

Results

All documentation layer numbers from the 30 layer database are increased by 100. For example, if you started with a 12 electrical-layer design, the first documentation layer, layer 13, becomes layer 113.

Designating a Board as Single-sided

You can only designate a board to be single-sided provided it has no more than two electrical layers.

Procedure

1. Click the **Setup > Layer Definition** menu item. The Layers Setup Dialog Box appears.
2. Select the Single-sided board support check box.
3. Click **OK**.

Results

- 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 (30 layer database) mode or increased-layer (250 layer database) mode.

For more information, see "Layer Modes".

Restrictions and Limitations

You can't modify the layer definition for electrical layers if a physical design reuse exists on the board.

Prerequisites

Before changing the number of electrical layers, delete all partial vias and their definitions from your design.

Procedure

1. In the Layers Setup Dialog Box, click **Modify**. The Modify Electrical Layer Count Dialog Box opens.
2. In the Modify Electrical Layer Count dialog box, type the new number of electrical layers 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. For more information, see "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 on page 1337.

Setting Up an Outer Layer

The outer layers of your design are the Top and Bottom layers. You can designate an outer layer as a component layer or non-component layer.

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. By default, the outer layer names are assigned 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.

       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.
    c. Click **OK**.
4. In the Layers Setup dialog box, click a Plane Type setting. If you select CAM or Split/Mixed Plane, you must assign the plane net(s) to the layer:
    a. Click the **Assign Nets** button. The Plane Layer Nets Dialog Box appears.
    b. Click a net from the All Nets list.
    c. Click **Add** to move the net to the Assigned Nets list. You can also assign a net to other layers as required. For example, if you have multiple ground plane layers, you can assign your ground net to those multiple layers.
    d. If the layer is a Split Mixed Plane type, you can associate additional nets to the layer.
    e. Click **OK**.
5. In the Layers Setup dialog box, click a Routing Direction setting. 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

Setting Up an Inner Layer

Setting Up a Documentation Layer

Association of Component and Documentation Layers

Choosing Between Split/Mixed and CAM Layers

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. By default, SailWind 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 a Plane Type setting. If you select CAM or Split/Mixed Plane, you must assign the plane net(s) to the layer:
    a. Click the **Assign Nets** button. The Plane Layer Nets Dialog Box appears.
    b. Click a net from the All Nets list.
    c. Click **Add** to move the net to the Assigned Nets list. You can also assign a net to other layers as required. For example, if you have multiple ground plane layers, you can assign your ground net to those multiple layers.
    d. If the layer is a Split Mixed Plane type, you can associate additional nets to the layer.
    e. Click **OK** to close the Plane Layer Nets dialog box.
4. In the Layers Setup dialog box, click a Routing Direction setting. 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

Setting Up an Outer Layer

Setting Up a Documentation Layer

Choosing Between Split/Mixed and CAM Layers

Setting Up a Documentation Layer

SailWind 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. 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. By default, SailWind 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 a Fab. Assembly and Documentation Layer Type setting. If you click SilkScreen, Paste Mask, Solder Mask, or Assembly, this layer is available for association with component layers.
4. Click **OK**.

Related Topics

Association of Component and Documentation Layers

Setting Up an Inner Layer

Setting Up an Outer Layer

Component Layer Associations Dialog Box

Hiding or Displaying Non-electrical Layers in Layer Lists

Default SailWind Layout designs come with many available layers and these layers all appear in the layer list of certain dialogs. If your design only uses a few layers, the result is a large number of unused layers appearing in the layer list. You can hide those unused layers from view to shorten the layer list wherever it displays—for example, the Layer list of the Standard Toolbar, the Layers Setup and Display Colors dialog boxes.

Tip
For instructions on hiding layers in the PCB design, see "Hiding Layers".

Restrictions and Limitations

You can hide only non-electrical layers from layer lists.

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 column. Click a column heading to sort by that column's values. Click the same column heading again to re-sort in the reverse order.

3. Click **OK** to return to the Layers Setup dialog box.

Setting Layer Parameters

Layer parameters are provided to confirm design compliance and calculate impedance. For example, when you verify your design, the electrodynamic check uses such information to confirm design compliance as electrical layer and dielectric material layer thickness and dielectric constant information defined in the Layer Parameter Setup dialog box. Besides, more layer parameters are provided for impedance calculation.

Traces on high-speed printed circuit boards can act like transmission lines that “broadcast” interference to adjacent conductors. With 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.

Prerequisites

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 **Advanced** button. The Layer Parameter Setup dialog box opens.
2. Set layer parameters as needed.
    - For each dielectric material layer, double-click the Type cell to select whether the “layer” is a Prepreg or Substrate layer.
    - For dielectric material and coating layers, fields Dk and Df are editable. For electrical layers, you can set such parameters as Drum SR, Matte SR, Ref.Plane, SE Width, DIFF Width, and DIFF Spacing.

    For detailed parameter description, refer to the table below.

Table 73. Layer parameter description

Field	Description
#	Displays the electrical layer number, which corresponds to **Lev.** in the Layers Setup Dialog Box.
Name	Displays the name of the layer, which corresponds to that defined in the Layers Setup Dialog Box.

Field	Description
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.
Material	Specifies the material of each layer. You can choose one from the drop-down list.
Thickness	Specifies the thickness of the layer. The input range varies from the material type and current design units: • For dielectric material, its thickness must be within 5.08 mm/200 mil/0.2 inch. • For copper, its thickness must be within 0.25 mm/10 mil/0.2 inch. **Tip:** If no coating is required, set thickness to zero.
Dk	Specifies the dielectric constant value.
Df	Specifies the dielectric loss tangent. Range: 0—1
Drum SR	Specifies the roughness of the drum side. The input range varies from the current design units: • Mils: 0—1 • Metric: 0—0.02540 • Inches: 0—0.001
Matte SR	Specifies the roughness of the matte side. The input range is silmilar to that of the drum side.
Ref. Plane	Specifies the electrical layer as reference layer or not. **Note:** Plane layers specified in the Layers Setup Dialog Box are set to reference layers by default.
SE Width	Specifies the single-ended trace width. The input range varies from the current design units as follows and the same rule applies to that of DIFF Width as well as DIFF Space. • Mils: 0—100 • Metric: 0—2.54 • Inches: 0—0.001
SE Z0 (ohm)	Displays the single-ended impedance calculated by clicking **Calculate Impedance**.
DIFF Width	Specifies the trace width of differential pairs.
DIFF Space	Specifies the spacing between differential pairs.

Field	Description
DIFF Z0 (ohm)	Displays the differential impedance calculated by clicking **Calculate Impedance**.
Edit	Makes the selected cell available for editing. **Exception:** Cells that are grayed out or blank 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.
Calculate Impedance	Clicks to start impedance calculation.
Import	Clicks to import a XML file.
Export	Clicks to export a XML file.

3. 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

Related Topics

Verify the Design

Setup of High Speed (Electrodynamic) Checking

Setup of EDC Parameters

Unassigning a Netname from a Plane Layer

You can remove a netname from its assignment to 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 in the Assigned Nets list.
4. Click **Remove**. The netname is moved back to the All Nets list.
5. Click **OK**.

Reassigning Electrical Layers

When you alter the number of electrical layers, you must reassign the old layers to locations in the new layer setup.

Restrictions and Limitations

- When reassigning layers, component keepouts are ignored. They remain on the layer on which you created them.
- You cannot modify the layer definition for electrical layers when a physical design reuse exists on the board.
- 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.

Results

The following layer properties are changed during layer reassignment:

- Name
- Plane status
- Layer (routing) direction
- Dielectric constant and thickness (associated with upper copper layer)

- Associated nonelectrical layers (for top and bottom)
- List of assigned nets
- Colors, including outline colors

Tip
Because both layer names and colors are reassigned, an object remains in the same color after reassignment.

The following objects have an assigned layer and are reassigned during layer reassignment:

- Traces
- 2D lines, free and in decal (part outlines)
- Copper, open and closed, free and in decals, but not pin-associated copper
- Keepouts, free and in decals
- Copper cutouts, free, associated, and in decals
- Texts, free, combined with 2D lines, and in decals
- Attribute labels
- Pour outlines and plane areas
- Pour or plane area hatch outlines and hatch voids
- Conditional clearance rules
- Layer mask in routing rules

The following objects have an assigned layer and are not reassigned during layer reassignment:

Tip
Before deleting an electrical layer, make sure that you first delete all of the following objects from, and any references to, the layer.

- Pad definitions on absolute layers in pad stacks (for decals and vias)
- Pin-associated copper in decals

- Start and end layers for partial vias
- Drill pairs

Chapter 15
Via Setup

Read the following topics to learn more about the setup of vias.

For information about adding virtual pins to nets in your design, see Virtual Pins.

Defining Drill Pairs

A *drill pair* is a pair of layer numbers specifying the starting layer and ending layer of a drilled hole. If you want to use one or more partial vias in your design, you must define a drill pair for each layer change specified in a via type.

Define drill pairs before you define partial via types or begin routing; after you define your drill pairs, SailWind Layout prevents you from performing actions (such as defining a via type, or making a layer change during routing) that do not conform to a defined drill pair.

If you are using both through-hole and partial vias in your design, you must also create an “all-layers” drill pair for the through-hole default via type; after any partial via type has been created, SailWind Layout requires that a drill pair be defined for all via types, including the through-hole default.

Procedure

1. Click the **Setup > Drill Pairs** menu item.
2. In the Drill Pairs Setup Dialog Box, 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. Repeat Steps 2 through 4 to add any additional drill pairs. You can change the starting or ending layer of a drill pair, or delete drill pairs.
6. Click **OK** to save the 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 Type

Creating a Through-hole Via Type

Drill Pairs Setup Dialog Box

Creating a Non-Drilled Single-Layer Via Type for Virtual Pins

Creation of Vias

You can create three basic via types: Through-hole, Partial, and Non-drilled Single-Layer (the Partial via type includes both blind and buried vias).

Creating a Through-hole Via Type

You can create a through-hole via type to use for vias and/or virtual pins.

Prerequisites

If any partial drill pairs exist in your design, an “all-layers” drill pair must also exist before you perform this procedure. See “Defining Drill Pairs”.

Procedure

1. Click the **Setup > Pad Stacks** menu item.
2. In the Pad Stacks Properties Dialog Box, in the Pad Stack Type area, click Via.
3. Click **Add Via** to add a line to the Decal name list. A list of all previously defined via types appears in the Decal Name list.
4. In the Vias area, in the Name box, type the name of the new via.
5. Click Through to indicate that this is a Through-hole via type.
6. If needed, customize the pads for individual layers, as follows:
    a. Select the layer from the “Sh: Sz: Layer: list”. For more information, see “Pad Stack Default Layers” and “Control of Solder Mask and Paste Mask on page 209”.
    b. In the Parameters area, specify the settings on all three **Pad**, **Thermal**, and **Antipad** tabs. The style of pad used is selected automatically in the design depending on the situation. The thermal and antipad styles are used when the pad is located within a plane. For more informations, see “Design Rule Versus Pad Stack - Thermals and Antipads” on page 797.
    c. Repeat Steps a and b for other layers, as needed.
7. Click **OK** to save the new via type.

Related Topics

Pad Sizes and Pad Stacks

Creating a Partial Via Type

Editing a Via Type

Creating a Non-Drilled Single-Layer Via Type for Virtual Pins

Defining Drill Pairs

Pad Stacks Properties Dialog Box

Creating a Partial Via Type

You can create a partial via type to use for vias and/or virtual pins.

Partial vias can be either a blind via or buried via.

For more information, see “Creating a Through-hole Via Type” and “Creating a Non-Drilled Single-Layer Via Type for Virtual Pins”.

Prerequisites

A drill pair specifying the starting and ending layers for the new partial via type must exist. For more information, see “Defining Drill Pairs”.

Procedure

1. Click the **Setup > Pad Stacks** menu item.
2. In the Pad Stacks Properties Dialog Box, in the Pad Stack Type area, click Via.
3. Click **Add Via** to add a line to the Decal name list. A list of all previously defined via types appears in the Decal Name list.
4. In the Vias area, type the name of the via type in the Name box.
5. Click Partial to indicate that this is a blind or buried via type.
6. Select the starting and ending layers from the Start Layer and End Layer lists.
7. If necessary, customize the pads for individual layers, as follows:
    a. Select the layer from the “Sh: Sz: Layer:” list. For more information, see “Pad Stack Default Layers” and “Control of Solder Mask and Paste Mask”.
    b. In the Parameters area, specify the settings on all three **Pad**, **Thermal**, and **Antipad** tabs. The style of pad used is selected automatically in the design depending on the situation. The thermal and antipad styles are used when the pad is located within a plane. For more information, see “Design Rule Versus Pad Stack - Thermals and Antipads” on page 797.
    c. Repeat Steps a and b for other layers, as needed.
8. If the partial via will be laser drilled, you can select the Laser check box to use the “Laser drill oversize” value in the Options Dialog Box, Design Category, for verifying spacing.
9. Click **OK** to save the new via type.

Results

If you are unable to use the new partial via type, check the following in the Default Routing Rules:

- Is your new via type 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

Creating a Through-hole Via Type

Editing a Via Type

Creating a Non-Drilled Single-Layer Via Type for Virtual Pins

Defining Drill Pairs

Pad Stacks Properties Dialog Box

Creating a Non-Drilled Single-Layer Via Type for Virtual Pins

You can create a non-drilled (single-layer) pad via type to be used by virtual pins. You must create one of these for each layer on which you will use virtual pins. Use the Through via setting to create a single pad non-drilled via type.

Procedure

1. Click the **Setup > Pad Stacks** menu item.
2. In the Pad Stacks Properties Dialog Box, in the Pad Stack Type area, click Via.
3. Click **Add Via** to add a line to the Decal name list. A list of all previously defined via types appears in the Decal Name list.
4. In the Vias area, in the Name box, type the name of the new via type.
5. Click Through. You must use this setting to create a non-drilled via type.
6. From the “Sh: Sz: Layer:” list, select the layer on which virtual pins using this via type will be installed. For more information, see “Pad Stack Default Layers”.
7. In the Parameters area, specify only the settings on the Pad tab. (The thermal and antipad styles are used only when the pad is located within a plane, and virtual pins are not allowed in plane nets.) For more information, see “Design Rule Versus Pad Stack - Thermals and Antipads” on page 797.
8. In the Drill size box, type 0 (zero).
9. For all other layers of the pad stack, in the Size box, type 0 (zero).
10. Click **OK** to save the new via type.

Related Topics

Creating a Through-hole Via Type

Creating a Partial Via Type

Adding a Virtual Pin to a Net

Editing a Via Type

You can make changes to a via type and apply the changes to all objects that use that via type.

Procedure

1. Select a via or virtual pin.
2. Right-click and click the **Properties** popup menu item.
3. In the Via Properties on page 1779 or Virtual Pin Properties on page 1789 dialog box, click the **Pad Stack** button.
4. In the Pad Stacks Properties Dialog Box, click the via type to edit in the Decal Name list.
5. Change via settings as needed. Use the Parameters and Drill Size areas to reset the size and shape options.
6. When you finish making changes, click **OK**.
7. Click **Yes** to change all vias of the selected type.
8. Click **OK** to close the Via or Virtual Pin properties dialog box.

Related Topics

Editing Pad Stacks

Creating a Through-hole Via Type

Creating a Partial Via Type

Creating a Non-Drilled Single-Layer Via Type for Virtual Pins

Defining Drill Pairs

Pad Stacks Properties Dialog Box

Deleting a Via Type

Delete via types by deleting a via definition in the pad stacks.

Procedure

1. Click the **Setup > Pad Stacks** menu item.
2. In the Pad Stacks Properties Dialog Box, in the Pad Stack Type area click Via.
3. Click a via type name in the Decal Name list.
4. Click **Delete Via**.
5. Click **OK**.

Related Topics

Editing Pad Stacks

Creating a Through-hole Via Type

Creating a Partial Via Type

Creating a Non-Drilled Single-Layer Via Type for Virtual Pins

Pad Stacks Properties Dialog Box

Tented Vias With Solder Mask

You can cover vias with solder mask (a process known as “tenting” the via.)

While using an attribute to control soldermask openings is a quick method, you are unable to see the shapes in the design since they are generated when creating the gerber files. Adding custom solder mask shapes to pad stacks is more labor intensive but also allows the most control and customization of each pad stack. See the topics below for more information.

Tenting Vias By Adding a Custom Solder Mask Shape
Tenting Vias By Adding an Attribute
Verifying Via Tenting Results

Tenting Vias By Adding a Custom Solder Mask Shape

Use the Pad Stacks dialog box to add a custom solder mask shape to one or both solder mask layers. Adding solder mask shapes to all pad stacks allows you to see solder mask openings in the design and also see when there are no openings in the solder mask. The Solder Mask layer also displays drill hole locations.

Procedure

1. On the **Setup** menu, click **Pad Stacks**.
2. In the Pad Stacks Properties Dialog Box, in the Pad Stack Type area, click **Via**.
3. Add a solder mask layer to the via type you want to be tented, as follows:
    a. In the Decal name list click the name of the via type.
    b. In the Vias area Name box, append “_OPEN” to the via type 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 Bottom, and click **OK**.
    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 type you just edited, as follows:
    a. With the <VIA_NAME>_ OPEN via type selected, click the **Add Via** button, and click **Yes** when prompted to apply pad stacks information and to change all vias of type <VIA_NAME>_ OPEN.
    b. In the Vias area, type “<ORIGINAL_VIA_NAME>_TENTED”.
    c. In the Sh: Sz: Layer area, select a Solder Mask layer.
    d. With the 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 and click the **Properties** popup menu item.

   c. In the Via Properties Dialog Box, in the Via Name list, select the name of the new tented via type.

   d. Click **OK**.

Results

To verify your results, perform the procedure in Verifying Via Tenting Results.

Tenting Vias By Adding an Attribute

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.

CAUTION:
If a solder mask shape exists in a solder mask layer of the via type pad stack, and the via type also has an assigned “CAM.Solder mask.Adjust” value, and you apply both values when creating the CAM document, the values are combined. See “Mask Hierarchy on page 209”.

Procedure

1. In the design, select the vias you want to be tented.

2. Right-click and click the **Attribute** popup menu item.

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 the Enter key. For example, enter -20 to tent a 20 mil via pad.

6. Click **Close**.

Results

To verify your results, perform the procedure in Verifying Via Tenting Results.

Verifying Via Tenting Results

You can verify the results of via tenting operations by previewing the solder mask area.

Procedure

1. Click the **File > CAM** menu item.
2. In the Define CAM Documents Dialog Box, click the **Add** button.
3. In the Add/Edit Document Dialog Box, from the Document Type list, select Solder Mask.
4. In the Layer Association dialog box, select the top/bottom layer.
5. In the Customize Document area, 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.

Troubleshooting Via Usage

Keep certain tips in mind when you try to place a via, and it fails, or you receive the message, “Via usage for this net is restricted.”

- When partial vias on page 330 are used, you must set up a drill pair on page 327 for all partial and through hole vias on page 329 in the design.
- In the Routing Rules Dialog Box, you can specify the vias that are to be used at all levels of the design rules hierarchy on page 404. This can, for example, prevent you from using the wrong via on a specific net or pin pair.

Chapter 16
Electrical Nets

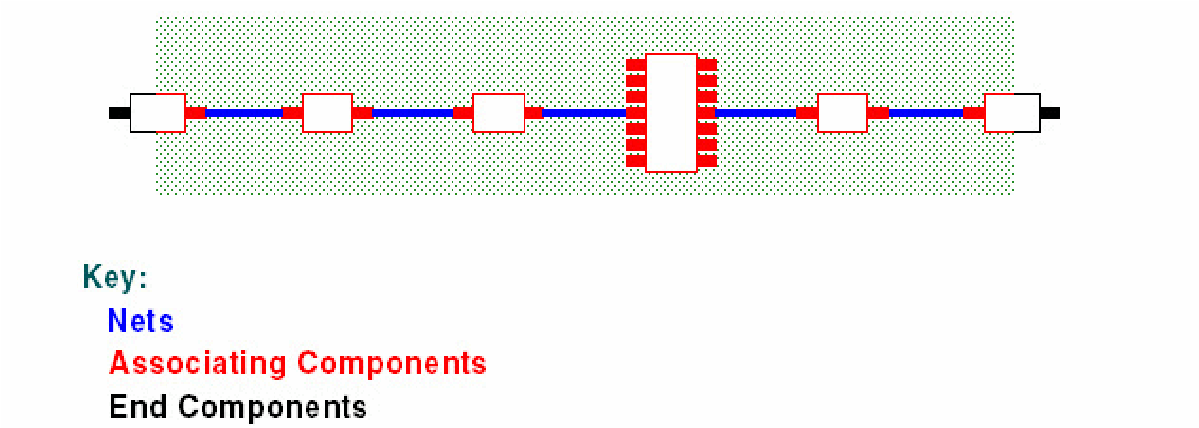
Read the topics that follow to learn how to create and manage electrical nets.

Electrical Net Creation

You can associate an array of nets joined by discrete components, creating an electrical net to which you can apply length, differential pair and matched length rules as you would to a single net. The length of an electrical net is the combined lengths of the nets and discrete components of which it is composed.

The figure below shows a simple example of an electrical net.

Figure 26. Electrical Net Components and Nets



There are three methods you can use to create and modify electrical nets.

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 Electrical Net Creation" on page 348.

Creating Electrical Nets by Selecting the Nets
Creating Electrical Nets by Selecting the Components
Creating Electrical Nets by Component Refdes Prefix

Creating Electrical Nets by Selecting the Nets

You can create electrical nets by selecting nets and associating them.

Restrictions and Limitations

- Plane nets cannot be part of an electrical net.
- Components through which an electrical net passes must be either discrete two-pin components, or multiple pin components if all the pins connect to a gate and each gate has exactly two pins.

Procedure

1. Click the **Setup > Electrical Nets** menu item.
2. In the Electrical Nets on page 1361 dialog box, set the "Maximum net count per electrical net" and "Maximum non-plane-net pin count" values for your design.
3. In the workspace, select the nets you want make into electrical nets.
4. Right-click and click the **Electrical Nets** popup menu item. As an alternative, in the Net Properties dialog box on page 1487, select the "Create electrical net" check box. The selected nets are made into electrical nets, and the Net Properties "Create electrical net" check box is selected for all the selected nets.
5. Whenever electrical nets are created, deleted or changed, all electrical nets in the design are regenerated. Verify that the results of an electrical nets action are what you expected by reading the messages in the Output Window.

Results

Are the results different from what you expected?:

- An expected electrical net might have been split or truncated, or not created at all, if it surpassed one of the maximum thresholds set in the Electrical Nets on page 1361 dialog box.
- Some nets cannot be made into electrical nets; for example, two selected nets from opposite sides of the board will have the Create electrical net check box selected, but no electrical net will be created.
- Mixed-method restriction — if a component to which one of the selected nets is attached has been excluded from electrical net creation, no electrical net can go through it. (A component or net is excluded from electrical net creation if its "Create electrical nets" and "Allow electrical net creation..." check boxes are both cleared.) For more information, see "“Conditions Governing Electrical Net Creation” on page 348".

Related Topics

Canceling Electrical Net Creation by Nets

Excluding Nets from Electrical Net Creation

Creating Electrical Nets by Selecting the Components

To base the creation of electrical nets on the selected components, use the connected physical nets on the components to create your electrical nets.

Restrictions and Limitations

- Plane nets cannot be part of an electrical net.
- Components through which an electrical net passes must be either discrete two-pin components, or multiple pin components if all the pins connect to a gate and each gate has exactly two pins.

Procedure

1. Click the **Setup > Electrical Nets** menu item.
2. In the Electrical Nets on page 1361 dialog box, set the “Maximum net count per electrical net” and “Maximum non-plane-net pin count” values for your design.
3. In the workspace, select the components whose nets you want to make into an electrical net.
4. Right-click and click the **Electrical Nets** popup menu item. As an alternative, in the Component Properties Dialog Box, select the “Create electrical net” check box. The nets attached to the selected components are made into an electrical net, and the Component Properties “Create electrical net” property is set for all the selected components.
5. Whenever electrical nets are created, deleted or changed, all electrical nets in the design are regenerated. Verify that the results of an electrical nets action are what you expected by reading the messages in the Output Window.

Results

Are the results different from what you expected?

- An expected electrical net might have been split or truncated, or not created at all, if it surpassed one of the maximum thresholds set in the Electrical Nets on page 1361 dialog box.
- Mixed-method restriction — if a net attached to a selected component is excluded from electrical net creation, it cannot be included in an electrical net. (A component or net is excluded from electrical net creation if its “Create electrical net” and “Allow electrical net creation...” check boxes are both cleared.) For more information, see “Conditions Governing Electrical Net Creation” on page 348.

Related Topics

Excluding Components from Electrical Net Creation

Canceling Electrical Net Creation by Components

Creating Electrical Nets by Component Refdes Prefix

You can create electrical nets automatically by specifying the refdes prefixes of the components through which the nets pass in the Electrical Nets dialog box.

For more information, see Electrical Nets Dialog Box.

Restrictions and Limitations

- Plane nets cannot be part of an electrical net.
- Components through which an electrical net passes must be either discrete two-pin components, or multiple pin components if all the pins connect to a gate and each gate has exactly two pins.

Procedure

1. Click the **Setup > Electrical Nets** menu item.
2. In the Electrical Nets on page 1361 dialog box, set the "Maximum net count per electrical net" and "Maximum non-plane-net pin count" values for your design.
3. In the "Discrete component prefixes" area, enter the refdes prefixes of the components that should create electrical nets. This creates electrical nets for all components that have the specified refdes prefixes. In order to constrain the number of electrical nets that are created, you could assign unique reference designators to those components that should create electrical nets, or simply create electrical nets by selecting the components in the design on page 339.
4. Click **OK**. Electrical nets are regenerated based on the new or changed set of specified refdes prefixes.
5. Whenever electrical nets are created, deleted or changed, all electrical nets in the design are regenerated. Verify that the results of an electrical nets action are what you expected by reading the messages in the Output Window.

   Rules (min/max length, matched length groups, differential pairs) for electrical nets that are unchanged in the regeneration are kept. Electrical 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.

Results

Are the results different from what you expected?

- An expected electrical net might have been split or truncated, or not created at all, if it surpassed one of the maximum thresholds set in the Electrical Nets on page 1361 dialog box.
- Mixed-method restriction — if any of the selected components, or any net attached to a selected component, has been excluded from electrical net creation, it cannot be included in an electrical net. (A component or net is excluded from electrical net creation if its "Create electrical net" and "Allow electrical net creation..." check boxes are both cleared.)

  For more information, see the "Conditions Governing Electrical Net Creation" on page 348.

Related Topics

Deletion of Electrical Nets

Excluding Nets from Electrical Net Creation

Excluding Components from Electrical Net Creation

Deletion of Electrical Nets

You can delete electrical nets manually—or automatically, if Ref. Des. Prefix assignment in the Electrical Nets dialog box created them.

Deleting Electrical Nets Manually

Select electrical nets in the design to delete them.

Procedure

1. In the workspace, select the electrical nets on page 345you want to delete.
2. Right-click and click the **Delete Electrical Net** popup menu item. Deleting an electrical net does not delete the physical nets.

Results

The Create electrical net check box is cleared for all the nets of the selected electrical nets. If any electrical nets were created by component (including by refdes prefix), for those nets the “Allow electrical net creation by component” check box is cleared as well.

Related Topics

Canceling Electrical Net Creation by Nets

Excluding Nets from Electrical Net Creation

Deleting Electrical Nets Automatically

You can “automatically” delete a group of electrical nets that the refdes prefix method created automatically.

Procedure

1. Click the **Setup > Electrical Nets** menu item.
2. In the Electrical Nets on page 1361 dialog box, delete the refdes prefixes of the associating components.
3. Click **OK**.
4. Whenever electrical nets are created, deleted or changed, all electrical nets in the design are regenerated. Verify that the results of an electrical nets operation are what you expected by reading the messages in the Output Window.

Results

The electrical nets created 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 “Create electrical net” check box selected, the physical nets will remain in an electrical net.
- If a component having the deleted refdes prefix has the “Allow electrical net creation by refdes prefix and by net” check box selected, and both nets attached to a two-pin component (or to a single gate of a multiple-pin component) have the “Create electrical net” check box selected, those nets will remain in an electrical net.

  For more information, see ““Conditions Governing Electrical Net Creation” on page 348”.

Related Topics

Excluding Components from Electrical Net Creation

Excluding Nets from Electrical Net Creation

You can exclude individual nets from inclusion in any electrical net.

Clear both the Net Properties “Create electrical net” check box and the “Allow electrical net creation by component” check box, removing the selected nets from any current electrical net, and preventing electrical net creation by component (including by refdes prefix). For more information, see “Conditions Governing Electrical Net Creation” on page 348.

Procedure

1. In the workspace, select the net(s) you want to exclude from electrical net creation.
2. Right-click and click the **Disable Electrical Net Creation** popup menu item. As an alternative, in the Net Properties dialog box, clear both the “Create electrical net” and “Allow electrical net creation by components” check boxes.
3. Whenever electrical nets are created, deleted or changed, all electrical nets in the design are regenerated. Verify that the results of an electrical nets operation are what you expected by reading the messages in the Output Window.

Results

The selected nets are removed from existing electrical nets, and the Net Properties “Create electrical net” and “Allow electrical net creation by components” check boxes are cleared for all the selected nets.

Related Topics

Excluding Components from Electrical Net Creation

Canceling Electrical Net Creation by Nets

Excluding Components from Electrical Net Creation

You can prevent an individual component (a component through which an electrical net passes) from creating electrical nets.

If you exclude a component that is not already part of an electrical net, it cannot become part of an electrical net created by net or by refdes prefix. In addition, if the component is already part of an electrical net, it is removed from the electrical net.

Clear both the Component Properties “Create electrical net” and “Allow electrical net creation...” check boxes, preventing the selected components from creating electrical nets. For more information, see “Conditions Governing Electrical Net Creation” on page 348.

Procedure

1. In the workspace, select the components you want to exclude from electrical nets.
2. Right-click and click the **Disable Electrical Net Creation** popup menu item. As an alternative, in the Component Properties Dialog Box, clear both the “Create electrical net” and “Allow electrical net creation by refdes prefix or by net” check boxes.
3. The selected components are excluded from electrical net creation. If a component is part of an existing electrical net, it is removed, that is, the electrical net no longer goes through it. Removing a component from an electrical net splits, truncates, or deletes the electrical net, depending upon its configuration and the position of the removed component.
4. Whenever electrical nets are created, deleted or changed, all electrical nets in the design are regenerated. Verify that the results of an electrical nets operation are what you expected by reading the messages in the Output Window.

Results

Are the results different from what you expected? Nets attached to the excluded component may still be part of an electrical net through other components.

Related Topics

Excluding Nets from Electrical Net Creation

Canceling Electrical Net Creation by Nets

You can cancel or prevent participating nets from creating an electrical net.

Procedure

1. In the workspace or Project Explorer, select the nets you want to remove from electrical nets.
2. Right-click and click the **Properties** popup menu item.
3. In the Net Properties dialog box on page 1487, in the Electrical Nets area, clear the Create electrical net check box. The selected nets are removed from existing electrical nets.

Tip
If you want to exclude the nets from electrical nets altogether, also clear the “Allow electrical net creation by components” check box.

4. Whenever electrical nets are created, deleted or changed, all electrical nets in the design are regenerated. Verify that the results of an electrical nets operation are what you expected by reading the messages in the Output Window.

Results

Are the results different from what you expected? Mixed-method restriction — nets that are also part of an electrical net by the component and/or the refdes prefix method remain part of an electrical net.

Related Topics

Conditions Governing Electrical Net Creation

Canceling Electrical Net Creation by Components

You can cancel or prevent components from creating an electrical net that passes through them.

Procedure

1. In the workspace or Project Explorer, select the components whose electrical net creation you want to cancel.
2. Right-click and click the **Properties** popup menu item.
3. In the Component Properties Dialog Box, in the Electrical Nets area, clear the Create electrical net check box. Electrical net creation by the selected component is canceled.

Tip
If you want to exclude the components from electrical nets any method, also clear the “Allow electrical net creation by refdes prefix and nets” check box.

4. Whenever electrical nets are created, deleted or changed, all electrical nets in the design are regenerated. So you should always verify that the results of an electrical nets operation are what you expected by reading the messages in the Output Window.

Results

Are the results different from what you expected? Mixed-method restriction — nets that are attached to selected components, and are also part of electrical nets by the net and/or the refdes prefix method remain in the electrical net through the selected components.

Selecting Electrical Nets

Use the shortcut menu to select an electrical net.

Procedure

1. In the workspace, select one of the nets in the electrical net you want to select.
2. Right-click and click the **Select Electrical Net** popup menu item.

Creating a Matched Length Group of Electrical Nets

Gather electrical nets into a matched length group and apply rules to the group.

Procedure

1. Click the **Setup > Design Rules** menu item.
2. In the Rules Dialog Box, click **Electrical Nets**.
3. In the Electrical Net Rules Dialog Box, select the electrical nets you want to include in the matched length group, and click the **HiSpeed** button.
4. Make appropriate settings in the HiSpeed Rules Dialog Box, and click **OK**.

Creating a Differential Pair of Electrical Nets

Add electrical nets into a differential pair and apply rules to the pair.

Procedure

1. Click the **Setup > Design Rules** menu item.
2. In the Rules Dialog Box, click **Differential Pairs**.
3. In the Differential Pairs Dialog Box, select the **Electrical nets** tab.
4. Select the electrical nets to make up the diff pair:
    a. Select the first electrical net from the Available list, and click the top **Select** button.
    b. Select the second electrical net from the Available list, and click the bottom **Select** button.
5. Click **Add**.
6. Set the properties of the diff pair as appropriate, and click **OK**.

Creating or Modifying Electrical Net Design Rules

Set or modify electrical net design rules. By default, electrical nets have no rules.

Procedure

1. You can assign rules to one or more electrical nets at a time. There are two ways to select the electrical net(s) before accessing the rule categories.

   - Select electrical nets in the design:

     i. In the design area, select one of the nets in the electrical net to which you want to apply rules. 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 the **Select Electrical Net** popup menu item.

     iii. Right-click and click the **Show Rules** popup menu item.

   - Select electrical nets from the list in the Electrical Net Rules dialog box:

     i. Click the **Setup > Design Rules** menu item.

     ii. In the Rules Dialog Box, click **Electrical Nets**.

     iii. In the Electrical Net Rules Dialog Box, in the Electrical Nets box, click one or more electrical nets. To display all electrical nets, clear the Show Electrical Nets with Rules check box.

2. Click the **HiSpeed** button.

3. In the HiSpeed Rules dialog box for electrical nets on page 1405, specify Length and/or Matching rules for the selected electrical net(s).

Results

After you define rules, the HiSpeed Rules indicator (H) appears beside the selected electrical nets in the Electrical Nets list.

Clearing Electrical Net Rules

Select an electrical net and return its rules setup to the default. By default, electrical nets have no rules.

Procedure

1. Click the **Setup > Design Rules** menu item.

2. In the Rules Dialog Box, click **Electrical Nets**.

3. In the Electrical Net Rules Dialog Box, select one or more electrical nets in the Electrical Nets list, click **Default**, and then click **Yes**. The Default button is unavailable if the selected electrical net has no rules assigned to it. You know you have successfully reset the net rules if the HiSpeed Rules indicator (H) is removed, and the Default button is no longer available.

4. Close all rules dialog boxes to return to the design.

Conditions Governing Electrical Net Creation

You can create electrical nets using one of three methods. To correctly maintain settings, avoid using a mixture of methods.

Settings That Control Electrical Net Creation

The following Net Properties dialog box on page 1487 settings control the creation of electrical nets:

- **Create electrical net check box** — creates electrical nets and is cleared by default for all nets.
- **Allow electrical net creation by components check box** — allows the selected net to be added to an electrical net by other methods, and is selected by default for all nets. So by default, you can create an electrical net by any of the three methods.

The following Component Properties dialog box on page 1191 settings control the creation of electrical nets:

- **Create electrical net check box** — creates electrical nets, and is cleared by default for all components.
- **Allow electrical net creation by refdes prefix and nets check box** — allows the selected component to be added to an electrical net by other methods, and is selected by default for components. So by default, you can create an electrical net by any of the three methods.

The following "Electrical Nets Dialog Box" on page 1361 Refdes prefixes and settings control the creation of electrical nets:

- **Maximum net count per electrical net value** — default value is 5.
- **Maximum non-plane-net pin count value** — default value is 25. Virtual pins are not included in a net's pin count.

Conditions Required for Electrical Net Creation

A discrete component and its attached nets become part of an electrical 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 Properties "Create electrical net" check box is selected and neither attached net is excluded from electrical net creation. (A component or net is excluded from electrical net creation if its "Create electrical net" and "Allow electrical net creation..." check boxes are both cleared.)
    - Net Properties "Create electrical net" check box is selected for both nets and the component is not excluded from electrical net creation (A component or net is excluded from electrical net creation if its "Create electrical net" and "Allow electrical net creation ..." check boxes are both cleared.)

- Component “Allow electrical net creation...” check box is selected and no attached net is excluded from electrical net creation and the component’s refdes prefix is specified in the Electrical Nets dialog box. (A component or net is excluded from electrical net creation if its “Create electrical net” and “Allow electrical net creation...” check boxes are both cleared.)

2. The component, its nets, and the potential electrical net it will be a part of, conform to the “Maximum net count per electrical net” and “Maximum non-plane-net pin count” settings in the “Electrical Nets Dialog Box” on page 1361.

Chapter 17
Setting Options

Read the topics that follow to learn more about multi-step procedures for setting options in SailWind Layout.

Creating a Backup File

SailWind Layout automatically backs up your files based on the settings you choose.

Procedure

1. Click the **Tools > Options** menu item > **Global** category > Backups subcategory on page 1527.
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. Backup files are named *<design_name>#.pcb*, 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**. 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. For 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. Clear the check box if you want your backup files in one, common backup directory.

Results

Table 74. 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.

Table 74. Backup File Creation (continued)

If this is selected:		The Backup File is Saved
	X	in one common directory using the design name.

Setting the Display Grid

You can set a visible dot grid in the design area to aid in component placement.

Procedure

1. Click the **Tools > Options** menu item > Grids and Snap category > Grids subcategory on page 1537.
2. In the Display grid area, type a value for the X and Y values. You can set this grid to the same setting as the design grid or give it a different set of values like a multiple of the design grid. For example if your design grid is 5,5 mils, set the grid to 20,20. You will see the grid when using a higher magnification and can easily perceive the invisible grid locations between the dot grid.

Tip
The fastest and most convenient way to change this grid is to use the Modeless Command on page 83 instead of using the Options dialog box; type GD <x> <y> and press the Enter key. For example, GD 100 100.

Setting the Design Grid

Set the design grid for placement of components, traces and drafting objects.

Procedure

1. Click the **Tools > Options**menu item > Grids and Snap category > Grids subcategory on page 1537.
2. In the Design grid area, type a value for the X and Y values. To improve routing, set the design grid to the minimum trace width + minimum trace to trace spacing or a multiple of this value.

Tip
The fastest and most convenient way to change this grid is to use the Modeless Command on page 83 instead of using the Options dialog box; type GR <x> <y> and press the Enter key. For example, GR 100 100. To set the design and via grid to the same setting, type G <x> <y> and press the Enter key.

Setting Up a Polar Grid

Before using Radial Move, set up a polar grid and the move options.

Procedure

1. Click the **Tools > Options** menu item > Grids and Snap category > Grids subcategory on page 1537.
2. Click **Radial Move Setup**.
3. In the Radial Move Setup Dialog Box, on page 1652set 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 click the **Setup > Set Origin** menu item to change the location of the polar grid origin, but you must Redraw to see the change.

Related Topics

Component Arrays

Moving Design Objects Radially

DRC and the Via Stitching and Shielding Operations

The Design Rule Checking setting determines how the Via Stitch and Add Via Shield operations can add vias.

The Design Rule Checking (On-line DRC on page 1503) 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?”

Chapter 18
Controlling Attributes

Read the topics that follow to learn more about creating, modifying, managing and using attributes in SailWind Layout.

Attributes

You can use *attributes* to associate information with an object in the design. Attributes are made of two parts: an attribute name and its corresponding value.

For example, you can create an IsSMD attribute to keep track of which parts are SMD and which are not.

You can assign attributes to the following objects:

- PCB (the board)
- Part Type
- Decal
- Part (component; instance of a part in the design)
- Net Class
- Net
- Pin, including jumper pin
- Via

Every attribute you add to a design is added to the Attribute Dictionary. Attributes are assigned for the entire design. Once you name an attribute and set its properties, that name and those properties apply throughout the design.

Attributes Workflow

Create and manage attributes using a structured workflow.

This is the general process for adding attributes to a design:

1. Create attributes. You can create attributes using the Attribute Dictionary. You can also create attributes and assign them to objects using the Object Attributes Dialog Box; however, you cannot modify the properties of the attribute with this dialog box. Therefore, it is easier to create all your attributes in the Attribute Dictionary. Define the attribute properties. You must set the kind of value the attribute should have, the design objects to which you want to assign the attribute, and the hierarchy for the attribute. For more information, see the Creating Attributes for the Design topic.

2. Assign attributes to objects in the design. For more information, see the Design Object Attributes topic.

3. You can assign attributes to multiple objects of multiple types. For more information, see the Attribute Manager on page 392 topic.

4. When it is necessary to change the attributes assigned to objects use the Properties dialog boxes.

Attribute Hierarchy

The attribute hierarchy is the search order in which SailWind Layout searches the database to find an attribute value. You can assign attributes using the default hierarchy or you can change the hierarchy, creating your own search order. The lowest level to which you can assign an attribute is the PCB. An attribute applied to the PCB applies to every object on the board, unless you set an attribute at a higher hierarchy level. When you set an attribute at a higher level in the hierarchy, it overrides the PCB level.

The levels in the attribute hierarchy are object dependent, that is, each object has a different hierarchy. You can modify the hierarchy for every attribute using the Objects tab on page 1111 on the Attribute Properties dialog box. The following table shows the attribute default hierarchy for each object type.

Table 75. Attribute Hierarchy

For Object	Default Hierarchy is
PCB	None. This is the lowest level. Attributes assigned at other levels in this hierarchy override attributes assigned at this level.
Part Type	Part Type, PCB
Decal	Decal, PCB
Part (Component)	Part, Decal, Part Type, PCB
Net Class	Net Class, PCB
Net	Net, Net Class, PCB
Via	Via, Net, Net Class, PCB

Table 75. Attribute Hierarchy (continued)

For Object	Default Hierarchy is
Pin	Pin, Net, Net Class, Part, Decal, Part Type, PCB

If you assign attributes to multiple levels and then delete an attribute, the attribute from the next level in the hierarchy is assumed. For example, if you assign an attribute at the Part Type level and at the PCB level, and you delete the attribute at the Part Type level, the attribute at the PCB level is then applied to the component.

Passing Attributes

You can pass attributes between SailWind Layout and other programs. SailWind Layout provides a default set of units (and unit prefixes) acceptable for input and for output among programs.

For more information, see Default Units.

Table 76. Passing Attributes to Other Programs

Program	Attribute Passing
SailWind Logic	You can pass the Value and Tolerance part type attributes in SailWind Logic forward to SailWind Layout as part of a netlist. SailWind Layout can accept the attributes from the netlist.
IDF	The default attribute Geometry.Height is automatically exported to IDF. This attribute replaces the ZHEIGHT functionality used in previous versions of SailWind Layout. For more information, see “Exporting IDF Files”.
BoardSim	You can pass the default attributes Value, Tolerance, Voltage, and PowerGround to BoardSim. For more information, see “Creating PADS HyperLynx BoardSim - HYP Files on page ”.

Attribute Dictionary

Although you can add new attributes to design objects using the Object Attributes dialog box (select an object > right-click > **Attribute** popup menu item), 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.

Use the Attribute Dictionary to create attributes in a design. SailWind Layout provides default attributes that are applied to every new design. Although the attributes are provided, they are not assigned to any objects.

You can automatically load attributes for part types and decals from the current libraries into PCB designs using the Attribute Dictionary dialog box. You can load attributes when you open files or you can load the attributes after you open a file. When you load attributes, the following actions occur:

- The Attribute Dictionary is updated with new attributes
- Each new attribute uses the default hierarchy
- Attributes are added to the current part types and decals, as appropriate, in the open design
- ECO registration is turned on for all new attributes
- ECO commands are not stored in the ECO journal file for these updates

For more information, see the "Attribute Dictionary Dialog Box" topic.

Default and Other Attribute Properties and Usage

SailWind Layout provides default attributes that are available to every design. You can change the default attribute dictionary to match library attributes or to suit other design needs.

For more information, see the "Modifying Design Attribute Properties" on page 389 and "Modifying the Default Attribute Dictionary" on page 371 topics.

SailWind Layout also provides a default set of units (and unit prefixes) that are accepted as input and used as output. For more information, see Default Units.

CAUTION:
Do not edit the ASSEMBLY_OPTIONS attribute. SailWind Layout automatically maintains this attribute.

Default Attribute Properties

Default attribute properties are defined and maintained in the design database.

In the following table column titles:

- **S** — System
- **H** — Hidden
- **RO** — Read-only

Table 77. Default Attribute Properties

Attribute	Type	Objects	ECO	Hierarchy	S	H	RO
Value	Free Text*	Part, Part Type	Yes	Part, Part Type	Yes	No	No
Tolerance	Free Text*	Part, Part Type	Yes	Part, Part Type	Yes	No	No
HyperLynx.Model	Free Text*	Part, Part Type	Yes	Part, Part Type	Yes	No	No
HyperLynx.Model File	Free Text*	Part, Part Type	Yes	Part, Part Type	Yes	No	No
HyperLynx.Function	Not supported by PADS HyperLynx. Values were: SIM_OUTSIM_BOTH and SIM_IN.						
HyperLynx.Frequency	Not supported by PADS HyperLynx						
HyperLynx.Duty Cycle	Not supported by PADS HyperLynx						
HyperLynx.Type	Not supported by PADS HyperLynx. Values were: Clock, Strobe, Data Address, Power Supply, Analog High Speed, Analog Low Speed, Do Not Analyze.						
HyperLynx.Default IC.Model	Not supported by PADS HyperLynx						
HyperLynx.Default IC.Model File	Not supported by PADS HyperLynx						
HyperLynx.Default IC.Model Pin	Not supported by PADS HyperLynx						
Part Number	Free Text*	Part, Part Type	Yes	Part, Part Type	No	No	No
Description	Free Text*	Part, Part Type	Yes	Part, Part Type	No	No	No

Table 77. Default Attribute Properties (continued)

Attribute	Type	Objects	ECO	Hierarchy	S	H	RO
Cost	Free Text*	Part, Part Type	Yes	Part, Part Type	No	No	No
Manufacturer #1	Free Text*	Part, Part Type	Yes	Part, Part Type	No	No	No
Manufacturer #2	Free Text*	Part, Part Type	Yes	Part, Part Type	No	No	No
DIE.xxx	Free Text*	Decal	No	Decal	Yes	Yes	Yes
ASSEMBLY_OPTIONS	Free Text*	PCB, Part	Yes	PCB	Yes	Yes	Yes
PowerGround	Yes/No	Net, Net Class, PCB	Yes	Net, Net Class	Yes	No	No
Voltage	Measure	Net, Net Class	Yes	Net, Net Class	Yes	No	No
Geometry.Height	Size/Dim. (Measure)	PCB, Part, Decal,Part Type	No	Part, Decal	Yes	No	No

- ***** — Free Text attributes are not case-sensitive

Other Attribute Properties

An extensive list of other attribute properties are defined and maintained in the design database. In the following table column titles:.

- **S** — System
- **H** — Hidden
- **RO** — Read-only

Table 78. Other Attribute Properties

Attribute	Type	Objects	ECO	Hierarchy	S	H	RO
AutoDimensioning.Line_Layer	Number	PCB	No	PCB	Yes	No	No
AutoDimensioning.Text_Layer	Number	PCB	No	PCB	Yes	No	No
AutoDimensioning.Arc_RadiusMode	Yes/No	PCB	No	PCB	Yes	No	No

Table 78. Other Attribute Properties (continued)

Attribute	Type	Objects	ECO	Hierarchy	S	H	RO
AutoDimensioning.Marker_Shape	Yes/No (each)	PCB	No	PCB	Yes	No	No
AutoDimensioning.Marker_Size	Measure	PCB	No	PCB	Yes	No	No
AutoDimensioning.Marker_Width	Measure	PCB	No	PCB	Yes	No	No
AutoDimensioning.Arrow_Shape	Number	PCB	No	PCB	Yes	No	No
AutoDimensioning.Arrow_Length	Measure	PCB	No	PCB	Yes	No	No
AutoDimensioning.Arrow_Size	Measure	PCB	No	PCB	Yes	No	No
AutoDimensioning.Arrow_LineWidth	Measure	PCB	No	PCB	Yes	No	No
AutoDimensioning.Arrow_TailLength	Measure	PCB	No	PCB	Yes	No	No
AutoDimensioning.Arrow_TextGap	Measure	PCB	No	PCB	Yes	No	No
AutoDimensioning.Text_Height	Measure	PCB	No	PCB	Yes	No	No
AutoDimensioning.Text_Width	Measure	PCB	No	PCB	Yes	No	No
AutoDimensioning.Text_Suffix	Free Text†	PCB	No	PCB	Yes	No	No
AutoDimensioning.Text_NumberPrecision	Measure*	PCB	No	PCB	Yes	No	No
AutoDimensioning.Text_AngularPrecision	Measure*	PCB	No	PCB	Yes	No	No
AutoDimensioning.Text_DisplacementCase	Number	PCB	No	PCB	Yes	No	No
AutoDimensioning.Text_DisplacementValue	Number	PCB	No	PCB	Yes	No	No
AutoDimensioning.Text_DefaultOrientation	Number	PCB	No	PCB	Yes	No	No
AutoDimensioning.Text_DefaultPosition	Number	PCB	No	PCB	Yes	No	No

Table 78. Other Attribute Properties (continued)

Attribute	Type	Objects	ECO	Hierarchy	S	H	RO
AutoDimensioning.Text_ManualMove	Yes/No	PCB	No	PCB	Yes	No	No
AutoDimensioning.Text_NoGenerate	Yes/No	PCB	No	PCB	Yes	No	No
AutoDimensioning.Extension_Draw1	Yes/No	PCB	No	PCB	Yes	No	No
AutoDimensioning.Extension_Draw2	Yes/No	PCB	No	PCB	Yes	No	No
AutoDimensioning.Extension_Width	Measure	PCB	No	PCB	Yes	No	No
AutoDimensioning.Extension_PickPointGap	Measure	PCB	No	PCB	Yes	No	No
AutoDimensioning.Extension_LineGap	Measure	PCB	No	PCB	Yes	No	No
AutoDimensioning.Preview_Type	Number	PCB	No	PCB	Yes	No	No
DFT.Nail Count Per Net	Number	Net, Net Class, PCB	Yes	Net, Net Class	Yes	No	No
DFT.Nail Diameter	Free Text†	Pin, Via	No	Pin, Via	Yes	No	No
DFT.Nail Number	Free Text†	Pin, Via	No	Pin, Via	Yes	No	No
DFT.Generate Test Points	Yes/No	PCB	No	PCB	Yes	Yes	Yes
DFT.Probe to Trace Clearance	Size/Dim.(Measure)**	PCB	No	PCB	Yes	Yes	Yes
DFT.Probe to Pad Clearance	Size/Dim.(Measure)**	PCB	No	PCB	Yes	Yes	Yes
DFT.Allow Stubs	Yes/No	PCB	No	PCB	Yes	Yes	Yes
DFT.Stub Length	Size/Dim.(Measure)***	PCB	No	PCB	Yes	Yes	Yes
DFT.Use Via Grid	Yes/No	PCB	No	PCB	Yes	Yes	Yes
DFT.Grid X-Coordinate	Size/Dim.(Measure)***	PCB	No	PCB	Yes	Yes	Yes
DFT.Grid Y-Coordinate	Size/Dim.(Measure)***	PCB	No	PCB	Yes	Yes	Yes

Table 78. Other Attribute Properties (continued)

Attribute	Type	Objects	ECO	Hierarchy	S	H	RO
Strategy.SplitPairs.Pass	List****	PCB	No	PCB	Yes	Yes	Yes
Strategy.SplitPairs.Protect	Yes/No	PCB	No	PCB	Yes	Yes	Yes
Strategy.SplitPairs.Pause	Yes/No	PCB	No	PCB	Yes	Yes	Yes
Strategy.SplitPairs.Priority	Number	PCB	No	Net, Net Class, Part, PCB	Yes	Yes	Yes
Strategy.SplitPairs.Intensity	List*****	PCB	No	PCB	Yes	Yes	Yes
Strategy.Fanout.PlanePriority	Number	PCB	No	PCB	Yes	Yes	Yes
Strategy.Fanout.Pass	List****	PCB	No	PCB	Yes	Yes	Yes
Strategy.Fanout.Protect	Yes/No	PCB	No	PCB	Yes	Yes	Yes
Strategy.Fanout.Pause	Yes/No	PCB	No	PCB	Yes	Yes	Yes
Strategy.Fanout.Priority	Number	PCB	No	Net, Net Class, Part, PCB	Yes	Yes	Yes
Strategy.Fanout.Intensity	List*****	PCB	No	PCB	Yes	Yes	Yes
Strategy.Fanout.PlanePriority	Number	PCB	No	PCB	Yes	Yes	Yes
Strategy.Patterns.Pass	List****	PCB	No	PCB	Yes	Yes	Yes
Strategy.Patterns.Protect	Yes/No	PCB	No	PCB	Yes	Yes	Yes
Strategy.Patterns.Pause	Yes/No	PCB	No	PCB	Yes	Yes	Yes
Strategy.Patterns.Priority	Number	PCB	No	Net, Net Class, Part, PCB	Yes	Yes	Yes
Strategy.Patterns.Intensity	List*****	PCB	No	PCB	Yes	Yes	Yes
Strategy.Patterns.PlanePriority	Number	PCB	No	PCB	Yes	Yes	Yes
Strategy.Route.Pass	List****	PCB	No	PCB	Yes	Yes	Yes
Strategy.Route.Protect	Yes/No	PCB	No	PCB	Yes	Yes	Yes
Strategy.Route.Pause	Yes/No	PCB	No	PCB	Yes	Yes	Yes

Table 78. Other Attribute Properties (continued)

Attribute	Type	Objects	ECO	Hierarchy	S	H	RO
Strategy.Route.Priority	Number	PCB	No	Net, Net Class, Part, PCB	Yes	Yes	Yes
Strategy.Route.Intensity	List*****	PCB	No	PCB	Yes	Yes	Yes
Strategy.Route.PlanePriority	Number	PCB	No	PCB	Yes	Yes	Yes
Strategy.Optimize.Pass	List****	PCB	No	PCB	Yes	Yes	Yes
Strategy.Optimize.Protect	Yes/No	PCB	No	PCB	Yes	Yes	Yes
Strategy.Optimize.Pause	Yes/No	PCB	No	PCB	Yes	Yes	Yes
Strategy.Optimize.Priority	Number	PCB	No	Net, Net Class, Part, PCB	Yes	Yes	Yes
Strategy.Optimize.Intensity	List*****	PCB	No	PCB	Yes	Yes	Yes
Strategy.Optimize.PlanePriority	Number	PCB	No	PCB	Yes	Yes	Yes
Strategy.Miters.Pass	List****	PCB	No	PCB	Yes	Yes	Yes
Strategy.Miters.Protect	Yes/No	PCB	No	PCB	Yes	Yes	Yes
Strategy.Miters.Pause	Yes/No	PCB	No	PCB	Yes	Yes	Yes
Strategy.Miters.Priority	Number	PCB	No	Net, Net Class, Part, PCB	Yes	Yes	Yes
Strategy.Miters.Intensity	List*****	PCB	No	PCB	Yes	Yes	Yes
Strategy.Miters.PlanePriority	Number	PCB	No	PCB	Yes	Yes	Yes
Strategy.TestPoint.Pass	List****	PCB	No	PCB	Yes	Yes	Yes
Strategy.TestPoint.Protect	Yes/No	PCB	No	PCB	Yes	Yes	Yes
Strategy.TestPoint.Pause	Yes/No	PCB	No	PCB	Yes	Yes	Yes
Strategy.TestPoint.Priority	Number	PCB	No	Net, Net Class, Part, PCB	Yes	Yes	Yes
Strategy.TestPoint.Intensity	List*****	PCB	No	PCB	Yes	Yes	Yes

Table 78. Other Attribute Properties (continued)

Attribute	Type	Objects	ECO	Hierarchy	S	H	RO
Strategy.TestPoint.Plane Priority	Number	PCB	No	PCB	Yes	Yes	Yes

- † — Free Text attributes are not case-sensitive
- * — Dimensioning text number and text angular precision values range from 0 to 8.
- ** — DFT.Probe to trace and probe to pad clearance values range from 1 to 1000
- *** — DFT.Stub length, grid x-coordinate, and grid y-coordinate values
- **** — Strategy.XXX.Pass values shown in the following table are available for all Strategy.XXX.Pass attributes, where XXX is the name of the pass type for SailWind Router to perform: Split Pairs, Fanout, Patterns, Route, Optimize, Miters, and Test Point.

Table 79. Values for Strategy XXX Pas Attributes

Status	Represents
Done	Indicates SailWind Router completed this pass.
Yes	Indicates SailWind Router should perform this pass.
No	Indicates SailWind Router should not perform this pass.

- ***** — Strategy.XXX.Intensity values where XXX is the name of the pass type for SailWind Router to perform: Split Pairs, Fanout, Patterns, Route, Optimize, Miters, and Test Point.

Default Attribute Usage

Default attributes are used according to specific rules.

Table 80. Default Attribute Usage

Attribute	Used For
Value	Replaces the value attribute that is usually assigned to the part name on the schematic level. When you bring parts into a design, value is converted to the new attribute format.
Tolerance	Replaces the tolerance attribute that is usually assigned to the part name on the schematic level. When you bring parts into a design, tolerance is converted to the new attribute format.
HyperLynx.Model	In the .ref file that lists reference designators and corresponding model information, this attribute provides the model name. Used for PADS HyperLynx BoardSim simulations. When an IBIS model file *(.ibs)* contains multiple component models, this attribute is necessary to specify which should be used.

Table 80. Default Attribute Usage (continued)

Attribute	Used For
HyperLynx.Model File	In the .ref file that lists reference designators and corresponding model information, this attribute provides the name of the IBIS model file *(.ibs)*. Used for PADS HyperLynx BoardSim simulations.
HyperLynx.Function	Not supported by PADS HyperLynx.
HyperLynx.Frequency	Not supported by PADS HyperLynx.
HyperLynx.Duty Cycle	Not supported by PADS HyperLynx.
HyperLynx.Type	Not supported by PADS HyperLynx.
HyperLynx.Default IC.Model	Not supported by PADS HyperLynx.
HyperLynx.Default IC.Model File	Not supported by PADS HyperLynx.
HyperLynx.Default IC.Model Pin	Not supported by PADS HyperLynx.
HyperLynx.Sim Direction	Not supported by PADS HyperLynx.
Part Number	Used for part ordering, accounting, and so on.
Description	Describes the purpose of the part.
Cost	Specifies the cost of the part.
Manufacturer #1	Specifies the primary manufacturer of the part.
Manufacturer #2	Specifies a secondary manufacturer of the part.
ASSEMBLY_OPTIONS	Indicates whether the part is part of an assembly variant. Existing assembly variants are converted.
PowerGround	Identifies nets as ground and power nets.
Voltage	Describes the voltage of the net.
Geometry.Height	Describes the part height (height above the PCB).

Other Attribute Usage

Other attributes are also used according to specific rules.

Table 81. Other Attribute Usage

Attribute	Used For
AutoDimensioning.Line_Layer	Indicates the layer on which the dimensioning lines appear.

Table 81. Other Attribute Usage (continued)

Attribute	Used For
AutoDimensioning.Text_Layer	Indicates the layer on which the dimensioning text appears.
AutoDimensioning.Arc_RadiusMode	Indicates whether to measure a radius or a diameter when you dimension a circle.
AutoDimensioning.Marker_Shape	Indicates the shapes of the alignment tool, horizontal, vertical, or diagonal crosshairs, and a square or circular bull's-eye.
AutoDimensioning.Marker_Size	Indicates the size of the alignment tool.
AutoDimensioning.Marker_Width	Indicates the line width of the alignment tool.
AutoDimensioning.Arrow_Shape	Indicates whether to draw open arrows, closed arrows, or datum lines.
AutoDimensioning.Arrow_Length	Indicates the length of the arrow.
AutoDimensioning.Arrow_Size	Indicates the width (height) of the arrow.
AutoDimensioning.Arrow_LineWidth	Indicates the line width of the tail and arrow lines.
AutoDimensioning.Arrow_TailLength	Indicates the minimum length of the arrow tail.
AutoDimensioning.Arrow_TextGap	Indicates the spacing between the tail and the measurement text.
AutoDimensioning.Text_Height	Indicates the height of dimensioning text.
AutoDimensioning.Text_Width	Indicates the width of one character in dimensioning text.
AutoDimensioning.Text_Suffix	Indicates the suffix to appear after the dimensioning measurement.
AutoDimensioning.Text_NumberPrecision	Indicates the number of decimal places, in mils, for linear measurements.
AutoDimensioning.Text_AngularPrecision	Indicates the number of decimal places, in degrees, for angular measurements.
AutoDimensioning.Text_DisplacementCase	Indicates the position of the dimensioning text.
AutoDimensioning.Text_DisplacementValue	Indicates the custom position of the dimensioning text.
AutoDimensioning.Text_DefaultOrientation	Indicates the orientation of the dimensioning text.

Table 81. Other Attribute Usage (continued)

Attribute	Used For
AutoDimensioning.Text_DefaultPosition	Indicates the position of the dimensioning text.
AutoDimensioning.Text_ManualMove	Attaches the dimensioning text to the pointer when you add dimensions.
AutoDimensioning.Text_NoGenerate	Creates only extension lines and arrows when you add dimensions.
AutoDimensioning.Extension_Draw1	Draws an extension line for the first point you select.
AutoDimensioning.Extension_Draw2	Draws an extension line for the second point you select.
AutoDimensioning.Extension_Width	Indicates the width of the extension line.
AutoDimensioning.Extension_PickPointGap	Indicates the gap between the selection point and the end of the extension line.
AutoDimensioning.Extension_LineGap	Indicates the overhang of the line beyond the arrow.
AutoDimensioning.Preview_Type	Indicates the contents of the preview window at various orientations depending on the current preference settings.
DFT.Nail Count Per Net	Indicates the maximum number of test points on a net.
DFT.Nail Diameter	Indicates the probe, or nail diameter, size for a test point.
DFT.Nail Number	Indicates the ID of the probe in the test fixture.
DFT.Generate Test Points	Indicates whether test points should be created.
DFT.Probe to Trace Clearance	Indicates the minimum probe-to-trace clearance.
DFT.Probe to Pad Clearance	Indicates the minimum probe-to-pad clearance.
DFT.Allow Stubs	Indicates whether stubs should be created.
DFT.Stub Length	Indicates the maximum stub length.
DFT.Use Via Grid	Indicates whether to use the via grid when adding and placing test points.

Table 81. Other Attribute Usage (continued)

Attribute	Used For
DFT.Grid X-Coordinate	Indicates the via grid size along the X axis.
DFT.Grid Y-Coordinate	Indicates the via grid size along the Y axis.
Strategy.SplitPairs.Pass	Indicates whether SailWind Router should perform the split pairs pass and whether it is complete.
Strategy.SplitPairs.Protect	Indicates whether SailWind Router should protect traces routed during the split pairs pass.
Strategy.SplitPairs.Pause	Indicates whether SailWind Router should pause routing after completing the split pairs pass.
Strategy.SplitPairs.Priority	Indicates the routing order of nets for the split pairs pass.
Strategy.SplitPairs.Intensity	Indicates the intensity, or level of effort, for SailWind Router to use when performing the split pairs pass.
Strategy.Fanout.Pass	Indicates whether SailWind Router should perform the fanout pass and whether it is complete.
Strategy.Fanout.Protect	Indicates whether SailWind Router should protect traces routed during the fanout pass.
Strategy.Fanout.Pause	Indicates whether SailWind Router should pause routing after completing the fanout pass.
Strategy.Fanout.Priority	Indicates the routing order of nets for the fanout pass.
Strategy.Fanout.Intensity	Indicates the intensity for SailWind Router to use when performing the fanout pass.
Strategy.Fanout.PlanePriority	Indicates the routing order for plane nets for the fanout pass.
Strategy.Patterns.Pass	Indicates whether SailWind Router should perform the patterns pass and whether it is complete.
Strategy.Patterns.Protect	Indicates whether SailWind Router should protect traces routed during the patterns pass.

Table 81. Other Attribute Usage (continued)

Attribute	Used For
Strategy.Patterns.Pause	Indicates whether SailWind Router should pause routing after completing the patterns pass.
Strategy.Patterns.Priority	Indicates the routing order of nets for the patterns pass.
Strategy.Patterns.Intensity	Indicates the intensity for SailWind Router to use when performing the patterns pass.
Strategy.Patterns.PlanePriority	Indicates the routing order for plane nets for the patterns pass.
Strategy.Route.Pass	Indicates whether SailWind Router should perform the route pass and whether it is complete.
Strategy.Route.Protect	Indicates whether SailWind Router should protect traces routed during the route pass.
Strategy.Route.Pause	Indicates whether SailWind Router should pause routing after completing the route pass.
Strategy.Route.Priority	Indicates the routing order of nets for the route pass.
Strategy.Route.Intensity	Indicates the intensity for SailWind Router to use when performing the route pass.
Strategy.Route.PlanePriority	Indicates the routing order for plane nets for the route pass.
Strategy.Optimize.Pass	Indicates whether SailWind Router should perform the optimize pass and whether it is complete.
Strategy.Optimize.Protect	Indicates whether SailWind Router should protect traces routed during the optimize pass.
Strategy.Optimize.Pause	Indicates whether SailWind Router should pause routing after completing the optimize pass.
Strategy.Optimize.Priority	Indicates the routing order of nets for the optimize pass.
Strategy.Optimize.Intensity	Indicates the intensity for SailWind Router to use when performing the optimize pass.
Strategy.Optimize.PlanePriority	Indicates the routing order for plane nets for the optimize pass.

Table 81. Other Attribute Usage (continued)

Attribute	Used For
Strategy.Miters.Pass	Indicates whether SailWind Router should perform the miters pass and whether it is complete.
Strategy.Miters.Protect	Indicates whether SailWind Router should protect traces routed during the miters pass.
Strategy.Miters.Pause	Indicates whether SailWind Router should pause routing after completing the miters pass.
Strategy.Miters.Priority	Indicates the routing order of nets for the miters pass.
Strategy.Miters.Intensity	Indicates the intensity for SailWind Router to use when performing the miters pass.
Strategy.Miters.PlanePriority	Indicates the routing order for plane nets for the miters pass.
Strategy.TestPoint.Pass	Indicates whether SailWind Router should perform the test point pass and whether it is complete.
Strategy.TestPoint.Protect	Indicates whether SailWind Router should protect traces routed during the test point pass.
Strategy.TestPoint.Pause	Indicates whether SailWind Router should pause routing after completing the test point pass.
Strategy.TestPoint.Priority	Indicates the routing order of nets for the test point pass.
Strategy.TestPoint.Intensity	Indicates the intensity for SailWind Router to use when performing the Test Point pass.
Strategy.TestPoint.PlanePriority	Indicates the routing order for plane nets for the test point pass.

Modifying the Default Attribute Dictionary

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:\<install_folder>\<version>\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. For more information, see "File Open Conversions" and "Attribute Dictionary Dialog Box".

Prerequisites

Back up your Default.asc or DefaultAttributeDictionary.asc before you overwrite it.

Procedure

1. Click the **File > New** menu item.
2. If you want to edit the default attributes to use with older files, click the **File > Import** menu item and import the file *DefaultAttributeDictionary.asc*.
3. Click the **Edit > Attribute Dictionary** menu item.
4. Modify the existing attributes or add attributes as needed. For more information, see "Modifying Design Attribute Properties".
5. Click **OK** to close the dialog box.
6. To overwrite the existing file, click the **File > Export** menu item. 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.
7. Click **ASCII** as the file type and click **Save**. The ASCII Output dialog box appears.
8. In the Sections list, select the Attributes check box.
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.

Assignment of Attributes

If you cannot select the object to which you want to assign an attribute, select the related object.

The following table defines the relationship of the objects. For more information, see "Applying an Attribute Value to All Other Objects" on page 395 for assigning to the same object types and Attribute Manager on page 392 topics to assign to multiple object types.

Table 82. Assignment of Attributes

Object	What to Select
PCB	Any object
Part Type	A design component

Table 82. Assignment of Attributes (continued)

Object	What to Select
Decal	A design component
Part	A design component
Jumper	A Jumper
Pin	A Pin
Net Class	A net that is a member of the class
Net	A Net
Via	A Via

Attribute Values

Attribute values can be 2047 characters long. You can use any printable character, including spaces, in an attribute value. You cannot, however, use a space as the first or last character in the value.

When you enter an attribute value, the exact value you type (in dialog boxes, ASCII files, or the library) is stored. This means that capitalization, leading and trailing zeros, embedded spaces, specific unit prefixes, and the presentation of the Yes/No value are all stored exactly as you type them. Leading and trailing spaces are not saved. Invalid values are not saved.

Exact values are saved for the following attribute types:

- Number
- Decimal umber
- Yes/No
- Measure

Exact values are not saved in the Attribute Dictionary entry for the List attribute type or when you set Limits for Number, Decimal Number, and Measure attribute types. For the Number attribute type, leading zeros are removed. For Decimal Number and Measure attribute types, leading zeros are removed, trailing zeros after the decimal point are removed, and numbers greater than 14 characters are rounded. Numbers with more than 14 zeros may be converted to scientific notation.

When using attribute values:

- Automation does not pass the exact attribute value for Yes/No, Number, or Decimal Number attribute types. Automation does, however, pass the exact attribute value for the Measure attribute type.
- Automation ignores whether an attribute is read only, system, or hidden. Therefore, Automation can change all attributes (properties, Attribute Dictionary entry, and value), regardless of their state.

Special Attribute Measurements
Number/Decimal Number Attribute Values and ECO
Exact Attribute Value Examples
List Exception
Measure, Geometry.Height (Size/Dimension) Attribute Exceptions
Default Units
Customizing Units for Attributes
.ini File Format for Units

Special Attribute Measurements

You can enter complex units like ounces/sq. foot for copper thickness; however, SailWind Layout does not input, process, or output prefixes for complex units.

- Size/Dimension. SailWind Layout accepts as input, processes, and outputs Size/Dimension units using the values shown in the table below.

Table 83. Size and Dimension Measurements

Unit set on Global Tab (Options)	Output Example	Comment
Mils	12 mil	1 mil = 25.4*10-6 m
Metric	3 mm	1 mm = 1*10-3 m
Inches	2"	1" = 25.4*10-3 m

- Percentages. SailWind Layout accepts as input, processes, and outputs a percentage, like 10%, if you use the percent symbol (%).

Number/Decimal Number Attribute Values and ECO

SailWind Layout automatically converts attribute values for the Number, Decimal Number, or Measure type properties during the ECO process.

For example, a frequency value, if entered as 100 at the schematic or library, is converted to .1 kHz by default. Also, leading and trailing zeroes are truncated. For example, the decimal number 123.400 becomes 123.4.

Although these conversions are correct, Compare Netlist and the ECO process, detect and report these conversions as differences. Therefore, a design populated with attributes could have thousands of warnings. To avoid this, do one of the following:

- Define attributes as Free Text type in the Attribute Dictionary. When you want to take advantage of the math functions in the Attribute Manager dialog box, go to the Attribute Dictionary and change the type to Number, Decimal Number, or Measure. Then, before performing a comparison or beginning an ECO, set the type back to Free Text.

- Use the Number, Decimal Number, or Measure types. Make sure the attributes are ECO-registered and then perform a backward annotation. The values are converted in the design and backward annotated to the schematic. The schematic and PCB layout will now be synchronized.

Exact Attribute Value Examples

The exact value required for an attribute varies depending on the type of attribute, such as number, decimal, or measurement.

The following tables provide some specific examples of exact attribute values.

Table 84. Yes/No Examples

You Type	V 3.5 and Higher Import and Export	V 3.0 Imports and Exports
y	Y	Yes

Table 84. Yes/No Examples (continued)

You Type	V 3.5 and Higher Import and Export	V 3.0 Imports and Exports
NO	NO	No
true	True	Yes
1	1	Yes

Table 85. Number Examples

You Type	V 3.5 and Higher Import and Export	V 3.0 Imports and Exports
0001	0001	1

Table 86. Decimal Number Examples

You Type	V 3.5 and Higher Import and Export	V 3.0 Imports and Exports
0001.5	0001.5	1.5
0.123456789	0.123456789	0.123457
0.000001	0.000001	1E-006
1d3	1d3	1000
12.3e7	12.3e7	1.23E+008
121.	121.	121
1.230000	1.230000	1.23

Table 87. Measure Examples

You Type	V 3.5 and Higher Import and Export	V 3.0 Imports and Exports
10	10	10V
1000V	1000V	1kV
1e-5V	1e-5V	10uV
12 volt	12 volt	12V
7 MILLIVOLT	7 MILLIVOLT	7mV

List Exception

Although SailWind Layout does not save exact List type values, it does change list entries to match the Attribute Dictionary entry.

For example, if the Attribute Dictionary entry for a list type attribute has Intel, IBM, and AMD as list choices, and you enter intel as a value, SailWind Layout changes the entry to Intel. The lowercase i is changed to uppercase. The following table lists exceptions for voltage measures.

Table 88. Measure, Voltage Attribute Exceptions

You Type	V 3.5 and higher Import and Export	V 3.0 Imports and Exports	Comment
" 10V"	"10V"	"10V"	Spaces before 10V are removed in both 3.0 and 3.5 **Note:** Quotation marks used only to show spaces
"10V "	"10V"	"10V"	Spaces after 10V are removed in both version 3.0 and 3.5 **Note:** Quotation marks used only to show spaces
ten volt			Invalid string. Input is ignored and no value is attached to the attribute.

Measure, Geometry.Height (Size/Dimension) Attribute Exceptions

If you do not specify a unit of measure for a Size/Dimension Measure type attribute, SailWind Layout does not save the exact value.

The value is not saved because a number in the attribute value without a unit creates confusion if you change the current units.

Table 89. Measure, Geometry Height Attribute Exceptions

V 4.0 and higher Input		V 4.0 and higher Output for Current Units		
String	**Current Units**	**Mils**	**Inches**	**Metric (mm)**
10	mils	10mil	0.01"	0.254mm
0.1	inches	100mil	0.1"	2.54mm
10	metric	39.37mil	0.3937"	10mm

Tip
Using a non-standard format for your value might cause SailWind Layout to change the value even if you do not change the current units. For example entering a value of 0001.2000mil, causes SailWind Layout to change the value to 1.2mil.

Default Units

You can include a unit with an attribute value. SailWind Layout provides a default set of units (and unit prefixes) acceptable for input and for output.

SailWind Layout uses Systeme Internationale units, or SI units. Units are exported with attributes and are converted appropriately. User-defined units (dollar, yen, feet, pound, ounces/sq. foot, and so on) are not converted. Also, you cannot use prefixes with user-defined units.

The following units are supported, but are either enabled or disabled for actual use within SailWind Layout. To change the units that appear in this list, see Customizing Units for Attributes.

Tip
An extra comma (,) means that you can enter the abbreviation for the unit without a prefix. For example, you can enter O for Ohm in an attribute value; it is a valid value. You cannot, however, add F for Farad in an attribute value; it is not a valid value. You must use Farad with a prefix.

Supported Units

The following table lists units that are supported in SailWind Layout.

Table 90. Supported Units

Abbreviation	Unit	Enabled	Allowed Prefixes	Quantity
O	Ohm	Yes	u,m,,k,M,G	Resistance
F	Farad	Yes	p,n,u,m	Capacitance
H	Henry	Yes	n,u,m	Inductance
Hz	Hertz	Yes	,k,M,G	Frequency
A	Ampere	Yes	u,m,,k	Electric current
V	Volt	Yes	n,u,m,,k	Voltage
W	Watt	Yes	p,u,m,,k,M,G	Power
s	Second	Yes	p,n,u,m	Time
g	Gram	No	u,m,,k	Mass
Wb	Weber	No	p,n,u,m	Magnetic flux

Table 90. Supported Units (continued)

Abbreviation	Unit	Enabled	Allowed Prefixes	Quantity
T	Tesla	No	p,n,u,m,,k	Magnetic flux density
C	Coulomb	No	p,n,u,m	Charge
S	Siemens	No	n,u,m	Electric conductance
J	Joule	No	p,n,u,m,,k,M,G	Energy
N	Newton	No	u,m,,k	Force
Pa	Pascal	No	,k,M	Pressure
K	Kelvin	No	u,m,	Temperature
rad	Radian	No	u,m,	Plane angle
sr	Steradian	No	u,m,	Solid angle
cd	Candela	No	u,m,,k	Luminous intensity
lx	Lux	No	u,m,,k	Illumination
lm	Lumen	No	u,m,,k	Luminous flux
mol	Mole	No	u,m,,k	Amount of substance
Gy	Gray	No	U,m,,k	Absorbed dose
Bq	Becquerel	No	U,m,,k	Activity
Sv	Sievert	No	U,m,,k	Dose equivalent
m	Meter	No	P,n,u,m,,k	Distance
l	Liter	No	U,m,	Liquid

Unit Prefixes

As shown below, each prefix has a symbol and is a power of ten. You cannot use prefixes with user-defined units, such as dollar, yen, feet, pound, and so on.

Tip
SailWind Layout never exports the prefixes h, da, d, and c.

Table 91. Unit Prefixes

Symbol	Prefix	Power of Ten
Y	Yotta	+24
Z	Zetta	+21
E	Exa	+18
P	Peta	+15
T	Tera	+12
G	Giga	+9
M	Mega	+6
k	kilo	+3
h	hecto	+2
da	deca	+1
d	deci	-1
c	centi	-2
m	milli	-3
u	micro	-6
n	nano	-9
p	pico	-12
f	fempto	-15
a	atto	-18
z	zepto	-21
y	yocto	-24

Customizing Units for Attributes

In addition to the standard attributes, you can add and modify custom attributes to a design to meet your own specialized requirements.

Procedure

1. Navigate to the *C:\<install_folder>\<version>\Programs* folder.
2. Open the *SailWindpcb.ini* file in a text editor, such as Notepad.

3. Add a new attribute unit section by typing the header [SI Units].
4. Make modifications as required:
   a. To Enable Units:
      i. To enable units, delete the “ignore;” variable from the line. For example, the line for the Gram unit reads:

      ```
      Gram=ignore;u,m,,k
      ```

      ii. Modify the line so it reads:

      ```
      Gram=u,m,,k
      ```

   b. To Disable Units:
      i. To disable units, add the “ignore;” variable to the line. For example, the line for the Farad unit reads:

      ```
      Farad=p,n,u,m
      ```

      ii. Modify the line so it reads:

      ```
      Farad=ignore;p,n,u,m
      ```

      iii. 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.
5. Save the *SailWindpcb.ini* file.

Related Topics

.ini File Format for Units

.ini File Format for Units

Specify units in the [SI Units] section of the *.ini* file.

Use the following format:

```
<full unit name>=[ignore;][input:<prefix list>;][output:<prefix list>]
```

Table 92. where:

<full unit name>	Specifies the name of the unit.
[ignore;]	Specifies whether to ignore the unit. If this variable is included, the unit is ignored. To enable the unit remove this variable.

Table 92. where: (continued)

[input:<prefix list>;]	Specifies that you are creating a list of prefixes that are valid for input into SailWind Layout.
[output:<prefix list>]	Specifies that you are creating a list of prefixes that are valid for output from SailWind Layout.

Tip
You can list the valid prefixes after the equal sign (=) and those prefixes are used for both input and output.

Attribute Types

You can assign many types of attributes to the objects in your designs, including the free-text type, the yes/no type, and the list type.

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,” meaning you can type the name of the net as an attribute, but renaming the net does not update the attribute.

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.

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.

The List Attribute Type

The List attribute type allows you to create a list from which to choose the value.

Creating a List Type Attribute

You can create a list of entries as options for the attribute value. For example, you can create a list of all of your part manufacturers that is used every time you assign the attribute.

Procedure

1. Click the **Edit > Attribute Dictionary** menu item.
2. In the Attribute Dictionary Dialog Box, click **New**.
3. In the Attribute Properties dialog box on page 1115, type the name of the new attribute to create in the Attribute text box.

   Tip
   Attribute 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.

4. On the **Types** tab, select the List Type.
5. Type an attribute value in the List box and click **Set** to add the item to the list. Repeat as necessary.
6. You can select the Case-sensitive check box to preserve the letter case of List entries. This setting affects sorting and matching in the Find Dialog Box and the Attribute Manager Dialog Box.
7. On the Objects tab on page 1111, assign settings and hierarchy to the objects to which you want to apply the attribute.
8. Click **OK** to close the Attribute Properties dialog box.
9. Click **Close** to close the Attribute Dictionary dialog box.

Results

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.

Deletion of List Entries

Click **Clear** or **Clear All** to delete an attribute item or items from a list.

Procedure

1. Click the **Edit > Attribute Dictionary** menu item.
2. In the Attribute Dictionary Dialog Box, select an attribute that contains a List item that you would like to delete.
3. Click the **Properties** button.
4. On the **Types** tab, select the item(s) that you wish to delete from the list box.
5. Click the **Clear** or **Clear All** button to delete the item(s) from the list.
6. Click **OK** to close the Attribute Properties dialog box and return to the Attributes Dictionary dialog box.
7. Click **Close** to close the Attribute Dictionary dialog box.

The Measure Attribute Type

A *Measure attribute type* is a physical value associated with units. It allows you to determine a measurement for the attribute value.

Creating a Measure Type Attribute
Deleting a Set of Units

Creating a Measure Type Attribute

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 on page 380. 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.

Procedure

1. Click the **Edit > Attribute Dictionary** menu item.
2. In the Attribute Dictionary Dialog Box, click **New**.
3. In the Attribute Properties dialog box on page 1115, type the name of the new attribute to create in the Attribute text box.

   Tip
   Attribute 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.

4. On the **Types** tab, select the Measure type and perform one of the following:

   - In the Measure list, select an existing measurement unit. The abbreviation appears in the Abbr box, the unit in the Unit box, and the measure or quantity in the Quantity box.
   - Enter the details for a new measurement unit.

     i. In the Abbr box, type the abbreviation to use for the unit.

     ii. In the Unit box, type the name of the unit.

     Note:
     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 Options dialog box > **Global** category > General subcategory on page 1531.

iii. In the Quantity box, type the quantity, or what it measures.

iv. Click **Set** to add the item to the list.

v. In the Limits area, type a minimum value in the Min box and/or a maximum value in the Max box. SailWind Layout checks against the Limits area values. If an attribute has the Number, Decimal Number, or Measure type property, SailWind 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.

5. On the Objects tab on page 1111, assign settings and hierarchy to the objects to which you want to apply the attribute.

6. Click **OK** to close the Attribute Properties dialog box.

   The new Attribute is added to the Attribute Dictionary with the measurement values that you assigned.

Deleting a Set of Units

You can delete a set of measurement units from an attribute.

Procedure

1. Click the **Edit > Attribute Dictionary** menu item.

2. In the Attribute Dictionary Dialog Box, select an attribute that contains a measurement item that you would like to delete.

3. Click the **Properties** button.

4. On the **Types** tab, the currently assigned measurement unit will be selected.

   > **Tip**
   > You can clear the measurement unit or assign a new set of values in the Abbr:, Unit: and Quantity: boxes.

5. 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.

6. The default units remain in the list.

7. Select a new measurement unit from the list or type in a new measurement unit in the Abbr:, Unit: and Quantity: boxes.

   > **Tip**
   > You must specify a measurement for the attribute in order to save it and close the Attribute Properties dialog.

8. Click **OK** to close the Attribute Properties dialog box and return to the Attributes Dictionary dialog box.

9. Click **Close** to close the Attribute Dictionary dialog box.

Related Topics

Default Units

Customizing Units for Attributes

Creating a Number or Decimal Number Type Attribute

You can type an integer number (Number) or a number with decimal or floating point numbers (Decimal Number) for the attribute value. SailWind Layout automatically converts attribute values for the Number or Decimal Number type properties during the ECO process.

Restrictions and Limitations

These are the valid ranges that are recognized:

- **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.

Procedure

1. Click the **Edit > Attribute Dictionary** menu item.

2. In the Attribute Dictionary Dialog Box, click **New**.

3. In the Attribute Properties dialog box on page 1115, type the name of the new attribute to create in the Attribute text box.

   Tip
   Attribute 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.

4. Click the **Properties** button.

5. On the **Types** tab, select the Number or Decimal number type.

6. Type a minimum value in the Min box and/or a maximum value in the Max box.

7. SailWind Layout checks against the Limits area values. If an attribute has the Number, Decimal Number, or Measure type property, SailWind Layout checks the attribute value against this property.

8. On the Objects tab on page 1111, assign settings and hierarchy to the objects to which you want to apply the attribute.

9. Click **OK** to accept the limits and close the Attribute Properties dialog box.

10. Click **Close** to close the Attribute Dictionary dialog box.

11. For more information, see "Number/Decimal Number Attribute Values and ECO".

Creating Attributes for the Design

SailWind Layout provides default attributes to apply to every new design you create. Although SailWind Layout provides attributes, it does not assign them to any objects.

For more information, see "Default and Other Attribute Properties and Usage".

Procedure

1. Click the **Edit > Attribute Dictionary** menu item.

2. In the Attribute Dictionary Dialog Box, click **New**.

3. In the Attribute Properties dialog box on page 1115, type the name of the new attribute to create in the Attribute text box.

   Tip
   Attribute 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.

4. On the **Types** tab, assign a type to the attribute and assign settings to the type if applicable. The default type is Free Text.

5. On the Objects tab on page 1111, assign settings and hierarchy to the objects to which you want to apply the attribute.

6. When you finish setting attribute properties, click **OK** to return to the Attribute Dictionary dialog box.

7. Click **Close**.

Modifying Design Attribute Properties

Use the Attribute Dictionary to modify design attribute properties.

Prerequisites

If an attribute is ECO-Registered, you must be in ECO mode to modify the attribute.

Procedure

1. Click the **Edit > Attribute Dictionary** menu item.
2. In the Attribute Dictionary Dialog Box, click the attribute to modify from the list. Default attributes and the design attributes are listed. You can select an attribute group on page 1811 from the Group list to filter the view and display only a select group of attributes. Attributes are grouped if they are structured attributes on page 1862.

   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.

   CAUTION:
   You can modify the default attributes; however, it is not recommended.

3. Click **Properties**.
4. In the Attribute Properties dialog box, on page 1115 click the **Types** tab, and click the type for the attribute.

   CAUTION:
   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 Options dialog box > **Global** category > General subcategory on page 1531).

5. Click the Objects tab on page 1111, and click the objects you want to restrict. You cannot 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 cannot have this attribute. Therefore, you would disable the Net object for the Manufacturer attribute.
6. To use the default hierarchy, click the Use Default Hierarchy check box. If you choose not to use the default hierarchy, you can modify it. Modifying the hierarchy is considered an advanced procedure.
7. If you want to enable ECO Registration on page 1824 of the attribute, click the ECO Registered check box. 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.
9. 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 now associate with non-decal attributes.

Procedure

1. Click the **Edit > Attribute Dictionary** menu item.

2. In the Attribute Dictionary dialog box, on page 1107click the attribute to delete from the list. You can select an attribute group on page 1811 from the Group list to filter the view and display only a select group of attributes. Attributes are grouped if they are structured attributes on page 1862. Clear the Show Hidden check box to filter the list to show only attribute groups that contain at least one visible attribute.

   CAUTION:
   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.

3. Click **Delete**. If you select a hidden attribute, this button is unavailable. The message "Are you sure you want to delete attribute type XXX?" appears.

4. Click **Yes** to delete the attribute.

Attribute Manager

Use the Attribute Manager to view a spreadsheet of all of the attributes on all objects 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.

CAUTION:
Hidden attributes are not available to the Attribute Manager and are not listed in the Show Attributes dialog box.

Note:
Rows in the multi-column list are unavailable if the attribute is read-only on page 1853. Rows are also unavailable if the attribute is ECO-registered on page 1824 and SailWind Layout is not in ECO mode on page 1824. 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.

Tip
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.

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.

Procedure

1. Click the **Edit > Attribute Manager** menu item.
2. In the Attribute Manager Dialog Box 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 open the Attribute Manager, 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.

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.

Procedure

1. Click the **Edit > Attribute Manager** menu item.
2. In the Attribute Manager Dialog Box, click **Show**.
3. In the "Show Attributes Dialog Box" on page 1720, select an attribute group in the Group list or <all> for all attributes. Use the Group list to filter the Attributes list. You can choose an attribute group on page 1811 to view.
4. 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.

Adding Attribute Values to Design Objects With the Attribute Manager

In the Attribute Manager, you can add an attribute and value to one or more objects that do 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 you have not already assigned it to an object.

You can add attribute values to a design object only 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. SailWind Layout also provides a default set of units (and unit prefixes) that are accepted as input and used as output. For more information, see "Default Units".

Restrictions and Limitations

You cannot add a hidden attribute, read-only attribute on page 1853, or ECO-registered attribute on page 1824 while not in ECO mode on page 1824.

Procedure

1. Click the **Edit > Attribute Manager** menu item.
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 the Enter key.

> **Tip**
> 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. SailWind Layout also provides a default set of units (and unit prefixes) that are accepted as input and used as output. For more information, see "Default Units".

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 assigned attribute has no value.

Restrictions and Limitations

You cannot edit a read-only attribute on page 1853, or an ECO-registered attribute on page 1824 while not in ECO mode on page 1824.

Procedure

1. Click the **Edit > Attribute Manager** menu item.
2. In the Attribute Manager Dialog Box, select a blank cell or a cell with a value.
3. Click **Edit**.
4. Type a value in the cell and press the Enter key.

> **Tip**
> You can include a unit with the value. SailWind Layout also provides a default set of units (and unit prefixes) that are accepted as input and used as output. For more information, see "Default Units".

Deleting Attribute Values

In the Attribute Manager, you can delete an attribute and value from a design object.

CAUTION:
You cannot delete a read-only attribute on page 1853, or an ECO-registered attribute on page 1824 while not in ECO mode on page 1824.

Procedure

1. Click the **Edit > Attribute Manager** menu item.
2. In the Attribute Manager Dialog Box, 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.

   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 remain unchanged.

Procedure

1. Click the **Edit > Attribute Manager** menu item.
2. In the Attribute Manager Dialog Box, 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.

Creating a Summary

Create and modify attribute summaries that are based on every value of the attribute assigned to objects of the same type. In other words, summaries apply per attribute and are summed up from all objects in the same tab of the multi-column list. You can also change the summary type. The summaries appear at the bottom of attribute columns.

Restrictions and Limitations

- Summaries are only available for Number, Decimal Number, and Measure attribute types. The type is an attribute property. For more information, see “Attribute Types”.
- Hidden attributes are not available to the Attribute Manager and are not listed in the Show Attributes dialog box.

Procedure

1. Click the **Edit > Attribute Manager** menu item.

2. In the Attribute Manager Dialog Box, click **Show**.

> **Note:**
> An attribute must be selected for viewing in the Attribute Manager. See "Selecting Attributes to List in the Attribute Manager" for more information.

3. In the "Show Attributes Dialog Box" on page 1720 select an attribute in the Attributes list.

4. Select the check boxes for the summary types you want to create for each attribute type.

5. 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.

6. You can create summaries or edit the summary types for other attribute columns without going into the Show Attributes dialog box. In another qualified attribute column (Number, Decimal Number, and Measure attribute types), double-click in the cell of the first summary row (used for the title of the summary type).

7. Choose a summary from the list, and press the Enter key. The new summary information appears.

Design 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.

CAUTION:
Rows in the multicolumn list are unavailable if the attribute is read-only on page 1853. Rows are also unavailable if the attribute is ECO-registered on page 1824 and SailWind Layout is not in ECO mode.

Note:
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.

Assigning Attributes to Design Objects With the Object Attributes Dialog

You can assign attributes to an object or objects of the same type using the Object Attributes dialog box.

For example, 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 apply to the objects you select.

Procedure

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 the **Attribute** popup menu item. The "Object Attributes Dialog Box" on page 1496 opens. For more information, see "Assigning Attributes to Design Objects With the Object Attributes Dialog".
3. From the Groups list click an attribute group on page 1811 to view. If you assign multiple attributes and some of those are structured attributes on page 1862, 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. For more information, see "Attribute Hierarchy".

Note:
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. For more information, see "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 on page 1853 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, cannot 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. SailWind 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. For more information, see "Default Units".

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.

Related Topics

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, the Attributes list displays the attributes that belong to all of the selected objects. When you add an attribute, you add it to all selected objects, and when you delete a value, you delete it from all objects with that attribute. You can edit the value of a selected attribute if the current level defines the attribute.

For example, if the text in the “Level” column matches the text in the “Attribute For” list, you can edit the value. If the current level does not define the attribute (the text in the Level column does not match the text in the Attributes For list), SailWind Layout adds a new attribute 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 on page 355, or add a new attribute with the correct attribute name.

Procedure

1. If the attribute is ECO-registered on page 1824, enter ECO mode. To enter ECO mode, click the ECO Toolbar button. If you do not enter ECO mode first, a message appears, informing you that you need to enter ECO mode.

   Restriction:
   You cannot modify read-only attributes on page 1853.

   For more information, see “ECO Options Dialog Box”.

2. Select the object(s) to modify.

3. Right-click and click the **Attribute** popup menu item. Attribute information about the selected object(s) appears in spreadsheet form in the “Object Attributes Dialog Box” on page 1496.

4. From the Groups list click an attribute group on page 1811 to view. If you assign multiple attributes and some of those are structured attributes on page 1862, 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. For more information, see “Attribute Hierarchy”.

   Note:
   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**.

8. Type or click the new attribute value and press the Enter key. The new value is added to the objects. For more information, see “Attribute Values”.

You can include a unit with the value. SailWind Layout provides a default set of units (and unit prefixes) that are accepted as input and used as output. For more information, see "Default Units".

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).

Related Topics

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 appears for the attribute at a higher level in the hierarchy, it applies to the current level.

To delete the attribute entirely from the design, use the Attribute Dictionary on page 355.

Procedure

1. If the attribute is ECO-registered on page 1824, enter ECO mode. To enter ECO mode, click the **ECO Toolbar** button. The ECO Options Dialog Box appears. If you do not enter ECO mode first, a message appears, informing you that you must enter ECO mode.

   Tip
   You cannot edit read-only attributes on page 1853.

2. Select the object(s) to edit.
3. Right-click and click the **Attribute** popup menu item. The "Object Attributes Dialog Box" on page 1496 appears. Attribute information about the selected objects appears in spreadsheet form in the list.
4. From the Groups list click an attribute group on page 1811 to view. If you assign multiple attributes and some of those are structured attributes on page 1862, 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**.

   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.

Procedure

1. If the attribute is ECO-registered on page 1824, enter ECO mode. To enter ECO mode, click the **ECO Toolbar** button. The ECO Options Dialog Box appears. If you do not enter ECO mode first, a message appears, informing you that you must enter ECO mode.

   Tip
   You cannot edit read-only attributes on page 1853.

2. Select the object(s) to edit.
3. Right-click and click the **Attribute** popup menu item. The "Object Attributes Dialog Box" on page 1496 appears. Attribute information about the selected objects appears in spreadsheet form in the list.
4. From the Groups list click an attribute group on page 1811 to view. If you assign multiple attributes and some of those are structured attributes on page 1862, 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 key and the press the Enter key. The attribute is still assigned to the objects, but it has no value.

Adding Height Information to Design Components and Jumpers

Height information prevents components from being placed in height-constrained areas. Height information is also important when exporting a design to a 3-dimensional modeling application.

Tip
To quickly add height information to multiple objects, use the procedure in "Adding Attribute Values to Design Objects With the Attribute Manager".

Procedure

1. Select the object in the design.
2. Right-click, then click the **Attribute** popup menu item.
3. In the Object Attributes Dialog Box, click the **Add** button.

   Note:
   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.

Related Topics

Adding Height Information to Library Parts

Restricting Heights on Component Layers

Restricting Heights in Areas of Component Layers

Adding an Attribute to All Objects in the Design

Add an attribute at the PCB level of the attribute hierarchy to apply the attribute to all objects in the design.

Tip
Attributes at this level have the lowest priority in the attribute hierarchy. For more information, see "Attribute Hierarchy."

Procedure

1. In the design, select an object (for example, a component, component pad, net, or via), right-click and click the **Attribute** popup menu item.
2. In the Attributes for list, click PCB. Attributes of the PCB at the PCB level of the attribute hierarchy are displayed.
3. Click the **Add** button to add a new attribute.
4. In the new row, type a new attribute, or use the list to select a predefined attribute.
5. Type a value.
6. Click **Close**.

Chapter 19
Setting Rules and Using Keepouts

Read the topics that follow to learn more about setting up design rules for your design. You can also import rules from the schematic.

Design Rules

Design rules represent constraints, such as clearances and available routing layers, that support manufacturing, signal integrity, and other design requirements. SailWind Layout can perform design rule checking as you interactively place and route the design (called on-line design rule checking or just DRC), or after you complete design operations.

Most rules that you set up are used by On-line Design Rule Checking and can actively prevent you from making design errors. Some rules are only checked as a post-process using the Verify Design utility. Lastly, some rules are only used by the advanced capabilities of SailWind Router.

Setup Strategy

The first step in your strategy for setting up the rules of your design is to create a set of default rules.

The default rules are a set of rules that apply to all the objects in your design that will not have any other rules assigned. Objects with hierarchical rules take precedence over the default rules. For example, a pin pair rule with a Trace Width of 10 will override the default rule with a Trace Width of 5.

Note:
Larger clearance rules always have precedence despite the order of the hierarchy. For example, if you use a Default clearance rule of 50 mils, but a higher Net clearance rule of 10 mils, the more restrictive default clearance rule of 50 mils has precedence and is applied to the net even though a hierarchical Net clearance was created. Clearance rules should be less restrictive at lower levels of the hierarchy. They should be a minimum clearance value.

After the Default rule set has been created, create hierarchical rules based on the requirements of your design. Any hierarchical rules that are created are based on the Default set.

Design Rule Hierarchy

The design rule hierarchy enables you to specify a default set of rules for the entire design and additional rules for individual objects or collections of objects.

For example, you might specify in the default design rules a small trace width that would be adequate for signal nets, and then specify a large trace width for a class on page 1816 composed of power nets.

Because objects can be subject to multiple design rules, the design rule hierarchy specifies the priority of the rule types, so you can predict which design rule is applied to an object.

Basic Object Hierarchy

The objects in the design are defined using a basic object or relational hierarchy.

The design hierarchy can be easily visualized in the Rules Dialog Box.

The lowest level of the hierarchy begins with the Default level. Once you set the Default rules, they apply to all design objects, unless you customize the rules at a higher level of the hierarchy. In the figure below, each object to the right of the Default rules takes precedence over the object before it. For example, the Net level has a higher priority than the Class level, which has only a higher priority than the Default level. The Component level of the hierarchy has the highest priority in the hierarchy.

Figure 27. Basic Object Hierarchy

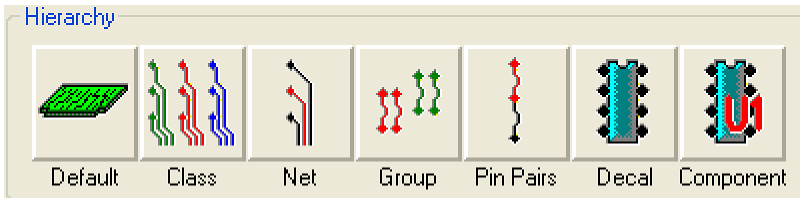


Table 93. Hierarchy Descriptions

Hierarchy level	Description
Default	Rules that apply to an object if there are no other individually defined rules.
Class	Rules for a collection of nets, called a class, that need identical rules.
Net	Rules for a specific net.
Group	Rules for a collection of pin pairs, called a group, that need identical rules.
Pin Pair	Rules for a specific pin pair. Usage example: A net needs different trace widths for certain pin-to-pin connections within it. You can apply rules to only those selected pin-to-pin connections.
Decal	Rules for all components using a specific decal.
Component	Rules for a specific component.

Complete Hierarchy

The complete hierarchy is not limited to the seven buttons in the Rules dialog box.

Conditional Rules add many levels in between and are used extensively in the complete design rule hierarchy. Conditional design rules are rules that come into effect when an object is adjacent to another object named in the conditional rule and/or on a layer named in the conditional rule. For example: Change a trace at any point in its length depending on the layer on which it resides. Net A may be assigned a default clearance of 12, but may need to reduce to 8 when it enters layer 4. In this case, you can assign 12 as the default for net A and make a conditional rule which applies an 8 mil clearance to net A against layer 4.

Although the hierarchy is more extensive than the basic object hierarchy as seen in the Rules dialog box, the relationships that are found in the basic object hierarchy are still maintained by the conditional rules. For example, Net rules have a priority over Class rules in the basic hierarchy. So, a net to net conditional rule also has priority over a class to class rule.

The following list is the complete hierarchy when all possible design rules are involved. This lists the priority for all rules from lowest to highest. Default rules, at level 1, have the lowest priority and represent the lowest level of the rules hierarchy. Component rules, at level 32, have the highest priority and represent the highest level of the rules hierarchy.

Note:
Electrical net rules are not part of the rules hierarchy.

1. Default rules on page 416
2. Default clearance rules on a specific layer (Conditional) on page 416
3. Class rules on page 418
4. Class clearance rules on a specific layer (Conditional) on page 422
5. Net rules on page 423
6. Net clearance rules on a specific layer (Conditional) on page 424
7. Group rules on page 426
8. Group clearance rules on a specific layer (Conditional) on page 430
9. Pin Pair rules on page 431
10. Pin Pair clearance rules on a specific layer (Conditional) on page 432
11. Class against Class rules (Conditional) on page 433
12. Class against Class rules on a specific layer (Conditional) on page 433
13. Net against Class rules (Conditional) on page 435
14. Net against Class rules on a specific layer (Conditional) on page 435
15. Net against Net rules (Conditional) on page 437
16. Net against Net rules on a specific layer (Conditional) on page 437
17. Group against Class rules (Conditional) on page 439
18. Group against Class rules on a specific layer (Conditional) on page 439
19. Group against Net rules (Conditional) on page 441
20. Group against Net rules on a specific layer (Conditional) on page 441

Restriction:
Decal rules are not used in SailWind Layout.

Restriction:
Component rules are not used in SailWind Layout.

33. Differential pair rules

Restriction:
Only the gap value applies to the hierarchy; it is considered as the equivalent of the trace to trace clearance.

Design Rule Categories

SailWind Layout provides five possible categories of design rules. Not all categories are available at all levels of the basic hierarchy.

For example, the Fanout and Pad Entry categories are only available at the Default, Decal, and Component levels.

Figure 28. Rule Category Icons

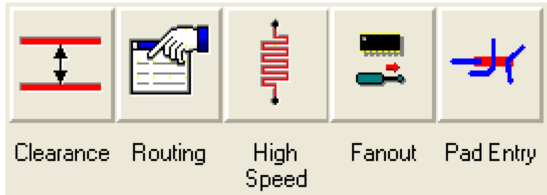


Table 94. Design Rule Category Descriptions

Rule Category	Description
Clearance rules on page 1167	Set the minimum allowable air gap between various object types in the design, such as trace to trace and via to trace. **Tip** You can also define conditional clearance rules that apply when an object is adjacent to a specified object, or when an object is on a specified layer. For example, you can use conditional rules to specify different clearance rules for a net when it is routed on different layers.
Routing rules on page 1666	Topology on page 1867 type, assign and prohibit via types, layers available for routing, specify maximum number of vias per net, specify topology types, and allow or prohibit autorouting.
High Speed rules on page 1402	Parallelism on page 1845, shielding on page 1859, geometric and electrical constraints, and length matching. on page 1837 Set the minimum and maximum parameters for advanced design rules, such as parallelism, delay, and capacitance. You can pass these rules from the schematic or assign them in SailWind Layout. You can also pass clearance and routing rules to autorouters. The router must be able to interpret the passed rules. **Tip** You can also define conditional high-speed rules that apply when an object is adjacent to a specified object, or when an object is on a specified layer. For example, you can use conditional rules to specify different high-speed rules for a net when it is routed on different layers.
Fanout rules on page 1375	Adding copper shapes automatically to SMD on page 1860 pads to make routing easier and to ensure that connections are made. **Restriction:** These rules are only used by SailWind Router
Pad Entry on page 1564	Location and angles for pad-to-trace intersections, and rules for placing vias on SMD on page 1860 pads. **Tip** These rules are only used by SailWind Router

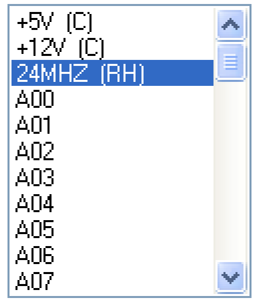
Non-Default Rules Indicators

When you change rules in the hierarchy from the default rules, two types of indicators appear.

The indicators visually show that rules have been customized from the default rules.

- The categories of rules customized at the current rule level are displayed parenthetically beside the class, net, group, or pin pair name in the list.

Figure 29. Non-Default Parenthetical Indicators



C = Clearance rules

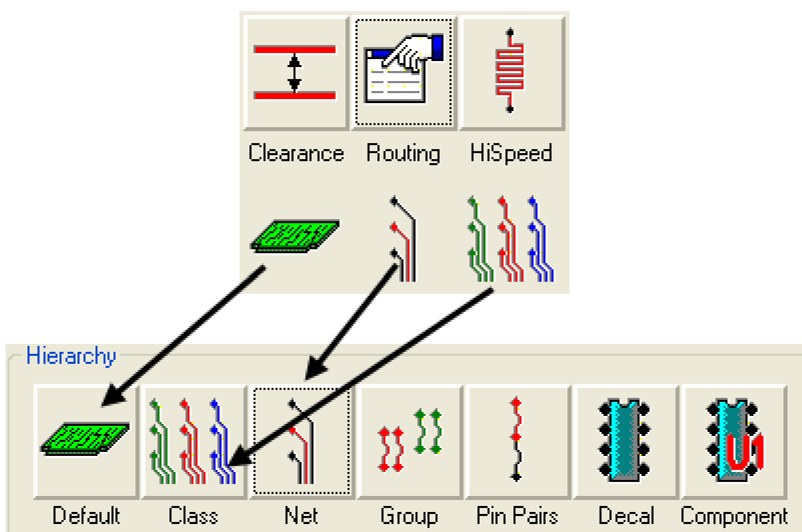
R = Routing rules

H = High speed rules

Example: A net listing followed by (RH) has non-default routing and high speed rules definitions.

- The icon that appears below each rule category button indicates whether the rules are customized at the current rule level or if it is only customized at another rule level.

Figure 30. Non-Default Icon Indicators



The meaning of the icon corresponds to the button in the Hierarchy area of the Rules dialog box.

Example: If a green polygon appears below the Clearance button, then the default values apply at the current level of the hierarchy for the object you've selected.

Advanced Rules Option

The Advanced Rules option is a license-enabled modular add-on that extends the restricted rule set to allow use of the full rule set.

The full set of constraints allow for control of more complex designs. Without the Advanced Rules option, you only have access to the Default, and Net design rules. With the Advanced Rules options, your license enables the Class, Group, Pin Pairs, Decal, Component, Conditional Rules, and Differential Pairs Rules.

When you use Advanced Rules, the Hierarchical dialog boxes on the Design Rules dialog box are available. You can read in hierarchically assigned rules from a netlist and edit and save rule changes using the hierarchy.

If you do not have the Advanced Rules option, the Hierarchical dialog boxes on the Design Rules dialog box are not available. However, schematic-applied hierarchical rules reside in the netlist, and they can be passed to any autorouter that can interpret them.

Design Rule Checking

On-line Design Rule Checking (DRC) provides continuous rule checking during routing and placement.

On-line DRC does not check clearance for text, open associated copper, or free copper belonging to a decal. The following table shows the four DRC modes.

Table 95. Design Rule Checking Modes

Mode	Purpose
Prevent Errors	Prevents you from completing any operation that will create errors. You cannot paste items from the paste buffer. You cannot create or change (move, split, or miter, for example) the following items: board outlines, board cutouts, text, copper, and keepouts. To edit the Properties of these items, turn off On-line DRC using the modeless command, DRO. You can also right-click and click the **Ignore Clearance** popup menu item to temporarily override DRC and complete an operation. When you are done, On-line DRC automatically resets to Prevent mode.
Warn Errors	Warns that a design rule violation has occurred by highlighting the rule obstacles.
Ign Clrn	Ignores clearance checking.
Off	DRC is completely deactivated. When On-line DRC is off, program performance is enhanced. However, you are limited to the Route command for trace editing. When you turn On-line DRC on again, SailWind Layout pauses to map the board. The undo buffer is cleared when you turn On-line DRC off, but is still available for further edits.

To turn On-line DRC on use the **Design** category in the Options dialog box or the modeless command, DRP or DRW. After activating On-line DRC, you can change the setting using the dialog box, or the modeless commands.

When On-line DRC is activated, an octagon shaped guard band appears at the end of the route to indicate any clearance violation. You can hide the guard band using the Show Guard Band option in the Options dialog box > **Routing** category > **General** subcategory.

On-line Design Rule Checking is available on most SailWind Layout configurations. For Advanced Design Rules package users, all hierarchical rules, except High Speed EDC settings, are monitored.

Clearance Checking Against Text

For clearance checking against text, a rectangle referred to as an extent box is drawn around the text string, calculated as a smallest rectangle that will contain the text string. If a text string is rotated, the extent box is also rotated.

Text strings included in clearance checking are those that exist on the same level as other checked objects, or created to appear and on all layers.

CAUTION:
Text belonging to a decal is not checked for clearance by On-line DRC.

Clearance Checking Against Copper

Clearance checking against copper includes closed or open copper that is not part of the current net being routed and exists on the same layer as other checked objects. Closed copper in a decal that is associated with a terminal is checked for clearance by On-line DRC.

CAUTION:
On-line DRC does not check the clearance of open associated copper and free copper belonging to a decal.

Design Rules Transfer

Schematic and design files store the design rules you create. Design rules set up in SailWind Logic transfer automatically when you import the netlist into SailWind Layout. Design rules you set up in SailWind Layout become available automatically when you open the design in SailWind Router.

Note:
SailWind Logic supports only default, class, and net rules.

Import and Export of Design Rules

You can export and import design rules for the purpose of reusing them in future designs that require the same constraints.

- To export design rules, follow the instructions to export an ASCII file on page 291, 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 on page 290.

Related Topics

Check Design Rules

Creation of Rules for Your Design

SailWind Layout provides many levels of rules in the rules hierarchy. You can create rules at each level of the hierarchy.

Restriction:
The links below are in hierarchical order from lowest to highest priority. For more information, see "Design Rule Hierarchy".

Links to Procedures for Creating all Rule Types

Default Rules

- Creating Default Rules
- Creating Default Clearance-Rules for a Specific Layer (Conditional Rule)

Class Rules

- Class Design Rules
- Creating Class Clearance Rules for a Specific Layer (Conditional Rule)

Net Rules

- Net Design Rules
- Creating Net Clearance Rules for a Specific Layer (Conditional Rule)

Group Rules

- Group Design Rules
- Creating Group Clearance Rules for a Specific Layer (Conditional Rule)

Pin Pair Rules

- 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

Design Rules Setup

Read the sections that follow to learn how to set up each of the different design rules types in SailWind Layout.

Default Design Rules
Class Design Rules
Net Design Rules
Group Design Rules
Pin Pair Design Rules
Class Against Class Design Rules
Net Against Class Design Rules
Net Against Net Design Rules
Group Against Class Design Rules
Group Against Net Design Rules
Group Against Group Design Rules
Pin Pair Against Class Design Rules
Pin Pair Against Net Design Rules
Pin Pair Against Group Design Rules
Pin Pair Against Pin Pair Design Rules
Decal Design Rules
Component Design Rules
Deletion or Modification of Conditional Design Rules
Differential Pair Design Rules

Default Design Rules

When it comes to creating default design rules, you can create default rules and apply them to all layers of a design or you can create default design rules and apply them to specific layers.

Creating Default Rules

Each new design already has a default set of rules or constraints. The default set either comes from the SailWind Layout *default.asc* template file or imports 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 on page 416.

Procedure

1. Click the **Setup > Design Rules** menu item.

2. In the Rules Dialog Box, click **Default**.

3. In the Default Rules Dialog Box, click any of the five rule category on page 407 buttons (Clearance, Routing, HiSpeed, Fanout, Pad Entry) to customize the rules from their default values and settings.

4. After you have customized the rule categories, close all Rules dialog boxes.

Related Topics

Design Rule Hierarchy

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 that apply to all layers.

Procedure

1. Click the **Setup > Design Rules** menu item, and then click the **Conditional Rules** button.

2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, click “All”.

3. In the Against rule object area, with “Layer” 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 **Matrix** 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

Class Design Rules

Read the sections that follow to learn more about the setup and application of the class design rules.

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 the **Make Class** popup menu item.

        iii. In the Add Net to Class on page 1058 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 on page 1162 dialog box:

        i. Click the **Setup > Design Rules** menu item, and then click **Class**.

        ii. In the Class name box, type the class name, and then click **Add**. The maximum class name length is 15 characters. You can use any alphanumeric characters except brackets {}, asterisks *, or spaces.

iii. 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>>**. 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. In the Class Rules Dialog Box (**Setup > Design Rules** menu item then **Class**), select the class in the Class box.

3. Click any of the three rule category on page 407 buttons (Clearance, Routing, HiSpeed) to customize the rules from their default values and settings.

4. After you have customized the rule categories, click **OK** to accept the changes and close the Class Rules dialog box.

Results

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

Deleting a Design Rule Class

Use the Class Rules dialog box to delete classes.

Procedure

1. Click the **Setup > Design Rules** menu item and then click Class.

2. In the Class Rules Dialog Box, in the Class box, select one or more classes, and then click **Delete**. 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

If you have already defined a class design rule, you can add nets to it as required.

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 the **Make Class** popup menu item.

     iii. In the Add Net to Class on page 1058 dialog box, click the Add to Existing Class option, and select the class in the Existing Classes list.

     iv. Click **OK**.

   - Select nets from the list in the Class Rules dialog box:

     i. Click the **Setup > Design Rules** menu item, then click **Class**.

     ii. In the Class Rules on page 1162 dialog box, in the Class box, select the class to which you want to add a net. To display all classes in the Class box, clear the Show Classes with Rules check box.

     iii. 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>>**. 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. Click the **Setup > Design Rules** menu item, then click the **Class** button.
2. In the Class Rules Dialog Box, in the Class box, select the class. 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

If desired, you can change or modify the rules of a class of nets.

Procedure

1. Click the **Setup > Design Rules** menu item, then click the **Class** button.
2. In the Class Rules Dialog Box, in the Class box, select the class.
3. Click any of the three rule category on page 407 buttons (Clearance, Routing, HiSpeed) to modify the values and settings.
4. After you have 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 change the name of a class.

Procedure

1. Click the **Setup > Design Rules** menu item, then click the **Class** button.
2. In the Class Rules Dialog Box, in the Class box, select a class. 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

If desired, you can reset the class rules to the same as the default rules.

Procedure

1. Click the **Setup > Design Rules** menu item, then click the **Class** button.
2. In the Class Rules Dialog Box, select one or more classes in the Class list, click **Default**, and then click **Yes**. The **Default** button is unavailable if the class already has only default rules assigned to it. You know you have successfully reset the class rules if all the icons below the rule categories on page 407 (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. Click the **Setup > Design Rules** menu item, then click the **Class** button.
2. In the Class Rules Dialog Box, in the Class list, click the name of the class. Nets in the class appear in the Selected list.
3. After you have 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 for a specific layer that take precedence over the class rules that apply to all layers.

Prerequisites

The Class of nets must already exist. For more information, see "Class Design Rules".

Procedure

1. Click the **Setup > Design Rules** menu item, then click **Conditional Rules**.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, click the Classes option and then select a class in the list.
3. In the Current rule set area, choose the Clearance option.
4. In the Against rule object area, choose the Layer option 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 **Matrix** 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

Net Design Rules

Read the topics that follow to learn more about the setup and application of net design rules.

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 the **Show Rules** popup menu item.

   - Select nets from the list in the Net Rules dialog box:

     i. Click the **Setup > Design Rules** menu item, then click the **Net** button.

     ii. In the Net Rules Dialog Box, in the Nets box, click one or more nets. To display all nets, clear the Show Nets with Rules check box.

2. Click any of the three rule category on page 407 buttons (Clearance, Routing, HiSpeed) to customize the rules from their default values and settings.

3. After you have customized the rule categories, close all rules dialog boxes to return to the design.

Results

After you customize a rule category for the net(s), Non-Default Rules Indicators appear in the dialog box.

Related Topics

Design Rule Hierarchy

Modification of Net Design Rules

You can modify net design rules to meet your design requirements.

Procedure

1. Click the **Setup > Design Rules** menu item, then click the **Net** button.
2. In the Net Rules Dialog Box, in the Nets box, click one or more nets with rules assigned.
3. Click any of the three rule category on page 407 buttons (Clearance, Routing, HiSpeed) to modify the values and settings.
4. If other nets have the same rules, you may be prompted with a dialog box to include the other nets in the changes you are making. Click one of the following:
    - **Yes** — the dialog goes away and the other nets are selected in the Nets list. You must click the rule cateory button again to proceed.
    - **No** — the rule category dialog opens for editing.
5. After you have customized the rule categories, click **OK** to accept the changes and close the Net Rules dialog box.
6. Click **Close** in the Rules dialog box.

Resetting Net Rules to Default Rules

You can reset the net rules to the same as the default rules.

Procedure

1. Click the **Setup > Design Rules** menu item, then click the **Net** button.
2. In the Net Rules Dialog Box, select one or more nets in the Nets list, click **Default**, and then click **Yes**. The **Default** button is unavailable if the net has only default rules assigned to it.

    You know you have 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. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 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, choose the Clearance option.
4. In the Against rule object area, choose the Layer option 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

Group Design Rules

The following topics describe the setup and application of the group design rules.

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 the **Make Group** popup menu item.

        iii. In the Add Pin Pairs to Group on page 1068 dialog box, choose the Create New Group option, 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:

        i. Click the **Setup > Design Rules** menu item and then click the **Group** button.

        ii. In the Group Rules dialog box, on page 1399 in the Group name box, type the group name, and then click **Add**. 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.

iii. 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. 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. In the Group Rules Dialog Box (**Setup > Design Rules** menu item and then click Group and select the group in the Group box.

3. Click any of the three rule category on page 407 buttons (**Clearance**, **Routing**, **HiSpeed**) to customize the rules from their default values and settings.

4. After you have customized the rule categories, click **OK** to accept the changes and close the Group Rules dialog box.

Results

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

Deleting a Design Rule Group

Use the Group Rules dialog box to delete a group in SailWind Layout, such as a group of pin pairs.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **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. 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

SailWind Layout allows you to group pin pairs together and apply design rules. You can create a new group or 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 the **Make Group** popup menu item.

     iii. In the Add Pin Pairs to Group on page 1068 dialog box, choose the Add to Existing Group option, and select the group in the Existing Groups list.

     iv. Click **OK**.

   - Select pin pairs from the list in the Group Rules dialog box:

     i. Click the **Setup > Design Rules** menu item and then click the **Group** button.

     ii. In the Group Rules Dialog Box, in the Group box, select the group to which you want to add a pin pair. To display all groups in the Group box, clear the Show groups with rules check box.

     iii. 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. 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

If necessary, you can remove a set of pin pairs from a group if you do not want to apply the group design rules to it. Use the Group Rules dialog box to remove pin pairs from a group.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Group** button.
2. In the Group Rules Dialog Box, in the Group box, select the group. 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 change or edit group design rules and apply them to all objects in the group.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Group** button.

2. In the Group Rules Dialog Box, in the Group box, select the group.

3. Click any of the three rule category on page 407 buttons (**Clearance**, **Routing**, **HiSpeed**) to modify the values and settings.

4. After you have 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

You can change the name of a design rules group by using the Group Rules dialog box.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Group** button.

2. In the Group Rules Dialog Box, in the Group box, select a group. 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

You can set a group's design rules back to their default rule settings.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Group** button.

2. In the Group Rules Dialog Box, select one or more groups in the Group list, click **Default**, and then click **Yes**. The **Default** button is unavailable if the class already has only default rules assigned to it.

   You know you have successfully reset the group rules if all the icons below the rule categories on page 407 (**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. Click the **Setup > Design Rules** menu item and then click the **Group** button.

2. In the Group Rules Dialog Box, in the Group list, click the name of the group. Pin pairs in the group appear in the Selected list.

3. After you have 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.

Prerequisites

The Group of pin pairs must already exist. For more information, see “Group Design Rules”.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.

2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Groups option and then select a group in the list.

3. In the Current rule set area, click Clearance.

4. In the Against rule object area, choose the Layer option 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

Pin Pair Design Rules

Read the topics that follow to learn more about the setup and application of the pin pair design rules.

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 the **Show Rules** popup menu item.

    - Select nets from the list in the Pin Pair Rules dialog box:

        i. Click the **Setup > Design Rules** menu item and then click the **Pin Pairs** button. If you want to display all pin pairs, clear the Show pin pairs with rules check box.

        ii. In the Pin Pair Rules Dialog Box, in the Connections box, click one or more pin pairs. To display pin pairs for a specific net, select the net in the From Net list.

2. Click any of the three rule category on page 407 buttons (**Clearance**, **Routing**, **HiSpeed**) to customize the rules from their default values and settings.

3. After you have customized the rule categories, close any rules dialog boxes to return to the design.

Results

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

Modification of Pin Pair Design Rules

If necessary, you can edit pin pair design rules to support your specific design requirements.

Follow the procedure in the topic "Creating Pin Pair Design Rules" on page 431.

Resetting Pin Pair Rules to Default Rules

If desired, you can reset the pin pair design rules to back to their default settings.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **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**. The **Default** button is unavailable if the pin pair has only default rules assigned to it.

   You know you have successfully reset the pin pair rules if all the icons below the rule categories on page 407 (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 that apply to all layers.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Pin pairs option and then select a pin pair in the list.
3. In the Current rule set area, choose the Clearance option.
4. In the Against rule object area, choose the Layer option 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

Class Against Class Design Rules

Read the topics that follow to learn more about the setup and application of the class against class design rules.

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.

Prerequisites

Both Classes of nets must already exist. For more information, see "Class Design Rules".

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Classes option and then select a class in the list.
3. In the Current rule set area, choose the Clearance or the High Speed option.
4. In the Against rule object area, choose the Classes option 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

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. Such a clearance rule takes precedence over class against class clearance rules that apply to all layers.

Prerequisites

Both Classes of nets must already exist. For more information, see "Class Design Rules".

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Classes option and then select a class in the list.
3. In the Current rule set area, choose the Clearance option.
4. In the Against rule object area, choose the Classes option 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

Net Against Class Design Rules

Read the topics that follow to learn more about the setup and application of the net against class design rules.

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.

Prerequisites

The Class of nets must already exist. For more information, see "Class Design Rules".

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Nets option and then select a net in the list.
3. In the Current rule set area, choose the Clearance or the High Speed option.
4. In the Against rule object area, choose the Classes option 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

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. Such clearance rules take precedence over net against class clearance rules that apply to all layers.

Prerequisites

The Class of nets must already exist. For more information, see "Class Design Rules".

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Nets option and then select a net in the list.
3. In the Current rule set area, choose the Clearance option.
4. In the Against rule object area, choose the Classes option 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

Net Against Net Design Rules

Read the topics that follow to learn more about the setup and application of the net against net design rules.

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. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Nets option and then select a net in the list.
3. In the Current rule set area, choose the Clearance or the High Speed option.
4. In the Against rule object area, choose the Nets option 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

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. Such design rules take precedence over net against net clearance rules that apply to all layers.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Nets option and then select a net in the list.
3. In the Current rule set area, choose the Clearance option.
4. In the Against rule object area, choose the Nets options 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

Group Against Class Design Rules

Read the topics that follow to learn more about the setup and application of the group against class design rules.

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.

Prerequisites

The Group of pin pairs and the Class of nets must already exist. For more information, see "Group Design Rules" and "Class Design Rules".

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Groups option and then select a group in the list.
3. In the Current rule set area, choose the Clearance or the High Speed option.
4. In the Against rule object area, choose the Classes option 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

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. Such clearance rules take precedence over group against class clearance rules that apply to all layers.

Prerequisites

The Group of pin pairs and the Class of nets must already exist. For more information, see "Group Design Rules" and "Class Design Rules".

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Groups option and then select a group in the list.
3. In the Current rule set area, choose the Clearance option.
4. In the Against rule object area, choose the Classes option 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

Group Against Net Design Rules

Read the topics that follow to learn more about the setup and application of the group against net design rules.

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.

Prerequisites

The Group of pin pairs must already exist. For more information, see “Group Design Rules”.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Groups option and then select a group in the list.
3. In the Current rule set area, choose the Clearance or the High Speed option.
4. In the Against rule object area, choose the Nets option 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

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. Such clearance rules take precedence over group against net clearance rules that apply to all layers.

Prerequisites

The Group of pin pairs must already exist. For more information, see “Group Design Rules”.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Groups option and then select a group in the list.
3. In the Current rule set area, choose the Clearance option.
4. In the Against rule object area, choose the Nets option 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

Group Against Group Design Rules

Read the topics that follow to learn more about the setup and application of the group against group design rules.

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.

Prerequisites

The Group(s) of pin pairs must already exist. For more information, see "Group Design Rules".

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Groups option and then select a group in the list.
3. In the Current rule set area, choose the Clearance or the High Speed option.
4. In the Against rule object area, choose the Groups option 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

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. Such rules take precedence over group against group clearance rules that apply to all layers.

Prerequisites

The Group(s) of pin pairs must already exist. For more information, see "Group Design Rules".

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Groups option and then select a group in the list.
3. In the Current rule set area, choose the Clearance option.
4. In the Against rule object area, choose the Groups option 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

Pin Pair Against Class Design Rules

Read the topics that follow to learn more about the setup and application of the pin pair against class design rules.

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.

Prerequisites

The Class of nets must already exist. For more information, see “Class Design Rules”.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Pin Pairs option and then select a pin pair in the list. 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, choose the Clearance or the High Speed option.
4. In the Against rule object area, choose the Classes option 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

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. Such rules take precedence over pin pair against class clearance rules that apply to all layers.

Prerequisites

The Class of nets must already exist. For more information, see “Class Design Rules”.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Pin Pairs option and then select a pin pair in the list. 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, choose the Clearance option.
4. In the Against rule object area, choose the Classes option 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

Pin Pair Against Net Design Rules

Read the topics that follow to learn more about the setup and application of the pin pair against net design rules.

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. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Pin Pairs option and then select a pin pair in the list. 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, choose the Clearance or the High Speed option.
4. In the Against rule object area, choose the Nets option 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

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. Such rules take precedence over pin pair against net clearance rules that apply to all layers.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Pin Pairs option and then select a pin pair in the list. 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, choose the Clearance option.
4. In the Against rule object area, choose the Nets option 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

Pin Pair Against Group Design Rules

Read the topics that follow to learn more about the setup and application of the pin pair against group design rules.

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.

Prerequisites

The Group of pin pairs must already exist. For more information, see "Group Design Rules".

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Pin Pairs option and then select a pin pair in the list. 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, choose the Clearance or the High Speed option.
4. In the Against rule object area, choose the Groups option 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

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. These rules take precedence over pin pair against group clearance rules that apply to all layers.

Prerequisites

The Group of pin pairs must already exist. For more information, see "Group Design Rules".

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Pin Pairs option and then select a pin pair in the list. 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, choose the Clearance option.
4. In the Against rule object area, choose the Groups option 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

Pin Pair Against Pin Pair Design Rules

Read the topics that follow to learn more about the setup and application of the pin pair against pin pair design rules.

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. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Pin Pairs option and then select a pin pair in the list. 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, choose the Clearance or the High Speed option.
4. In the Against rule object area, choose the Pin Pairs option and then select a pin pair from the list. 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

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. These rules take precedence over pin pair against pin pair clearance rules that apply to all layers.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Source rule object area, choose the Pin Pairs option and then select a pin pair in the list. 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, choose the Clearance option.
4. In the Against rule object area, choose the Pin Pairs option and then select a pin pair from the list. 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

Decal Design Rules

Read the topics that follow to learn more about the setup and application of the decal design rules.

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.

Restrictions and Limitations

- You can define Decal Rules in SailWind Layout; however, these rules are used in SailWind Router only.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Decal** button.
2. In the Decal Rules Dialog Box, in the Decals box, click one or more decals. To display all decals, clear the Show decals with rules check box.
3. Click any of the four rule category on page 407 buttons (Clearance, Routing, Fanout, Pad Entry) to customize the rules from their default values and settings.
4. After you have customized the rule categories, close any rules dialog boxes to return to the design.

Results

After you customize a rule category for the class, Non-Default Rules Indicators appear in the dialog box.

Related Topics

Design Rule Hierarchy

Modification of Decal Design Rules

If necessary, you can edit or change decal design rules to support your specific design requirements.

Follow the procedure in the topic “Creating Decal Design Rules”.

Resetting Decal Rules to Default Rules

If desired, you can reset the decal rules back to their default settings.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Decal** button.
2. In the Decal Rules Dialog Box, select one or more decals in the Decals list, click **Default**, and then click **Yes**. The **Default** button is unavailable if the decal has only default rules assigned to it.

   You know you have successfully reset the decal rules if all the icons below the rule categories on page 407 (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. SailWind Layout saves the design rules with the decal in the library. The design rules become active whenever you use the decal in a design.

Restrictions and Limitations

You can define Decal Rules in SailWind Layout; however, these rules are used in SailWind Router only.

Procedure

1. In the PCB Decal Editor, click the **Setup > Decal Rules** menu item.
2. In the Decal Rules dialog box on page 1231, click any of the four rule category on page 407 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.

Results

After you customize a rule category for the decal(s), Non-Default Rules Indicators appear in the dialog box.

Related Topics

Design Rule Hierarchy

Component Design Rules

Read the topics that follow to learn more about the setup and application of the component design rules.

Creating Component Design Rules

Use the Component Rules dialog box to create design rules for one or more components.

Restrictions and Limitations

- You can define Component Rules in SailWind Layout; however, these rules are used in SailWind 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 the **Show Rules** popup menu item.

   - Select components from the list in the Component Rules dialog box:

     i. Click the **Setup > Design Rules** menu item and then click the **Component** button.

     ii. In the Component Rules Dialog Box, in the Components box, select one or more components. To display all components in the list, clear the Show components with rules check box. To display components using a specific decal, select the decal in the "Using decal" list.

2. Click any of the four rule category on page 407 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.

Results

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

Modification of Component Design Rules

You can edit or change component design rules as necessary to support the specific requirements of your design.

Follow the procedure in the topic “Creating Component Design Rules”.

Resetting Component Rules to Default Rules

If necessary, you can set the component rules back to their default settings.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Component** button.
2. Select one or more components in the Components list, click **Default**, and then click **Yes**. The **Default** button is unavailable if the component has only default rules assigned to it.

   You know you have successfully reset the component rules if all the icons below the rule categories on page 407 (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.

Restrictions and Limitations

- When you open the Routing Rules dialog box from the Decal Editor, only options in the Vias area are available.

Procedure

1. In the PCB Decal Editor, click the **Setup > Design Rules** menu item and then click the **Routing** button.
2. In the Routing Rules Dialog Box, in the Vias area, click the **Via definition** button.
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. 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

Creating and Editing PCB Decals

Deletion or Modification of Conditional Design Rules

Read the topics that follow to learn more about deleting or modifying conditional design rules.

Deleting a Conditional Rule

If necessary, use the Conditional Rule Setup dialog box to delete conditional rules that you have created.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 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.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Conditional Rules** button.
2. In the Conditional Rule Setup on page 1199 dialog box, in the Existing rule sets list, select the rule to modify. 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.

   Note:
   For reporting purposes, nets and pin pairs in the Source rule object list are identified as aggressors on page 1808. If a class is in the Source rule object list, all nets in the class are identified as aggressors.

   CAUTION:
   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

Differential Pair Design Rules

Read the topics that follow to learn more about the setup and application of the differential pair design rules.

Creating Differential Pair Design Rules

You can define differential pair rules for nets, pin pairs, or electrical nets that behave electrically as differential pairs.

Restrictions and Limitations

- You can define differential pair rules in SailWind Layout or SailWind Router; however, these rules are used in SailWind Router only. To verify differential pairs in the design, you must check them in Latium checking on page 883.

Procedure

1. Click the **Setup > Design Rules** menu item and then click the **Differential Pairs** button.
2. In the Differential Pairs Dialog Box, choose the **Net**, **Pin pairs**, or **Elecrical Nets** tab.
3. In the Available list, double-click the first net, and then double-click the second net. Double-clicking is a shortcut instead of clicking the **Select>>** button.

Note:
Nets, pin pairs, and electrical nets 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. Click the **Setup > Design Rules** menu item and then click the **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. Click the **Setup > Design Rules** menu item and then click the **Report** button.

2. In the Rules Report dialog box, on page 1677 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**.

Results

The report is written to *C:\SailWind Projects\rules.rep* and displayed by the default text editor.

Turning on Design Rule Checking

With *Online DRC*, you can enable live design rule checking as you work on your design. Online DRC prevents or reports design rule violations during interactive placement or routing.

Use Online DRC to constrain actions and avoid making design errors. You can run the Verify Design on page 1775 tool to search for and locate errors at any time.

Restrictions and Limitations

- There are many design rules that are not checked by Online DRC. Some are only checked by SailWind 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 Online DRC.

- Online DRC does not prevent you from moving objects in a flooded area. If you don't re-flood the area, Verify Design reports errors. If objects within a flooded area were moved, or thermal settings were changed since the original flooding, you need to flood the area again.

Procedure

1. Click the **Tools > Options** menu item > Design category on page 1503.

2. In the Online DRC area, select a setting.

> **Tip**
> You can also initiate design rule checking using one of the Design Rule Checking modeless commands (drp, drw, dri, or dro). on page 1479

Check Design Rules

You can perform a design rules check during interactive place and route operations, or after place and route operations are complete.

For more information, see "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. For more information, see "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.

> **Restriction:**
> You cannot use a height restriction of 0 (zero) to prevent placement on a layer. A height restriction of 0 specifies 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.

> **Tip**
> 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.

Restrictions and Limitations

- A board height restriction does not 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. Click the **Tools > Options** menu item, then click the **Text and Lines** 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

Adding Height Information to Design Components and Jumpers

Adding Height Information to Library Parts

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 and Limitations

- A board height restriction does not 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 on page 465 in the area of the board where you want to restrict component height.

3. In the Add Drafting dialog box, in the Restrictions area, select the Placement and the Component height check boxes.
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

Adding Height Information to Design Components and Jumpers

Adding Height Information to Library Parts

Restricting Heights on Component Layers

Keepouts

Keepouts prevent the placement of design items within a specified area. Keepouts appear as one pixel-width lines during interactive placement, cluster placement, routing, and other operations. The outer edge of design objects and board outlines/cutouts appear for use in clearance calculations against keepouts. Both the Online Design Rule Checking and Verify Design tools 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.

CAUTION:
The three component type keepouts can only be created in the Layout Editor.

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.

Note:
When defining keepouts in the PCB Decal Editor, you can also assign keepouts to an <Opposite Side>. You cannot do this in the Layout Editor. Placement, Component Height, and Component Drill check boxes are unavailable in the PCB Decal Editor.

Procedure

1. On the Drafting Toolbar, then click the **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 on page 1054that 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.

Results

The keepout is displayed with a widely spaced hatch filling the area. If the fill you are using for Copper or Copper Planes is similar and you find it difficult to distinguish them, you can change the direction of the keepouts to be the reverse in the “Hatch and Flood” on page 1511 options.

Related Topics

Creating Keepout Areas in a Decal

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 edge of a keepout, right-click and click the **Select Shape** popup menu item.
2. Right-click and click the **Properties** popup menu item.
3. In the Drafting Properties dialog box, turn restrictions on or off and modify the layer settings. For more information, see “Drafting Object Properties.”
4. Click **OK**.

Tip
You cannot 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

Restricting Heights on Component Layers

Attribute Dictionary

Chapter 20
AI-Powered Advanced Functionality

Read the topics that follow to learn more about the procedures for executing algorithmn-driven layout.

Layout Area Assessment

Use this feature to automatically place components within the board outline, adhering to the layout rules while minimizing the occupied space. It can be triggered by the following two methods:

Triggered by Menu Bar

Prerequisites

There are unglued components outside the board outline.

Procedure

1. Click the **AI > Layout Area Assessment** menu item.
2. Configure layout rules, including:
    - **One-sided Layout** — Select this option to enable the single-sided layout.

      By default, this option is unchecked for double-sided layout.
    - **Allow 45 Degrees** — Select to allow rotating components by 45 degrees.
    - **Border Spacing** — Specify the minimum clearance from the component body to the board outline.
    - **Max Iteration Number** — Specify the number of algorithm iterations, with an upper limit of 100.
3. Add classes to include decals and define spacing rules between classes.
    a. Add classes: In the Class Decal Name box, type the name of the class, and then click **Add** to add the class name to the Class list.

       Select a class name in the Class list, you can also:
       - Click **Rename** to rename the class selected in the list with the text in the Class Decal Name box.
       - Click **Delete** to delete the selected item.

    b. Include PCB decal types to the class: Select a class in the list, and then choose decals from the Usable list to add to the Selected list.
       - Use Ctrl/Shift for multiple selections.
       - Click **Remove** to move the class from the Selected list to the Usable list.

c. Define spacing rules between classes: In the Spacing Rule of the Same Surface Element area, define the clearance rule between classes by double-clicking the table cell.

> **Tip**
> To set the same value for an entire column or the table, click on a column heading or All.

4. Click **Run** to perform the layout operation. A hyperlink to the report file appears in the Output window, on the **Status** tab.

   During runtime, you can:

   - Check both the current and maximum iteration count, as well as area utilization rate in the Status Bar.
   - Click the **Cancel** button or close the dialog box to terminate the process, placing components based on the current operating state.

5. (Optional) Click the **Save** button to save all settings for the next run.

Triggered by Right-Click Menu

Prerequisites

There are unglued components outside the board outline.

Procedure

1. Select components for layout. Use Ctrl+click to select multiple non-sequential items, or drag the cursor to select a range of items.
2. Right-click and select the **Layout Area Assessment** popup menu.
3. Click the board outline or 2D line item to define the layout area outline as prompted.

   The selected component is placed based on the options configured in the Layout Area Assessment Dialog Box, except for the maximum iteration number (1 for this case).

Intelligent Layout

Use the Layout Area Assessment feature for minimal layout area.

Prerequisites

- The **AI layout reference data** option is enabled when sending a SailWind Logic netlist using the SailWind Layout Link.
- A board outline must be defined in the design.

Procedure

1. Click the **AI > Intelligent Layout** menu item.
2. In the Intelligent Layout dialog box, a table lists the intelligent identified component groups along with their layout state and rule set correspondingly.

   Note:
   Rule set is not currently supported.

3. (Optional) Click the **Rule Setup** button to configure layout options on demand, including:
   - **Allow rotation** — Select to allow rotating components by 90 degrees.
   - **Layout style** — Select the layout style from the drop-down list.
   - **One-sided Layout** — Select this option to enable the single-sided layout.

     By default, this option is unchecked for double-sided layout.
   - **Key part safety spacing** — Specify the minimum clearance from the intelligently identified key component within a group.

   Tip
   TSkip this step t set.

4. Click the **Module Analysis** button to initiate the module analysis.
5. Click the **Module Layout** button to initiate the layout process.

   When the process is completed, the Layout State changes to "Layout completion".
6. Click the **Cluster Create** button to create clusters from the component groups.
7. Click the **Cluster Layout** button to open the Cluster Placement Dialog Box.

Chapter 21
Part Placement

Read the topics that follow to learn more about the multiple methods and strategies available for part placement operations. You can also learn more about the operations of the part placement commands, as well as information on changing or modifying your placement, dispersion, and the use of unions and clusters.

Placement Strategy

Parts placement depends on multiple factors. In addition to connection length and spacing tolerances, manufacturing considerations such as accessibility for pick-and-place machines and solder treatment also affect placement. After you determine your manufacturing strategy and import your netlist, use the interactive placement tools to optimize your placement scheme.

Tools to aid in parts placement include:

- Find function that locates and selects parts by reference designator or location
- Separate grid settings for x and y coordinates, set by a modeless command
- Interactive part alignment function
- Nudge interactive parts shoving, with DRC enabled
- Net length minimization
- Connection length minimization
- Automatic and interactive part, pin, and gate swapping
- Automatic part renumbering

Remember to backward annotate engineering change order (ECO) changes to the schematic.

Placement Guidelines

A number of guidelines will aid you in the development of your component placement design.

A good placement scheme:

- Observes all the restrictions placed on component location and trace routing.
- Provides the easiest routing solutions.

Follow this general order for placement:

1. Place all components that have a fixed location, such as connectors, ICs that require a specific location, and mounting holes.
2. Because of their typically heavy connectivity pattern, connectors need sufficient room between other connectors and the first row of ICs to exit the connector. Allow at least 0.5 mils between the connector and the nearest row of ICs.
3. Place components that have engineering restrictions, such as parts that must be a certain distance from each other, parts that cannot be located next to each other, and parts that must be near the connector.
4. Place components with no overriding placement requirements. Follow these general guidelines:

◦ A part's pads should fall on the grid used during autorouting. Some standard increments are 100, 50, 25, 20, 16.67, 12.5, or 10 mils. Set up the design grid before placing components.

◦ The flow of connections should be vertical and horizontal. Minimize diagonal connections.

◦ Component pads should line up vertically and horizontally so they do not block routing channels. If two ICs are side by side with their pads staggered 50 mils, the pads of one IC block the routing channels of the other.

◦ For manufacturing reasons, it is better to have all of the ICs on the board oriented in the same direction, rather than some vertical and some horizontal.

Component Placement Process

SailWind Layout initially overlays all parts 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 SailWind Router and use it to do all of your placements.

Restriction:
You cannot use unions, clusters, or reuse items in SailWind Router to assist with placement.

- Click the **Tools > Disperse Components** menu item to disperse all the components around the board outline.

Tip
Click the **View > Extents** menu item after you disperse components to bring all design objects into view.

- Move components in the design. For more information, see "Moving Components".
- Orient parts radially on a polar grid. For more information, see "Setting Up a Polar Grid".
- Move multiple components sequentially. For more information, see "Sequential Component Placement".
- Flip components to the bottom layer. For more information, see "Modify the Board-Side Location of Components, Bottom View".
- Array components. For more information, see "Creating a Component Array".
- Align edges or centerlines of multiple components. For more information, see "Aligning Objects".

- Rotate objects in 90 degree increments or at any angle. For more information, see "Rotating an Object, Spinning an Object".
- Swap the locations of two parts. For more information, see "Swapping 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. For more information, see "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.

You can set the origin of an object using the popup menus when creating a Dimensioning Shape or Drafting Shape.

Procedure

1. Select the whole shape.

Tip
Right-click and click the **Select Shape**, **Select Documentation**, or **Select Board Outline** popup menu selection shortcuts to select the whole shape. As an alternative, you can press the Shift key and select a segment - this selects the shape rather than a segment or corner.

---

2. Right-click and click the **Set Origin** popup menu item.
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.
5. To exit Set Origin mode, press the Esc key.

Placement and Length Minimization

Before you begin placement, set the topology types you want to use for nets, net classes, or the Default level of the rules hierarchy. *Topology* is the pattern of the trace and the order in which to connect pins in a net. The Length Minimization tool reorders pin connections in a net to support the topology you set.

Length minimization does not change the netlist, it just finds better places to make connections required by the netlist. For example, if you specify the "minimized" topology for a net, length minimization finds the pin order that produces the shortest pin-to-pin connections.

When you move a part around the layout, length minimization happens on the fly. If the topology type is set to something other than "protected," you can see ratsnest connections linking and unlinking pins on the moved part to the nearest viable terminals on other parts. Also, as you move the part, a running measurement called New Length/Old Length appears on the message line. 100 equals 100 percent of all nets connected to the part when you picked it up. If the percentage becomes less than 100, the length is getting better. If the percentage becomes more than 100, the length is getting worse. When you have a placement you think will work, you can also run length minimization for the entire board by using the Length Minimization command.

Generally, placement that minimizes connection length should be your primary consideration. However, using minimum connection length to determine part location can result in dense areas of connections, usually in the center of the board. These connections can outnumber the routing channels available in the area. Consequently, after you minimize the connection length, study the board to see whether there are critical dense areas. Do not hesitate to make local adjustments to the component locations to spread the connections away from the dense areas.

Part swapping also depends on topology type. No ECO file is required or generated for length minimization or part swapping. On the other hand, gate swapping, pin swapping, and ECL terminator swapping, are all recorded in an ECO file for backward annotation. There may be, however, some high-speed or critical nets in your design for which you want to set to the “protected” topology type before you begin placement.

Controlling Length Minimization

You can control length minimization by using different methods, each having a distinct effect on the your design environment.

There are two levels of control for length minimization.

- The Length minimize setting in the Design category of the Options dialog box that determines if and when length minimization occurs when moving a component - during a move, or after a move which consumes less display memory.
- The more detailed controls are configured at the Default or a hierarchical rules level, in the Routing Rules dialog box. For more information, see the Routing Rules Dialog Box on page 1666topic.

Minimizing Unrouted Connection Length

Before you begin placement, consider setting the topology types you want to use. Running a length minimization 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.

Procedure

Click the **Tools > Length Minimization** menu item.

The nets are reordered to minimize the length between connection points.

Related Topics

Placement and Length Minimization

Routing Rules Dialog Box

Placement Related ECOs

An engineering change order (ECO) is a change in design that either comes to the board from the schematic or occurs in the layout and effects the schematic.

Because ordinary length minimization and part swapping do not change the netlist, you do not need to backward annotate them to the schematic. You must, however, record in an *.eco* file length minimization and placement optimization tools like gate and pin swapping and reference designator renumbering. For this reason, these commands are located on the ECO Toolbar. When you access the ECO Toolbar you are automatically prompted to make this file.

Use of 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.

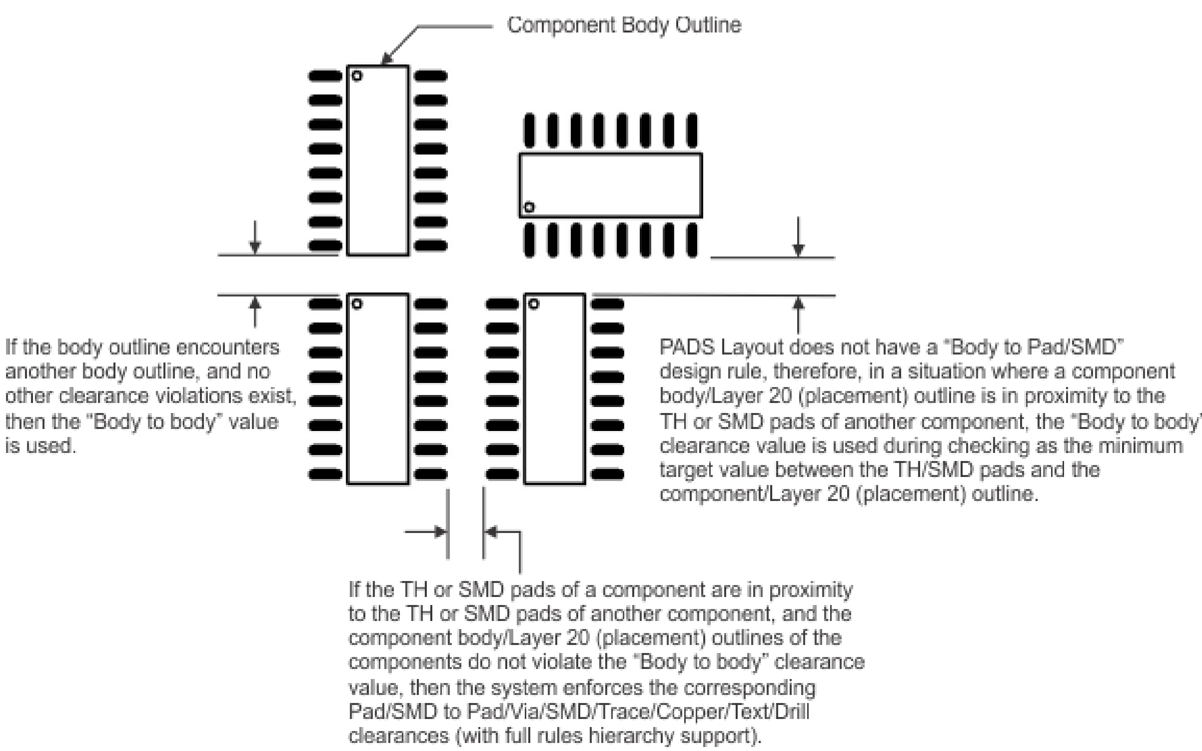
Component Spacing

SailWind Layout provides automated rules checking to ensure adequate spacing between components.

Spacing Basics

The software places components using the units of the Design grid when you enable the snap to grid setting of the Design grid.

The software prevents components from overlapping if you have set up a Body to body Clearance on page 1167 rule and the Online DRC on page 1503 (design rule checking) is in Prevent errors mode.



The preceding illustration shows examples of how some of the design clearance rules are interpreted by DRC in various design situations.

During placement operations (with Online DRC enabled), the system examines the design rules for the component being placed and compares them against other design objects including other components, the board outline and board cutouts.

All of the design rules are equally and separately enforced. If values are specified for each of the design clearance rules, then they are individually checked. During placement operations (with Online DRC enabled), the system does not allow a part to be placed if any design clearance rule is in violation of its specified value.

Typically, when a component is moved into the proximity of another component, the system will check the following design rules (if values are assigned):

- Body/Layer 20 (placement) outline against other body/Layer 20 (placement) outlines
- Body/Layer 20 (placement) outline against board outline or board cutout(s)
- Pad/SMD to Pad/Via/SMD/Trace/Copper/Text/Board/Drill clearances (with full rules hierarchy support)
- Pad and SMD Same Net clearances (with full rules hierarchy support)

The component body is considered to be the furthest extent of any 2D line object on the following layers:

- Component layer (top or bottom)
- Associated silkscreen layer for component layer (silkscreen top or bottom)
- All Layers/Layer 0 (zero).

There is only one “Body to body” clearance value that must apply to all components.

Body-to-Body Checking Rules

The “Body to body” check has specific rules associated with it during placement operations with DRC enabled:

- If the body outline encounters another body outline, and no other clearance violations exist, then the “Body to body” rule is applied.
- SailWind Layout does not have a “Body to Pad/SMD” design rule, therefore, in a situation where a component body/Layer 20 (placement) outline is in proximity to the TH or SMD pads of another component, the “Body to body” clearance value is used during checking as the minimum target clearance value between the TH/SMD pads and the component body/Layer 20 (placement) outline.
- If the TH or SMD pads of a component are in proximity to the TH or SMD pads of another component, and the component body/Layer 20 (placement) outlines of the components do not violate the “Body to body” clearance value, then the system enforces the corresponding Pad/SMD to Pad/Via/SMD/Trace/Copper/Text/Board/Drill clearances (with full rules hierarchy support).

DRC checks the “Body to body” spacing between:

- Component body outline to component body outline
- Layer 20 outline to Layer 20 outline
- Component body outline to Layer 20 outline

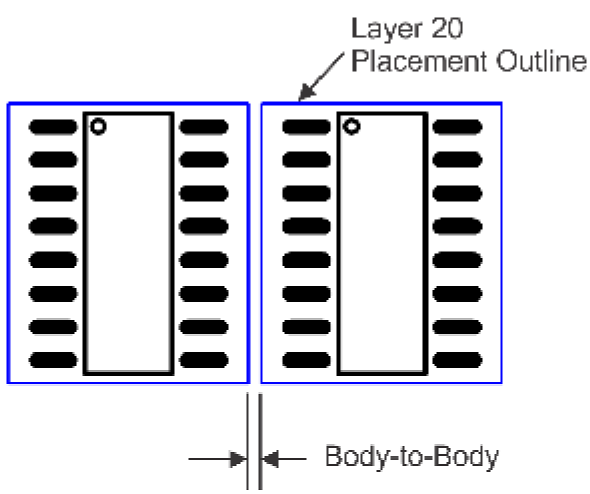
The software references the “Body to body” measurement from the edges of the 2D lines that define the body of the components.

Advanced Spacing

While a single “Body to body” clearance may suit many of your components, you may have many parts that require extra spacing, for the pick and place process, or for rework.

You can create a custom clearance outline in the decal on Layer 20. The software uses this larger placement outline, called a “component courtyard” or “nudge outline”, during the placement process to create more spacing around components. The software considers the Layer 20 (placement) outline as part of the component body. Whenever it is encountered, the “Body to body” clearance rule is invoked.

Tip
In extended layers mode on page 315, Layer 20 becomes Layer 120.



Collision Prevention

If you set the Nudge option to “automatic” or “prompt”, when you attempt to overlap parts, the component either nudges automatically or you are prompted for the direction to move the component away from the one you are placing.

If you disable Nudge and turn off the Online DRC during placement, as a post-process, you can nudge all offending components with the **Tools** menu > **Nudge Components** command, but you have no control over the offending components and where they move.

You can create arrays of parts that automatically space rows and columns of components by values that you supply using the Create Array Dialog Box.

Spacing Verification

If you are unsure if components have been placed outside of the design rules that you have set up, you can run a clearance check (click the **Verify Design > Clearance** menu item). Be sure that all aspects of your components, including the outline color for Layer 20 are visible since the clearance check only verifies objects that are visible and within view during its checking routine.

In the Clearance Checking Setup on page 1164, select the check box for the following:

- Body to body to check for the “Body to body” clearance value between component bodies.
- Placement Outline to check for the “Body to body” clearance value between the Layer 20 outlines

Note:
If you select both the Body to body and the Placement outline checks, the system also checks for clearance violations between the component body outlines and the Placement (Layer 20) outlines.

Moving Components

You can move components as required to optimize part placement.

Procedure

1. Attach the component(s) to your pointer using one of the following methods:
   - On the Standard Toolbar, click the **Design Toolbar** button to display the Design Toolbar. On the Design Toolbar, click the **Move**, and then select a component. You can only move one component at a time.
   - With nothing selected, right-click and click the **Select Components** popup menu item, then select one or more components to move and drag the component(s) to the desired location. The Drag moves option on page 1531 must be set to Drag and attach or Drag and drop. For more information, see “Drag Moves area of the Global > General options on page 1531”.
   - With nothing selected, right-click and click the **Select Components** popup menu item, then select one or more components to move, right-click and click **Move**.
   - With nothing selected, right-click and click the **Select Components** popup menu item, then select one or more components to move and on the Design Toolbar, click the **Move** button.

   If you move a component with a pin that is a locked test point, a Warning appears. For more information, see “Troubleshooting”.

   When you move a group, an error may appear in a 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.
2. The component attaches to your pointer according to the Move preference option on page 1503.

Tip
While the component is attached to the cursor, you can right-click and click commands like **Rotate 90**, **Spin**, **Flip Side**, change the Move preference, activate Snapping to objects on page 1540, or set the On-line DRC on page 1503 to ignore clearance rules. Use Bottom View for a more realistic view when placing components on the bottom side of the board.

3. To place the component, use one of the following methods:

- Move the pointer to the new location and click to place the component.
- Use one of the S Modeless Commands to proceed to a specific coordinate. Press the Spacebar key to place the component. This substitutes a mouse click that could move the pointer off the coordinate that you specified.

Moving Items With Move by Origin

When you select several objects at once, you can use different origins for the move.

If you select Move by Origin in the Design category of the Options dialog box, the pointer snaps to the center of the group, calculated from the extents of the group, and uses it as the origin for the move.

Moving Items With Stretch Traces During Component Move

When you select a component with partial traces connected to the pins of the component, the traces stretch to remain connected to the pins as long as you have “Stretch traces during component move” selected in the Design category of the Options.

Each trace connected to a pin of the moving component connects to the new pin position. If DRC is off, straight-line segments are added. If DRC is on, traces are rerouted with smoothed patterns using AutoRoute. When “Stretch traces during component move” is not selected, unroutes are created between the tacks of the trace connections.

Moving Design Objects Radially

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.

Restrictions and Limitations

- You must set up a Polar Grid on page 352.
- In the PCB Decal Editor, you can only use Radial Move with terminals.

Procedure

1. Select single or multiple components or unions. In the PCB Decal Editor, select single or multiple terminals.

2. On the Standard Toolbar, click the **Design Toolbar** button to display the Design Toolbar.

3. On the Design Toolbar, click the **Radial Move** button. The status bar displays the current polar radius and polar angle position in addition to the angular and radial increments.

 Tip
 You can alternately move design objects radially using verb mode. First, on the Design Toolbar, click the **Radial Move** button and then select the desired design object(s) to move. This method will keep the Radial Move command active and allow you to repeat the operation again after you have finished moving you first selection(s).

4. If you did not previously set up the polar grid to match your design requirements, click the **Tools > Options** menu item, select the **Grids** category, then click **Radial Move Setup**. The Radial Move Setup dialog box appears. For more information, see "Radial Move Setup Dialog Box".

 Set up the polar grid and move options and click **OK**. SailWind Layout saves these settings to apply to any future use of Radial Move in the current session.

 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.

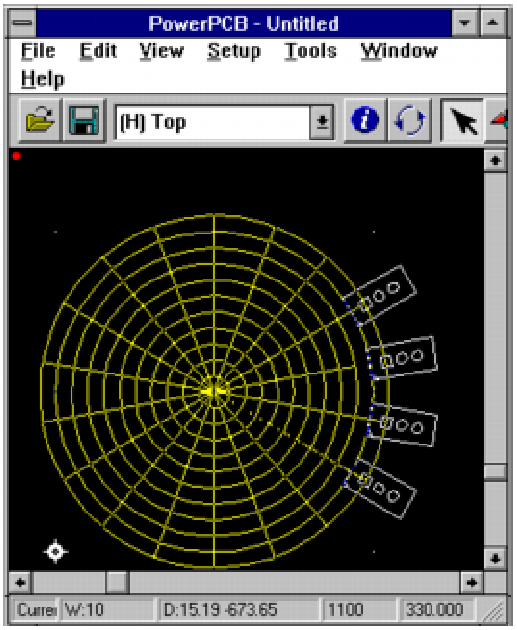
 Tip
 When you exit Radial Move, the polar grid is turned off. Use the GP modeless command on page 83 to turn the polar grid on and off independently of Radial Move.

Examples

Placing or moving parts on a radial grid utilizes a slightly different approach than that of the standard x/y grid. Examples will illustrate the behavior of the placement and move commands when using the polar grid.

The following graphic shows an example polar grid definition.

Figure 31. An Example Polar Grid Definition



The following table shows the Radial Move settings used to create the polar grid in the figure above.

Table 96. Radial Move Settings

Option	Setting
Polar Grid Origin	X=1000, Y=1000
Inner Radius	100 (mils)
Delta Radius	100 (mils)
Start Angle	30.000 (degrees)
Angle Range	360.000 (degrees)
Delta Angle	20.000 (degrees)
Sites Per Ring	18
Direction	Clockwise
Auto Rotate	On
Disperse	On
Use Discrete Radius	On
Use Discrete Angle	On
Polar Orientation	Let Me Specify, 0.000 (degrees)

The Radial Move Shortcut Menu

The Radial Move shortcut menus provides a specific selection of commands for use during radial move operations, such as rotating or flipping an object or specifying where to grab the object during movement.

Right-click to open the Radial Move shortcut menu (Table 97).

Table 97. Radial Move Shortcut Menu Items

Command	Description
Properties	Queries the component or union. For more information, see "Modifying Component Properties".
Rotate 90	Rotates the part while calculating the orientation of the part relative to the radial direction, from the grid origin to the current position of the part's move origin. Each part or union rotates using its individual origin point. For more information, see "Rotating an Object".
Flip Side	Flips each object individually around its radial direction. For more information, see "Flipping a Component".
Move by Cursor Location	Moves objects according to the relationship between the cursor location and the origin of the moved object. The default move preference is found in the Options Dialog Box, Design Category.
Move by Origin	Moves objects by the origin of the object. The default move preference is found in the Options Dialog Box, Design Category.
Move by Midpoint	Moves objects by the midpoint of an object or a group of objects. Snaps to the current pointer position. The default move preference is found in the Options Dialog Box, Design Category.
Ignore Clearance	Temporarily ignores clearances for design rules. The default On-line DRC setting is found in the Options Dialog Box, Design Category.

Sequential Component Placement

After selecting a predetermined set of components, you can use the “Move Sequential” command to sequentially place components on your pointer for placement in the design.

Placing Components Sequentially Using the Find Dialog Box

Use the Find dialog box to choose components by selecting them from a list. This method places the components on your cursor in alphanumeric order.

Procedure

1. Click the **Edit > Find** menu item.
2. In the Find Dialog Box, set Find by to Ref. Designator, Part Type, Decals, Unions or Clusters.

   Tip
   Select multiple Ref. Des. prefixes, part types or decals in order to expand the list of available components.

3. From the resulting list, select all the parts you want to move. Drag or use Shift-click for a range of components, and/or Ctrl+click for multiple components.
4. In the Action list, choose Move Sequential.

   The Move by ascending Ref Des order has no function when used with the Find dialog box. Components are always added to the pointer in ascending Ref Des order. This feature is only used by the Placing Components Sequentially in the Workspace method.
5. Click **OK**.
6. In the dialog box, “Proceed with next object? <value>” click:

   - **Yes** — to attach parts to the cursor successively, but prompt after every placement to allow you to skip over components if desired
   - **Yes to All** — if the alphanumeric order of the selection is okay and you do not intend to skip over components.
   - **No** — to skip this part, and go to the next.

   The first part is attached to the pointer for placement. Parts are attached to the pointer in ascending Ref Des order.

   With a part attached to the pointer, you can right-click to access the Rotate 90, Spin mode, Flip, move preferences, and Snap to features and commands. You can also use modeless commands.

If a part selected for Move Sequential has a pin that is a locked test point, a Warning dialog box appears informing you of changes you are making to the test point. Test points are locked in the Routing options on page 1542.

7. Click to place each component, union, or cluster.

Placing Components Sequentially in the Workspace

You select components by either using the Project Explorer or by right-clicking directly in the working area to apply the Move Sequential command. This method places the components on your cursor in the reverse order of your selection order - last component selected is attached to the cursor first.

Procedure

1. Use one or both of the following methods to select the components:

   - In the Project Explorer with Allow Selection on page 53 enabled, use Ctrl+click to select multiple components.

   - In the working area, with nothing selected, right-click and click the **Select Components**, **Select Unions/Components**, or **Select Clusters** popup menu item. Using Ctrl+click, select multiple components, unions or clusters.

   The parts are attached to the pointer in the order in which they are selected. The first component selected is the first one attached to the pointer. You can override this in a later step.

2. In the working area, right-click and click the **Move Sequential** popup menu item.

   > **Note:**
   > The Move by ascending Ref Des order can override the order in which you selected the components and place the components on your pointer in ascending Ref Des order.

3. In the dialog box, “Proceed with next object? <value>” click:

   - **Yes** — to attach parts to the cursor successively, but prompt after every placement to allow you to skip over components if desired

   - **Yes to All** — if the alphanumeric order of the selection is okay and you do not intend to skip over components.

   - **No** — to skip this part, and go to the next.

   The first part is attached to the pointer for placement. With a part attached to the pointer, you can right-click to access the Rotate 90, Spin mode, Flip, move preferences, and Snap to features and commands. You can also use modeless commands.

   You can set the Move preference to select a part by its Origin or Midpoint in the Design options on page 1503.

   Placement uses the current design grid. To place components with a Radial Move, turn on the polar grid on using the GP modeless command on page 83.

If a part selected for Move Sequential has a pin that is a locked test point, a Warning dialog box appears informing you of changes you are making to the test point. Test points are locked in the Routing options on page 1542. For more information, see “Troubleshooting”.

4. Click to place the component.

Related Topics

Creating a Component Array

Automatic Placement

Use of the Find Dialog Box During Placement

Modify the 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 on page 488 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 on page 488 before using Flip Side or Flip Group.

CAUTION:
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.

Flipping a Component

You can move a component to the opposite side of the board using its origin as the mirror point.

Procedure

1. Select one or more components to move to the opposite side.
2. Use one of the following:
    - Right-click and click the **Flip Side** popup menu item.
    - Right-click and click the **Properties** popup menu item. In the Component Properties Dialog Box, select the opposite layer in the Layer list.

Results

If DRC is on and the system encounters a clearance violation, the flip is canceled. Test points flip with the flipped component.

Tip
Use Bottom View for a more realistic view when placing components on the bottom side of the board.

Flipping a Group

You can move a group of components to the opposite side of the board by specifying the mirrored location.

Procedure

1. Select the components, right-click, then click the **Flip Group** popup menu item.
2. Click to indicate the mirror location.

Use of the /NTL Switch

Layer associations automatically switch with a component when you flip it to another side.

Typically you would want to layer associations to move with the component. For example, you move a surface mounted component to the bottom of the board. The assembly drawing, silkscreen, solder mask and paste mask layer information moves from the top side to the bottom side documentation layers.

If for some special reason, you do not want this to occur, set the /NTL switch before starting SailWind Layout to turn TrueLayer associations off. For more information, see “Software Launch Options”.

Component Arrays

You can arrange parts by creating *arrays*. A component array is a union with members placed on sites of a user-defined matrix.

Component Array Examples

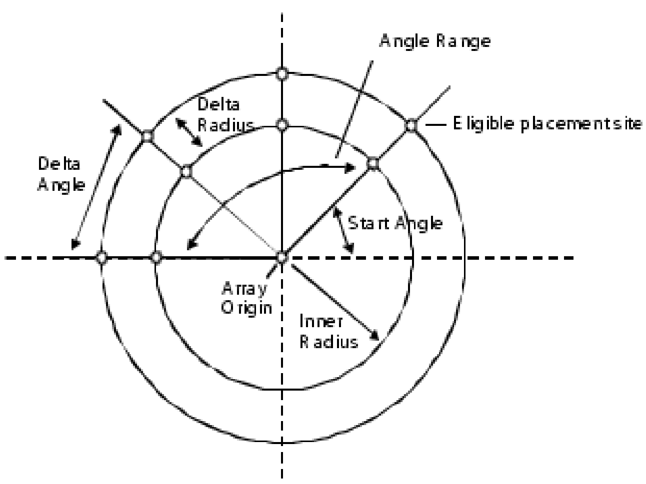
You can create either planar arrays or circular arrays.

Polar Grid or Circular Array

A number of specific values are utilized in the configuration of a polar grid or circular array.

The figure below shows the different values that you set to create a polar grid or a circular array. In a circular array, components are placed equal distances from each other on one or several rings. By turning on the polar grid and Snap to Grid, you can create drafting items with corners located at the nodes of the polar grid.

Figure 32. Polar Grid Array

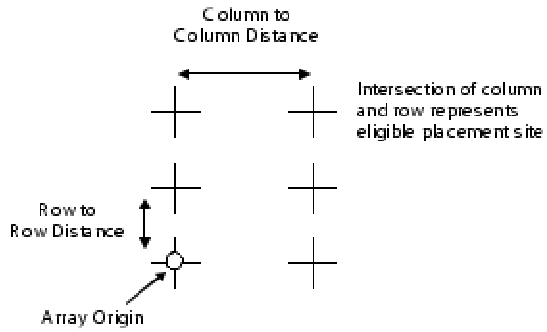


Planar Array

A planar array is a union of components that are placed on the intersections of equidistant parallel lines.

The figure below shows the different values that you set to create a planar array. The parallel lines can exist in both the X and Y directions.

Figure 33. Planar Array

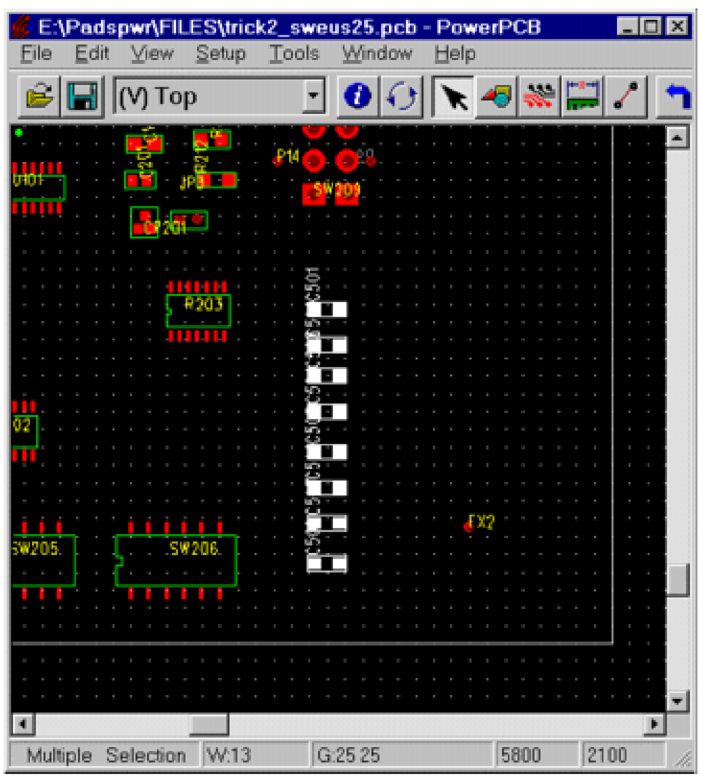


Component Array Examples

Examples will illustrate different methods that can be utilized to setup component arrays.

The figure below represents an original PCB design with eight components selected to create an array.

Figure 34. PCB Design with an Array

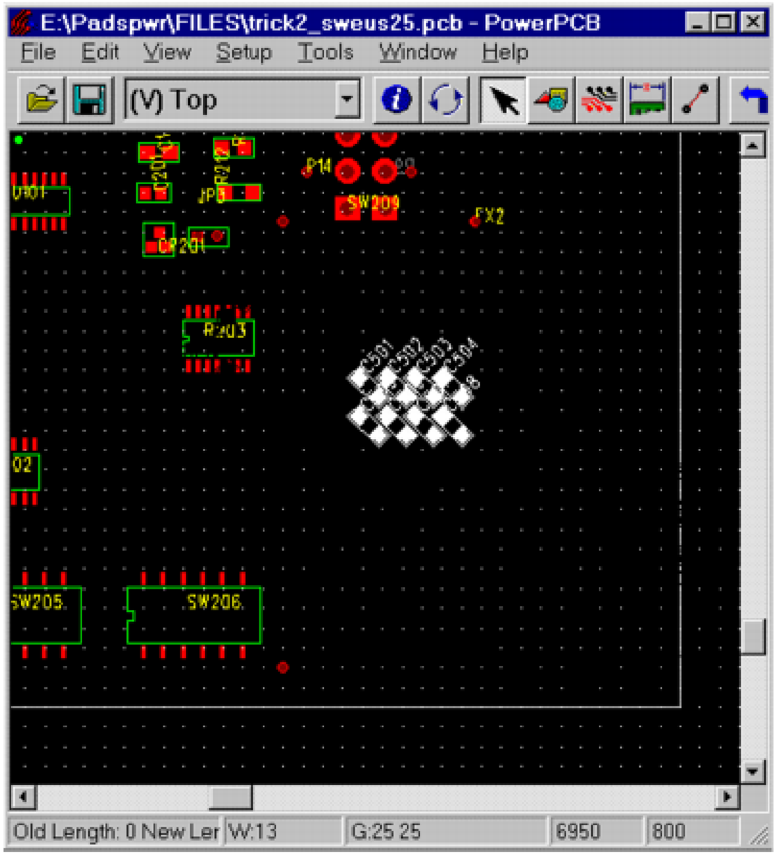


The following examples represent the same eight components using different grid and array settings.

Example 1

The figure below shows a planar array of the original eight components with a minimum body-to-body clearance of 6 mils and 45-degree rotated components:

Figure 35. An Example Planar Array (Minimum Body-to-Body Clearance of 6 mils)



The table below shows the settings used on the Planar Array tab to create the planar grid in the figure above.

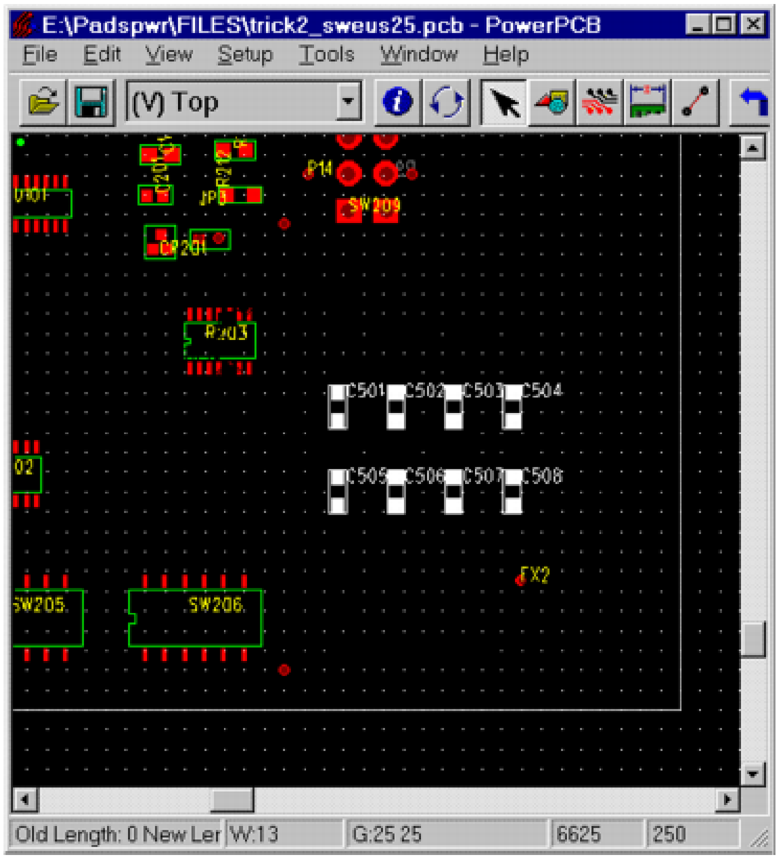
Table 98. Planar Array Settings (6 mils)

Option	Setting
Row to Row	193.7 (mils)
Column to Column	140.24 (mils)
Number of Columns	4
Place by Columns	On
Rotate	On
Orientation	45 (degrees)
Align by	Origin

Example 2

The figure below shows the example planar array of the original eight components with a minimum body-to-body clearance of 200 mils.

Figure 36. An Example Planar Array (Minimum Body-to-Body Clearance of 200 mils):



The table below shows the settings on the Planar Array tab used to create the planar grid in the figure above.

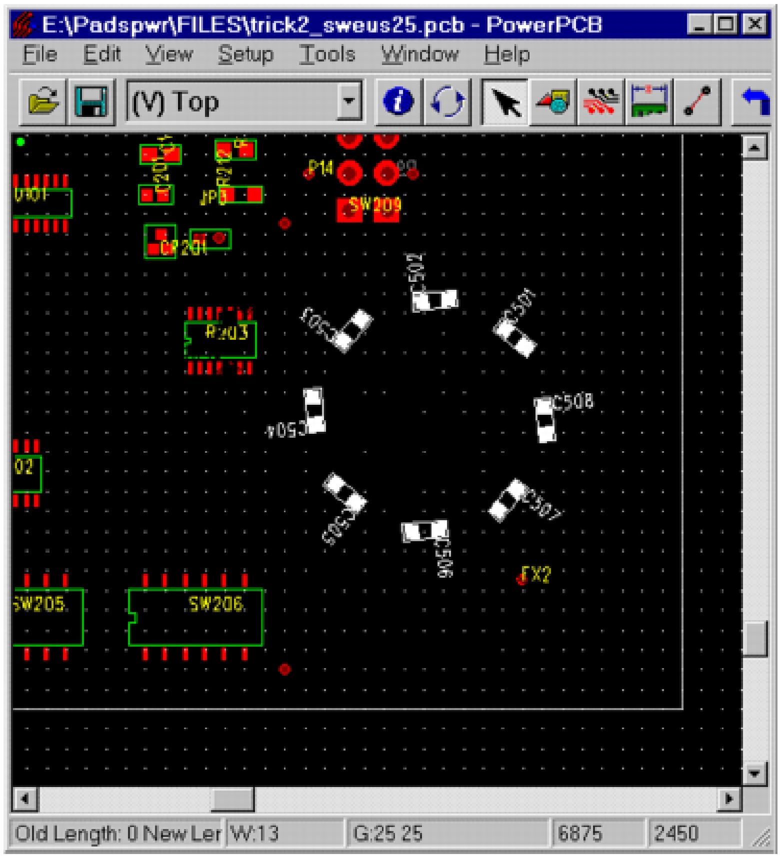
Table 99. Planar Array Settings (200 mils)

Option	Setting
Row to Row	430.15 (mils)
Column to Column	293.4 (mils)
Number of Columns	4
Place by Columns	On
Rotate	On
Orientation	0.000 (degrees)
Align by	Origin

Example 3

The figure below shows an example circular array of the original eight components with a minimum body-to-body clearance of 200 mils:

Figure 37. An Example Circular Array



The table below shows the settings on the Planar Array tab used to create the circular array in the figure above.

Table 100. Circular Array Settings

Option	Setting
Inner Radius	576.68 (mils)
Delta Radius	0 (mils)
Start Angle	5 (degrees)
Angle Range	360 (degrees), Locked
Delta Range	45.000
Sites per Ring	8
Direction	Counterclockwise

Table 100. Circular Array Settings (continued)

Option	Setting
Rotate	On
Orientation	0.000
Align by	Origin

Creating a Component Array

You can arrange parts by creating either circular or planar arrays.

Procedure

1. Select the components you want, right-click, then click the **Create Array** popup menu item.
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 dialog box appears asking you for an array or union name. For more information, see “Component Arrays”.
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.

   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. For more information, see “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.
5. Click to indicate a location for the array.

Modifying a Component Array

Using Modify Array, you can rearrange the components in an array after placing it, or you can modify a union.

Restrictions and Limitations

When modifying a component array:

- You must select Unions in the Selection Filter to select arrays and unions.
- You can only modify one array or union at a time.

Procedure

1. Select an array or union, right-click, then click the **Modify Array** popup menu item.

    - If you are modifying an array, the Modify Array dialog box appears with the tab displayed and the options used to create the array.

    - If you are creating an array from a union, a dialog box 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.

    Modify Array does not enter Move mode as Create Array does.

    Modify Array does not warn against nor prevent placement violations unless On-line DRC is in Prevent errors or Warn errors mode.

    If you modify the location of an array that contains a component with a pin that is a locked test point, a warning appears. For more information, see "Troubleshooting."

3. Click **OK**. The array is modified or created from the union.

4. If On-line DRC is in Prevent errors or Warn errors mode, click to indicate a new position for the modified array.

Aligning Objects

Alignment snaps selected parts to an imaginary line coming from a side or center of a guide component. After placing objects 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 the **Align** popup menu item.

    Restriction:
    You cannot align elements in a physical design reuse. Align becomes unavailable when you select physical design reuse elements.

3. In the Align dialog box on page 1080, click the alignment scheme you want to use. The objects are automatically aligned according to the position of the last one selected.

    Automatic alignment does not guarantee minimum spacing if on-line DRC is not enabled, but you can use Nudge to accomplish this interactively. If On-line 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.

Rotating an Object

You can rotate objects in 90 degree increments.

Procedure

1. Select the object to rotate. You can select more than one object at a time.
2. Right-click and click the **Rotate 90** popup menu item. The selected object rotates 90 degrees counterclockwise.

   If you rotate a component with a pin that is a locked test point, a warning appears. For more information, see "Troubleshooting."

   When you rotate a group, an error may appear in one of the Trace Copy dialog boxes. For more information, see "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.

   Tip
   To use Rotate 90 in Verb Mode, click the **Rotate** button on the Design Toolbar and then select the part to rotate.

3. Repeat step 2 to continue rotating the objects.

Spinning an Object

You can spin an object on its axis during part placement.

Procedure

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.

   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. For more information, see "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.

3. Click to indicate a new angle.
4. To use Spin in Verb Mode, click the **Spin** button on the Design Toolbar, and then select the part to spin. click to indicate a new angle.

Swapping Parts

Sometimes simply switching neighboring parts' positions can improve connection length. You can swap the position of two parts anywhere on the board using the “Swap Part” command. When Online DRC is selected, the swap is examined for placement violations.

Procedure

1. To begin, use one of the following methods:

   - **For a single swap** — Select a part > Design Toolbar button > **Swap Part** button.

   - **For multiple swaps** — **Design Toolbar** button > **Swap Part** button > select the first part to switch.

2. Select the second part to switch.

3. You are prompted to allow the swap and at the same time, the X + Y connection lengths before and after the swap appear in the status bar so you can see whether the lengths improve.

4. Click **Yes** to complete the swap. Parts are swapped origin-to-origin.

   If you chose the multiple swap method and you are still in Swap Part mode, you can continue to choose parts to swap.

Interactive Placement Tools

You can place parts to exact tolerances using the alignment and nudge tools.

When placing parts interactively, note the following effects of alignment and nudge:

- Alignment snaps selected parts along an imaginary line coming from a side or center of a guide component.
- Nudge eliminates overlaps and clearance violations by pushing parts aside to make room for a crowded component.

Tip
For nudging, all component elements are considered glued. Align and Nudge are unavailable when you select physical design reuse elements.

Nudging Parts

Nudging Parts

When parts are close enough to violate either pad-to-pad or body-to-body clearance rules, they are considered overlapping.

- Nudge and Design Rule Checking (DRC)
- Nudge Overlapping Parts
- Nudging All Components
- Nudging Single Parts

Nudge and Design Rule Checking (DRC)

When DRC is set to Prevent, SailWind Layout actively prevents overlapping by canceling the completion of an illegal move and returning the selected component to Move mode.

This automatic checking can be further refined using Nudge. Nudge is a shove function for parts; it resolves overlapping by moving surrounding parts away from the part you are trying to place. Nudge has three operational modes that you set on the **Design** category of the Options dialog box. As shown in the table below, Nudge operation is dependent on the current DRC setting:

Table 101. DRC Setting and Nudge Modes

DRC Setting	Prevent	Warn	Off or Ign Clrn
Auto	Overlapping parts are automatically moved and unhighlighted. If relocation is not possible due to available space, a message appears.		
Warn	The Nudge dialog box appears for user interaction.		
Off	No Nudge operations occur. The selected part remains attached to the cursor.	The Clearance Violation dialog box appears. The choices are: **Ignore** — Places part without adjusting overlapping parts. **Explain** — Lists the intersecting parts. **Cancel** — Cancels the command; the part remains attached to the cursor.	The part is placed without moving overlapping parts.

When you use Nudge for placement, SailWind Layout approximates the component size. For example, if you placed an L-shaped component, SailWind Layout approximates that part area as a square, not as an L. Nudge then uses the greater of the pad-to-pad or body-to-body clearances defined in Design Rules setup to determine the distance that must exist between two components.

In many cases, several parts are adjusted to accommodate the moved part. Nudge does not move glued parts or parts outside the board outline. Nudge treats test points as glued objects. Nudge does not move parts inside the board outline outside the outline. When an overlap exists, the Nudge Parts and Unions dialog box appears.

Nudge does adhere to the current grid setting, so be sure you set the grid fine enough for the command place within the restrictions of your routing strategy.

Nudge Overlapping Parts

Use Nudge to move overlapping parts. The manner in which Nudge operates is dependent on the current DRC setting.

For more information, see Nudging Parts.

Nudging All Components

You can move all overlapping parts at once using the Nudge Components command. Nudge eliminates overlaps and clearance violations by pushing parts aside to make room for a crowded component.

Procedure

Click the **Tools > Nudge Components** menu command.

All components are automatically nudged according to the design rules.

Nudging Single Parts

Nudge eliminates overlaps and clearance violations by pushing parts aside to make room for a crowded component. You can use Nudge to move an individual part that overlaps another part.

Procedure

1. Select a part.
2. Right-click and click the **Nudge** popup menu item. For more information, see

   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 cannot nudge elements in a physical design reuse. All component elements are considered glued. Nudge is unavailable when you select physical design reuse elements.
3. In the Nudge Parts and Unions Dialog Box on page 1495, click the nudge direction and the direction to move overlapping parts.
4. Click **Run** to perform the nudge.
5. 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.

Changing the Part Outline Width

You use this procedure to change the width of the 2D lines that indicate the part outline on Silkscreen Top/Bottom layers.

Restrictions and Limitations

- This procedure does not apply to placement (nudge) outlines created on Layer 20.

Procedure

1. Select a component, right-click, and then click the **Properties** popup menu item.
2. Type a width (between 0 and 250) in the Part Outline Width box.
3. Click **OK**.

   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.

   CAUTION:
   In SailWind Layout, the "Body to body" clearance checks are performed between the edges of the lines used to define the component body outlines. Increasing the widths of these lines may impact the spacing between existing components in the design and require you to perform a Verify Design Clearance check to assure that all of the "Body to body" clearances are maintained.

4. To update all parts that have the same decals as the selected parts with the new part outline width, click **Continue**.

Modifying Component Properties

You can edit or change component properties to reflect changes in the design.

Procedure

1. Select a component.
2. Right-click and click the **Properties** popup menu item. The Component Properties Dialog Box appears.

   Restriction:
   Several of the options on the dialog box are unavailable when the component is part of a physical design reuse.

3. Make any necessary changes and click **OK**.

Creating a Label

You can create labels using the **Labels** tab in the Component Properties dialog box.

Procedure

1. Select a component.
2. Right-click and click the **Properties** popup menu item. The Component Properties Dialog Box appears.
3. On the **Labels** tab, in the Label list, click <new> and click the **Label** button. For more information, see Adding a New Part Label.
4. 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.

   Restriction:
   The only attribute available for jumpers is the reference designator.

5. 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 on page 1853. This box is also unavailable if the attribute is ECO-registered on page 1824 and SailWind Layout is not in ECO mode.
6. Set the visibility status and the placement information for the label.
7. Set the justification, right-readability, height, and width of the label.
8. Click the layer on which to place the label, and then click **OK** to create the label.

   Tip
   If you do not set visibility information (such as in steps 4, 5, and 6), default positions are used. For more information, see Label Defaults.

   Note:
   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 designators, part types, and attributes.

Procedure

1. Select a component.
2. Right-click and click the **Properties** popup menu item. The Component Properties Dialog Box appears.
3. On the **Labels** tab, click a label in the list and click the **Label** button in this tab.

4. A label is selected instead of the component, and the corresponding Part Label Properties dialog box on page 590 appears where you can modify the label.

Restriction:
When modifying the Properties of a jumper name, Reference Designator is the only available label.

Related Topics

Component Properties Dialog Box

Cluster and Union Placement

Use cluster placement features to create associations or groupings of connected parts.

Two object types are used:

- **Unions** — are user-created part associations that have a strict relationship with each other. An example of this is associating a filter capacitor to reside on top of an IC. When you move or place a selected union, the physical relationship between the parts, or union members, remains unchanged. Unions are not created using the Cluster Placement routines, but they are considered during cluster creation and automatic placement operations. For information, see the Unions topic.

- **Clusters** — are collections of individual parts, unions, and other clusters, based on connectivity. A series of ICs and associated discrete components that make up a memory array could make up a cluster. Clusters differ from unions because the parts that belong to the cluster, or cluster members, are rearranged within the cluster to improve placement.

  Clusters are useful in very large designs or in designs where areas of the board are separated into different functions. You can create or modify clusters manually or automatically. You can also automatically place individual parts using this feature. For more information, see the Placing Parts Automatically and Moving Clusters Interactively topics.

  Any operation that creates, displays, or modifies clusters (except the Cluster Manager) automatically places you in Cluster View mode. For more information, see the Cluster View Mode topic.

Cluster Display Settings

The circle that represents a cluster is assigned in the Display Colors Setup dialog box as a 2D Lines property on the Top layer. The text representing the cluster name is assigned as a Text property on layer 20.

The connection lines originating from the center of the cluster (the same as in the normal view) are assigned with the Connection setting in the Display Colors Setup dialog box.

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.

Creating a New Union

You can create a union consisting of two or more parts.

Procedure

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 the **Create Union** popup menu item. 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.

Procedure

1. Select a union, right-click, and then click **Create Like Unions**.

   A dialog box appears to “Confirm Creating Like Unions”.
2. Click **Yes** to continue.
3. In the dialog box that appears prompting “OK to Disperse new unions?” do one of the following:

   - Click **Yes** to disperse the newly created unions around the board outline.
   - Click **No** to disperse parts, and 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.

Selection of Unions

You can select unions either manually or automatically.

Select a Union From a Union Member

Select the first item in a union and then find the related objects.

Procedure

1. Select a member of a union.
2. Right-click and click the **Select Union** popup menu item.

Automatically Select a Union When One of Its Members is Selected

Automatically select all members of a union when you select the first object.

Procedure

1. Click **Edit > Filter**.
2. Click both the Parts and Unions options. For more information, see Refined Object Selection

Adding a Part to a Union

If desired, you can add parts to an existing union.

Procedure

1. Set the filter to select the entire union when you select a member. For more information, see Refined Object Selection
2. Use multiple selection, Ctrl+click, to select the parts to add.
3. Right-click and click the **Create Union** popup menu item. 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 that prevents you from modifying unions sets automatically when you start SailWind Layout. You can alter this setting.

Procedure

1. Select a member of any union.
2. Right-click and click the **Modify Union Member** popup menu item. This option remains active for the entire SailWind Layout session. Click the option again to return it to the default state.

Delete a Union or Union Members

You can remove a part from the union or remove the entire union.

You can choose the method that meets your design requirement:

- Remove a Part from a Union — Use this method to remove an individual part from a union.
- Remove an Entire Union — Use this method to remove the entire union from the design.

Remove a Part from a Union

If desired, you can break a part out from a union.

Procedure

1. Select the part to remove.
2. Right-click and click the **Break From Union** popup menu item.

Remove an Entire Union

If desired, you can break apart a union.

Procedure

1. Select the union.
2. Right-click and click the **Break** popup menu item.

> **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.

Creating New Clusters

You can manually create new clusters.

Procedure

1. Select the parts, unions, and other clusters to include in the cluster.
2. Right-click and click the **Create Cluster** popup menu item. 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.

Procedure

1. Select the clusters to change.
2. Right-click and click the following popup menu items:

- **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 on page 1177 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 on page 1177 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 on page 1171 to modify existing clusters using a dialog box.

Cluster View Mode

SailWind Layout automatically switches to Cluster View Mode any time you create or modify clusters, unless you are using Cluster Manager. Click **View > Clusters** 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

You can display parts belonging to a cluster while in Cluster View Mode.

Procedure

1. Click **Edit > Filter**.
2. Click **Clusters**.
3. Select the cluster.
4. Right-click and click the **Show Contents** popup menu item. 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

If necessary, you can reposition a cluster.

Procedure

1. Select a cluster, right-click, and then click the **Move** popup menu item.
2. Move the cluster to a new location and click. The cluster moves to the new location and the members of the cluster collapse on page 1816. You can set SailWind Layout so that clusters do not collapse after moving. For more information see Collapse Clusters.

> **Tip**
> Autoplacement features ignore components that are part of a physical design reuse.

Delete a Cluster

You can remove an individual cluster or all clusters.

Remove a Single Cluster

You can remove a single cluster in a design.

Procedure

1. Select the cluster. You can make multiple selections.
2. Right-click and click the **Break** popup menu item.

Remove All Clusters

If desired, you can remove all clusters in the design at once.

Procedure

1. Select any cluster.
2. Right-click and click the **Break All Clusters** popup menu item.

Collapse Clusters

The "Collapse" command relocates the cluster members 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 the **View > Clusters** menu item.

Collapsing All Clusters

You can collapse all clusters in the design at one time.

Procedure

1. With nothing selected, right-click and click the **Select Clusters** popup menu item.
2. Right-click and click the **Select All** popup menu item.
3. Right-click and click the **Collapse Members** popup menu item.

Collapsing Cluster Members

Be default, clusters in SailWind Layout collapse after moving them. However, you can change this setting.

Procedure

1. While moving a cluster, right-click and click the **Collapse Cluster Members** popup menu item.
2. The cluster is collapsed.

Tip
Collapse Cluster Members is set to on each time you start SailWind Layout.

Related Topics

Cluster and Union Placement

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

Automatic Placement

Use the Cluster Placement dialog box to build new clusters, place clusters within the board outline, and place parts within the board outline.

Preparing for Automatic Placement
Placing Parts Automatically

Preparing for Automatic Placement

After you create a board outline and import a netlist, SailWind Layout locates all parts at the board origin. You can then prepare your design for automatic placement and begin the automatic placement process.

Procedure

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 the **Tools > Disperse Components** menu item 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. On the Drafting Toolbar, click the **Component Keepout** button to create areas where parts should not be placed during automatic placement.
7. Use the **Rotate** and **Flip** buttons to arrange parts that should be placed at an angle or reside on the bottom side of the design.

Placing Parts Automatically

As a time-saving option, you can choose to place parts on your PCB design automatically.

Procedure

1. Click the **Tools > Cluster Placement** menu item.
2. Click the **Place Parts** button. To modify the automatic placement options, click the **Setup** button (below the Place Parts 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 "Automatic Placement".

Cluster Management

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.

Adding a Component to a Cluster

If desired, you can add additional components to an existing cluster.

Procedure

1. Click the **Tools > Cluster Manager** menu item.
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** button.

Making Members of One Cluster Part of Another

If desired, you can merge members of one cluster into another.

Procedure

1. Click the **Tools > Cluster Manager** menu item.
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** button.

Results

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 22
SailWind 3D

As an alternative to standard two dimensional board layout, SailWind Layout provides a three dimensional, graphical representation of your project in the SailWind 3D window. In addition to allowing interactive component placement, the SailWind 3D window also renders your board layout in a realistic three dimensional view, thereby letting you preview your design, perform physical measurements from one component to the next, and check clearances with imported mechanical designs, such as mounting brackets.

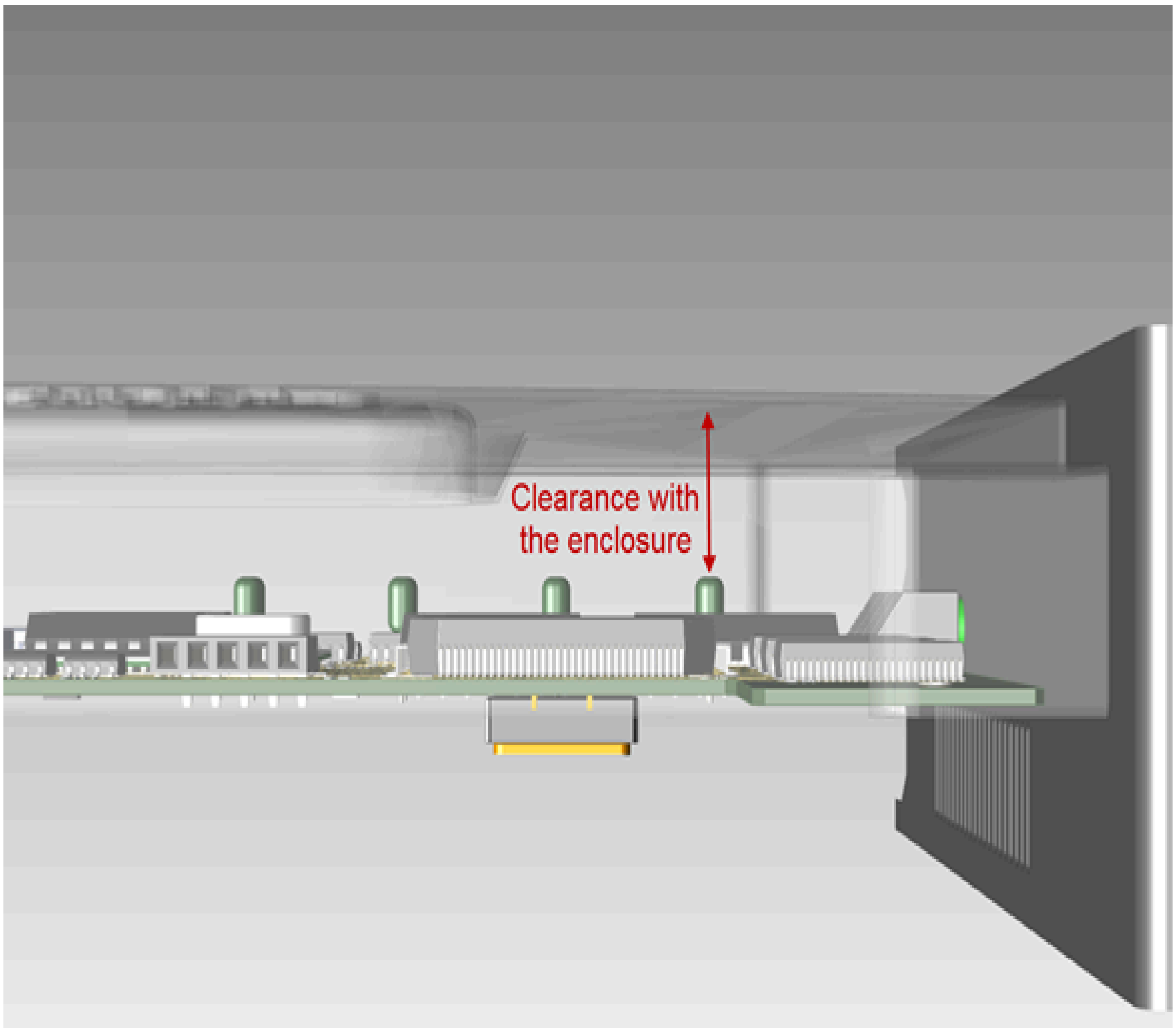
SailWind 3D Overview

The SailWind 3D window provides a three-dimensional rendering of your PCB layout, including electronic components and mechanical assemblies (such as mounting brackets or connectors). This three-dimensional representation allows you to examine the PCB construction, move and align components, and check clearances between components on your PCB and adjacent assemblies (such as EMI shields, heat sinks, and covers). The SailWind 3D window also provides tools for aiding in the design collaboration process with mechanical (MCAD) designers.

Part Placement in SailWind 3D

One of the primary purposes of the SailWind 3D window is to allow you to view a three-dimensional representation of your design to check clearances between components and surrounding assemblies or mechanical parts in the X, Y, and Z direction.

Figure 38. Checking PCB Component Clearance with an Enclosure



However, if desired, you can synchronize the SailWind 3D window with the two-dimensional layout design view, meaning selecting and moving a part in one view updates the part's position in the other view. This synchronization feature becomes helpful if you choose to place parts in your PCB design using the SailWind 3D window instead of the standard two-dimensional layout design view.

Using the SailWind 3D window, you can assign or "map" 3D models to your parts, thereby giving them an accurate three-dimensional representation (see Assigning a 3D Model to a Component). You then have the option of saving the 3D model mapping for re-use with other designs by saving it to the Reuse folder location for a netlist project — see 3D Model Mapping. Alternatively, you can keep it mapped to part in the PCB design only.

For 3D model alignment, the SailWind 3D window appears adjacent to the Decal Editor along with the Align 3D Models dialog box (see Align 3D Models Dialog Box). The Align 3D Models dialog box provides you with the ability to align the pads of a 3D model with the pads of a part decal, as displayed in the SailWind 3D window.

Clearance Checking Features

For checking clearances, the SailWind 3D window provides options:

- Manually check distances between a component and a point on the PCB or an assembly using the "Measure Distance" and "Measure Minimum Distance" features. This option is especially useful when aligning a 3D model with decal pads.
- Perform automatic design rule checking across the entire design through the use of 3D clearances Design Rule Checks (DRC).

Note:
Use the 3D Clearances dialog in the SailWind 3D window for defining your 3D clearances.

MCAD Collaboration

To aid in the collaboration effort with mechanical designers, the SailWind 3D window features the MCAD Collaborator, a tool for tracking and annotating design change proposals. With the MCAD Collaborator, you can establish an initial or "baseline" design with the mechanical designers. When you make a change from the baseline layout design (for instance, you move a part), the MCAD Collaborator lists the affected part as a change proposal and allows you to exchange the proposal with the mechanical design team as an *.idx* file. Upon acceptance from the mechanical designers, the proposal becomes the new baseline.

Related Topics

Assigning a 3D Model to a Component

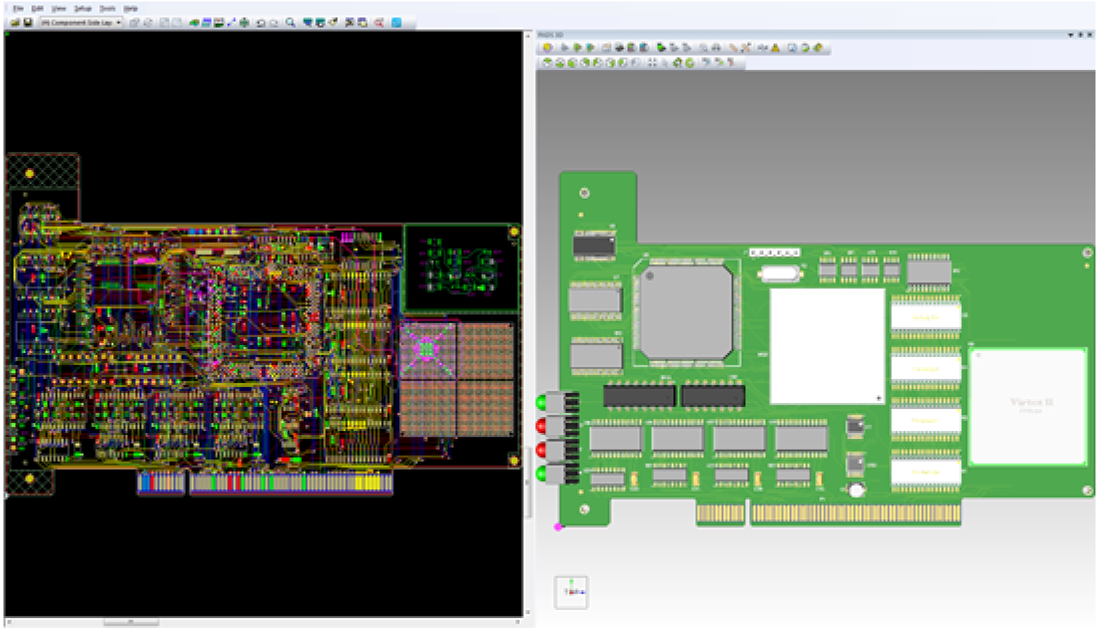
SailWind 3D Object Manipulation

You can rotate the viewing angle of your PCB design in the SailWind 3D window along three axes. You can also zoom in or out to change the magnification of the rendered PCB design. Use the viewing controls in tandem with the SailWind 3D window display options to view various features of your PCB design.

Rotating the View

Choose the **View > SailWind 3D** menu item to open the SailWind 3D window. The window displays next to the SailWind Layout design window.

Figure 39. The SailWind 3D Window



You can resize the SailWind 3D window as necessary by dragging its border with the mouse.

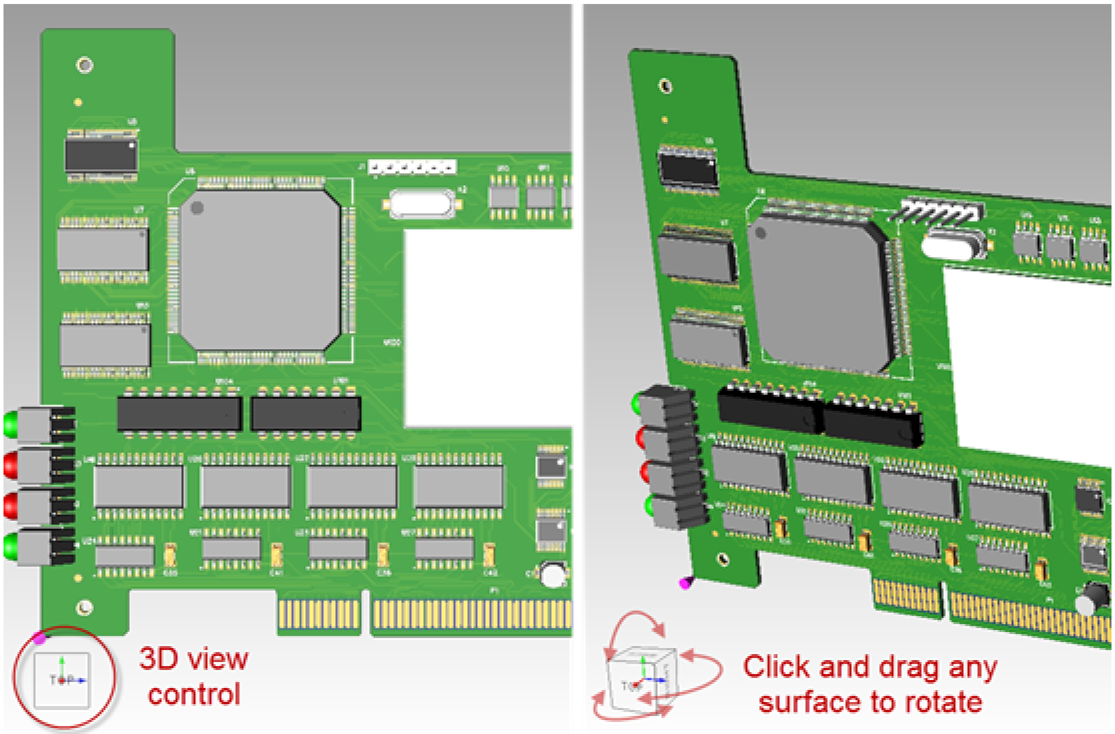
The SailWind 3D window features a 3D view “cube” control for rotating, zooming, and panning the three-dimensional PCB. Rotate the PCB in the SailWind 3D window to a fixed or orthogonal view (such as front view, top view, left-hand view, and so forth) by either clicking a corresponding surface of the cube, or by clicking a rotational arrow to rotate the PCB view in 90° increments.

Note:
Clicking a cube surface also changes the view to the default magnification.

Click the “home” icon at the corner of the cube for an isometric view of the PCB.

Alternatively, you can rotate the PCB dynamically by clicking any surface of the cube control and rotating it in the desired direction. The PCB moves as you rotate the cube.

Figure 40. Rotating the PCB with the 3D View Control



Zooming and Panning

Use the 3D view control to zoom in on the PCB or pan across to view certain details as described in the following table:

Table 102. Manipulations with the 3D View Control

If you want to...	Do the following...
Zoom in or out of the current view.	Press the Ctrl key and drag the 3D view controller up or down.
Pan across the current view.	Press the Shift key and drag the 3D view controller in any direction.

Using the Middle Mouse Button

Alternatively, if you want to rotate the board along a single axis or manipulate the 3D view from anywhere on the display using the middle mouse button, do so as follows:

Table 103. Manipulations with the Middle Mouse Button

If you want to...	Do the following...
Zoom in or out of the current view.	Press the Ctrl key and rotate the middle mouse wheel up or down.

Table 103. Manipulations with the Middle Mouse Button (continued)

If you want to...	Do the following...
Rotate the current view around the X or Y axis.	Press the Shift key and press and hold the middle mouse button as you drag the cursor up and down for X axis rotation or sideways for Y axis rotation.
Rotate the current view around the Z axis.	Press the Ctrl key and press and hold the middle mouse button as you drag the cursor in a circular motion.
Pan across the current view.	Press the Alt key and press and hold the middle mouse button as you drag the cursor up and down or sideways.

The SailWind 3D window supports most zooming functions available in the 2D workspace. For more information, see Zooming Overview.

Tip
Rotation with this method is limited to one axis at a time. To rotate the PCB in a dynamic direction, use the 3D view control instead.

Preset Views

The SailWind 3D window provides preset views (View Top, View Bottom, View Front, View Back, View Left, and View Right) for rapid access to standard views. Access these preset views by clicking the  associated buttons in the 3D window toolbar.

While these preset views do not adjust the image magnification, they are useful for returning the view to an established angle, such as the top or front view.

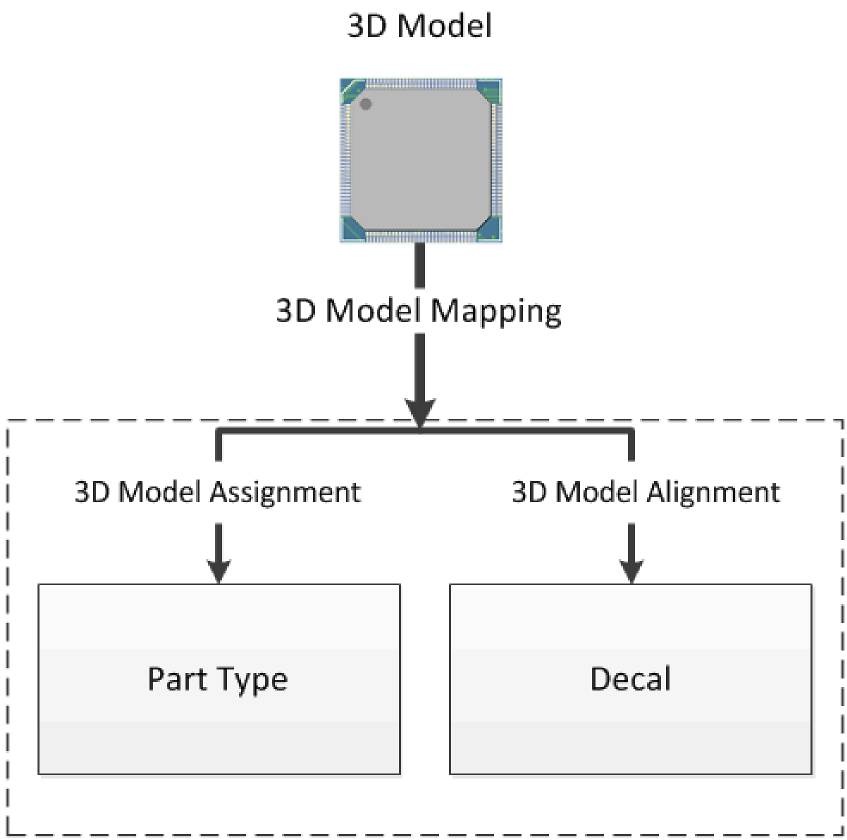
3D Model Mapping

The SailWind 3D window allows you to assign a 3D model to a PCB component and align it with the decal pads. This alignment and assignment information is known as model *mapping*.

Mapping 3D models begins with assigning a 3D model to a component in the SailWind 3D window—see Assigning a 3D Model to a Component. After assigning and aligning the 3D model, you have the option of saving the mapping information to the Reuses folder location (for a netlist project).

When you map a 3D model to a component, and you choose to save the model mapping to the *Reuse* folder location (using the Update Library Dialog Box), SailWind Layout associates the 3D model assignment with the part type. It also associates the 3D model alignment information to the decal.

Figure 41. SailWind 3D Model Mapping



The mapping function allows you to re-use the 3D model with any other design that has the same part type and decal combination. For more information, see Reusing 3D Models with Other Components in a Design.

Related Topics

Saving 3D Models for Re-use in Other Designs

Update Library Dialog Box

Assigning a 3D Model to a Component

Assign or "map" 3D models to component decals and align them so the SailWind 3D window accurately represents them in the rendered image. Import a new 3D model from an external source and map it to a decal then align the 3D model so that it is placed correctly on the PCB.

You can map 3D models in one of two ways: directly within the 3D window or within the Decal Editor.

Within the 3D window:

- You can perform mapping more quickly than editing in the Decal Editor and reloading the design in the 3D window after returning
- Changes do not modify the decal

- Changes apply to all components with the same part type-decal pairing
- You can make alignment changes in delta offset values only

Within the Decal Editor:

- You can apply a model mapping to multiple Part Types
- Changes modify the decal, possibly making it appear in the list in the Update from Library Dialog Box
- You can make alignment changes in absolute or delta offset values
- When exiting, you can choose to apply mapping to the selected component or to all components using the same part type-decal pairing

Procedure

1. Select a component in the SailWind 3D window.

   If you select an ECO-registered component, ensure the it meets “certain requirements” on page 531 so that it displays correctly in the SailWind 3D window.

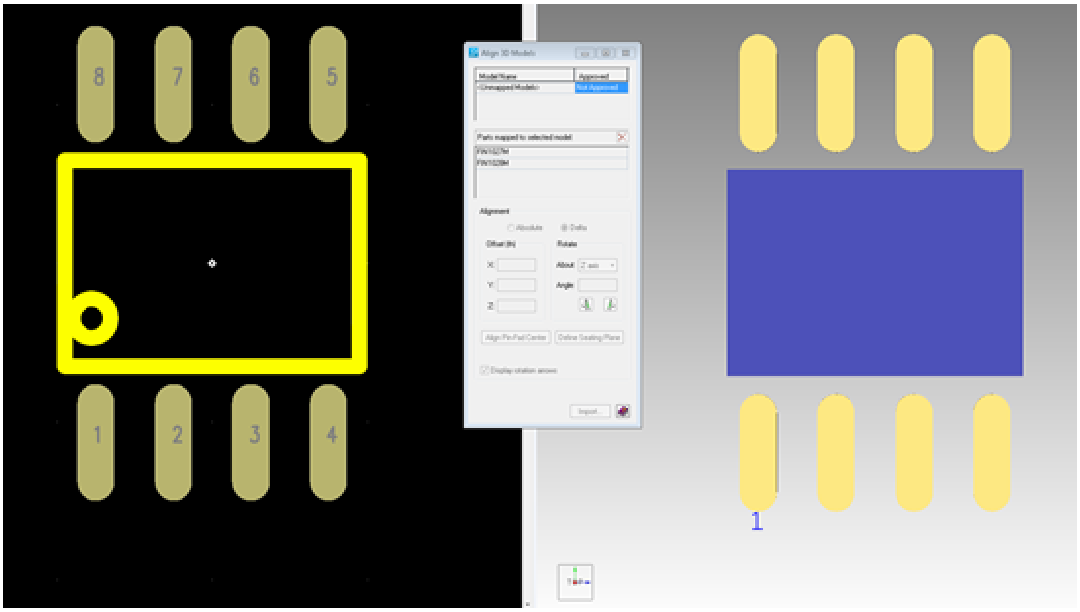
2. Click the **Import Part Model** button on the 3D window toolbar. .

   Alternatively, if you want to use the Decal Editor, perform the following steps:

   a. Right-click on a component and click the **Edit decal** popup menu item.

   > **Restriction:**
   > If your software license does not enable this feature, click in the standard two-dimensional layout design view to access the **Edit decal** command.

   The Decal Editor opens and displays the default two-dimensional decal of the component and a corresponding representation in the SailWind 3D window adjacent to it. The Align 3D Models Dialog Box also opens.



b. In the Align 3D Models dialog box, select the part name (or names, if you want to map the same 3D model to more than one part) in the “Parts mapped to selected model” list and click **Import**.

3. In the Import Part Model dialog box, select the correct model definition file you want to import and click **Import**.

   If you are assigning a 3D model through the Decal Editor, the new model appears in the Model Name list of the Align 3D Models dialog box as the model mapped to the decal.

4. Rotate or reposition the imported 3D model as necessary so it aligns correctly with the decal pads as follows:

   a. Rotate the model so it is oriented correctly over the pads (its pins do not need to be aligned yet —you will align them in the next sub-step).

   b. Click the **Align Pin-Pad Center** button. The decal pads hide temporarily to simplify alignment.

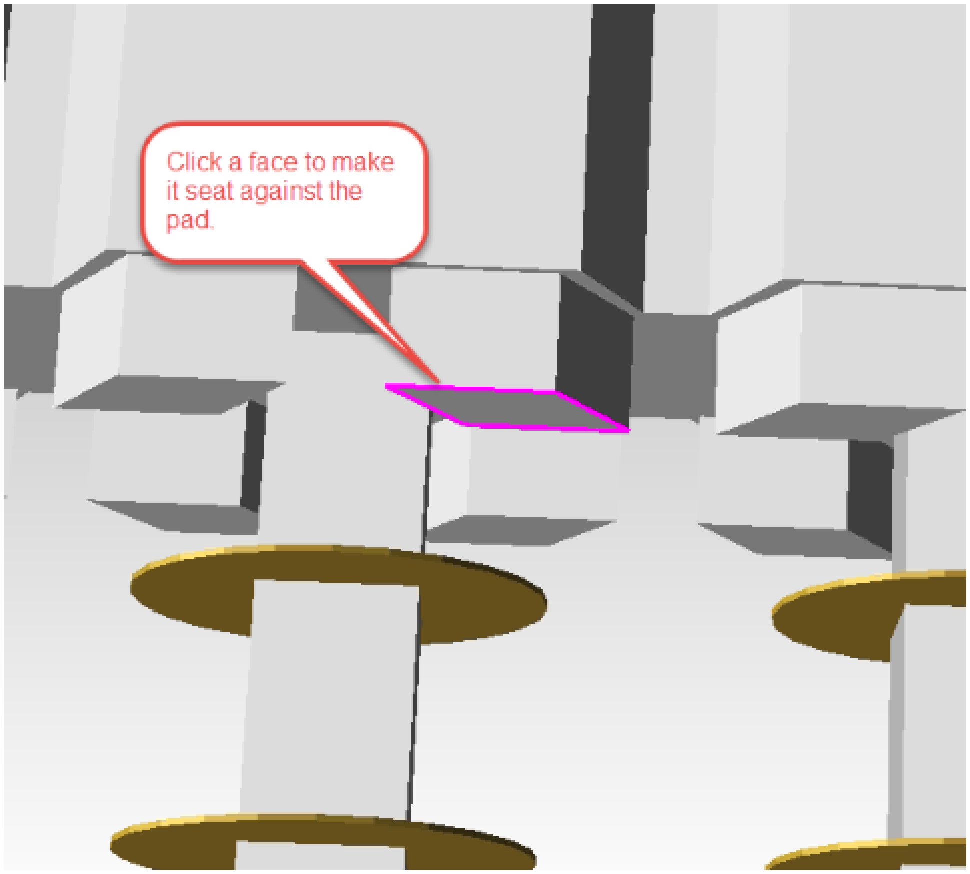
   c. Select a face on a model pin to use for alignment, adjusting the 3D view as necessary for access. For example, if you are aligning a SO-14 package, select the bottom surface of pin 1 that you expect to make contact with the pad.

      The decal pads return to view after you select an alignment face.

   d. Click the corresponding decal pad to which you want to align the center of the selected pin face. The model moves so the center of the pin face is aligned with the center of the pad.

   e. In some cases—such as when you are mapping a through-hole component—you may also need to define which model face you want to make “seat” against the pad.

Figure 42. Selecting a Model Face



i. Click **Define Seating Plane**.

ii. Rotate the 3D view as required to access the model face you want to seat against the pad.

iii. Click the face to select it. The model adjusts to seat the face against the pad.

Note:
If you close the 3D Window before exiting the Decal Editor, the Decal Editor prompts you to save your data. Click **Yes** at the prompt to save the 3D model mapping and apply the 3D model to all parts in the design that have the decal. Otherwise, click **No** to abandon your changes.

---

5. Click **OK** when finished aligning and centering the model with the decal pads.

   In the 3D window, you can save changes to the library using the Update Library Dialog Box. You can also use the Manage Mappings Dialog Box to apply mappings to other components in the design.

   After updating the library with your changes, you can later "change the 3D model" on page 532 associated with the component but you can no longer delete all model associations.

6. (Decal Editor only) Choose the **File > Exit Decal Editor** menu item when you are finished aligning the imported model.

   In the Apply Decal Changes prompt, click **All** to apply the 3D model to all parts using the decal in the design or **Selected** to apply the model to the selected component only.

   Tip
   Click **All** if you change only the model alignment.

Results

The SailWind 3D window displays changes in the three-dimensional representation of your design and its components.

Related Topics

SailWind 3D Object Manipulation

Align 3D Models Dialog Box

Saving 3D Models for Re-use in Other Designs

Reusing 3D Models with Other Components in a Design

As a time-saving measure, you can reuse or reassign 3D models to components on the rendered PCB design that share a common physical footprint. For example, you can assign the 3D model from a surface mount resistor to a surface mount capacitor if the two components appear physically identical.

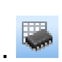
Restrictions and Limitations

You cannot use the Manage Mappings dialog box to reassign a 3D model to a part if the part is already saved in the library with a 3D model.

Prerequisites

You have already mapped the 3D model you want to reuse to at least one component.

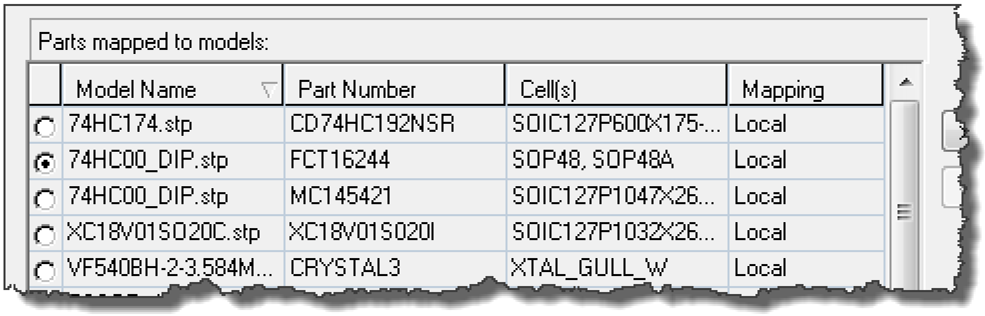
Procedure

1. In the SailWind 3D window, click the **Manage Mappings** button. 

2. Select the “Enable cross probing” check box in the Manage Mappings Dialog Box.

3. Click a component in the 3D window that has the 3D model you want to reuse and make note of the corresponding entry indicated in the “Parts mapped to models list” of the Manage Mappings dialog box.

Figure 43. Selected Model Entry in the Manage Mappings Dialog Box



Alternatively, you can click the model entry directly in the list to select it.

4. In the "Parts without model" list of the Manage Mappings dialog box, select the check box of the component or components that will reuse the 3D model.

5. Click the left arrow to map the selected component in the "Parts without model" list to the 3D model.

6. If a message appears cautioning about differences in alignment data between decals ("cells"), click **No** to retain the current alignment of the 3D model you are reusing or click **Yes** and choose another alignment as follows:

   a. Click in the "Use alignment" box in the Choose Alignments Dialog Box.

   b. Select an existing decal alignment from the list or click **None** to "align the 3D model" on page 524 manually.

7. Click **OK** to map the component to the model.

8. If you need to re-align the 3D model, right-click it and click the **Edit Decal** popup menu item to open the Decal Editor. See Assigning a 3D Model to a Component for more information about aligning 3D models.

Related Topics

Saving 3D Models for Re-use in Other Designs

Update Library Dialog Box

Update Models Dialog Box

Holes in SailWind 3D

In some cases, you must add certain attributes to components if you want the holes to display properly in SailWind 3D.

Holes Displayed in the 3D Window

Pin holes, mounting holes, and other component holes do not always render as voids in SailWind 3D. The display of holes depends on several factors, such as whether or not the component is an ECO-registered part or a non-ECO registered part, the number of pins, and whether or not it has a pin with a drill specification.

Generally, the pin holes of a component (such as a through-hole capacitor) always appear filled in SailWind 3D. For all other components, you must assign either the “HOLE” attribute or the “MountingHolePins” attribute for holes to display as voids.

The following sections summarize how holes render in SailWind 3D under various conditions.

ECO-Registered Parts

- Pin holes display as filled by default, even for through-hole components and components with only a single pin.
- Components with multiple pins that have mounting holes still render as filled.
- You can make the voids visible in SailWind 3D by applying either the “"HOLE" attribute” on page 531 or the “"MountingHolePins" attribute” on page 532.

Non-ECO Registered Parts

- Pin holes display as filled by default UNLESS the component has a single pin with a non-zero drill specification, in which case it converts automatically into a mounting hole and displays as such.
- SailWind 3D automatically renders a non-ECO component with a single pin as a mounting hole if the pin has a non-zero drill specification. This automatic rendering prevents you from selecting the mounting hole in the 3D window (to assign a 3D model to it, for example).

Note:
A mounting hole generally displays as a hole in the board, nothing else.

- You can assign a 3D model, such as a standoff, to a non-ECO component if you first “apply the MountingHolePins attribute” on page 530. Assigning this attribute enables you to select the component in the 3D window. Assigning the attribute also effectively converts the component from a mounting hole to a mechanical component with one mounting hole inside it.

Note:
SailWind 3D renders any non-ECO registered part with the "MountingHolePins" attribute as a mechanical component if the part has only a single pin (such as is the case for a mounting hole or fiducial). Be aware that turning off the display of mechanical models in the SailWind 3D settings hides such parts. Also be aware that these parts export as mechanical parts when exporting the design to MCAD software.

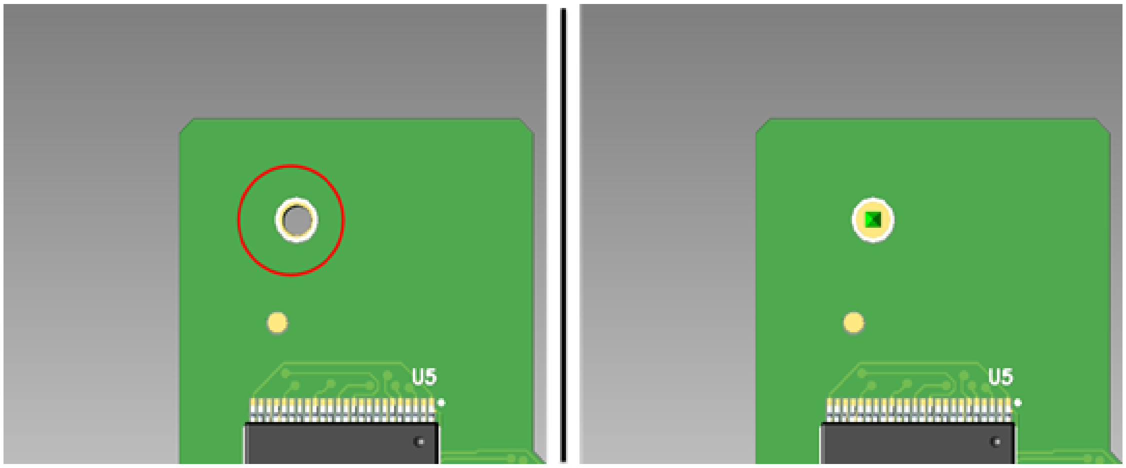
Assigning a 3D Model to a Non-ECO Mounting Hole

In some cases, you may want to assign a 3D model, such as a standoff, to one or more board mounting holes. You can assign a 3D model after first applying the "MountingHolePins" attribute.

Procedure

1. In the 2D workspace, right-click the mounting hole then click the **Attribute** popup menu item.
2. In the "Object Attributes dialog box" on page 1496, select the component level from the "Attributes for" list.
3. Click **Add**.
4. In the new Attribute box, type, "MountingHolePins" and press the Enter key.
5. (Optional) If you want the hole to continue to display as a void, type the pin number in the Value box.
6. Close the Object Attributes dialog box.
7. In the 3D window, select the corresponding mounting hole (note that it now displays as filled unless you specified the pin number in the Value box). Apply and "assign a new part model" on page 523.

Figure 44. 3D Mounting Hole Before and After Applying the Attribute



Rendering Mounting Holes on ECO-Registered Parts as Voids

In some cases, ECO-registered parts may not appear correctly in the SailWind 3D window; specifically, if a part is a single pin part, you must add certain attributes to the part decal to ensure a correct appearance.

Procedure

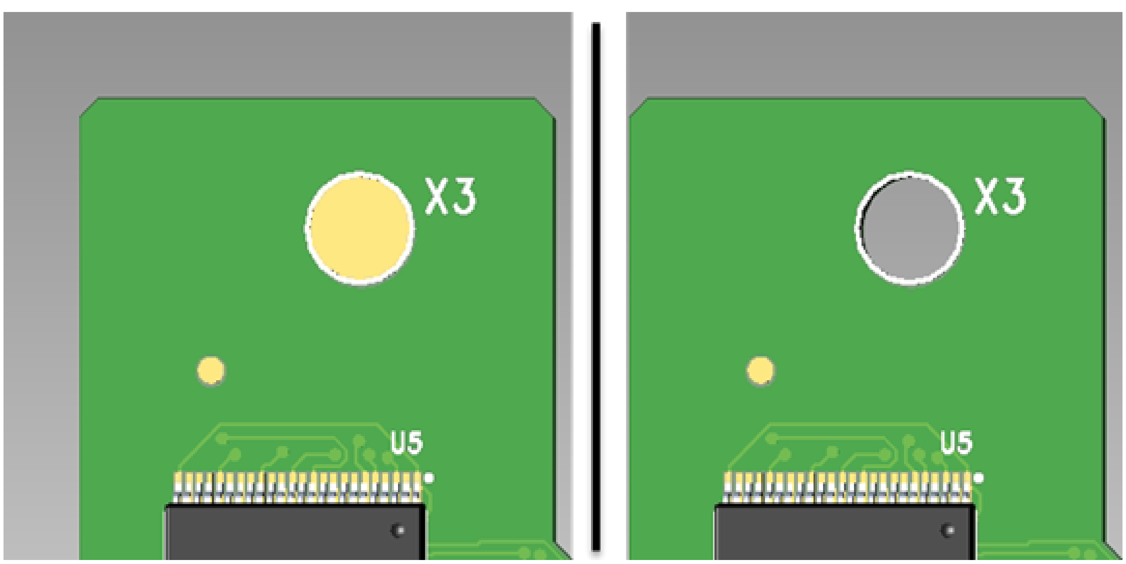
1. Right-click the component in either the 2D workspace or the 3D window, then click the **Attribute** popup menu item.
2. In the “Object Attributes dialog box” on page 1496, click **Add**.
3. Type one of the following attributes:

Table 104. Required Attributes

Pin Type	Attribute
Through-hole	HOLE
SMD	FIDUCIAL

For a single-pin, through-hole part, adding the “HOLE” attribute displays the hole as a void in the SailWind 3D window. For SMD single-pin parts, add the “FIDUCIAL” attribute in cases where you want the part treated as a fiducial.

Figure 45. ECO-Registered Mounting Hole Before and After Applying the HOLE Attribute



Setting Multi-Pin Components to Display Holes for Some Pins

You can set attributes on an ECO-registered part so that the holes associated with one or more selected pins render in SailWind 3D as voids.

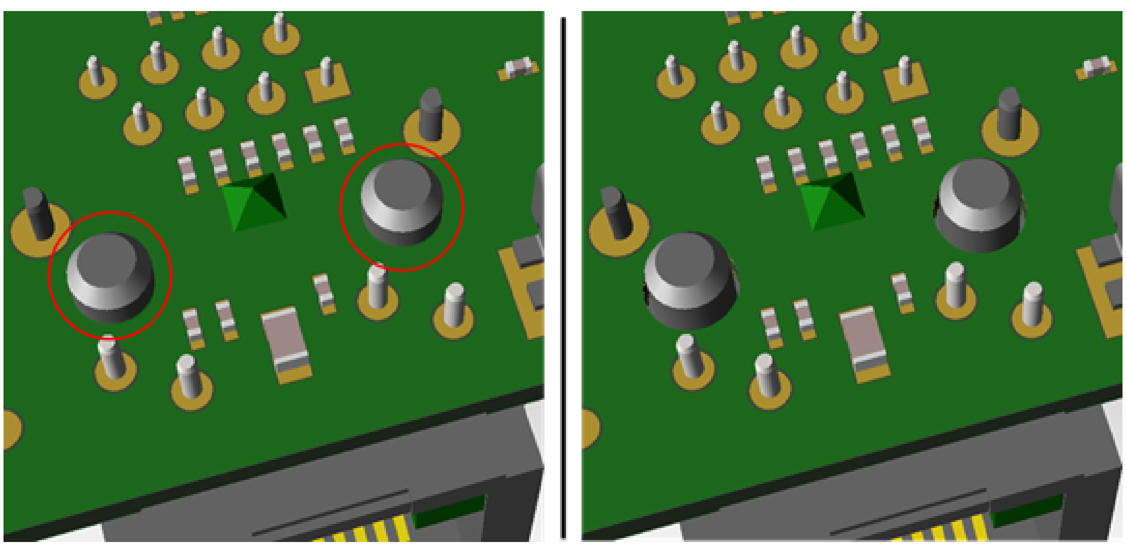
Procedure

1. Right-click the component in either the 2D workspace or the 3D window, then click the **Attribute** popup menu item.
2. In the “Object Attributes dialog box” on page 1496, click **Add**.
3. Type “MountingHolePins” as the new attribute name and press the Enter key. In the Value box, type the pin number for each pin in the component where you want the corresponding hole to display as a void. Ensure you separate each pin number with either a comma or a space (for example, “P001,P003,P005,” and so on).

Note:
Keep the following in mind

- Ensure you specify the pin number of the part, NOT the decal pin number
- If you specify all pin numbers for an ECO-registered part, SailWind 3D renders the part as a mechanical model. Be aware that these parts export as mechanical parts when exporting the design to MCAD software.
- The software takes longer to render the 3D image of your PCB the more pin numbers you specify for hole rendering.

Figure 46. Mounting Pin Holes Before and After Applying the Attribute



Changing a 3D Model Assignment

You can change or assign a different 3D model to a component decal.

Restrictions and Limitations

- Mapping through the Decal Editor allows you to choose to apply changes to only a component instance, if desired. Mapping in the 3D window applies changes to all instances of a component.
- If you have already updated the library with a 3D model assignment, you can change the assigned 3D model but you cannot remove all 3D model associations from the component.
- Be aware that any mapping changes made in the Decal Editor—including assignment and alignment of 3D models—affect the decal timestamp, which may affect the schematic during backward annotation.
- In the 3D window, you can use the Manage Mappings Dialog Box to apply mappings to other components in the design.

Procedure

1. Select the part to which you want to assign a different 3D model.
2. Click the **Import Part Model** button on the 3D window toolbar. .

   Alternatively, if you want to use the Decal Editor, perform the following steps:

   a. Right-click on a component and click the **Edit decal** popup menu item.

      The Decal Editor opens and displays the component decal and its currently mapped 3D model.

   b. In the Align 3D Models Dialog Box, select the part or parts in the “Parts mapped to selected model list” that you want to reassign then click **Import**.
3. In the Import Part Model dialog box, browse to and select the correct model definition file you want to import and click **Import**.

   If a dialog box appears with a caution about rewriting a file, click **Yes** to overwrite the model currently in the local design library, **No** to import the file as a new model, or **Cancel** to exit without making changes.
4. “Rotate or reposition” on page 525 the imported 3D model as necessary so it aligns correctly with the decal pads; then click **OK**.
5. (Decal Editor only) Choose the **File > Exit Decal Editor** menu item when you are finished aligning the imported model.

   In the Apply Decal Changes prompt, click **All** to apply the 3D model to all parts using the decal in the design or **Selected** to apply the model to the selected part only.

   The new 3D model appears in the Model Name list of the Align 3D models dialog box as the model mapped to the decal.
6. In the “Apply Decal Changes” dialog box, click **All** to apply the 3D model to all parts using the decal in the design.
7. If you previously saved the model mapping to the Reuses file location, you can update the mapping:

a. Click the **Update Library** button in the SailWind 3D window toolbar.

b. In the Update Library Dialog Box, select the check box of the part with the new mapping.

c. In the Overwrite policy list, choose “Overwrite all in Library.”

Importing and Aligning 3D Mechanical Models

Add mechanical designs such as mounting brackets to your PCB design in the SailWind 3D window to examine clearances and construction with the rest of the PCB. Adding 3D mechanical designs helps to ensure the PCB parts fit with the mounting hardware or enclosures in the final, packaged PCB assembly.

You can import mechanical designs in a wide variety of file formats, including *.asat*, *.iges*, *.sat*, *.step* (*.stp*), and *.xtda*.

Prerequisites

- You have opened a design in SailWind Layout and you have also opened the SailWind 3D window.

Procedure

1. In the SailWind 3D window, click the **Mechanical Model Properties** button .
2. In the Mechanical Model Properties Dialog Box, click the **Add** button ; then browse to and select the mechanical model file and click **Open**.

   Tip
   You can also import a mechanical model from the SailWind 3D window by clicking the **Import Mechanical Model** button .You can then modify its properties by selecting the model and clicking the **Mechanical Model Properties** button.

   The imported mechanical model appears in the SailWind 3D window and a temporary instance name appears in the **Instance** dropdown list.
3. Rename the mechanical model by typing a more descriptive name in the **Instance** dropdown list.
4. Select a model type to determine which 3D clearance rules to apply and where to display the model in the 3D Display Control settings (see 3D Display Control Window).
5. Rotate and align the 3D mechanical model to its proper position in relation to the PCB design using the Rotate and Offset controls. Alternatively, select the Allow interactive mechanical model movement check box to reposition the mechanical model by dragging it with the mouse. Rotate the PCB angle as required to align the mechanical model in the X, Y, and Z direction. See SailWind 3D Object Manipulation.
6. Select or clear the check boxes for your save options.
7. Close the dialog box and save the PCB design when you are through adding the mechanical model.

Results

You have added and aligned a mechanical model to your 3D PCB layout. If necessary, you can use the commands on the mating toolbar to ensure an accurate fit between mechanical models. For more information, see Positioning 3D Mechanical Models Using Mating Commands.

Positioning 3D Mechanical Models Using Mating Commands

You can use the mating commands (on the 3D window Toolbar) to accurately mate the surface of an imported 3D mechanical model with another surface. Two mated surfaces connect physically and remain in contact. Attempts to manipulate movement of one mated model is thus limited to permit only movement that keeps the mated surfaces in contact.

Note:
The mating commands apply to mechanical models only. To move or manipulate components, use the Decal Editor.

Prerequisites

You have already imported a 3D mechanical model and rotated or aligned it roughly in the position where you want to place it.

Procedure

1. In the SailWind 3D window, click to choose the mechanical model that you want to mate.
2. On the 3D window toolbar, click the **Mate Selected Models** button. 
3. On the mechanical model, click the surface, edge, or other feature (such as a hole) that you want to mate to another surface.

   To aid in choosing the feature, right-click and click a popup menu item to limit feature selection to the type of feature you want to align:

   - **Snap to Closest** — Enables you to click any mechanical feature of the model, whether it is an axis, edge, or face.
   - **Snap to Axes** — Enables you to click the discernible X, Y, or Z axis of a feature on the model, such as a hole, cutout, or corner.
   - **Snap to Edges** — Enables you to click any edge of the mechanical model.
   - **Snap to Faces** — Enables you to click one of the non-edge surfaces of the mechanical model.

   SailWind Layout highlights the chosen feature after you click it.
4. Click a corresponding surface or mechanical feature on the other model where you want the mechanical model to mate. For example, if you choose a standoff on the mechanical model, and you want to mate it to a mounting hole on the PCB, click the desired mounting hole on the PCB.

The mechanical model moves or snaps to the selected mechanical feature. In addition, in the Mechanical Model Properties Dialog Box, the alignment and movement features become disabled. SailWind Layout permits only movements that allow the two mated surfaces to remain in contact.

5. To disassociate or "unmate" the two surfaces at any time, click the mechanical model again to choose it and click the **UnMate selected models** button.

   If you want to unmate all models at one time (without having to choose them all first), click the **UnMate all models** button.

Creating a Board Outline From an Imported Mechanical Model

You can import a mechanical model into the SailWind 3D window for use as a physical board outline in your design. If you want to use any of the holes already in the mechanical model, you must map them to their proper type, such as cutouts or mounting holes, to ensure proper import.

Note:
If you re-create a board outline using the same mechanical model, SailWind Layout discards your previous hole mappings.

Prerequisites

You have imported the mechanical model that you want to use for your board outline.

Procedure

1. In the SailWind 3D window toolbar, click the **Create Board Outline** button.
2. Select the surface of the imported mechanical model that you want to use for the board outline.
3. Click the location on the mechanical model that you want to use as the origin of the board outline.
4. A list of holes already in the imported mechanical model appears in the Map Hole Features Dialog Box. Select each hole that you want to use for the board by clicking the Hole Type box and choosing the corresponding hole type from the dropdown list:

   - <Discard> — The default setting. The software ignores holes set to "Discard" when it generates the board outline.
   - Contour — Indicates the hole is a cutout.
   - MountingHole — Indicates the hole as a board mounting hole. Choose this option only if you want to map a hole in the mechanical model to an existing hole in your PCB design. If you choose this hole type, click the Hole Attribute box and select the corresponding mounting hole attribute from the dropdown list.

   You can clear the Highlight check box for each hole individually to make identification easier. The SailWind 3D window centers a hole on the display if you select its check box again. Alternatively, you can click any hole on the mechanical assembly to cross probe with the holes in the dialog box.

5. Click **OK**. The Map Hole Features dialog box closes and the design updates with the new board outline.

   The updated board outline appears in both the SailWind 3D window and the layout design view.

Creation of 2.5D Models

SailWind Layout creates a “2.5D” model automatically by combining a component’s outline (such as that used for silkscreen) with its Geometry.Height attribute. When you open the SailWind 3D window, any part that does not already have a 3D model assigned is represented by a 2.5D outline. The component outline must meet certain requirements to ensure proper generation of the 2.5D model.

Component Outline Hierarchy

SailWind Layout locates component outlines using a layer hierarchy:

1. Layer defined in the *SailWindpcb.ini* file
2. Layer 25 (125)
3. Top associated assembly layer
4. Top layer
5. <All Layers>
6. Layer 20 (120)

To define a component outline layer in the SailWindpcb.ini file, open the file (at *<drive>\<install_folder>\<release>\Settings\SailWindpcb.ini*) with a text editor and add the following line:

```
3D_Shape_Layer = <layer number>
```

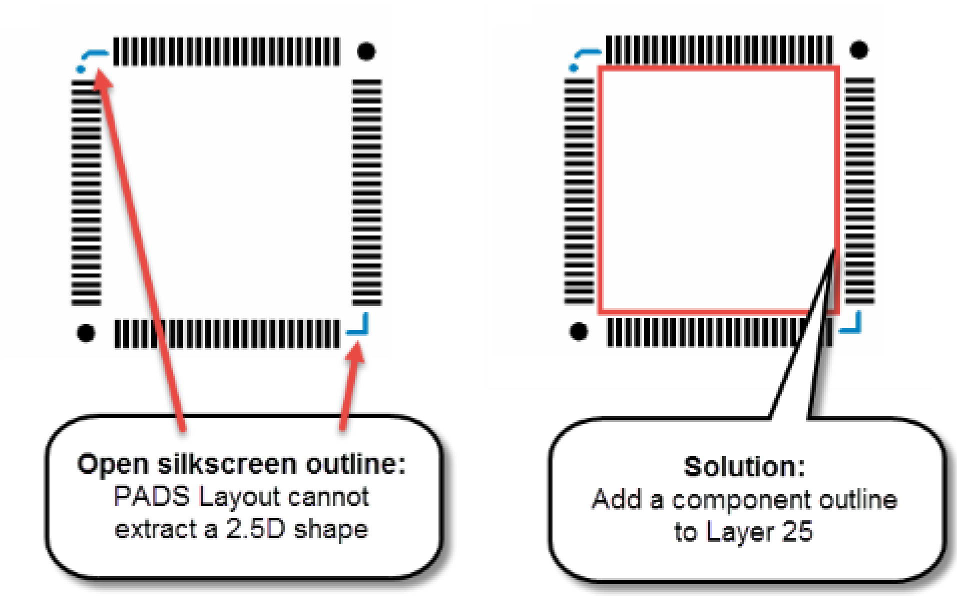
Where <layer number> is the component outline layer.

Component Outline Requirements

Component outlines must be closed in order for SailWind Layout to generate a 2.5D model. If you place any open component outlines on a layer defined in the *SailWindpcb.ini* file, for example, SailWind Layout disregards the open outlines and selects only those that are closed.

To ensure SailWind Layout generates 2.5D models for all components, create closed component outlines and place them on Layer 25. SailWind Layout locates the closed component outlines as it searches through the component outline hierarchy. For example, if you have open silkscreen outlines for a component on the top layer—and the top layer is defined as the component outline layer in the *SailWindpcb.ini* file—ensure you add corresponding closed outlines of the component to Layer 25.

Figure 47. Open and Closed Component Outlines



Related Topics

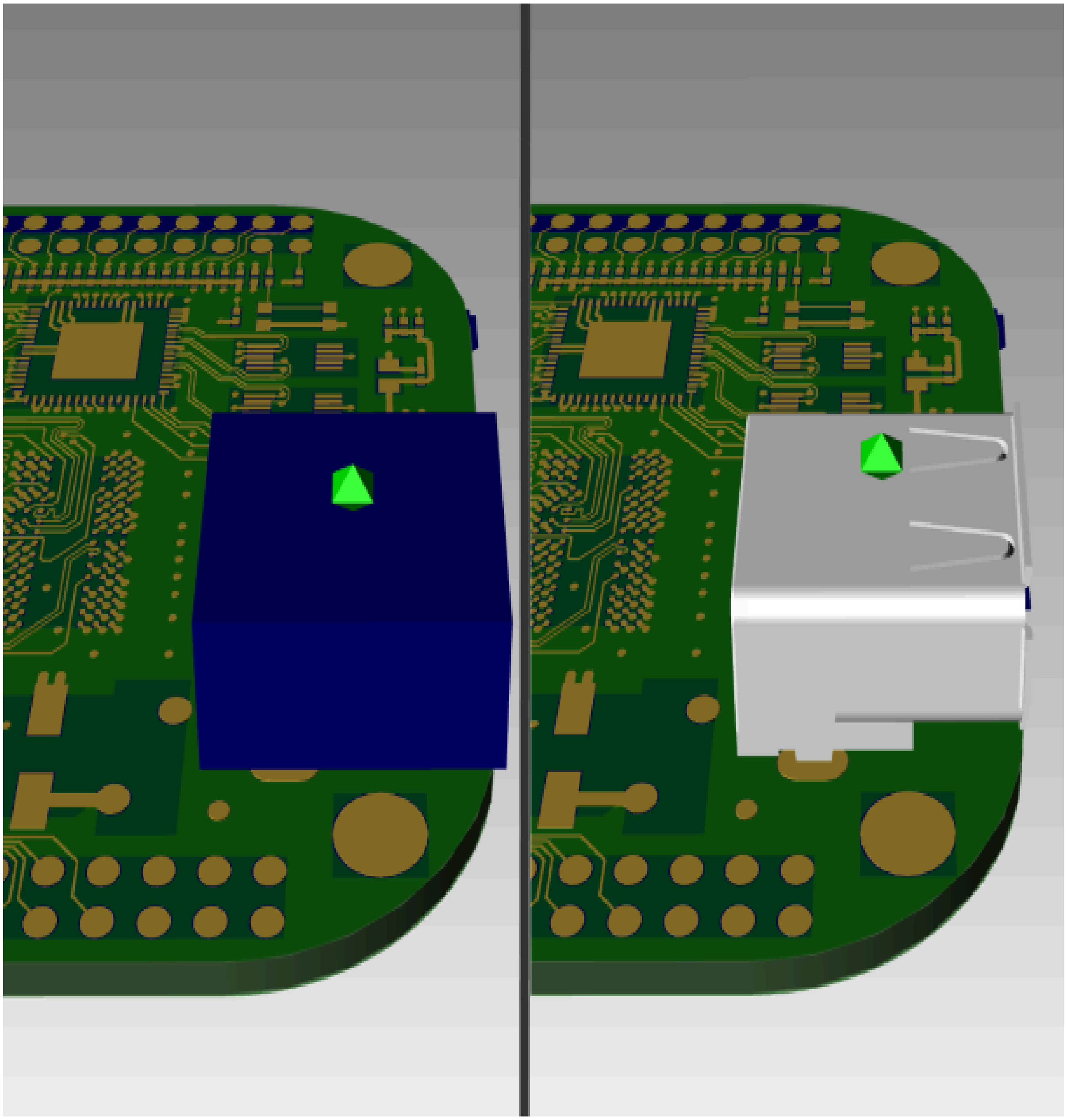
Compare 3D and 2.5D Models

Compare 3D and 2.5D Models

As an additional aid for laying out parts on your PCB, you can view the “2.5D” models in tandem with 3D parts in the SailWind 3D window. Enabling the 2.5D view allows you to check the alignment and positioning of a 3D model in relation to the original 2D layout to verify you have mapped the correct model.

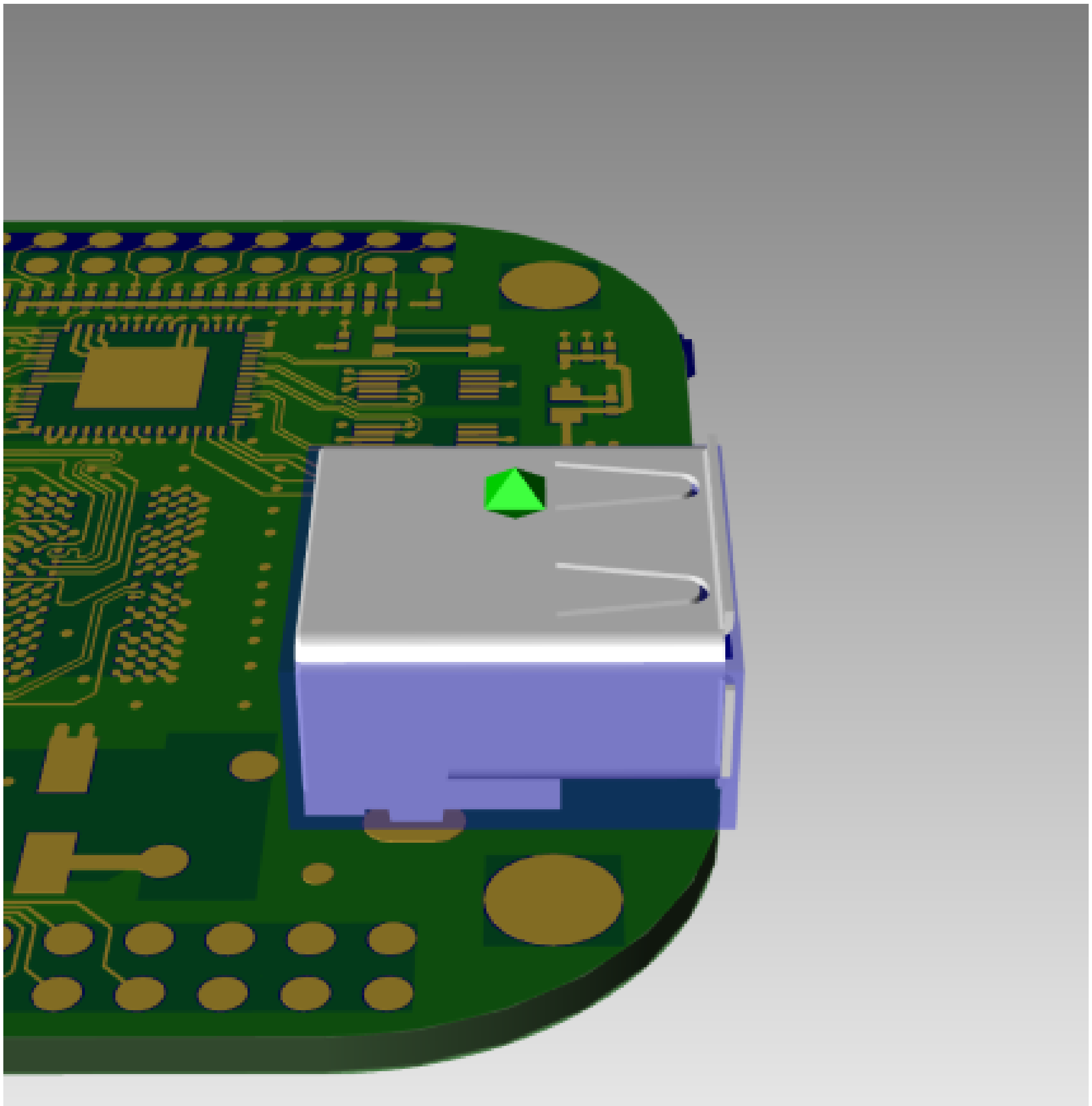
The “2.5D” model is an extrusion of the decal’s silkscreen outline and the Geometry.Height attribute. When you open the SailWind 3D window, any part that does not already have a 3D model assigned is represented by a 2.5D outline of its decal. Any 3D model you assign to a part replaces the 2.5D model.

Figure 48. A 2.5D Model Replaced by a 3D Model



As a layout verification, you can turn on the 2.5D model after assigning a 3D model by selecting the Overlay 3D Models with 2.5D check box in the 3D Display Control Window. The correctly aligned 3D model overlays the 2.5D outline.

Figure 49. The Original 2.5D Model Overlaying the 3D Model



Reposition or realign the 3D model as necessary to reconcile any significant placement differences between the 2.5D and 3D models. If the models are not the same shape, ensure you have mapped the correct 3D model to the part.

See "Comparing 2D Components with 3D Models" on page 541 for instructions.

Related Topics

Assigning a 3D Model to a Component

Decal Attributes Dialog Box

Creation of 2.5D Models

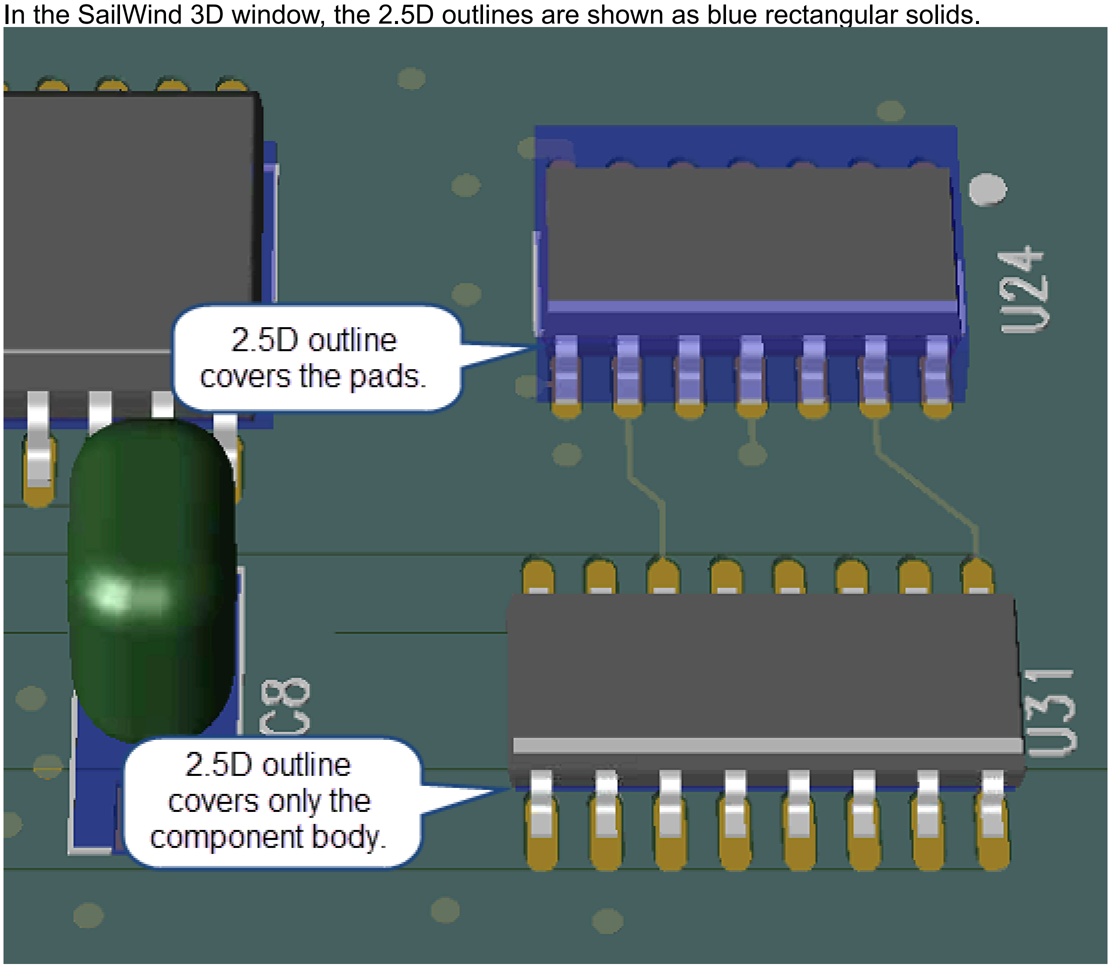
Comparing 2D Components with 3D Models

Overlay an extruded 2D outline on a 3D model to verify the correct alignment and positioning of the 3D model.

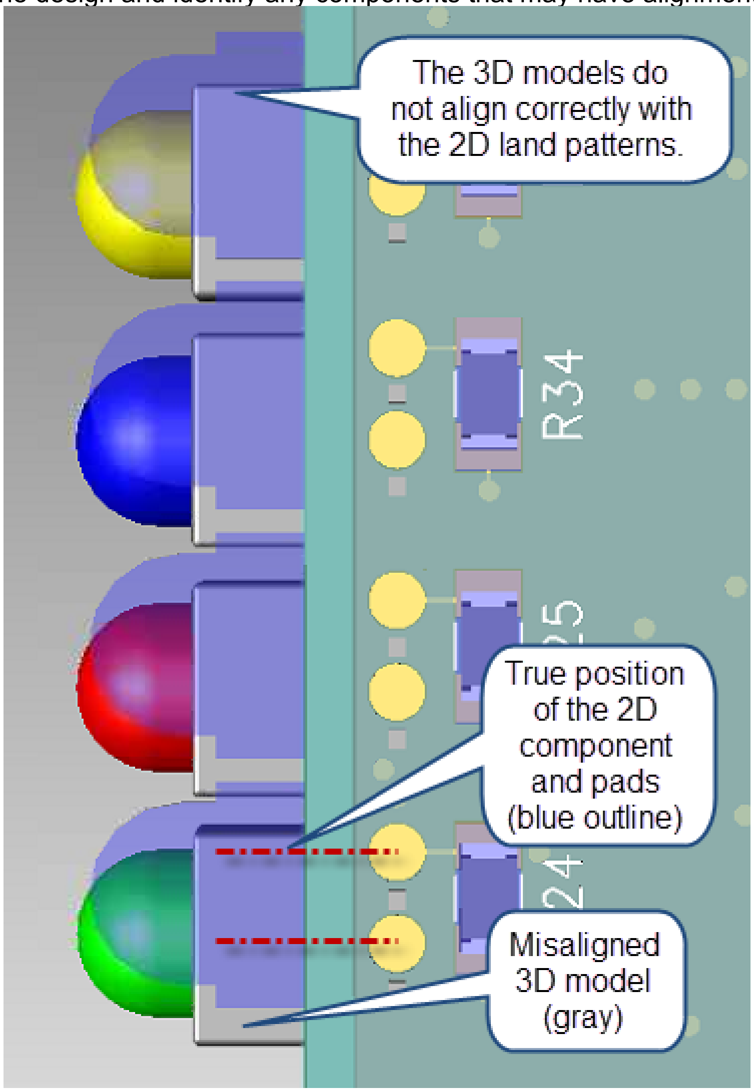
Procedure

1. In the 3D Display Control Window, Options section, select the Overlay 3D models with 2.5D check box.

   In the SailWind 3D window, the 2.5D outlines are shown as blue rectangular solids.



2. Examine the design and identify any components that may have alignment or positioning problems.



Tip
To better evaluate the alignments, rotate the 3D image or change the view settings as needed. If necessary, clear the Perspective check box in the 3D Display Control dialog box so you can see the alignments and positions more clearly.

3. Correct any problems by adjusting the alignment of the 3D models so they are positioned correctly relative to the 2D components and land patterns.

Saving 3D Models for Re-use in Other Designs

You can save 3D (“map”) model-to-decal alignments from one PCB design and re-use them in an existing design wherever the part types and decal names are the same. For example, if you have two PCB designs that share a specific IC with an SO-16 decal, mapping a 3D model to the IC in one PCB design allows you to use the same 3D model with the IC/decal combination in the other PCB design. SailWind

Layout saves your 3D model mappings to the Reuses file location specified in the **Tools > Options > Global > File Locations** tab.

Prerequisites

- You are viewing a PCB design in the SailWind 3D window.
- You have mapped at least one 3D model to a decal in the PCB design.

Procedure

1. In the SailWind 3D window, click the Update Library button. 
2. In the Update Library Dialog Box, select the 3D model mappings that you want to save. Mappings are listed in the dialog box according to the part and decals to which they are mapped.

   Select the Enable cross probing check box if you want the ability to click on a part in the design to interactively locate its corresponding mapping in the list.
3. Select an overwrite policy from the Overwrite policy list.
4. Click **Apply** to save the 3D model mapping or mappings to the Reuse file location.
5. Click **OK** to close the message prompt.
6. Click **Cancel** to close the dialog box.

Results

You have mapped 3D models to part types in the Reuse file location and you have saved the 3D model alignment information with the part decals. The 3D models automatically appear in other designs wherever the part and decal combination is the same. If changes occur to any of the mapped 3D models externally to the PCB design, you can import the changes by clicking the **Update Models** button in the SailWind 3D window. 

Related Topics

Reusing 3D Models with Other Components in a Design

3D Model Mapping

Component Placement in the SailWind 3D Window

As an additional tool for PCB layout, you can use the SailWind 3D window for component placement. Using the SailWind 3D window allows you to check the component placement in three dimensions.

After you import a design, SailWind Layout initially places all the design components on the PCB origin. With the SailWind 3D window, you can disperse the components and place them around the PCB as you would in the standard two-dimensional design view. The SailWind 3D window supports many of the standard component placement functions through use of a context menu.

For example, you can spin, rotate, nudge, disperse, and move components to the PCB flip side from withing the SailWind 3D window by right-clicking on a component and selecting an option from the popup menu. For more information about component placement, see “Part Placement” on page 471.

Restriction:
Use the standard two-dimensional design view to route and connect your components; the SailWind 3D window does not support component routing.

As with the two-dimensional design view, you can use on-line Design Rule Checking (DRC) to alert you to clearance warnings or violations while placing components in the SailWind 3D window. See "Viewing DRC Violations in SailWind 3D" on page 545 for instructions.

Synchronizing the Design and SailWind 3D Viewing Areas

Set options to enable the views in the Design space and the SailWind 3D window to mirror one another in zoom level and pan area.

Procedure

1. In the SailWind 3D window with nothing selected, right-click and click the **3D Display Control** popup menu item, or on the 3D General toolbar, click the **3D Display Control** button.
2. In the "3D Display Control Window" on page 1034, in the Options section:
    - Select the Drive 2D View check box to send synchronization commands to the Design view. Zoom levels and pan moves made in SailWind 3D will be matched in the Design space.
    - Select the Follow 2D View check box to receive synchronization commands from the Design view. Zoom levels and pan moves made in the Design space will be matched in SailWind 3D.
3. Click **OK**.

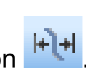
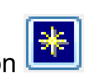
Defining 3D Clearance Constraints

Define 3D clearance X,Y, and Z axis constraints for the objects in SailWind 3D. You can define generic, mechanical model type, and component specific constraints.

Restrictions and Limitations

- 3D clearance constraints affect only models in SailWind 3D and do not conflict with or override design constraints.

Procedure

1. In the SailWind 3D window, on the 3D General toolbar, click the **Clearances** button .
2. In the 3D Clearances Dialog Box, to add a new row, click the **Add** button .
3. In the new row, click the From and To fields to choose objects from the lists.

4. Type values where needed in the Minimum XY, Minimum Z, Optimal XY, and Optimal Z boxes. If there are blank values the clearance is not checked. For example, this allows you to omit the XY clearance for a connector sticking out past the board edge. Specify a blank XY value for the specific component to board edge constraint.

5. Repeat as necessary.

6. Click **OK**.

7. For the next step, see either “Running Batch 3D Clearance Checking” on page 546 to check all design objects or “Viewing DRC Violations in SailWind 3D” on page 545 to view clearance violations as you move objects.

Viewing DRC Violations in SailWind 3D

Enabling standard on-line Design Rule Checking (DRC) in the **Tools > Options** menu also allows you to view 3D clearance violations in the SailWind 3D window while you position components on the PCB.

You can define your 3D clearances using the 3D Clearances Dialog Box.

Restrictions and Limitations

DRC in the SailWind 3D window checks only the 3D clearances; they do not conflict with or override design constraints.

Prerequisites

You have already defined one or more 3D clearances for your project (see “Defining 3D Clearance Constraints” on page 544).

Procedure

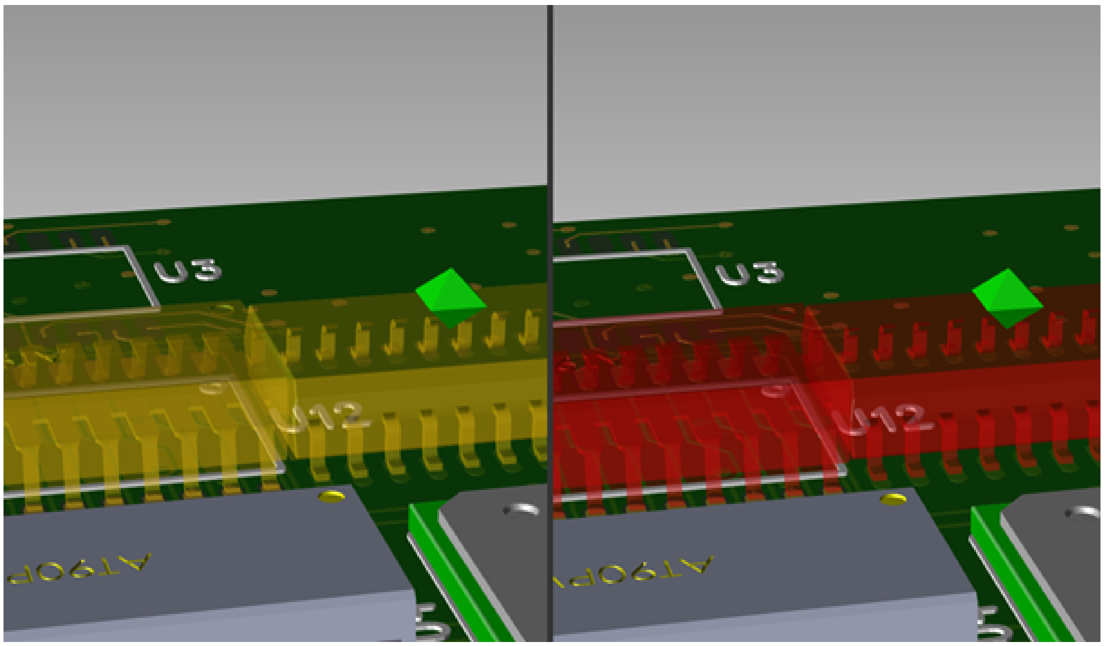
1. Enable on-line DRC in either the “Prevent Errors” or “Warn Errors” mode—see “Design Rule Checking” on page 410.

2. Position your PCB components as required. While positioning your a component, the component changes color (you may have to hold to component with the cursor at a position and allow it to “hover” a moment) to indicate 3D clearance violations as follows:

- **Optimal XY or Optimal Z violations** — Affected components or surfaces highlight in yellow when placement exceeds any “optimal” clearance.

- **Minimum XY or Minimum Z violations** — Affected components or surfaces highlight in red when placement exceeds any minimum clearance.

Figure 50. 3D Optimal and Minimum Violation Examples



3. Reposition the affected component as required until the highlight disappears, indicating the 3D clearance is no longer exceeded.

Running Batch 3D Clearance Checking

Check all 3D objects against the 3D Clearances constraints.

Prerequisites

- 3D Clearances must be defined. For more information, see “Defining 3D Clearance Constraints” on page 544.

Procedure

1. In the SailWind 3D window, on the 3D General toolbar, click the **3D Design Rule Check** button . A Processing dialog box opens displaying the elapsed time for the process. Then a window opens displaying the number of hazards found and showing a path to the log file.

2. In the Output Window, click the link to view the contents of the log file.

Changing the Appearance of Components and Models in 3D

You can alter the appearance of components and assemblies in the SailWind 3D window to make it quicker to identify key groups or components. For example, you can color code all 3D models of high frequency parts to make grouping them easier prior to routing. You can also make components transparent to view details beneath them.

Procedure

1. Open the 3D Display Control Window.
2. Select a component or mechanical model using one of the following methods:
   - Click on a component or mechanical model on the PCB in the SailWind 3D window; then in the appropriate Components, Assemblies, Mechanicals, or PCBs area of the 3D Display Control dialog, click **Add selected** to add it to the list.
   - If the component does not appear in the list, click **Add** and select it.
3. Locate the component in the appropriate list (Assemblies, Components, Mechanicals, or PCBs) of the 3D Display Control dialog box and select its corresponding Appearance checkbox. Ensure the Appearance check box is selected for the component or assembly and make the following changes are desired:
   - Assign a color from the Color palette and set the transparency to the desired level.
   - Clear the corresponding Visibility check box to hide a component or assembly.
4. Click **Apply** after making your changes.
5. Click **OK** to close the dialog box when you are done.

Measuring In 3D with Measure Distance

Measure distances between features of the 3D models when working in the SailWind 3D window.

Note:
The units of measure are the design units you define in the Design units area of the **Tools > Options > Global/General** tab in SailWind Layout.

Prerequisites

- Open the Display Control dialog box by clicking the 3D **Display Control** button on the 3D General Toolbar and enable the correct display settings so the objects you want to measure are visible.

- Use the 3D rotate, pan, and zoom controls to orient the design in the SailWind 3D window so the objects you want to measure are contained within the view.

Procedure

1. On the 3D General Toolbar, click the **Measure Distance** button .

2. Choose one of the following options from the popup menu:

If you want to...	Choose...
Measure between any two features	**Snap to Closest** (Default) **Note:** Snap to Closest automatically determines and highlights the nearest axis, edge, face or point on the features.
Measure between the central axes of two circular features	**Snap to Axes**
Measure between the edges of two rectangular features	**Snap to Edges**
Measure between the flat surfaces of two rectangular features	**Snap to Faces**
Measure between the points (corners) of two rectangular features	**Snap to Points**

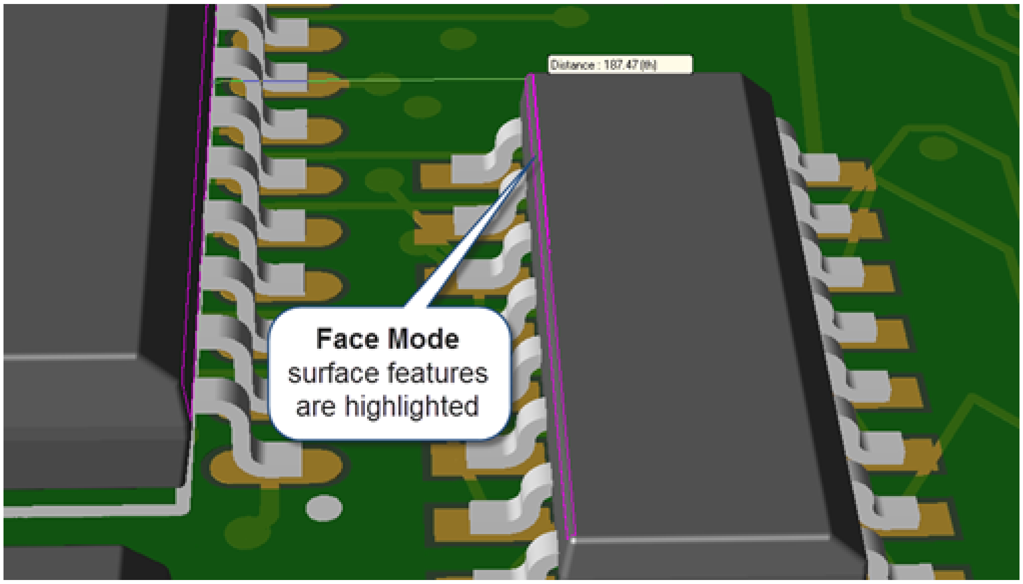
3. Click on a feature to mark the starting point of the measurement.

   The mode you select determines the appropriate feature automatically and highlights it. The distance from the starting point appears dynamically in a small readout attached to the cursor as you drag the cursor.

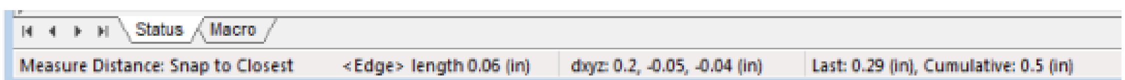
4. Hover over another feature to mark the ending point of the measurement.

   Note:
   Before selecting the second feature, (optionally) you can use the RMB menu to change the selection options.

   The second feature highlights. The final measured distance between the two features appears when you hover over the ending point.



5. You can sequentially select additional features and the status bar will display accumulative results of the measurement including the x, y and z offsets.



6. Press the Esc key once to release the currently selected feature so you can make additional measurements. Press the Esc key a second time to exit the Measure command.

Tip
You can right-click and click the **Cancel** popup menu item to release the current feature. Choose **Cancel** a second time to exit the Measure command.

Related Topics

SailWind 3D Object Manipulation

Assigning a 3D Model to a Component

3D Display Control Window

Measuring In 3D with Measure Minimum Distance

Measure the minimum distance between features of the 3D models when working in the SailWind 3D window.

Note:
The units of measure are the design units you define in the Design units area of the **Tools > Options** menu item, **Global/General** tab.

Prerequisites

- Open the Display Control dialog box by clicking the 3D **Display Control** button on the 3D General Toolbar and enable the correct display settings so the objects you want to measure are visible.
- Use the 3D rotate, pan, and zoom controls to orient the design in the SailWind 3D window so the objects you want to measure are contained within the view.

Procedure

1. Click the **Measure Minimum Distance** button to quickly measure the minimum distance between the closest features of two selected objects.

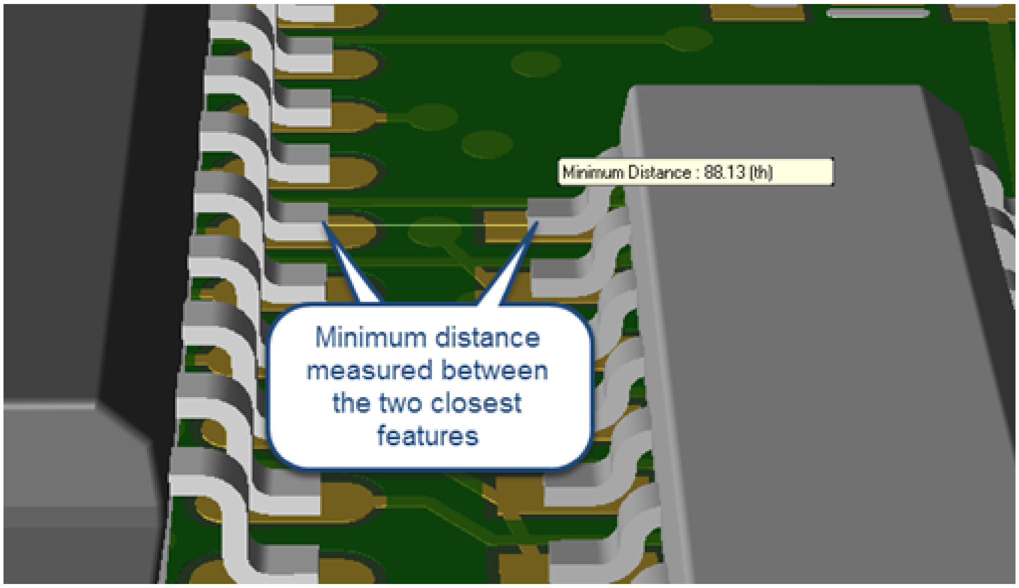
   Tip
   Choose either **Mechanical Model Selection** or **Board Selection** from the popup menu to limit your selections to one of those design levels.

2. Click on a feature to mark the starting point of the measurement.

   The mode you select determines the appropriate feature automatically and highlights it. The distance from the starting point appears dynamically in a small readout attached to the cursor as you drag the cursor.

3. Hover over another feature to mark the ending point of the measurement.

   The second feature highlights. The final measured distance between the two features appears when you hover over the ending point.



4. Press the Esc key once to release the currently selected feature so you can make additional measurements. Press the Esc key a second time to exit the Measure Minimum Distance command.

Tip
You can right-click and click the **Cancel** popup menu item to release the current feature. Choose **Cancel** a second time to exit the Measure Minimum Distance command.

Related Topics

SailWind 3D Object Manipulation

Assigning a 3D Model to a Component

3D Display Control Window

Viewing Internal Layers in SailWind 3D

The SailWind 3D window provides a virtual preview of the final manufactured PCB, allowing you to see into the layer stackup and inspect internal routing, via connections, and other design factors not visible in the 2D view.

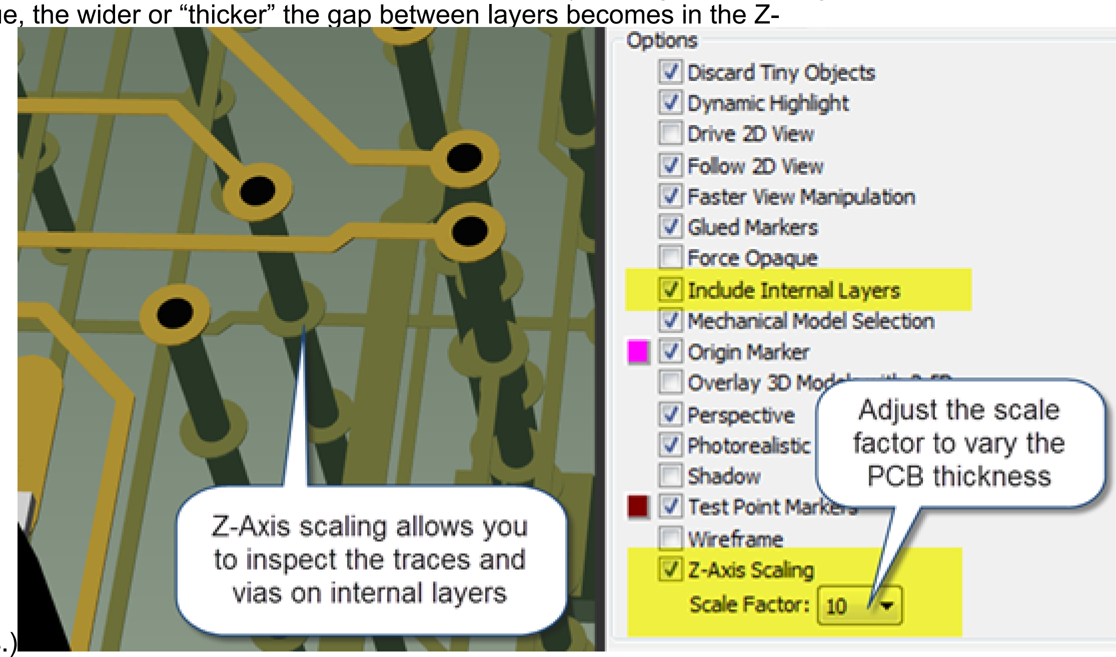
Use the SailWind 3D window to analyze how blind and buried vias are connected through the layer stackup on a routed design. By scaling the design and zooming into areas of dense routing, you can verify that via connections on internal layers meet design specifications. Manipulating or rotating the design in the SailWind 3D window may also help you identify regions in high density designs where you can improve the placement or routing (see "SailWind 3D Object Manipulation" on page 519).

Procedure

1. In the SailWind 3D window, open the 3D Display Control Window.

2. Select the "Include Internal Layers" check box to view the traces and pads on the internal layers.

3. In the Options section, select the “Z-Axis Scaling” check box and choose a value for the Scale Factor from the dropdown list. (The larger the scaling value, the wider or “thicker” the gap between layers becomes in the Z-axis.)



Tip
Selecting the Z-Axis Scaling check box also displays via barrels in the inner layers that are not otherwise visible when selecting the “Include Internal Layers” check box alone.

4. Use the 3D view controls to orient the design in the SailWind 3D window and focus on a specific area you want to analyze.

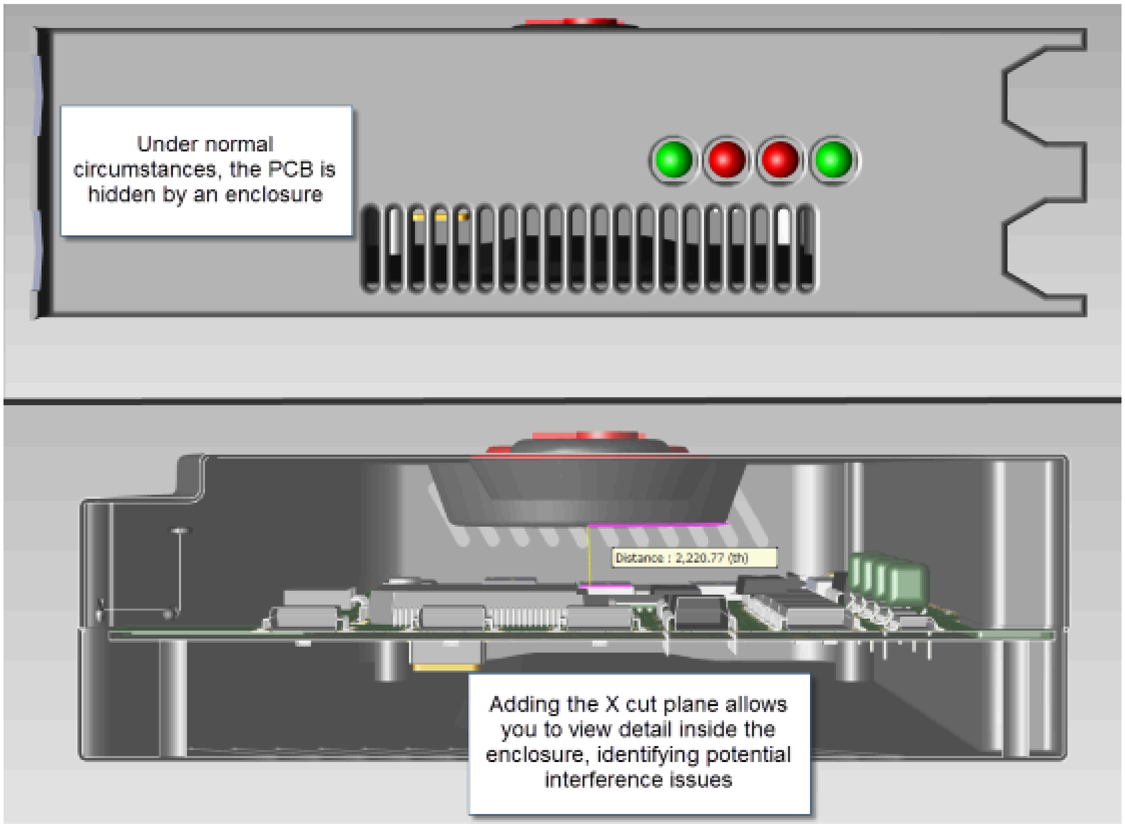
5. Visually examine the critical traces and vias to determine if the inner layer structures meet the design requirements. Note that the SailWind 3D window measuring tools (Measure Distance and Measure Minimum Distance) continue to report true material measurements regardless of z-axis scaling.

   If necessary, you can select the Copper Planes check box (in the Objects area) to view internal layer copper planes. Ensure you click **Refresh** after selecting the Copper Planes check box to ensure the copper planes are visible.

Creating 3D Cross Sections

Create cross-sectional “cutaway” views in the SailWind 3D window to view details such as the fit between the PCB components and mechanical assemblies or enclosures.

Figure 51. Using a Cut Plane to Create a Cutaway View

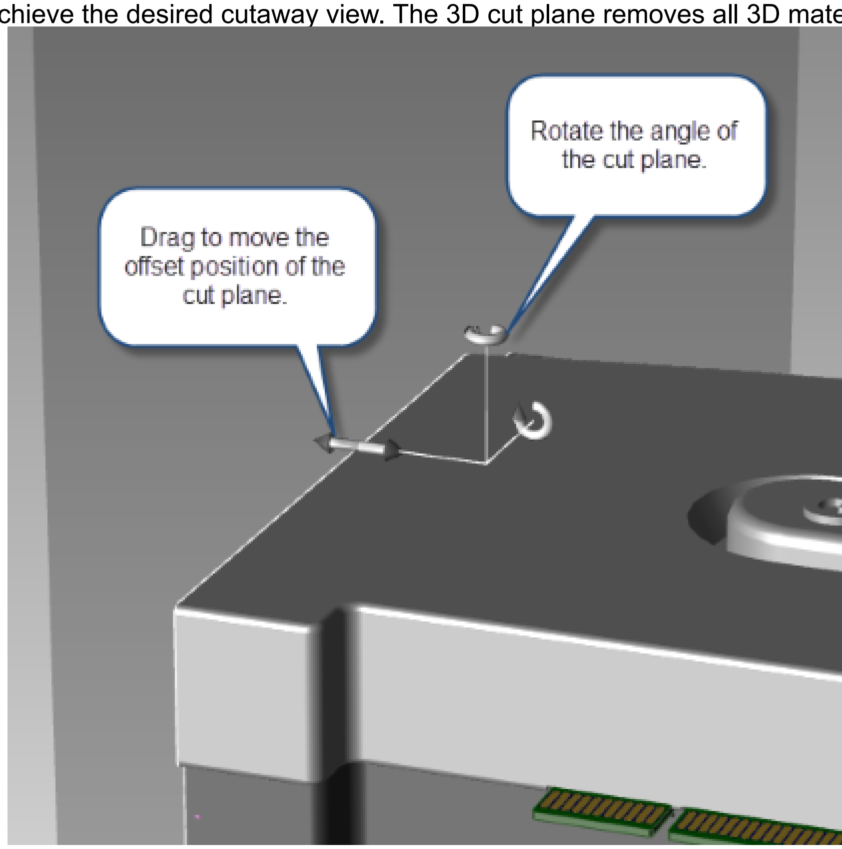


Creating a cross sectional view allows you to access components, vias, traces, and other details hidden by enclosures or other objects.

Procedure

1. In the SailWind 3D window, click the **X Cut Plane**, **Y Cut Plane**, or **Z Cut Plane** button, depending upon which direction you want to "cut." 

   You can click one, two, or all three cut plane buttons and have more than one active cut plane at one time (though only one cut per plane type).

2. When the cut plane appears, drag the cut plane to the location where you want it and rotate it as necessary to achieve the desired cutaway view. The 3D cut plane removes all 3D material to one side from view. 

3. Click the cut plane button again to remove the cut plane.

   The cut plane returns to its original pre-defined starting location the next time you click its corresponding button.

Checking the Status of the 3D Design

As you make changes in the SailWind 3D window, you may lose track of the parameters you have set and how the design is currently configured. You can generate a report of the 3D statistics and verify that the 3D design is acceptable.

Procedure

1. Click the **Design Status** button  on the 3D General Toolbar.

   A 3D Design Status Report file appears.

```
=======================================================================
[illegible]
=======================================================================
Job Name:                    C:\Candy\PCB\Candy.pcb

3D Design Status Report:     C:\Candy\PCB\LogFiles\DesignStatus3D_00.txt

04:59:34 PM Monday, October 05, 2015

=======================================================================
3D DESIGN STATUS
=======================================================================

Design Z Extents ......... -3.99 mm to 14.17 mm

Part models found ........ 20
   Part Number               3D Model
   1N5822                    STPS3150
   2N2222                    BAT54S
   74LS03                    TLC2652AMJB
   74LS12                    SSM2166S
   CAP                       0603G102J500S
   LED                       STPS3150
   LT1129-5                  AN6780S
   MAX2003                   MAX2003CPE
   NICD                      PTF5615R000TZEK
   NMOSH                     BAV99-V
   RES_0.294                 PTF5615R000TZEK
   RES_100K                  PTF5615R000TZEK
   RES_150                   PTF5615R000TZEK
   RES_20K                   PTF5615R000TZEK
   RES_28K                   PTF5615R000TZEK
   RES_9                     PTF5615R000TZEK
   SCONN9                    5745438-1
   THRMNTC                   PTF5615R000TZEK
   TIP115                    BAV99-V
   mux4x2                    TLC2652AMJB

Part models not found .... 3
   Part Number
   CAP, .1UF
   PCAP, 100UF
   PCAP, 3.3UF

Assemblies
   Filename: "C:Candy\PCB\Mech\#4_screw_resized_for_enclosure.stp",
   Scale: 1.00 Instance: Type=Assembly  "Screw4",
   Rotate(X, Y, Z in degrees): (0.00, -0.00, 0.00),
   Translate(X, Y, Z in th): (0.00, 0.00, 0.00) (saved with design)
   Filename: "C:\Candy\PCB\Mech\30mm_heatsink.stp",
   Scale: 1.00  Instance: Type=Assembly "HeatSink30",
   Rotate(X, Y, Z in degrees): (0.00, -0.00, 0.00),
   Translate(X, Y, Z in th): (807.41, 34.07, 0.00) (saved with design)
```

2. Review the contents of the report file to verify the 3D parameters.

Status Parameter	Description
Design Z Extents	Shows the maximum range in the Z-axis of all models in the design.

Status Parameter	Description
	Note: The beginning value of the range shows a negative number if models are mounted on the rear of the PCB. The Design Z Extents range tells you the overall thickness so you can verify that you have not exceeded the overall thickness (height) restriction for the assembled PCB.
Parts Models Found	Lists the 3D model assignments for each part in the design. Review this list to verify that you have the correct model assignment for each part.
Parts Models Not Found	Lists all parts that do not have a 3D model assignment. By default, these parts are represented by 2D extruded models. To assure the greatest accuracy possible in the 3D View, assign 3D models to all of the parts in this list when possible.
Assemblies	Lists the mechanical assemblies that you have imported into the design. Review this list to verify that you have imported all of the correct assemblies in the design.

3. (Optional) Make any changes necessary. For example, change model assignments, assign 3D models to parts that do not have model assignments, or change the Z-axis clearances to accommodate height restrictions for the design. After making the changes, generate a new 3D Design Status Report and review it again.

Related Topics

Assigning a 3D Model to a Component

Defining 3D Clearance Constraints

SailWind 3D Overview

Exporting the 3D Image to a 3D PDF File

Capture the current SailWind 3D window and export it as a 3D PDF file that allows the recipient to view, rotate, and query the objects in the design in 3D.

Procedure

1. In the Display Control dialog box, use the display options in the Components area to highlight specific parts or toggle their visibility if necessary.

   Tip
   Use the 3D rotate, pan, and zoom controls to orient the design in the SailWind 3D window the way you want it to appear in the exported image.

   ---

2. On the 3D General Toolbar, click the **Export** button.
3. In the Export dialog box, choose the directory path, enter the file name, and in the “Save as type:” dropdown list, choose 3D PDF (*.pdf*) as the file type.
4. Click **Save**.

Related Topics

SailWind 3D Object Manipulation

Creating 3D Cross Sections

3D Display Control Window

Exporting the 3D Image to a Graphics File Type

Capture the current SailWind 3D window and export it as an graphics image file for use in assembly drawings and other manufacturing or test documentation.

Procedure

1. In the Display Control dialog box, use the display options in the Components area of the to highlight specific parts or toggle their visibility if necessary.

   Tip
   Use the 3D rotate, pan, and zoom controls to orient the design in the SailWind 3D window the way you want it to appear in the exported image.

   ---

2. On the 3D General Toolbar, click the **Export** button.
3. In the Export dialog box, choose the directory path, enter the file name, and in the “Save as type:” dropdown list, choose the desired graphics image format (*.bmp, .gif, .jpg, .png or .tif*) for the exported file.
4. Click **Save**.

Related Topics

SailWind 3D Object Manipulation

Creating 3D Cross Sections

3D Display Control Window

Exporting the 3D Image as a Mechanical Model

Capture the content of the SailWind 3D window and export it as a mechanical model for use in other PCB designs or in 3D mechanical design applications. You can choose to export all mechanical models in the design or just selected models.

Note:
You can also choose to export the design as a PDF or image file but your export options are limited in such a case.

Procedure

1. In the Display Control dialog box, use the display options in the Components area of the to highlight specific parts or toggle their visibility if necessary.

   Tip
   Use the 3D rotate, pan, and zoom controls to orient the design in the SailWind 3D window the way you want it to appear in the exported image.

2. On the 3D General Toolbar, click the **Export** button.
3. In the Export dialog box, choose the directory path, type the file name, and in the “Type” dropdown list, choose SAT (*.sat*) or STEP (*.step*) as the file type.
4. In the “Mechanical Model Options” area, determine the type of design objects you want to export by selecting either “All mechanical models” (any mechanical models you have added plus the board with the components) or “Include with board export items only” (just the models with the “Include with board export” check box selected in the Properties dialog box).
5. Select the check box for each element type you want to export.
   - The Silkscreen and Soldermask check boxes apply to STEP file export only.
   - You can choose to export element types in separate files by selecting the “Export as Sheets” check box. For more information, see Export Dialog Box.
6. Click **Save**.

Results

SailWind Layout saves the specified mechanical models as one or more files, depending on your export selections. Note that if you export certain mechanical model elements as individual sheets (files), and later you clear one or more of the element type check boxes in the Export dialog box, SailWind Layout deletes the corresponding SAT or STEP file.

Related Topics

SailWind 3D Object Manipulation

Creating 3D Cross Sections

3D Display Control Window

Chapter 23
Exchanging Data with Mechanical Designers

Use MCAD Collaborator in SailWind Layout to transfer, record, manage, and preview change requests between the PCB design team and the mechanical design team.

Supported Design Objects for Data Exchange

MCAD Collaborator supports the processing and exchange of design objects using IDX Schema 2.0 or 3.0 formats.

The following tables list various design objects and whether or not they are supported for collaboration in IDX Schema 2.0 or IDX Schema 3.0.

Restriction:
You cannot collaborate by applying an IDX Schema 3.0 baseline file and then export an IDX Schema 2.0 proposal from that baseline.

Table 105. Objects Supported in IDX Schema 2.0

Design Object	IDX 2.0
Single Boards	
Cutouts	yes
Traces	
Export external layers data through baseline	yes
Pads	
Export external layers data through baseline	yes
Vias	

Table 105. Objects Supported in IDX Schema 2.0 (continued)

Design Object	IDX 2.0
Export thru-vias through baseline	yes
Component Pins/Pin Holes/Mounting Holes	
Export one location of component pin through baseline	yes

For IDX 3.0, objects are supported for different collaboration processes:

- **Export** — means you can export this data in IDX 3.0 format and save it as a baseline file.
- **Collaborate** — means you can import and export these objects. The Collaborator tracks them so you can send and receive updates (additions, deletions, changes) on the objects.

Table 106. Objects Supported in IDX Schema 3.0

Design Object	IDX 3.0	
	Export	Collaborate
Board Objects		
Board Outline	yes	yes
Board Thickness	yes	no
Board Stackup	yes	no
Through Mounting Holes	yes	yes
Cutouts (Contours)	yes	yes
Cavities	yes	yes
Copper		
Traces	yes	yes
Planes (Area Fills)	yes	yes
Via Pads	yes	no
Via Holes	yes	no
Blind/Buried Vias	yes	no
Fiducials	yes	no
Component Features		
Electrical Components	yes	yes

Table 106. Objects Supported in IDX Schema 3.0 (continued)

Design Object	IDX 3.0	
	Export	Collaborate
Mechanical Components	yes	yes
Component Pads	yes	no
Component Pin Holes	yes	no
Primary Pin	yes	no
Keepouts		
Placement Keepouts	yes	yes
Plane Keepouts	yes	yes
Trace Keepouts	yes	yes
Via Keepouts	yes	yes

Guidelines for Using Data Exchange Objects

Board Outlines/Board Thickness:

- MCAD Collaborator shows a different board thickness than the stackup thickness. The Collaborator does not include soldermask in the thickness calculation so it represents proper part mounting (parts actually mount on the copper). The stackup thickness, minus the thicknesses of the soldermask, should equal the board thickness shown in MCAD Collaborator.

Component Pins/Pin Holes/Mounting Holes:

- The IDX format contains information on the location of Pin One for a component.

  The Pin One coordinates are shown in the Properties section. The report of the coordinate location of Pin One is relative to the part origin, not the board origin. This is used to assist in aligning models.

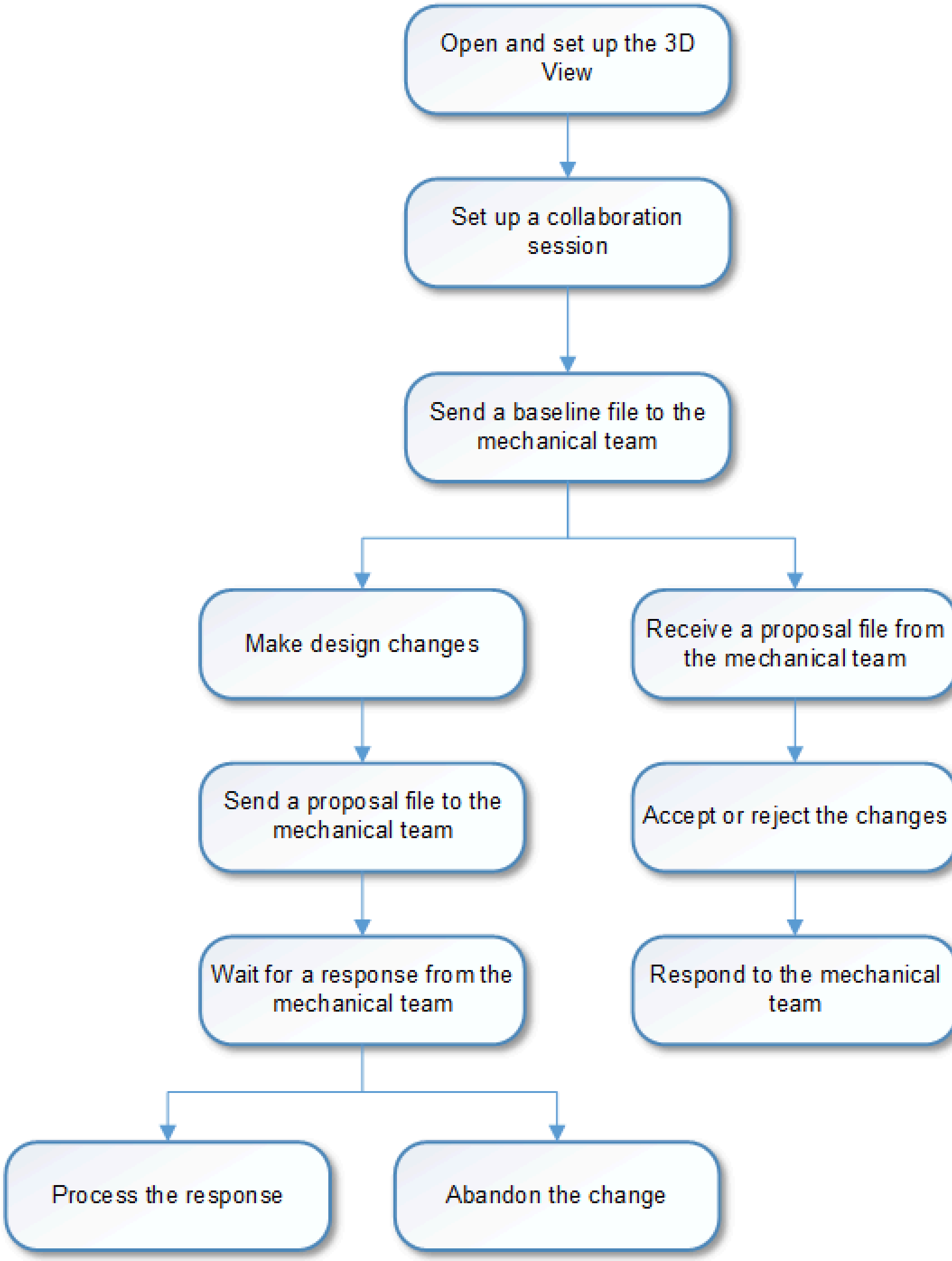
Collaboration Workflow Diagram

The typical collaboration workflow involves setting up a collaboration session, establishing a common design baseline, and then exchanging mechanical change requests.

You start the exchange process by opening a collaboration session and submitting a baseline data file to the mechanical design team (or they start the process by submitting their baseline file to you). The baseline and change data files are in IDX format. The baseline file updates the design with the changes; you do not have the option of rejecting the proposed changes in a baseline file. The side receiving a proposal has the option of either accepting or rejecting the proposed changes. Either side can then create and submit additional proposals by sharing new change data files.

You exchange the data files either by email or by storing them in a common directory accessible by both teams. Either team can initiate a change, and the change request is approved or rejected by the other team. MCAD Collaborator saves and tracks the status of each change request so you have a continuous record of what was done. Once a change request has been sent and received, you can close the collaboration session.

Figure 52. Typical Collaboration Workflow Diagram



Related Topics

Launching the Collaboration Tool

Launching the Collaboration Tool

Launch the collaboration tool from the SailWind 3D window within SailWind Layout.

Procedure

1. From within a design in SailWind Layout, open SailWind 3D (**View > SailWind 3D**).

   SailWind 3D opens and displays the 3D General toolbar.

2. Open the MCAD Collaborator by clicking on the **MCAD Collaborator** button  on the toolbar.

Results

You are ready to set up a collaboration session.

Related Topics

Setting Up a Collaboration Session

Setting Up a Collaboration Session

Before you transmit changes, set up a collaboration session and define how you want to exchange data with the mechanical design team.

Restrictions and Limitations

- You must name (or rename) all components and part names so they do not contain spaces or non-alphanumeric characters. If you send a data file to the mechanical design team that has a space or a non-alphanumeric character in a name, those characters are automatically replaced with underscores. These automatic changes in the names are then marked as modifications even though no design change has actually occurred.

Procedure

1. In MCAD Collaborator, expand the Communication section.

2. In the Write Options section, choose the desired output and Email notification options from the dropdown lists.

3. Choose one of the following from the Data Transfer dropdown list to define how you want to exchange the baseline and change request files:

If you want to...	Do the following...
Exchange data files by storing them in a shared directory	1. Choose File from the dropdown list. 2. In the Data Path text box, enter the path name for the directory where you want to store the data exchange files (or browse to locate the directory).

If you want to...	Do the following...
Exchange data files as email attachments	1. Choose Email from the dropdown list. 2. Choose either Outlook or SMTP from the Email Server dropdown list. 3. Enter the appropriate contact information in the Receiver Info and Sender Info text boxes.

4. Click **Filter objects** to open the Object Filter dialog box.
5. In the Component Options section, do the following to specify how to generate component data:
   - Check “Write Mounting Holes” to generate mounting hole data as part of the decal.
   - Check “Write Pin Holes” to generate pin hole data as part of the decal.
   - Under Replace 0 Height Components With, check “Top or Bottom Copper Thickness” or enter a custom height.

   Note:
   The collaboration exchange may not transmit zero height values correctly. By defining a default height for zero height components, you avoid potential errors that may appear on the mechanical design side of the exchange.

6. In the Board Options section, check Write Cutouts (Contours and Cavities) to include cutouts.
7. In the Collaboration Objects Filter list, filter what is displayed in the Collaboration Data list by checking the desired object types. (Unchecked object types will not appear in the Collaboration Data list.)
8. Click **OK** to save the Object Filter settings and close the dialog box.

   Note:
   If you made changes to the Collaboration Objects Filter list, the **Update Tree Data** button in MCAD Collaborator turns yellow. Click **Update Tree Data** to update the Collaboration Data list.

9. (Optional) In the Scheme section, click **Save** and enter a scheme name to save the settings you define so you can reuse them later in other designs.

   The Communication and Object Filter settings are saved as a scheme in the directory you specify.

Results

The Collaboration Data list displays the design objects you checked in the Objects Filter list.

The collaboration settings are stored with your design project.

You are ready to send a baseline request.

Related Topics

Launching the Collaboration Tool

Sending a Baseline Request

Receiving a Baseline Request

Creating and Sending a Change Request

Reviewing and Applying a Change Request

Sending a Baseline Request

Start a collaboration session by creating and sending a baseline request to the mechanical design team.

The baseline request establishes a common database that is shared between the PCB design team and the mechanical design team so that both are synchronized. Future change requests are referenced against the baseline data.

Prerequisites

- A collaboration session has been set up that defines how the request information is exchanged. See “Setting Up a Collaboration Session” on page 565.

Procedure

1. In MCAD Collaborator, choose Baseline from the File Type Selector dropdown list.
2. Click **Object Filter** to open the Object Filter dialog box.

    a. Check the Component Options you want to apply.

    a. In the Collaboration Objects Filter list, filter what is displayed in the Collaboration Data list by checking the desired object types. (Unchecked object types will not appear in the Collaboration Data list.)

    a. (Optional) In the Board Options section, uncheck “Write Cutouts (Contours and Cavities)” to send only the Board Outline without cutout and contour data.

    a. Click **OK** to apply the settings and close the Object Filter dialog box.

    Note:
    If you made changes to the Collaboration Objects Filter list, the **Update Tree Data** button in MCAD Collaborator turns yellow. Click **Update Tree Data** to update the Collaboration Data list with the objects you checked in the Object Filter dialog box.

    The Properties section for the main design item shows the File type as “Baseline” and the Status as “To be sent”.

    The status bar reads “Ready” and is green.

3. In the Collaboration Data list, check the objects you want to include in the baseline; uncheck the objects you want to exclude.

4. Click **Send**.

   MCAD Collaborator generates the specified PCB data.

Results

The baseline request file is saved and appears in the History section.

The status bar reads “Baseline sent...Ready to send or receive more changes” and is green.

If you enabled email notification, the mechanical design team receives your baseline request.

Related Topics

Receiving a Baseline Request

Creating and Sending a Change Request

Receiving a Baseline Request

If you receive a baseline request from the mechanical design team, you must accept the entire baseline database as it is submitted to you so you can proceed with the collaboration session.

The baseline establishes a common database that is shared between the PCB design team and the mechanical design team so that both are synchronized. Future change requests are referenced against the baseline data.

Note:
The MCAD Collaborator monitors the shared directory where the baseline files are stored and updates the Files section automatically whenever a new change request file (baseline or proposal) is placed in the directory.

Procedure

1. In the Files section of MCAD Collaborator, select the baseline file.

   The Collaboration Data section shows a list of the design objects contained in the baseline file.

2. (Optional) For detailed information about a design object, select it and review the properties for that object in the Properties section.

3. Click **Apply**.

   Note:
   If a baseline has a DRC error, a warning message appears. You can proceed and apply items that partially succeed instead of failing the entire transaction. If you proceed, the MCAD and ECAD baseline files become out-of-sync.

Results

Your design database is updated with all of the changes contained in the baseline file. Error messages and processing reports appear in the Message Window.

The baseline file moves to the History section.

The status bar reads: "Baseline applied. Ready to send or receive more changes" and is green.

Related Topics

Sending a Baseline Request

Creating and Sending a Change Request

Reviewing and Applying a Change Request

Reviewing the Change History

Creating and Sending a Change Request

You can make changes to the placement and objects in your design then send a change request to the mechanical design team that describes the proposed changes.

Prerequisites

- A baseline file has been exchanged and accepted.

Procedure

1. In MCAD Collaborator, choose Proposal from the Proposal Type dropdown list.
2. Click **Update tree data** .

   Note:
   Any time you make a change in the PCB design, the **Update tree data** button turns yellow to indicate that the data in MCAD Collaborator is out-of-sync and needs to be udpated.

   MCAD Collaborator generates the revised PCB data with any changes you have made and shows the changes in the Collaboration Data list.

   The Properties section shows the File type as "Proposal", the Status as "To be sent", and the File Name as "proposal_nn".
3. (Optional) Do the following to limit the Collaboration Data list to only the types of objects you want to display:

   a. Click **Filter objects** to open the Object Filter dialog box.

   b. In the Collaboration Objects Filter list, check only the types of objects you want to display in the Collaboration Data list, then click **OK**.

      The **Update tree data** button turns yellow to indicate that the list needs updating.

   c. Click **Update tree data** to update the Collaboration Data list and show only the object types you selected in the Collaboration Objects Filter list.
4. In the Collaboration Data list, check the objects you want to include in the proposal; uncheck the objects you want to exclude.

5. Click **Send**.

> **Tip**
> If you want to abandon the proposal after sending it, click **Abandon**. This cancels the proposal file and changes its Status to "Abandoned" in the History section.

Results

The proposal file is submitted to the mechanical design team for their approval or rejection. Error messages and processing reports appear in the Message Window.

The proposal file appears in the History section.

The status bar shows: "Proposal sent. Awaiting response" and is yellow.

You can continue to make additional design changes or process other change requests sent to you from the mechanical design team. (You cannot send a new proposal until you have processed previous proposals and any responses you receive from the mechanical design team.)

Related Topics

Reviewing and Applying a Change Request

Sending a Baseline Request

Reviewing and Applying a Change Request

After you receive a change request (proposal) from the mechanical design team, preview the proposed changes and determine whether to accept or reject them.

The 3D View shows what has been added, modified, or deleted by highlighting the proposed changes.

Prerequisites

- A baseline file has been exchanged and accepted.

Procedure

1. In the Files section of MCAD Collaborator, select the proposal file you want to preview and validate.

   The Collaboration Data section shows a list of the design objects contained in the proposal.

2. Click **Highlight/Unhighlight**  to activate highlighting and enable animation of the selected changes.

3. In the Collaboration Data list, select the object you want to preview.

   The 3D View shows an animated playback of the change you select.

Tip
For more detailed information about the selected object, review the properties for that object in the Properties section. This shows a “From” / “To” comparison of the changes made to the object.

4. Based on what you see in the 3D View, determine if the change you previewed is acceptable.

If you want to...	Do the following...
Accept the change	Check the change item.
Reject the change	Uncheck the change item.

5. Repeat Steps 3 - 4 for each object in the Collaboration Data list.
6. Click **Apply**.

   Your design database is updated and the items you checked are changed in your design.

   Note:
   If a change creates a DRC error, you are notified and the change is not applied. For example, the change would create an "unresolvable metal error". In this case, you would need to send a "Reject" response.

   The proposal file moves to the History section.

   The status bar shows: “Proposal Applied. Ready to send a response” and is green.
7. (Optional) Repeat Step 3, as needed, to review the changes that have been made and verify that the design has been updated correctly.
8. Click **Send**.

   A *response_nn.idx* file is sent.

Results

If you are collaborating by means of email notification, the mechanical design team receives a response telling them that you have processed the proposal. The email contains a file (*response_nn.idx*) that the mechanical team loads to their design environment to see the changes.

The status bar shows: “Transaction complete. Ready to send or receive more changes” and is green.

Related Topics

Controlling the Display of Different States

Controlling the Display of Different States

Display different change items in a proposal based on their states to quickly determine the items to accept.

You can control which objects appear in the Collaboration Data list of MCAD Collaborator according to their status within a proposal file. You can also highlight a selected object in the SailWind 3D window to locate it and analyze its impact.

For example, you can choose to view only those objects in a change request that are added, and not view objects that are deleted or changed. This helps you evaluate a proposed change request more efficiently by focusing on only one state at a time.

Procedure

1. In the Files section of MCAD Collaborator, select the proposal file you want to review.

   The Collaboration Data section shows a list of the objects contained in the proposal file.

   > **Tip**
   > You can manage the files in the Files section in the following ways:
   > - Select a file and choose **Clear** from the popup menu to remove it from the Files list.
   > - Choose **Clear All** from the popup menu to remove all of the files from the Files list.
   > - Choose **Refresh** from the popup menu to update the Files list.

2. In the display toolbar, click the appropriate buttons to toggle the display of objects in the Collaboration Data list:

If you want to...	Do the following...
Show (or hide) all of the objects that have been added.	Click .
Show (or hide) all of the objects that have been deleted.	Click .
Show (or hide) all of the objects that have been modified.	Click .

3. (Optional) Click , then select an object in the Collaboration Data list that you want to preview in the 3D View.

   The selected object is highlighted in the 3D View to help you locate it more quickly in the design.

Related Topics

Reviewing the Change History

Reviewing and Applying a Change Request

Sending a Baseline Request

Reviewing the Change History

You can review the history of change requests for the current collaboration session and look at previous proposals to recall what has been changed.

Procedure

1. In the History section of MCAD Collaborator, select the proposal you want to review.

   The Collaboration Data section updates to show a list of the objects contained in the selected proposal.

2. In the Collaboration Data list, select the object you want to review.

   The 3D View shows an animated playback of the change you select.

3. Continue to review other individual change items in the proposal, as needed, or select a different proposal and review those changes.

4. (Optional) Click **Clear** in the History section to clear the entire change history and reset the current collaboration session.

   The change request files are moved to the Files section and the History section becomes empty. The collaboration state is cleared as if no exchange of data has occurred.

   CAUTION:
   You cannot undo this action.

Related Topics

Reviewing and Applying a Change Request

Controlling the Display of Different States

Managing the Change Files

You can manage the list of files to work more efficiently with large lists of baseline and change request files.

Procedure

1. In the Files area of the MCAD Collaborator, select a file.

   Tip
   To make sure the Files area is updated and current, choose **Refresh** from the popup menu.

2. Do any of the following:

If you want to...	Choose the following from the popup menu...
Remove a single file.	**Clear**
Remove all of the files.	**Clear All**

Chapter 24
Virtual Pins

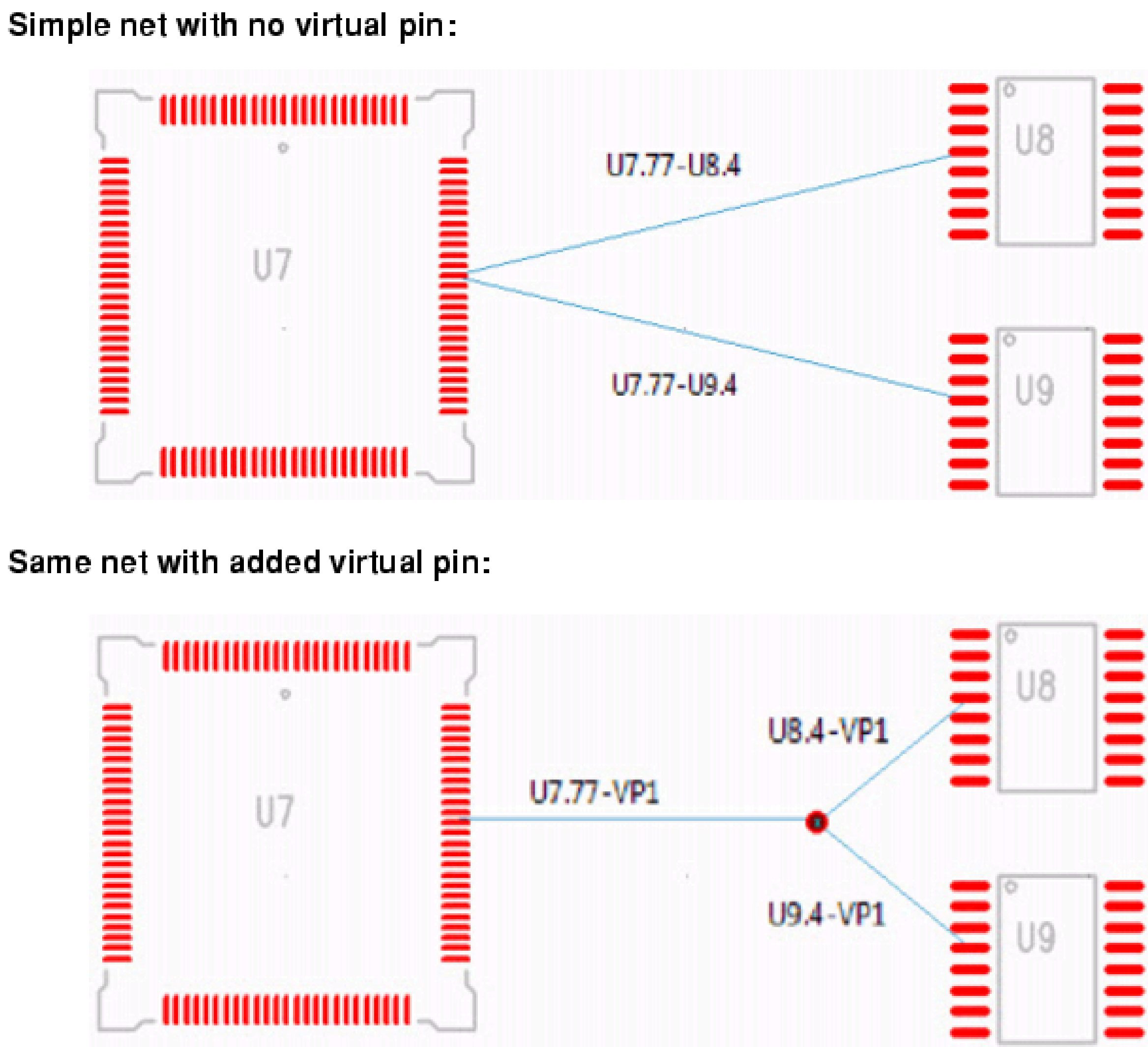
Read the topics that follow to learn more about the setup and management of virtual pins.

The Virtual Pin

A virtual pin is like a component pin that you can add to a net. It becomes a start point and/or end point of one or more new pin pairs. This gives you increased (sub-pin-pair) control over the net. For example, you could use a virtual pin to create distinct matched length pin pairs out of the net's branches.

Tip
You can also add virtual pins in SailWind Router.

A virtual pin uses the pad stack of a via. The pad stack can be through-hole or partial, or it can be a single-layer pad. Virtual pins are added to nets. When you add a virtual pin to a net, the total number of pin pairs in the net increases by 1, as shown in this example:



You can assign rules to pin pairs ending on virtual pins in the same way as to pin pairs ending on component pins.

In the display, virtual pins are identified with a marker that uses the color of the board outline.

Figure 53. Virtual Pin Marker

To work with virtual pins, you need the Advanced Rules license.

Virtual Pin Setup

Virtual pins use via types defined in the Pad Stacks Properties dialog box.

If you want to create unique via types to use for virtual pins in your design (a non-drilled single-layer pad, for instance) follow the procedures in "Via Setup".

Adding a Virtual Pin to a Net

You can add a virtual pin to any net—unrouted, partially-routed, or wholly routed.

When you add a virtual pin to a net, it is added as follows:

- If the virtual pin you are adding is the first virtual pin in the net, and the net has no pin pair rules and no routing, and has Minimized topology, pin pairs are created between the new virtual pin and each component pin in the net in a "starburst" pattern, with the virtual pin in the center. This is the only case in which a starburst pattern is created.

  Note that if a virtual pin has been previously added to the net and deleted, the net topology will have been set to Protected; so the virtual pin you are adding, even though it appears to be the first virtual pin in the net, will be connected only to the nearest component pin, not in a starburst pattern. You must reset the net topology to Minimized to create the starburst pattern with a central virtual pin.

- If the virtual pin you are adding is the first virtual pin in the net, and the net has pin pair rules, or routing, or a topology other than Minimized, the new virtual pin is connected to the nearest component pin only. No starburst pattern is created.

- If the virtual pin you are adding is not the first virtual pin in the net, it is connected to the nearest virtual pin only. No starburst pattern is created.

Restriction:
You cannot create a starburst pattern by placing a virtual pin on a trace of a routed net.

Restrictions and Limitations

You cannot add a virtual pin to a plane net.

Procedure

1. In the design area, select a net.

2. Right-click and click the **Add Virtual Pin** popup menu item. The new virtual pin appears on the pointer.

3. Position the virtual pin, and click to place it. To stop adding virtual pins, right-click and click the **Cancel** popup menu item.

Results

- If you have created a starburst pattern, the new pin pairs are created, and the net topology is set to Protected. If you set your filter to Select Pin Pairs, you can select each connection. The status bar displays the connection to the virtual pin. Instead of a component and pin number (for example U2.8), it displays the instance of the virtual pin (for example VP2). The same format (VP<number>) is used in pin pair names, for example VP7—U2.1.

  For information on setting design rules for the new pin pairs, see "Pin Pair Design Rules."

- If you have created a non-starburst configuration, or if you have created a starburst configuration and want to reconfigure your pin pairs, you will need to reschedule the net in SailWind Router. See "Rescheduling Nets" in the *SailWind Router Guide* for instructions.

Moving a Virtual Pin

Move virtual pins just as you would move a via.

Restrictions and Limitations

You cannot move more than one virtual pin at a time.

Procedure

1. In the design area with nothing selected, right-click and click the **Select Virtual Pins** popup menu item.
2. Select a virtual pin to move, right-click and click the **Move** popup menu item.
3. Move the virtual pin to a new position. Virtual pins adhere to the via grid for initial placement but adhere to the design grid when moving them.

Deleting a Virtual Pin

You can remove or delete a virtual pin from a routed or unrouted connection.

Procedure

1. In the design area with nothing selected, right-click and click the **Select Virtual Pins** popup menu item.
2. Select the virtual pin, right-click, and click the **Delete** popup menu item.

Results

- If the deleted virtual pin has only one connection (routed or unrouted), the virtual pin and its connection are deleted.
- If the deleted virtual pin has two or more connections, all unrouted, the virtual pin is deleted, and new connections are made between the component pins and virtual pins it was connected to.

- If the deleted virtual pin has two or more connections, some or all of which are routed:
    - The virtual pin and all its attached pin pairs are deleted.
    - Routing (traces and vias) of the deleted pin pairs remains, and is assigned new pin pairs.
    - If traces from different layers were attached to the deleted virtual pin, one or more zero-length unroutes will be created between junctions on different layers.
    - Unroutes might be connected to junctions that replace the virtual pin.

Converting a Virtual Pin to a Via

Select a virtual pin in the design and right-click to convert it to a via.

Procedure

1. In the design area with nothing selected, right-click and click the **Select Virtual Pins** popup menu item.
2. Select the virtual pin, right-click and click the **Convert to Via** popup menu item.

Results

The virtual pin is converted to a via, traces and unroutes attached to the converted virtual pin remain attached to the new via, and new pin pairs passing through the new via are assigned.

Gluing a Virtual Pin

Select a virtual pin in the design and enable the Glued property.

Procedure

1. In the design area with nothing selected, right-click and click the **Select Virtual Pins** popup menu item.
2. Select the virtual pin, right-click and click the **Properties** popup menu item.
3. In the Virtual Pin Properties Dialog Box, select the Glued check box.
4. Click **OK**.

Changing a Virtual Pin’s Via Type

If desired, you can change the via type that a virtual pin uses by right-clicking on it.

Procedure

1. Select the virtual pin, right-click and click the **Properties** popup menu item.
2. In the Virtual Pin Properties Dialog Box, select the new via type from the Via Name list.
3. Click **OK**.

Support for Export to Other Formats

Virtual pins are recognized and exported when a design is exported to the PADS-ASCII format (of PADS 9.5 or later), or to a DXF file. When a design is exported to any other format, including PADS-ASCII formats of releases earlier than PADS 9.5, the export process converts virtual pins to vias.

For more information, see "File Export Formats".

ECO Import Support

ECO ignores operations on virtual pins, but in cases of removal of components or connections from nets containing virtual pins, importing the ECO file affects the nets.

Nets are affected as follows when you import the ECO file:

- If, after applying all ECO commands that remove components, connections, or nets, a net is left with less than two component pins, the net and all its virtual pins are deleted.
- If a net with virtual pins is left with two or more component pins, its virtual pins remain, and new pin pairs are created.
- If a net is split at a connection that has a virtual pin, the virtual pin remains connected to one of the new nets.

Reuses Support

Physical design reuses do not support virtual pins.

Chapter 25
Working With Labels

Read the topics that follow to learn more about the methods used to create, modify, and control the visibility of labels. These sections include a selection of graphical examples to guide you in setting up your label properties.

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 on page 1813 around the label itself.

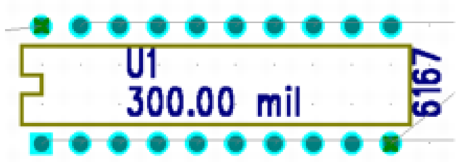
Label Defaults

If you create a label, but do not provide display information in the decal or in the component, SailWind Layout uses default visibility properties.

The default position for the first label is at the decal origin with no orientation, and it uses the default height and width. If a label already exists in the first position, the second position is used. The default position for the second label is under the first label position.

The figure below shows how a new height attribute label is placed in the second default position. Note that the second position is based on the first position, not where other labels are placed.

Figure 54. New Height Attribute Label (Second Default Position)



The figure below shows how a new height attribute label is placed in the third default position because the first and second positions are already filled.

Figure 55. New Height Attribute Label (Third Default Position)

Related Topics

Labels

Creating Attribute Labels in the PCB Decal Editor

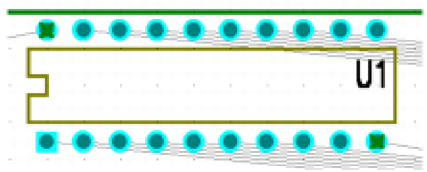
Justification Examples

You can justify free text and attribute, reference designator, and part type labels. You can set justification options when you create the label or text.

For more information, see Justifying a Label.

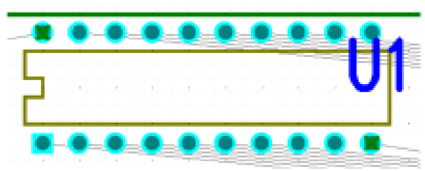
In the figure below, a label appears within a part outline. The label uses the default justification, which is Left, Down; meaning that the label was placed by its lower left corner.

Figure 56. Label Within a Part Outline



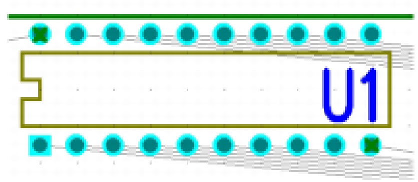
The figure below shows what happens when you change the Height and Width of the label. The label now overlaps the part outline.

Figure 57. Label After Changing the Height and Width (with Overlap)



The figure below shows how you can change the Height and Width of the label and prevent the label from overlapping outlines. By changing the justification to Top, Right you change the location by which the label is placed, preventing overlap.

Figure 58. Label After Changing the Height and Width (no Overlap)



Related Topics

Labels

Creating Attribute Labels in the PCB Decal Editor

Right Reading Examples

You can set the left or right reading status for free text and attribute, reference designator, and part type labels using the Label Properties dialog box. You can also use the Properties dialog box for any of the objects.

For more information, see Adding a New Part Label or the appropriate Properties object topic.

The figures below show different reading orientations.

Figure 59. Normal Rotation

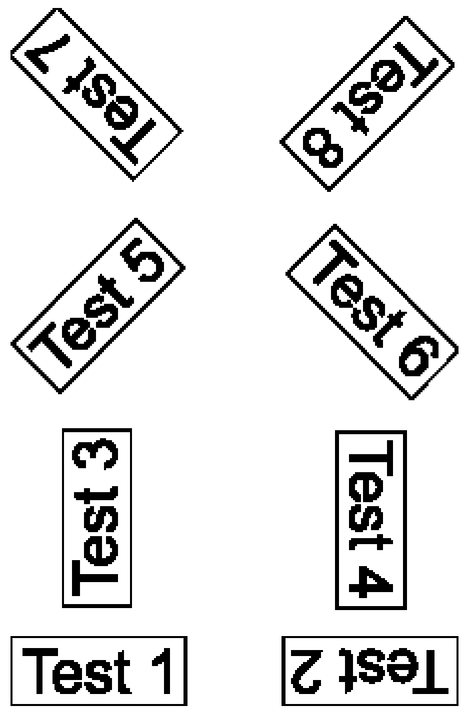


Figure 60. Mirrored

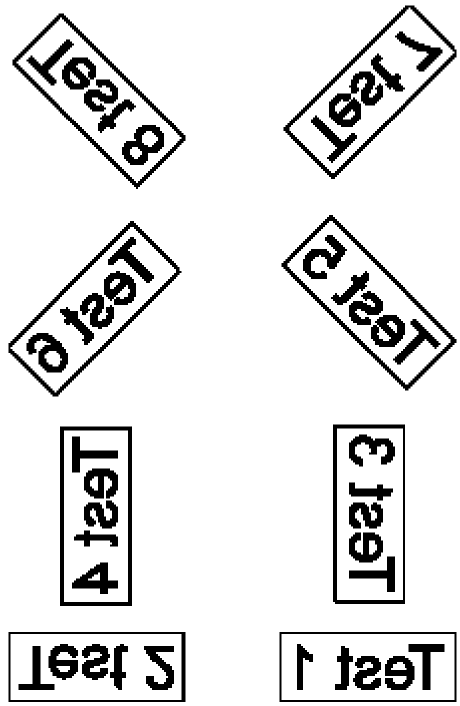


Figure 61. Right Reading - Orthogonal

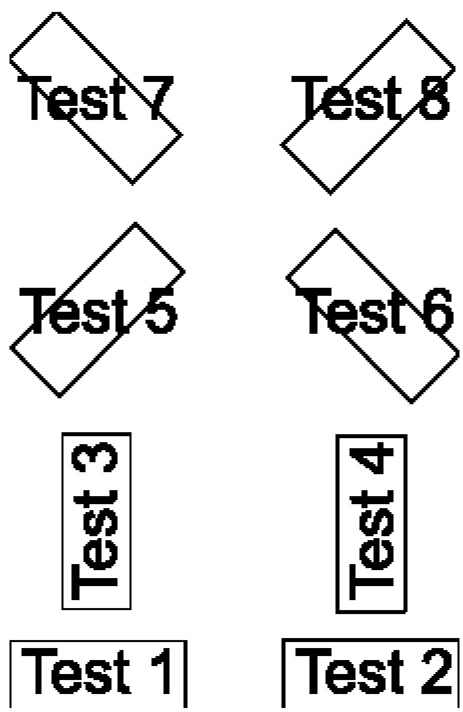


Figure 62. Right Reading - Orthogonal, Mirrored

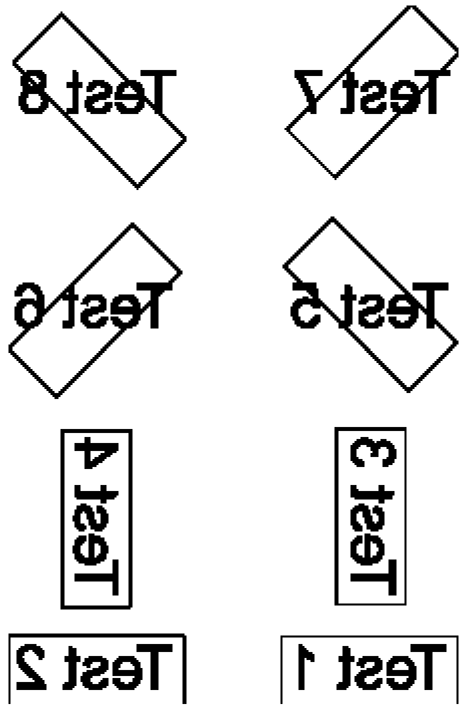


Figure 63. Right Reading - Angled

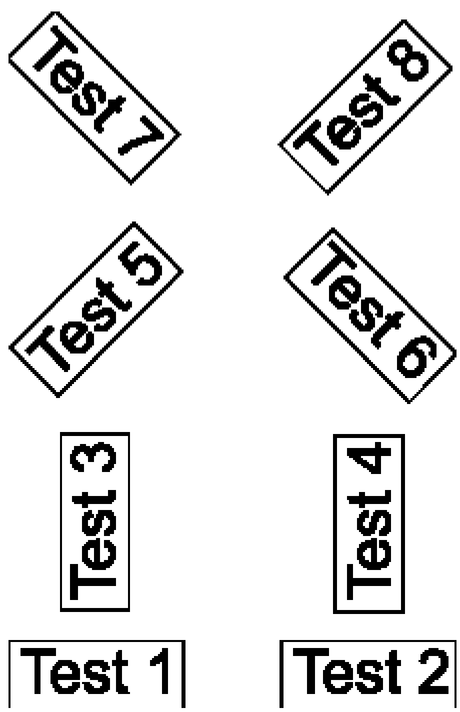
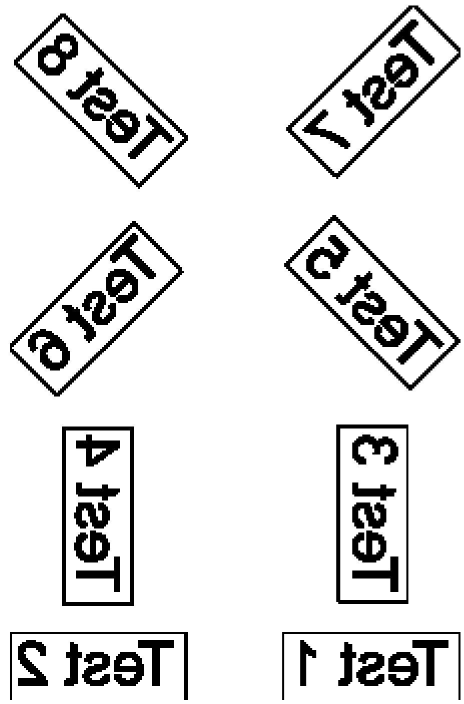


Figure 64. Right Reading - Angled, Mirrored



Related Topics

Labels

Creating Attribute Labels in the PCB Decal Editor

Managing Reference Designators

SailWind Layout handles reference designators in the same manner as it handles attribute labels, providing flexibility in visibility and placement.

You have many options when creating and using reference designators in your design:

- Create a reference designator label and set it up for silkscreen layers. For more information, see the Moving Reference Designators to the Silkscreen Layer on page 810topic.
- Create a second reference designator label and set it up for assembly drawing layers. For more information, see the Generating a Second Set of Reference Designators for Assembly Drawings on page 808 topic.
- Hide reference designators. For more information, see the Disabling Reference Designators of Selected Components topic.

Related Topics

Labels

Creating Attribute Labels in the PCB Decal Editor

Labels in the PCB Decal Editor

Work with labels in the PCB Decal Editor exactly as you do 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. You can, for example, 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.

For more information, see the Creating Placeholder Attribute Labels topic.

When you create labels, they may not be visible. Turn on label visibility in the Display Colors Setup dialog box. On the Display Colors Setup dialog box, you can set the color for reference designators, part type, and attribute labels.

If you created a placeholder label for an attribute that exists in the Attribute Dictionary, the label is associated with the existing attribute, but it is not visible. The attribute is not, however, assigned to the part using the decal. You must manually assign it.

Related Topics

Labels

Creating Attribute Labels in the PCB Decal Editor

Adding a New Part Label

Add labels of attributes from components or jumpers.

Restrictions and Limitations

When working with labels, some restrictions apply:

- Reference designator is the only label available for use when you are creating labels for jumpers.
- If you do not set visibility information, default positions are used. For more information, see “Label Defaults.”
- Unlike free text, when you add a label, there is no invisible bounding rectangle on page 1813 around the label itself.

Procedure

1. Select a component or jumper, right-click, and then click the **Add New Label** popup menu item.
2. In the Add New Part Label Dialog Box, in the Attribute list, select the attribute you want.

Restriction:
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 for Component or Value for Jumper box lists the value of the selected attribute. Accept this value, or type a new one.

Restriction:
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.

Tip
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 SailWind 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. Make any additional settings required for the label.

6. Click **OK**.

Selecting a Label

Set the selection filter and select a label.

Procedure

1. Use one of the following methods to set the Selection Filter:

   - With nothing selected, right-click, and then click the **Select Documentation** popup menu item.

   - Click the **Edit > Filter** menu item. In the Selection Filter dialog box on page 114, on the **Object** tab, select the Labels check box. Click **Close**.

2. Select the label.

Deleting a Label

If desired, you can delete or remove a label from your design.

Procedure

1. Select the label to delete. Use the selection filter if you need help selecting the label.
2. Press the Delete key, or click the **Edit > Delete** menu item.

Justifying a Label

You can justify free text and attribute, reference designator, or part type labels. You can set justification options when you create a label. You can also justify text and labels after you create them.

Procedure

1. Select the label or free text.
2. Choose one of one of these methods to justify the label:
   - Right-click and click the **Justify Horiz** popup menu item. Then click **Left**, **Center**, or **Right** to set the horizontal Justification. Right-click again and click the **Justify Vert** popup menu item. Then click **Up**, **Center**, or **Down** to set the vertical justification.
   - Right-click and click the **Properties** popup menu item to open the Properties dialog box for the object and set the appropriate justification setting.

Note:
You can also set justification options when you spin or move labels and text. For more information, see "Moving Text and Labels and Rotating an Object."

Related Topics

Adding a New Part Label

Component Properties Dialog Box

Labels in the PCB Decal Editor

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 the selected labels.

Procedure

1. Select a part label, right-click, and then click the **Properties** popup menu item.
2. In the Part Label Properties Dialog Box, in the Attribute list, select the attribute you want.

Restriction:
If you are creating or modifying 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.

Restriction:
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.

Tip
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 SailWind 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. Make additional edits to text settings as necessary.

6. Click **OK**.

Modifying Labels using the Component Properties Dialog Box

You can modify labels from the Component Properties dialog box.

Procedure

1. Select a component, right-click, and then click the **Properties** popup menu item.

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.

4. For information on deleting labels, see "Deleting a Label."

Chapter 26
Reusing Designs or Parts of Designs

Read the topics that follow to learn more about the creation and management of physical design reuses.

Physical Design Reuse

The most common use of the reuse feature is to clone the placement and layout of multi-channel design elements. To describe the feature more generally, if you have portions of your design that have a strict relationship with each other, you can group those elements of your design to lock them together to re-use them, or to clone them. The physical design reuse includes properties and attributes of these objects, such as test point status. You can create a reuse of any design elements except for the board outline and board cutouts.

Reuse a circuit in other designs to effectively reuse proven and tested elements and shorten the design cycle of new designs.

A design may have a common circuit that you want to repeat a number of times in the same design. Place the components and interconnect traces for the common circuit, save that as a physical design reuse, and then clone it using other existing design elements.

Each time you create a physical design reuse, it is assigned a reuse type, a unique name that describes it. The reuse type is similar to the library part type. Each physical design reuse also has an origin. The origin is visible only when you reset the origin of the reuse. You cannot display the reuse name and reuse type in the design.

Here are the three ways a reuse can be used:

- Like a union on page 504, the elements of a reuse are glued together and are moved as a group. For example, if you have not finalized placement, but want to start routing, you could route associated elements and then move them later if needed.
- Save the reuse to a file, and in any design, use the reuse as a template to create a clone of the layout using existing design elements. For example, if you have a redundant or multi-channel area in your design, you could place and route the interconnects of one channel, and then use it as a template to automatically place and route the other identical channels. This accelerates placement and routing of the design.
- Save the reuse to a file, and in ECO mode, add the reuse elements to your design. For example, if you have no schematic to supply the parts and netlist, you can add the reuse into the design.

Elements in a Physical Design Reuse

You can include certain elements in a physical design reuse. The physical design reuse includes properties and attributes of these elements, such as test point status. Each element is assigned a unique ID in the reuse. You must completely select an item to include it in a reuse. SailWind Layout does not include partially selected items in a reuse.

Physical design reuse elements include:

- Component Elements
- Routing Objects
- Drafting Objects
- Unions and Arrays
- Net-Based Design Rules

The physical design reuse opens as a new design. The start-up file information and design rules saved with the physical design reuse load as well. When you load a physical design reuse, single pin components are added to the ends of the single pin nets to preserve the net objects. These components have no drill size, pad size, outline, and so on. They are glued by default, non-ECO registered, and located at the 0,0 origin. The component name includes the entire netname, or reasonable portions of it, to easily identify the component to net associations.

Component Elements

The table below shows component elements eligible for inclusion in a physical design reuse.

Table 107. Component Elements Used in Physical Design Reuse

Eligible Element	Description
Reference Designator	Includes the reference designator of a component.
Part Type Definition	Includes all details of the part type definitions assigned to components.

Table 107. Component Elements Used in Physical Design Reuse (continued)

Eligible Element	Description
Location	Includes the location of component elements relative to the origin of the physical design reuse; including X,Y coordinates, rotation, and the side of the design on which the component is mounted.
Reference Designator and Type Label Properties	Includes each component element's reference designator and part type labels.
PCB Decal and Alternates	Includes the PCB decal and alternates assigned to each component element.
Component Attributes	Includes all attributes associated with each component element as well as their labels.
Other Properties	Includes other details of the component and its pins, including the test point status, whether the test point is Top Access, thermal eligibility, and thermal and antipad pad stack for each pin.

Routing Objects

The path of route segments, or the route pattern, is stored with width, layer, and netname information. For a physical design reuse, it is assumed that a net consists of one or more pin pairs. Nets are not included directly in a reuse. Instead, the physical properties of the net such as traces, vias, jumpers, coppers, and signal names are included, so the net is preserved.

Tip
Because only net properties are included in a physical design reuse, unrouted pin pairs are not included. The signal name of a pin pair, however, is included and is assigned to pins of the pin pair.

The table below shows route elements you can include in a physical design reuse.

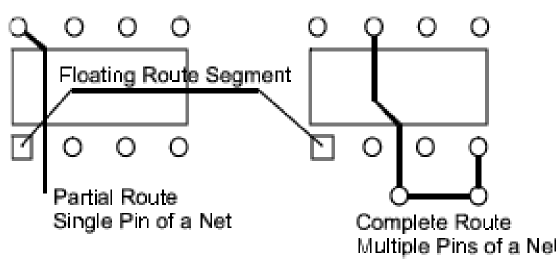
Table 108. Route Elements Used in Physical Design Reuse

Route Element	Description
Complete Routes and Route Paths	Contiguous routed or unrouted segments of a pin pair between two component pins. This includes all trace segments, vias, and jumpers.
Partial Routes	Incomplete, but contiguous, route segments of a pin pair that start or terminate on one component pin. The unrouted segments are also saved in the physical design reuse.
Floating Segments	Segments of a pin pair that neither start nor end on a component pin. The path of the segments is stored in the physical design reuse with width, layer, and netname attributes. This may require generating new pin pairs when the physical design reuse is added to the design.
Multiple Pins of a Net	Complete or partial route of the same net, with more than one component pin of the net included in the selection. When included in the physical design reuse, pin pairs may be created

Table 108. Route Elements Used in Physical Design Reuse (continued)

Route Element	Description
	between the pins included in the physical design reuse in order to associate and preserve the route paths.
Single Pin of a Net	Partial route of the same net with only one component pin of the net included in the selection. When included in the physical design reuse, a pin pair is created. If you cannot create the pin pair, the route is not included in the physical design reuse.
Vias	Including any via in the route with the via name as an attribute.
Via Definitions	Including the via definition (pad stack) in the physical design reuse for each element added to the physical design reuse.
Via Attributes	Including test point status, Top Access status for test point, thermal status, and user-defined attributes.
Teardrop Settings	Including teardrop status of route end points at vias and pads. Teardrop display is maintained according to the **Tools > Options** menu item, **Routing** tab, Generate Teardrops setting.
Jumper Component and Pins	Including a jumper component with its reference designator, reference designator labels, pad stack definition of its vias (pins), and attributes.

Figure 65. Route Elements



Drafting Objects

The definition of the path of the line item, or shape, with width, layer, and netname stored in the physical design reuse. This includes both open and closed shapes.

The table below shows line elements eligible for inclusion in a physical design reuse.

Table 109. Line Elements Used in Physical Design Reuse

Line Element	Description
2D Lines	Including the shapes of all drawing pieces, layer for all drawing pieces, and a list of all text items associated with the 2D line.
Dimensions	Including the X,Y location, layer, and individual dimension element settings, like text and arrow extensions.

Table 109. Line Elements Used in Physical Design Reuse (continued)

Line Element	Description
Text Items	Including the text string, X,Y location, orientation, layer, height and width, mirror flag, and justification settings.
Complete Copper Polygons	Size and shape of polygons, stored with line width and netname.
Complete Copper Plane Polygons	Shape of copper plane polygons, stored as corner coordinates with attributes for polyline width and netname. Hatch outlines are not included in a physical design reuse but pour outlines are included. Editing to hatch outline is lost when you add a physical design reuse to a design.
Cutouts and Keepouts	Any completely selected keepout, copper plane cutout, and copper cutout. Board cutouts are not stored in a physical design reuse.

Restriction:
Board outlines, board cutouts, hatch outlines, hatch voids, via thermals, and pad thermals are not included in the physical design reuse.

Unions and Arrays

Any completely selected union or component array is stored in a physical design reuse. Clusters are not included in a physical design reuse.

Net-Based Design Rules

Design rules for nets and net classes, default rules, and net-based pin pair and group rules are included in the physical design reuse. No other design rules, including pin pair-based rules, are included. Design rules are not added to a design from the reuse, but are used for comparison purposes.

Process of an Added Physical Design Reuse

When you add a physical design reuse to a design, SailWind Layout compares the design and the physical design reuse and records the results in the report file.

Information is processed in the following order:

- Layer Definitions Compared
- Part Types Compared
- PCB Decals Compared
- Components Added
- Pin Pairs Added

- Routes and Design Rules Added
- Polygon and Text Items Added

When you add a second instance of a physical design reuse to the same design, the checking that occurs (described below) is skipped. However, if you copy a physical design reuse and paste it into a different design, the checking is performed.

A report file is created logging any errors or warnings. The report file is named *Layout.err* and located in *..\SailWind Projects*. If an error is encountered, the add reuse process cancels. If you receive warnings, you can choose whether to cancel the process or add the reuse.

Layer Definitions Compared

The layer arrangements in both the physical design reuse and the design are compared. If both are identical, the physical design reuse is added.

Errors include:

- Layer counts not matching.
- Layer types not matching. Documentation layer type differences are not treated as errors. The layer type in the reuse is ignored in this case.

Warnings include:

- Layer names not matching. The layer name in the reuse is ignored. The report file details the layer name change.
- Other Layer Definition Data. Layer thickness, layer associations, and so on, are stored in the reuse, but the design settings are always used. This includes netnames associated with plane layers.

Part Types Compared

The part type names in the physical design reuse and the design are compared. If both are the same, the two part type definitions are compared. The definitions must match exactly, or the process cancels. Any differences found are recorded as errors in the report file for each part type.

PCB Decals Compared

If the PCB decals share the same name in the physical design reuse and in the design, the two decal definitions are compared. The definitions must match, or the add reuse process cancels. Slight differences in the decal definitions are tolerated. The location and orientation of the reference designator, part type, and attribute labels do not have to be identical. The attribute name and value, however, must match. Also, if the decal definitions differ, but the pin counts match, the decal definition in the physical design reuse is added to the design with a new name (its original name with an “A” suffix).

When the reuse is added, the PCB decal definition in the design is used.

When the decal pin counts do not match, errors are recorded in the report file. When PCB decals are renamed, warnings are recorded in the report file.

Components Added

After the Reuse Properties dialog box settings are accounted for, component elements are added. If a part cannot be added, the add reuse process cancels.

Warnings include:

- Start at, Increment by, or Suffix/Prefix assigns a used reference designator. If a current setting in the Reuse Properties Dialog Box creates a reference designator conflict, a message immediately appears and you can return to the Reuse Properties dialog box to choose different settings.
- Start at or Increment by assigns a new reference designator. If a reference designator is renamed (for example with a prefix), a message is recorded in the report file.
- Same or Next Highest assigns a new reference designator. If a reference designator is renamed, a message is recorded in the report file.
- Assign Prefix assigns a new reference designator. If a reference designator is renamed, a message is recorded in the report file.
- Assign Suffix assigns a new reference designator. If a reference designator is renamed, a message is recorded in the report file.

Pin Pairs Added

Pin pairs are added to the design. Pin pairs are also added to the design to merge reuse nets with existing nets in the design. If all pin pairs can be added to the design without conflict, the Add Reuse process continues. If a pin pair cannot be added, the Add Reuse process continues but the net is not added to the design.

When a net is renamed, a message is recorded in the report file.

Routes and Design Rules Added

Route elements are added based on the same rules and checking procedures for the route copy command. If any portion of the route patterns in the physical design reuse cannot be added to the design, the Add Reuse process continues and a message is recorded in the report file.

Nets added to the design assume the design rules for the net with which they merge. Otherwise, nets assume the default net rules. Rules saved with the physical design reuse are not used.

Jumpers are added to the design similarly to components and vias. The pad stacks and components are added as defined in the reuse definition, including any settings for thermal status, test point status, and so on.

Warnings include:

- Single pin nets not added without an existing net. Single pin nets are only added to the design when a net of the same name already exists in the design. Otherwise, single pin nets are not added. If single pin nets are removed, a warning is recorded in the report file.
- Via definitions not matching. If via definitions are the same in the physical design reuse and the design, the two via definitions are compared. The definitions must match or the via is added to the design with a new name (its original name with an “A” suffix). When the reuse is added, the

via definition in the design is used. If a new via definition is created, a message is recorded in the report file.

- Add route fails. If any portion of the route cannot be added, a message is recorded in the report file describing the failure.
- Add jumper fails. If a jumper cannot be added, a message is recorded in the report file describing the failure.

Polygon and Text Items Added

Polygon and text items are added to the design. There are no restrictions on adding polygons to a design, since adding a reuse must be performed with DRC off.

Tip
Pour cutouts are saved in the physical design reuse; however, board cutouts are not.

Creating a Physical Design Reuse

Select design items and add them to a reuse.

Procedure

1. Select the objects you want to include in the physical design reuse. You must select the entire object - partial selections are not included.

   Tip
   Copper planes must be in Pour Outline mode to be included in the Reuse. The copper plane cannot be flooded or in Hatch Outline mode when the Reuse is created.

2. Right-click and click the **Make Reuse** popup menu item. Ineligible items are removed from the selection. Among them, 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.

   Tip
   If one of the items you selected is not a valid physical design reuse element, 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, in the Reuse Type box, type a description for the reuse.
4. Optionally, in the Reuse Name box, type a different reuse name if the default name based on the Reuse Type is not acceptable.
5. If you want to save the reuse to a file to use it as a template for cloning or to use it in other designs, select the Save to File check box.

6. Optionally, click to view the Selection Report or Deselection Report, to ensure that all expected elements are in the reuse. The report file *report.rep* is created in the *C:\<install_folder>\<version>\Settings* folder and opens in the default text editor. 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.

7. Click **OK**.

    - 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.

8. If you selected the Save to File check box, the Reuse Save As dialog box appears.

    a. 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 *\SailWind Projects\Reuse*.

    b. 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.

    c. 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.

Results

Now you can:

- Move the reuse on page 602.
- Create clones of the reuse on page 603.
- Add the reuse to a design on page 606.

Selecting a Reuse in the Design

Click on a reuse in the design to modify it or to view its properties.

Procedure

1. Select any element in the physical design reuse except for component pins.

2. Right-click and click the **Select Reuse** popup menu item. The physical design reuse is selected.

Tip
Alternatively, to select physical design reuse blocks you can set the Selection Filter on page 114 to Reuse or the Find Dialog Box to Reuse type.

Results

Now you can:

- Modify the properties of the reuse on page 609
- Move the reuse on page 602
- Rotate the reuse
- Create clones of the reuse on page 603
- Break the reuse on page 610
- Set the origin of the reuse on page 610
- Save it to a file on page 602 for use in other designs or to add a reuse to the design in ECO mode
- Report on the contents of the reuse on page 611

Moving a Reuse

When you move a physical design reuse, traces routed to elements outside the reuse stretch, similarly to moving a group.

Procedure

1. Type the DRO modeless command on page 83 and press the Enter key to turn off Design Rule Checking. DRC must be off to move a reuse.
2. Select the physical design reuse. on page 601
3. Right-click and click the **Move** popup menu item.
4. Click to indicate a new location for the physical design reuse. You can right-click and click the **Cancel** popup menu item to cancel the move.

Saving a Reuse

You can save a physical design reuse to a file at any time. The saved version of the physical design reuse becomes known as the *reuse definition*.

Procedure

1. Select a reuse on page 601.
2. Right-click and click the **Save to File** popup menu item.
3. Optionally, you can type a new name for the reuse type from what it was named within the design.

4. Browse for a location in which to save the selected physical design reuse. The default file name is the reuse type name with a *.reu* file extension. The default folder is *\SailWind Projects\Reuse*.

5. Click **Save**.

   - If you changed the reuse file name, the message “Do you want to change reuse type name to XXX?” appears. 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. Click **Yes** to overwrite the reuse file or click **No** to specify another type.

Results

Now you can:

- Use the reuse as a template to create clones in another design. on page 603
- Add the reuse to any design in ECO mode. on page 606

Design Elements Made Like a Reuse Definition

The Make Like Reuse command uses existing design components with their logical interconnects to create a clone of a reuse definition. SailWind Layout automatically creates all other physical design reuse elements not yet present in the design—for example, traces, vias, coppers, 2D lines, and text.

The Make Like Reuse process maps components and interconnects of the physical design reuse circuit and searches for a match among the subcircuits within the design. Components are filtered based on the part type, the number of connections, the decal type, and the value/tolerance attributes. Nets are filtered based on the number of connections. Successive passes filter components and nets based on neighboring net and neighboring component characteristics, including terminal types and connection count. Parallel circuits are handled, and the filtering is independent of reference designators and netnames.

Default reuse properties are assigned to the new physical design reuse, including the reuse name. The Designator enumeration and Net Preferences are ignored since the elements already exist in the design.

Make Like Reuse skips glued components, components that are union or cluster members, component elements in another physical design reuse, and components with attached traces.

You can add a physical design reuse in either of these ways:

- Using Make Like Reuse in Object Mode — Use this method to allow the software to find any compatible elements to use in the reuse clone.
- Using Make Like Reuse in Verb Mode — Use this method to specify which design elements are used in each reuse clone. By choosing the components based on their schematic relationship, you can maintain the expected placement of the components.

Using Make Like Reuse in Object Mode

Select a reuse within the design and create a clone of it using design elements that the software selects.

Restrictions and Limitations

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.

Procedure

1. Type the modeless command DRO and press the Enter key to turn off Design Rule Checking. 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 "Selecting a Reuse in the Design."
3. To use the Reference Designator positions and style of the reuse, click the **Tools > Options** menu item and then click the **Design** category, and select the Apply reuse Ref Des placement check box.
4. In the Matching Result Dialog Box, see whether a match to the selected reuse is found.
    - If yes, each part has an equivalent one in the Equivalent Ref. Des. list, and the new physical design reuse dynamically attaches to the pointer after clicking **OK**.
    - If not, some parts have no match. In this case, you can clear the Value check box to have another try.

    For more parameter description, see Table 189.
5. Click **OK** to close the dialog box. If a match is found in Step 4, with the like reuse on the pointer, click to place it.

Results

If you have bridge copper in your reuse block, you should ensure that it has retained its bridge status and assigned nets.

Using Make Like Reuse in Verb Mode

Unlike other buttons, the Make Like Reuse button is not "sticky" in verb mode, but instead it allows you to pre-select the components to use in the reuse before applying the Make Like Reuse command.

Restrictions and Limitations

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.

Procedure

1. Type the modeless command DRO and press the Enter key to turn off Design Rule Checking. 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. Optionally, select the group of components you want to include in the new reuse.

> **Tip**
> If you do not select a specific group of components to use in the reuse, any available components that match the reuse are used. This can group components that are not grouped together in the schematic. Skip this step if it does not concern you.

- If you want to pre-select a particular group of components to use in the reuse, you must save your reuse definition before making a like reuse.
- Use cross-probing with the schematic software to select components that are grouped in the schematic.
- 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.

3. To use the Reference Designator positions and style of the reuse, click the **Tools > Options** menu item and then click the **Design** category, and select the Apply reuse Ref Des placement check box.
4. Click the **Design Toolbar** button and then click the **Make Like Reuse** button on the Design Toolbar. The Open Reuse File dialog box appears.
5. Select a reuse in the Select Reuse Module Dialog Box.
6. In the Matching Result Dialog Box, see whether a match to the selected reuse is found.
   - If yes, each part has an equivalent one in the Equivalent Ref. Des. list, and the new physical design reuse dynamically attaches to the pointer after clicking **OK**.
   - If not, some parts have no match. In this case, you can clear the Value check box to have another try.

   For more parameter description, see Table 189.
7. Click **OK** to close the dialog box. If a match is found in Step 6, with the like reuse on the pointer, click to place it.

Results

If you have bridge copper in your reuse block, you should ensure that it has retained its bridge status and assigned nets.

Reuse Blocks Added in ECO Mode

You can add reuse blocks to your design. Since adding reuse blocks will add new netlist elements, it must be done in ECO mode.

When the elements already exist in the schematic, use the Make Like Reuse command instead.

The methods, to adding a reuse in ECO mode, are different depending if it is the first instance of the reuse or if you are adding an existing reuse.

Adding the First Instance of Design Reuse

After you create a physical design reuse and save it to a reuse definition file, you can add the elements of the reuse as new design elements in ECO mode.

Restrictions and Limitations

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.

Procedure

1. Click the **ECO Toolbar** button. To add or copy a reuse, you must be in ECO mode; therefore, all components, pin pairs, and nets added to the design can be recorded in the *.eco* file using standard ECO commands.
2. Type the DRO modeless command on page 83and press the Enter key to turn off Design Rules Checking. 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 on page 1491 appears.
7. Resolve net conflicts, if any, and click **OK**.
8. In the Reuse Properties dialog box, 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.

   For more information, see “Process of an Added Physical Design Reuse”.

   The results of this comparison are recorded in the *Layout.err* report file in the *C:\<install_folder>\<version>\Programs* folder.

- 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.

- 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.

9. When all physical design reuse elements are successfully added to the design, the physical design reuse dynamically attaches to the pointer. With the Reuse on the pointer, click to place it.

Results

If you have bridge copper in your reuse block, you should ensure that it has retained its bridge status and assigned nets.

Related Topics

Adding an Existing Reuse

Process of an Added Physical Design Reuse

Adding an Existing Reuse

Add a copy of an existing physical design reuse to your design in any of three ways.

Procedure

1. Click the **ECO Toolbar** button. To add or copy a reuse, you must be in ECO mode; therefore, all components, pin pairs, and nets added to the design can be recorded in the *.eco* file using standard ECO commands.

> **Tip**
> You can copy and paste a reuse outside of ECO mode if the reuse contains only drafting items and no netlist items.

2. Type the DRO modeless command on page 83and press the Enter key to turn off Design Rules Checking. 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. Select the reuse using one of the following methods:

- **Copy and Paste** — select the reuse in the design:

    i. Select an existing physical design reuse in the design as described in “Selecting a Reuse in the Design”. You can only copy one reuse at a time.

    ii. Click the **Edit > Copy** menu item.

    iii. Click the **Edit > Paste** menu item.

- **Add Reuse button in Object mode** — select the reuse in the design:

    i. Select an existing physical design reuse in the design as described in “Selecting a Reuse in the Design”.

    ii. Click the **Add Reuse** button on the ECO Toolbar.

- **Add Reuse button in Verb mode** — select the reuse definition file:

    i. Click the **Add Reuse** button from the ECO Toolbar. The Add Reuse dialog box appears.

    ii. Click the reuse file you want to add and click **Open**. Since 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.

    iii. Click **OK**.

4. In the Reuse Properties Dialog Box that appears, modify the physical design reuse properties as needed.

5. Click **OK**.

6. With the Reuse on the pointer, click to place it.

Results

If you have bridge copper in your reuse block, you should ensure that it has retained its bridge status and assigned nets.

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.

Related Topics

Process of an Added Physical Design Reuse

Editing a Physical Design Reuse Definition

You can edit a reuse definition and update the reuse file (*.reu*), modifying it like a PCB design.

The reuse definition on page 1854 is the master copy of the physical design reuse that saves 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.

Procedure

1. Open the physical design reuse file:
    a. Click the **File > Open** menu item.
    b. Click SailWind Layout Reuse Files (*.reu) as the file type.
    c. Navigate to the appropriate folder and click the reuse file to open.
    d. Click **Open**. The reuse is selected after you open it.
2. Select the physical design reuse. on page 601
3. Right-click and click the **Break Reuse** popup menu item to break the reuse on page 610.
4. Edit the reuse, as necessary.
    - Change the layer stackup on page 315.
    - Change the design rules on page 412.
    - Modify the reuse elements.
5. When finished changes to the reuse, recreate the physical design reuse:
    a. Select all of the reuse elements.
    b. Right-click and click the **Make Reuse** popup menu item. The Make Reuse Dialog Box appears.
    c. Select the Save to File check box.
    d. Type the Reuse Type and Reuse Name or if you do not remember it, you can type any name and select the correct file name in the next step.
    e. 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.
    f. Click **Yes** to overwrite the existing file. The physical design reuse definition is updated.

Modifying Reuse Properties in the Design

You can modify a physical design reuse, but you cannot 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 record in the *.eco* file using standard ECO commands.

Procedure

1. Select a physical design reuse. on page 601
2. Right-click and click the **Properties** popup menu item.
3. Modify the location, rotation, glued status, and reuse name, if necessary.
4. In ECO mode on page 1824, you can modify the reference designator preferences or net properties, if necessary.
5. Click **OK**.

Results

If you changed the reference designator preferences or net properties, the component elements and private nets are renamed.

Resetting the Origin of a 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.

Procedure

1. Select a physical design reuse on page 601 whose origin you want to reset.
2. Right-click and click the **Reset Origin** popup menu item. 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.

Breaking a Reuse

You must break a physical design reuse before you can edit the elements within it. 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 return to their pre-reuse state; SailWind Layout maintains route protection and other settings. You do not need to be in ECO mode to break a physical design reuse.

Procedure

1. Select a physical design reuse on page 601
2. Right-click and click the **Break Reuse** popup menu item. The message “OK to Break Reuse(s)?” appears.
3. Click **Yes** to break the physical design reuse.

Note:
Reuses are automatically broken when you import an *.eco* file.

Deleting a Reuse

You can delete a reuse in ECO mode. Doing so removes all components and nets in the reuse.

If you want to simply break the reuse, see "Breaking a Reuse".

Restrictions and Limitations

You cannot delete physical design reuses that contain glued components or protected routes.

Procedure

1. Click the **ECO Toolbar** button. To delete a reuse, you must be in ECO mode; therefore, all components, pin pairs, and nets deleted from the design can be recorded in the *.eco* file using standard ECO commands.
2. Select the physical design reuse on page 601 you want to delete.
3. Click the **Edit > Delete** menu item, or press the Delete key.
4. In the confirmation message that appears, clear the Delete Routes check box if you want to preserve routes connected to the physical design reuse. It is not always possible to maintain traces connected to physical design reuse components. Delete Routes is selected by default, meaning routes are deleted with the physical design reuse.
5. Click **OK** to delete the physical design reuse.

Creating a Reuse Report

You can create a report that contains the name, type, and elements of the selected physical design reuse.

Procedure

1. Select the physical design reuse on page 601.
2. Right-click and click the **Report Contents** popup menu item.

Results

The report is written to *C:\SailWind Projects\rules.rep* and is displayed by the default text editor.

Chapter 27
Drafting Operations

Read the topics that follow to learn more about the drafting commands that are available on the Drafting Toolbar. Use drafting commands to create and edit drafting objects, including the board outline, copper areas, keepout areas, simple line shapes, text, and all other items not generally associated with part placement or routing.

Creating a Drafting Object

The methods for creating all drafting objects apply in the same manner. These instructions apply to creating 2D lines, copper, copper cutouts, board outlines, board outline cutouts, keepouts, copper planes, and copper plane cut outs.

Procedure

1. After you click a button on the Drafting Toolbar, right-click to set any values before creating the drafting object on page 614.

2. Right-click and choose any of the following shapes to draw from the popup menu.

- Circle on page 624
- Polygon or Path on page 623
- Rectangle on page 624
- Chamfered path on page 656 (copper only)

Set 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. You can set various values after you click the button for the type of item to add, but before you place the drafting object.

Set Line Width

Choose one of the following methods to set the width of the lines in the drafting object you are creating:

- Type W<nn>, where <nn> is the width value
- Right-click and click the **Width** popup menu item and type a value.
- Set the Default width value in the Text and Lines options on page 1553.

Set Line Style

For 2D lines only, choose one of the following methods to set the line style:

- Right-click and click the **Line Style** popup menu item, then choose a line style from the list.
- Type one of the LS Modeless Commands to set the line style.

Set Layer Placement

Choose one of the following methods to select the layer for the placement of your drafting object:

- Type L<n>, where <n> is a layer number
- Right-click and click the **Layer** popup menu item to specify the layer where you want to place the Drafting object.
- Specify the layer using the Layer list on the Standard Toolbar.

Set Auto Mitering of Corners

Choose one of the following methods to auto miter corners (diagonal miter or arc):

- Right-click and click the **Auto Miter** popup menu item to miter all added corners. Define the miter type and size in the Design options on page 1503.
- In the Design options on page 1503 in the Miters area, select the Auto miter check box.

Set Snap to Objects

You can snap to existing objects instead of to locations on the design grid. You can begin drawing from an object snap point or draw to an object snap point. You can enable snapping to objects in the Object Snap Options on page 1540 or on the popup menu while drawing drafting objects. You can also use the OS Modeless Commands.

- Right-click and click the **Snap to Objects** popup menu item.

Note:
To filter the objects you can snap to, right-click, choose the **Snap to** popup menu item and then click to add or remove the check mark beside the types of objects you want to snap to.

Set Corner Angle

Choose one of the following methods to select a corner angle.

- Right-click and click the **Orthogonal**, **Diagonal**, or **Any Angle** popup menu item.
- In the Design options on page 1503, in the Line/trace angle area, choose a setting.

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

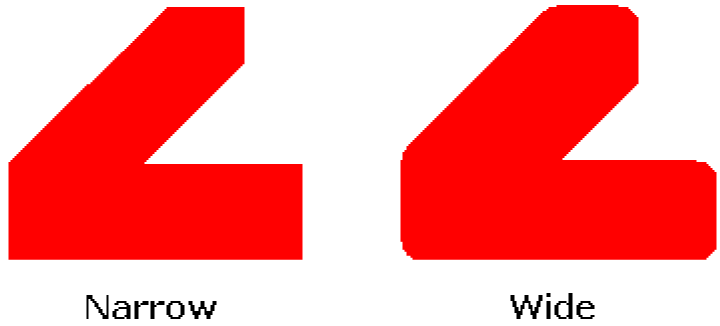
Edge Precision of Drafting Shapes

You often need to precisely control the edge of drafting shapes, as many factors contribute to that edge.

Line Width

The precision of edges is controlled by the line width used to create the shape. The default line width is set on the **Text and Lines** category of the Options dialog box. However, you can change the line width prior to, or after creating your shape. Use a very narrow outline width to achieve a sharp corner or increase the value for more blunt corners. All corners are rounded with a radius equal to one half of the outline width.

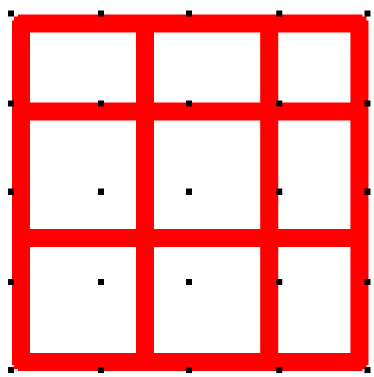
Figure 66. Narrow Versus Wide Outline Width



Design and Hatch Grids

Lines are used to create the outline of the shape and are also used to create the fill inside electrical drafting items. Lines or outlines are placed on the design grid. Then shapes are filled using horizontal lines placed on the Hatch Grid.

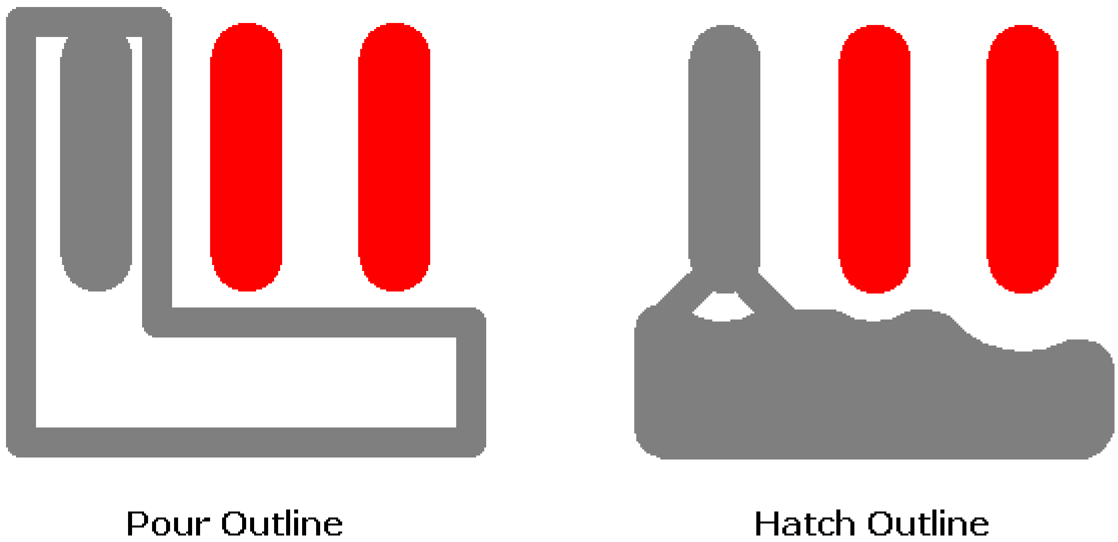
Figure 67. Hatch Grid



Clearance Rules

If you are creating a copper plane and your drafting shape is near another object, the resulting poured outline could be displaced by the clearance rules of that object.

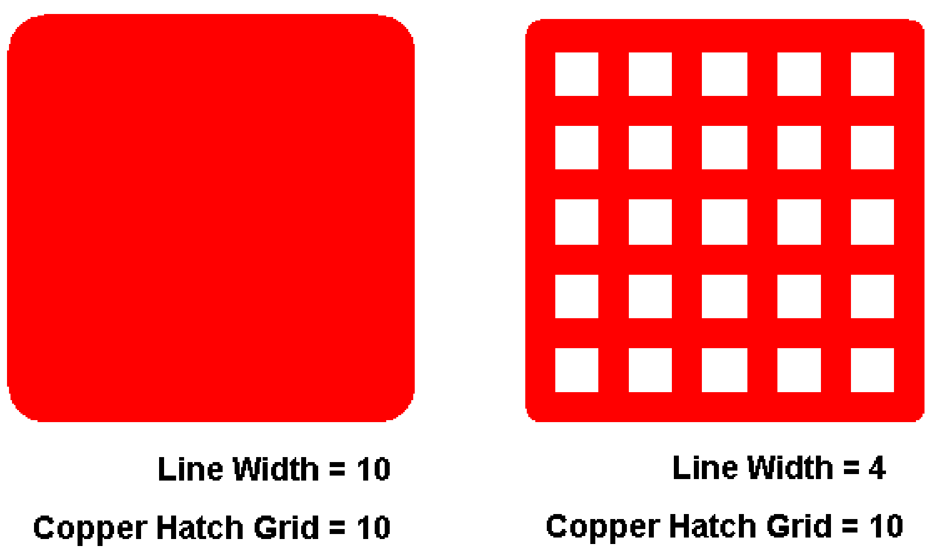
Figure 68. Clearance Rule Effect on the Hatch Outline



The Fill of Copper, and Copper Planes

SailWind Layout generates the fill of copper and copper plane shapes using drafting lines drawn on the copper hatch grid. If you create the shape using a line width value less than the copper hatch grid value, it results in a hatch pattern. If you use a line width value equal to, or greater than the copper grid value, it results in a solid copper shape.

Figure 69. Example of Solid and Hatch Pattern



Shapes can be flooded with copper in varying degrees of hatch through to a solid pattern. The pattern used by default when you flood a shape is determined by the relative values of two settings in the Options dialog box:

- The Default width value on the Drafting / Text and Lines options category on page 1553

- The Copper hatch grid value on the Grids options category on page 1537

You can set line width and grid settings before or after creating the object. After you've drawn the shape, you can change the settings in the Drafting Properties on page 1328. If you need to customize the copper hatch grid for a copper plane shape, you can set the grid value in the Flood on page 1381.

For more information on creating shapes with the desired flood pattern, see Creating a Copper Shape and "Creating a Copper Plane Manually" on page 667.

Migration to Copper Planes

Prior to the VX.2.4 release, SailWind Layout used copper pours on "No Plane" layers and plane areas on "Split/mixed" layers. For all later releases, the software unifies copper pours and plane areas into "copper planes." If you open a design saved in an earlier version, PADS migrates or converts all copper pours and plane areas to copper planes and applies design rules accordingly. Certain clearance and connectivity changes in your design may result.

The following changes take place during the migration to copper planes:

- The "Remove unused pads" setting, found in the "Thermals Options" on page 1514, is extended to "No Plane" layers. With this setting enabled, all pads are removed on all internal "No Plane" and "Split/Mixed" layers unless the pin is connected to a trace or a copper plane on that layer. Prior to the PADS VX.2.4 release, unused pads were removed only from "Split/Mixed" layers.

- Copper pours converted to copper planes are affected by the Plane Thermal check box (in the Pin, or Via Properties dialog boxes) on page 797. Prior to PADS VX.2.4, the setting applied only to plane areas on "Split/Mixed" layers. This means, for example, if you have disabled thermal connections for a pad or via by clearing the "Plane Thermal" check box, the software removes the thermal connection to copper planes (old copper pours) on "No Plane" layers when you import the design into a newer release.

- You can apply custom thermals and custom antipads to copper planes on "No Plane" layers in addition to "Split/Mixed" layers. Prior to PADS VX.2.4, the custom thermals and antipads applied only to plane areas on "Split/Mixed" layers.

- Prior to PADS VX.2.4, a pin could receive a thermal connection to a plane area on a "Split/Mixed" layer even though a trace was connected to it. With later releases, any component pin with an attached trace does not receive a thermal connection if the "Add thermals to routed component pads" setting is disabled in the "Thermals Options" on page 1514. This was already the case for copper pours beginning with the PADS VX.2.3 release, and has been extended to copper planes on "Split/Mixed" layers in subsequent releases. Prior to PADS VX.2.3, a pin could receive a thermal connection to a copper pour even though a trace was connected and the "Add thermals to routed component pads" setting was disabled if an additional unrouted pin pair connection existed on the pin. Via pads will still receive thermal connections to copper planes regardless of the setting.

- The plane indicators, previously only shown on "Split/Mixed" layers and "CAM Plane" layers are extended to pins and vias surrounded by a copper plane on "No Plane" layers. The Show general copper plane indicators setting is found in the "Thermals Options" on page 1514.

- One smoothing radius setting in the "Hatch and Flood Options" on page 1511 applies to all copper planes, regardless of the layer on which they reside. Previous releases had separate smoothing radius settings for copper pours and plane areas. The value from the Hatch and Flood of an existing design is retained in VX.2.4 and later releases. It may be necessary to adjust the smoothing radius value if you experience connectivity errors in a later release that were not present in the prior version.

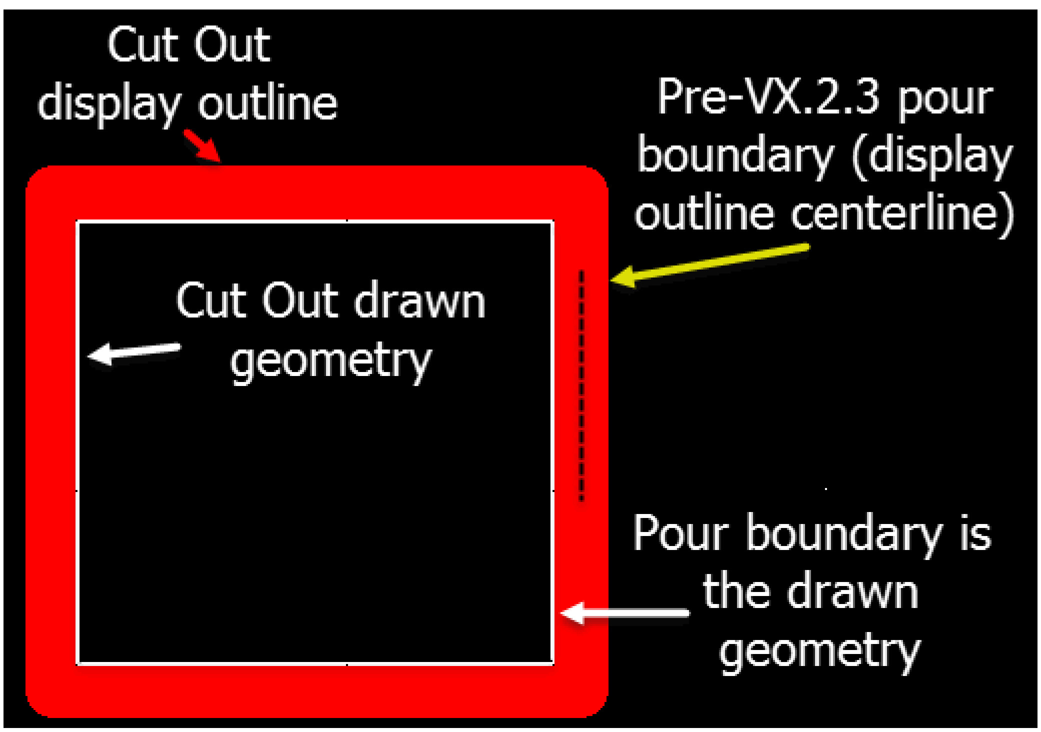
- The setting "Remove violating thermal spokes" is removed. All violating thermals spokes are removed by default. This is a hard-coded setting that cannot be modified.

- The modeless command "PO" toggles copper planes between pour outline and the flooded hatch display. The "SPO", "SPI", and "SPD" commands are no longer used.

- The "Create cutouts around embedded planes" setting in the "Hatch and Flood Options" on page 1511 works for "No Plane" layers in addition to "Split/Mixed" layers. Previously, that setting worked for only "Split/Mixed" layers.

- Starting with VX.2.3, copper floods to the inner edge of completed cut out outlines instead of the centerline. Prior to VX.2.3, the cut out would grow by half the line width since copper would flood to its centerline rather than the inner edge which was the intended boundary. For more information, see "Creating a Copper Plane Cut Out" on page 686.

Figure 70. Cut Out Pour Boundary Changes



- Beginning with PADS VX.2.6, the origin of the copper plane hatch grid is calculated from the lower left corner (if one exists or calculated if one does not exist) of the pre-flooded shape. If you used a hatched fill instead of a solid fill in a pre-VX.2.6 design, the hatch grid might not have the same origin and placement of the lines of the hatch grid. As a result, pins or vias that touched and connected to the hatch grid might not be connected. Other pins that did not touch or connect to the hatch grid might be in a position to connect.

The origin of the hatch grid pattern is defined by an imaginary box surrounding the extents of the copper plane shape. The extents box is reduced by ½ the line width of the copper plane shape and the hatch grid origin is placed in the lower left corner of the extents box. For a simple rectangular copper plane shape the hatch grid origin will be in the lower left corner of the shape. For more complex shapes, the lower left corner of the extents may be a point outside of the copper plane shape. This is not the origin of the copper plane shape, only the origin of the hatch pattern for the copper plane.

Figure 71. Real Lower-Left Hatch Origin

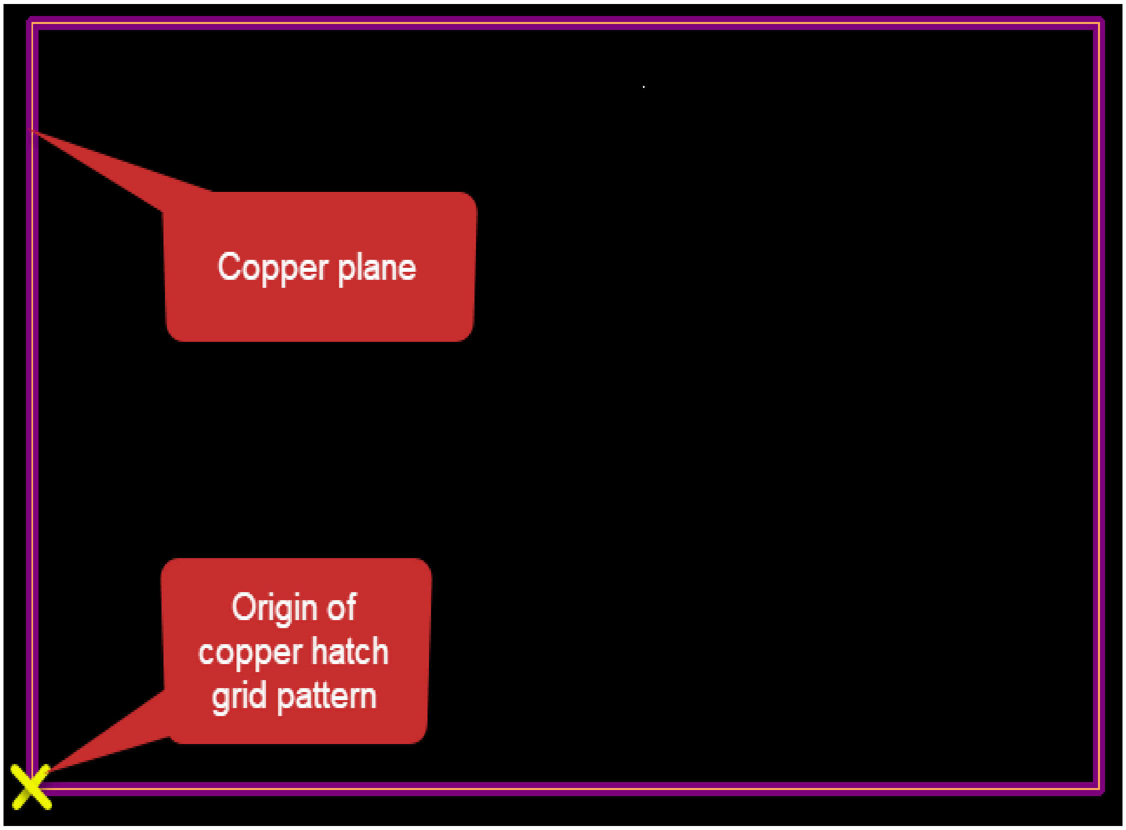
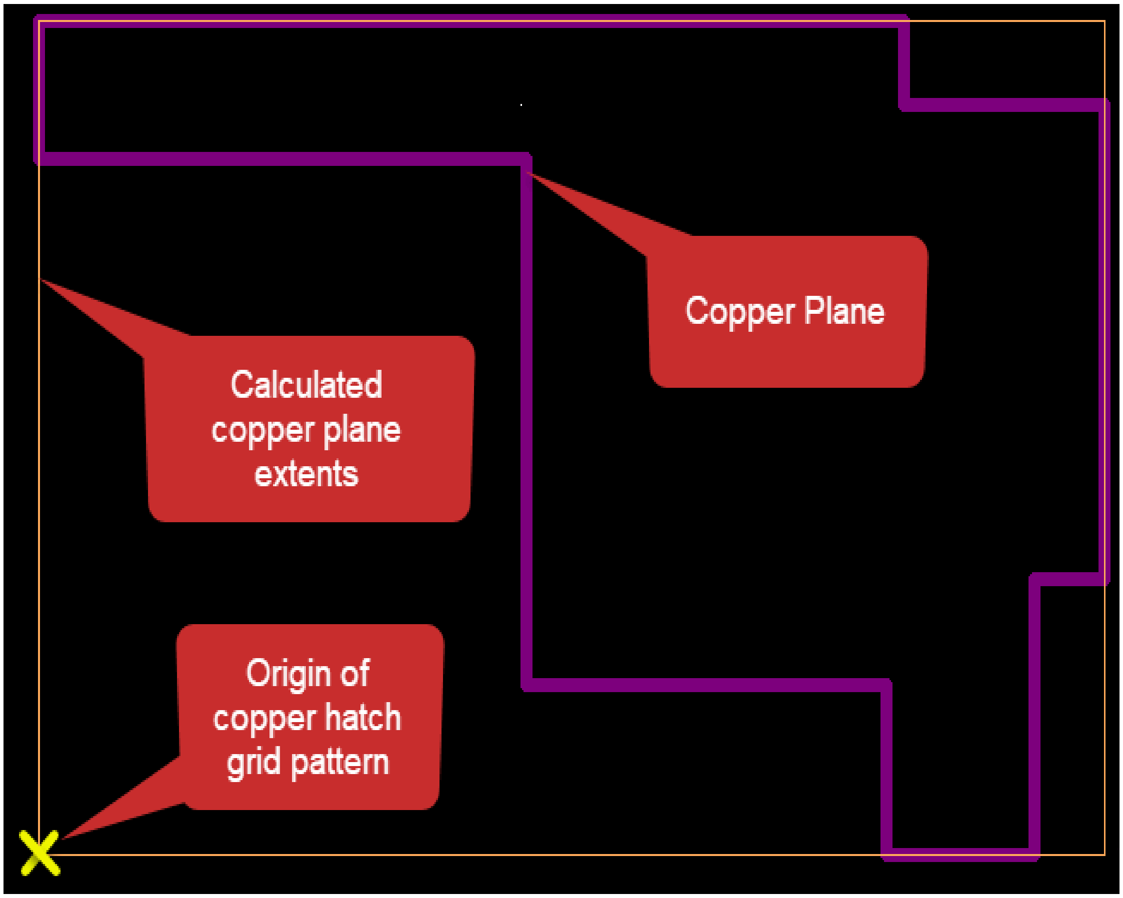


Figure 72. Calculated Lower-Left Hatch Origin



Plane and Net Connectivity

A *plane* is a large copper area that provides access to universally necessary nets, like power or ground. One design may use several plane areas.

Most PCB designs locate planes 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 only part of the layer.

Copper planes may have insulated traces and vias passing across the plane area, as long as the traces do not divide the plane to break connectivity.

Pins that are supposed to connect to the plane are usually plotted for manufacturing using spoked thermal relief pads. Pins that do not connect to the plane are referred to as unused pads, and are typically removed from inner layer planes and replaced with antipads.

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 Split/Mixed type and draw a Copper Plane
- Draw a Copper Plane on a No Plane layer

Displaying Connectivity with General Plane Indicators

Use the “Show general copper plane indicators” setting in the **Copper Planes** category > **Thermals** subcategory of the Options dialog box to see which pins have thermals. The “Plane thermal” Pin Properties setting determines whether the thermal is generated for the pin.

Set the Plane Thermal option using the Properties dialog boxes for pins, vias, and jumper pins.

Tip
Copper Planes on a No Plane or Split/Mixed layer are functionally equivalent. Using a Split/Mixed layer provides the “Show plane nets only” check box in the Drafting Properties and use of the Plane check in Verify Design. Some other software tools may make use of a layer specifically set as Split/Mixed to utilize certain functions.

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**, **Copper Cut Out**, **Keepout**, **Copper Plane**, **Copper Plane Cut Out** or **Board Outline and Cut Out**. The add command for that object type attaches to your pointer.

2. Right-click and set your drafting values on page 614.

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.

    Tip
    Right-click and click the **Add Arc** popup menu item to add an arc instead of a straight-line segment.

    Note:
    You can press the BackSpace key to remove the last corner entered. You can also press the Esc key to cancel the operation.

    ---

4. Double-click or right-click and click the **Complete** popup menu item 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

Set 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**, **Copper Cut Out**, **Keepout**, **Copper Plane**, **Copper Plane Cut Out** or **Board Outline and Cut Out**. The add command for that object type attaches to your pointer.
2. Right-click and set your drafting values on page 614.
3. Click to indicate the location of the circle's center. 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 the Esc key 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

Set 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**, **Copper Cut Out**, **Keepout**, **Copper Plane**, **Copper Plane Cut Out** or **Board Outline and Cut Out**. The add command for that object type attaches to your pointer.

2. Right-click and set your drafting values on page 614.

3. Click to indicate one corner of the rectangle. 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 the Esc key 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

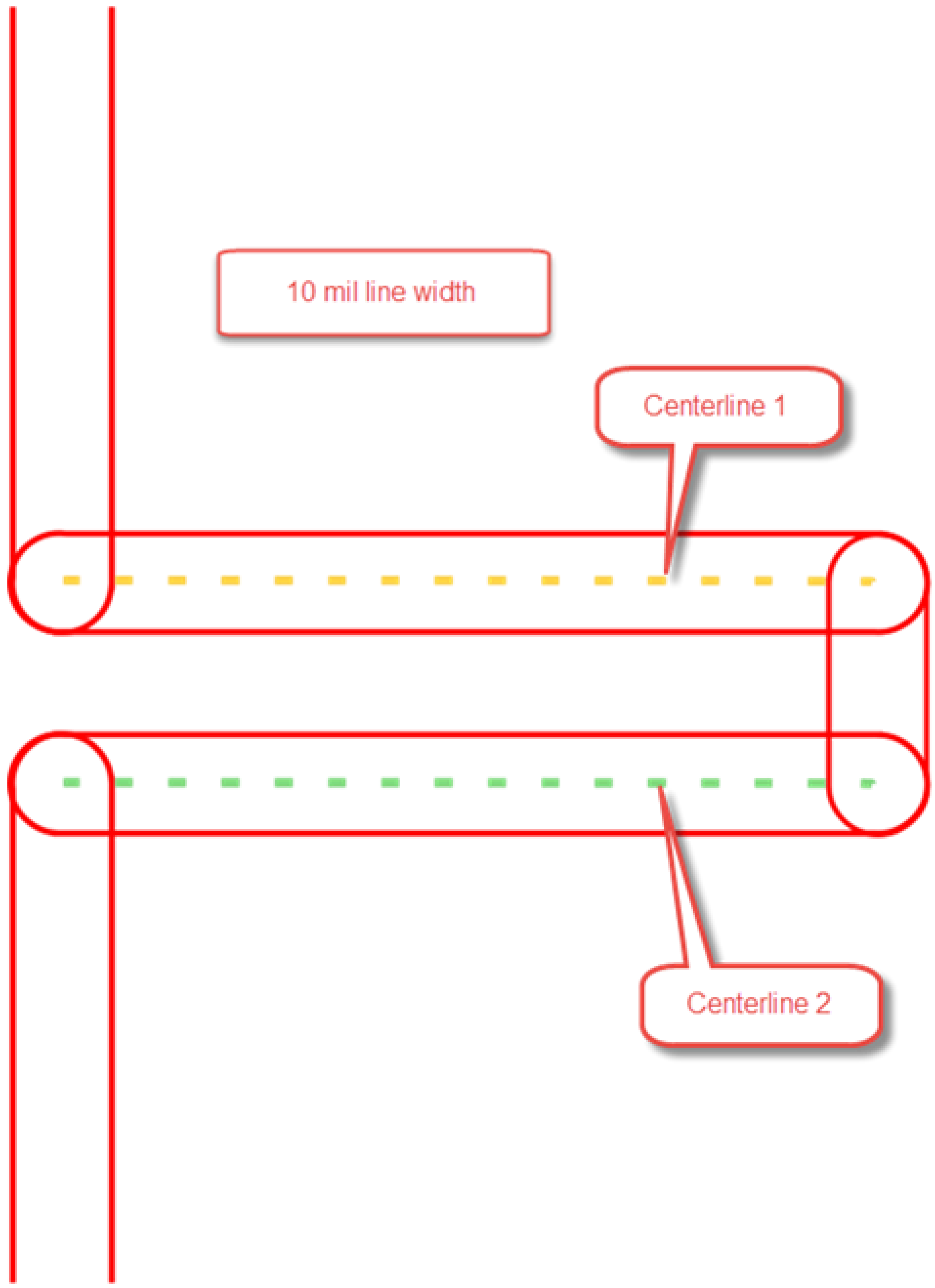
Set Values Before Creating a Drafting Object

Self-Intersecting Polygons

A *self-intersecting polygon* occurs whenever a portion of a shape outline overlaps with another portion of the same shape. In some cases, creating a polygon with too large a line width for the outline can also cause overlapping of some portions.

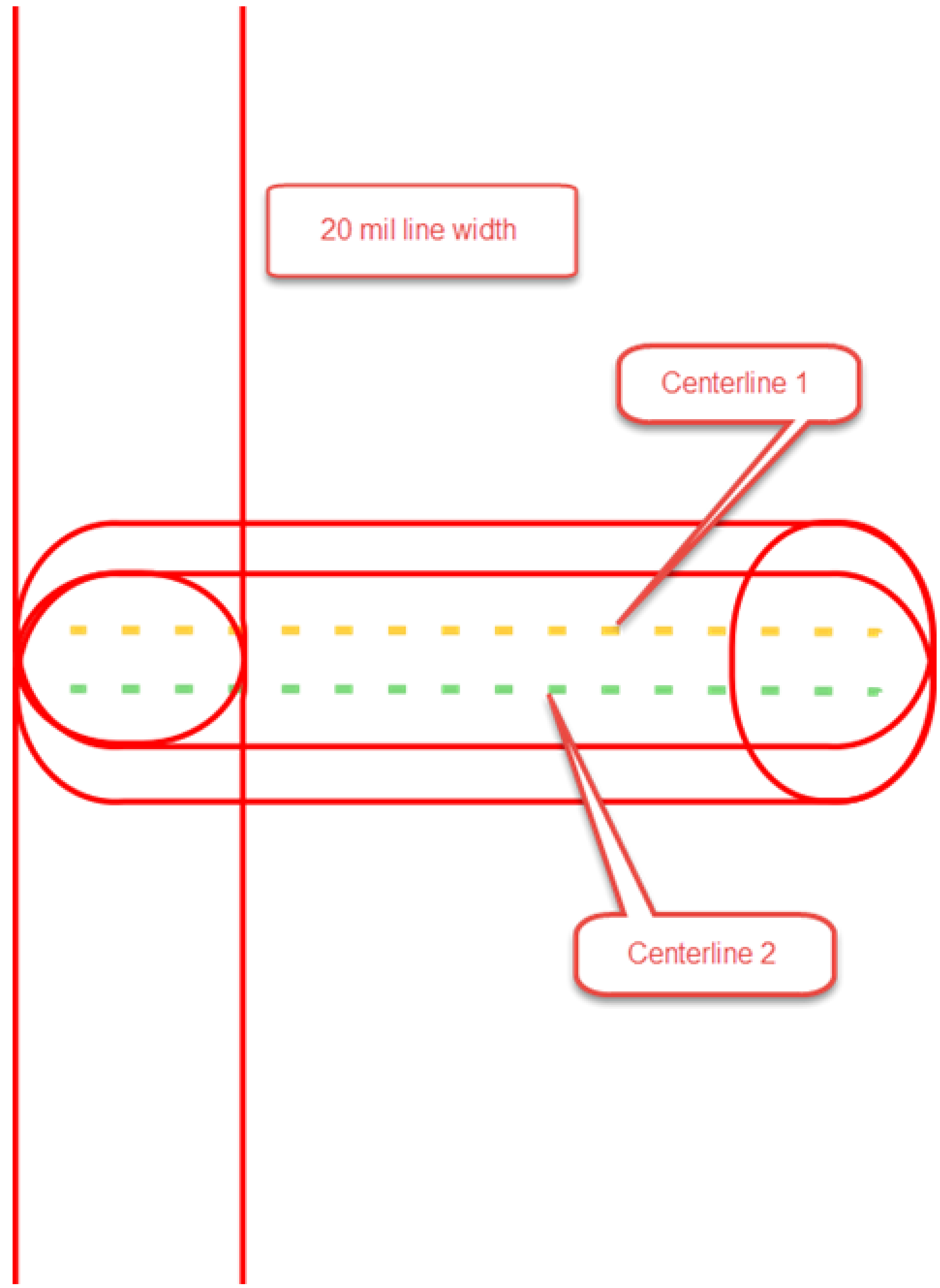
When drafting copper planes and shapes using polygons, you can set the outline line width in the Drafting Properties Dialog Box. Choosing a narrow line width enables you to create shapes with greater accuracy in detailed areas, as shown below.

Figure 73. Close up of Shape Detail



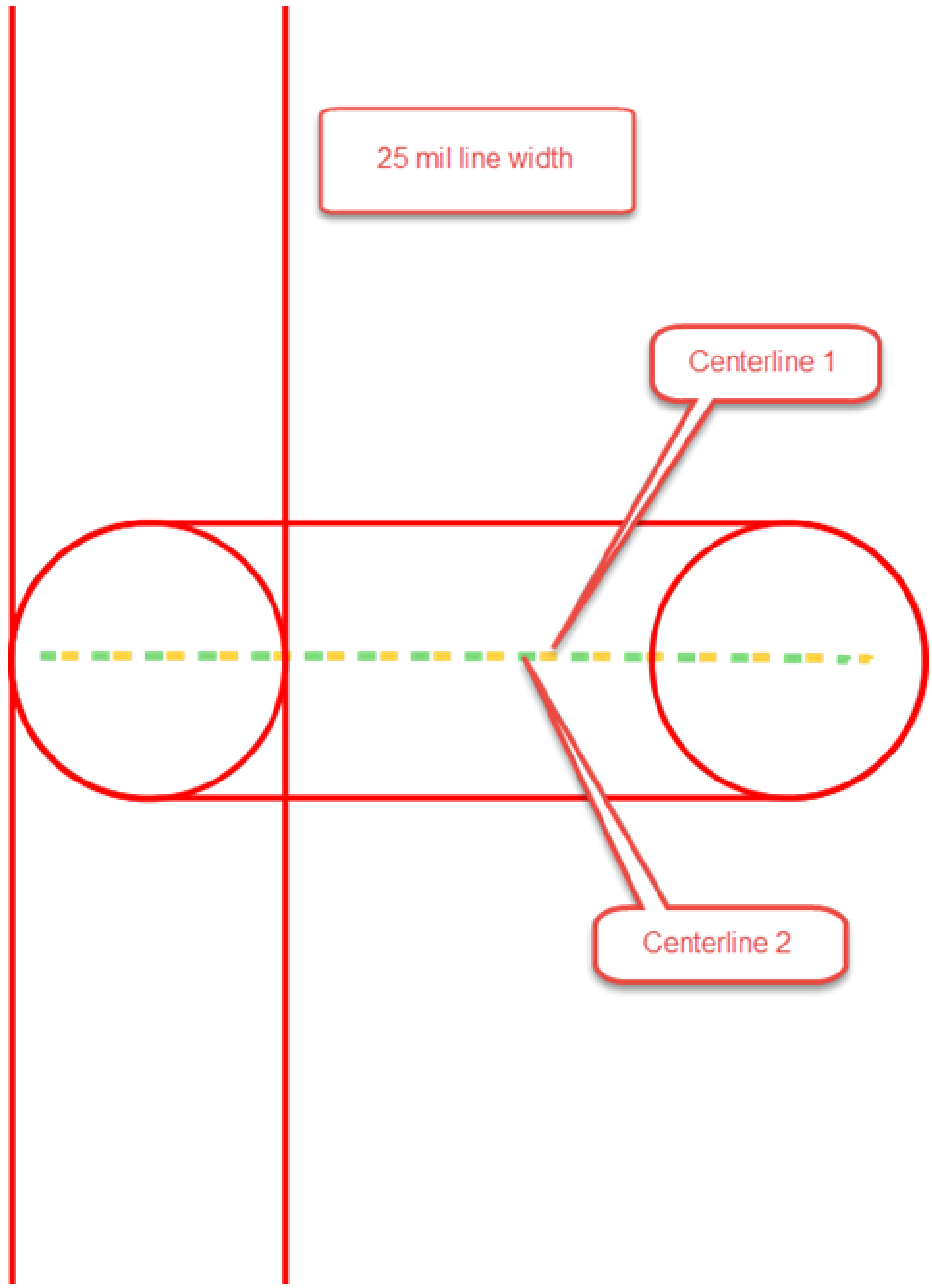
You can set the line width as required for your design; however, as the width increases, the shape resolution decreases. Referring to the example in the preceding figure, as the outline line width of a copper shape increases, the outer edge of the lines remain in place and the inner edges expand inward. The result appears in the figure below.

Figure 74. Shape Detail with Increased Line Width



If you set the outline line width too wide, the centerlines of two edge segments may overlap one another, resulting in a self-intersection polygon.

Figure 75. Overlapping Centerlines



Note:
The overlap is most visible when you turn on copper flooding.

Keep in mind that a self-intersecting polygon can occur with planes created from the board outline shape when the plane area shrinks by the clearance value. Similarly, a self-intersecting polygon can also occur when using the auto separate functionality because of the auto separate gap.

Text

Use free text to add text information to your design. Free text is not associated to a particular design object; you can use it to add descriptions such as board identification nomenclature.

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 on page 1813 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 "Embedded Text Documents on page 986".

Procedure

1. Click the **Drafting Toolbar** button to open the Drafting Toolbar, and then click the **Text** button.
2. In the Add Free Text Dialog Box, type the text string you want to use in the Text box. There is a maximum of 128 characters per text string.
3. In the Font list, select the font you want to use. The default font name and style appear in the Font box when the dialog box opens. Select a stroke font or a system font.

   Tip
   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 a line width.
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.

    - For vertical justification, click **Left**, **Center**, or **Right**.
    - For horizontal justification, choose **Up**, **Center**, or **Down**.

    Tip
    You can set justification in the design after closing the Add Free Text dialog box. Select the text, then right-click and click **Justify Horizontally**, and then click the **Left**, **Center**, or **Right** popup menu item; and right-click and click **Justify Vertically,** and then click the **Up**, **Center**, or **Down** popup menu item.

11. Click **OK** to close the dialog box. 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.

    Tip
    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

Mirroring Text

Moving 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.

Procedure

1. Select a text string, right-click, and then click the **Properties** popup menu item.
2. The Text Properties dialog box displays the selected text string. Modify the existing text string, or type a new text string.

    Restriction:
    Several options in this dialog box are unavailable if the text is part of a physical design reuse.

3. In the Font area, select the font you want to use. Select stroke font or a system font.

Tip
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 a line width.
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.
    - For vertical justification, click **Left**, **Center**, or **Right**.
    - For horizontal justification, choose **Up**, **Center**, or **Down**.

Tip
You can set justification in the design after closing the Add Free Text dialog box. Select the text, then right-click and click **Justify Horizontally**, and then click the **Left**, **Center**, or **Right** popup menu item; and right-click and click **Justify Vertically,** and then click the **Up**, **Center**, or **Down** popup menu item.

11. Click **OK**. If you select another text object before closing the dialog box, the text information is updated.

Related Topics

Text

Mirroring Text

Moving Text and Labels

Mirroring Text

You can mirror text objects in your design.

Procedure

1. Select the object.
2. Right-click and click the **Mirror** popup menu item.

Related Topics

Text

Modifying Text Properties

Combining Line and Text Objects

Moving Text and Labels

Moving Text and Labels

You can move text and labels around in your design.

Procedure

1. Select a text or label, right-click, and then click the **Move** popup menu item.

   The object attaches to the pointer. If you move a text object that is combined on page 642 with other drafting objects, all the combined objects 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 moving reference designators.

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.

Related Topics

Text

Modifying Text Properties

Combining Line and Text Objects

Mirroring Text

Modifying Copper Chamfered Paths Properties

You can select the copper chamfered paths of a net to change its properties.

Procedure

1. Select a net in the design.

2. Right-click and click the **Select Chamfered Paths** popup menu item.

   Alternatively, you can click  under the RF Toolbar.

3. Right-click and click the **Properties** popup menu item.

Related Topics

Drafting Object Properties

Scaling 2D Line Objects and Dimensions

Use Scale in the 2D line or Dimension object Properties to change the size of objects.

An arc is too large if its radius is greater than 14 inches or its center is outside the database coordinate area. The database coordinate area is (-28, -28) to (28, 28) inches. Arcs are approximated with multiple straight segments.

When you scale text or dimensions, combine them with other line objects. Line widths are not scaled.

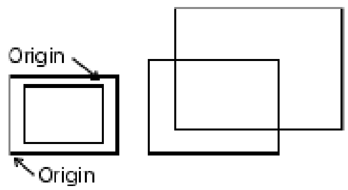
When combined text or dimension text is scaled, the maximum text height is 1000 mils and the maximum text width is 50 mils.

Location of Scaled Objects

If the scale model contains multiple objects and you want to maintain their relative positions, combine the objects before using Scale.

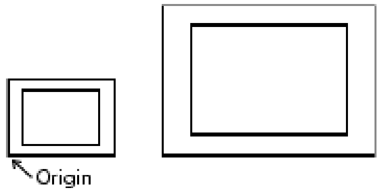
In the following figure, the two noncombined objects are scaled. The objects have individual origins, as shown on the left. These different origins are used when scaling, creating overlapping objects, as shown on the right.

Figure 76. Noncombined Objects



In the following figure, the two combined objects are scaled. The objects have a single origin, as shown on the left. This origin is used when scaling, creating a larger copy of the original objects, as shown on the right.

Figure 77. Combined Objects



Modification of Drafting Objects

After adding drafting objects to a design, you can move, edit, or delete them as your design needs change.

Moving a Drafting Object

You can move a whole drafting object while maintaining its shape.

Procedure

1. Select the drafting object to move using one of the following methods:

    - With nothing selected, right-click, then click the **Select Shapes** popup menu item, and click on a segment of the object. The Select Shapes selection filter setting is the best filter for selecting Drafting Objects even if it is an open shape like a single line.

    - Select a segment of the object, right-click, and then click the **Select Shape** popup menu item.

    - Shift-click a segment of the object

2. Right-click and click the **Move** popup menu item.

    Alternatively, you can drag the selected object, if allowed by your Drag moves setting on page 1531.

3. The shape attaches to the cursor by the Move preference setting on page 1503.

    Tip
    If you move a drafting object that is combined with other drafting objects, all the combined objects move.

4. If you need to make additional changes to the component before you move it, you can right-click and activate the rotate, spin, or mirror commands, or manage the Object snapping settings on page 1540.

Restriction:
You cannot modify, move, or delete a polygon of any type (2D line, copper, or pour) that is part of a physical design reuse. 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.

5. Move the drafting object to its new location and click to place it.

Modifying Drafting Edge Properties

You can edit or change the properties of drafting edges.

Procedure

1. To select an edge, click on it once.
2. Right-click and click the **Properties** popup menu item. 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

You can edit or change the properties of drafting corners, such as location.

Procedure

1. Select a drafting corner, right-click, and then click the **Properties** popup menu item.
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

Drafting Object Properties

Deleting a Corner

Moving a Corner

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.

Note:
Several of the options in this dialog box are unavailable if the object is part of a physical design reuse.

Converting Drafting Shapes

You can convert a closed drafting object to another shape. In the Layout Editor, you can convert any closed drafting object to a 2D line, board outline, board cut out, copper, copper cut out, keepout, or copper plane. In the PCB Decal Editor, you can convert any closed drafting object to a 2D line, copper, copper cut out, or keepout.

Restrictions and Limitations

When converting shapes, the following restrictions apply:

- 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. You must first have a board outline before you can create a board cut out.
- If there is already a board outline in the design, the Board Outline option does not appear in the dialog box because you cannot have two board outlines.

Procedure

1. Select a drafting item, right-click, and then click the **Properties** popup menu item.
2. In the Type list, select a new shape type.

Tip
The Copper Cut Out type is also used for cut outs in Copper Planes.

Changing Line Widths

You can modify the line width of drafting object outlines and fill lines as applicable.

Procedure

1. Select a drafting item, right-click, and then click the **Properties** popup menu item.
2. In the Drafting Properties Dialog Box, in the Width box, type a value.

> **Restriction:**
> Some drafting object line widths cannot be changed.

> **Note:**
> 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

Resize a drafting object.

Procedure

1. Select a drafting item.
2. The object will scale up or down from the origin of the shape. If necessary, right-click and click the Set Origin on page 474 popup menu item to set the origin of the scale model.
3. Right-click, and click the **Properties** popup menu item.
4. In the Drafting Properties Dialog Box, set scale options.

   Enter a floating-point number greater than 0. Fractions are not supported. Values greater than 1 will increase the size while values less than 1 will shrink the object. A scale value of 2 will double the shape's size, a scale value of 0.5 will halve the shape's size. The line width does not scale.
5. Click **OK**.

Results

- If scaling changes a dimension, select the dimension, right-click, and click the **Reset Measurement** popup menu item to update the dimension measurement.

Filling a Shape with Solid Copper

SailWind Layout fills copper using the settings of the Copper Hatch Grid and the Drafting Default width. If desired, you can ignore these settings and fill a shape with solid copper. When you create chamfered paths, they appear as solid copper.

Procedure

1. Select a copper drafting object, right-click and then click the **Properties** popup menu item
2. In the Drafting Properties dialog box on page 1328, select the **Solid Copper** check box.

> **Tip**
> The Solid Copper check box forces copper to use trace clearance rules.

Changing Layers

You can change the layer on which the drafting object exists.

Procedure

1. Select a drafting item, right-click, and then click the **Properties** popup menu item.
2. Drafting Properties Dialog Box, 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.

Moving a Miter

You can expand or contract a miter or arc on a drafting segment with the Move Miter command.

Procedure

1. Select a miter segment or arc, right-click, and then click the **Move Miter** popup menu item. 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.
4. Use the Stretch Command to modify an arc or miter on a route.

> **Restriction:**
> You cannot modify, move, or delete a polygon of any type (2D line, copper, or copper plane) that is part of a physical design reuse. 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, and then click the **Pull Arc** popup menu item.
2. The segment or corner attaches to your pointer and changes to an arc.
3. Click to indicate the new position for the arc.
4. Use the Move Miter on page 639 command to modify the radius of a created arc or miter.

Restriction:
You cannot modify, move, or delete a polygon of any type (2D line, copper, or copper plane) that is part of a physical design reuse. 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 Arc Miters

Creating Arcs

Deleting a Drafting Segment or Object

If desired, you can delete drafting segments or objects from your design.

Procedure

1. Select the segment or object. See “Selection of Drafting Objects”.
2. Press the Delete key or click the **Edit > Delete** menu item.

Restriction:
You cannot modify, move, or delete a polygon of any type (2D line, copper, or copper plane) that is part of a physical design reuse. 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

Set Values Before Creating a Drafting Object

Deleting an Item

Use the Delete command to remove a selected item from the design.

Procedure

1. Select the item you want to delete.
2. Click the **Edit > Delete** menu item. The item is removed from the design.

Combine Drafting Objects

You can combine drafting objects so that they can be moved as a group. If the design changes, you can explode or un-combine the groups of 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 the Combine command in both 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 the **Combine** popup menu item. You cannot combine text objects without including a drafting object.

Results

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 on page 647 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 Objects

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.

Procedure

1. Select the combination.

> **Tip**
> To select a combination, select any part of the combination, right-click, and click the **Select Shape** popup menu item.

2. Right-click and click the **Explode** popup menu item.

Related Topics

Combining Line and Text Objects

Uncombining Drafting Objects

Uncombining Drafting Objects

Use the Uncombine command—accessed in the Decal Editor—to remove a single item from a combined object. Combined objects are made of multiple drafting and text objects that you have combined using the Combine command.

Procedure

1. Select the combination.

> **Tip**
> To select a combination, select any part of the combination, right-click, and click the **Select Shape** popup menu item.

2. Right-click and click the **Uncombine** popup menu item.
3. Select the object.

Removing a Text Object From a Combination

If necessary, you can remove a text object by uncombining a combination.

Procedure

1. Select the text object.
2. Right-click and click the **Uncombine** popup menu item.

Related Topics

Combining Line and Text Objects

Exploding Combined Objects

Join and Close 2D Lines and Copper Shapes

You can join and close 2D lines and copper shapes so that changes applied to one object apply to all of the joined objects.

Use Join and Close to Connect 2D Lines and Copper Shapes

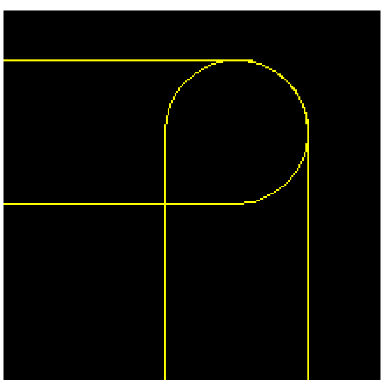
Use Join and Close commands to connect 2D lines and copper shapes to create single objects for the purpose of changing their properties. For example, you can create a closed 2D line and then change its properties to “copper.”

Tip
These commands are especially useful for after you have imported a *.dxf* file.

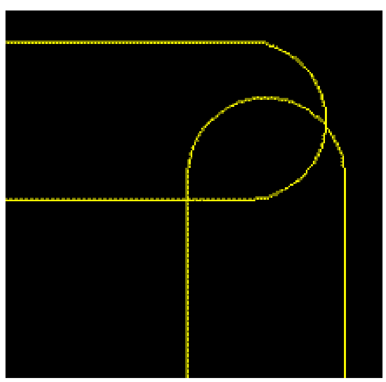
You can use Join and Close when the line ends are “near enough”: meaning: touching, crossing slightly, or perfectly matched. If the lines are not touching or crossed too far, Join and Close will not work.

Examples of Near Enough

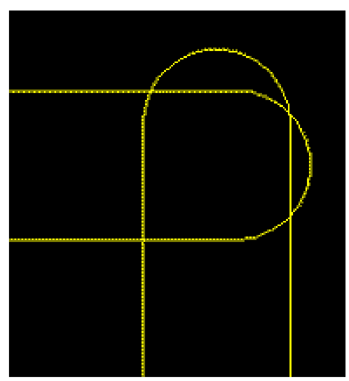
Join and Close will work on any of the following examples:



Perfect match



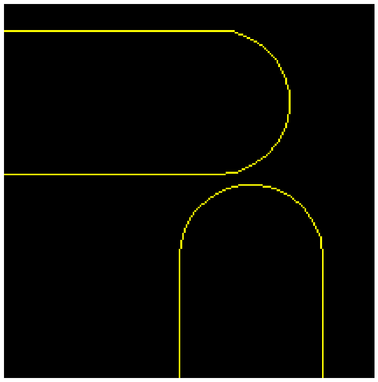
Lines touch



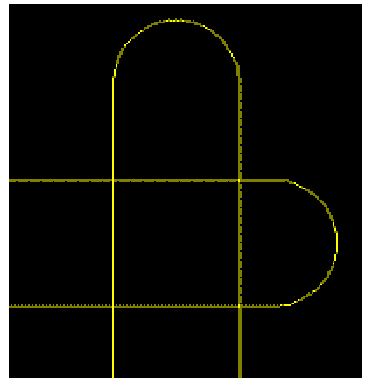
Lines cross slightly

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

Note:
The connection is available if both ends are not 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.

Joining 2D Lines and Copper Shapes

Use the Join command to connect any 2D lines or copper shapes that are “near enough.”

For more information, see “Join and Close 2D Lines and Copper Shapes”.

Procedure

1. Select the two 2D lines or copper shapes you want to join. The objects must be near enough.
2. Right-click and click the **Join** popup menu item.

- 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 do not cross, an additional segment is added

Results

If the command worked, you should see "2D Line" in the Status bar. You can now select the entire joined object.

If the command did not work, you should see the error message: "Join command aborted" in the Output Window.

Closing 2D Lines and Copper Shapes

Use the Close command to create a closed (and joined) shape that has contiguous lines. The objects must be "near enough" for the command to be effective.

For more information, see "Join and Close 2D Lines and Copper Shapes."

Procedure

1. Select the 2D lines or copper shapes you want to close. The objects must be near enough.
2. Right-click and click the **Close** popup menu item.

Results

If the command worked, you should see "2D Line" in the Status bar. You can now select the entire closed object.

If the command did not work, you should see the error message: "Close command aborted" in the Output Window.

Breaking an Object

You can break any 2D Line or copper shape that is not a single straight line or a single arc into multiple 2D Lines. The Break command creates a "break point" at every corner of the shape.

Restrictions and Limitations

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 the **Break** popup menu item.

Results

If the command worked, you should see "Multiple Selection" in the Status 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 a current design or any other design. SailWind Layout stores all of the objects with the Lines filter in the Library.

Procedure

1. Press the Shift key and select the item to select the whole shape rather than a segment or corner of the shape.

   You can select a board outline, 2-D lines, copper objects, keepout objects, and dimension objects. You cannot save text alone to the library. Text must be combined on page 642 with a 2D line in order to save it to the library.

2. Right-click and click the **Save to Library** popup menu item.

3. In the Save to Library dialog box, select the library you want, and type the name of the item.

4. Click **OK**.

   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 select a drafting item from a library and add it to your design.

Prerequisites

You can place a board outline, copper object, or keepout object from a library only while on-line DRC is off.

Procedure

1. On the Standard Toolbar, click the **Drafting Toolbar** button.

2. On the Drafting Toolbar, click the **From Library** button.

3. In the Get Drafting Item from Library Dialog Box, select an item from a library.

   Use the Library list to view drafting items in all libraries or in a specific library. Use wildcards on page 155in the Filter to limit the objects that appear in the Drafting Items list.

4. Click **OK**. The item attaches to your pointer.

5. Move the item to where you want and click to place the item.

Restriction:
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.

Related Topics

Saving a Drafting Item to a Library

Modifying Objects in a 2D Lines Library

In order to edit an object that is stored in a 2D Lines library, you must first bring that object into the design workspace, make the desired edits and then save it back into the library.

Procedure

1. In the SailWind Layout workspace, on the Drafting Toolbar, click the **From Library** button.
2. In the Get Drafting item from Library dialog box, select the desired object from the Drafting Items list and click **OK**. The item will attach to the cursor.
3. In the design workspace, click a location to place the item.
4. Make the desired edits using the commands available on the Drafting Toolbar and the right mouse button menus.

   Tip
   To save multiple objects that include lines and text, you must first select all the items, choose **Combine** from the right mouse menu, and then choose **Save to Library** from the right mouse button menu.

5. Once you have completed the desired edits, select the objects that make up the drawing item, right click and choose **Save to Library**.
6. You can choose to either overwrite the item or save it as a new item. Click **OK** to save the 2D line object back into the 2D Lines library.

Change the Width of a Trace or Drafting Object

During a design session, you can change the width of drafting object or a trace using one of two available methods.

Use one of the following methods changing the current width of traces or drafting objects:

Changing the Width of a Segment, Pair or Net Once It Is Routed

If necessary, you can change the width of a segment, pin pair, or net after routing it by changing its properties.

Procedure

1. Select the item, right-click and select the **Properties** popup menu item.
2. In the Drafting Properties dialog box on page 1328, in the Width box, type a new value.

Finding All Traces of a Similar Width

You can locate and change objects all traces of a similar width.

Procedure

1. Click the **Edit > Find** menu item.
2. In the Find on page 1378 dialog box, in the Find by list, select Line Width.
3. In the Line Type box, select Trace.
4. In the Line Width box, select the width and click **OK**.

Drawn Line Width

The *drawn line width* 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 category on page 1553in 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 the **Properties** popup menu item. Use the Find dialog box to search for and change all lines of a similar width.

Pasting Items by the Pointer Location

The default origin for pasting contents is the lower left corner of the area that encompasses all information in the buffer (clipboard). You can paste design items using the location of the pointer when the item is copied or cut.

Procedure

1. Click the **Tools > Options** menu item, and then select the Design category on page 1503.
2. In the “Move preference” area, click “Move by cursor location” and then click **OK**.
3. In the design, select the design item(s) to cut or copy.
4. Place the pointer at the new origin location and press Ctrl+X (for Cut) or Ctrl+C (for Copy).
5. Press Ctrl+V. The item appears on the cursor.
6. Find the new location and click to place the object.

Selection of 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

Select the outlines of a drafting object to move the outline and change the shape of the object.

Procedure

1. Right-click and click the **Select Anything** popup menu item.
2. Click the outline of the shape.
3. Right-click and click a command to alter the segment as needed.

Selecting Whole Drafting Objects

Select the outlines and fill of a drafting object to move the object or change the properties of the whole object.

Procedure

Use one of the following methods:

- With nothing selected, right-click and click the **Select Shapes** popup menu item and then click the object.
- With nothing selected, right-click and click the **Select Anything** popup menu item and then shift-click along the outline of the object.
- With nothing selected, right-click and click the **Select Anything** popup menu item and then drag a selection box over an outline segment.

Chapter 28
Clearances and Spacing

Read the topics that follow to learn more about the procedures for using the **View > Clearance** menu item to check distances between objects.

Viewing the Clearance Between Nets

Use Net-to-Net Clearance to detect the location of the smallest spacing between two specified nets.

Procedure

1. Click the **View > Clearance** menu item.

2. In the View Clearance Dialog Box, click **Net to Net**.

3. To center the minimum clearance marker in the workspace, select the “Pan to Minimum Clearance Marker” check box.

4. Select two nets.

   A minimum clearance marker displays the minimum clearance location between the nets in the design using the Selections color from the Display Colors dialog box.

Related Topics

Viewing the Clearance Between Items and Annotating Dimensions

Viewing the Clearance Between Items and Annotating Dimensions

Use Item-to-Item clearance to measure the spacing between items and annotate a PCB layout design with dimensioning that indicates critical or problem clearances.

Procedure

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 the **View > Clearance** menu item.

3. In the View Clearance Dialog Box, click **Item to Item**.

4. If you intend to annotate the design with the dimensions of the spacing between your two items, click **Options** to set the placement layer and style of the dimensions.

5. 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.

   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.

6. To place the dimensions in the design move the pointer to where you want the dimensions and click. If you do not want the dimensions in the design, click Cancel.

   Dimensions are added to the layers for dimensioning 2D Lines and Text as set in the Options.

Viewing the Clearance Between a Net and an Item

Use Net-to-Item Clearance to detect the location of the smallest spacing between the net and all objects.

Procedure

1. Click the **View > Clearance** menu item.
2. In the View Clearance Dialog Box, click **Net to Item**.
3. To center the minimum clearance marker in the workspace, select the “Pan to Minimum Clearance Marker” check box.
4. Select a net.

   A minimum clearance marker displays the minimum clearance location using the Selections color from the Display Colors dialog box.

Related Topics

Viewing the Clearance Between Items and Annotating Dimensions

Chapter 29
Copper Operations

Read the topics that follow to learn more about the methods used to add copper shapes to your design, including how to assign names to copper objects and how to create cutouts within the copper shapes.

Creating a Copper Shape

Copper is used to create heat sinks, shielding, and net bridges. You can assign nets to copper shapes.

Prerequisites

- 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 a copper plane. For more information, see "Creating a Copper Plane Manually" on page 667.

Procedure

1. On the Drafting Toolbar, click the **Copper** button.

2. Right-click and click a command to change the values of the drafting object. For more information, see "Set Values Before Creating a Drafting Object" on page 614.

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.

> **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 in the Options dialog box > **Text and Lines** category.

5. In the Add Drafting properties dialog box on page 1328, 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. For more information, see “Drafting Object Properties” on page 637.

   Is the resulting shape what you expected?

   - If the fill is not correct, see “The Fill of Copper, and Copper Planes” on page 617.
   - If the shape edges are not correct, see “Edge Precision of Drafting Shapes on page 615”.
   - If the shape needs to be modified, see “Drafting Object Properties”.
   - If you want to start over, see “Deleting a Drafting Segment or Object”.

Related Topics

Creating a Copper Cut Out

Assigning a Net to Existing Copper or Copper Planes

Routing To a Copper Shape

Routing From a Copper Shape

Creating Copper in the Decal

Bridging Nets with Copper

Cut Outs Absorbed by Copper

Creating Copper Chamfered Paths

Because some designs (such as RF designs) require specialized trace corner geometries, you can add chamfered copper as an alternative for regular traces. Using Chamfered Path, SailWind Layout squares or chamfers orthogonal corners. SailWind Layout also chamfers wide angle corners and acute corners.

As copper is created with an outline and a fill, you can specify a very narrow outline width to achieve very sharp corners. SailWind Layout creates all Chamfered Paths with a Solid Copper fill.

Prerequisites

This feature requires the Advanced Editing/RF license option on page .

Procedure

1. On the Drafting Toolbar, click the **Copper** button.

   If you have Online Design Rule Checking on page 461 enabled, in the prompt that appears, click **OK** to switch off Design Rule Checking.

2. Right-click and click the **Chamfered Path** popup menu item.

3. In the Add Chamfered Path dialog box, set the Chamfered Path Parameters on page 658 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.

   Note:
   To back up and remove the last corner, press the Backspace key.

7. To end the copper, either double-click or right-click and click the **Complete** popup menu item.

8. In the Properties dialog box, you can assign a net and modify other settings.

Results

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”.

Restriction:
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 when you select Chamfered Path - copper shape. Set the Chamfered Path parameters before you add the copper to the design.

Procedure

1. On the Drafting Toolbar, click the **Copper** button.

   If you have Online Design Rule Checking on page 461 enabled, in the prompt that appears, click **OK** to switch off Design Rule Checking.

2. Right-click and click the **Chamfered Path** popup menu item.

3. 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.

   ---

4. In the “Chamfered path width” box, type a width value for the overall width of the copper path.

5. 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.

6. Clear the “Chamfer corners with angles less than or equal to” check box to chamfer only 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.

7. Click **OK**.

Bridging Nets with Copper

You can create a physical connection between objects of two different nets with design rule checking enabled.

Restrictions and Limitations

- 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 the **Properties** popup menu item.
    - If the copper does not already exist, Create the copper shape. on page 655
2. In the Add Drafting or Drafting Properties dialog box on page 1328, select the “Bridge” check box.
3. Click the **Nets to bridge** button.
4. In the Net Association dialog box on page 1486, 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. The following exceptions will apply:

- 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.

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.

You can copy bridge coppers and they retain their bridge status, but their net associations are removed. For more information, see “Copied Bridge Copper in ECO Mode” on page 840.

CAUTION:
Bridged nets are combined when you export the design into the *.hyp* (PADS HyperLynx), *.ipc* (IPC356), and *.tgz* (ODB++) file formats.

Assigning a Unique Netname to Copper or Copper Planes

You can assign a unique netname to copper or copper planes, enabling you to assign rules and properties to the copper shape independent of the netlist.

Tip
To assign an existing netname to a selected copper shape, use the Drafting Properties Dialog Box.

Procedure

1. Select a shape, right-click, and then click the **Add Net** popup menu item.
2. In the Define Name of New Net Dialog Box, do one of the following:
    - To specify a custom netname, choose the Typed New Name option and type the netname in the Name of New Net box. The maximum netname length is 47 characters. You can use any alphanumeric characters except braces { }, asterisks *, spaces, question marks, or commas.
    - Choose the Autogenerated New Name option to use the default netname.

Results

The new name is added to the netlist, and you can access the net in the rules dialogs to add constraints to the new net. Because this action does not affect the schematic logic, the new netname is not recorded as an ECO operation.

Related Topics

Finding by Hatch Outline

Finding by Isolated Hatch Outline

Assigning a Net to a Copper Shape or Copper Plane

Creating a Copper Cut Out

Use copper cut outs to create voids inside copper shapes. Creating a cut out involves combining the fixed copper shape with the copper cut out.

Restrictions and Limitations

A cut out does not create a void in the outline of the copper shape. Create the copper as a polygon shape to create features in the outline.

Prerequisites

You must put the cut out on the same layer as the copper.

Procedure

1. On the Drafting Toolbar, click the **Copper Cut Out** button.
2. Create one or more cut out areas for the copper.

   See "Creating a Drafting Object" for more information. You can create the cut out before the copper or the copper before the cut out.
3. Right-click and click the **Select Shapes** popup menu item.

   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 in the Options dialog box > **Copper Planes** category > "Hatch and Flood subcategory" on page 1511.
4. Drag a box to group-select the copper and cut out(s).

   Note:
   Some circumstances do not 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 the **Combine** popup menu item. Copper inside the cut out area is automatically removed.

Results

Is the result what you expected?

- Do you need to move or edit a cut out? You will 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).

Related Topics

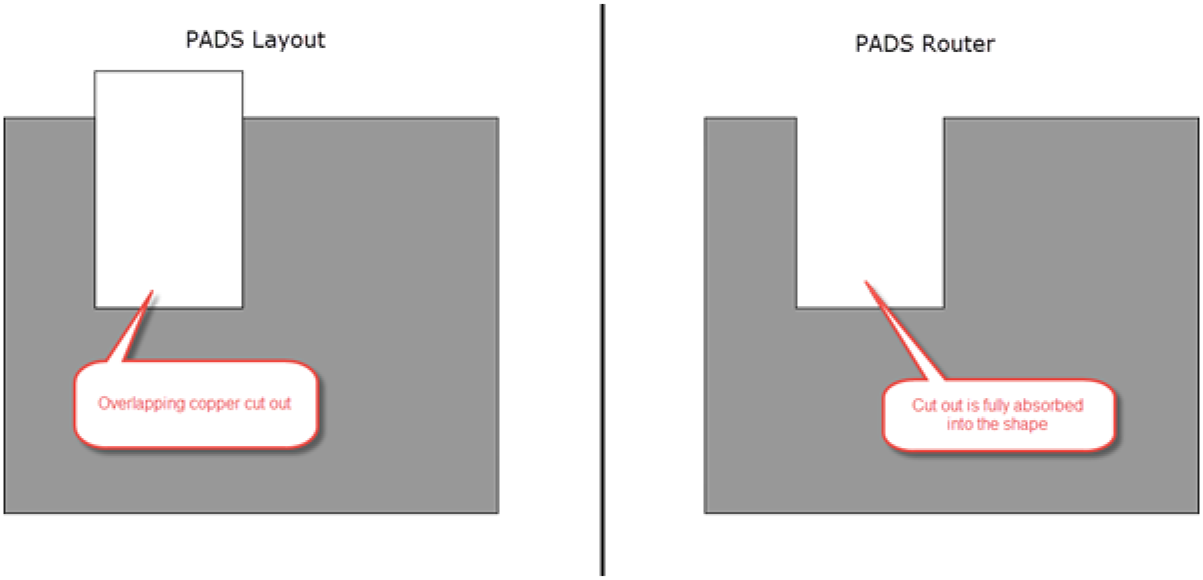
Cut Outs Absorbed by Copper

Cut Outs Absorbed by Copper

When you combine coppers and copper cut outs in SailWind Layout and take the design into SailWind Router, if any cut outs cross the copper outline, they are no longer cut outs. Instead, the software absorbs them into the copper shape.

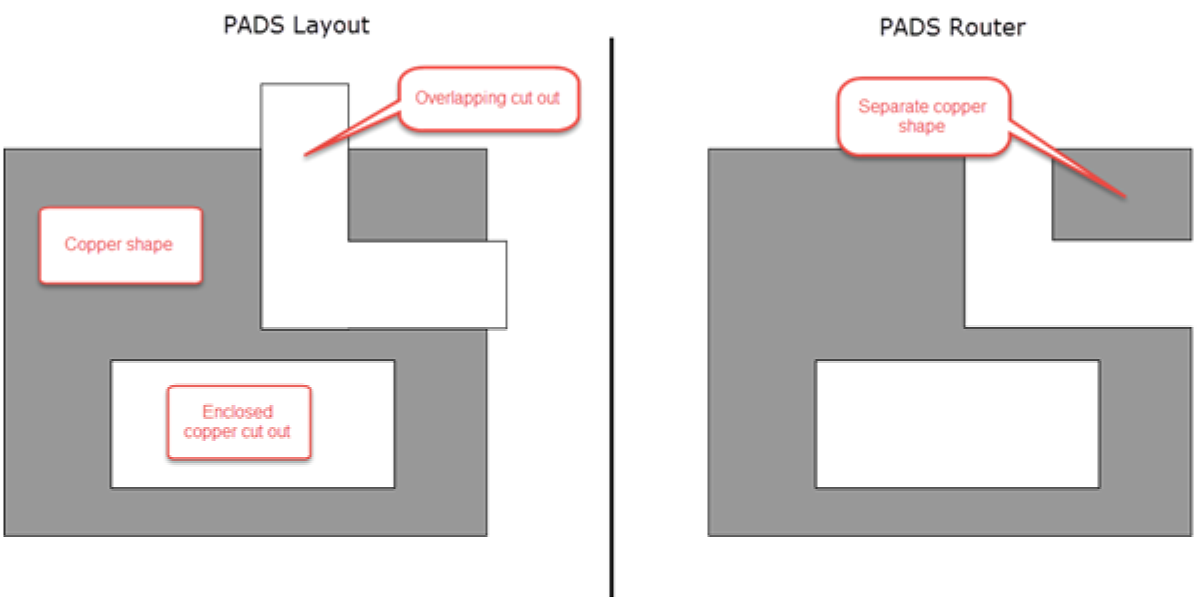
In some cases, copper cut outs are completely enclosed within the confines of a copper shape. If the copper cut out extends beyond the boundaries of the copper shape, however, and you merge the cut out with the copper shape in SailWind Layout (by right-clicking and clicking the **Combine** popup menu item), when you move the design over to SailWind Router, the software “absorbs” it into the outline of the copper shape.

Figure 78. Absorbed Copper Cut Out



If the cut out completely divides a copper shape, two or more separate copper shapes may result in SailWind Router.

Figure 79. Subtracted Copper Cut Outs in SailWind Router



Certain other changes take place:

- If you completely cover a copper shape with a cut out then combine the two, taking the design into SailWind Router deletes the copper shape in its entirety.
- SailWind Router removes any copper cut outs that lie outside the boundary of a copper shape.
- If a copper shape has two or more overlapping cut outs within its boundaries, the copper cut outs merge into one shape when you move the design to SailWind Router.

The software generates a report automatically when you open the design, detailing any instances where the software absorbs one or more shapes during the subtraction process.

Note:
If SailWind Router is running in the foreground, the link to the report displays in the SailWind Router Output Window. If SailWind Router is running in the background (for example, during an autoroute routine or during Latium Design Verification), the link to the report displays in the SailWind Layout Output Window.

Creating Nested Copper

You can create a copper shape within another "outer" copper shape. You can do so by creating a copper cut out inside the outer shape, combining it with the outer shape, and then creating the new nested copper shape inside the copper cutout.

Procedure

1. In the outer copper shape, Create a copper cut out on page 660 of appropriate size and shape, and combine it with the outer shape.
2. Create the new nested copper shape on page 655 within the copper cut out.

Results

The new nested copper shape displays inside the cut out of the outer shape.

Chapter 30
Assigning a Net to a Copper Shape or Copper Plane

You can assign any net in the design to copper or copper planes so that the net maintains proper connectivity as it transitions across many design objects.

Assigning a Net to Existing Copper or Copper Planes

You can assign a net to copper or a copper plane by selecting a design object with the net already assigned, thus assigning the copper to the net.

Procedure

1. Select the copper shape or copper plane.
2. Assign the net using one of the following methods:
   - Right-click and click the **Properties** popup menu item. In the Drafting Properties dialog box, in the Net Assignment area, choose a net from the list of nets in the design.
   - Right-click, and then click the **Assign Net By Click** popup menu item. Click a design object, such as a pin, via, trace, net, copper, or unroute, with the netname you want to assign to the copper.

Restriction:
If a net has already been assigned to the copper or copper plane, **Assign Net By Click** is unavailable on the popup menu. You must right-click and click the **Properties** popup menu item, and in the Drafting Properties dialog box, click the **Assign Net By Click** button.

Results

The status bar displays the netname assigned to the copper or copper plane.

Creating Copper or Copper Planes Based on the Net of an Object

If desired, you can assign a net to a new copper shape or pour area before you start drafting it.

Procedure

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 click the **Add Copper Area** or **Add Copper Plane** popup menu item.

   Restriction:
   If you select multiple design objects, the **Add Copper Area** and **Add Copper Plane** popup menu items are unavailable.

3. Create the copper or copper plane.

Results

The copper inherits the netname from the selected object.

Chapter 31
Copper Plane Operations

A *copper plane* is a closed, non-self-intersecting polygon that provides access to universally necessary nets, like power and ground. One design may use several copper planes. Most PCB designs locate planes 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. Such a configuration is called a *split plane*.

Copper planes 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 on page 667 using the drafting tools or automatically on page 668 using Create Copper Plane, using the board outline as a guide. After identifying a layer as a split/mixed plane layer and assigning nets to the layer, you can add plane areas.

For more information, see “Differences Between Copper Shapes and Copper Planes” on page 779, and “Split/Mixed Plane Layers” on page 683.

The following topics describe the methods used to create copper planes and how to assign nets to them. They also provide detailed information related to copper plane filling, splitting plane areas, thermal connectivity and troubleshooting.

Creating a Copper Plane Manually

Use a copper plane to create a copper area that avoids objects not connected to its associated net while simultaneously making thermal connections to objects that are connected to its associated net.

Flooded copper planes create automatic clearance to traces, pins, and other objects that do not share the same net assignment.

For more information, see “Differences Between Copper Shapes and Copper Planes” on page 779.

Procedure

1. On the Drafting Toolbar, click the **Copper Plane** button.
2. Right-click and click a command to change the values of the drafting object. For more information, see “Set 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 on page 1328that displays:
   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.

      When you create a copper plane on a split/mixed layer, you can select the “Show plane nets only” check box to limit the nets to only those assigned to the split/mixed plane layer in the Layers Setup.

   b. Make any other appropriate changes to the copper plane properties.
5. Click **OK**.
6. View the resulting shape.
   - If the gap between the pad and the copper for thermals or antipads is incorrect, see “Customizing Design Rule Thermals” on page 669 or “Customizing Design Rule Antipads” on page 670.
   - If the shape edges are not correct, see “Edge Precision of Drafting Shapes on page 615”.
   - If the shape needs to be modified, see “Drafting Object Properties”.
   - If you want to start over, see “Deleting a Drafting Segment or Object”.

Related Topics

Flooding Copper Planes

Creating a Copper Plane Automatically

You can create a copper plane 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.

Procedure

1. On the Standard Toolbar, set the active layer to either a non plane layer or a Split/Mixed plane layer in the Layer list.

2. Select the board outline.

3. Right-click and click the **Create Copper Plane** popup menu item.

4. In the Add Drafting dialog box, in the Net assignment area, assign a net to the copper plane using either the net dropdown list or the **Assign Net by Click** button.

   You can use the Show plane nets only check box to reduce the list of nets to only those assigned to the layer in the Layers Setup. You can also choose "None" from the dropdown list and associate a net later. If only None appears in the list, the net has not been assigned to the Split/Mixed plane layer in the Layers Setup. When you assign the net to the copper plane, pads associated with these nets will get thermal indicators if they are inside the appropriate polygons.

Tip
When you use Create Copper Plane, you may receive an error message indicating a self-intersecting polygon on page 625. This occurs because the generated plane area is proportionally smaller than the board outline and the software cannot create small miters and/or notches in the board outline using the specified line width. Correct this error by reducing the distance between the board outline and the plane. In the **Tools > Options** (menu item, **Copper Planes** category, **Hatch and Flood** subcategory, type a smaller value for the Auto Separate Gap setting. Alternatively, you can use a smaller line width for the copper plane (if you choose this method, you also need to reduce the copper hatch grid to match in order to maintain a solid fill for the copper plane. Generating more fill lines consumes more memory and could slow operations if the line width is greatly reduced).

Related Topics

Flooding Copper Planes

Customizing Design Rule Thermals

You can customize the default thermals on pins, virtual pins, and vias within copper planes. The default thermals depend 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.

Procedure

1. Choose the **Tools > Options** menu item and then select the **Copper Planes** category, then **Thermals** subcategory to view the Thermals options on page 1514.

   There are separate settings for Drilled (Through-hole) and SMT (surface mount) pads.

2. Set the width of the spokes in the Spoke width box.

3. Set the minimum allowed number of spokes in the Spoke minimum 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 plane. 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 "Add thermals to routed component pads" check box.

6. Open the Design Rules on page 1674. In any applicable level of the hierarchy of the design rules, in the Clearance on page 1167 rules, set the Copper-to-Pad and Copper-to-Via value according to the needs of your design.

Results

- After you fill the copper plane, a*therm.err* report generates if any pads receive fewer spokes than the Min. spoke value. For example, if the pad intersects the boundary of a copper plane, it may not be possible to create all four spokes.

- After filling a copper plane, if any pad fails to receive a thermal, use the Selection Filter to check for trace stubs at the pad. Small leftover trace segments attached to a pad prevent the pad from receiving thermals if the "Add thermals to routed component pads" check box is selected (in the Options dialog box, **Copper Planes** category, **Thermals** subcategory). Set the Selection Filter to Traces, Corners, and Tacks to select and remove trace segments attached to the pad.

Customizing Design Rule Antipads

You can customize the default antipads that appear around pins, vias, or drill holes within copper planes. Default antipads depend 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 copper plane or the net of the pad/via.

Procedure

1. Click the **Setup > Design Rules** menu item.

2. In the Design Rules on page 1674 dialog box, in any applicable level of the hierarchy of the design rules, in the Clearance on page 1167 rules, set the Copper-to-Pad, Copper-to-Via, and Copper-to-Drill values according to the needs of your design.

Tip
By applying the clearance to the net of the copper plane rather than the pads, you can affect all pads, vias, and drills within the copper plane. But by applying the clearance to the net of pads and vias rather than the copper plane, you can achieve different antipad spacings. Or you can use a combination - apply a general minimum clearance to the net of the copper plane and apply a larger clearance to nets of objects that need larger antipads.

Results

- When you are dealing with antipads, there are two nets involved. If you are not getting the results you expect, ensure that the clearance values of the other net involved are not conflicting with the values you have already changed. Larger clearance values always have precedence.

- With through hole pins and vias, the copper-to-drill value applies if the “Use design rules for thermals and antipads” check box is selected.

- The **Tools > Options** (menu item), **Copper Planes** (category), **Thermals** (subcategory) option “Remove unused pads” and “Use design rules for thermals and antipads” both affect the clearance of the antipad.

Related Topics

Customizing Design Rule Thermals

Associating a Net to a Copper Plane

You can associate a net with a copper plane at any point in the design process. This allows you the flexibility to assign a net to a copper plane at the time of creation, or to create the copper plane and then go back at some future time and associate a particular net with the copper plane.

Procedure

1. Select the copper plane outline.
2. Right-click and click the **Properties** popup menu item.
3. Click a net from the **Net** list.
4. Click **OK**.

Examples

Use the Drafting Properties dialog box to associate a net to a copper plane if:

- You created a copper plane before you assigned nets to the copper plane in the Layer Setup dialog box.

- You chose None from the Assign Net to Selected Polygon dialog box when you split a copper plane.

- You want to associate a different net to a copper plane.

Assigning Nets to Split/Mixed Plane Layers

After defining your plane layers, you can assign the appropriate net(s) to those layers to establish the desired connectivity in the design.

Use Assign Nets in the Layers Setup Dialog Box to assign one or more nets to the split/mixed plane layer.

Procedure

1. Click the **Setup > Layer Definition** menu item.
2. Select a split/mixed plane layer and click the **Assign Nets** button.
3. Select a net from the All Nets list.

> **Note:**
> You can assign a net to as many layers as required.

4. Click **Add** to move the net to the Assigned Nets list box.
5. Repeat the steps above to associate additional nets.
6. Click **OK** to close the Plane Layer Nets dialog box.
7. Click **OK** to close the Layer Setup dialog box.

Related Topics

Creating a Copper Plane Manually

Creating a Copper Plane Automatically

Creation of Split Planes

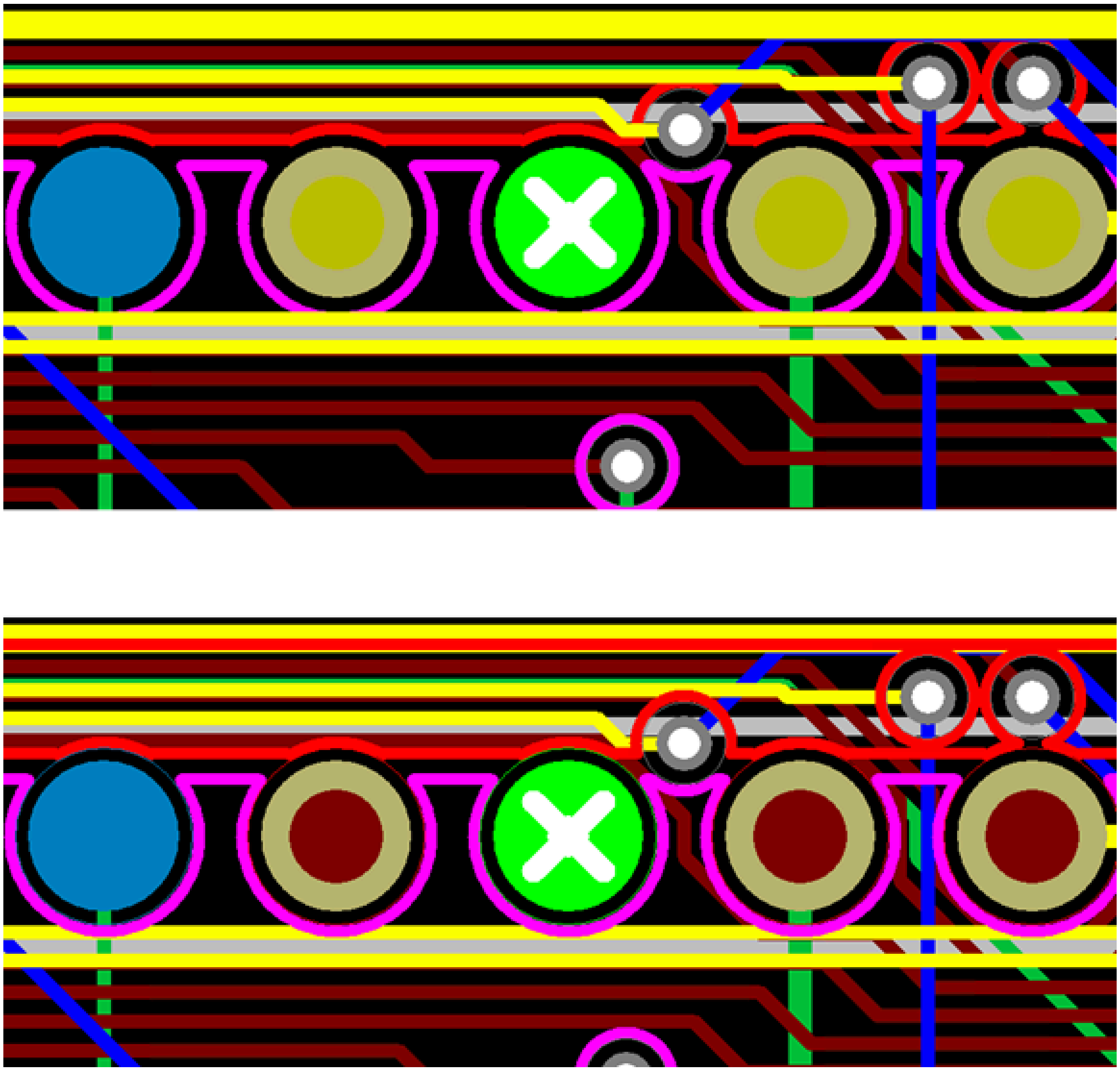
Control the Display of Thermals in Copper Planes

Use the Options dialog box > **Copper Planes** category > **Thermals** subcategory to control the display of thermal information.

For more information, see "Assigning Copper Plane Thermal Attributes."

Use the "Active Layer Comes to Front" check box in the **Tools > Options** (menu item), **Global** (category), General on page 1531 subcategory to view more detail for the thermal and antipad clearances. When this check box is selected, 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 copper plane net.

Figure 80. Active Layer Setting Disabled (Top) and Enabled (Bottom)



System-prompted Copper Plane Filling

When you use certain command operations, you are prompted to fill a copper plane. The fill prompts appear when you verify your design or run CAM generation commands.

Verify Design

When you use the **Tools > Verify Design** menu item, and you perform a Copper Plane check (and also select the “Check clearance and connectivity” option in the Setup), the software checks for proper setup, creates the copper plane (if only one net is assigned), if necessary. The software then prompts you to fill the copper plane automatically. For more information, see “Verify the Design.”

Run CAM

When you run CAM on a split/mixed plane layer, the software checks for proper setup and creates the copper plane if necessary. The software then prompts you to fill the copper plane automatically. For more information, see “Creating All Outputs.”

Troubleshooting Copper Plane Fills

Copper plane fills require special attention to assure that the connectivity matches the design intent.

Solution

The following situations can prevent a copper plane from filling.

No pads of the net are located within the copper plane

Ensure that at least one pad of the net assigned to the copper plane is within the shape.

No drilled pads are located within the copper plane

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 copper plane with thermals. Select the pad(s), right-click and click the **Properties** popup menu item. 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.

No pads or vias located within the copper plane have the Plane Thermal property enabled

Enable the Plane Thermal property for pads and vias to allow them to connect to the copper plane with thermals. Select the pad(s), right-click and click the **Properties** popup menu item. 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).

No copper plane is defined

Ensure you can see the outline of the copper plane.

Multiple copper planes are intersecting or defined on top of each other

One of the copper planes must take priority and you must create a cutout of the other plane. You might benefit from enabling the “Create cutouts around embedded copper planes” option found by choosing the **Tools > Options** menu item, Copper Planes (category), “Hatch and Flood” on page 1511 subcategory. For information about giving a copper 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 copper planes and greatest for the outer copper planes.

You have modified the copper plane outline while the copper plane area is filled

Return the plane area to the basic plane outline using the modeless command PO before modifying the shape of the outline.

You have embedded/nested copper planes

You must create the largest copper plane first, then the smaller inner copper plane(s) next. (As long as you select the **Tools > Options** menu item, then **Copper Planes** (category), **Hatch and Flood** (subcategory), then "Create cutouts around embedded copper planes" check box, this automatically assigns the correct flood priority and creates the required copper plane cutout within the larger split plane.)

If you neglected to select the "Create cutouts around embedded copper planes" option or you simply created the inner copper plane area first, the copper plane areas have conflicting flood priority values and are unable to fill. You must decide the order in which the copper planes should be filled and adjust their flood priorities accordingly. For information about giving a copper plane a flood priority, see the Flood Priority setting in the "Flood and Hatch Options dialog on page 1381" box. If you have only one inner copper plane, there is a quicker way to give it the higher flood priority. Use the PO modeless command to return the plane to outline mode. Select the plane area shape, right-click and click the **Bring to Front** popup menu item. The system recognizes the new flood priority and fills the copper plane.

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.

Your minimum hatch area setting is too large

In the **Tools > Options** (menu item), **Copper Planes** (category), "Hatch and Flood" on page 1511 subcategory, check the "Min. Hatch Area" and ensure the value entered is not larger than the copper plane size.

Your keepout is restricting the copper plane

Assign a color to Keepouts for all layers and ensure they do not restrict copper planes for the location and layer.

You have hidden the fill

Ensure the fill is visible. **Tools > Options** (menu item), **Copper Planes** (category), "Hatch and Flood" on page 1511 subcategory, in the Hatch View area, click Normal instead of No Hatch.

You have set Thermal options to no connect

Ensure that drilled and non-drilled pads have orthogonal, diagonal, or flood over settings in the Thermals Options on page 1514.

You have 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 Add thermals to routed component pads check box is selected in the Thermals Options on page 1514.

Troubleshooting Thermal Results

When you fill (flood/pour) a copper plane, a Thermal Relief Errors Report (*therm.err*) might appear in your default text editor. This file appears when pads within the copper plane do not receive the minimum number of spokes according to your settings in the **Tools > Options** menu item, Copper Planes (category), Thermals (subcategory).

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 plane outline width and hatch grid are too large for the copper to flood between the component pins, etc.

Interpreting Thermal Results

You can interpret the thermal results to detect problematic thermal issues in your design.

If results are not as expected, consider the following:

- If a large number of pads and/or vias have zero thermal spokes being generated, there may be a copper plane that is not filling. For more information, see “Troubleshooting Copper Plane 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 copper plane with thermals. Select the pad(s), right-click and click the **Properties** popup menu item. 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).

Clearances Between Copper Planes and the Board Outline

A number of factors can influence the way you construct copper planes and how they relate to other objects in a design.

Plane Edge Setback

For any plane area, it is desirable for the edge of the flooded plane to be set back slightly from the edge of the board. This prevents the edge of the plane from being exposed during the board trimming process and eliminates the potential of the plane being accidentally shorted to another design or packaging object.

There are two important measurements that you must consider when auto-generating a plane area:

- The distance between the board outline and the plane area outline
- The distance between the board outline and the copper within the plane area after flooding (hatch outline)

Board Outline

A drawn 2D line represents the board outline. The Default Width setting on the **Text and Lines** category of the Options dialog box determines the width of the line at the time of generation. You can change the board outline width by selecting the board outline and changing the width in the Drafting Properties dialog box.

The line used to represent the board outline is intended to be a visual reference that indicates the perimeter of the board in the design workspace as well as on the CAM plots. Changing the width of this line does not change the dimensions of the board as the actual board edge is at the centerline of this 2D line.

Plane Area Pour Outline

You can create a plane area by defining its pour outline. When you create the plane area pour outline, you specify the width of the line that represents the visible boundaries of the shape. The outer edge of this outline represents the maximum extents of the copper when the shape is flooded. The Board to Copper clearance rule influences how the design references the edges of this shape and can modify the extents of the copper flooding when required.

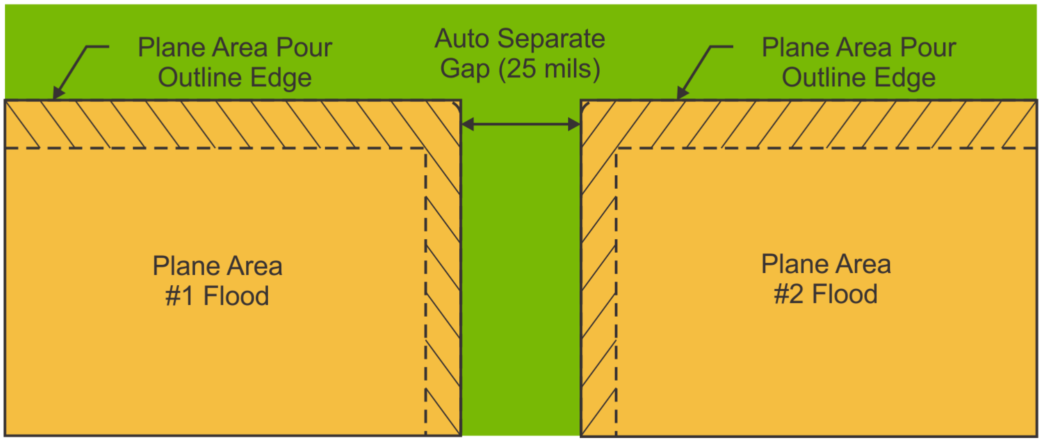
For more information on how plane area pour outlines are created, see Edge Precision of Drafting Shapes on page 615.

Auto Separate Gap

You can split a single plane area shape into two or more shapes. When you divide the plane area, you can control the desired spacing between these multiple plane area shapes by specifying a value for the Auto Separate Gap (choose the **Tools > Options** menu item and click the **Copper Planes** category > **Hatch and Flood** subcategory to access this setting).

The lines used to create the plane area pour outlines are Drafting objects and have width. If you have specified a value for the Auto Separate Gap, this value is used to compute the distance between the edges of these lines. Therefore, if you specify a 25 mil value for the Auto Separate Gap, the plane area pour outlines will be separated from one another by a distance of 25 mils. This distance is measured between the outer edges of the lines used to draw the plane area shapes.

Figure 81. Auto Separate Gap



CAUTION:
When multiple plane areas exist on the same layer, the flood priority assignments determine the final separation gap (see Flood Priority Settings).

When auto-generating plane areas using the board outline shape, the Auto Separate Gap is considered during the calculations used to determine the distance that a plane area will be set back from the edge of the board.

Restriction:
If a value of 0 is specified for the Auto Separate Gap, a non-editable system default value of 10 mils is used in this calculation.

For additional information on how split planes are created, see "Split/Mixed Plane Layers" on page 683.

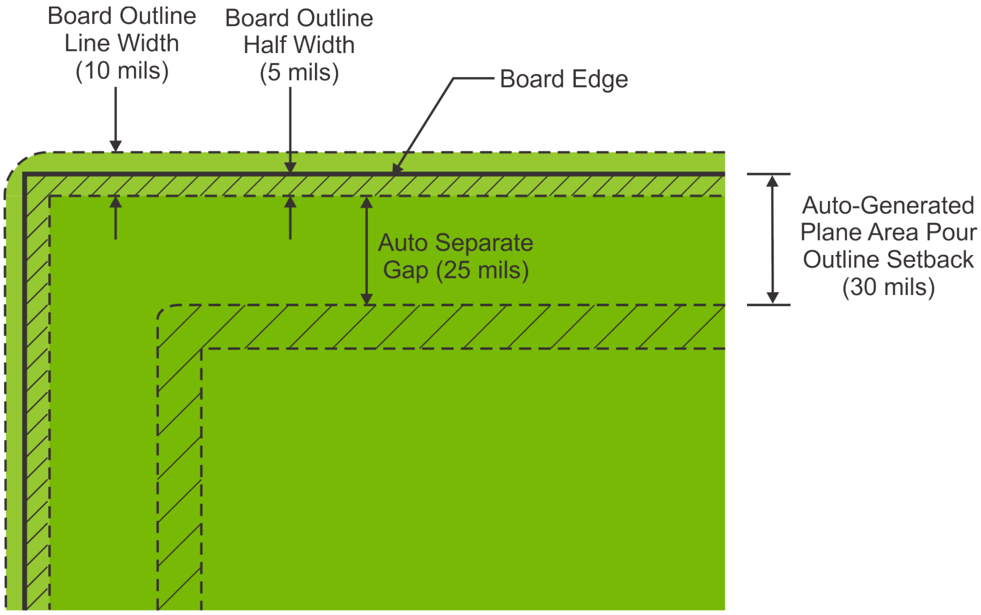
Plane Area Pour Outline Setback from the Board Outline

The line that draws the board outline is a Drafting object and thus it has a width, so any plane areas sharing a boundary with the board outline use the Auto Separate Gap value to determine the minimum distance that they must pull back from the board edge.

This setback distance is measured from the inside edge of the line used to draw the board outline. Since the board edge is at the center of the line defining the board outline, half of this line width is inside the edge of the board, so the resulting plane area pour outline setback distance is the sum of the value specified in the Auto Separate Gap plus one half of the line width used to draw the board outline.

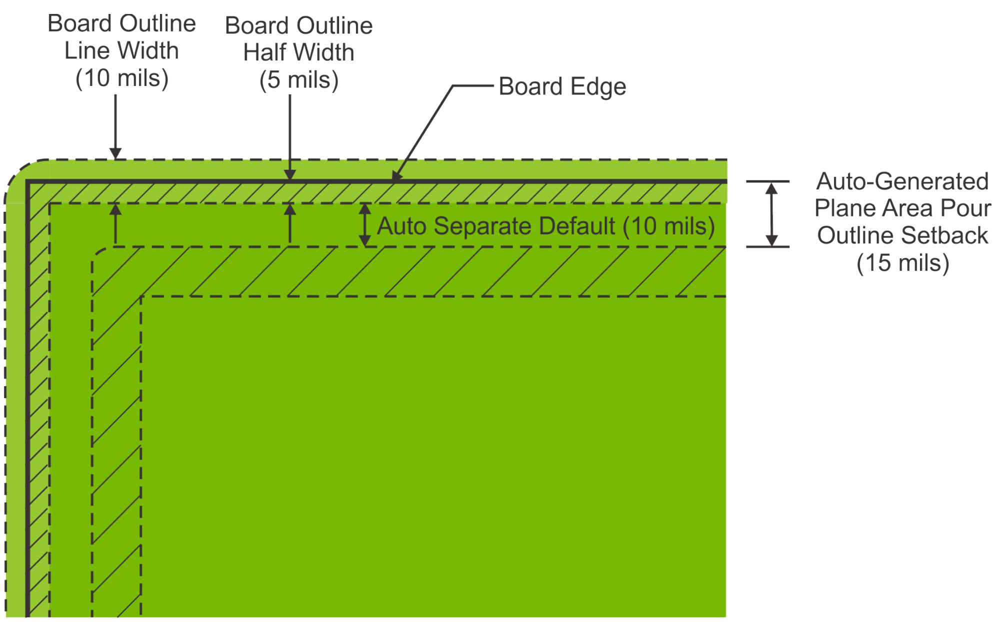
Example: If the board outline width is 10 mils and the Auto Separate Gap is set to 25 mils, then the resulting plane area pour outline setback distance would be the Auto Separate Gap value (25 mils) plus half of the line width value (5 mils) for a total distance of 30 mils from the board edge.

Figure 82. Plane Area Pour Outline Setback from the Board Outline



Exception Example: If no value is specified for the Auto Separate Gap, the system uses the system default value and places the plane area pour outline 10 mils from the inside edge of the line used to draw the board outline, so the resulting plane area pour outline setback distance would be the system default value (10 mils) plus half of the line width value (5 mils) for a total distance of 15 mils from the board edge.

Figure 83. Plane Area Pour Outline Setback from the Board Outline Exception



Plane Flood Hatch Outline Setback From the Board Outline

When plane areas are in proximity to the board edge, the Board to Copper clearance value in the design rules normally controls the plane flood hatch outline setback. However, if the results of the calculation that determines the plane area pour outline setback has a value larger than the Board to Copper design rule, it takes precedence.

Plane areas are flooded based upon the following conditions:

1. If the Board to Copper value is larger than the Auto Separate Gap + half the board outline width, then the plane flooding boundary (hatch outline) is pulled back and the copper flood is set back from the board edge to a distance equal to the Board to Copper Clearance value.

2. If the value set in the Board to Copper Clearance is equal to or smaller than the Auto Separate Gap + half the board outline line width, then the plane flooding boundary is not reduced and the area is flooded to the existing plane shape edge.

For additional information related to how the plane flood hatch outlines are filled, see "The Fill of Copper, and Copper Planes" on page 617

Examples of Plane Flooding Setback from the Board Outline

SailWind Layout calculates plane flooding setback distances based upon various design conditions. The following table provides examples of how plane flooding setback distance is calculated.

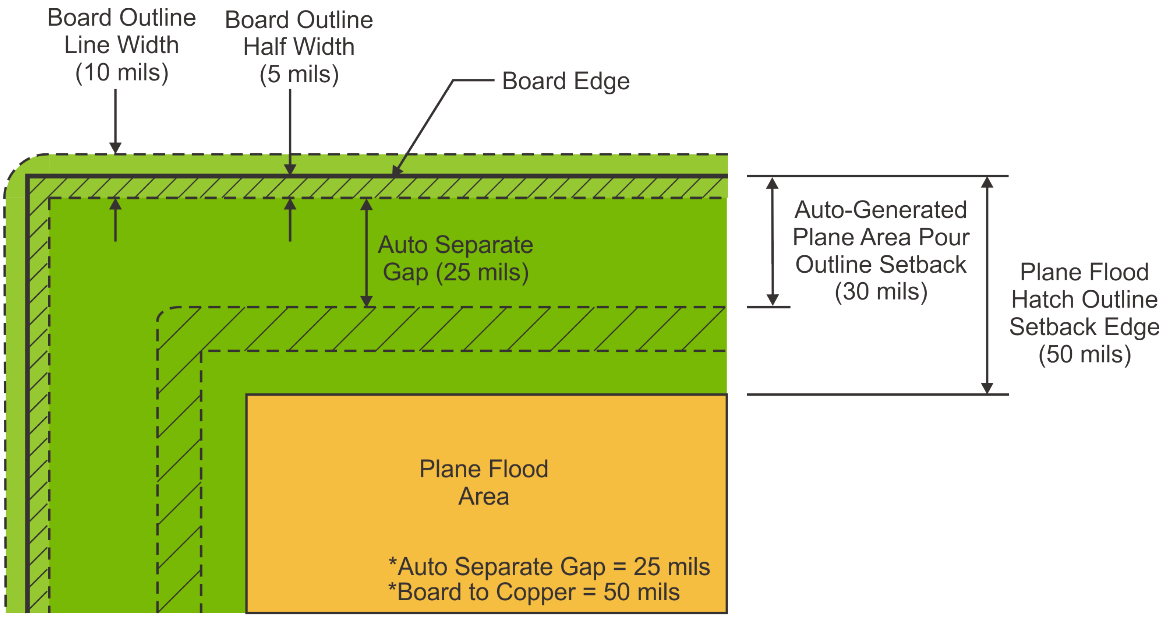
Table 110. Example Data for Determining Plane Flooding Setback Distance

Parameter	Example A	Example B	Example C	Example D
Board Outline Width	10 mils	10 mils	10 mils	10 mils
Auto Separate Gap	25 mils	0	25 mils	0
Board to Copper	50 mils	50 mils	0	0
Results				
Distance to Pour Outline	30 mils	15 mils	30 mils	15 mils
Distance to Hatch Outline	50 mils	50 mils	30 mils	15 mils

Example A

The Board to Copper value is greater than the calculated plane area pour outline setback, so the Board to Copper value is used for the hatch outline setback.

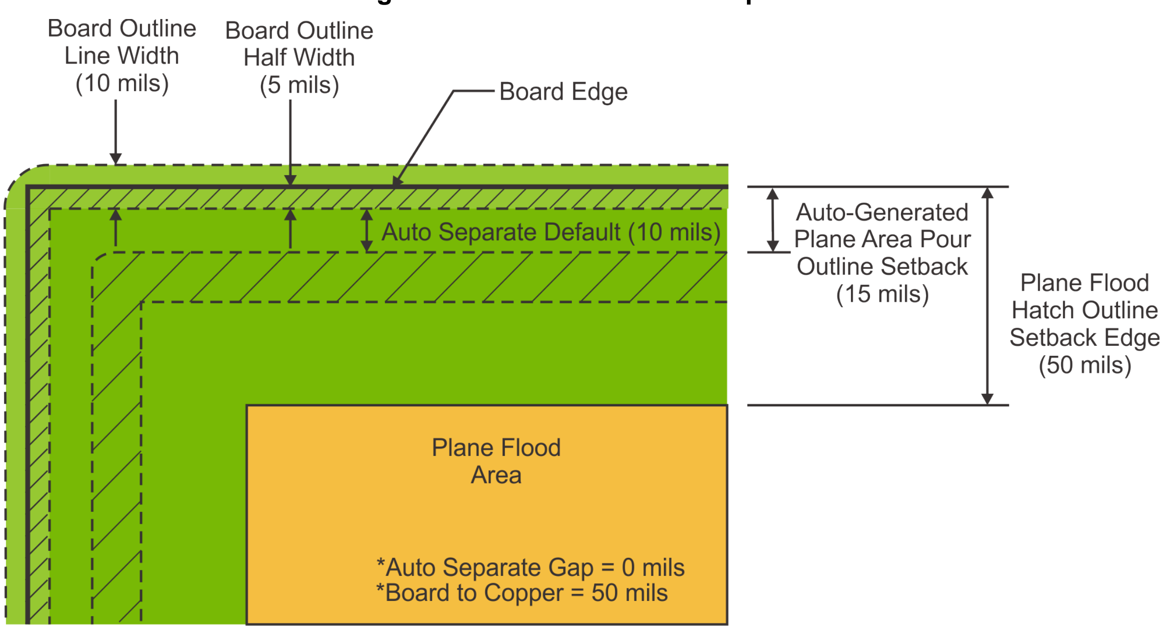
Figure 84. Plane Setback Example A



Example B

The Board to Copper value is greater than the calculated plane area pour outline setback (using the Auto-Separate Gap default value), so the Board to Copper value is used for the hatch outline setback.

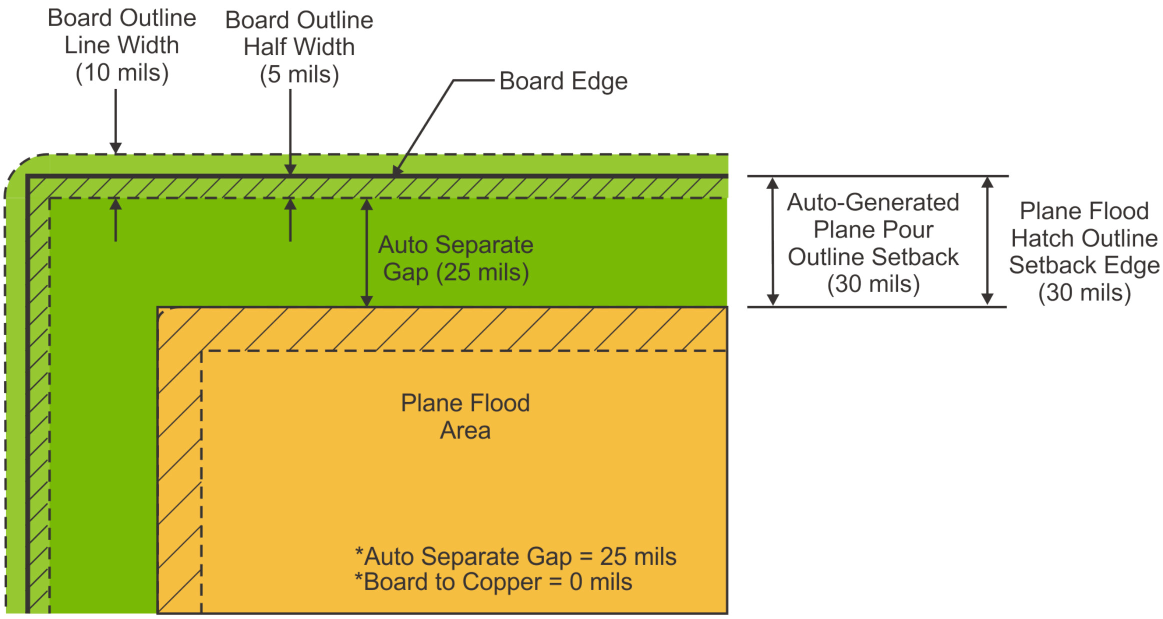
Figure 85. Plane Setback Example B



Example C

The Board to Copper value is equal to or smaller than the calculated plane area pour outline setback, so the plane area pour outline setback value is used for the hatch outline setback (coincident with the pour outline).

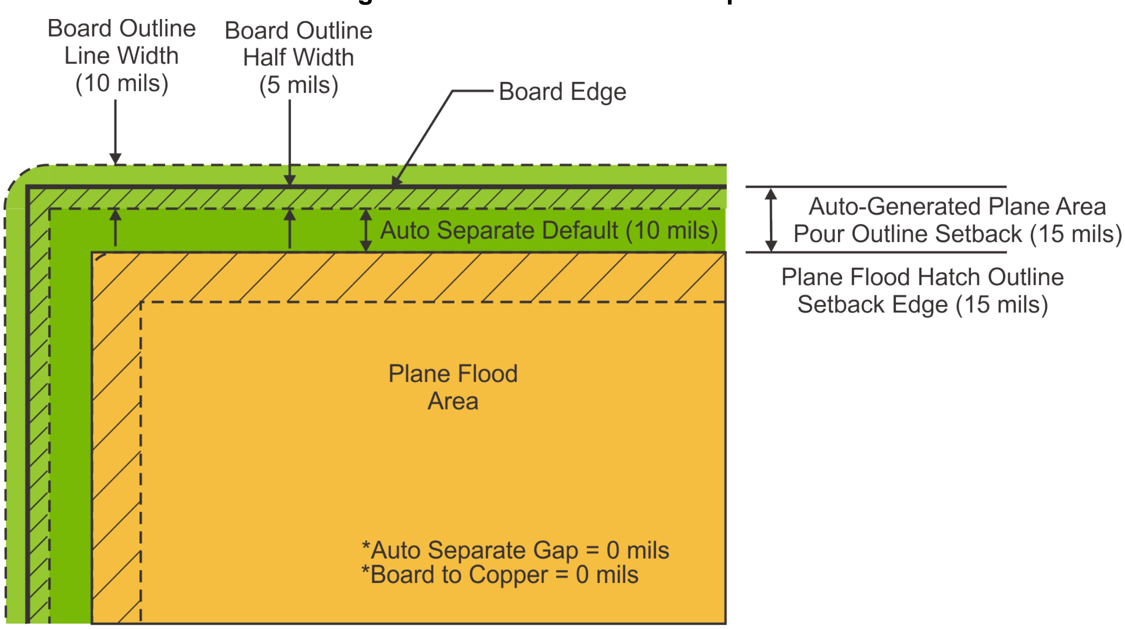
Figure 86. Plane Setback Example C



Example D

The Board to Copper value is greater than the calculated plane area pour outline setback (using the Auto-Separate Gap default value), so the Board to Copper value is used is used for the hatch outline setback.

Figure 87. Plane Setback Example D



CAUTION:
As the flooding operation can reduce the actual flooded area, always check plane connectivity after the flooding operation to be sure there are no discontinuities or starved thermals.

Split/Mixed Plane Layers

SailWind Layout supports the creation of split and mixed plane layers. A split plane layer is a plane layer with one or more isolated areas of copper that have different net assignments. A mixed plane layer is a plane layer with one or more copper planes and any number of signal routes.

In many designs it is common to require large copper areas for voltage and ground nets. To do this, create an internal layer dedicated to a single net (typically power or ground nets). Although it is common to have one plane layer dedicated to a single net, designs with multiple voltage requirements require you to separate, or split, the plane layer into isolated areas. When designing very dense designs, you can use split/mixed plane layers for normal signal routing.

Setup > Layer Definition menu item includes an option for identifying split/mixed plane layers and the 2-D drafting commands contain options for creating closed shapes to define copper planes and voids (areas with no copper). When you route on a layer identified as a split/mixed plane layer, clearances are automatically created around the trace and pin pair: the traces are actually plowed through the copper plane.

Copper plane operations also take full advantage of design rule-driven spacing and thermal relief generation. When you create copper planes with embedded traces, their spacing can be based on custom thermal clearances, design rule clearances or global clearances.

If you use split/mixed planes and you also route using SPECCTRA, see "SPECCTRA and Split/Mixed Planes".

Creation of Split Planes

You can create more than one copper plane on a split/mixed plane layer.

There are three recommended methods to split a plane layer:

- **Creating Multiple Copper Planes** — For more information, see "Creating a Copper Plane Manually" on page 667, and "Creating a Copper Plane Automatically" on page 668.
- **Separating Copper Planes Automatically** — Use the Auto Plane Separate tool to split a copper plane that covers the whole layer or a large area into two or more copper planes. For more information, see "Separating Copper Planes Automatically" on page 683.
- **Creating an Embedded Copper Plane** — Embed a copper plane within another copper plane. For more information, see "Embedded Copper Planes" on page 685.

Separating Copper Planes Automatically

Use the **Auto Separate** button to divide a copper plane into two new copper planes. Using Auto Separate, you draw a path from one side of a copper plane to another side of the same plane. This path defines the center of the separation between the two new copper planes and divides the copper plane into two distinct shapes.

Restrictions and Limitations

- You must start and end the separation on the same copper plane polygon.

Procedure

1. On the Standard Toolbar, in the Layer list, click the split/mixed plane layer.

2. Use one of the following:

   - For object mode, right-click and click the **Select Anything** popup menu item. Click the starting point of the split on a segment of the copper plane. 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 Copper Plane Separate** button. Click to indicate a starting location somewhere on the perimeter of the copper plane polygon.

3. Double-click at the ending point on the copper plane outline to complete the Auto Separate command.

   The Assign Net to Selected Polygon dialog box displays and automatically selects one of the new copper planes. The netnames that appear in the dialog are those that were assigned to the Split/Mixed layer in the Layers Setup.

4. From the Choose Existing Net area, click a net to assign to the selected copper plane.

   This assigns the net, deselects the copper plane, and selects the other copper plane.

   Tip
   No segments of the auto-separate path may be outside of the polygon or the message “All segments must be within the copper plane polygon” appears.

   ---

5. Click a net for the other copper plane. This assigns the net, deselects the copper plane and closes the dialog box.

   Tip
   Use the **Tools > Options** (menu item), Copper Planes (category), Hatch and Flood on page 1511 (subcategory), Auto Separate Gap option to define the distance between the two copper planes.

   ---

6. 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).

Embedded Copper Planes

Embedded copper planes enable you to create multiple copper planes on a specific layer and control how their constructions interact with each other.

You can embed copper planes (create a copper plane within another copper plane) using one of the following methods:

Creating an Embedded Copper Plane from the Outermost to the Innermost Area

When creating an embedded copper plane, work from the outermost area to the innermost area to ensure that all areas flood as expected.

Procedure

1. Create the outermost copper plane by drawing a copper plane as described in "Creating a Copper Plane Manually," and "Creating a Copper Plane Automatically."
2. Create the inner copper plane using the **Copper Plane** button on the Drafting Toolbar.
3. Continue nesting copper planes until you define all the embedded copper planes for the layer.

Tip
The embedded copper 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.

Creating an Embedded Copper Plane by Prioritizing, Adding Cut Outs and Combining

You cannot ensure correct flooding if you draw the innermost copper planes before the outermost copper planes. To ensure proper flooding of embedded copper planes, you can perform one of two methods: You can change the flooding priority of the embedded copper plane, or you can create a copper plane cut out around the perimeter of an embedded copper planes and then combine it with the outer copper plane.

Procedure

Choose one of the following:

- Select the embedded copper plane, right-click and click the **Bring to front** popup menu item. This sets the embedded copper plane to a lower "Flood priority", ensuring it floods before the outer copper plane.
- Alternatively, perform the following steps:

i. Create a copper plane cut out on the outside perimeter of all embedded copper planes. (For more information, see Creating a Copper Plane Cut Out.)

ii. Select the copper plane cut out polygon shape.

iii. Press the Ctrl key and select the next outer copper plane.

iv. Right-click and click the **Combine** popup menu item.

Creating a Copper Plane Cut Out

A copper plane cut out is a closed, non-self-intersecting polygon that defines a void within a copper plane. Creating a cut out involves combining the copper plane shape with the copper plane cut out shape.

Tip
You can create the cut out before the copper plane or the copper plane before the cut out.

Restrictions and Limitations

- Although it appears to create a void in the edge or outline of a copper plane when in hatch outline mode, a copper plane cut out does not actually alter the copper plane outline. Create the copper plane outline as a polygon shape to create features in the outline.
- You must put the copper plane cut out on the same layer as the copper plane.

Procedure

1. On the Drafting Toolbar, click the **Copper Plane Cut Out** button.
2. Create one or more cut out areas for the copper plane.

   See "Creating a Drafting Object" for more information.
3. Right-click and click **Select Shapes**.

   Tip
   If you cannot see a cut out within the copper plane shape because the shape is flooded, type the modeless command **PO** to return the copper plane to pour outline display mode. You will need to re-flood the copper plane once you have combined the cut out with the copper plane.
4. Drag a box to group-select the copper plane and cut out(s).

   Note:
   Some circumstances do not allow for dragging a group selection box. Instead, use Ctrl +click to select the copper plane and the cut out.

5. Right-click and click the **Combine** popup menu item.

6. "Flood the copper plane" on page 780.

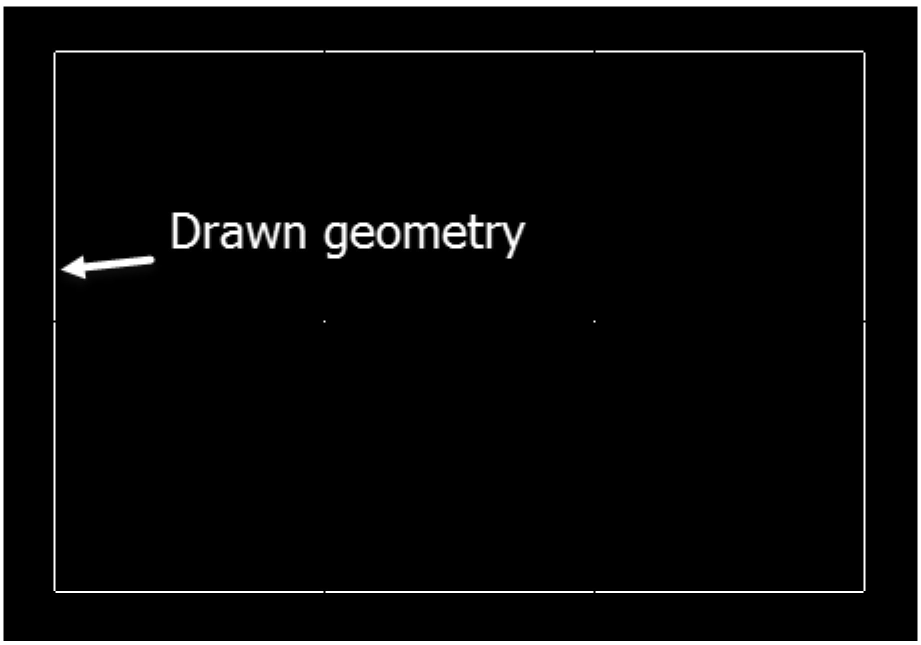
   The copper plane area inside the cut out area is automatically removed. If you need to move or edit the copper plane or the cut out, be aware that any modifications you make to the copper plane shape (move, rotate, miter, and so on,) also affect the combined cut out(s). You may need to uncombine it first. For instructions, see "Uncombining Drafting Objects" on page 643.

Results

CAUTION:
Starting with VX.2.3, copper floods to the inner edge of completed cut out outlines instead of the centerline. Prior to VX.2.3, the cut out would grow by half the line width since copper would flood to its centerline rather than the inner edge which was the intended boundary.

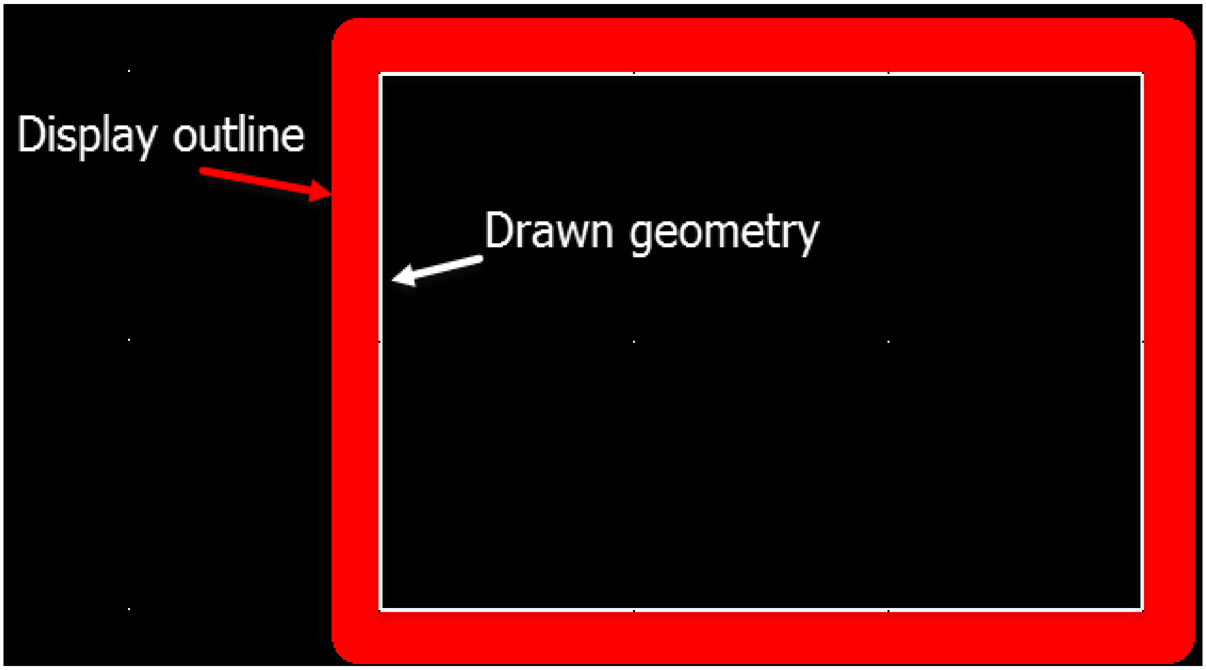
As you draw the cut out shape, the drawn geometry appears in a single screen pixel line.

Figure 88. Cut Out Outline While Drawing the Shape



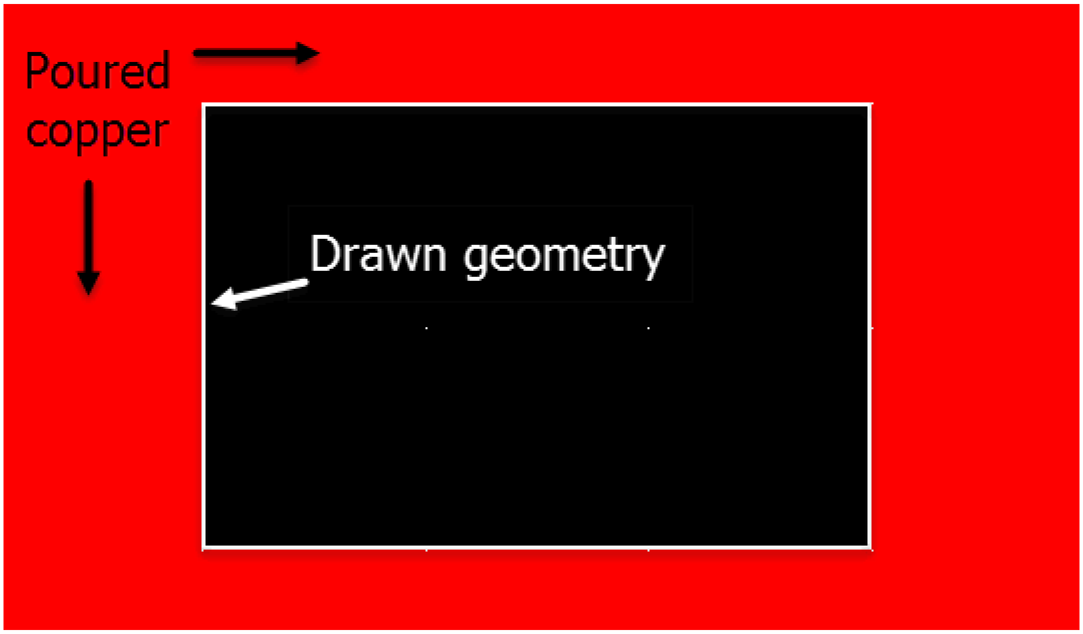
Once completed, the outline uses a real line width but is positioned completely outside the perimeter of the drawn geometry to help visualize the cut out prior to flooding.

Figure 89. Composite of Drawn and Display Outlines



Once the copper is poured, it pours to the perimeter of the drawn geometry - the inside edge of the cut out display outline.

Figure 90. Pour Surrounding Cut Out Geometry



Assigning Copper Plane Thermal Attributes

A thermal setting is assigned to all pad stacks in the design automatically.

Use the Plane Thermal check box in the Pin Properties on page 1632, Via Properties on page 1779, and Jumper Pin Properties on page 1422 dialog boxes to determine whether the pad stack is an eligible thermal candidate. Turn the Plane Thermal check box off if you do not want any type of thermal relief on any layer of the pad stack.

For more information, see “Generating Thermals”.

Related Topics

Control the Display of Thermals in Copper Planes

Discarding Copper Plane Data on Save

If desired, you can discard copper plane data and save only the plane polygons when you save the design file.

Procedure

1. Click the **Tools > Options** menu item, **Copper Planes** (category), **Hatch and Flood** (subcategory).
2. In the Save to PCB File area, select the “Copper Plane polygon outlines” option and click **OK**.
3. Click the **Tools > Copper Plane Manager** menu item to flood the copper plane on page 780.
4. Save the design.

   The “Discarding Copper Plane Data Dialog Box” on page 1314 appears.
5. Click **Proceed** to save only the copper plane polygons. Click **Save All** to save all copper plane data and change the option for future saves.

Display of Connections for Pads Connected to a Copper Plane

You can hide the connection lines (ratsnest or unroute connections) of nets assigned to a copper plane. Plane nets are typically power or ground nets; hiding them can help to clear the design space and reduce the connections to those that require more difficult routing.

Plane net connections can be made by through-hole pins connecting directly to the copper plane, or through fanout (short trace ending with a via) connections. These nets do not usually require long traces and are often not important in the strategic placement of the part - unless the copper 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.

For more information, see “View Nets Dialog Box”.

Chapter 32
Routing The Design

Read the topics that follow to learn more about routing connections in the design.

Routing Setup Considerations

Consider certain SailWind Layout setup options before you begin routing your design.

- Angle Modes
- Starting Layer for Routing
- Via Types
- Via Modes
- Trace Width
- Length Minimization
- Routing Colors

- Display of Trace Width
- Route and Unroute Protection

Angle Modes

Angle modes constrain the trace angle as you route the design and also impact the entry into pads. All routing modes are subject to the angle constraints for pad entry that you set in the Line/Trace area in the Design category of the Options dialog box on page 1503. As shown in the table below, the routing modes are Orthogonal, Diagonal, and Any Angle. The modes determine how a trace follows the cursor from corner to corner and enters a pad on completion. To set these modes quickly type the AO, AD, or AA modeless commands on page 83. All angle modes are constrained by the grid size if “Snap to grid” is enabled for the “Design grid” in the Grids options on page 1537. The Status dialog box on page 1733 is a useful tool for easy access to toggling the Snap to grid feature.

Table 111. Angle Modes

Mode	Description
Orthogonal (AO)	Prevents you from entering diagonal lines and 45-degree corners.
Diagonal (AD)	Limits you to 45- and 90-degree angles on the grid.
Any Angle (AA)	Allows placing routes at any angle.

Starting Layer for Routing

To set the current routing layer before you start routing, select the layer from the Layer list on the Standard Toolbar. A layer's current Routing Direction setting appears next to its entry in the Layer list. When you select a connection and select a routing mode, the route begins on that layer.

When you end a route on a layer different from where you began it, the layer you ended on becomes the active layer and appears in the Layer list. The next connection you select starts on this active layer. You can set a different active layer by using the list or by using the L modeless command on page 83: type L; type the Layer number, and press the Enter key.

Via Types

A via is a drilled and plated hole used to connect copper objects on different layers of a printed circuit board.

In general, there are two types of vias:

- **Through-hole vias** — Vias that are drilled through every layer of the board.
- **Partial vias** — Vias drilled through a select number of layers. Partial vias include:
    - **Blind vias** — Vias that pass through an outer layer and connect to an inner layer.
    - **Buried vias** — A partial via that passes through only the inner layers.

Vias have a variety of uses in a layout design:

- **In Routing** — The most common use for vias is in routing traces, to continue the connectivity of traces to other layers. When you add a via during routing, SailWind Layout connects it to the trace and considers it a routing via.

- **In stitching and shielding** — Vias used aside from routing are sometimes called free vias because they can be added anywhere in the design.

  Free vias are more often called stitching vias because when added to a design, they may resemble a row or pattern of stitches. In addition, they are often used to "stitch together" copper to a plane. For example, you might add vias to a small copper area to give it more numerous connections to a ground plane, insuring a solid connection.

  When you add these vias, the Stitching property is enabled, which distinguishes them from routing vias.

  There are several methods you can use to place stitching vias in a layout design:

  - Add single vias repetitively. Vias appear on your cursor, ready to be placed. For instructions, see "Adding Stitching Vias".

  - Surround a net, pin-pair, or copper area with vias to act as a via shield (**Add Via Shield** command). For example, you might place a via shield around a pin-pair to create an RF trough. For instructions, see "Adding a Via Shield".

  - Stitch a copper or copper hatch shape with a pattern of vias (**Via Stitch** command). There are two modes for stitching the patterns:

    - You can fill the shape with vias. For instructions, see "Filling a Shape with a Pattern of Vias".

    - You can also add vias inside the perimeter of the shape. For instructions, see "Placing Vias Inside the Perimeter of a Shape".

Via Modes

Use the Vias Dialog Box to determine what kind of via to install when you change levels while routing. Use the V modeless command on page 83 to open the dialog box.

The table below lists the via routing modes. To set the via type quickly, use the VA, VP, or VT modeless commands on page 83.

Table 112. Via Routing Modes

Mode	Description
Automatic (VA)	SailWind Layout chooses from all vias, through or partial, that can handle the particular layer change. If SailWind Layout finds partial vias dedicated to the layer change, it chooses from them even if the via is not one that is specifically selected in the Vias dialog box. If SailWind Layout cannot 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, SailWind Layout installs the one with the smallest drill diameter or smallest pad size. Automatic allows only vias

Table 112. Via Routing Modes (continued)

Mode	Description
	that begin and end on the “Layer pair” setting in the Options dialog box > **Routing** category > General subcategory on page 1542. To use automatic via mode, the layer pair for routing and the layer pair for the partial via definition must 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 (VP)	The automatic via selection still occurs, but it is limited to the partial vias only.
Through (VT)	The automatic via selection still occurs, but it is limited to the through hole vias only.

In any case, 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 rule takes precedence. If On-line DRC is set to Prevent Errors, the layer will not change if the trace or via creates a clearance problem.

You can change an installed via to another type using the Via Properties Dialog Box. For more information, see the “Modifying Via Properties” topic. You can also change the via type during routing. For more information, see the “Changing the Via Type While Routing” topic.

Allow Vias by Net

Use the Routing Rules Dialog Box to control which vias are eligible for which nets. This is one of the main criteria for the automatic via selection.

Trace Width

You can pass a value for trace width with the netname from the schematic. You can also enter or edit the width as a line in the schematic ASCII file for nets. If widths are assigned to nets when the netlist is read in, the connections automatically assume those widths when routing. If no widths are associated, SailWind Layout provides a default width of 12 mils for all nets. You can change the default trace width at any time by changing the Recommended Trace Width. For more information, see the Clearance Rules Dialog Box.

The recommended width is in effect unless you override it locally. For example, a net may have a width of 12 assigned to it, but you need a certain pin pair within the net to have a width of 10. Use the Pin Pair Properties Dialog Box to assign a local trace width to the pin pair in the Trace Width box.

You can see a connection's assigned width when you select it. The width appears on the Status Dialog Box on page 1733prefixed with “W:” between the pin pair and the trace-to-trace clearance.

If you have the Advanced Rules license option, you can assign different widths within a net by making the width assignments on the net class, pin pair, or group of pin pairs level.

Length Minimization

When you start interactive routing, the flightline to the terminating pin may jump from the pin to which it is connected to another pin. This on-the-fly length minimization shows that the pin indicated by the flightline is part of the same net and is closer to the cursor than the original terminating pin.

You can disable length minimization by setting the topology type to “protected.” In this case, the flightline appears but maintains its original connection. You can set this property for Net, Class, or the Default level of the rules hierarchy. For more information, see the Routing Rules Dialog Box.

Routing Colors

Use the Display Colors Setup Dialog Box to set different colors for routed traces on different layers. You can also set different colors for copper and for pads on pins or vias per level. If “Active layer comes to front” is selected on the Global category > General subcategory of the Options dialog box on page 1531, when you start a routing command on that layer, all the traces drawn in the color for the current layer come forward and overlap traces on other layers.

To selectively hide or display traces or connections or to display nets with specific colors, use the View Nets Dialog Box. For more information, see “Assigning Colors to Nets”.

Display of Trace Width

To speed scrolling, panning, and screen regeneration, you can set a minimum displayed width for all lines. Any line widths under this width value appear as 1-pixel centerlines.

You can set the “Minimum display width” in the Global category> General subcategory of the Options dialog box on page 1531 or use the R modeless command on page 83. Set the value to your narrowest line width or less to see the design's true widths.

Route and Unroute Protection

You can set options that prevent the modification of fully routed connections or the routed portion of partial routes, and also unroutes at the pin pair or net level. Protection of routed traces passes bidirectionally to and from SailWind Router to protect critically placed routes during interactive routing, and batch routing.

The protection setting is maintained when a reuse is created or broken apart. For more information, see the Breaking a Reuse topic.

Tip
In the Routing category > General subcateory on page 1542 of the Options dialog box, select the Show Protection check box to display traces with an outline pattern opposite that of other traces. In other words, if normal traces are solid, protected traces are outlined. If in Outline View Mode, protected traces are solid.

Protecting Pin Pair Trace Segments

You can protect or prevent modification of fully routed connections or the routed portion of pin pairs.

Procedure

1. Select a trace segment or unroute.
2. Right-click and click the **Select Pin Pair** popup menu item.
3. Right-click and click the **Properties** popup menu item.
4. In the Pin Pair Properties Dialog Box, select the Protect Routes check box.
5. Click **OK**.

Protecting Entire Nets

You can protect or prevent the modification of fully routed connections or the routed portion of nets.

Procedure

1. Select a trace segment or unroute.
2. Right-click and click the **Select Net** popup menu item
3. Right-click and click the **Properties** popup menu item.

4. In the Net Properties dialog box on page 1487, select the Protect Routes check box.

5. Click **OK**.

Protecting Unroutes in a Pin Pair

You can protect or prevent the modification of fully unrouted connections and the unrouted portion of partial routes of pin pairs. Protecting unroutes between pin pairs prevents routing of the protected connections.

Restrictions and Limitations

You cannot protect the unrouted portion of a partial route unless you first protect the routed portion.

Procedure

1. Select an unroute.

2. Right-click and click the **Properties** popup menu item. The Pin Pair Properties Dialog Box opens.

3. To protect a fully unrouted pin pair, select the Protect Unroutes check box. If the pin pair contains partial routes, this option is unavailable. To protect the unrouted portion of a partially routed pin pair, select the Protect Routes check box and then you can select the Protect Unroutes check box.

4. Click **OK**.

Protecting Unroutes in a Net

You can protect or prevent the modification of fully unrouted connections and the unrouted portion of partial routes in nets. Protecting unroutes in a net prevents routing of the protected connections.

Restrictions and Limitations

You cannot protect the unrouted portion of a partial route on page 1846 unless you first protect the routed portion.

Procedure

1. Select an unroute.

2. Right-click and click the **Select Net** popup menu item.

3. Right-click and click the **Properties** popup menu item. The Net Properties dialog box on page 1487 opens.

4. To protect a fully unrouted net, select the Protect Unroutes check box. If the net contains partial routes, this option is unavailable. To protect the unrouted portion of a partially routed net, select the Protect Routes check box and then you can select the Protect Unroutes check box.

5. Click **OK**.

Selecting Routing Objects

Select one of several route objects before you begin routing. You can confirm what you have selected in the design.

Procedure

1. With nothing selected, right-click and either choose one of the selection presets or click the **Filter** popup menu item and then select specific routing objects for selection. For more information, see "The Selection Filter."
2. Click one of the following objects:
   - **Connections** — if there is a color assigned to Connections in the Display Colors dialog box, you can toggle the display of unrouted connections using the Z U modeless command on page 83. When you select a connection, right-click and click the **Select Net** popup menu item to select the entire net.
   - **Pins** — for rectangular and oval pins, or pads, click in the center of the object or use area select. The trace begins on the active layer.
   - **Traces** — a tack appears at the starting point of routing.
   - **Vias** — the trace begins on the active layer.
   - **Tacks** — if SailWind 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.

Results

To confirm basic information for a selected object, you can 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**.

Use Transparent Mode (T modeless command on page 83) 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.

Related Topics

Typing Modeless Commands

Routing Tools

SailWind Layout provides multiple routing tools that each have their own special uses.

Tip
As an alternative, you can use the more powerful interactive and auto-routing capabilities of SailWind Router to route your design. Set up a routing strategy and options; indicate whether to autoroute with SailWind Router in the background or foreground. For more information, see Autoroute using SailWind Router on page 776.

Table 113. Routing Tools

Topic	Description
Routing Manually	With the Add Route button on the Design Toolbar, click to indicate each corner in the trace. This type of routing works in any DRC mode.
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" On-line DRC mode.
Routing with the Single Layer Pin-to-Pin Autorouter	The Auto Route tools is a single layer, pin-to-pin autorouter that automatically adds traces to a selected connection or pin pair. Existing impeding traces are shoved when possible. Auto Routing is available only in "Prevent errors" On-line DRC mode.
Bus Router	The bus router is a dynamic routing tool that creates buses. Use the bus router to quickly route data lines, memory arrays, or similar connections where several routes need to flow together from one part to another or to a set of parts.
Routing Buses	The Bus Route tool is 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" On-line DRC mode.

Routing Manually

With the **Add Route** button on the Design Toolbar, click to indicate each corner in the trace. This type of routing works in any DRC mode.

You can add arcs, miters, and vias while a trace is in progress or after it is completed.

Prerequisites

- Set the routing direction on page 1441 for all layers.
- Set your grids on page 1537: design, via, and display.
- Set your design rules on page 412 and enable DRC on page 461.

Procedure

1. On the Design Toolbar, click the **Add Route** button.

 Tip
 You can set the Add Route command to start automatically when you double-click an unrouted path (connection). In the Options dialog box, Routing category > General subcategory on page 1542, in the Unrouted path double click area, select the Add Route check box.

2. Select an object on page 698 to route. 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" on page 1808. When you start routing from a trace, a tack on page 1864 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. The trace can move in one of the three Angle Modes: Orthogonal, Diagonal, or Any Angle. See the Line/Trace Angle setting in the "Options Dialog Box, Design Category."

 Tip
 As you route, the "guard band" on page 1832 appears whenever the head of the trace meets a clearance obstacle that it cannot shove. With design rules in prevent mode, you cannot complete the trace without changing the clearance rules or removing the obstacle.

4. Click to indicate a corner. To remove unwanted corners, press the Backspace key to remove the last corner and make the previous segment active.

5. Right-click and click the **Add via** popup menu item to change layers. For more information, see "Adding a Via While Routing."

 Tip
 Use Transparent Mode, T modeless command on page 83, to view obstacles that may lie under traces on the active layer.

6. Proceed to your destination until you see the target on page 1854 shape over a pad, trace, or via.

7. Click to complete the trace.

Results

If you disabled on-link design rule checking at any time when routing with the Add Route command, you should check the design for clearance violations. For instructions, see "Running a Design Check."

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" On-line DRC mode.

The Dynamic Route tool is a gridless, shape-based router. If the routing grid is set at less than clearance values, the Dynamic Route tool uses the clearance settings to decide the path.

Because corners install automatically, you quickly route a selected connection by pulling the head of the trace through obstacles with the cursor, guiding the path you want to use. Although you can, you do not have to manually install corners. Established traces move out of the way to make room for a path you are routing, providing there is room to move without creating clearance violations.

Prerequisites

- Set your design rules on page 412
- Set the routing direction on page 1441 for all layers. Each layer has a preferred Routing Direction. Route on a layer that is set in the direction you need. For more information see "Layers Setup Dialog Box."
- Set your grids on page 1537: design, via, and display.

Procedure

1. Type the drp modeless command on page 83 and press the Enter key to enable the online design rule checking Prevent mode and the **Dynamic Route** button. For more information, see "Turning on Design Rule Checking" on page 461 and "Design Rule Checking."
2. On the Design Toolbar, click the **Dynamic Route** button.

Tip
You can set Dynamic Route to start automatically when you double-click an unrouted path (connection). In the Routing Options > General subcategory on page 1542, in the "Unrouted path double click" area, select the Dynamic Route check box.

3. Select an object on page 698 to route. 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" on page 1808. When you start routing from a trace, a tack on page 1864 appears at the selection point.
4. Guide the pointer through the items you want to bypass without creating corners. To remove unwanted corners, slowly move the pointer back over the unwanted traces. The trace can move in one of the three Angle Modes: Orthogonal, Diagonal, or Any Angle. See the Line/Trace Angle setting in the Options dialog box Design category on page 1503.

Tip
As you route, the guard band on page 1832 appears whenever the head of the trace meets a clearance obstacle that it cannot shove. With design rules in prevent mode, you cannot complete the trace without changing the clearance rules or removing the obstacle.

5. Click to indicate a corner; these act as tacked corners to prevent preceding segments from moving. To remove unwanted corners that were placed by clicking, press the Backspace key to remove the last corner and make the previous segment active.
6. Right-click and click the **Add via** popup menu item to change layers. For more information, see "Adding a Via While Routing."

Tip
Use Transparent Mode, T modeless command on page 83, to view obstacles that may lie under traces on the active layer.

7. Proceed to your destination until you see the target on page 1854 shape over a pad, trace, or via.

8. Click to complete the trace.

Related Topics

Rerouting with Sketch Route

Routing with the Single Layer Pin-to-Pin Autorouter

The Auto Route tools is a single layer, pin-to-pin autorouter that automatically adds traces to a selected connection or pin pair. Existing impeding traces are shoved when possible. Auto Routing is available only in “Prevent errors” On-line DRC mode.

Restrictions and Limitations

- The auto router operates on one layer only; it will not add vias.
- The auto router follows only the current design grid.
- Each layer has a preferred routing direction, the auto router works only in the preferred routing direction. For more information, see “Layers Setup Dialog Box”.

Prerequisites

Set your grids on page 1537: design and display.

Procedure

1. Type the drp modeless command on page 83 and press the Enter key to enable the online design rule checking Prevent mode and the **Auto Route** button. For more information, see “Turning on Design Rule Checking” and “Design Rule Checking”.

2. To improve route completion results, click the **Tools > Length Minimization** menu item. The Auto Route tool follows the unrouted connections to add traces. If you run Length Minimization, you can provide shorter and more direct paths for the Auto Route tool to follow.

3. On the Design Toolbar, click **Auto Route**.

4. Select an object on page 698 to route.

Results

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 and it leaves the unrouted connection.

Bus Router

The bus router is a dynamic routing tool that creates buses. Use the bus router to quickly route data lines, memory arrays, or similar connections where several routes need to flow together from one part to another or to a set of parts.

Tip
You need the Dynamic Route Editing (DRE) license option to use the bus router.

To use the bus router, identify multiple pin pairs to route and then place one route. This route acts as a guide route. The other routes, called follow routes, are based on the established path of the guide route. The bus router creates the guide route using the same automatic corner creation and trace shoving as the Dynamic Route tool.

To take full advantage of the bus router's automatic routing features, make sure selected pin pairs are adjacent to each other at both ends of the connection and that connections do not cross. Selecting pins with connections that cross each other invokes a manual bus route mode where you dynamically route each connection in sequence. For more information, see the Routing Buses topic.

Selection Criteria for Bus Routing

The bus router creates routes from selected pins, vias, and tacks. You can set the Selection Filter to select only these objects. You can also right-click and click the Select Pins/Vias/Tacks popup menu item when nothing in the work area is selected. The bus router does not work if you select connections or traces.

While selected pins are the most commonly used objects for bus routing, adding selected vias and tacks lets you route a portion of a bus, end it, and then begin routing again.

Selection Rules

The following rules govern selection for bus routing:

- Select at least two objects. This can be a combination of pins, vias, and tacks.
- Selected pins can have through-hole or surface mount pads. Selected surface mount pads must exist on the same layer.
- Selected pins must all belong to the same part.
- Selected objects must have assigned nets.
- Nets to which the selected objects are assigned cannot be assigned to a plane layer.
- Selected vias or tacks must be attached to a dangling route.

Object Filtering

When you start the bus router, the following are automatically cleared:

- Pin selections on multiple parts. The part with the most selected pins remains selected and pins belonging to other parts are deselected.
- If two or more parts have the same number of selected pins, the part added to the design is chosen.
- Pins assigned to a plane net.
- One of two objects if they belong to the same net.

Bus Route and the Active Layer Setting

When you select surface mount pads or tacks at the ends of a trace, the layer on which these objects exist is detected and the layer is automatically set as the active routing layer.

Tip
If a selected pin has a layer restriction on the active layer, it is deselected.

Bus Route Tips for Adding Corners

While bus routing, you can cycle through all of the connections in the bus to use a different connection for the guide route.

- Keep guide routes as short as possible — Long guide routes can be difficult for the bus router to resolve, so indicate corners for bus turns as often as possible. If a Bus Route failure message appears on the status bar, press the Backspace key to undo the bus to the last indicated position. Then insert the corner at a different location.
- Switch the guide properly — Proper control of the guide route is necessary for successful bus routing. The guide route should be internal when you indicate a corner. If you use an external guide route to create a corner, the internal follow routes shrink to make a turn, as long as there is sufficient space.

Via Pattern Mode

You can choose from five via patterns when you add vias to a bus. The chosen via pattern is used until you choose a different pattern or exit the program. The available via patterns are:

Parallel

Vias are added in a column, parallel to the direction in which the bus is traveling. Parallel is the default via pattern (Figure 91). If the guide route is one of the middle routes, the remaining routes adjust on either side of the pattern (Figure 92).

Figure 91. Routing when Via Pattern is Parallel to Bus Direction

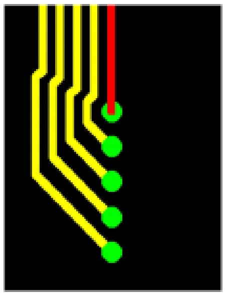
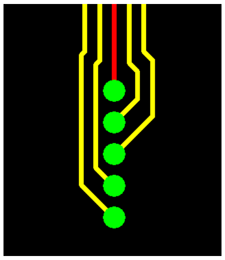


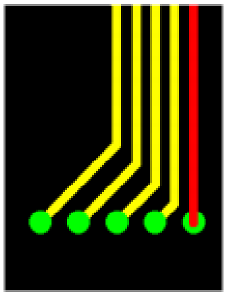
Figure 92. Parallel Vias from Middle Guide Route



Perpendicular

Vias are added in a row, perpendicular to the direction in which the bus is traveling. When the guide route is one of the middle routes, the remaining routes adjust on either side of the pattern (Figure 93).

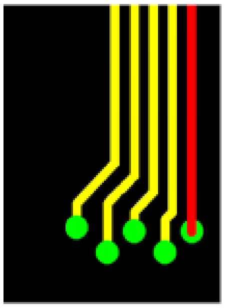
Figure 93. Routing when Via Pattern is Perpendicular to Bus Direction



Staggered

Vias are added in staggered rows to minimize space requirements. If the guide route is one of the middle routes, the remaining routes adjust on either side of the pattern (Figure 94).

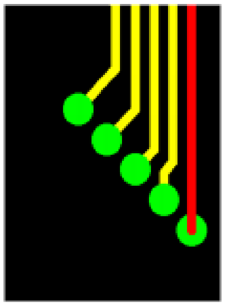
Figure 94. Routing when Via Pattern is Staggered



Minus 45 Degrees

Vias are added in a diagonal, at a negative 45-degree angle to the direction in which the bus is traveling. If the guide route is one of the middle routes, the remaining routes adjust on either side of the pattern (Figure 95).

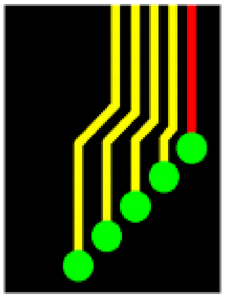
Figure 95. Routing when Via Pattern is Minus 45 Degrees



Plus 45 Degrees

Vias are added in a diagonal, at a positive 45-degree angle to the direction in which the bus is traveling (Figure 96).

Figure 96. Routing when Via Pattern is Plus 45 Degrees



Related Topics

Routing Buses

Routing Buses

The Bus Route tool is 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” On-line DRC mode.

Prerequisites

- Set your grids on page 1537: design, via, and display.
- Set the routing direction on page 1441 for all layers. Each layer has a preferred routing direction, be sure to route on a layer that is set in the direction you need. For more information, see “Layers Setup Dialog Box”.
- Set your design rules on page 412.

Procedure

1. Type the drp modeless command on page 83 and press the Enter key to enable the online design rule checking Prevent mode and the **Bus Route** button. For more information, see “Turning on Design Rule Checking” and “Design Rule Checking” on page 410.

   CAUTION:
   If the Smooth bus route traces option on page 1542 is enabled, the Smooth on page 1860 command is applied to all Bus Route traces once they are complete.

2. **Design Toolbar** button > **Bus Route** button.

   You can also activate the bus route mode from the shortcut menu or the toolbar after you have selected the objects for routing in the design.

3. Select multiple objects to route by holding the mouse button down and dragging a selection rectangle around them.

   For example, draw a selection rectangle around a group of pins whose connections terminate on the same component. A route segment for the first connection, or guide route on page 1832, attaches to the pointer (and follows pointer movement).

4. To use a different connection for the guide route, right-click and click the **Next Guide** (or **Previous Guide**) popup menu item to cycle through all of the connections.

   Tip
   For a list of all Bus Routing keyboard shortcuts see the “Keyboard Shortcuts for Routing Operations on page 100” table.

5. As you move the pointer, you route in a dynamic route mode where corners are added automatically. The trace can move in one of the three Angle Modes: Orthogonal, Diagonal, or Any Angle. For more information, see the Line/Trace Angle setting in the Options dialog box > Design subcategory on page 1503.

Tip
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.

6. 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 other bus connections, or follow routes on page 1829, 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 on page 700. When you complete a connection in manual mode, the next connection becomes the new guide route.

   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.

Tip
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.

7. To change layers for all connections at once:

   a. Right-click, point to the **Via Pattern Mode** popup menu item, then click the pattern you want.

   b. Move the pointer to where you want to add the via pattern.

   c. Right-click and click the **Add Vias** popup menu item.

      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.

      The spacing between vias is based on the via size, the via grid, and the widths and clearances of the traces.

   d. If the via pattern you just added is not 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. The Cycle Via Pattern menu item is only available if Add Vias or Add Via to Guide was the last command used. For more information, see “Via Pattern Mode on page 703.”

8. 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 the **Add Via to Guide** popup menu item. A via is added to the current guide route and the next connection becomes the new guide route.

The spacing between vias is based on the via size, the via grid, and the widths and clearances of the traces. For more information, see "Adding a Via While Routing."

9. To end the trace short of the target (leaving a dangling connection, or ending at a via or testpoint):

a. Right-click, point to the **End Via Mode** popup menu item, and then click an End mode. For more information, see "End Via Mode."

b. Move the pointer to where you want to add the tack or via.

c. Right-click and click the **End** popup menu item.

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 right-click and click the **End Guide** popup menu item to end the routes individually.

The spacing between vias is based on the via size, the via grid, and the widths and clearances of the traces.

10. To complete the bus route, proceed to the final destination and do one of the following:

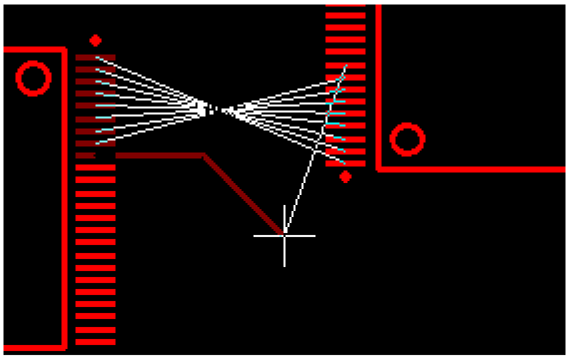
- Right-click and click the **Complete** popup menu item (or double-click).

- 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 on page 1854 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.

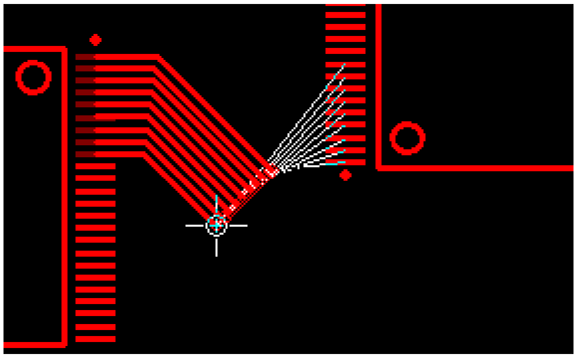
Examples

Listed below are examples of basic bus router operations:

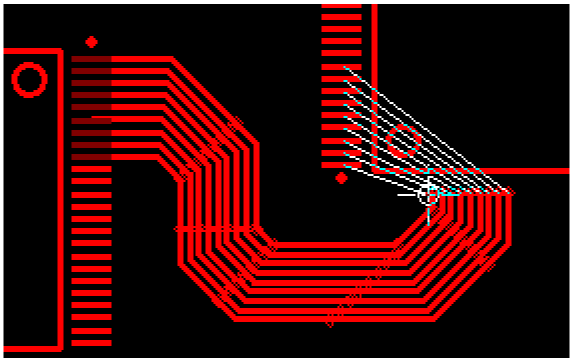
- Moving the pointer before indicating a corner: only the guide route appears.



- 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. For best results when adding corners, use the inside connection as the guide.



Related Topics

Bus Router

Via Types

Design Routing Visibility Options

SailWind Layout provides several visibility options to make routing your connections easier. By assigning specific colors to nets or unrouted connections, for example, you can quickly distinguish the net in the design. Similarly, by hiding unrouted connections that you do not need to see, you can make the design less cluttered.

Assigning Colors to Nets

You can assign colors to pads, vias, and unrouted connections of a specific net. By default, pads, vias and unrouted connections assume the color given to them in the Display Colors dialog box. For example, assign a color to a plane net after placement. With all plane net pads appearing in a specific color, you can locate them quicker for fanning out to the plane layer.

There are many options available for net color assignment. See the table below for instructions.

Table 114. Available options for net color assignments

#	Option	Operation
1	Use the View Nets Dialog Box	1. Click the **View > Nets** menu item. 2. In the View Nets Dialog Box, scroll through the Net List on the left and add one or more nets to which you want to assign color to the View List. 3. In the View List, select one or more nets. You cannot assign a color to Default. 4. Select the **Enable Color by Net** check box. 5. In the Color by Net area, select a color for the net. This color palette comes from the Display Colors Setup Dialog Box. If you select the “None” option, however, the net does not have a color assignment and it appears as it normally does. All nets are assigned None by default. 6. Click **OK**. The color change takes place.
2	Use the Assign Color to Net Dialog Box	It works in both Verb Mode and Object Mode. Here takes operations in Verb Mode as example. 1. With nothing selected, right click, and click the **Assign Color to Net** popup menu item. 2. In the View Nets Dialog Box, select a color you want from the color palette.

Table 114. Available options for net color assignments (continued)

#	Option	Operation
		If you select the "None" option, however, the net does not have a color assignment and it appears as it normally does. 3. Select a net you want to color in the workspace. To check if the color change takes place, click at the blank space. **Tip** Colors of those nets locked in the View Nets Dialog Box remain unchanged. 4. Optionally: • Repeat Steps 2-3 for other nets that require individual colors. • Click **Default All** to return net colors to their default assignments (as defined in the Display Colors Setup Dialog Box), except those nets locked in the View Nets Dialog Box. • Click **More** to open the Assign Color to Netlist Dialog Box, where you can assign color for nets in a batch.
3	Use the Assign Color to Netlist Dialog Box	1. In the Assign Color to Netlist Dialog Box, add net(s) for which you want to assign color from the Net List to the Selected list. You can use wildcards or expressions for filter, and Ctrl/Shit for multiple selections. 2. In the color palette, select a color for all nets in the Selected list. 3. Click **Apply** to make the color assignment effective. **Tip** Colors of those nets locked in the View Nets Dialog Box remain unchanged. 4. (Optional) To assign individual colors for other nets, you can: • Click **Remove all** to clear the Selected list, and then repeat Steps 1-3. • Click **Return** to use the Assign Color to Net Dialog Box for color assignment.

Highlighting a Net

Highlight a net to quickly locate it on a dense board.

Tip
To search for and select several nets, use the Find dialog box on page 1378. Finding by nets finds and either highlights or selects nets.

Procedure

Type the N modeless command on page 83, type a space after the N modeless command, type the netname and press the Enter key. As an alternative, you can select a net in the design by clicking on an unrouted connection, or pad and then click the **Edit > Highlight** menu item.

Results

The net appears in the Highlight color that is specified in the Display Colors on page 1318. The net is not selected: you have to select a segment if you intend to route it. The highlighting turns off when you select the net or a pin pair and click the **Edit > Unhighlight** menu item.

Changing the Visibility of All Unrouted Connections

You can turn the display of all the unrouted connections (also called the “ratsnest”) on and off by display color or commands.

By color:

1. Click the **Setup > Display Colors** menu item.
2. In the Display Colors Setup Dialog Box, set the **Connection** color tile:
    - To the background color to make the connections invisible.
    - To a specific color from the color palette to distinguish it in the design.

By commands:

Available options are shown as below, which take no effect on nets that are locked in the View Nets Dialog Box.

Table 115. Available options for ratsnest visibility control

Option	Operation
Function buttons	Click the following buttons on the Standard Toolbar: • : Show All Connection • : Hide All Connection
Modeless commands	Use modeless commands: **VUA** for Show All Connection; **HUA** for Hide All Connection.
Shortcuts	Use the shortcuts for **Show All Connection** and **Hide All Connection** as defined in the Customize Dialog Box, Keyboard and Mouse Tab, where these two commands are under the list item "Standard Toolbar".

Changing the Visibility of Selective Unrouted Connections

You can also turn the display of selective unrouted connections on and off by display color or commands.

Table 116. Available options

Option	Operation
By display color	1. Click the **View > Nets** menu item. 2. In the View Nets dialog box, select a net and move it to the View List to change its visibility. Use this dialog box to hide or show ratsnest connections by netname. You can also select nets by net class, nets with rules, and plane net status. For example, it is often convenient to hide the unrouted connections of the ground net or other plane nets. 3. With the net or nets selected in the View List: • To hide the net's unrouted connections, in the View Unroutes Details area, click None. **CAUTION:** When View Nets hides unrouted connections, neither Verify Design nor Find can see them. Be sure View Nets does not disable nets or traces you want to search for or select. • To give the net's unrouted connections a custom color, select a color in the palette. Pads, vias and unrouted connections take on the custom color.
By commands	1. Select one or more design items with net property from the workspace or the Project Explorer. 2. Apply commands to the selected objects by one of the following means: **Note:** These comamnds are only available for design items with net property. • Right-click menu: Right click, and click the **Show Connection** or **Hide Connection** popup menu item. **Note:** To apply commands to multiple objects selected in the Project Explorer, you must right click at the blank space. Otherwise, it only works on the object at the cursor position on click. • Use modeless commands: **VUS** for Show Connection; **HUS** for Hide Connection. • Use the shortcuts for commands **Show Connection** and **Hide Connection** as defined in the Customize Dialog Box, Keyboard and Mouse Tab, where these two commands are under the list item "View".

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 appear solid, protected routes have outlines, and so forth.

For more information, see “Route and Unroute Protection” on page 696.

Procedure

1. Click the **Tools > Options** menu item.
2. In the Options dialog box, click the Routing category > General subcategory on page 1542.
3. Select the Show Protection check box.
4. Click **OK**.

Routing To a Copper Shape

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.

Restrictions and Limitations

While in ECO - Add Route mode, you cannot terminate a newly created connection on copper. The connection must go between two component pins.

Prerequisites

- You must assign a net to the copper shape before you can route to it. For more information, see “Assigning a Net to a Copper Shape or Copper Plane” on page 665.
- 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. For more information, see “Associating Copper with Terminals”.

Procedure

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 on page 1854-like shape indicates that you have connected with the copper.

   If you are routing on another layer, you can end the trace with a via into the copper. For more information, see “End Via Mode”.
3. Click to finish.

Related Topics

Routing From a Copper Shape

Routing From a Copper Shape

There are limitations when 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 limitation, the recommended practice is to route to the copper on page 715 whenever possible for more flexible connection points.

Prerequisites

- You must assign a net to the copper shape before you can route to it. For more information, see “Assigning a Net to a Copper Shape or Copper Plane” on page 665.
- 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. For more information, see “Associating Copper with Terminals”.

Procedure

1. While in a routing mode, locate the point where a connection (unroute) is exiting the copper.
2. Click to begin the trace.

> **Tip**
> You can route and add a via within the copper if you need to immediately change layers.

3. Route your trace.

Results

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.

Related Topics

Routing To a Copper Shape

End a Trace on a Different Net

You can end a trace on a segment with a different netname, but you must use the Add Route button on the ECO Toolbar.

For more information, see the “Adding a Route in ECO Mode” topic.

Vias

Read the topics that follow to learn more about using vias when routing.

For topics covering the design of vias, see "Via Setup".

You can find more information about using vias to change layers in "Layer Changes While Routing".

Via Types

If you have different types of vias available to a net, you can switch between via modes to help select which via to use when routing.

You can use the V, VA, VP, VT modeless commands on page 83 as shortcuts to specify the Via Mode in the Vias dialog box on page 1782when you add a via or a virtual pin. When you choose a via type, it is used with the Add Via, Add Via to Guide, and Add Virtual Pin commands. For procedures, see "Changing the Via Type While Routing".

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 will not complete if the trace, via or virtual pin creates a clearance problem.

After you install a via or virtual pin, you can change it to another via type using the Via Properties Dialog Box or Virtual Pin Properties Dialog Box.

Adding Vias in Pads

You can add vias to SMD pads to avoid fanouts when there is no room for a regular fanout in the design.

Procedure

1. In the Clearance Rules Dialog Box, in the Same Net area, the SMD to Via clearance must be 0 (zero) at the required level of the rules hierarchy to enable the via to be placed in the pad.

   Tip
   Also consider other via rules, such that the via cannot create a clearance violation. It must be allowed for the net and in drill pairs.

2. Select one or more SMD pads, right-click and click the **Add Via at SMD** popup menu item.

   The via is added to the center of the SMD. Component pads always overlay vias and it may be difficult to determine if the via is actually inside the pad. Enable "Transparent mode" with the T modeless command on page 83 to see vias in pads.

Related Topics

Connecting SMD Pads to Planes

End Via Mode

To end a trace with vias, tacks, or test points, use the commands on the **End Via Mode** submenu in the shortcut menu.

The available modes are listed below. The selected mode is on until you select a different one or exit SailWind Layout.

Table 117. 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. For related information, see "Placing Test Points", and "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

Routing Buses

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 the **Layer Toggle** popup menu item, or shift-click to install a via and change to the other layer.

Set the layer pair in the Routing category > General subcategory on page 1542 of the Options dialog box or as a shortcut, use the modeless command on page 83 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 the **Layer** popup menu item to specify the layer number.

Related Topics

Layer Changes While Routing

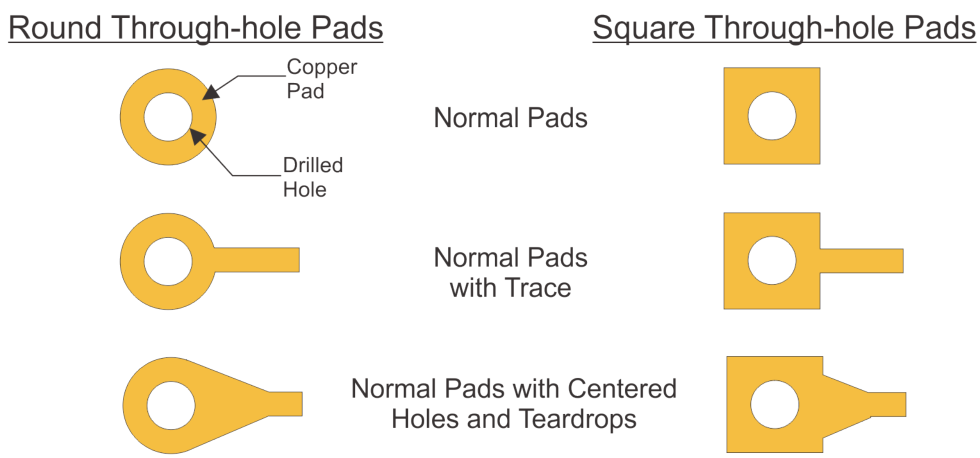
Teardrops

A *teardrop* (also referred to as a *pad fillet*) is typically a v-shaped copper reinforcement that you add to the junction where a trace enters a pad. Most PCB designs use teardrops primarily on pads that have a drilled hole, though there are circumstances where they may be beneficial on surface mount technology (SMT) pads.

When you choose to generate teardrops, they are created automatically on all traces leading into pads and vias. If pads and vias are not yet routed, the teardrop is added once the trace into the pad or via is completed. You do not add teardrops selectively, but you can remove teardrops selectively.

In the Options dialog box, use the **Routing** category > **General** subcategory to generate teardrops. Use the **Teardrops** category to set options only for teardrops on new traces that you route. These settings are saved with the design.

Figure 97. Pads with Teardrops



Benefits of Using Teardrops

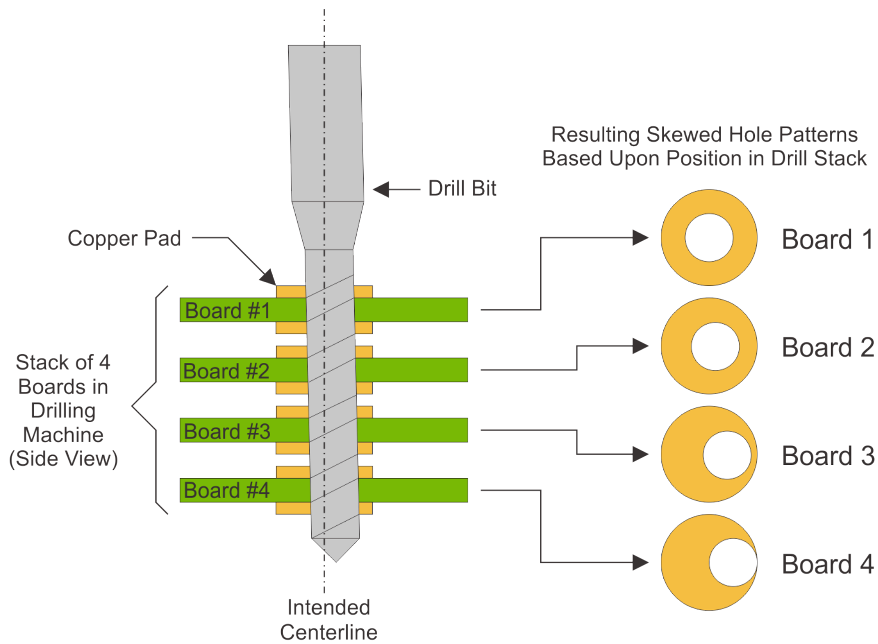
Teardrops provide solutions for a number of design issues, such as improving yield, preventing acid traps, and reducing flex circuit stress.

Yield Improvement

Teardrops can improve yield during the board manufacturing process when stacks of boards are drilled simultaneously.

A typical printed circuit board design can contain hundreds (and in some cases thousands) of drilled holes. If a manufacturer can drill a stack of 3 or 4 (or more) boards at the same time, they can improve throughput, and therefore reduce the overall cost of the board run. Unfortunately, as the height of the board stack increases, the potential for the drill to "wander" off of its centerline also increases. The farther the drill has to travel, the more potential there is for it to veer off its intended path, and produce a hole that is off-center on the pads of the lower board(s) in the stack. Though the variance is typically only a few thousandths of an inch, this small movement can cause the drill hole to become tangent to the edge of the pad, and if it occurs at the location where a trace connects to the pad, it can result in breakout or an open circuit.

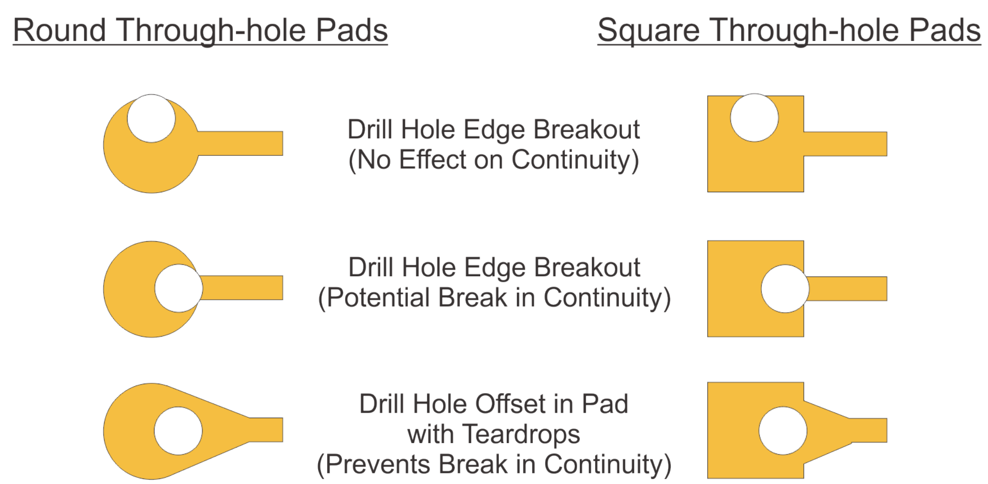
Figure 98. Drilling a Stack of Boards



Adding a teardrop to the pad increases the target area around the trace-to-pad connection and can compensate for some amount of the drill drift. This allows the connection to be made even if the drill is tangent to the pad edge and encroaches on the connection area.

This technique is effective on both round and square through-hole pads.

Figure 99. Pads with Breakouts



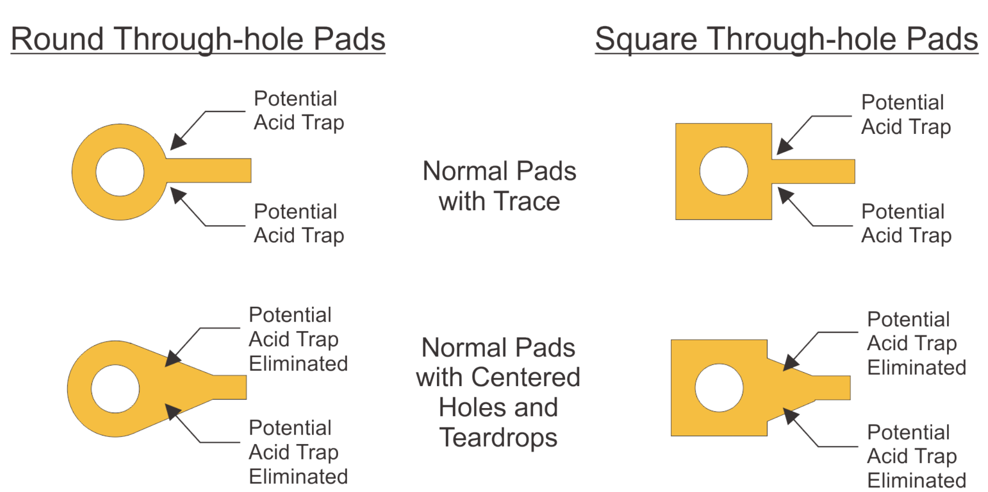
Acid Trap Prevention

Teardrops can reduce the potential of acid traps at the trace-to-pad junction.

The location where the trace enters the pad poses a potential for the etching solution (acid) to build up in the sharp corners and over-etch the junction. This can result in a slight narrowing of the trace as it enters the pad and potentially creating a connection that is more prone to fracturing. This over-etching can also slightly reduce the current handling capacity of the trace at the junction creating a potential failure point.

Adding a teardrop to the connection point eliminates the potential problem. It does not matter if the drill is centered or offset in the pad.

Figure 100. Acid Trap Reduction



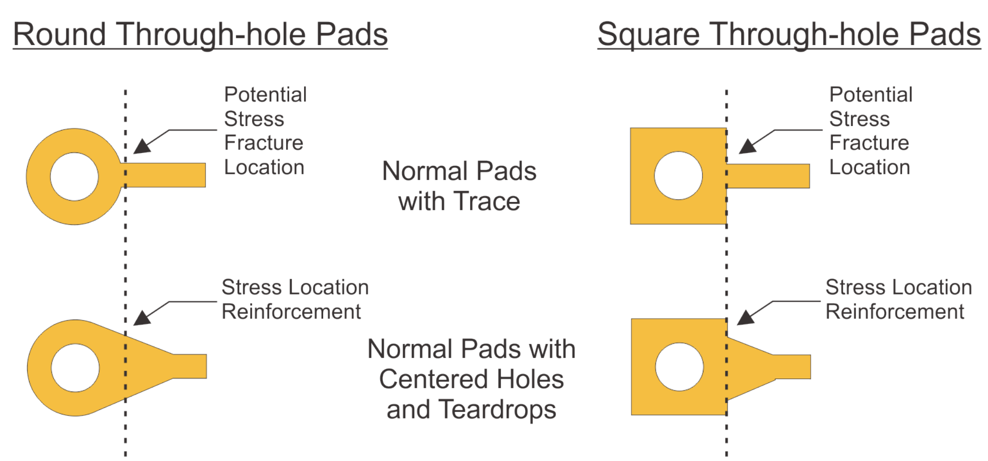
Stress Failure Reduction

Teardrops can reduce the potential for stress failures on flex circuits.

The location where the trace enters the pad can be subject to constant stress on a flex circuit. As the flex circuit is bent, it stresses this connection, and if the movement is repeated enough times, it could eventually cause the junction to fracture and fail.

Adding a teardrop to the connection point adds additional surface area to this potential stress point and greatly improves the ability of the board to withstand the effects of the repeated stress. It does not matter if the drill is centered or offset in the pad.

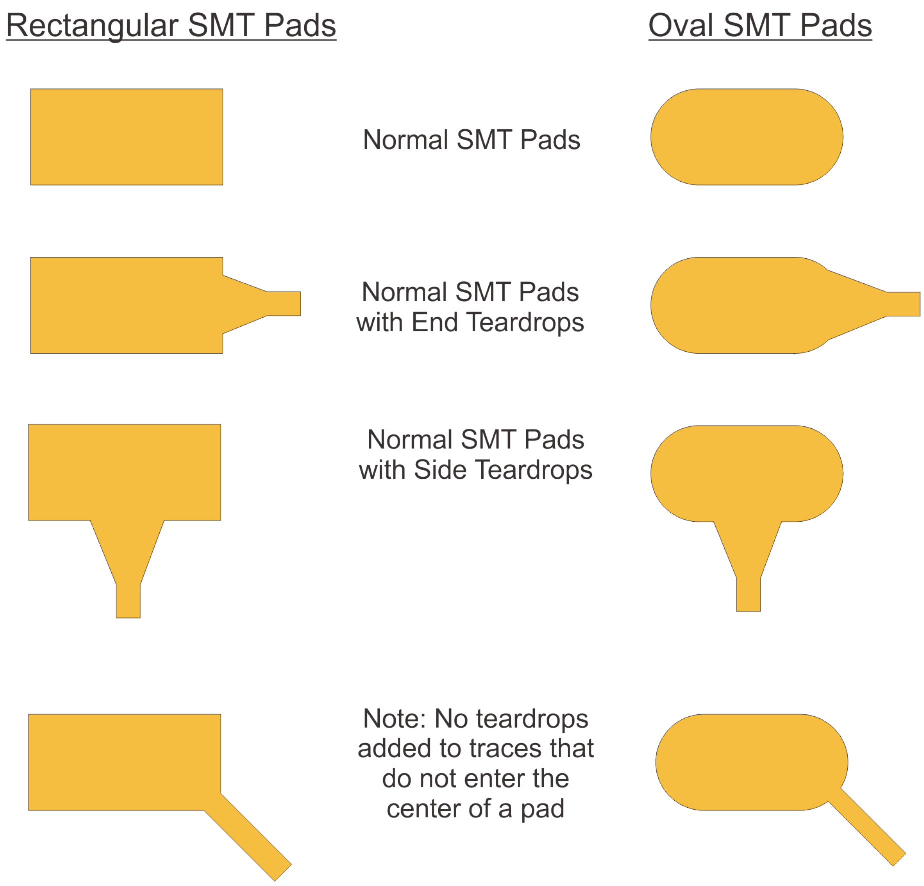
Figure 101. Stress Failure Reduction



Teardrops on SMT Pads

The same design benefits of acid trap reduction and stress fracture reduction obtained from using teardrops on through-hole pads can also be applied to surface mount design objects.

Figure 102. Teardrops on SMT Pads



Related Topics

Teardrops

Generating Teardrops

You can create teardrops on trace connections to pads and vias. If desired, you can also enable connections not yet routed to automatically create teardrops after they are routed.

Restrictions and Limitations

- 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 **Routing** category > **Teardrops** subcategory of the Options dialog box. Use Auto Adjust to set

a custom length and width ratio. With Auto Adjust selected, SailWind 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.

Procedure

1. Click the **Tools > Options** menu item.
2. In the Options dialog box, on the Routing category > General subcategory on page 1542, select the Generate Teardrops check box.
3. Click **OK**.

   Teardrops are automatically added to existing trace segments that enter or leave a pad or via or whenever you route that connection. 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.

Tip
If you do not want a teardrop on a specific trace you are routing, after you begin routing the trace, right-click and click the **Ignore Teardrop** popup menu item to turn on the Ignore Teardrop mode. A teardrop will not appear on traces that you route until you right-click and click the **Ignore Teardrop** popup menu item again, to turn off Ignore Teardrop mode. A check appears beside the command on the menu to indicate that the mode is turned on.

Results

If you have not changed the teardrop options in a previous SailWind Layout session, the Default teardrop shape is used for all newly routed trace connections to pads and vias. You can set more specific options for teardrops if you have the additional teardrop functionality using the **Routing** category > Teardrops on page 1546 subcategory in the Options dialog box.

Removing All Teardrops

You can delete or remove teardrops from a design by reversing the process for creating them.

Procedure

1. Click the **Tools > Options** menu item.
2. In the Options dialog box, on the **Routing** category > General subcategory on page 1542, 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.

Procedure

1. Click the **Tools > Options** menu item.
2. In the Options dialog box, on the **Routing** category > General subcategory on page 1542, in the Parameters area, clear the Display Teardrops check box.

Selectively Disabling Teardrops

You can turn teardrops off selectively at the pin level.

Procedure

1. Select one or more pins. To clear multiple pins, set the Selection Filter on page 1710 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.
2. Right-click and click the **Properties** popup menu item.
3. In the Pin Properties Dialog Box, clear the Teardrops check box.

Teardrops in CAM

When you plot teardrops, the line width you use is the same as the track width. If the teardrop is on an annular-shaped pad, SailWind 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. You can modify teardrops individually, teardrops on all layers, or all teardrops at one time. You can also remove an individual teardrop from a design.

Procedure

1. Select one or more traces to which teardrops are 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 the **Teardrop Properties** popup menu item.

   The Teardrop Properties on Traces Dialog Box appears.
3. Make changes and then click **OK**.

Results

- If you selected multiple traces and modify teardrops on those traces or apply teardrop property changes to multiple teardrops, only existing teardrops are modified. Newly created teardrops always use the settings in the **Routing** category > Teardrops subcategory on page 1546 of the Options dialog box.

- If you selected 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.

Checking Teardrops for Errors

You can run a check on teardrops—using a similar method to the Verify Design dialog box—to check for and view teardrop errors. SailWind Layout gives the error location, layer, and a short explanation of the error. Start the teardrop checking process through the **Routing** category > **Teardrops** subcategory of the Options dialog box instead of the Verify Design dialog box.

Procedure

1. Click the **Tools > Options** menu item. The Options dialog box opens. You must enable teardrops before you can check them. On the **Routing** category > General subcategory on page 1542, select the Generate Teardrops check box.
2. In the Options dialog box, click the **Routing** category > Teardrops subcategory on page 1546.
3. Click the **Check** button.
4. In the Check Teardrop Dialog Box, click **Start** to run the check. Error results are listed in the Location box. Listed errors contain their (X,Y) location, layer, and error type.
5. Select an error from the list.

    - In the Explanation box, view the detailed information about the error selected in the Location list. The explanation includes information about any conflicting object. The current error list remains whether the dialog box is open or closed.

    - The design window pans to the area of the error (unless the Disable Panning check box is selected). This is more obvious if you are zoomed into your design and the marker is set to show up in the design. The error markers remain until you click **Clear Errors**, which clears the error markers, but does not fix the errors.

Tip
Colors must be assigned properly in the Display Colors Setup 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.

Results

You can 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.

To change the default teardrop report file name, click **Report File**. This opens the Save As dialog box where you can change the name of the report. The changed name remains the default until you change it again.

Tacks

SailWind Layout generates tacks, which most commonly occur at corners where traces are routed against the layer direction.

You can also add Tacks on page 1864 manually. They can even be deleted in some cases, but it is not recommended since it will typically remove the corner at the same time and change the path of the routed trace.

You can add tacks manually to lock a corner or prevent rerouting when using the dynamic route or bus route tool. While routing click to lock down a corner with a tack. When there are no tacks, traces can be shoved to allow room for another trace to be routed.

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. The Selection Filter on page 1710 contains an option for tacks so you can set the filter to select only tacks. To delete a manually added tack select it and press the Delete key.

Jumpers

Read the topics that follow to learn more about creating, adding, and modifying jumpers.

Setting Up Jumpers

When you add a jumper to the design, SailWind Layout uses the Default jumper. You can change the Default jumper settings or change the settings for a specific jumper in the design. The attributes of the default jumper save to the *SailWindpcb.ini* file. Open the pin properties to change the setting of a specific pin in the design.

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.

Procedure

1. Click the **Setup > Jumpers** menu item.
2. In the Jumpers Dialog Box, in the Apply to area click:
   - **Default** — to change the settings of the Default jumper - any new jumper added to the design. The settings are saved in the *SailWindpcb.ini* file.
   - **Design** — and then select a jumper name in the Reference Name list—to change the settings of an existing jumper in the design.
3. Modify the jumper size settings. To modify the jumper size settings of Design Jumpers, use the Jumper Properties on page 1425. In the Jumper Sizes area:
   a. In the Min. Length and Max. Length boxes, type the minimum and maximum length of the jumper.
   b. 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.
4. Set up the pad stacks on page 729.
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.
6. Select the Display Silk check box to display the silkscreen outline for the jumper. The outline for jumpers is set at 10 mils; you cannot modify this setting. To modify the Display Silk settings of Design Jumpers, use the Jumper Properties on page 1425. For CAM output, you must enable Component Outlines for the layer on which the jumper resides before jumper outlines will plot.

Setting Up Jumper Pad Stacks

You can set up pad stack information for jumpers or individual jumper pins.

Procedure

1. Click the **Setup > Jumpers** menu item.
2. In the Jumpers Dialog Box, in the Apply to area click:

   - **Default** — to change the settings of the Default jumper - any new jumper added to the design. The settings are saved in the *SailWindpcb.ini* file.
   - **Design** — and then select a jumper name in the Reference Name list—to change the settings of an existing jumper in the design.
3. In the Pin list, select both jumper pin (All) or a pin to change. You can add individual pins of the jumper to the list for customizing. Use the **Add** or **Delete** button to maintain the Pin list.
4. In the Shape/Size/Layer list select a layer on which to make jumper pad stack changes. 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.
5. In the Parameters area, select the type of pad from the Pad Style list.

   Choose from (normal) pad, thermal (pad), or antipad. 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. 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 > Copper Planes category > Thermals subcategory on page 1514. The **Use Global Defaults** 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).
6. Select a Pad Shape.

   Choose from: Round, Square, Annular, or Odd. Annular and Odd shapes are only available for the normal pad style. You can select the Pad Size Relative to Drill Size check box to display inner and outer pad values relative to the drill size.
7. In the Diameter box, type the diameter for the pad style.

   Thermals require additional settings. Read “Design Rule Versus Pad Stack - Thermals and Antipads” on page 797for details.
8. Square pads only: Select the Corner type and enter the Radius.

Results

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

Adding Jumpers

You can place jumpers in a trace either while routing or after you finish routing. SailWind 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
SailWind Router can load and autoroute jumpers on page 1835.

Restrictions and 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.

Procedure

1. Unless you want to use the default jumper, set up a jumper:
   a. Click the **Setup > Jumpers** menu item.
   b. In the Jumpers Dialog Box, change the stackup of the jumper pins and the jumper size settings. For more information, see "Setting Up Jumpers".
2. Start the jumper in one of the following ways:
   - While routing — locate the pointer at the point where you want the first pin of the jumper to appear.
   - On an existing trace — select the trace at the point where you want the first pin of the jumper to appear. As an alternative, on the Design Toolbar click **Add Jumper** and then click a point on a trace where you want the first pin of the jumper to appear.
3. Right-click and click the **Add Jumper** popup menu item. The first pin of the jumper appears.

4. 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.

5. Click to complete the jumper.

   - If you are adding a jumper to an existing trace, you must enter the second pin on a trace in the same net.

   - If you added a trace while routing, regular routing continues.

Creating Jumper List Reports

You can use the Reports dialog box to create a report containing information about all jumpers in the design.

Procedure

1. Click the **File > Reports** menu item.

2. In the Reports Dialog Box, in the Select Report Files for Output list, select “Jumper List”.

3. Click **OK**. The report is written to *C:\SailWind Projects\report.rep* and displayed in the default text editor.

Related Topics

Reports

Modifying Jumper Properties

Open the Jumper Properties to view information, change coordinates, change jumper size settings, control the display of the jumper outline, glue the jumper in place, or modify the pad stack and reference designator of the jumper.

Procedure

1. Use one of the following methods to access the Jumper Properties:

   - Select a jumper pin, right-click and click the **Properties** popup menu item. In the Jumper Pin Properties Dialog Box, click the **Jumpers** button. The Jumper Properties Dialog Box appears.

   - Select the outline of a jumper, right-click and click the **Properties** popup menu item.

   - 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 the **Properties** popup menu item.

   - Select the jumper name and then right-click and click Properties. In the Jumper Name Properties Dialog Box, click the **Jumper** button.

2. In the Jumper Properties Dialog Box, view properties or make changes.

3. Click **OK**.

Related Topics

Modifying Jumper Name Properties

Modifying Jumper Pin Properties

Modifying Jumper Name Properties

Open the Jumper Name Properties to change the value or properties of the reference designator.

Procedure

1. Use one of the following methods to access the Jumper Name Properties:
   - Select a jumper reference designator. Right-click and click the **Properties** popup menu item.
   - Select the jumper. Right-click and click the **Properties** popup menu item. The Jumper Properties Dialog Box appears. Select the name to modify from the Label list. Click the **Label** button.
2. In the Jumper Name Properties Dialog Box, view properties or make edits.
3. Click **OK**.

Related Topics

Modifying Jumper Pin Properties

Modifying Jumper Properties

Modifying Jumper Pin Properties

In the Jumper Pin Properties, you display jumper pin information, change the coordinates of a jumper pin, and activate the pin as a test point.

Procedure

1. Select the jumper pin.
2. Right-click and click the **Properties** popup menu item. The Jumper Pin Properties Dialog Box appears.
3. View Jumper Pin properties or make changes.
4. Click **OK**.

Results

If you change the pad stack of a jumper pin that is a locked test point, the Warning: Test Point Locked dialog box on page 995 appears.

Related Topics

Modifying Jumper Name Properties

Modifying Jumper Properties

Operations While Routing

You can access certain operations or features while routing in SailWind Layout.

Trace Length Monitor

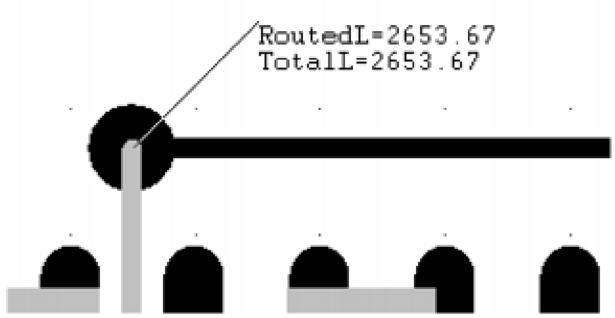
The trace length monitor calculates and shows the trace length (on the status bar and near the pointer) as you route. When design rules for the net specify a length, then red or green text in the monitor indicates whether the length is within the specified rules.

- Red text indicates violations of the rule.
- Green text indicates trace lengths within the rule.

The status bar shows detailed length information.

The trace length monitor shows the routed length and the estimated total length of the net. For more information, see "Trace Length Monitor".

Figure 103. The Trace Length Monitor



Color Customization for the Trace Length Monitor

You can set the colors of the text in the monitor that indicate whether the length is within the specified rules. Set the colors in the *SailWindpcb.ini* file. In the *.ini* file, look for the [monitor] section that resembles the following:

```
[monitor]
```

```
normal = 10
```

```
norules = 15
```

```
error = 12
```

The numbers represent colors from the [Colormap] section of the *.ini* file. To change the colors used in the monitor, change the numbers in the [monitor] section to a color from the [Colormap] section. Values may be 0-31.

Effects of Reroute and Smooth on Trace Length

During reroute operations, the trace length monitor may show a length greater that the actual final length. In addition, for dynamic routing, smoothing operations and pad entry corrections may be performed on traces after you finish routing (depending on your Options dialog box > **Routing** category > General subcategory on page 1542 settings). These operations may change trace length slightly.

Trace Length Monitor Reports

As mentioned, the trace length monitor shows the net length rules, the current routed length, and the estimated total length. The rules that are reported are net length rules. As shown in Table 118, the trace length monitor reports different information depending on the length rule.

Table 118. Trace Length Monitor Reports

Length Rule	Information in Report
Minimum/Maximum Length Rules	Shows the current routed length, the total length, the minimum net length rule, and whether it's applied at the net or pin pair level of the rules hierarchy.
Matched Length Rules	Shows the current routed length, the total length, and the matching net length rule.
Differential Pair Rules	Shows the current routed length, the total estimated length, the minimum length rule, the maximum length rule, the gap rule, and whether it's a net or pin pair rule.
No Length Rules	Shows the current routed length and the total estimated length.

Activating 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

1. Click the **Tools > Options** menu item.
2. In the Options dialog box, on the **Routing** category > General subcategory on page 1542, select the Show Trace Length check box.
3. Click **OK**.

   This turns the display of the trace length monitor on or off at the pointer only. The trace length always displays on the status bar.

   Tip
   You can also use the shortcut on page 100 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.

Related Topics

Trace Length Monitor

Routing Direction

When you start routing, SailWind Layout automatically chooses the end from which you start. If you want to start from the opposite end of the connection, right-click and click the **Swap End** popup menu item. Also, at any point while you are routing, click (add a corner) to preserve what you have already routed and switch to the opposite end of the connection to route from the opposite direction.

Swap End is available only in Add Route on page 699 and Dynamic Route on page 700 modes.

Layer Selection for Starting a Trace

When you start routing a trace, it assumes the layer of the object from which it began the connection, such as a pad, via, or copper area. If it starts routing from a through-hole pad or a via, you can set the current routing layer before you start routing.

Choose one of the following methods to set the layer before routing:

- On the Standard Toolbar, in the Layer list, choose the layer.
- Change the current layer by typing the L modeless command on page 83, typing the layer number, and pressing the Enter key.

A layer's current direction setting appears next to its entry in the Layer list. 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

You can add a via during routing to either continue routing on another layer or to connect to a plane area.

Prerequisites

- You must have created a via type for use with your design. For more information, see "Creating a Through-hole Via Type" and "Creating a Partial Via Type".
- The via must be available for use, and both source and destination layers must be available for routing according to the applicable Routing Rules on page 1666.

Procedure

1. While routing, use one of the following methods to add a via:

   - Right-click and click the **Add Via** popup menu item to continue routing on another layer.
   - Click to place a tack and then change to the desired new layer on page 738 to continue routing. The via is added automatically in the location of the tack.

   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.

Tip
Instead on continuing routing after you add a via, you can end the trace with a via. You would use this method to fan out a plane net to the plane layer. While routing, right-click, point to the **End Via Mode** popup menu item, and then click the **End Via** popup submenu item to set the mode. Right-click and click the **End** popup menu item to end the trace with a via. 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 should not connect.

2. Continue routing on the new layer.

Related Topics

Troubleshooting Routing on Another Layer

Via Types

End Via Mode

The Layer Pair

Creating Arcs

While routing, you can create arcs instead of corners.

Procedure

1. While routing, right-click and click the **Add Arc** popup menu item. 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.

Related Topics

Stretching an Arc or Miter

Layer Changes While Routing

To change layers while routing, use the L modeless command while the trace is active. The “L” command installs a via at the last corner and moves the segment of your trace (stretching from the via to the pointer) onto the specified layer. The Via Type dialog box settings control the type of via you place. The trace color changes accordingly and the new layer appears in the Layer list on the Standard Toolbar.

If the layer change is allowed, a via is installed at the last corner when you change layers either while routing or after entering the trace. You can use the design rules to use a specific via by default. Since partial vias undergo a checking process, you cannot set them this way by default; they are chosen automatically depending on the circumstances of layer change.

Conditions for Changing Layers

Operations for changing layers in interactive routing are the same for the Route command and Dynamic Route Editor.

When you change layers, you must meet several requirements before you can place a via. First, the layer change must be a legal drill pair, the start and finish layer numbers. This check affects multiple layer boards. Establish pairs by clicking the **Setup > Drill Pairs** menu item, according to your projected manufacturing plan. Once you enter drill pairs, you are warned if you are joining two layers that do not match your manufacturing scheme.

For more information on setting up through hole vs. partial vias, see Via Setup.

Layer Change Methods

There are several ways to install a via and change layers while routing:

- If you are working extensively between two layers, set them as the default “Layer Pair” in the Options dialog box > **Routing** category > General subcategory on page 1542. You can also use the PL modeless command on page 83. While routing, right-click and click the **Layer Toggle** popup menu item or press the F4 key.
- Type the L modeless command on page 83, the new layer number, and press the Enter key.
- Right-click and click the **Layer** popup menu item. This opens the Modeless Command window with the L command already typed for you. After the L command, type the new layer number and then press the Enter key.
- On the Standard Toolbar, in the Layer list, select the new layer.

The type of via placed is determined by the Vias Dialog Box settings. You can also use the V modeless commands on page 83.

Related Topics

Adding a Via While Routing

Changing the Via Type While Routing

While routing, you can change the type of via that SailWind Layout uses for via additions.

Procedure

1. While routing, right-click and click the **Via Type** popup menu item.

   The Vias Dialog Box appears. You can leave the Vias dialog box open to quickly change from one via type to another.

2. In the Via Mode area click a via type.
3. If the via is a through hole, click the via type to use in the Via List list.

   If you click automatic, SailWind Layout selects a via type based on which via types are allowed for the net in the Routing Rules on page 1666 dialog box and which via types are legal for the layer change according to the Drill Pairs Setup on page 1337 dialog box. If more than one via type qualifies on both counts, the product uses the smaller drill diameter and/or smaller pad size.

4. Click **OK**. The new via type is used until you change it.

Changing 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 “Recommended” trace width setting on page 1167 for the net stays unchanged. If you end with a partial route and use it later, the width you set separately is still in effect.

Procedure

1. While routing, type the W modeless command on page 83. This changes the “Default width” setting in the Options dialog box > **Drafting** category > Text and Lines subcategory on page 1553. You can also change the width in the Grid/Width dialog box on page 1398.

2. Type the new width and then press the Enter key. The line width area on the status bar changes accordingly.

Related Topics

Changing the Width of an Existing Trace

Troubleshooting Routing on Another Layer

You can use a via to switch to another layer while routing but many rules or settings 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 on page 1666 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 on page 1666 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 on page 1708, 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 Options dialog box > Global category > General subcategory on page 1531, 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?

Traces Ended on Different Nets

To end a trace on a segment with a different netname, use the ECO Toolbar routing command. SailWind Layout assumes you are intentionally joining two nets to make one, and considers it an engineering change. SailWind Layout records it 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. For instructions, see “Adding a Route in ECO Mode”.

Refined Object Selection

You can use additional selection filter shortcuts after selecting a design object. Right-click and click a selection shortcut.

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.

Procedure

1. Select an object.
2. Right-click and click the **Select Net** popup menu item, or press the F6 key. The entire net is selected, including routes, connections, and pins.

Selecting Pin Pairs From an Object

You can select all pin pairs attached to a pin, trace segment, corner, tack, or via.

Procedure

1. Select an object.
2. Right-click and click the **Select Pin Pairs** popup menu item, or press the F5 key.

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.

For more information about classes, see "Design Rule Hierarchy."

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.

Procedure

1. Select a net.
2. Right-click and click the **Select Class** popup menu item.

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.

For more information about groups, see "Design Rule Hierarchy".

Procedure

1. Select a pin pair of a group.
2. Right-click and click the **Select Group** popup menu item. All pin pairs of the group are selected.

Selecting Drafting Objects from Segments/Corners

You can use the Select Shape command to select an entire drafting object from one of its segments or corners.

Procedure

1. Select the drafting segment or corner.
2. Right-click and click the **Select Shape** popup menu item.

Operations After Routing

SailWind Layout offers detail tools to help you clean up or modify routed trace patterns, perform a 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 View Mode, modeless command on page 83 T.

Moving a Trace Segment to Another Layer

Instead of rerouting an entire trace, you can move a trace segment to another layer and allow SailWind Layout to add vias automatically at the opposite ends of the trace segment.

Procedure

1. Select one or more continuous trace segments.
2. Right-click and click the **Properties** popup menu item.
3. In the Trace Properties Dialog Box, select the layer you want from the Layer list.
4. Click **OK**.

Results

The trace segment moves to the selected layer and vias are automatically placed at the segment end points.

If nothing happens, design rules may have been violated. For example, there may not be enough room for a via or traces may not be permitted on the layer you have chosen.

Rerouting 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.

Prerequisites

- 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 on page 1666 dialog box for the net and select Copper Sharing. To enable sharing for all nets, in the Rules dialog box, on page 1674 click **Default** and select Copper Sharing in the Routing area.

Procedure

1. Select a trace segment at the point where you want to start the reroute.

   Tip
   Use Transparent View Mode, modeless command on page 83 T, to view obstacles that may lie under traces on the current active layer.

2. Right-click and click the **Route** or **Dynamic Route** popup menu item. The new trace begins at the selection point.

- 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.

3. Route the new trace segment. 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-connect. 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.

4. To complete the route, double-click or right-click and click the **Complete** popup menu item.

Rerouting with Sketch Route

Use Sketch Route to reroute a set of trace segments between two pins. A pin in this case may be a component pin, via, or tack.

Restrictions and Limitations

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.

Procedure

1. Type the drp modeless command on page 83 and press the Enter key to enable the online design rule checking Prevent mode and the **Sketch Route** button.

   For more information, see “Turning on Design Rule Checking,” and “Design Rule Checking.”

2. Activate the Sketch Route command using one of the following:

   - Object mode — select the first segment of the series you want to edit, then right-click and click the **Sketch Route** popup menu item.

   - Verb mode — on the Design Toolbar, click the **Sketch Route** button and then click on a trace segment at the point where you want to change the trace pattern.

   Restriction:
   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.

3. A thin dotted line appears that moves with the pointer. Use this line to mark the general location of the new trace. You do not have to enter any corners.

4. Click to indicate the end of the sketched route. This completes the trace, routing through the defined path.

Smoothing 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 with the Dynamic Route Editing licensing option.

Procedure

1. Type the drp modeless command on page 83 and press the Enter key to enable the online design rule checking Prevent mode.

   For more information, see "Turning on Design Rule Checking," and "Design Rule Checking."

2. Select a trace segment.

3. Right-click and click the **Smooth** popup menu item.

> **Restriction:**
> 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.

Changing the Pad Entry Angle

After you complete routing a connection from pin to pin, you can change the pad entry angle into the pin.

The Trace/Line Angle setting in the Options dialog box > Design subcategory on page 1503 determines the angle of pad entry when you complete a trace. You can also use Trace/Line Angle in the Status Dialog Box.

Procedure

1. Select the segment.

2. Right-click and click the **Pad Entry** popup menu item.

   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.

> **Restriction:**
> 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.

Copying and Pasting 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. The selection includes layer changes and vias in the path.

Memory patterns or SMD fanouts are ideal applications for copied routes. When you select and copy a trace or a trace and via combination, you start a specialized operation that immediately attaches a copy of the selected routing to the pointer and lets you repeatedly paste with a mouse click.

Procedure

1. Select anything from a single trace segment up to a pin pair in length.

   Your selection can include vias. The copy feature does not work on trace patterns longer that a single pin pair. Select multiple items by Ctrl+click. Selections can include vias.

2. Click the **Edit > Copy** menu item, or press Ctrl+C.

   All included segments and vias are copied and attached to your pointer in Move mode.

3. If needed, right-click and use the shortcut menu commands to rotate or flip the copy.

4. Position the pointer over the location of 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. 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.

   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 the Esc key to cancel further copies.

Related Topics

Copying and Pasting Routes in BGAs

Cut, Copy, and Paste Commands

Creating 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 the **Route Loop** popup menu item.

   A new route starts at the selection point. You can also start Route Loop using the Ctrl+J shortcut keys.

   Restriction:
   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.

   ---

3. Indicate the location for each corner of the new route.
4. Indicate the location where the loop branches back into the existing route.

Moving a Trace Segment

You can move traces if you need to make room for other routing objects.

Procedure

1. Select the trace segment.
2. Use one of these methods to start the move.
    - Right-click and click the **Move** popup menu item.
    - Press Ctrl+E.
    - Select the trace segment and drag. Use the Options dialog box > Global category > General subcategory on page 1531 to click a drag and drop method, or to prohibit pointer-drag moves.

    The selection attaches to the pointer.
3. Click to indicate the new route location.

Related Topics

Stretching an Arc or Miter

Trace Shove During a Move

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, when the trace is in Move mode, right-click and click the **Shove** popup menu item to enable the shove feature.

This feature requires that you have the Dynamic Route Editing installed option.

The following restrictions apply:

- If you set DRC to Prevent or Warn, you cannot 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. For more information, see “Modifying a Route Attached to a Locked Test 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.

Tip
Use Transparent View Mode, the modeless command on page 83 T, to view obstacles that may lie under traces on the current active layer.

Deleting a Trace Segment

You can delete trace segments to restore the unroute connection.

Procedure

1. Select the segment.
2. Press the Delete key, or click the **Edit > Delete** menu item. When you delete a trace segment, an unroute connection line is restored.

   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.

Unrouting All Segments Attached to the Pads of a Component

You can unroute the first segment attached to each pin of a component. Use the “Unroute Attached Segments” command to reroute each connection that terminates at the part, or when replacing a component in the design. When you unroute a segment, tacks appear at the end of the connection where it meets the route. If the unrouted segment ends at a via, SailWind Layout creates a connection between the via and the unrouted pin.

Procedure

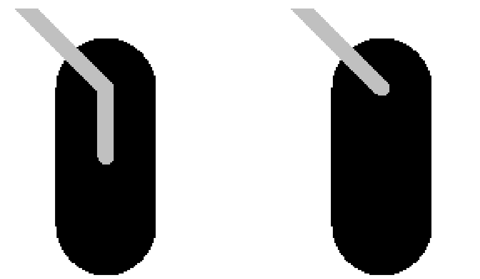
1. Select the component from which you want to unroute attached segments.
2. Right-click and click the **Unroute Attached Segments** popup menu item.

   The first segments are unrouted and tacks appear at the end of the routes.

Restriction:
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.

Results

When you unroute all final segments into a component, in some cases it may appear as though a trace is still connected to a pad. This occurs when the final two segments end within the bounds of the pad. See the example in the graphic below.



Changing the Width of an Existing Trace

You can change the width of routed traces at the segment, pin pair, or net level.

Tip
You can also use the **Edit > Find** menu item to find all traces of a similar width, select them, and change them to a new width.

Procedure

1. Select a segment, pin pair, or net.
2. Right-click and click the **Properties** popup menu item.
3. In the Trace Properties on page 1757, Pin Pair Properties on page 1627, or Net Properties dialog box on page 1487, in the Width or Trace Width box, type the new width and click **OK**.

Restriction:
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.

Creating Arc Miters

You can create a small, smooth arc effect by mitering a routed 90 degree corner. Such an ability is useful if you need to automatically add arcs to many corners in the design.

If you want to manually add an arc, see “Converting a Trace Corner to an Arc”.

Procedure

1. Click the **Tools > Options** menu item. The Options dialog box opens.
2. On the Design category on page 1503, in the Miters area, click Arc.
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 the **Add Miter** popup menu item.

   Trace segments that are too short for the specified ratio are not mitered. You can also select one or more pin pairs, right-click and click **Add Miter**.

   Restriction:
   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.

Converting a Trace Corner to an Arc

Use the “Convert to Arc” command to create an arc from two intersecting route segments. You can manually pull the arc to the radius that you need.

If you need to automatically add arcs to many corners in the design, see “Creating Arc Miters”.

Procedure

1. Click the **Tools > Options** menu item. The Options dialog box appears.
2. On the Design category on page 1503, in the Miters area, use the Angle value to control 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.
3. Select the corner of a trace.
4. Right-click and click the **Convert to Arc** popup menu item.

   Note:
   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.

   ---
5. Move the pointer to adjust the radius of the new arc and click to finish.

Results

If you need to adjust the radius of a created arc or to convert the arc back to a corner, select the arc segment, right-click and click Stretch on page 753.

Stretch Command

Use the Stretch command to stretch arcs or miters and to move route segments.

Stretching an Arc or Miter

Use Stretch to expand or contract a miter or arc on a trace segment.

Procedure

1. Select the miter segment or arc.
2. Right-click and click the **Stretch** popup menu item. The object attaches to your pointer.

   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. For more information, see "Modifying a Route Attached to a Locked Test Point."

   Restriction:
   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.

3. Move the pointer to expand or contract the object. Click outside the point of intersection of the two lines to return the arc or miter to a corner.
4. Click to finish.

Using Stretch to Move a Route Segment

Use Stretch to move a straight route segment that has corners at the end points.

Tip
To modify arcs and miters of drafting objects, use the Pull Arc on page 640 and Move Miter on page 639 commands.

Restrictions and Limitations

You cannot use Stretch to move a straight route segment with an arc at its end point.

Procedure

1. Select the trace segment.
2. Right-click and click the **Stretch** popup menu item. The segment attaches to your pointer.

If you move or segment 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. For more information, see "Modifying a Route Attached to a Locked Test Point."

Restriction:
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.

3. Move the pointer to move the segment. The corner angles are maintained and the length of the segment is adjusted during the stretch. If you stretch a segment so that it reaches another segment's corner, corner angles may change.
4. Click to finish.

Related Topics

Creating Arcs

Creating Arc Miters

Moving a Corner

You can move any corner on a trace or drafting object.

Procedure

1. Select the corner to move.
2. Right-click and click the **Move** popup menu item.

   When moving a corner of a drafting object, you can also use the Move button on the Design Toolbar. Click to select the corner and it attaches to your pointer.

Restriction:
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.

3. Optionally, you can right-click and apply any of the following:
   - **Convert To Arc** — To convert the corner to an arc. The corner becomes an arc and remains attached to your pointer. Move the arc to the desired position, and click to complete.
   - **Shove** — To shove other routes as you move the selected corner.
   - **Orthogonal**, **Diagonal** or **Any Angle** — To select an angle setting.
   - **Ignore Clearance** — To ignore clearance rules regardless of the On-line DRC setting on page 1503 in the Options dialog box > **Design** category.

4. Move the corner with the pointer.

5. Click to place the corner at the new location.

Related Topics

Deleting a Corner

Modifying Drafting Corner Properties

Moving a Via or Tack

You can use the pointer to move a via or tack in your design.

Procedure

1. Select the via or tack.

2. Right-click and click the **Move** popup menu item.

   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. For more information, see "Modification of a Via or Virtual Pin That is a Locked Test Point" or "Modifying a Route Attached to a Locked Test Point."

   > **Restriction:**
   > 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.

3. Optionally, you can right-click and apply any of the following:

   - **Shove** — To shove other routes as you move the selected tack or via.
   - **Orthogonal**, **Diagonal**, or **Any Angle** — To select an angle setting.
   - **Ignore Clearance** — To ignore clearance rules regardless of the On-line DRC setting on page 1503 in the Options dialog box > **Design** category.

4. Move the pointer to reposition the via or tack. 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.

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 SailWind Layout you can end the route on a via by choosing End With Via from the Route shortcut menu. This creates a dangling via and dangling route.

Procedure

1. Select one or more nets in the design.
2. Right-click and click the **Select Dangling Routes** popup menu item.
3. Press the Delete key or right-click and click the **Unroute** popup menu item.

> **Restriction:**
> 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.

Splitting a Trace

The Split command divides a selected line segment into two segments and makes one of the segments move dynamically with the pointer by adding a third segment at the location of the split.

Procedure

1. Begin the split using one of the following methods:

   - Object mode — select a trace segment, right-click and click the **Split** popup menu item.
   - Verb mode — on the Design Toolbar click the **Split** button and then select a trace segment.

   The segment dynamically attaches to the pointer. The current Angle Mode setting effects the movement of the attached segments.

   > **Restriction:**
   > 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.

2. Optionally, while splitting a trace, you can switch the trace side that is attached to the pointer. Before completing the split, right-click and click the **Swap** popup menu item. The pointer attaches to the other end of the segment.
3. Click to indicate where to split the segment. One of the two segments is divided into two segments.
4. If you selected Verb mode, add a split to another trace segment or press the Esc key to cancel the Split mode.

Adding a Corner to a Trace Segment

Add Corner divides the selected line segment into two segments and makes both line segments move dynamically with the pointer to create the new corner.

Procedure

1. Begin the corner using one of the following methods:

   - Object mode — select a trace segment, right-click and click the **Add Corner** popup menu iem.

   - Verb mode — on the Design Toolbar click the **Add Corner** button and then select a trace segment.

   The segments dynamically attach to the pointer with the corner at the position of the pointer. The current Angle Mode setting effects movement of the attached segments.

   Restriction:
   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.

   ---

2. Click to indicate the location for the new corner.

   When On-line DRC is set to Prevent Errors and errors occur, (guard band on page 1832 appears), the corner is added at the closest legal position to the indicated point.

Adding Vias to an Existing Trace

Add a feed-through via to the selected trace at the selection point location. This action does not change the current layer setting.

Procedure

1. Select a trace segment, trace corner, or trace end point at the location of the via you want to place.

2. Right-click and click the **Add Via** popup menu item.

   The via is added at the point where you selected the trace object. The via is a plain via. The Stitching or Glued settings are not enabled for the via. To add traces to unrouted connections, pin pairs or nets, see “Adding Stitching Vias”.

   Note:
   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.

Stitching Vias

Use stitching vias to interconnect copper areas of nets including copper planes or CAM planes.

Note:
SailWind Layout does not consider stitching vias to be orphaned vias and does not delete them during routing optimization. If you use the Select Dangling Routes command, it will select stitching vias. Glue stitching vias to keep them from being deleted if you select dangling routes and delete them as part of your design process.

Adding Stitching Vias

You must associate stitching vias with a net. You can add them to the design through the net selection of an object in the design.

Stitching vias interact with many commands in SailWind Layout:

- The All Except Connected Plane Nets option in the View Nets dialog box disables unroute display to stitching vias that are embedded in copper planes, or CAM planes. Also, Length Minimization calculates both the pads and drills of the stitching via to determine the unroute visibility to a stitching via.
- When you copy trace patterns, stitching vias are also copied. Copied stitching vias snap to the via grid when pasted.

Procedure

1. Select a net (you can click trace segments, vias, pins, unroutes, copper, and copper planes), pin pair, or unroute in the design.
2. Right-click and click the **Add Via** popup menu item to attach the new via to the pointer.
3. Click to place the stitching via.

   Another stitching via attaches to the pointer. You can place vias repeatedly. If unroutes are visible for the selected net, a new unroute will connect to the stitching via.
4. Right-click and click the **Cancel** popup menu item to stop adding stitching vias.

Results

When you add a via to an unroute, pin pair or net, the result is a via with a Stitching setting enabled. By default, stitching vias added to the design are always given the Stitching setting as they are added. Vias added to traces are not. When you add a via to a trace segment or trace corner, the result is a plain via (unglued). For more information, see “Adding Vias to an Existing Trace”.

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 Options dialog box > Design category on page 1503.

Related Topics

DRC and the Via Stitching and Shielding Operations

Selecting Stitching Vias

You must enable the selection of stitching vias before you can select a stitching via.

Procedure

1. With nothing selected, right-click and click the **Filter** popup menu item.
2. In the Selection Filter, on the **Object** tab, select the Stitching vias check box. Clear the check boxes of other object types for easier selection of the Stitching vias.
3. Click **Close**.
4. Select a stitching via.

Changing 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.

Procedure

1. Select one or multiple stitching vias.
2. Right-click and click the **Properties** popup menu item.
3. In the Via Name list, select the correct via.

Tip
You can use the Pad Stacks dialog box to create new vias.

Deleting Stitching Vias

Delete stitching vias by pressing the Delete button on the keyboard.

Procedure

1. Select one or more stitching vias on page 1862 to delete.
2. Press the Delete key. A dialog opens with the message, “Delete all orphaned vias?”. Click **Yes** to delete the stitching via.

Converting Routing Vias into Stitching Vias

Vias that you add to a trace segment, corner, or end point, are not considered stitching vias but routing vias. Like stitching vias, you can use these routing vias to interconnect areas of a net, but SailWind Layout does not consider them plain vias. Enable the Stitching setting to make a routing via into a stitching via.

Procedure

1. Select one or multiple vias.
2. Right-click and click the **Properties** popup menu item.
3. Select the Stitching check box.
4. Click **OK**.

Related Topics

Gluing a Via

Stitching Vias

Adding a Test Point to an Existing Trace

You can manually add a test point along an existing segment or trace of a route. Test points, by default, do not glue in a design. You can access them from the bottom.

Tip
You can also make an existing via, pin or virtual pin a test point. See the Test Point check box in the following object Properties dialog boxes: Via Properties Dialog Box, Pin Properties Dialog Box, Jumper Pin Properties on page 1422

Procedure

1. Unless the via used in the design can also be used for test points, create a via on page 329 to use as a test point.
2. If there is more than one via available to the design, you must specify the via that should be used for test points.
   a. Click the **Tools > DFT Audit** menu item.
   b. In the DFT Audit dialog box on page 1273, select the Add Test Points to Existing Traces check box. This check box must be selected to choose a test point in the Use Test Point Via list, but is not required to be active to add test points to traces in SailWind Layout.
3. Use one of the following methods to add a test point:

- Object mode — Select a trace segment or corner. You cannot use multiple selection when adding test points to a route. Right-click and click the **Add Test Point** popup menu item.

- Verb mode — On the Design Toolbar, click the **Add Test Point button**. Select any trace segment or corner to which you want to add a test point. Repeat or right-click and click **Cancel** to exit Add Test Point mode.

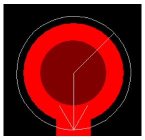
4. A test point is added to the route.

   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.

Restriction:
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.

Results

When the via or pin is flagged as a test point, and the Show Test Points check box is selected on the Options dialog box > **Routing** category > General subcategory on page 1542, an arrow marker overlays the via:



Related Topics

Performing a Test Point Audit

Deleting a Corner

You can delete corners of traces or drafting objects.

Procedure

1. Select the corner.

2. Press the Delete key. The two segments joined at the corner become one segment.

Restriction:
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

Moving a Corner

Modifying Drafting Corner Properties

Deleting a Via

You can remove a via by selecting it and pressing the Delete key.

Procedure

1. Select the via.
2. Press the Delete key. The via is deleted and the layers of the attached segments are changed if necessary.

Note:
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.

Gluing a Via

Glue a via to prevent it from being moved for any reason.

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.

Procedure

1. Select a via, right-click and click the **Properties** popup menu item.
2. Select the Glued check box.
3. Click **OK**.

Results

If you also select the Test Point check box, 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.

Deleting Miters

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.

Procedure

1. Select the routed pin pairs, drafting paths, or polygons.
2. Right-click and click the **Delete Miters** popup menu item. All miters in the selected path are deleted and replaced by a 90-degree corner.

Deleting a Route from a Pin Pair

You can delete all traces and vias in a routed pin pair.

Procedure

1. Select the pin pair.
2. Press the Delete key. A ratsnest line between the two pins replaces the routed pin pair.

Connecting SMD Pads to Planes

Routing an SMD pad to a plane involves placing a via to the plane under or somewhere near the pad.

Vias you add to plane nets receive a thermal relief pad for the plane layer based on the either the design rules or the pad stack. For more information, see "Design Rule Versus Pad Stack - Thermals and Antipads" on page 797.

Prerequisites

- You must have created a via type for use with your design. For more information, see "Creating a Through-hole Via Type", and "Creating a Partial Via Type".
- The via must be available for use, and both source and destination layers must be available for routing according to the applicable Routing Rules on page 1666.

Procedure

1. Optionally, if you are fanning out power and ground connections, you can hide all unrouted connections except the power and ground nets and give the pads of those nets unique colors to quickly identify them in the design:
    a. Click the **View > Nets** menu item.
    b. Select "Default" in the View List list.
    c. Select the "Traces Plus the Following Unroutes" check box, if necessary, and in the View Unroutes Details area, click **None**.
    d. In the Select By list, click Plane Nets. Locate your plane nets in the Net List and add them to the View List.
    e. Select the plane nets and in the View Unroutes Details area, click "All Except Connected Plane Nets". This shows full pin-to-pin connections and partially routed copper traces, but not partial connections.
    f. Select each plane net individually and in the Color by Net area, give each net a unique color.

g. Click **OK**.

All the unrouted connections from the display except for the plane net connections are visible and the pads of those nets have unique colors. These turn off as you link them to the plane with vias.

2. Add a via using one of the following methods:

- Fanout via:

  i. Route a small trace segment out from the pad.

  ii. While routing, right-click, point to **End Via Mode**, and then click the **End Via** popup menu item to set the mode.

  iii. Right-click and click the **End** popup menu item to end the trace with a via. 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 should not connect.

- Via in pad:

  i. In the Clearance Rules Dialog Box, in the Same Net area, the SMD to Via clearance must be 0 (zero) at the required level of the rules hierarchy to enable the via to be placed in the pad.

  ii. Select the pad, right-click and click the **Add Via at SMD** popup menu item. The via is added to the center of the SMD. Component pads always overlay vias and it may be difficult to determine if the via is actually inside the pad. Enable Transparent mode with the T modeless command on page 83 to see vias in pads.

Results

When you install the via, either under the pad or a short distance away, the connection for that pin-to-pin unroute connection disappears if you route the connection to the plane layer. For more information, see "Check the Plane Connection for Continuity."

High Speed and RF Routing Features

Read the topics that follow to learn more about RF features for use when routing.

Converting a Trace to a Copper Chamfered Path

Converting traces to a copper chamfered path has two advantages over simply creating a copper chamfered path. You can use the more powerful interactive router to initially route the trace, and when you convert the trace to a chamfered path, SailWind Layout assigns the net automatically.

Procedure

1. Select a pin pair, multiple pin pairs, or a net.

   Unrouted or partially routed pin pairs, or pin pairs belonging to reuse blocks, are excluded from selection.

2. Right-click and click the **Convert to Chamfered Paths** popup menu item.

   Alternatively, you can click  under the RF Toolbar.

3. In the Convert Pin Pairs to Chamfered Paths dialog box, select the pins pairs desired using Ctrl +click, Shift+click, or the **Select All** and **Unselect All** buttons as shortcuts.

   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.

   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. Specify a trace width using one of the following:

   - 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.

   - 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.

   For example, 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. Specify the degree of corners to chamfer using one of the following:

- Clear the “Chamfer corners with angles less than or equal to” check box to chamfer only angles less than 90 degrees (acute angles).

- 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**. 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

Drafting Object Properties

Restoring Traces After Conversion to Copper Chamfered Paths

Converting original traces to copper unroutes the traces. You must re-route traces to replace chamfered paths.

Procedure

1. Use the Select Nets selection filter to select the net.

2. Right-click and click the **Select Chamfered Paths** popup menu item.

   Alternatively, you can click  under the RF Toolbar.

3. Press the Delete key.

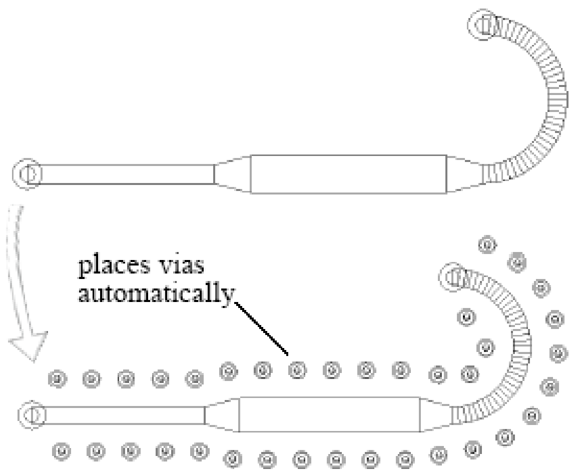
   Vias in the chamfered path are not deleted since they are glued during conversion from trace to chamfered path.

4. Re-route the pin pair, pin pairs, or net.

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:



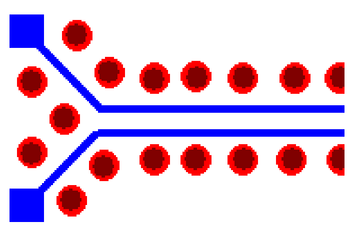
Procedure

1. If you have not done so, set the options for shielding on the Options dialog box > Via Patterns category on page 1556.

   If you need to create a new via type for shielding, see “Via Setup”. Your Via Pattern settings are stored for future operations. You need to perform this step only if you want to change the settings you selected previously.

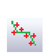
2. Select a net, pin-pair or copper area.

   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 the **Add Via Shield** popup menu item.

   Alternatively, you can click  under the RF Toolbar.

Results

Vias are placed around the selected net, pin-pair, or copper. If there is adequate room for the vias, but no vias appear, your View > Nets menu item, View Details setting on page 1786 could be set to not show Traces Plus the Following Unroutes.

If the Design Rule Checking setting is Prevent Errors, the Add Via Shield operation does not add any via that violates design rules.

When you edit traces, SailWind 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

DRC and the Via Stitching and Shielding Operations

Properties of Routing Objects

Use the Properties dialog boxes to make selective modifications to any routing objects.

- 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

In the Net Properties dialog box, view net information, control the protection of the net and unroutes, control the trace width, and electrical net status.

Procedure

1. Select a net, right-click and click the **Properties** popup menu item.
2. In the Net Properties dialog box on page 1487, modify settings if necessary.

   Several of the options in this dialog box are unavailable if the net is part of a physical design reuse.
3. Click **OK**.

Related Topics

View Nets Dialog Box

Modifying Pin Properties

In the Pin Properties dialog box, view pin information, and control teardrop, plane thermal, and test point status.

Procedure

1. Select a pin, right-click and click the **Properties** popup menu item.
2. In the Pin Properties Dialog Box modify settings if necessary.

   Several of the options in this dialog box are unavailable if the pin pair is part of a physical design reuse. 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. For more information, see "Modification of a Via or Virtual Pin That is a Locked Test Point".
3. Click **OK**.

Modifying Pin Pair Properties

In the Pin Pair Properties dialog box, view pin pair information, control protection of traces and unroutes, and control trace width.

Procedure

1. Select a pin pair, right-click and click the **Properties** popup menu item.
2. In Pin Pair Properties Dialog Box, modify the settings, if necessary.
3. Click **OK**.

Modifying Trace Corner or Tack Properties

In the Trace Corner or Tack Properties dialog boxes, view the trace corner or tack information, and modify coordinates.

Procedure

1. Select a trace corner, right-click and click the **Properties** popup menu item.
2. In the Tack Properties or Trace Corner Properties dialog boxes on page 1744, modify the settings, if necessary.

   Several of the options in this dialog box are unavailable if the corner is part of a physical design reuse. If you select another trace corner while the dialog box is open, the selected corner updates with the information.
3. Click **OK**.

Related Topics

Design Rule Hierarchy

Modifying Via Properties

In the Via Properties dialog box, view via information, modify coordinates, change via types, set the glued, stitching, plane thermal, and test point status.

Procedure

1. Select a via, right-click and click the **Properties** popup menu item.
2. In the Via Properties Dialog Box, modify the settings, if necessary.

   Several of the options in this dialog box are unavailable if the via is part of a physical design reuse. If you select another via while the dialog box is open, the information updates for the selected via. 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. For more information, see "Modification of a Via or Virtual Pin That is a Locked Test Point".
3. Click **OK**.

Modifying Trace Segment Properties

In the Trace Properties dialog box, view trace segment information, modify the coordinates, specify the trace width, or change the layer on which the segment resides.

Procedure

1. Select the trace segment, right-click and click the **Properties** popup menu item.
2. In the Trace Properties Dialog Box, modify the settings, if necessary.

   If you change the layer, vias are automatically added at the segment endpoints. If you select another trace segment while the dialog box is open, the information is updated for the selected segment. Several of the options in this dialog box are unavailable if the trace is part of a physical design reuse.
3. Click **OK**.

Troubleshooting Constraints While Routing

When you are routing and the design rules constrain you, you can view the rule hierarchy to which the object is subjected and access the rules for a design object (class, net, group, pin pair, component, or unroute). You can do so by selecting the object in the design and displaying its 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.

Procedure

1. Select the object whose rules you want to display.
2. Unless you are already selecting a pin pair or net, right-click and click the **Select Pin Pair** popup menu item.
3. Right-click and click the **Properties** popup menu item.
4. In the Pin Pair Properties Dialog Box, in the Rules Data area, view the level from which the Pin Pair is getting the design rules.
5. Click the **Rules** button to access the Pin Pair level of design rules.

   If you need to access a different level of the hierarchy, close all open dialog boxes and click the **Tools > Rules** menu item.

Clearance and Checking After Routing

Trace Clearance is the distance allowed between a trace edge and the edge of another conducting item. You can run a verification check to find clearance violations.

Assign clearance values in the Clearance Rules Dialog Box. If you have the Advanced Rules Installed Option on page , you can assign non-default clearances on the pin pair, pin pair group, net class, and all 25 levels of the conditional rules.

Use the **Tools > Verify Design** menu item menu to search the design for clearance violations. Verify Design marks each violation and enables you to center the view on each violation. Verify Design only checks for clearance violations on what you choose to display in the design. If you want to check the entire design for clearance violations, make sure colors are applied to all object and that nothing is invisible in the design. Also bring all design items into view by zooming out to the point where all items are in view.

Setting up clearances for traces, as well as pads and vias, is described in "Design Rules on page 1821". Checking for errors (spacing errors, airgap errors, and high-speed problems) is discussed in "Verify the Design".

Related Topics

Design Rule Hierarchy

Chapter 33
SailWind Router

This section discusses using SailWind Router in your design process. You can connect to SailWind Router from within the SailWind Layout environment. Once connected, you can launch a SailWind Router autorouting session or you can initiate Synchronization Mode where you can work interactively and simultaneously in SailWind Layout and SailWind Router.

Synchronization Mode

Synchronization Mode links SailWind Layout and SailWind Router such that as you work in one program, you see the changes reflected in the other. For example, if SailWind Layout is the “active” program, changes made there are reflected in SailWind Router but you are restricted in what you can do in SailWind Router since it is the “inactive” application. When you are 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.

Note:
You must enable Synchronization mode on page 774 and then restart SailWind Layout before Synchronization mode takes effect.

Despite the synchronization, you must actively switch back to SailWind Layout in order to work on your design in SailWind Layout.

Automation can still be used; however, most Automation methods will not return any valuable results and will not make any changes in the database.

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, SailWind Layout and SailWind Router become out of sync.

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

Unsupported commands

- Changing a drafting object: copper, copper planes, text, 2D lines, keepouts, labels, board outlines, dimensioning. However, 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
- Hatch and Flood
- Thermals
- ECO Options
- Via Patterns

Related Topics

Cut Outs Absorbed by Copper

Enabling Synchronization Mode

Synchronization Mode links SailWind Layout and SailWind Router so that you see the design in both programs and can work in either tool without moving the design back and forth.

Procedure

1. Click the **Tools > Options** menu item.
2. In the Options Dialog Box, click the Synchronization category on page 1535.
3. In the Layout and Router Synchronization area, click Enable.
4. Switching to SailWind Router automatically places your design in SailWind Layout in DRC Off mode; the DRC mode in SailWind Router is not affected.

   To restore the DRC mode when you switch back to SailWind Layout (to the setting that existed before switching to SailWind Router) click Restore DRC Mode on return. Depending on the size of your design, restoring the DRC mode may take a few minutes.
5. To receive a warning that your design is 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 SailWind Layout.

Related Topics

Sending a Design to SailWind Router in Synchronization Mode

Sending a Design to SailWind Router in Synchronization Mode

Use Synchronization Mode to quickly transfer your design to SailWind Router, but still keep your design open in SailWind Layout.

Procedure

1. Enable Synchronization Mode on page 774.
2. Switch to SailWind Router using one of the following:

   - On the Standard Toolbar, click the **Route** button.
   - Click the **Tools > Router** menu item, and in the SailWind Router Link dialog box select an Action and click **Proceed**.

   Note:
   To exit Synchronization mode, close one of the programs. Since in Synchronization Mode the Layout button (on the SailWind Router Standard Toolbar) does not send your design to SailWind Layout and then close SailWind Router, you must close the application manually. However, if one of the programs is out of sync, make sure you save your changes if you want to keep them. For example, if you want to continue working in SailWind Layout, but no longer want to be in Synchronization Mode, close SailWind Router. When prompted, save your design.

Results

SailWind Router opens and your design is loaded. SailWind Layout is switched to DRC Off mode and is now the inactive program. Most menu items and commands are disabled or are read-only for the inactive program. As you work in SailWind Router, you will see the changes reflected in SailWind Layout. You can quickly switch between the two programs and continue to be in Synchronization Mode. Look at the status bar to quickly determine which mode the program is in: Active, Inactive, or Out of sync.

If an unsupported command is performed or an unsupported option is changed, the inactive program will become out of sync. To re-synchronize the two programs, if SailWind Router is active, click the Layout button on the main toolbar in SailWind Router; or if SailWind Layout is active, click the Route button on the main toolbar in SailWind Layout.

To turn off the display of licensing warnings when switching between SailWind Layout and SailWind Router, type the following text in the *Blazerouter.ini* file:

```
[Security]
```

```
DisplayWarnings=0
```

(You do not need to restart SailWind Router.)

Related Topics

Enabling Synchronization Mode

Cut Outs Absorbed by Copper

Autorouting Your Design Using SailWind Router

Using the link you can either run SailWind Router and automatically open the current design file in the foreground, so you can view the autorouter's progress, or you can run SailWind Router in the background.

Procedure

1. Click the **Tools > Router** menu item.
2. In the SailWind Router Link Dialog Box, in the Action area, select the action you want to perform:

   Autoroute in Background — Opens SailWind Router and SailWind Router Monitor, but runs SailWind 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 SailWind Router and SailWind Router Monitor, and runs SailWind Router in the foreground making it the active program.
3. In the Routing Strategy area, click **Setup** to set the autorouting strategy on page 777.
4. Set any required settings before autorouting:

a. In the Options area, click one of the selections to specify which settings you want to configure and then click **Setup**.

b. Repeat for other selections as required.

5. Click **Proceed**.

Results

If you clicked Run in Background, the SailWind Router Monitor on page appears, but SailWind Router does not appear on your desktop.

If you clicked Run in Foreground, the SailWind Router Monitor on page appears. SailWind Router also opens and automatically autoroutes the current SailWind Layout design.

Related Topics

Cut Outs Absorbed by Copper

Setting up a Routing Strategy

Use the Routing Strategy dialog box to define a strategy for autorouting a design. A routing strategy is a detailed set of instructions that SailWind Router uses to complete the autorouting of your design. You can create very detailed sets of instructions that can precisely control the routing order of specific nets, net classes, layers and components.

The strategy defines the sequential operations to perform during autorouting, including:

- Passes that the autorouter should run, including the routing intensity, whether or not to protect the generated traces, and whether to pause after a pass completes.
- Order in which to autoroute nets, net classes, differential pairs, and matched length groups and components.

Procedure

1. Click the **Tools > SailWind Router** menu item.
2. In the SailWind Router Link Dialog Box, in the Routing Strategy area, click **Setup**.
3. In the Routing Strategy Dialog Box, in the Pass column, select the check box for 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.

Tip
To continue after a paused pass when autorouting in the foreground, click the Resume Autorouting button on the Routing Toolbar in SailWind Router (or press the F10 key).

To continue after a paused pass when autorouting in the background, click the Stop button in the SailWind 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. You cannot set an intensity for the Center pass.

7. Set the routing order for the passes:

    a. In the Routing Order column, click the field for the Pass Type you want to set.

    b. At the bottom left of the dialog box in the Available list, select items to add to the Routing Order. The list is filtered between Components and different types of Nets using the “Select by” list above it. For example:

        i. For the Fanout pass, click **Clear**, and then click the **Plane Nets** button to fanout only those nets.

        ii. For the Patterns pass, click **Clear**. Set the “Select by” list to Nets and in the Available list choose any known bus nets in your design and then click the **Selected** button.

        iii. For the Route pass, set the “Select by” list to Components and choose the most dense components in your design and then click the **Selected** button. In the Routing Order area, make sure the All Nets item is at the bottom of the routing order. If needed, use the down arrow to move it down in the order.

    c. Use the Routing Order area buttons to delete and re-order the items.

8. Click **OK**.

Results

When you autoroute the design using SailWind Router on page 776, the autorouter follows the routing order when routing your design.

Related Topics

Autorouting Pass Types [SailWind Router User's Guide]

Chapter 34
Filling Copper, and Copper Planes

This section describes the methods used to fill coppers, and copper planes. It describes their differences, how connectivity is established, and how vias interact with the flooding operations.

Differences Between Copper Shapes and Copper Planes

There are many differences between copper shapes and copper planes. This topic presents a feature comparison table that explains the differences for understanding when to use one instead of the other.

Table 119. Feature Comparison of Copper Shapes and Copper Planes

Feature	Copper Shape	Copper Plane
Can be filled without assigning a net	X	X
Is filled automatically	X	
Can be created in the PCB Decal Editor	X	
Can be created on non-electrical layers	X	X
Can have thermals on routed pads		X
Must encompass an electrical object of the net to which it's assigned when using the "Remove isolated copper" option		X

Table 119. Feature Comparison of Copper Shapes and Copper Planes (continued)

Feature	Copper Shape	Copper Plane
Avoids objects of any net to which it is not assigned		X
Has thermal spokes based on the Thermal Options		X
Can generate thermals and antipads from the design rules*		X**
Is not filled automatically, but must be poured		X
Can generate custom thermals and antipads from pad stack settings*		X
Can automatically create cutouts around embedded copper planes*		X
Can remove unused pads on inner layers*		X
Can be automatically generated based on the shape of the board outline		X
Can be split into two areas using the Auto Plane Separate tool		X
Can save hatch outlines with the design		X

*These features are controlled by settings.

If you use the “Use design rules for thermals and antipads” setting in the Options dialog box > **Copper Plane category > **Thermals** subcategory, antipads use the Copper-to-Drill clearance rule rather than the Copper-to-Pad clearance.

Flooding Copper Planes

Flooding recalculates the copper plane 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 copper plane that might create clearance violations.

Flooding fills the copper plane with copper in a hatched or solid pattern on page 617, creating isolation areas around copper, traces, and pads that are inside the copper plane 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 Options dialog box settings in the Copper Planes category > “Hatch and Flood subcategory” on page 1511 and Thermals subcategory on page 1514, and Grids on page 1537 category, as well as the flood and hatch options on page 1381.

Tip
If you have overlapping outlines on the same layer having the same flood priority, the software 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.

Procedure

1. Use one of the following methods:

   - Flooding a copper plane using select and flood — Use this method to select the copper plane and then apply flooding.

     i. Select the copper plane.

     ii. Right-click and click the **Flood** popup menu item.

   - Flooding a copper plane using the Flood button in verb mode — Use this method if you have more than one copper plane to flood. You first activate the Flood button in verb mode and then you can sequentially click copper planes and apply flooding.

     i. On the Drafting Toolbar, click the **Flood** button. 

     ii. Select a copper plane to flood.

     iii. Click and flood other copper planes as needed.

   - Flooding a copper plane using the Copper Plane Manager — Use this method to apply flooding to all copper planes in the design.

     i. Click the **Tools > Copper Plane Manager** menu item.

     ii. In the Copper Plane Manager Dialog Box, click Flood.

     iii. Select one or more layers having copper planes to flood.

     iv. Click **Start** to start the flood process.

   - Flooding all copper planes in the design by clicking the **Flood All** button on the main toolbar. 

Tip
To see the plane data in a negative mode, use the C modeless command.

2. Inspect the resulting shape:

   - If the copper plane did not fill, see "Troubleshooting Copper Plane Fills" on page 674.

   - If the copper plane filled, but opened a Thermal Relief Errors Report (*therm.err*), see "Troubleshooting Thermal Results" on page 675.

   - 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".

- If the fill is not correct, see "The Fill of Copper, and Copper Planes" on page 617.
- If the shape edges are not correct, see "Edge Precision of Drafting Shapes on page 615".
- If the shape needs to be modified, see "Drafting Object Properties".
- If you want to start over, see "Deleting a Drafting Segment or Object".

For more information, see "Split/Mixed Plane Layers" on page 683.

Note:
Online DRC does not prevent you from moving objects in a flooded area. If you do not re-flood the area, Verify Design reports errors. If objects within a flooded area were moved, or thermal settings were changed since the original flooding, you need to flood the area again.

Related Topics

Setting Flooding Order of Overlapping Copper Planes

Assigning a Net to a Copper Shape or Copper Plane

Hatching Copper Planes

Hatching Copper Planes

Hatching refills (with hatch lines) existing poured copper planes for the current session; it does not recalculate the copper plane area.

Each time you open a design file, you must hatch the design; the fill is not saved (you can set the file to hatch automatically each time you open it, however, using the "Autohatch on file load" option in the **Tools > Options** menu item, **Copper Planes** category > **Hatch and Flood** subcategory. Use Flood if you change clearance rules, or make changes to—or within—a copper plane that might create clearance violations.

Restrictions and Limitations

- You can only hatch a copper plane that has already been flooded. This converts the pour outline to a hatch outline.

Procedure

1. Use one of the following methods:

   - Hatching a copper plane using select and hatch — Use this method to select the copper plane and then apply hatching.

     i. Select the copper plane.

     ii. Right-click and click the **Hatch** popup menu item.

- Hatching a copper plane using the Hatch button in verb mode — Use this method if you have more than one copper plane to hatch. You first activate the Hatch button in verb mode and then you can sequentially click copper planes and apply hatching.

    i. On the Drafting Toolbar, click the **Hatch** button.

    ii. Select a copper plane to hatch.

    iii. Click and hatch other copper planes as needed.

- Hatching a copper plane using the Copper Plane Manager — Use this method to apply hatching to all copper planes in the design.

    i. Click the **Tools > Copper Plane Manager** menu item.

    ii. In the Copper Plane Manager Dialog Box, click Hatch.

    iii. Click **Start** to start the hatch process.

2. Inspect the resulting shape:

- 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".

- If the fill is not correct, see "The Fill of Copper, and Copper Planes" on page 617.

- If the shape edges are not correct, see "Edge Precision of Drafting Shapes on page 615".

- If the shape needs to be modified, see "Drafting Object Properties".

- If you want to start over, see "Deleting a Drafting Segment or Object".

Note:
Online DRC does not prevent you from moving objects in a flooded area. If you do not re-flood the area, Verify Design reports errors. If objects within a flooded area were moved, or thermal settings were changed since the original flooding, you need to flood the area again.

Related Topics

Flooding Copper Planes

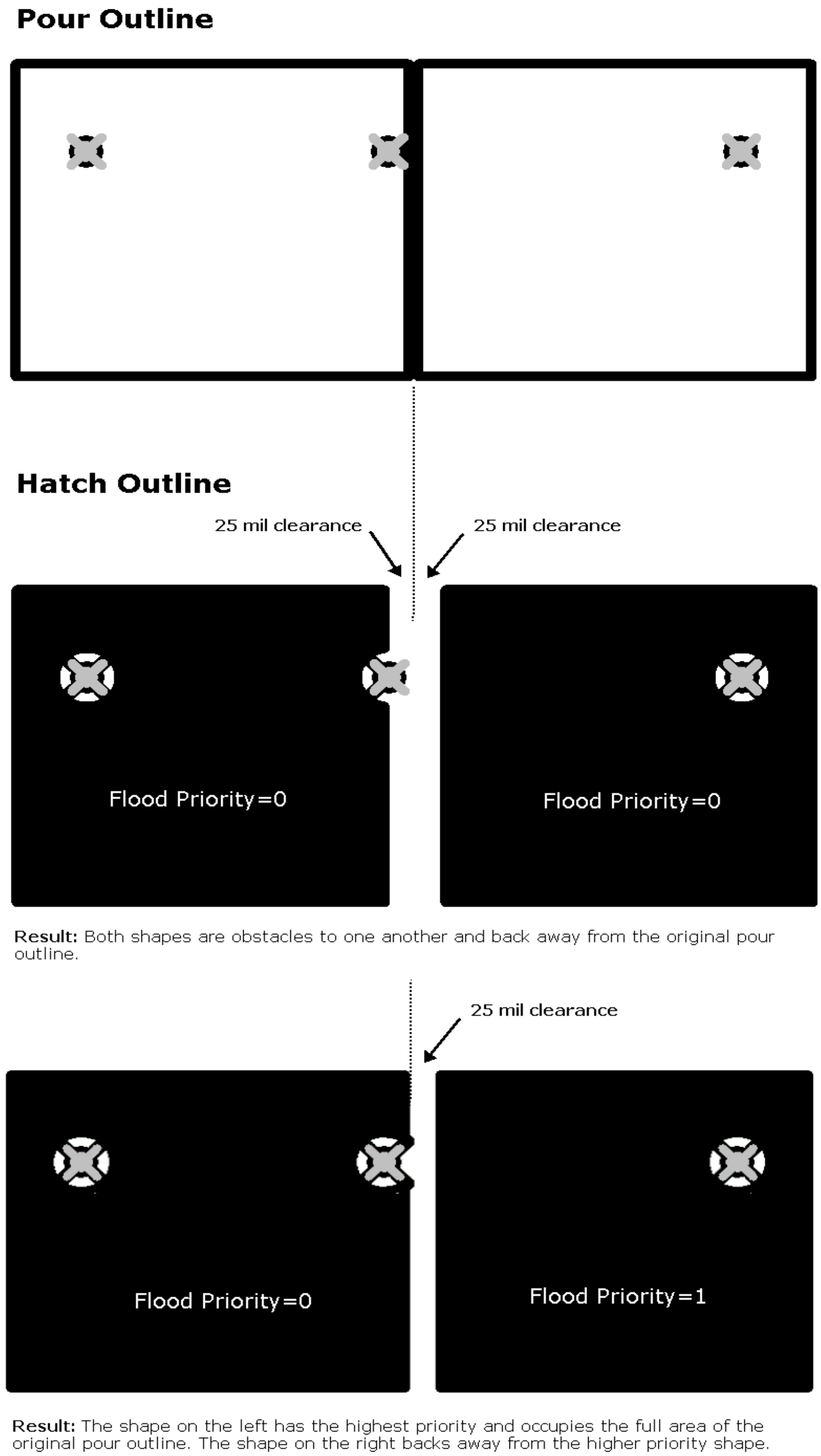
Copper Plane Flood Priorities

To determine which shape should be flooded first, you must set a flood priority to each shape. A shape with a lower priority number will be flooded before an object with a higher priority number. Copper planes on different layers are processed independently.

Scenario 1

You have adjacent shapes with a 25mil copper-to-copper clearance rule value.

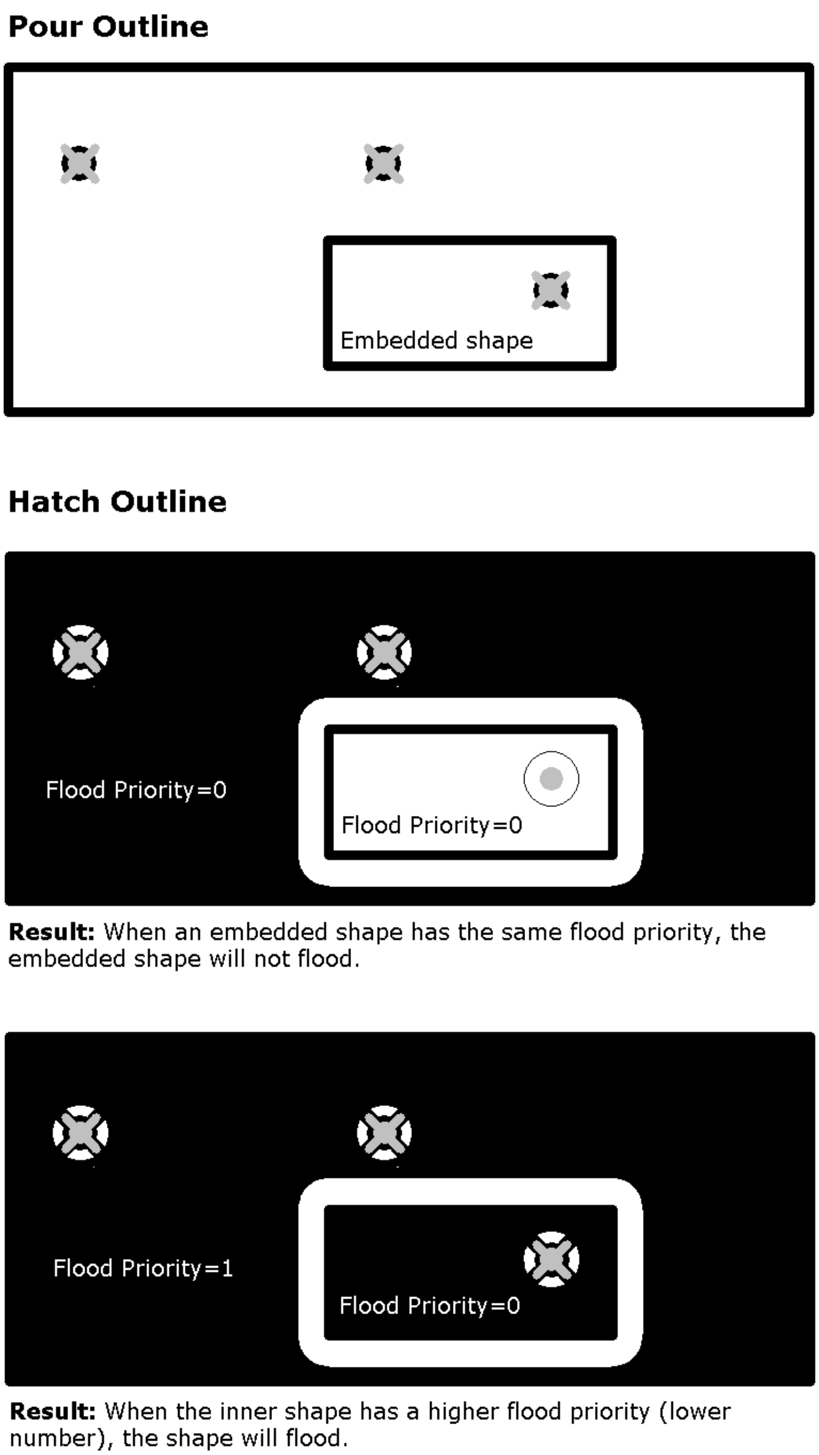
Figure 104. Scenario 1 with Same and Different Flood Priorities



Scenario 2

You have an embedded shape within a copper plane.

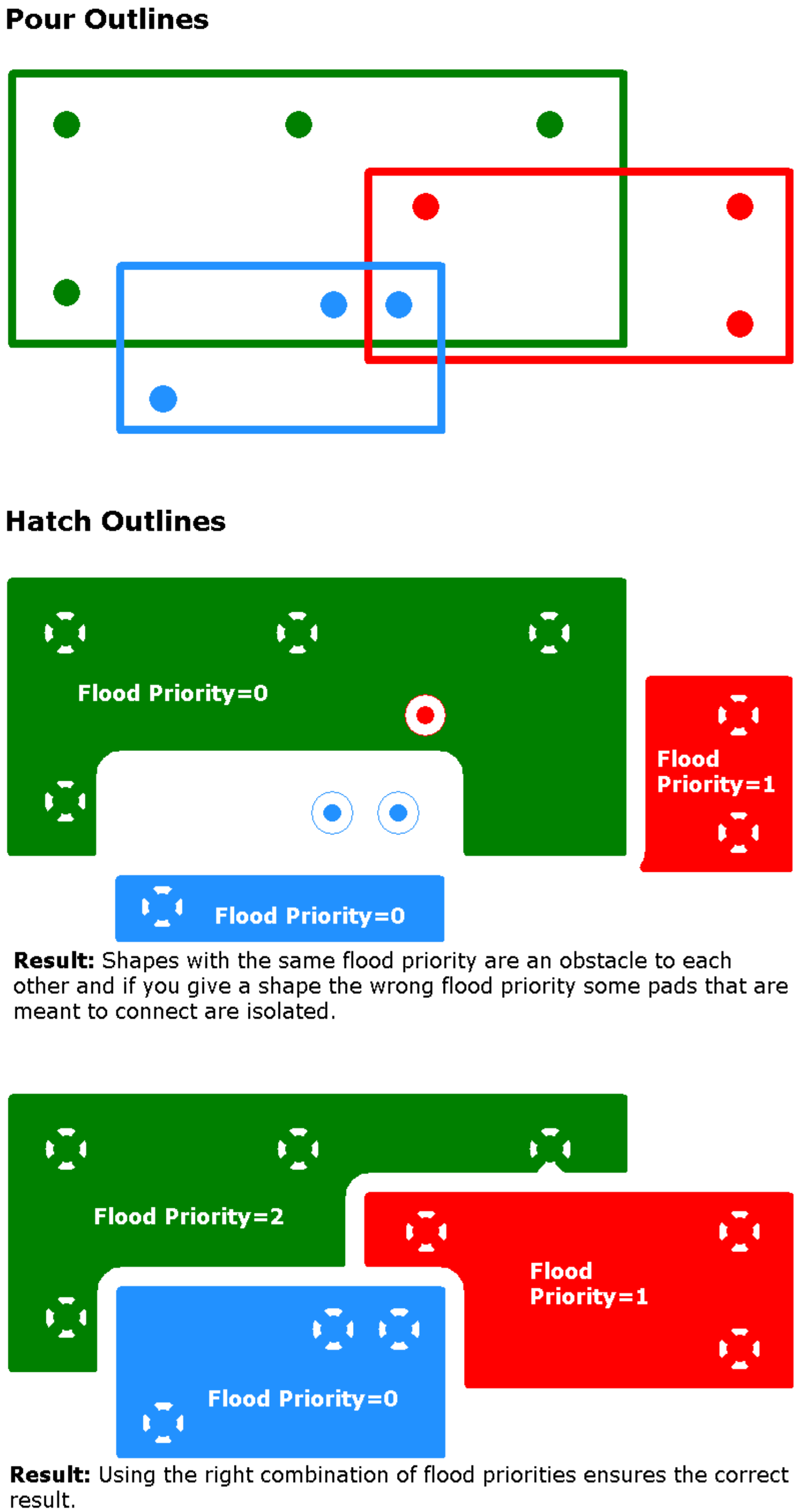
Figure 105. Scenario 2 with Same and Different Flood Priorities



Scenario 3

You have multiple overlapping shapes.

Figure 106. Scenario 3 with Wrong and Correct Flood Priorities



Related Topics

Setting Flooding Order of Overlapping Copper Planes

Flooding Copper Planes

Setting Flooding Order of Overlapping Copper Planes

The order in which copper planes are flooded will determine how they will appear in the design. You must set a flood order, (called the “priority”) for each object to define which copper plane should be flooded first. An object with a lower priority number will be flooded before an object with a higher one. Copper planes on different layers are processed independently.

Procedure

1. Right-click and click the **Select Shapes** popup menu item.
2. Select the copper plane.

   > **Restriction:**
   > The area you select cannot be a “plane hatch outline” on page 1849.

   > **Tip**
   > To quickly give the area the highest priority, bring it to the front (right-click and click the **Bring to front** popup menu item); to give it the lowest priority, send it to the back (right-click and click the **Send to back** popup menu item).

3. Right-click, then click the **Properties** popup menu item.
4. In the Drafting Properties Dialog Box, click the **Options** button. If the button does not appear, you have 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 cannot be flooded over), and 250 the lowest.

   > **Tip**
   > You can also see the 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 Plane Flood Priorities

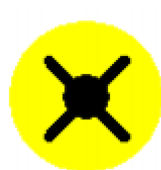
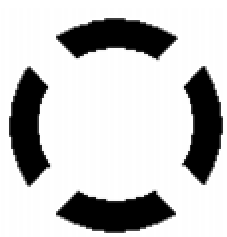
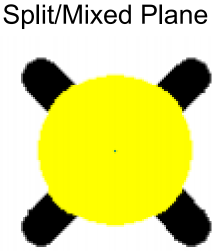
Keepouts

Plane Thermal Indicators

Plane thermal indicators are graphic images that show the thermal attribute status of pads.

There are two types of plane thermal indicators, general and specific. Sometimes during editing, thermal markers are covered; redraw the screen to view all of the markers.

Table 120. Plane Thermal Indicators

Indicator	Description
General Plane Thermal	General plane thermal indicators show that a connection to a CAM or split/mixed plane exists somewhere within a pad stack. This indicator shows as a small "x" in the center of the pad: 
Specific Plane Thermal	Specific plane thermal indicators show that a specific pad on a specific layer is connected to a plane. These indicators appear in the color assigned to the board outline in the Display Colors Setup dialog box. There is a different specific plane indicator for CAM planes and split/mixed planes: CAM Plane Indicator:  Split/Mixed Plane Indicator: 
Plane Antipad	Pads that do not belong to a CAM or split/mixed plane net and are present on one of these layer types appear with a circle to represent the antipad. These pads use the color specified for that layer in the Display Colors Setup dialog box.

Understanding How to Manipulate Copper Plane Thermals and Antipads

This section presents conceptual information that applies to the creation of copper planes and the use of custom thermals.

For procedures to create copper planes, see "Creating a Copper Plane Manually" on page 667 and for custom thermals, see "Creation of Thermals in the Pad Stacks" on page 186.

Common Copper Plane Thermal and Antipad Clearance Scenarios

This section discusses the four most common setting-combinations that affect a design's thermal and antipad clearances. Understanding these settings enables you to predictably create the thermal and antipad clearances that you desire.

Default Thermals and Antipads

There are a number of common settings for the default thermals and antipads. Once familiar with these settings, you can determine which ones you should use to accomplish your design goals.

Setup required:

1. Remove Unused Pads on page 1514 check box selected (enabled)
2. Use Design Rules for Thermals and Antipads on page 1514 check box cleared (disabled)
3. No custom thermals defined in Pad Stacks Properties on page 1566
4. No custom antipads defined in Pad Stacks Properties on page 1566

The Remove Unused Pads setting removes the pad from any component pin or via on an inner layer if that pin or via is not thermally tied to the copper plane or there are no routes coming off that pin or via on the plane layer.

Default thermals and antipads use the following basic formulas for calculating the clearance to the drill for antipads and the clearance to the pad for thermals:

- Default component through-hole antipad diameter = (2 x Copper-to-Pad clearance value) + pad size
- Default via antipad diameter = (2 x Copper-to-Via clearance value) + via pad size

Example: +5V via in a +12V copper plane with 35 mil pad, 20 mil drill, and 8 mil copper-to-via clearance value would result in a 51 mil diameter antipad, (2 x 8) + 35 = 51. Default copper to pad or copper to via clearance on thermals is simply controlled by the copper-to-pad or copper-to-via values.

Note:
The drill-to-copper clearance is used when the drill size is larger than the pad size, as defined in the pad stack setup.

The rules hierarchy on page 404 influences the default antipad and thermal clearance sizes. A Net rule with a higher copper-to-pad value than the default rule will create larger antipad sizes and greater clearance on thermals for the pins of that net. A net rule created for the plane net will not affect the antipad clearance size, but will affect the copper clearance on thermals. Net rules for the opposing nets will affect antipad size.

Conditional rules have some special behaviors. A Conditional rule in which a net is set against a layer will utilize the default antipad calculation. However, a Conditional rule in which one net is set against another net will not affect the antipad size unless the “Use Design Rules for Thermals and Antipads” setting is enabled - see “Design Rules for Thermals and Antipads”.

Design Rules for Thermals and Antipads

These are a number of common settings for Design Rules for thermals and antipads. Once familiar with these settings, you can determine which ones you should use to accomplish your design goals.

Setup required:

1. Remove Unused Pads on page 1514 check box selected (enabled)
2. Use Design Rules for Thermals and Antipads on page 1514 check box selected (enabled)
3. No custom thermals defined in Pad Stacks Properties on page 1566
4. No custom antipads defined in Pad Stacks Properties on page 1566

The thermals and antipads are now independently controlled by the Design Rules. This method enables you to create large antipad sizes without affecting the clearance on pads with thermals. The antipad size created affects both vias and component pads.

- Antipad Diameter for component pads and vias = (2 x Copper-to-drill) + Drill size

Thermal clearance is controlled by design rules copper-to-pad and copper-to-via settings. The rules hierarchy influences the default antipad and thermal clearance sizes. A net rule for the plane net affects the antipad size of any opposing net if you adjust the drill-to-copper value.

Example: A copper plane is defined for +12v, a net rule for +12v with an increased drill-to-copper value will cause the antipad size to increase for all non-connected pins or vias within the +12v copper plane.

Conditional rules have some special behaviors. A Conditional rule in which a net is set against a layer will utilize the default antipad calculation - see “Default Thermals and Antipads.” However, a Conditional rule in which one net is set against another net will not affect the antipad size unless the “Use Design Rules for Thermals and Antipads” setting is enabled.

Custom Thermals and Antipads

Custom thermals and antipads both have some common settings. Familiarize yourself with these settings to determine best which ones to use for accomplishing your design goals.

Setup required:

1. Remove Unused Pads on page 1514 check box selected (enabled)
2. Use Design Rules for Thermals and Antipads on page 1514 check box cleared (disabled)

3. You have custom thermals defined in Pad Stacks Properties on page 1566

4. You have custom antipads defined in Pad Stacks Properties on page 1566

Design rule settings do not affect the custom antipad size you define for the decal(s). The custom pad stack settings determine thermal or antipad clearances as well as any clearance checks.

Note:
You must select the Remove Unused Pads check box to enable custom antipad creation.

Hierarchy Between Custom Thermals and Antipads and the Use of Design Rules for Thermals and Antipads

There are a number of common settings for establishing the hierarchy between custom thermals and antipads and the use of Design Rules for thermals and antipads. Once familiar with these settings, you can determine which ones you should use to accomplish your design goals.

Setup required:

1. Remove Unused Pads on page 1514 check box selected (enabled)

2. Use Design Rules for Thermals and Antipads on page 1514 check box selected (enabled)

3. You have custom thermals defined in Pad Stacks Properties on page 1566

4. You have custom antipads defined in Pad Stacks Properties on page 1566

All custom antipad settings are ignored on pins and vias. The antipad size are determined by the copper-to-drill value in design rules.

- Antipad Diameter for component pads and vias = (2 x Copper-to-drill clearance value) + Drill size

A custom thermal's inner diameter is still used (the actual pad size), however the copper-to-pad clearance is used for component pins and the copper-to-via clearance is used for vias. The outer diameter value from a custom thermal is ignored in favor of the design rules.

Common Thermal Style Scenarios

This section discusses the three most common situations that affect thermal styles such as orthogonal, diagonal, flood over and no connect. Understanding these situations enables you to predictably create the thermal styles that you desire.

Thermal Style Changes in the Thermals Options

There are a number of common settings for changing thermal styles in the Thermals options. Once familiar with these settings, you can determine which ones you should use to accomplish your design goals.

Setup required:

1. You have made thermal style changes in the **Copper Planes** category > Thermals on page 1514 subcategory.
2. No custom thermals are defined in Pad Stacks Properties on page 1566
3. The “Flood over vias” check box is unchecked (disabled) in the Flood on page 1381.

The thermal style you choose affects all component pins and all vias for all layers and for all copper planes. You can set different thermal styles for each of the four pad shapes: round, square, oval, and rectangle. You can also differentiate between through-hole and surface mount components.

For example: It is possible to have a through-hole round pad with orthogonal spokes and a through-hole oval pad with a flood-over. You could also set up a through-hole round pad to be diagonal while an SMD round pad is set to no-connect.

Hierarchy between Custom Thermals and Thermal Style Changes in the Thermals Options

There are a number of common settings for establishing the hierarchy between custom thermals and thermal style changes in the Thermals options. After you become familiar with them, you can determine which settings you should use to accomplish your design goals.

Setup required:

1. You have made thermal style changes in the **Copper Planes** category > Thermals on page 1514 subcategory.
2. You have custom thermals defined in Pad Stacks Properties on page 1566
3. The Flood over vias check box is cleared (disabled) in the Flood and Hatch Options on page 1381.

Settings adhere to those components with custom thermals defined. All other components and/or vias will follow the thermal styles defined in the **Tools > Options** menu item > **Copper Planes** category > **Thermals** subcategory. You can create a flood-over of specific pins or vias by creating a custom thermal and reducing the outer diameter size to be equal to the inner diameter size. You may find this very helpful if, for instance, you only want the two mounting hole pins of your component to flood-over when connecting to GND.

Hierarchy Between Custom Thermals, Style Changes in the Thermals Options and Flood-over Vias in Copper Planes

There are a number of common settings for establishing the hierarchy between custom thermals, style changes in the Thermals options and flood-over vias in the copper planes. Once familiar with these settings, you can determine which ones you should use to accomplish your design goals.

Setup required:

1. You have made thermal style changes in the **Tools > Options** menu item, **Copper Planes**category > Copper Planes / Thermals on page 1514 subcategory.

2. No custom thermals are defined in Pad Stacks Properties on page 1566

3. The Flood over vias check box is selected (enabled) in the Flood and Hatch Options on page 1381

Settings adhere to those components with custom thermals defined. For those vias that are within the copper plane in which you specified for “Flood Over Vias,” they all flood over, no custom thermals adhere within that copper plane polygon. To set Flood Over Vias, select the copper plane in pour outline mode, right-click and click the **Properties** popup menu item, click on the **Flood & Hatch Options** button, and in the Flood & Hatch Options dialog box, select the “Flood over vias” check box. The thermal styles defined in the Options dialog box > **Copper Planes** category > **Thermals** subcategory are adhered to by all remaining components and vias.

Generating Thermals

Component pins and vias automatically receive thermals if they are associated with a plane net and if they have pads on the associated CAM plane layer. For component pins and vias that receive thermals, the Plane Thermal option is automatically turned on in the Pin Properties dialog box.

- When you unassign a plane net from a plane layer, the thermals are removed for all of the pins in the net. The Plane Thermal option is automatically cleared in the Pin Properties dialog box.

- If during routing you add a via to a plane net, the via automatically receives a plane thermal. The Plane Thermal option is automatically selected in the Via Properties dialog box.

For information on how thermals translate to and from SPECCTRA, see the SPECCTRA Translator Help.

Check the Flood Priorities by selecting the shape, opening the Properties and clicking on the Options button. The Flood Priority numbers should be lowest for inner nested planes and greatest for the outer planes.

Symptoms of Poor Thermal Results

Poor thermal connectivity can cause signal integrity problems in your design.

If the following symptoms occur, see “Troubleshooting Copper Plane Fills” on page 674:

- Copper plane will not flood
- Problems with split/mixed planes

Thermals on CAM Planes

When you generate thermals for CAM plane plots, SailWind Layout looks for plane netnames associated with CAM plane layers. SailWind Layout also checks that pins or vias with pads on the CAM plane layer have the Plane Thermal option selected in the Pin Properties and Via Properties dialog boxes. Use the “Show General Copper Plane Indicators” option in the **Tools > Options** menu item, **Copper Planes** category > **Thermals** subcategory to display CAM plane thermals.

When a pin exists in a net that is associated with a CAM plane layer and Plane Thermal is selected for the pin, a thermal appears on the pin. The ratsnest connections still appear. Use plane check in Verify Design to verify thermal generation for CAM plane plots.

CAM Plane Thermals

CAM plane thermals are displayed in the work area and the CAM preview area.

These calculations are also used for output during printing, pen plotting, and photo plotting operations:

- The outer diameter of the thermal matches the width of the aperture set in the Photo Plotter Setup Dialog Box.
- The inner diameter is 75% of the outer diameter.
- The number of spokes is always four, arranged diagonally.
- The spoke width is 1/6 of the outer diameter.

Tip
The inner width for custom CAM plane thermals is set as the pad size defined in the Pad Stacks Properties dialog box. The outer width is set as the default same-net pad to corner rule.

Viewing Plane Layer Connectivity

For CAM plane layers, the Plane Thermal option determines whether the thermal is generated for the pin. The Plane Thermal option signals CAM output to assign a D-code for a thermal relief aperture around the pins. Set the Plane Thermal option using the Properties dialog boxes for pins, vias, and jumper pins.

If you set up a plane connection successfully, a D-Code number is assigned for a thermal relief pad in your photoplotter aperture table, one that matches each pad size required. For more information, see the Results section of “Creating a CAM Plane Gerber-format File”.

Thermals on Copper Plane Areas

SailWind Layout follows certain procedures for thermals appearing in copper planes.

SailWind Layout looks for the same net name for objects to connect with a copper plane thermal:

- A component pin receives a thermal connection to a copper plane if there is no trace connected to it and the net name of the pin matches the net name of the copper plane.
- A component pin with a trace connected to it may receive a thermal connection if the "Add thermals to routed component pads" check box is selected in the **Tools > Options** menu item > **Copper Planes** category > **Thermals** subcategory.
- During copper plane flooding, vias always get copper plane thermals if the netname of the via matches the netname of the surrounding copper plane, whether they show unroutes or not.

Ratsnets connections still appear after the software pours the copper plane and installs the copper plane thermals. Click the **View > Nets** menu item to open the View Nets dialog box and assign the net to the View list. In the View Details area, select the "Traces Plus the Following Unroutes" check box and set the corresponding option to "All Except Connected Plane Nets. Choose the **Tools > Verify Design** menu item to check connectivity for copper planes.

Setting Pins and Vias as Thermals

Jumper pins, pins, and vias can all have thermals. For these objects to be eligible for a thermal, the Plane Thermal check box in the Jumper Pin Properties, Pin Properties, and Via Properties dialog boxes must be enabled.

If you select more than one pin or via where both plane nets and non plane nets are included, and not all pins and vias are marked as eligible for indicators, the Plane Thermal check box is neither selected or cleared but shows is a solid square showing a mixed state. If you click to select Plane Thermal, the selected plane net pins or vias are updated.

Design Rule Versus Pad Stack - Thermals and Antipads

There are two ways to apply thermal and antipad settings. One is design-based, and uses the design rule hierarchy to apply the settings. The second is component-based, and uses the pad stacks to customize the thermals and antipads of component pins and vias. You can use a combination of design rule and pad stack thermal and antipad settings.

Design Rule Thermals and Antipads

Design rule thermals are defined by the settings of the Thermals Options and the Copper-to-Pad or Copper-to-Via clearance rules. In the Thermals Options you define the spokes, and using the clearance rules, you define the gap between the pad and the copper.

Design rule antipads are defined by the Pad to Copper and Via to Copper clearance rules. You can use the hierarchy of the design rules to create different gaps between the drill and copper of antipads. For example, using the Net level of the Rule hierarchy, you could apply a larger gap to any pad connected to a particular net. Antipads can be defined by the Drill to Copper clearance rule when you select the "Use design rules for thermals and antipad" check box in the Options dialog box > **Copper Planes** category > **Thermals** subcategory.

You can only apply separate spoke settings to drilled and non-drilled thermals. The spoke settings globally apply to all pads of either type. If you need to customize the spoke settings of individual pads, you should use the pad stack thermal and antipad settings.

Pad Stack Thermals and Antipads

The pad stack thermals and antipads offer full control over all aspects of thermals and antipads. Unlike the design rule thermals and antipads, you can control the spokes of individual pads.

If you apply pad stack thermals and antipads to decals in the library, those settings may not apply to different PCB designs. You may want to only apply the pad stack settings to instances of the decal in the design rather than the decal in the library.

Priority

Although you can use a combination of design rule and pad stack thermal and antipad settings, the pad stack settings take priority over the more global design rule settings.

Flood Over Pads in a Copper Plane

You can flood over pads in a copper plane. There are two methods that you can use to flood over pads.

To flood over vias, see "Flood Over Vias in a Copper Plane."

Related Topics

Flood Over Vias in a Copper Plane

Flooding Over Pads Using the Thermals Options

You can use the Flood over option in the Thermals options to flood over all pads in a specific copper plane.

Restrictions and Limitations

These settings apply only when you are not using custom thermals.

Procedure

1. Click the **Tools > Options** menu item.
2. Click the **Copper Planes** category > Thermals on page 1514 subcategory.
3. For the pad shape, select the cell in the Drilled Thermals or SMT Thermals column to expand the list.
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.

Flooding Over Pads Using a Custom Thermal in the Pad Stack

You can create flood-over pads for component pads on any layers containing copper planes. Use this procedure if you do not want to flood over every object of a certain shape; you can flood over selected pad stacks.

Procedure

1. Click the **Setup > Pad Stacks** menu item.
2. In the Pad Stacks Properties Dialog Box, in the Pad Stack Type area, select Decal.
3. Select a decal in the Decal Name list.
4. Select a pin from the “Pin: Plated:” list
5. Select a layer from the “Sh: Sz: Layer:” list.
6. In the “Pad style” area, click the **Thermal** tab and click a pad shape from the buttons.
7. Select the “Flood Over” check box.
8. Repeat steps 5-7 for other layers of the decal pin that require thermal settings.
9. Click **OK** then click **Yes** to apply this update to the selected component or all components with this decal.

   Note:
   Clicking **Selected** appends the decal name with a letter to differentiate the padstack settings for the selected component from those applied to all other components with the same decal.

10. Repeat steps 1-9 for other decals for which you want flood-over pads.
11. 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 steps 1-8 but select “Via” as the pad stack type in step 2.

    Tip
    You can flood-over vias in copper planes using the Flood and Hatch Options Dialog Box. You can also flood over all drilled or SMT pads of a particular shape by setting options in the **Tools > Options** menu item > **Copper Planes** category > **Thermals** subcategory.

Flood Over Vias in a Copper Plane

You can flood over pads or vias in a copper plane. There are two methods that you can use to flood over vias.

- Flooding over vias by setting the drafting properties of the copper plane — Use this procedure to flood over vias in a copper plane area.
- Flooding over vias using a custom thermal in the pad stack — Use this procedure to flood over vias in a copper plane. Use this procedure if you do not want to flood over every object of a certain shape; you can flood over selected pad stacks.

To flood over pads, see "Flood Over Pads in a Copper Plane".

Related Topics

Flood Over Pads in a Copper Plane

Flooding Over Vias By Setting the Drafting Properties of the Area

You can use the Drafting Properties of the area to flood over vias in a copper plane.

Procedure

1. Select the copper plane outline.
2. Right-click and click the **Properties** popup menu item.
3. Click the **Flood & Hatch Options** button.
4. In the Flood and Hatch Options Dialog Box, select the Flood over vias check box.

   If the **Flood & Hatch Options** button is unavailable, close the dialog box, use the modeless command "PO" to switch to the pour outline mode, and repeat steps 1-4.
5. Click **OK** and then click **Yes** to proceed with the flood.

   Tip
   Click **No** to flood the area later on.
6. Close the Drafting Properties dialog box.

Results

The selected area floods over with copper. Any vias connected to the net also flood over, and there are no longer any thermal spokes on the vias.

Flooding Over Vias Using a Custom Thermal in the Pad Stack

You can use the Thermal Pad style option in the Pad Stacks Properties dialog box to flood over vias in a copper plane. Use this procedure if you do not want to flood over every via of a certain shape; you can flood over selected via types.

Procedure

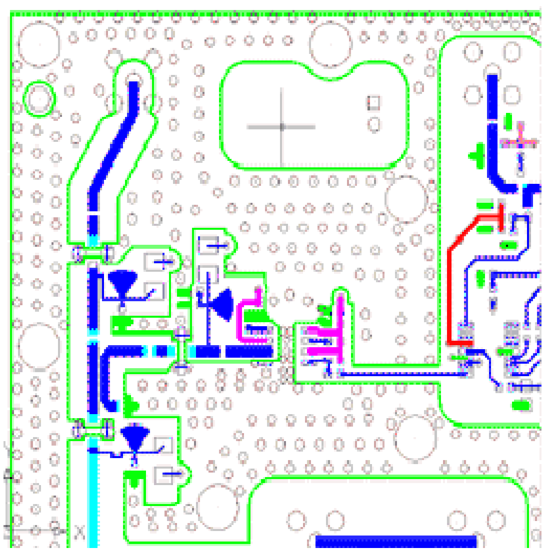
1. Click the **Setup > Pad Stacks** menu item.
2. In the Pad Stacks Properties Dialog Box, in the Pad Stack Type area, select Via.
3. Select the via name in the **Decal name** list.
4. Choose the layer you want from the **Sh: Sz: Layer** list.
5. In the Parameters area, select **Thermal** from the **Pad 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.

   The selected pads stacks are flooded over in the copper plane.

Filling a Shape with a Pattern of Vias

To fill a copper plane with a pattern of stitching vias, use the Via Stitch operation in Fill mode.

Figure 107. Shape Stitched with Vias



Procedure

1. If you have not done so, set the options for stitching shapes in the **Tools > Options** menu item > **Via Patterns** category. For information, see "Setting Options for Via Patterns on page 1556".

   Set the Pattern setting to Fill on the **Via Patterns** tab or set it after you select the object. (See Step 3.)

   Tip
   Your Via Pattern settings are stored 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 plane
   - A hatch outline for a copper plane

   Restriction:
   The copper or copper hatch outline must be associated with a net.

3. Right-click and click the **Via Stitch** popup menu item.

   Tip
   To override the Pattern (Fill or Perimeter) setting in the Via Patterns Options, right-click and click the Via Stitch Mode popup menu item, then select the pattern.

Results

The selected copper or hatch outline is filled with the pattern of vias.

Note:
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

DRC and the Via Stitching and Shielding Operations

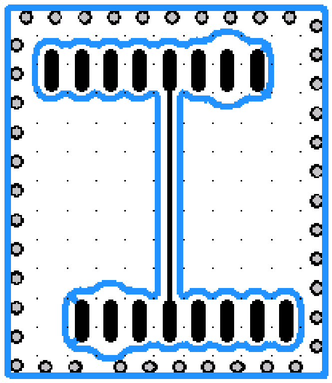
Vias

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 plane or a hatch outline by using the Via Stitch command in Perimeter mode.

Figure 108. Vias Inside Perimeter



Procedure

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 on page 1556".

   Set the Pattern setting to Perimeter on the **Tools > Options** menu item > **Via Patterns** category or set it after you select the object. (See Step 3.)

   Tip
   Your Via Pattern settings are saved 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 plane
   - A hatch outline for a copper plane

3. Right-click and click the **Via Stitch** popup menu item.

Tip
To override the Pattern (Fill or Perimeter) setting in the Via Patterns Options, select Via Stitch Mode, then select the pattern.

Note:
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

DRC and the Via Stitching and Shielding Operations

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. You can change the default behavior for the Via Stitch operation.

CAUTION:
Exit SailWind Layout before you edit the *SailWind.ini* file.

Procedure

1. In a text editor, open the *SailWindpcb.ini* file. The file's location is:

   C:\<install_folder>\<version>\Settings

2. Change the VS_StitchVoids value to 1. For example:

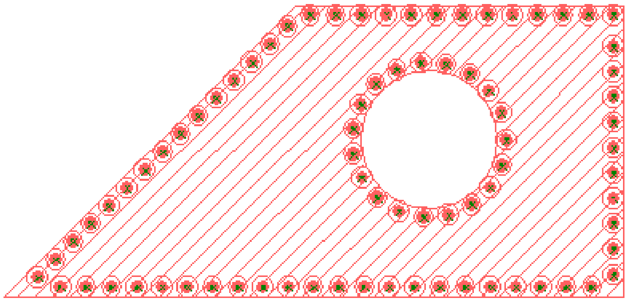
   ```
   [ViaShield]
   VS_StitchVoids=1
   ```

3. Restart SailWind Layout.

Results

The Via Stitch operation not only places vias inside the perimeter of a shape but also surrounds voids within the shape.

Figure 109. Vias Surrounding a Void



Chapter 35
Reference Designators

Reference designators are used to identify individual design objects. This section describes the use of reference designators in the design.

Standard Reference Designators

In order to promote continuity across the industry, standardized reference designators have been created for each of the major component and design object types. This topic presents a table containing the standard reference designator definitions from the IPC-2612 standard for schematic symbol generation.

Table 121. Standard Reference Designators

Prefix	Description	Prefix	Description
A	separate assembly	LS	loudspeaker, buzzer
AR	amplifier	M	meter
AT	attenuator, isolator	MG	motor-generator
B	blower, motor	MH*	mounting hole
BT	battery	MK	microphone
C	capacitor	MP	mechanical part
CB	circuit breaker	P	connector, plug, male
CP	connector adapter, coupling	PS	power supply
CN	capacitor network	Q	transistor
D or CR	diode	R	resistor
D or VR	breakdown diode	RN	resistor network
DC	directional coupler	RT	thermistor
DL	delay line	S	switch
DS	display, lamp	T	transformer

Table 121. Standard Reference Designators (continued)

Prefix	Description	Prefix	Description
E	terminal	TB	terminal board, terminal strip
F	fuse	TC	thermocouple
FD	fiducial	TP**	test point, in-circuit test points
FL	filter	TZ	transzorb
G	generator, oscillator	U	inseparable assembly, IC package
GN	general network	V	electron tube
H	hardware	VR	voltage regulator
HY	circular, directional coupler	W	wire, cable, cable assembly
J	connector, jack, female	X	fuse holder, lamp holder, socket
K	contactor, relay	Y	crystal, magnetostriction oscillator
L	coil, inductor, bead, ferrite bead	Z	miscellaneous

*These class letters would not appear in a parts list as they are part of a PCB and not an active electronic component.

**Not a class letter, but commonly used to designate test points for maintenance purposes.

Note:
Note: The above list is not exhaustive. See the standard list of class designation letters in ANSI Y32.2/IEEE Std 315, Section 22 and the Index.

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 either the top, or bottom layer.

   For selection instructions, see "Object Selection." You might need to hide the opposite mounted layer or set your selection filter to select only objects on the one layer.

   Tip
   Use the Z modeless command on page 83 to hide and reveal specified layers.

   ---

2. Right-click and click the **Add New Label** popup menu item.

3. In the Add New Part Label Dialog Box, ensure the Attribute list is set to "Ref.Des".

4. In the Layer list, select either the Assembly Drawing Top or Assembly Drawing Bottom layer.

   The layer to which you are moving the labels needs to be set as visible and a non-background color applied to reference designators on that layer in order for you to see the new labels on the layer. For more information, see "Setting Global Colors of Design Objects on page 131."

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 at the center of the component.

   See "Moving a Reference Designator."

Related Topics

Disabling Reference Designators of Selected Components

Moving Reference Designators to the Silkscreen Layer

Moving a Reference Designator

As you reposition objects in your design, reference designators may have to be moved so that they remain legible and avoid conflict with other design objects. You can move reference designators to optimize their location on the PCB.

Procedure

1. Use one of the following methods to begin moving the reference designator:

   - Object mode — Select a reference designator then right-click and click the **Move** popup menu item.

   - Verb mode — On the Design Toolbar, click the **Move Reference Designator** button, then select the reference designator that you want to move.

2. With the reference designator attached to the pointer, click to indicate a new location for the reference designator.

Moving Reference Designators to the Silkscreen Layer

If your components have been designed with reference designators included on layers other than the silkscreen layer(s) such as Layer Top or Layer Bottom, you can move them to the silkscreen layer(s) in the design.

Procedure

1. Select all components on either the top, or bottom layer.

   For selection instructions, see "Object Selection." You might need to hide the opposite mounted layer or set your selection filter to select only objects on the one layer.

   Tip
   Use the Z modeless command on page 83 to hide and reveal specified layers.

2. Right-click and click the **Properties** popup menu item.

3. In the Component Properties Dialog Box, click the **Labels** tab.

4. In the Label list, select "Ref.Des".

5. Click the button underneath the Label list.

6. In the Part Label Properties Dialog Box, in the Layer list, select the correct silkscreen layer for the components you have selected at the beginning of this procedure.

7. Click **OK** to move the reference designator labels to the specified layer.

8. Repeat this procedure for labels on the opposite layer.

Related Topics

Generating a Second Set of Reference Designators for Assembly Drawings

Disabling Reference Designators of Selected Components

Object Selection

Disabling Reference Designators of Selected Components

You can disable the display of the reference designator label for a component. If you turn off the reference designator label, it is also not visible in the CAM outputs for manufacturing the PCB.

This feature could be used to disable the reference designators of mounting holes placed in the design from parts stored in the library. As parts, they have reference designators, but they do not need reference designators to show up in the silkscreen.

Tip
You can hide the display of all reference designators by layer using the Display Colors Setup Dialog Box.

Procedure

1. Select a reference designator.
2. Right-click and click the **Properties** popup menu item.
3. In the Part Label Properties Dialog Box, in the Show list, click None.
4. Click **OK** and the reference designators label is turned off.

Related Topics

Generating a Second Set of Reference Designators for Assembly Drawings

Moving Reference Designators to the Silkscreen Layer

Object Selection

Chapter 36
ECO (Engineering Change Order)

As you develop new iterations of your design, you may need to create an ECO (Engineering Change Order) to capture the changes and forward or back annotate them between the schematic and the printed circuit board designs. This section discusses the purpose and use of Engineering Change Order operations.

Engineering Change Order Operations

ECO operations include any processes that modifies the connection list or parts list. These operations include deleting, adding, and changing various aspects of decals, parts, nets, pin pairs, pad stacks, attributes, or design rules.

You must be in ECO mode before you can make such edits. In ECO mode, SailWind Layout can record your changes in a text file called the ECO file. You can use the ECO file as a reference to update, or back-annotate, the schematic.

ECO Toolbar

The ECO Toolbar provides quick access to commands that allow you to create and update Engineering Change Orders associated with your design.

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.

  You can also create an ECO file by using the Compare/ECO dialog box (**Tools** menu > **Compare/ECO** menu item).

- 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 do not need to save changes in an ECO file.

Recorded Versus Generated ECO Files

It is always a best practice to record the engineering changes you make to your design. When you record the changes, an exact list is kept of the before and after change to objects.

For example, the *.eco* file will list that capacitor C42 was renamed to C71. If instead, you generate a list of engineering changes by simply comparing two designs, there is no way for the software to determine the changes that were made to parts that are electrically identical. If there are four identical capacitors in the design with identical connections, it might report that C28 changed to C71 when that isn't exactly what happened. Electrically, there's nothing wrong with these arbitrary decisions since the capacitors have the same value and are attached to the same nets. But if you compared how those capacitors are placed in the layout relative to how they're placed in the schematic, it might not make sense and it could lead you to be concerned that something terrible has occurred with the synchronization of the design.

So, depending on how you create the *.eco* file, a renumbered back annotation can have different effects, one of which can be inconvenient to cross probing or double-checking the results of the renumbering - comparing the schematics to the layout. Using a recorded *.eco* file is the best option and can be used in all three methods of back annotation including manual and automated methods. If you record the renumbering of the reference designators in the *.eco* file while you use the autorenumber tool, you will have a perfect (was...is) before and after listing of the changed reference designators.

But if you generate the *.eco* by comparing netlists, you will not have an exact (was...is) before and after list of the discretes in the schematic. Discretes that are identical components and have identical net connections are indistinguishable in the netlist. Electrically the changes will be correct. But since there is no placement information in netlists, an *.eco* file generated by comparison won't be able to determine the exact refdes exchange of these discretes.

Recording ECO Changes

If you are working on a schematic-driven design, you will need to back-annotate design changes to the schematic. You can record your netlist changes and then save them to an ECO file.

Procedure

1. On the Standard Toolbar click the **ECO Toolbar** button.

   If the ECO Options dialog box does not appear, click the **ECO Options** button on the ECO Toolbar to open it.

2. In the ECO Options dialog box:

   a. Select the “Write ECO file” check box.

   b. Make sure that the pathname of the *.eco* file in the “Filename” text box is correct. (If the file does not exist, it is created and recorded changes are written to it. Existing *.eco* files are left untouched.)

   c. 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 the Append to file check box.

      - To overwrite the existing file with the changes you make in this session, clear the Append to file check box.

d. 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 the Write ECO file after closing ECO toolbox check box.
- To save ECO changes only when you save the *.pcb* file, open a new file, or exit SailWind Layout, clear the Write ECO file after closing ECO toolbox check box.

e. Set the other options as appropriate. See the "ECO Options Dialog Box" topic.

f. Click **OK**.

Results

Any netlist changes you make are recorded. If you have selected the Write ECO file after closing ECO toolbox check box, the changes are written to the ECO file when you close the ECO toolbox; otherwise they are 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 check box during a session, all recorded change data, both in memory and in the ECO file, are deleted.

Related Topics

Recorded Versus Generated ECO Files

Backward Annotation from SailWind Layout to SailWind Logic

ECO-Registered Parts

A part is ECO-registered when you edit the part with the Library Manager, select ECO Registered Part on the General tab of the Part Information dialog box, and then save the part to the library. Once a part has been assigned the ECO Registered status, it can be identified for inclusion or exclusion during the ECO update processes.

When updating a design with changes from a schematic or updating a schematic with changes from a design, you can exclude or include non-ECO-registered parts from ECO processing. In the ECO Options Dialog Box, do one of the following:

- If you want ECO processing to exclude non-ECO-registered parts, select the "Output Only ECO Registered Parts" check box.
- If you want ECO processing to include non-ECO-registered parts, which includes non-electrical parts, clear the "Output Only ECO Registered Parts" check box.

You must be in ECO mode to add, delete, rename, or alter a part regardless of whether it is registered or not.

Avoid connecting non-ECO-registered parts, such as mechanical hardware, to ECO-registered netlist items, such as GND. A conflict may occur when an ECO-registered item is connected to a non-ECO-registered item.

ECO-Registered Attributes

Both ECO-registered and non-ECO-registered attributes can be added, deleted, or changed in ECO mode.

To turn on ECO Registration for attributes, use the Objects tab of the Attribute Properties dialog box on page 1111. Also turn on ECO Registration for any attributes you want to backward annotate to the schematic. Via attributes are not registered attributes, therefore you can add, delete, or change them in ECO mode or non-ECO mode.

You can exclude non-ECO-registered attributes from ECO processing by clearing the “Compare Only ECO Registered Attributes” check box in the **Comparison** tab of the Compare/ECO Tools dialog box on page 1181.

When updating a design from a schematic, a report automatically appears indicating the ECO registration of imported attributes. When an attribute does not exist in the Attribute Dictionary, it is added with ECO registration turned off. If the attribute already exists in the dictionary, the existing attribute and ECO registration in the dictionary are used.

ECO Mode Operations (Layout-Driven Design Tools)

This section describes specific operations that you can perform only in ECO mode.

CAUTION:
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.

Predefined Netnames

When you add a net to the design using the Add Route, Add Connection, or Copy Route command, the Derive Net Name from Pin Function shortcut menu option controls how the added net is named. When this option is enabled, the pin's pin name determines the net name. If no pin name exists, or this option is disabled, the net name is automatically generated.

Use the Derive Net Name from Pin Function option when creating the design on the fly instead of creating the design from the netlist generated by the schematic tool. For example, in a typical BGA design you manually connect each die part's substrate bond pad to a BGA component pad using the Add Route or

Add Connection command. To give the added net a meaningful name, enable Derive Net Name from Pin Function to derive the net name from the die part pin name, for example, GND.

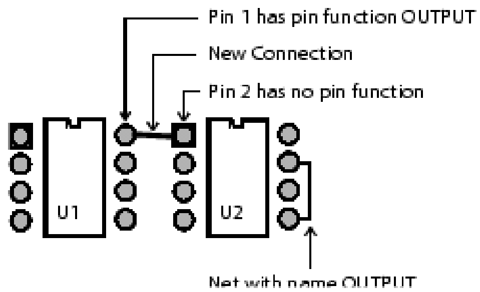
Added Connection

When adding connections, the command behavior is dependent upon the condition of the pins that are being connected such as pins not in a net, one pin in a net, one pin defined as an output or merging nets.

In example 1 shown in Figure 110, Pin 1 and Pin 2 are not in a net. A pin name (function) exists for one of the pins.

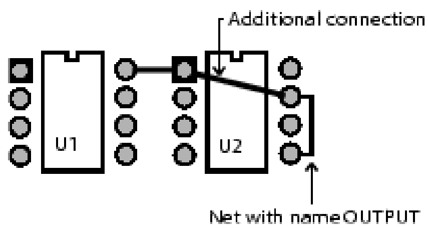
Connecting Pin 1 and Pin 2 opens the Define Name of Net dialog box.

Figure 110. Added Connection When Pins are Not in a Net



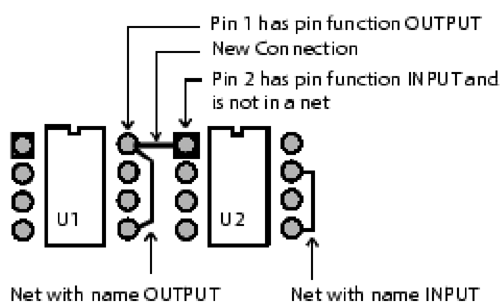
Clicking Add Pins to OUTPUT in the Define Name of Net dialog box adds Pin 1 and Pin 2 to netname OUTPUT (Figure 111).

Figure 111. Added Pin 1 and Pin 2 to Netname OUTPUT



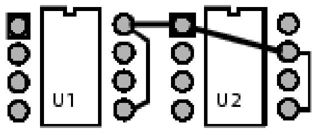
In example 2 shown in Figure 112, one of the pins is in a net. One of the pins has a pin function defined.

Figure 112. Added Connection When One of the Pins is in a Net



When you connect Pin 1 and Pin 2, the Define Name of Net dialog box appears. Choose the name you want to assign to the merged nets. The nets are then merged as shown in Figure 113.

Figure 113. Merging Nets



Adding a Connection in ECO Mode

When you create a new pin pair in your design and it will change the current netlist. ECO Mode captures this change and preserves it for later back annotation to the schematic. While adding the new connection, you have a number of naming options available.

CAUTION:
When you modify or delete design objects, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Procedure

1. Click the **ECO Toolbar > Add Connection** button0

   If this is the first time in the current session you are using 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.

   While adding a connection:

   - You can right-click and click the **Rename Current Net** popup menu item to rename the net of the current pin. In the Rename Net prompt, type the new name and click **OK**. The maximum netname length is 47 characters. For more information, see "Illegal Characters in Netnames and Part Names".

   - You can rename the net of the current pin, as follows:

     - To use the pin function as the net name, right-click and click the **Derive Net Name from Pin Function** popup menu item. For more information, see "Predefined Netnames".

     - To type in a new name, right-click and click the **Rename Current Net** popup menu item. In the Rename Net prompt type the new name, and click **OK**. The maximum netname length is 47 characters. For more information, see "Illegal Characters in Netnames and Part Names."

     - To use the pin function as the net name, right-click and click the **Derive Net Name from Pin Function** popup menu item. For more information, see "Predefined Netnames."

- To type in a new name, right-click and click the **Rename Current Net** popup menu item. In the Rename Net prompt, type the new name, and click **OK**. The maximum netname length is 47 characters. For more information, see "Illegal Characters in Netnames and Part Names."

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 or type a new netname. (Netnames are automatically generated for each newly created connection.) For more information, see "Predefined Netnames" on page 817.

4. Click another pin to continue adding connections to this net, or right-click and click the **Cancel** popup menu item to stop.

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, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Procedure

1. Click the **ECO Toolbar > Add Route** button.

   If this is the first time in the current session you are using 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.

3. Begin routing.

   Add corners and vias as described in "Routing Manually". You can also use the shortcut menu to control routing operations. While routing:

   - You can rename the net of the current pin. Right-click and click the **Rename Current Net** popup menu item. In the Rename Net prompt type the new name, and click **OK**. The maximum netname length is 47 characters. For more information, see "Illegal Characters in Netnames and Part Names".

     As an alternative you can use the pin function as the net name. Right-click and click the **Derive Net Name from Pin Function** popup menu item. For more information, see "Predefined Netnames".

   - Right-click and click the **Cancel** popup menu item to exit Add Route mode.

- With Online DRC disabled, the pointer changes to a bull's-eye when it is near any eligible completion point, regardless of the net.

- With On-line DRC in Prevent errors mode on page 1503, when 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 the **Select Target** popup menu item and select the pin at which you want to end the trace.

- If you are connecting to a pin of a different net (that is, merging two nets), the Define Name of Merged Net Dialog Box appears. Choose or type a new netname. (Netnames are automatically generated for each new connection.) For more information, see "Predefined Netnames."

4. Select the pin on which to end the trace.

Related Topics

Illegal Characters in Netnames and Part Names

Adding a Component in ECO Mode

You can add a new component to your design in ECO Mode and it will change the current netlist. ECO Mode captures this change and preserves it for later back annotation to the schematic.

CAUTION:
When you modify or delete design objects, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Restrictions and Limitations

- 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.

- A part must be ECO-registered on page 1824 to be included in the ECO file. You set the ECO registration for parts in the Part Information dialog box on page 1590.

Procedure

1. Click the **ECO Toolbar > Add Component** button.

   If this is the first time in the current session you are using 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 on page 155 in the Items box and click **Apply**.

   If the Filter Items box is blank, no items will be displayed. Add an asterisk (*) to see all items of the library or library partition in the Part Types list.

3. Select the part name in the Part Types list box and click **Add** to attach the component 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.

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 the **Rename Current Part** popup menu item. In the Rename Part prompt, type the new reference designator, and click **OK**.

6. Move the part to the appropriate location and click to place the part.

Related Topics

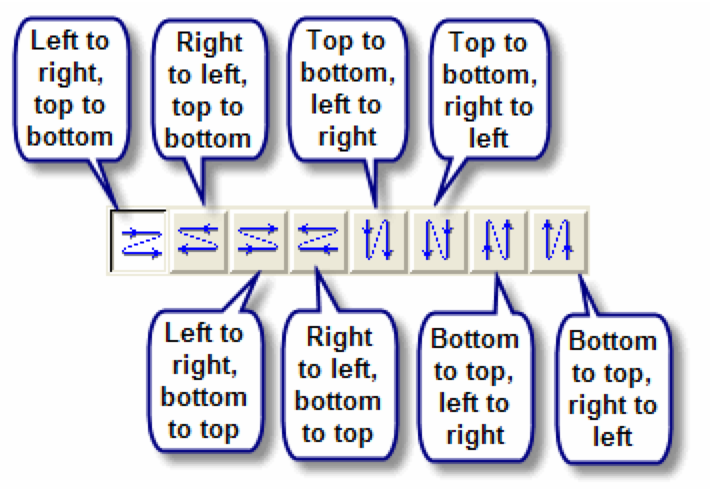
Updating a Design from the Library

AutoRenumber Sweeps

The AutoRenumber result depends on the matrix created by the cell size and the directional pattern of the renumbering sweep. The values used in the Cell Size area of the AutoRenumber dialog box allow you to create custom renumbering sweeps of the board. Here we provide two usage examples and then recommend a best practice.

You can select one of eight different directional numbering patterns. Choose the pattern that best suits the orientation and shape of your board.

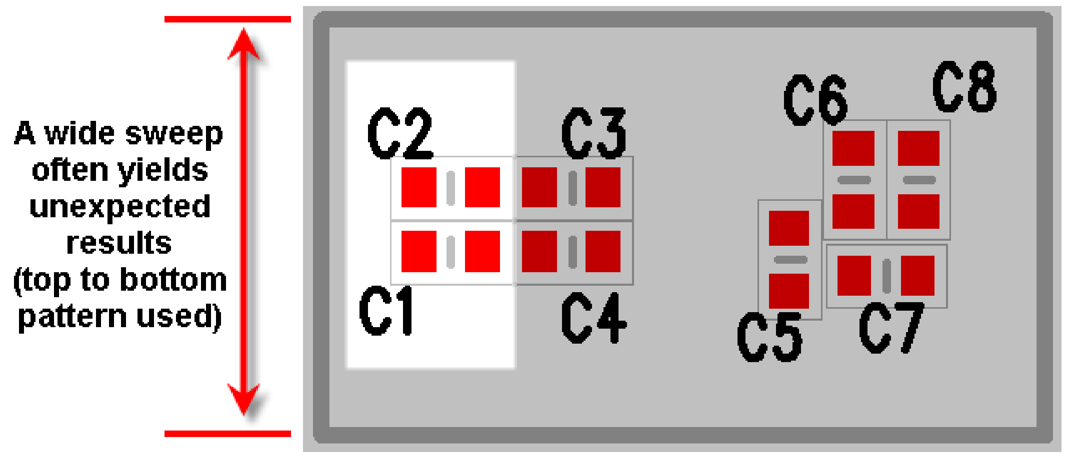
Figure 114. AutoRenumber Patterns



Wide Sweep Example

In this example, you accept the default Cell Size settings of the AutoRenumber dialog box. The On x and On y values are set to encompass your entire board. This produces one long, wide sweep. The result of using a wide sweep frequently results in unexpected numbering of the reference designators.

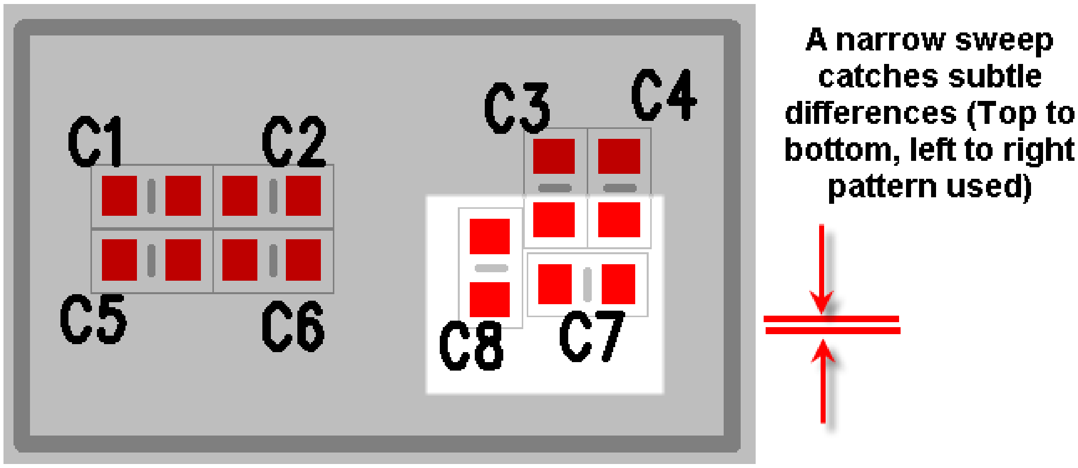
Figure 115. Wide Sweep Results



Narrow Sweep Example

In this example, you type a small value for the On y value of the Cell Size. The On y value is something small like 10 mils and it catches even the smallest variations in the alignment of components. The result of using a sweep that is too narrow results in numbering of components that don't appear to be offset.

Figure 116. Narrow Sweep Results



Best Practice

Based on the examples shown, the best practice to follow with renumbering is to create cells that do span your entire board, but in narrow sweeps. You may need to adjust the width of your sweep to achieve the desired result. When renumbering, the sweep uses the bottom pin of components as the origin.

For example, your board measures 4000 mils wide and 2000 mils high. You are renumbering left to right, top to bottom. You want to set your cell size to On x=4000 and On y=10.

Tip
If the reference designator renumbering isn't exactly perfect, you can always manually renumber a few components that weren't renumbered according to what you expected.

Related Topics

Changing the Reference Designators of Multiple Components in ECO Mode (Autorenumbering)

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 "Recorded Versus Generated ECO Files".

CAUTION:
When you modify or delete design objects, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Restrictions and Limitations

- 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.
- Jumpers are excluded from autorenumbering.

Procedure

1. Click the **ECO Toolbar > Auto Renumber** button.

   (If this is the first time in the current session you are using 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.

   For information about selecting cell sizes and renumbering patterns, see “AutoRenumber Sweeps”.

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 to 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**.

Results

Are the results not what you expected? Click **Edit > Undo** to undo the renumbering and start over. (You many want to revisit the “AutoRenumber Sweeps”.)

Related Topics

Recorded Versus Generated ECO Files

Recording ECO Changes

Backward Annotation from SailWind Layout to SailWind Logic

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 on page 827.

CAUTION:
When you modify or delete design objects, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Restrictions and Limitations

- 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. Click the **ECO Toolbar > Change Component** button.

   (If this is the first time in the current session you are using the ECO Toolbar, the ECO Options Dialog Box appears. Set the options you want and click OK to use the ECO tools.)

2. Type DRO and press the Enter key to turn off On-line DRC (Design Rules Checking).

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 the **Find** popup menu item to use the Find dialog box. For more information, see the “Find Objects dialog box on page 1378” topic.

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 the **Library Browse** popup menu item. The Get Part Type from Library Dialog Box appears. To filter the part types, in the filter area, type a wildcard or expression on page 155 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. on page 1104

Results

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.

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 on page 826 to another part type in the design or in the library.

CAUTION:
When you modify or delete design objects, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Restrictions and Limitations

- 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. Click the **ECO Toolbar > Change Component** button.

   (If this is the first time in the current session you are using the ECO Toolbar, the ECO Options Dialog Box appears. Set the options you want and click OK to use the ECO tools.)

2. Type DRO and press the Enter key to turn off On-line DRC (Design Rules Checking).

3. Select the component or components whose part type you want to update.

4. Right-click and click the **Library Browse** popup menu item.

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**. Note that you must update all components that 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. on page 1104

Results

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

Updating a Design from the Library

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, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Procedure

1. In the design, select the part to copy.

2. Click the **ECO Toolbar > Add Component** button.

   (If this is the first time in the current session you are using the ECO Toolbar, the ECO Options Dialog Box appears. Set the options you want and click **OK** to use the ECO tools.) As an alternative, click the **Edit > Copy** menu item, then **Edit > Paste** menu item.

   The component attaches to the pointer.

3. Optional: Before placing the component, you can assign a reference designator. If you do not, the new part is automatically annotated with the next available reference designator.

   Right-click and click the **Rename Current Part** popup menu item. The Rename Part prompt appears asking for a new reference designator. Type the new reference designator, and click **OK**.

4. Move the part to the appropriate position and click to place it.

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, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Restrictions and Limitations

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. Click the **ECO Toolbar > Delete Component** button.

   (If this is the first time in the current session you are using 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.

   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(s).

5. Click **OK** to confirm the deletion.

   Test points are deleted with the part.

6. If the deletion of the component(s) creates single-pin nets, a second Delete Part dialog box on page 1267 appears and displays which nets will also be deleted. This is not an optional step. Click **Close**.

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, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Restrictions and Limitations

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. Click the **ECO Toolbar > Delete Connection** button.

   (If this is the first time in the current session you are using 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 a Pin Pair — Select the connection or trace between the pins of 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" for information on how to do this.

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.

Warning:
If you split a net associated to bridge copper, the association is removed from the copper.

CAUTION:
When you modify or delete design objects, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Restrictions and Limitations

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.

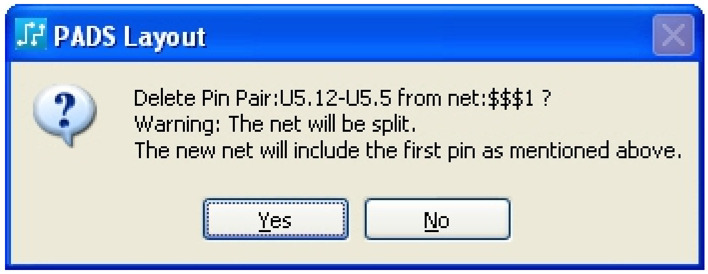
Procedure

1. Click the **ECO Toolbar > Delete Connection** button.

   (If this is the first time in the current session you are using 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.

You are prompted with a message like this:



Click **Yes**.

3. In the Define Name of New Net Dialog Box, name the new net or let SailWind Layout automatically generate the new netname.

   Netnames:

   - Have a maximum length of 47 characters.
   - Can contain any alphanumeric characters except brackets { }, asterisks *, commas (,), spaces, and question marks.

Results

- If a Color by Net setting exists for a net that is split, the new net uses the same color setting. For more information, see "Assigning Colors to Nets".
- 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, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Restrictions and Limitations

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. Click the **ECO Toolbar > Delete Net** button.

   (If this is the first time in the current session you are using 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.

3. Click **OK** to confirm the deletion.

   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, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Restrictions and Limitations

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. Click the **ECO Toolbar > Rename Component** button.

   (If this is the first time in the current session you are using 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

Recorded Versus Generated ECO Files

Recording ECO Changes

Backward Annotation from SailWind Layout to SailWind Logic

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, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Restrictions and Limitations

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.

   All selected components must currently have the same prefix.

2. Click the **ECO Toolbar > Rename Component** button.

   (If this is the first time in the current session you are using 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

Recorded Versus Generated ECO Files

Recording ECO Changes

Backward Annotation from SailWind Layout to SailWind Logic

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.

Restrictions and Limitations

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. Click the **ECO Toolbar > Rename Net** button.

   (If this is the first time in the current session you are using 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. For more information, see "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, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Restrictions and Limitations

This command ignores objects that are part of a physical design reuse. 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. Click the **ECO Toolbar > Terminator Assign** button.

   (If this is the first time in the current session you are using the ECO Toolbar, the ECO Options Dialog Box appears. Set the options you want and click OK to use the ECO tools.)

2. Click **Yes** to the displayed prompt.

Swapping Gates

This section describes the processes used to swap gates manually or automatically.

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, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Restrictions and Limitations

- 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. Click the **ECO Toolbar > Swap Gate** button.

   (If this is the first time in the current session you are using 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 the highlighted pin to swap with.

Results

If the Stretch Traces During Component Move check box is selected (on the Options dialog box > Design category on page 1503), 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.

Tip
You can undo your last Swap Gate command immediately after performing the swap. Right-click and click the **Undo Last Swap** popup menu item while still in Swap mode, or exit Swap mode and click the **Undo** button on the Standard Toolbar.

Related Topics

Part Information Dialog Box, Gates Tab

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, electrical nets are regenerated. This can cause existing electrical nets that include these objects to be truncated, split, or deleted altogether.

Restrictions and Limitations

- 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.

   (If this is the first time in the current session you are using 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.

Results

If the Stretch Traces During Component Move check box is selected (on the Options dialog box > Design on page 1503), 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.

Tip
You can undo your last Auto Swap Gate command immediately after performing the swap. Just click the **Undo** button on the Standard Toolbar.

Related Topics

Part Information Dialog Box, Gates Tab

Swapping Pins

This section describes the processes used to swap pins manually or automatically.

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 and Limitations

- 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. Click the **ECO Toolbar > Swap Pin** button.

   (If this is the first time in the current session you are using 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.

   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 ought to be swappable, the correct process would be to give them identical swap IDs in the Pins tab of the Part Information dialog box on page 1590.

4. Click **OK**.

Results

If the Stretch Traces During Component Move check box is selected (on the Options dialog box > Design category on page 1503), 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.

Tip
You can undo your last Swap Pin command immediately after performing the swap. Right-click and click the **Undo Last Swap** popup menu item while still in Swap mode, or exit Swap mode and click the **Undo** button on the Standard Toolbar.

Related Topics

Part Information Dialog Box, Pins Tab

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 and Limitations

- 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.

   (If this is the first time in the current session you are using 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.

Results

If the Stretch Traces During Component Move check box is selected (on the Options dialog box Design category on page 1503), 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.

Tip
You can undo your last Auto Swap Pin command immediately after performing the swap. Just click the **Undo** button on the Standard Toolbar.

Related Topics

Part Information Dialog Box, Pins Tab

Copied 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 a few noted exceptions.

The following characters are illegal in netnames:

- SpacesCommas (,)
- Braces ({ })
- Asterisks (*)
- Question marks (?)

The following characters are illegal in part names:

- SpacesCommas (,)
- Braces ({ })
- Asterisks (*)
- Question marks (?)
- Periods (.)
- Ampersands (&)

Chapter 37
Comparing Designs

This section discusses how to compare two versions of a design. It explains how to generate a differences report and the files you can use to update the older design to match the newer one.

Design Comparison

You can compare two versions of a design and create the files needed to update the original design to match the new design. Before comparing a schematic to a PCB layout, create a PADS-format ASCII netlist file (*.asc*) by generating a netlist in SailWind Logic or other schematic tool.

When you compare the updated schematic to the original PCB layout and then update the PCB layout to match the schematic, the process is called forward to layout or forward annotation. Similarly, when you compare the updated PCB layout to the original schematic and then update the schematic to match the PCB layout, the process is called backward from layout or backward annotation.

If SailWind Layout and the schematic tool are on the same computer, you can use more convenient automated tools to compare and update design versions. If SailWind Layout and the schematic tool are not on the same computer, you can use SailWind Layout to compare two versions of the design. See the following for more information:

- For SailWind Logic, see “Working with SailWind Logic”.

Design comparison can handle unused pins nets. An unused pins net contains all the component pins with no assigned net and groups them into one large net.

Design comparison does not do any of the following:

- Add pins removed from logic nets to the unused pins net.
- Use the reuse definition; the actual elements in the physical design reuse are used during comparison.

During comparison, it is assumed the new design contains the most current Attribute Dictionary. If an attribute is not ECO registered in the new design, the attribute is backward annotated only if you clear Compare only ECO Registered Parts check box on the Comparison tab of the Compare/ECO Tools dialog box on page 1181. If an attribute is ECO registered in the new design, the attribute is backward annotated and the value in the old design is updated, but the ECO registration for the attribute in the old design is not updated.

Comparing Two Versions of a Design

To compare two versions of a design, generate a new *.eco* file, a differences report, an ASCII file for back-annotating to a schematic, or all of them.

Tip
As an alternative, you can use the ECOGEN on page 844 command in SailWind Layout to do this.

You can compare two versions of a design in any of the following forms:

- The PCB design that is currently open in SailWind Layout
- A PADS-format ASCII netlist file (*.asc*) representing the schematic or the PCB layout
- A PCB layout file (*.pcb*)

If you are trying to forward or backward annotate design changes between SailWind Layout and SailWind Logic, you should record changes in an *.eco* file as you make them. This gives the best back-annotation results. For more information, see "Recording ECO Changes or " Backward Annotation from SailWind Layout to SailWind Logic.

For more details on the differences of these methods, see "Recorded Versus Generated ECO Files."

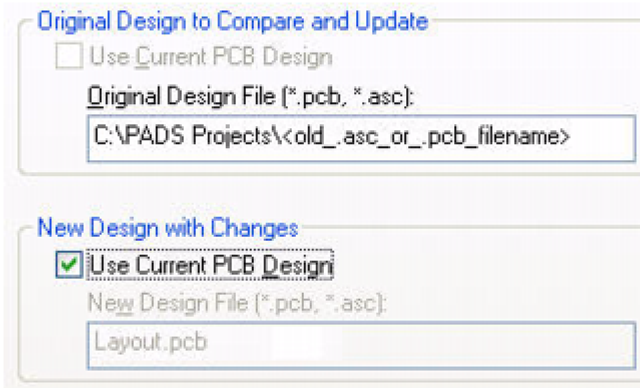
Tip
If SailWind Layout and the schematic tool are on the same computer, you can use SailWind Layout Link in SailWind Logic to both create the ECO file and forward or back-annotate the schematic. For more information, see "Working with SailWind Logic".

Procedure

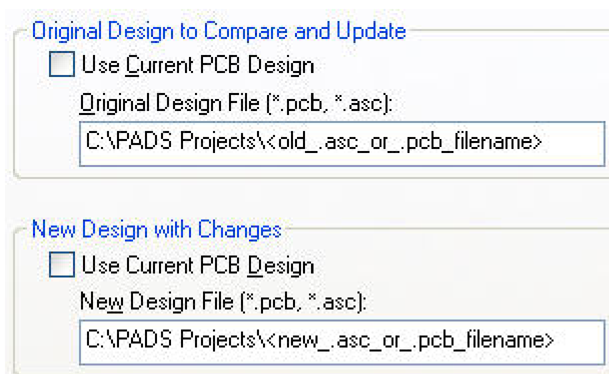
1. Click the **Tools > Compare/ECO** menu item.
2. In the Compare/ECO dialog box, on the Documents tab on page 1185, select the designs to compare as follows:

   - 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. In the Output Options area, select the files you want to generate.

4. Set the appropriate options on the Comparison on page 1181and Update on page 1187 tabs.

5. Click **Run** to compare the designs and create the files you specified. (The **Run** button becomes available when you select an option in the Output Options area.)

6. Click the **Show Report** buttons in the Process Status dialog box to view the generated files.

7. Click the **Close** button.

Results

Any messages or errors that occur during comparison are written to *Layout.err*, which is stored in the *\SailWind Projects* folder.

If the original design is in memory and you selected Update Original Design on the **Update** tab, the updates are automatically imported into the design when comparison is completed. To confirm that the updated design matches the new design, run the compare again, and select only the Generate Differences Report check box. If there are still differences, use the ECO Toolbar commands to correct the design.

If you did not select Update Original Design to automatically update the original design, click the **File > Export** menu item to import the *.eco* file and update the design.

After verification is complete, click the **Tools > Verify Design** menu item to check the design integrity.

Tip
If commands in the ECO file added any parts at the origin in the updated original design, remember to place these parts and route any newly unrouted pin pairs.

Related Topics

Design Comparison

Comparing Designs Using ECOGEN

Comparing Designs Using ECOGEN

After you make changes to a design, you can use the ECOGEN command to compare the new version of the design to the original, and create the files needed to update the original design to match the new design.

Tip
As an alternative, you can use the "Compare/ECO Tools dialog box" on page 1185 in SailWind Layout to do this.

It is good practice to record changes in an *.eco* file as you make them. This gives the best back-annotation results. For more information, see "Recording ECO Changes". For more information on the differences between these methods, see "Recorded Versus Generated ECO Files".

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.

Restrictions and Limitations

- 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.
- ECOGEN ignores virtual pins.
- ECOGEN ignores the reuse definition on page 1854 and uses the actual elements in the physical design reuse for comparisons.

Prerequisites

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.

   Click the **Start > Windows Accessories > Windows System > Command Prompt** menu item.

2. Change to the folder where the *ecogen.exe* file resides, for example, C:\<install_folder>\<version>\Programs.

3. Run ECOGEN in one of the following two ways:

   - 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>]
     ```

     For example:

     ```
     ecogen \ePD\3.1\project\logic.asc C:\SailWind 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.

     In the command line:

     - You must enter the newDesign, originalDesign, and ecoFile arguments in the given order.
     - If an argument contains spaces, enclose the argument with double-quotes " ".

   - Type ECOGEN@<command_file>, where <command file> is a pathname to a file containing all the parameters on one line.

Examples

The following table contains the parameter usage. Square brackets [ ] enclose optional parameters.

Table 122. 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.
[-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 SailWind Logic schematic, use the -aPART,NET switch value. To compare board, part, net, and pin attributes, for example, for a PADS Designer 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.
[-g], [-q]	**Tip** • 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.
	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.

Table 122. Parameter Usage (continued)

Parameter	Specifies
	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.
[-lRulesMask]	Specifies rules comparison and the object types, rule types, and rule kinds to be compared (RulesMask). For RulesMask, 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 PADS Designer 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.

Table 122. Parameter Usage (continued)

Parameter	Specifies
	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.

Related Topics

Design Comparison

Comparing Two Versions of a Design

Differences Report

The Differences Report (*Layout.rep*) reports the results of a comparison of two design files. (You perform the comparison using the **Tools > Compare/ECO** menu item or the ecogen command in the Windows command prompt.) For example, you can compare a new design (one with changes) to the original design (before changes). The Differences Report lists the changes found in the new design as compared in the original design.

Part Differences

Lists part type information and part placement information in separate sub-sections.

The part type information sub-section lists the reference designator and the part type for both the old and new designs. Parts that exist only in the old design are listed under the New Design column as <none>. Parts that exist only in the new design are listed under the Old Design column as <none>. Parts that are renamed are listed on the same line. Parts that have new part types are listed on the same line. Parts that have new assigned decals are listed on the same line. Parts that are identical by reference designator and part type in both designs are not listed.

The part placement information sub-section lists the differences for each part's x/y coordinates, glue status, and mirror (flip) status. Part placement information is reported only for parts that exist in both the old design and the new design.

Net Differences

Lists names of the nets that do not exist. Lists the nets that match, but have different names, including nets in the old design that have been combined in the new design. A net split operation appears as pin differences. Nets are listed alphabetically under the Old Design column, except where multiple nets are combined, when they are listed in succession. Nets that do not exist in the old design are listed at the end of this section.

Swapped-Gate Differences

Lists any gates from the old design that are swapped with gates in the new design. The report lists reference designators for the parent components in the design followed by the pins in the gate.

Swapped-Pin Differences

Lists any swapped pins in the old design that are swapped with pins in the new design. This list provides reference designators for the components in the design followed by the swapped pins.

Unmatched Net Pins in Old Design

Lists any connected pins in the old design that are missing or connected to other nets in the new design. These are the pins that are deleted from nets during the ECO process. This list provides net names in the old design followed by unmatched pins in the net. If the net does not exist in the new design, all pins in the net are listed.

Unmatched Net Pins in New Design

Lists any connected pins in the new design that are missing or connected to other nets in the old design. These are the pins that are added to nets during the ECO process. This list provides net names in the new design followed by unmatched pins in the net. If the net does not exist in the old design, all pins in the net are listed.

Attribute Differences

Lists each object under the following headings: Attribute Name, Old Value, and New Value. Attribute differences are included only for objects that exist in both the old design and the new design. If an attribute is missing in either design, the value is listed as <no attr>. If the attribute exists, but has no value, it is listed as <no value>.

Tip
To generate a report containing the mechanical (nonelectrical) parts in the design, clear the “Compare only ECO Registered Parts” check box on the Comparison tab on page 1181 of the Compare/ECO Tools dialog box.

Unmatched Net Pin Pairs in Old Design

Lists any pin pairs in the old design that are missing, connected to other nets, or connected to the same scheduled net in a different place in the new design. These would be the pin pairs that would be deleted from nets during the ECO process.

The report lists net names in the old design followed by the unmatched pin pairs in the net. If the net is missing in the new design, then all the pin pairs in the net are listed.

Unmatched Net Pin Pairs in New Design

Lists any connected pin pairs in the new design that are missing, connected to other nets, or connected to the same scheduled net in a different place in the old design. These would be the pin pairs that would be added to nets during the ECO process.

The report lists net names in the new design followed by the unmatched pin pairs in the net. If the net is missing in the old design, then all the pin pairs in the net are listed.

Rules Differences

This section reports differences in design rules between two designs. This section lists each object that has rules differences as a subheading. The subheading has three columns: Rule Name, Old Value, and new Value. (The Rules Differences section heading shows the units in which the values for rules are displayed.)

The subheading for each object type includes:

- The object type
- The object name in the original design
- The object name in the new design (if the name is different)
- The rule type

For example:

```
RULES DIFFERENCES (Values in mils)
          Old Object Name -> New Object Name -> Rule Type
          Rule Name           Old Value         New Value
          NETCLS              CLASS1 -> MYCLS   ROUTING
          PIN_SHARE           ON                OFF
          VALID_LAYERS        Top               Bottom
```

This example shows a routing rule set change for the CLASS1 class. The change renamed CLASS1 to MYCLS and included changed in the pin share and valid layers rules. No other routing rules were changed.

Differences in Net Classes

This section lists names of net classes that:

- Do not exist in one design or another. (Classes that do not exist in the original design appear at the end of the section.)
- Match but have different names

For example:

```
NET CLASS DIFFERENCES
Old Design      New Design
CLASS1          MYCLASS
CLASS2          <none>
<none>          NEWCLS
```

This example shows that in the new design:

- The CLASS1 net class was renamed to MYCLASS.
- The CLASS2 net class was removed.
- The NEWCLS net class was added.

Class Nets That Were Removed

This section reports nets that are in the original design but were removed from the new design. (The net class in the original design has nets either not included in the new design or included in different net classes in the new design.)

This section lists:

- Each net class in the original design from which nets were removed and the names of those removed nets
- All of the nets in the net class if the net class does not exist in the new design

For example:

```
REMOVED CLASS NETS

CLASS1         $$$1879    $$$1906    $$$1928
               $$$1920
CLSMESH        GNC        VCC
```

This example shows:

- Nets $$$1879, $$$1906, $$$1928, and $$$1920 in the CLASS1 net class of the original design are missing from that net class in the new design.
- Nets GND and VCC in the CLSMESH class of the original design are missing from that net class in the new design.

If you use the ECO operation to update the original design, the operation deletes the nets from net classes in the original design.

Class Nets That Were Added

This section reports nets that are not in the original design and are added in the new design. (The net class in the new design has nets either not included in the original design or included in different net classes in the original design.)

This section lists:

- Each net class that has added nets in the new design and the names of added nets
- All of the nets in the net class if the net class is new (does not exist in the original design)

For example:

```
ADDED CLASS NETS

CLASS1          ANDROID    BAJOR    SPOT
                DATA00     DATA01
CLSMESH         ZORG       GND2
```

This example shows that in the new design:

- ANDROID, BAJOR, SPOT, DATA00, and DATA01 nets were added to CLASS1.
- ZORG and GND2 nets were added to the CLSMESH class.

Differences in Pin-Pair Groups

This section lists pin-pair groups that:

- Do not exist in one design or the other. Pin-pair groups that do not exist in the original design appear at the end of the section.
- Match but have different names.

For example:

```
PIN-PAIR GROUP DIFFERENCES
Old Design       New Design
GROUP1           <none>
GROUP2           GROUPB
<none>           NEWGRP
```

This example shows that in the new (changed) design:

- The GROUP1 pin-pair group was removed.
- The GROUP2 pin-pair group was renamed to GROUPB.
- The NEWGRP pin-pair group was added.

Pin Pairs That Were Removed From Groups

This section reports pin-pairs that were in the original design but removed from the new design. (The pin-pair group in the original design has pin-pairs either not included in the new design or included in different groups in the new design.)

This section lists:

- Each group in the original design from which pin-pairs were removed and the names of removed pin-pairs
- All of the pin-pairs in the group if the group does not exist in the new design

For example:

```
REMOVED GROUP PIN-PAIRS

GROUP1        U2.2-U1.2 U3.3-U2.2 U3.2-U4.2
              U1.5-U2.5 U4.5-U1.5
GLMESH        R102.1-C2.2
```

This example shows that in the new (changed) design:

- Five pin-pairs were removed from GROUP1.
- The pin-pair R102.1-C2.2 was removed from the GLMESH group.

Pin-Pairs That Were Added to Groups

This section reports pin-pairs in the new design that are not in the original design. (The pin-pair group in the new design has pin-pairs either not included in the original design or included in different groups in the original design.) An ECO operation adds these pin-pairs.

This section lists:

- Each group with added pin-pairs in the new design and the names of added pin-pairs.
- All of the pin-pairs in the group if the group is new (does not exist in the original design).

For example:

```
UNMATCHED GROUP PIN-PAIRS

G         U2.2-U1.2 U3.2-U2.2
DASL      R29.2-U23.17
```

This example shows that in the new (changed) design:

- Two pin-pairs were added to the G group.
- The pin-pair R29.2-U23.17 was added to the DASL group.

Chapter 38
Reports

You can generate reports for the currently loaded design. Several report formats are provided, plus you can create reports formatted to match existing design standards. Each report pass creates a file called report.rep, which is stored in the *\SailWind Projects* folder.

Report Types

SailWind Layout includes two types of reports; predefined reports and customizable reports. The predefined reports provide useful statistical information about your design. The customizable reports allow you to create and configure a report that contains design-specific information that you desire for checking and testing your design.

- Predefined Reports
- Customizable Reports

Predefined Reports

The following table lists the predefined report types, which cannot be deleted.

Table 123. Predefined Reports

Report	Description
Unused	Provides a listing of all unused pins for each package in a design.
Statistics	Provides a variety of statistical information in a design such as number of layers, drill locations, and routed connections.
Limits	Provides maximum numbers of the various design items, based on your program's package limits.

Customizable Reports

SailWind Layout includes report format files that you can customize to fit specific output requirements. These files are located in *C:\<install_folder>\<version>\Settings* and were created using the Report Generation Language (RGL). The file name extension used for these files is *.fmt*.

To help you select the proper report, report files are listed in the Reports dialog box by description of output, instead of by file name. Table 124 lists the formats, file names, and their descriptions.

Table 124. Customizable Reports

Report	Format File	Report Description
Net List w/o pin info	*netlist.fmt*	Signals by netname without pin information
Net List w/pin info	*netlistp.fmt*	Signals by netname with pin information
Parts List 1	*parts1.fmt*	Parts by reference designator
Parts List 2	*parts2.fmt*	Reference designator by part type
Test points report	*testpnts.fmt*	Test point locations and netname
Jumper List	*jumpers.fmt*	Jumper locations and netnames
SailWind Format Netlist	*padsnet.fmt*	Netlist in current SailWind Layout format
PADS Format Netlist	*padsnetV2.fmt*	Netlist in PowerPCB 2.0 format
DFT Extended Test Point	*testpoint.fmt*	Test points by nets, nets without test points, and number of test points per net **Note:** You must have the DFT Audit licensing option to select DFT Extended Test Point.

Jumper List Report

You can create a Jumper List Report that lists all jumpers and their characteristics. Table 125 shows a sample report file.

```
JUMPER LIST REPORT -- a b c d e f g h i j.pcb
```

```
TOTAL = 3 jumper(s)
```

Table 125. Jumper List Report

Ref.Nm	Angle	Length	X1	Y1	X2	Y2	Signal
JMP1	90.000000	350	1825	3200	1825	3550	GND
JMP2	0.000000	250	2600	3275	2850	3275	GND
JMP3	90.000000	525	2725	2400	2725	2925	DA01

Running a Report

You can choose to run any of the 12 pre-configured reports. Two of the reports produce old format netlists.

Procedure

1. Click the **File > Reports** menu item to open the Reports Dialog Box.
2. In the Select Report Files for Output list, select one or more report formats.

   If the report format you want is not available, you can create a custom format, see "Adding or Removing Report Formats" or "Creating Custom Reports Using the Report Wizard".
3. Click **OK**.

Results

The report is written to *C:\SailWind Projects\report.rep* and displayed in the default text editor. If you selected more than one report format, *report.rep* contains all the reports.

Running a Report Using an Assembly Variant

If your design uses assembly variants, you can create a report based on an assembly variant instead of the base design.

Procedure

1. Click the **File > Reports** menu item to open the Reports Dialog Box.
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.

   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 on page 1853 data is reported.
4. Click **OK**.

Results

The report is written to *C:\SailWind Projects\report.rep* and displayed in the default text editor. If you selected more than one report format, *report.rep* contains all the selected reports.

Adding or Removing Report Formats

You can add or remove report formats from Reports dialog box. If you want to add a report format, you first create the format file using Report Generation Language.

For more information, see Customizable Reports.

Procedure

1. Click the **File > Reports** menu item to open the Reports Dialog Box.
2. Modify the list of report formats:

- Add to the list a custom report format, or import a report format file:

    i. Click **Add**.

    ii. In the Report Format File dialog box, browse to the format file and click **Open**. The format is added to the list and copied to the *C:\<install_folder>\<version>\Settings* folder.

- Remove a report format from the list:

    i. Select the format.

    ii. Click **Delete**. You cannot delete the Unused, Statistics, and Limits report types. The format is removed from the list and the file name extension is changed from *.fmt* to *.del*.

3. Click **OK**.

Related Topics

Creating Jumper List Reports

Creating Custom Reports Using the Report Wizard

In a maximum of 8 steps, the Script Wizard prompts you to choose information and once complete, it creates a reusable script of the report format, runs the script against the current design, and generates your report in Plain text, RTF, Excel, and HTML formats. This method is easier than manually writing the report formats in Report Generation Language for the Reports dialog box.

Procedure

1. Click the **Tools > Basic Scripts > Basic Scripts** menu item.
2. In the Basic Scripts Dialog Box, scroll through the list of scripts and select SailWind Layout Script Wizard.
3. Click **Run**.
4. Proceed through the screens of the Wizard and on the final screen, either click Finish to create the script of the report or click Finish & Run Report Now to see the report also.

   The basic script that generates the report appears in the Basic Scripts dialog box for reuse and the report opens in the chosen format. You can delete the basic script of the report if you do not intend to use it again.

Examples

Create and run a component height report listing the height of each part in the design.

1. Click the **Tools > Basic Scripts > Basic Scripts** menu item..
2. In the Basic Scripts Dialog Box, scroll through the list of scripts and select SailWind Layout Script Wizard.
3. Click **Run**.
4. On the Introduction screen, click **Next**.
5. On the Format screen, click Microsoft Excel and then click **Next**.
6. On the Report Type screen, click PCB-Based Reports and then click **Next**.
7. On the Database Object screen, click Parts and then click **Next**.
8. On the Data Type screen, click General Part properties in table format and then click **Next**.
9. On the Object Properties screen, in the Attribute (Select existing or type any valid name) list click Geometry.Height and then click **Add**.
10. On the Report Options screen, click **Next**.
11. On the Output Files screen, in the “Enter the name of the resulting VB script (without BAS extension)” box type a name for the report to be listed in the Basic Scripts dialog box.
12. Click **Finish & Run Report Now**.
13. An Excel report is generated for the current design reporting on the value of the Geometry.Height attribute of each part. When you need to generate the report again, select it in Basic Scripts dialog box and click Run.

Chapter 39
Checking a Design for Errors

This section describes the methods for checking a design for errors. It covers the Verify Design process as well as the interactions presented by Design For Test (DFT), Design For Fabrication (DFF), CAM, Thermal Analysis and the 3D PCB Viewer.

Design for Test

To support In Circuit Testing (ICT) procedures, SailWind Layout's DFT Audit can help you manage in-circuit test points. Using parameters that you set, DFT Audit can analyze all nets in the design, automatically assign test point attributes to the appropriate vias and component pins on accessible (adaptable) nets, add test points to adaptable nets that are already routed, and report inaccessible (non-adaptable) nets.

For non-adaptable nets, DFT Audit can add test point vias and place them outside the board outline. These capabilities help you consider ICT early in the design process, improving your productivity by reducing potential iterations of a manual DFT Audit.

To manually assign a component pin or via as a test point, you add a test point setting to the object. A test point can be one pin of a multiple pin component, the only pin of a single test point component, or a via.

DFT Audit assigns vias and component pins as test points rather than adding several single pin components. Therefore, you avoid backward annotation of test point information to the schematic. For more information, see "Test Point Definition".

When you run DFT Audit, SailWind Layout automatically transfers the design to SailWind Router. Using parameters that you set in SailWind Layout, SailWind Router analyzes all nets for adaptability and adds test points to routed adaptable nets. Note that SailWind Router may reroute nets during DFT Audit. When SailWind Router is done, it transfers the design back to SailWind Layout. For nets that SailWind Router determines to be non-adaptable, SailWind Layout can optionally add test points, which are placed outside the board edge. When DFT Audit finishes, the DFT Audit Board Report appears.

To access the DFT Audit, click DFT Audit on the Tools menu. For information about running DFT Audit, see the Performing a Test Point Audit topic.

Restriction:
DFT Audit tolerates slotted holes, but does not test them for adaptability.

While you can run DFT Audit in either SailWind Layout or SailWind Router, the dialog box used to set a DFT Audit option depends on the program you are running. For more information, see "Mapping SailWind Layout DFT Audit Settings to SailWind Router".

- Test Point Definition
- DFT-Related Options
- Compare Test Points
- Creating a Test Point ASCII File
- Performing a Test Point Audit
- Placing Test Points
- Setting Test Point Properties
- Setting Test Point Assignment Eligibility
- Probing the PCB Top Side Only
- Modifying a Jumper Pin that is a Locked Test Point
- Modifying a Pin that is a Locked Test Point
- Modifying a Route Attached to a Locked Test Point
- Modification of a Via or Virtual Pin That is a Locked Test Point
- Moving a Via or Virtual Pin That Is a Locked Test Point

Move Sequential to Move Components, Unions, or Clusters with a Locked Test Point
Moving, Dispersing, or Aligning a Component, Cluster, or Union with a Locked Test Point

Test Point Definition

Specific test point terminology is used to describe the various parameters used in setting up your test points in the design.

Figure 117 show the different parts of a test point, while Table 126 describes the parts.

Figure 117. Different Parts of a Test Point

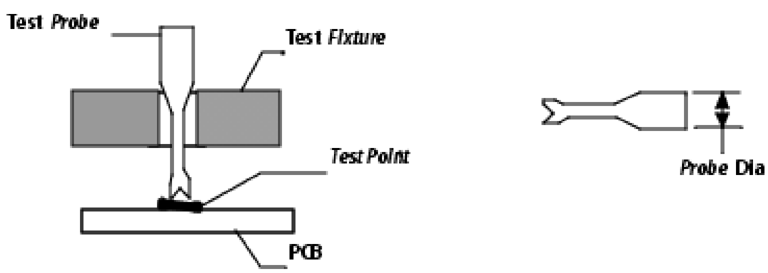


Table 126. Parts of a Test Point

Part of Test Point	Description
Test Probe	Also known as the probe, nail, nail pin, or tester pin. This object accesses the test point through the test fixture and makes contact between the test point on the PCB and the test equipment.
Test Fixture	A thick metallic plate attached to the ICT equipment that is customized for each PCB. The test fixture accurately positions test probes to their respective test points on the PCB. Test fixtures can be designed for a single side of the PCB, typically the bottom side, or for both sides of the board, which is called a clam fixture.
Test Point	The point on the net you are accessing, typically a via or component pin.
Head Type	The type of head style or contact point on the test probe. The head is the part of the probe that makes contact with the test point. You cannot set the Head Type in SailWind Layout.
Nail Diameter	An ASCII string assigned to a test point via or pin that equals the probe diameter.
Nail Number	A unique label assigned to a probe.

DFT-Related Options

In addition to adding more test point capabilities through DFT Audit, SailWind Layout contains additional features available within other functionality to support test points.

This test point information is discussed with the specific SailWind Layout topic and context-sensitive help is available from any dialog boxes you may encounter when working with test points. The table below shows impacted functionality.

Table 127. DFT-Related Options

Option	Impact
Add a test point	Manually adds a test point attribute to an existing via, jumper pin, or component pin.
ASCII I/O	Exports and imports test points.
Cluster Placement and Cluster options	Prompt you when you move, disperse, or collapse clusters with a locked test point.
CAM	Select Items dialog box displays test points. NC Drill Options dialog box plots test point locations.
Compare Test Points	Compares test point locations in two files.
ECO	Delete Connection, Delete Net, Delete Part, and Change Part handle test points differently.
End Test Point	Manually ends a route with a test point via on a dangling route.
Find	Finds by test points.
Modifying	Prompts you if you modify an object with a test point attribute; for example, changing the pad stack of a component pin that is a test point or changing the via type. See the Modifying Via Properties, Customizing Pad Stacks of Decal Pins, and Modifying Pin Properties topics.
Moving	Prompts you if you move a locked test point. This includes all moving commands for via, pin, cluster, union, or reroute, including spinning, rotating, and flipping objects. See the Moving Components, Moving, Dispersing, or Aligning a Component, Cluster, or Union with a Locked Test Point on page 873, Cluster Placement, and Moving a Trace Segment topics.
Reports	Extended reports for test points, including more keywords.
Routing tab	Displaying test points and locking test point locations.
SPECCTRA Translator	The SPECCTRA Translator supports via keepouts and a net of unused pins.
Verify Design	Checks for test point probe violations on the entire design.

Compare Test Points

Use Compare Test Points to compare test point settings in ASCII format, between the current file and another file.

Procedure

1. You need an ASCII file of test points from an older version of the design file.

   For more information, see "Creating a Test Point ASCII File".

2. Click the **Tools > Compare Test Points** menu item to open the Compare Test Points dialog box.

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**. SailWind 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.

Creating a Test Point ASCII File

You can create an ASCII test point list in an older design format.

Procedure

1. Open the *.pcb* file in SailWind Layout.

2. Click the **File > Export** menu item to open the File Export dialog box.

3. Type a name in the File name box.

4. Click **Save** and 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**.

Performing a Test Point Audit

Use DFT (Design For Test) 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, SailWind Layout automatically transfers the design to SailWind Router. Using options you set in SailWind Layout, SailWind Router analyzes all nets for accessibility and adds test points to routed accessible nets. Note that SailWind Router may reroute nets during DFT Audit. When SailWind Router is done, it transfers the design back to SailWind Layout. For nets that SailWind Router determines to be inaccessible, SailWind Layout can optionally add test points, which are placed outside the board edge. When DFT Audit finishes, the DFT Audit Board Report appears.

Prerequisites

- SailWind Router must be installed and licensed for you to run DFT Audit in SailWind Layout. You can run DFT Audit regardless of the layer limits in the SailWind Router license.
- If you are using BGAs, fan out the BGAs before running DFT Audit.

Procedure

1. Click the **Tools > DFT Audit** menu item.
2. On the DFT Audit dialog box, click the tab for which you want to modify the test point placement options and properties:
   - Options on page 866
   - Properties on page 867
   - Assignment on page 868
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

Automatic Test Point Placement [SailWind Router User's Guide]

Placing Test Points

Several options for placing test points are available to you on the **Options** tab of the DFT Audit dialog box. Choose the options that best fit the requirements of your current design.

Tip
To reduce design iterations, consult with your automated test engineers when setting DFT Audit options.

Procedure

1. Click the **Tools > DFT Audit** menu item, then select the **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.

   Tip
   In DFT, virtual pins are treated as component pins.

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 that allow you to specify many of the physical characteristics of test points.

Tip
To reduce design iterations, consult with your automated test engineers when setting DFT Audit options.

Procedure

1. Click the **Tools > DFT Audit** menu item, then select the **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 interference from any obstacles. This means that test points must keep a fixed distance from component bodies, pads, mounting holes, and the 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 do not want any nail pins on a net, double-click the Nail Pins cell for the net and type zero (0). 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.

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.

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.

Procedure

1. Click the **Tools > DFT Audit** menu item, then select the **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.

   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

You can restrict test point probing to the PCB top side only. This is useful for preparing a test setup when physical restrictions exist that will not let you access the bottom of the board during testing.

Procedure

1. Create a test point keepout on page 465 on the bottom side of the board.
2. Click the **Tools > DFT Audit** menu item to open the DFT Audit dialog box.
3. Click the Options on page 866 tab.
4. In the Probe through area, select the Probe Top Side check box.
5. Click **Run**.

Results

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

When modifying or moving a jumper pin that is a locked test point, a message is presented indicating that the object is locked and you will be required to confirm your design intent.

Procedure

You will be modifying a locked test point in one of the following ways:

- 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 Options dialog box > **Routing** category > General subcategory on page 1542.
    - **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

When modifying a pin that is a locked test point, a message will presented indicating that the object is locked and you will be required to confirm your design intent.

Procedure

You will be modifying a pin that is a locked test point in one of the following ways:

- 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 one of the following:
  - **All —** Applies the change to all decal types.
  - **Selected —** Applies the change to the selected decal.
  - **Cancel —** Cancels 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

When modifying a route that is attached to a locked test point, a message will presented indicating that the object is locked and you will be required to confirm your design intent.

Procedure

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 Options dialog box > **Routing** category > General subcategory on page 1542.
- **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.

Modification of a Via or Virtual Pin That is a Locked Test Point

When modifying a via or a virtual pin that is a locked test point, you can choose to modify the pad stack, or assign a different via type.

Modifying the Pad Stack of a Via or Virtual Pin That Is a Locked Test Point

When modifying the pad stack of a via or virtual pin that is a locked test point, a message will presented indicating that the object is locked and you will be required to confirm your design intent.

Procedure

You are modifying the pad stack of a locked test point in one of the following ways:

- If you modify the pad stack of a via or virtual pin in the Pad Stacks Properties dialog box, the “Are you sure you want to change all vias of type xxx?” message appears. Click one of the following:
    - **Yes** — applies the change.
    - **No** — cancels the change.
- If the via/virtual pin is a locked test point, the “Include locked test points in Pad Stack update?” message appears. Click one of the following:
    - **Yes** — Applies the change and maintains the locked test point status.
    - **No** — For vias, cancels the change. For virtual pins, the change is made (that is, as though Yes had been selected).

Results

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.

Assigning a Different Via Type To a Via That Is a Locked Test Point

When assigning a different via type to a via that is a locked test point, a message will presented indicating that the object is locked and you will be required to confirm your design intent.

Procedure

If you assign a different via type to a via that is a locked test point, by selecting a new Via Name in the Via Properties dialog box, 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 Options dialog box > **Routing** category > General subcategory on page 1542.
- **OK** — Applies the change and maintains the test point status.
- **Cancel** — Cancels the change.

Assigning a Different Via Type To a Virtual Pin That Is a Locked Test Point

If you assign a different via type to a virtual pin that is a locked test point, by selecting a new Via Name in the Virtual Pin Properties dialog box, no message appears, and the change is made.

Procedure

Assign a different via type to a virtual pin that is a locked test point, by selecting a new Via Name in the Virtual Pin Properties dialog box.

Moving a Via or Virtual Pin That Is a Locked Test Point

When moving a via or virtual pin that is a locked test point, a message will presented indicating that the object is locked and you will be required to confirm your design intent.

Procedure

If you move a via or virtual 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 Options dialog box > Routing category > General subcategory on page 1542.
- **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

When using Move Sequential to move a components, union or cluster that contains a pin that is a locked test point, a message will presented indicating that the object is locked and you will be required to confirm your design intent.

Procedure

When using Move Sequential with a locked test point, one of the following occurs:

- 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

When moving, dispersing, or aligning a components, union or cluster that contains a pin that is a locked test point, a message will presented indicating that the object is locked and you will be required to confirm your design intent.

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.

Procedure

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. Click one of the following:

- **Disable Lock Test Points** — Turns the Lock Test Points check box off in the Options dialog box > **Routing** category > General subcategory on page 1542.

- **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.

DFF, Design For Fabrication

To support fabrication design rules, SailWind Layout provides fabrication checks with Verify Design. This functionality, called DFF Audit (Design For Fabrication) , lets you either check for fabrication errors within SailWind Layout or backward annotate errors from the CAM product, CAM350. DFF Audit detects potential errors in a design, so that you can identify these problems prior to board fabrication.

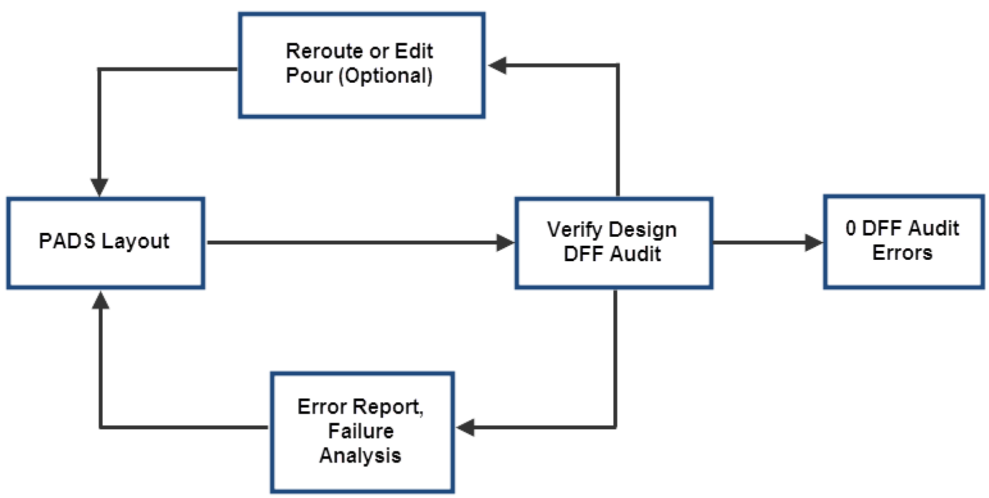
The checking within SailWind Layout uses the CAM document definitions to determine if fabrication errors exist. The CAM documents determine the photoplotter output and include layer composites, oversize, suppression, and other masking preferences. All electrical layers are analyzed to check acid traps and copper sliver fabrication. To check mask sliver and solder mask bridge fabrication, the solder mask layers are analyzed. To check silkscreen over pads, silkscreen layers are compared to solder mask layers.

Design for Fabrication Workflow

Design For Fabrication is best accomplished using a structured workflow to define the various steps of the process.

The table below shows the basic workflow for auditing your design for fabrication.

Figure 118. Design for Fabrication Workflow



Process Flow for Using DFF Audit

You can use DFF Audit to detect and resolve any fabrication errors. The audit process analyzes, verifies and displays any errors in the design related to the specific checks that you specify.

Prerequisites

You need the CAM350 Link license option to use this.

Procedure

1. Lay out the design using SailWind Layout on page 253 and SailWind Router on page 776.
2. Create CAM documents on page 934.
3. Set up Verify Design preferences for fabrication checks on page 894.
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

If you use CAM350 in your design process, you can use DFF Audit to detect and resolve any fabrication errors. The audit process analyzes, verifies and displays any errors in the design related to the specific checks that you specify.

Prerequisites

You need the CAM350 Link license option to use this.

Procedure

1. Lay out the design using SailWind Layout on page 253 and SailWind Router on page 776.
2. Create CAM documents on page 934.
3. Output the CAM documents to CAM350.
4. Set up Verify Design preferences for fabrication checks on page 894.
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 on page 876 to view errors. Use SailWind Layout to correct fabrication errors.
8. Repeat steps 3 through 7 until fabrication errors are resolved.

Exporting to CAM350

You can translate a SailWind Layout design using CAM350 Link. This allows you to perform additional design analysis on your design in the CAM350 environment.

Prerequisites

You need the CAM350 Link license option to use this.

Procedure

1. Create a SailWind Layout design.
2. Click the **File > Export** menu item, and in the Save as type list, click CAM350 and then click **Save**. The CAM350 dialog box appears.
3. Choose a mode - to either create the CAM350 *.cam* file only, or create the file and automatically launch CAM350.
4. In the Layer Options area, select an option for the amount of PADS design detail to translate to the CAM350 database.
5. Arcs in polygon shapes, such as copper planes, 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 Options dialog box > **Global** category > **General** subcategory.
6. 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.
7. Click **OK**.
8. Run your manufacturing process in CAM350.

Results

Once the design file is in CAM350, you can perform DFM analysis on the design. If DFM errors exist, you can back-annotate on page 889 the errors to SailWind 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 SailWind Layout or CAM350.

The processing routines that translate SailWind Layout data into CAM350 *.cam* files detect errors and warnings and report them to the standard SailWind Layout error file (*Layout.err*).

Working with Markups

You can use markups to define and document specific issues related to your design. This lets you notate design concerns and easily share them with associates or downstream process personnel.

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.

Prerequisites

You must create at least one topic section to begin adding issues and 2D line markups.

Procedure

1. Click the **Edit > Markups** menu item.
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 on page 881 to the markup.
8. Save the markups by choosing one of the following:

- Click the **Export** button in the Markups dialog box to export the markups into a *.cle* encrypted collaboration file.
- Click the **File > Save as** menu item to generate a snapshot of the design and create a *.cle* file to match it.

Note:
The markups are not saved when you click the **File > Save** menu item (or the **Save** button on the dialog box, or Ctrl+S).

Results

- 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 Using the Markups Dialog Box

Importing Markups Using File > Import

Exporting Markups Using the Markups Dialog Box

You can use the Markups Dialog Box to export markups for use in another software tool like CAMCAD and visECAD. The markup is exported into a simple *.cle* file which contains only the markup information.

Alternatively, you can export a *.CCE* file which includes most design elements along with the markups. See "Exporting Markups Using File > Export".

Procedure

1. Click the **Edit > Markups** menu item.
2. In the Markups dialog box, click the **Export** button.
3. In the Collaboration Data Save As dialog box, type a name for the *.cle* file.
4. Click **Save**.

Exporting Markups Using File > Export

You can use the **File > Export** command to export markups for use in another software tool like CAMCAD and visECAD. The markup is exported in a *.CCE* file which includes most design elements along with the markups.

Alternatively, you can export a simple *.cle* file which contains only the markup information. See "Exporting Markups Using the Markups Dialog Box".

Procedure

1. Click the **File > Export** menu item.
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**.

Results

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 Using the Markups Dialog Box

You can use the Markups Dialog Box to import markups created in another software tool, like CAMCAD or visECAD.

Restrictions and Limitations

Any markups that exist in the Markups dialog box will be deleted before the new data is imported.

Procedure

1. Click the **Edit > Markups** menu item.
2. In the Markups dialog box, click the **Import** button.
3. In the Collaboration Data Import dialog box, browse for and select the *.clb* or *.cle* file to import.
4. Click **Open**.

Related Topics

Importing Markups Using File > Import

Importing Markups Using File > Import

You can use the **File > Import** menu item to import markups created in another software tool, like CAMCAD or visECAD.

Restrictions and Limitations

Any markups that exist in the Markups dialog box will be deleted before the new data is imported.

Procedure

1. Click the **File > Import** menu item.
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**.

Results

- The Markups dialog box is populated with the imported collaboration data.
- Some shapes imported from other software have no equivalent in SailWind Layout (for example, sticky notes). They are represented as accurately as possible with 2D lines in SailWind 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.

Related Topics

Importing Markups Using 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 the **Link Selected** popup menu item.

Results

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 the **Unlink** popup menu item.

Related Topics

Linking Design Objects to Markup Issues

Verify 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.

Running a Design Check

You can check for individual or all design errors using the Verify Design dialog box:

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Check area of the Verify Design dialog box, choose a checking option.
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**.

Results

Error markers appear in the design at error locations and error details populate the Verify Design dialog box. For more information, see “Interpreting Error Markers”.

Review of the Design Error Results

Use Verify Design to examine error markers and view the descriptions of each error.

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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. Select an error from the list.

   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 including 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 workspace indicate the error type.

Prerequisites

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.

Procedure

1. You open the Verify Design database to perform a design check and generate a list of any errors occurring in your design.

   See “Running a Design Check” on page 883 for instructions.
2. With the “Verify Design dialog box” on page 1775 still open, select an error from the Location list.
3. The design window pans to the area of the error (unless you select Disable Panning first).

   The pan to the area of the area is more obvious if you are zoomed into your design.

Tip
You can prevent the design area from panning to errors when you select them if you select the Disable Panning check box in the Verify Design dialog box.

The generated error list remains whether the dialog box is open or closed. If you close the dialog box, you can view the list in the error report file. You can access the report file by clicking the link that appears in the Output window.

Note:
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.

Interpreting Error Markers

After verifying the design, error markers are inserted in a SailWind Layout database as database objects. Error objects present in a SailWind Layout database transfer into SailWind Router and are converted into SailWind Router error objects during file load.

Tip
Within SailWind Router you can assign an error as ignored. An ignored error status makes the error marker invisible but leaves the error in the database. SailWind Layout supports storage of the ignored status on errors; however, ignored status can only be assigned or unassigned within SailWind Router. In SailWind Layout, ignored errors are not invisible. Ignored errors invisible in SailWind Router saved to a SailWind Layout file will be visible when the file is loaded into SailWind Layout. But ignored errors remain as ignored after a file is saved in SailWind Layout and loaded SailWind Router.

Most error objects in SailWind Layout are converted to an identical SailWind Router error object. The exceptions to these conversions are listed in the Notes column of the table below.

Table 128. SailWind Layout/Router Error Objects

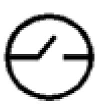
Marker	Error	Notes
	Testability and Connectivity	
	High-speed	
	Fabrication	

Table 128. SailWind Layout/Router Error Objects (continued)

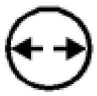
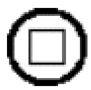
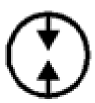
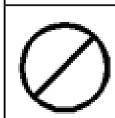
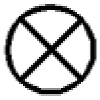
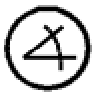
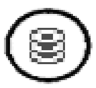
Marker	Error	Notes
	Minimum/Maximum Length	
	Assembly (Latium Only)	Component height errors become assembly errors in SailWind Router. Consequently, SailWind Router assembly errors in SailWind Router are converted to SailWind Layout keepout errors in SailWind Layout when the file is saved.
	Drill to Drill	Drill to Drill errors in SailWind Layout become fabrication errors in SailWind Router. Consequently, drill to drill errors in SailWind Router are converted to drill to drill errors in SailWind 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 SailWind Layout. "Body to body" checking is not performed in SailWind Router, only placement outline checking is performed as an assembly check. Therefore, "Body to body" errors cannot exist in SailWind Router. SailWind Layout "Body to body" errors are converted to placement outline errors in SailWind Router - identified by Assembly error markers. Consequently, SailWind Router Placement Outline errors in SailWind Router are converted to SailWind Layout placement outline errors in SailWind Layout when the file is saved. **Tip** "Body to body" errors are not returned from SailWind Router. **CAUTION:** When a design is loaded into SailWind Router, the SailWind Layout "Body to body" clearance rule value is converted to the SailWind Router "Minimum spacing between components" value. In SailWind Layout, the spacing between component outlines is performed between the edges of the lines used to define the component body outlines. In SailWind Router, the Placement outline checking is done using the centerline of the lines used to create the Placement outlines. Due to these subtle differences, components that meet the DRC spacing requirements in SailWind Router may have to be re-checked when the design returns to SailWind Layout.
	Keepout	Component height errors become assembly errors in SailWind Router. Consequently, SailWind Router assembly errors in SailWind Router are converted to SailWind Layout keepout errors in SailWind Layout when the file is saved.

Table 128. SailWind Layout/Router Error Objects (continued)

Marker	Error	Notes
	Board Outline	
	Maximum Angle	
	Maximum Via Number	One marker is added for every net with too many vias.
	Latium Check	The Latium error marker indicates that you have Latium rules on page 1835 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.

Troubleshoot Design Verify Errors

There are different types of errors such as clearance errors, subnet errors and untied plane pin errors. Interpreting them correctly will help you clear them from the design quickly.

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 on page 1328 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 Setup 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 stacks properties on page 1566 of the component pin and ensure the Plated check box is selected.
- **A via or pad without a plane thermal** — open the Via Properties on page 1779 or Pin Properties on page 1632 dialog box and ensure the Plane Thermal check box is selected.

A Full Plane Check with Same Layer Connectivity Check will also display a subnet error which is caused only by isolated planes. For more information, see “Plane Checking Setup”.

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

Saving and Printing Error Results

You can print error results, or alternatively, view the current error results in your default text editor.

Procedure

1. Click the **Tools > Verify Design** menu item.
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.

Related Topics

Setting Default Report File Names

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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. To specify the default file name of each report type click **Report File**.

This opens the Save As dialog box where you can name the report that you will run, if you do not want to use the default reference name for the report type.

If you do not give the report a unique name, it is always saved to the default file name.

Tip
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.

Prerequisites

You need the CAM350 Link license option to use this.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. Select an error in the Location box.
3. Click **Preview**.

Results

The CAM Preview Dialog Box opens.

Tip
Since error markers are not added to the CAM document, it helps to zoom into the design area before you click Preview. If the CAM Preview does not open zoomed into the same area, you can click the Workspace button in the Cam Preview window to match the workspace view.

Note:
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 SailWind Layout database, at the error location.

Prerequisites

- You need the CAM350 Link license option to use this.
- Existing DFM error markers in the SailWind 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 SailWind Layout.
3. Click the **Tools > Verify Design** menu item.
4. In the Verify Design Dialog Box, in the Check area, choose the Fabrication option.
5. Click **Setup**.
6. In the Fabrication Checking Setup Dialog Box choose the Annotate DFF Errors option.
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 on page 885 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the High Speed option.
3. Click **Setup**, then click **Add Nets** or **Add Classes**.
4. Select a net or class and then click **OK**.

Tip
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

Setup of High Speed (Electrodynamic) Checking

Setup of EDC Parameters

Setting Up Clearance Checking

Use the Clearance Checking Setup dialog box to specify which clearances to check during a Clearance verification.

CAUTION:
Clearance checks are only applied to design objects that are visible in the workspace view at your current zoom level and only those design objects with their colors are turned on. If you are zoomed in or have objects turned off, they are ignored by the clearance check.

Restrictions and Limitations

- 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Clearance option, and then click **Setup**.
3. Select the check box beside any types of clearance checks you want to enable and then click **OK**.

Tip
Acute angles create solder bridging between conductive objects during board manufacturing.

Related Topics

Verify the Design

Design Rule Hierarchy

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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, 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 and click **OK**.

Results

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

Latium checking performs advanced design checking using the Latium technology contained with SailWind Router.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Latium Design Verification option, and then click **Setup**.
3. Select the “Net to All” check box to check clearance rules on each net or hierarchical level against any other obstacle type.
4. Select the Board Outline check box to check clearance rules for the board outline and board cut outs.
5. 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.
6. Select the Keepout check box to check for keepout restriction violations.
7. Select the Same Net check box to check clearances between objects along the same net, as specified in the Clearance Rules dialog box.

   For more information, see “Design Rule Hierarchy”.
8. 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.
9. Select the Trace Width check box to check traces in excess of minimum and maximum widths, specified in the Clearance Rules dialog box.

   For more information, see “Design Rule Hierarchy”.
10. Select the Placement Outline check box to check outline against outline with the following considerations:

- 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.

11. Select the Via at SMD check box to check for via at SMD restriction violations.

    For more information, see "Pad Entry Rules Dialog Box".

12. Select the Differential Pairs check box to check for differential pair restriction violations.

    For more information, see "Creating Differential Pair Design Rules".

13. Select the Trace Length check box to check for length restriction violations.

    For more information, see "HiSpeed Rules Dialog Box".

14. 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. For more information, see "Performing a Test Point Audit".

Results

Latium design checks are performed with the following restrictions:

- 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 SailWind Layout and SailWind Router by running test point checking with the other Latium checks.

Related Topics

Verify the Design

Fabrication Checking

Use the Fabrication Checking Setup dialog box to enable fabrication checks, or to load DFF errors from a pre-existing CAM350 database and annotate into the design.

Note:
You need the CAM350 Link license option to use this.

Fabrication Checks Definition

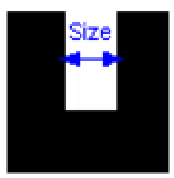
This section describes the DFF Audit options in the Fabrication Checking Setup dialog box.

Acid Traps

An acid trap is a location where, due to the surface tension of the etching, acid gets trapped in an area. This acid causes over-etching, which hurts yield. The acid trap runs on all visible electrical layers as defined by CAM documents.

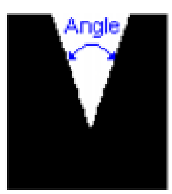
Acid Trap Maximum Size indicates the maximum size of the acid traps to flag. The area of pools that are flagged will be less than this value (Figure 119).

Figure 119. Acid Trap Maximum Size



Acid Trap Maximum Angle is an angle from 1 to 89 degrees. Any copper items (traces, pads, or any other objects that exist on the layer) that form an angle smaller than this are flagged as an acid trap (Figure 120).

Figure 120. Acid Trap Maximum Angle



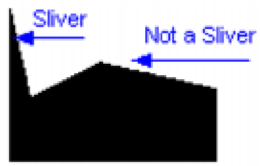
Slivers

Copper slivers are areas in the copper that are so narrow they may flake off. This check detects potential slivers on the electrical and composite layers in the design.

Copper slivers (Figure 121) should be eliminated whenever possible to prevent defect from occurring during the etching process.

Minimum Copper indicates the maximum size of the copper slivers to flag. This flags slivers of a width less than this value. This check runs on all visible electrical layers as defined by CAM documents.

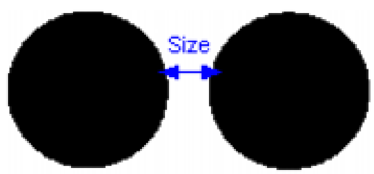
Figure 121. Sliver



Mask slivers in the solder mask layer are areas where the solder mask is so narrow they may flake off. These flakes float around and may drop into an area that needs to be soldered later, resulting in a bad board.

Minimum Mask (Figure 122) indicates the maximum size of the slivers to flag. This flags slivers of a width less than this value. This check runs top and bottom solder mask layers, if visible as defined by CAM documents.

Figure 122. Minimum Mask Size



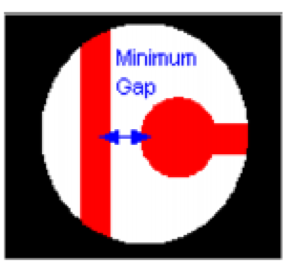
Solder Bridges

When a mask layer is created, openings for pads may be oversized too much and expose an adjacent trace or other conductive object. Therefore, during fabrication, the copper for that pad may become too close and create a bridge to the adjacent object. Solder bridges are usually caused by problems during mask data creation.

The CAD system used may be unable to validate that what was created is going to work.

The Minimum Gap (Figure 123) is the maximum distance the 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 the mask layer exposes it, it will not be identified as a bridge.

Figure 123. Minimum Gap



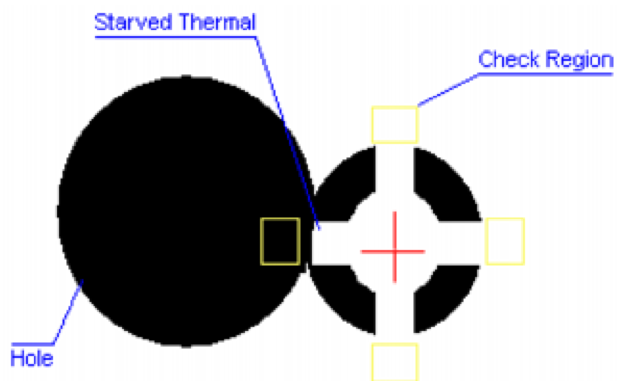
Starved Thermals

Many designs are plagued by thermal pad problems for negative CAM planes because the CAD system did not verify whether the thermals were going to make good connections to the copper plane.

The Starved Thermals fabrication check verifies whether each thermal connection to the negative CAM plane is valid, or if it has been constricted by adjacent data that is too close or overlapping – effectively starving out the ties. This check runs on all visible CAM negative plane layers as defined by CAM documents.

Starved Thermal Minimum Clearance is the percentage of the area next to the spoke of the thermal that must not be blocked by another object. Any smaller opening is considered starved.

Figure 124. Starved Thermals



Starved Thermal Minimum Spokes is the number of thermal spokes that cannot be blocked by another object. Any less will be considered starved. The number of spokes is specified as EVERY, meaning all spokes must not be blocked, or as an integer from 1 to 4.

Annular Ring

The Annular Ring area lets you set up annular ring checks by comparing data on different layers. This test checks both the size and the offset between the two layers.

The checks (described in the table below) assure that proper clearances are observed to alleviate manufacturing problems in the board fabrication processes. Layers to be tested are derived from CAM documents and pad stack data. This area provides selections for pad, mask, and drill checks. When drill sizes are analyzed for annular rings, the drill oversize setting on the Tools > Options > Design tab is not considered.

Table 129. Annular Ring Checks

Annular Ring Check Type	Checks
Annular Ring–Pad to Mask	The clearance between a pad and its solder mask opening. The offset and the annular ring are checked against the specified clearance value. This check is run on top and bottom electrical layers against their associated solder mask layers.

Table 129. Annular Ring Checks (continued)

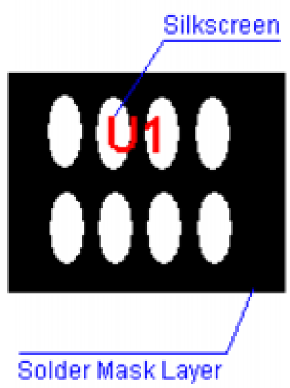
Annular Ring Check Type	Checks
Annular Ring–Drill to Mask	The clearance between a drill and its solder mask opening. The offset and the annular ring are checked against the specified clearance value. This check is run on the top and bottom drill layers against their corresponding solder mask layers.
Annular Ring–Drill to Pad	The clearance between a drill and its associated pad calculated as <pad size - drill size>.The offset and the annular ring are checked against the specified clearance value. This check can be run on any specified electrical layer.

Silkscreen Over Pads

The Silkscreen Over Pads check lets you set up the clearance for comparing data on silkscreen layers against top and bottom electrical layers. This check analyzes both the size and the offset between the two layers.

Layers to be tested are derived from CAM documents and pad stack data. This check is run for top and bottom electrical layers against their associated silkscreen layers.

Figure 125. Comparing Data on Silkscreen Layers



Trace Width/Pad Size

The Trace Width/Pad Size check runs minimum trace width and minimum pad size checks for electrical layers.

The Trace Width check detects small electrical traces on the electrical layers in the design. Minimum Trace indicates the maximum size of the traces to flag. Traces with a width less than this value will be flagged. This check runs on all visible electrical layers as defined by CAM documents.

The Pad Size check detects small pads on the electrical layers in the design before the board is manufactured. Minimum Pad indicates the maximum size of the pads to flag. Pads with a diameter less than this value will be flagged. This check runs on all visible electrical layers as defined by CAM documents.

Fabrication Check Types

Fabrication checks are used to verify that a design is manufacturable and that typical fabrication issues have been eliminated from the design.

Running Fabrication Checks

Fabrication checks are used to analyze a design and report potential fabrication issues that might create defects during the manufacturing process. Use the results of the fabrication checks to verify the manufacturing integrity of your design.

Prerequisites

Fabrication checks require certain CAM documents, which are listed below. You may consider creating these files before running the Fabrication check.

- To check Acid Traps, all electrical layer CAM documents are required.
- To check Copper Slivers, all electrical layer CAM documents are required.
- To check Solder Mask Slivers, the top and bottom Solder Mask layer CAM documents are required.
- To check Solder Mask Bridges, the top and bottom electrical and solder mask CAM documents are required.
- To check Starved Thermals, all negative CAM Plane layer CAMD documents are required.
- To check Minimum Annular Rings, all electrical layer and solder mask layer CAM documents are required.
- To check Silkscreen Over Pads, the top and bottom solder mask and silkscreen CAM documents are required.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Fabrication option, and then click **Setup**.
3. Choose the type of check that you want to run and set the required parameters.

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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Fabrication option, and then click **Setup**.
3. Select the Acid Trap check box.
4. 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.
5. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Fabrication option, and then click **Setup**.
3. Select the Slivers check box.
4. In the Minimum Copper box, type a minimum value for copper slivers.

   This flags slivers with less area than this value.
5. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Fabrication option, and then click **Setup**.
3. Select the Starved Thermals check box.
4. 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.”
5. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Fabrication option, and then click **Setup**.
3. Select the Trace Width/Pad Size check box.
4. 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.
5. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Fabrication option, and then click **Setup**.
3. Select the Silkscreen Over Pads check box.
4. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Fabrication option, and then click **Setup**.
3. Select the Annular Ring check box to enable the Pad to Mask, Drill to Mask and Drill to Pad checks.
4. The series of pad checking options are enabled for further definition.

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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Fabrication option, and then click **Setup**.
3. Select the Pad to Mask check box.
4. In the box, type the minimum clearance value.
5. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the **Fabrication** option, and then click **Setup**.
3. Select the Drill to Mask check box.
4. In the box, type the minimum clearance value.
5. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Fabrication option, and then click **Setup**.
3. Select the Drill to Pad check box.
4. In the box, type the minimum clearance value.
5. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Fabrication option, and then click **Setup**.
3. Select the Solder Bridges check box.
4. In the Minimum Gap box, type the minimum clearance value.
5. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Fabrication option, and then click **Setup**.
3. If not already selected, click the Annotate DFF Errors option.
4. 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 SailWind Layout for design verification.

   The file will be generated when the checks are run.

Related Topics

Fabrication Checks Definition

Verify the Design

Setup of 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.

Note:
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.

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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the High Speed option, and then click **Setup**.
3. 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.

4. Select an item from the Task List to enable High Speed checks for that item.
5. Select the check boxes for the types of checks you want to enable.

   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.
6. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the High Speed option, and then click **Setup**.
3. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the High Speed option, and then click **Setup**.
3. Click **Parameters**.

Related Topics

Setup of 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the High Speed option, and then click **Setup**.
3. If you need to edit or assign rules for checking, click the **Rules** button as a shortcut to the Rules dialog box.

Related Topics

Design Rule Hierarchy

Reusing Electrodynamic Check Settings

You can reuse your electrodynamic check settings. You can save the current electrodynamic check task, parameter and rules settings.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the High Speed option, and then click **Setup**.
3. Click **Save** or **Save As** to store your settings to an *.edp* file.
4. Click **Open** to retrieve settings from an *.edp* file.

Related Topics

Setup of EDC Parameters

Verify the Design

Setup of 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.

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.

Note:
Board Thickness is the total value of material and layer thicknesses in the current design units.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the High Speed option, and then click **Setup**.
3. In the Electrodynamic Check dialog box, click the **Parameters** button.
4. 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.

   Note:
   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.

5. For each dielectric material layer, double-click the Type cell to select whether the “layer” is a Prepreg or Substrate layer.

   Although it is a longer process, you can click a box and then click Edit instead of using the double-click method.

6. For each layer, double-click the Thickness cell and type a value.

   If no coating is required, set thickness to zero.

7. For dielectric material layers, double-click the Dielectric cell and type a dielectric constant value.

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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the High Speed option, and then click **Setup**.
3. In the Electrodynamic Check dialog box, click the **Parameters** button.
4. 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.
5. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the High Speed option, and then click **Setup**.
3. In the Electrodynamic Check dialog box, click the **Parameters** button.
4. 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 in the Design Verify process such as capacitance, impedance, delay, and length.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the High Speed option, and then click **Setup**.
3. In the Electrodynamic Check dialog box, click the **Parameters** button.
4. 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.
5. 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.
6. Select the Include Copper check box to include copper polygons with signal names in the capacitance calculations.
7. 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).

   For more information, see the PADS HyperLynx SI/PI 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the High Speed option, and then click **Setup**.
3. In the Electrodynamic Check dialog box, click the **Parameters** button.
4. Select the Report Segment Coordinates check box.

Listing Violations Only

You can list only items that contain violations in the high-speed report.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the High Speed option, and then click **Setup**.
3. In the Electrodynamic Check dialog box, click the **Parameters** button.
4. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the High Speed option, and then click **Setup**.
3. In the Electrodynamic Check dialog box, click the **Parameters** button.
4. Select the Remove Segments under Pads check box.

Related Topics

Verify the Design

Plane Checking Setup

Use the Mixed Plane Setup dialog box to set the type of Plane checking.

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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Plane option, and then click **Setup**.
3. Click “Check thermal connectivity only”.
4. Click **OK**.
5. 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.

Related Topics

Assigning Copper 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, the plane will be flooded before the check is run.

Prerequisites

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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Plane option, and then click **Setup**.
3. Click **Check clearance and connectivity**.

4. Click **OK**.

5. 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.

Procedure

1. Click the **Tools > Verify Design** menu item.

2. In the Verify Design Dialog Box, in the Check area, select the Plane option, and then click **Setup**.

3. Click Check clearance and connectivity.

4. Select the Same Layer Connectivity check box.

5. Click **OK**.

6. 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

Troubleshoot Design Verify 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.

Procedure

1. Click the **Tools > Verify Design** menu item.
2. In the Verify Design Dialog Box, in the Check area, select the Wire Bonds option, and then click **Setup**.
3. Select the check box beside any types of checks you want to enable and then click **OK**.

   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

Check the Plane Connection for Continuity

When you run a continuity check in Verify Design, 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 40
Dimensions

This section describes the many dimensioning methods available and defines how the dimensioning objects interact with the design elements. It also discusses dimension styles, snap modes and how to delete or modify dimensions.

Dimensioning

Dimensioning measures distances or angles on points you select, creates a text string containing the measured value, and creates the associated extension lines and arrows. This information is considered mechanical documentation for PCB designs.

The following table shows the setup options that let you match existing drafting standards and control how to add new dimensions.

Table 130. Dimensioning Set Up Options

Option	Description
Options	Establishes the appearance of newly added dimensions. For more information, see the Options Dialog Box, Dimensioning Category, General Subcategory.
Snap Mode	Controls how you select data items.
Edge Preference	Sets whether to measure lines from edge or centerline.

Dimensioning also lets you create Baseline dimensions for annotating multiple items from the same starting point. Use Continue to create daisy-chained dimensions.

Tip
You can use Dimensioning within the PCB Decal Editor. However, saving the decal converts dimensions to 2-D lines and text.

Dimensioning Modes

All of the various data items, routes, parts, and drafting items in SailWind Layout are referred to as objects. For dimensions, the extension lines, dimension lines, arrows, and text strings that make up dimensions are called dimension elements. Collectively, they create a dimension object.

Tip
When selecting and modifying, the software considers dimension lines as part of the arrows.

The following lists examples of how you can combine these separate elements to indicate dimensions:

- The standard, or default, dimension is comprised of lines extending from each point of measurement, a dimension line with arrows at each end running between the extension lines, and a text string identifying the length.
- Datum line dimensions are comprised of one extension line and one text string.
- Leader line dimensions consist of a dimension line with an arrow at one end and a text string at the other end. This category includes radius type dimensions.

You can combine dimension elements in various ways to create a dimension object. Many of the modification commands use these combinations, for example:

- You can select the entire dimension object from one of its selected elements using Select Parent.
- If you open the Properties of a selected dimension element, a Parent button appears in the subsequent dialog box. Click the **Parent** button to consider the entire dimension object for modifications.

Dimensioning Process

The Dimensioning Toolbar provides quick access to tools for dimensioning your design. The Dimensioning Toolbar is also available in the PCB Decal Editor.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button to display the Dimensioning Toolbar.
2. Set your preferences for dimensions on the Options dialog box > **Dimensioning** category.

   Give specific attention to the Layers where the dimensioning objects are added. For more information, see "Dimensioning Options on page 1520."

   CAUTION:
   Text and lines added to electrical layers will be fabricated in copper! Select a documentation layer for documentation objects.

3. Before attempting to dimension a design object, set the selection filter to the type of object for dimensioning.

For more information, see "The Selection Filter."

4. Based on the type of object to be dimensioned, select a dimensioning button.

   For more information, see "Creation of Dimensions."

5. If you are creating multiple linear dimensions, right-click and click one of the measurement styles.

   For more information, see "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 one of the Edge Preferences.

   For more information, see "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.

   For more information, see "Snap Mode for Dimension Points."

8. Select the object to dimension and then click to place the dimension.

9. When finished dimensioning, click the **Select** button to end dimensioning operations.

Tip
You can also use the View Clearance dialog box (**View > Clearance** menu item) to add dimensions to your design.

Related Topics

View Clearance Dialog Box

Creation of 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.

Note:
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** menu item to close the toolbar.

Adding Dimensions 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.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button, then on the Dimensioning Toolbar, click the **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.

Adding 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.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button, then on the Dimensioning Toolbar click the **Horizontal** button.
2. If needed, select a Snap Mode on page 925 and Edge Preference on page 923 popup menu item.
3. Select the line segment to dimension, or indicate two points and the dimension dynamically attaches to the pointer.
4. Click to indicate a location for the dimension line and then you can continue to dimension additional data items.

Adding 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.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button and then on the Dimensioning Toolbar click the **Vertical** button.
2. If needed, select a Snap Mode on page 925 and Edge Preference on page 923 popup menu item.
3. Select the line segment to dimension, or indicate two points and the dimension dynamically attaches to the pointer.
4. Click to indicate a location for the dimension line, and then you can dimension additional data items.

Adding 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.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button, and then on the Dimensioning Toolbar click the **Aligned** button.

2. If needed, select a Snap Mode on page 925 and Edge Preference on page 923 popup menu item.

3. Select the line segment to dimension, or indicate two points and the dimension dynamically attaches to the pointer.

4. Click to indicate a location for the dimension line and then you can dimension additional data items.

Adding 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.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button, and then on the Dimensioning Toolbar click the **Rotated** button.

2. If needed, select a Snap Mode on page 925 and Edge Preference on page 923 popup menu item.

3. Select the line segment to dimension, or indicate two points.

4. Type the rotation value you want to use and click **OK** and the dimension dynamically attaches to the pointer.

5. Click to indicate a location for the dimension line and then you can dimension additional data items.

Adding 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 on page 925.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button, and then on the Dimensioning Toolbar click the **Angular** button.

2. If needed, right-click and click a Snap Mode on page 925 popup menu item.

3. Select the lines to dimension, or indicate two points to define the angle and the angular dimension dynamically attaches to the pointer.

4. Click to indicate a location for the dimension line, and then you can dimension additional data items.

Adding 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.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button, and then on the Dimensioning Toolbar click the **Arc** button.

2. If needed, right-click and click an Edge Preference on page 923 popup menu item.

3. Select the arc or circle to dimension and the radial dimension dynamically attaches to the pointer.

4. Click to indicate a location for the dimension line, and then you can dimension additional data items.

Tip
Arc mode works best when you select the “Manual position” check box in the Displacement area of the Options dialog box > Dimensioning category > Text subcategory on page 1522. This lets you create a leader line effect for documented arcs and circles.

Adding 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.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button, and then on the Dimensioning Toolbar click the **Leader** button.

2. Right-click and click a Snap Mode on page 925 popup menu item 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 and a line with an orthogonal corner dynamically attaches to the pointer.

4. Click to add a vertex and anchor the leader line, and then 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

Dimensions can be created using a chained or baseline style. This section provides detailed descriptions of each method.

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.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button to display the Dimensioning Toolbar.
2. Click the appropriate dimensioning button on the Dimensioning Toolbar, such as **Horizontal**, **Vertical**, or **Aligned**.
3. Right-click and click the **Continue** popup menu item and a check mark appears next to **Continue**, indicating that it is active.
4. Click to indicate the location of the first dimension point and 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 and the check mark next to Continue is removed.

Creating Baseline Dimensions

Baseline dimensioning defines several measurements from the same starting point.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button to display the Dimensioning Toolbar.
2. Click the appropriate dimensioning button on the Dimensioning Toolbar, such as **Horizontal**, **Vertical**, or **Aligned**.
3. Right-click and click the **Baseline** popup menu item. 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 the **Baseline** popup menu item again. The check mark next to Baseline is removed.

Related Topics

Setting an Existing Extension Line as the Baseline

Setting an Existing Extension Line as the Baseline

You can set an existing extension line as the new baseline.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button, and then on the Dimensioning Toolbar, click the **Select** button.
2. Select the extension line to act as the new baseline.
3. Right-click and click the **Baseline** popup menu item and 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 the **Baseline** popup menu item again and the check mark next to Baseline is removed.

Related Topics

Creating Baseline Dimensions

Dimensioning Process

Options Dialog Box, Dimensioning Category, Alignment and Arrows Subcategory

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.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button, and on the Dimensioning Toolbar, click a dimensioning style button.
2. Right-click and click an Edge Preference popup menu item described in the table below.

Table 131. 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.

This mode remains in effect until you select another. Also use the shortcut menu to set the snap mode on page 925.

Tip
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.

A check mark appears next to the selected popup menu item to indicate that it is in use.

Related Topics

Dimensioning Process

Dimensioning

Snap Mode for Dimension 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.

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.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button, and on the Dimensioning Toolbar, click a dimensioning style button.
2. To disable auto-snap, right-click and click one of the snap mode popup menu items and a check mark appears next to the selected mode.

   This mode remains active until you select another one, or until you exit SailWind Layout.
3. To re-enable auto-snap, right-click and click the snap mode popup menu item to remove the check mark 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.

Procedure

1. On the Standard Toolbar, click the **Dimensioning Toolbar** button, and on the Dimensioning Toolbar, click a dimensioning style button.
2. If needed, right-click and click a snap mode popup menu item.

   The following table lists the snap modes and their descriptions.

Table 132. 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.
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.

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 another snap mode and then select the second dimensioning point.
5. If you selected the Angular dimensioning style, repeat steps 4-5 as needed to select the other side of the angle.

Related Topics

Dimensioning Process

Selecting the Parent Dimensioning Object

You can select the parent dimension object from a selected dimension element.

Procedure

1. Select one of the items in the dimension object.
2. Right-click and click the **Select Parent** popup menu item. The entire dimension is selected.

Related Topics

Dimensioning

Moving Dimensions and Dimension Objects

You can move Dimensions or Dimension Objects using the Move command or by dynamically dragging objects.

Moving an Entire Dimension

When required, you can move an entire dimension. This is useful when you need to create additional space for another dimension or callout.

Procedure

1. Select the dimension object through multiple selection, or by using the Select Parent on page 926 button in the Properties dialog box.

   Note:
   To move dimension objects using keyboard entered coordinates, use the Dimension Object Properties dialog box on page 1311.

2. Right-click and click the **Move** popup menu item and 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.

Moving a Dimension Object

You can move individual dimension objects, including: arrows (with their lines), text strings, extension lines, and leader segments.

Procedure

1. Select a dimension object.
2. Right-click and click the **Move** popup menu item.
3. Click to indicate the new location for the dimension object.

The new position of the text and the location of the arrows are determined by the current settings in the Options dialog box > **Dimensioning** categories.

Tip
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 on page 927 command. This command works in the same manner as moving arrows, but works with selected extension lines.

Moving Text to Its Default Location

If you have repositioned a text object, you can move the text to its default location.

Procedure

1. Click the dimensioning text to select it.

   If you cannot select the text, check the selection filter settings.

2. Right-click and click the **Default Location** popup menu item.

Tip
To add a new segment to an existing leader, right-click and click the **Add Corner** and **Split** popup menu items.

Dynamically Drag Objects

Objects can be dynamically dragged to reposition them. This allows you to quickly and intuitively reposition objects.

Prerequisites

You must set the “Drag moves” setting to “Drag and drop” in **Tools > Options** menu item, **Global** category > **General** subcatory.

Procedure

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.

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.

Procedure

1. Select the extension line for which you want to change the length.
2. Right-click and click the **Change Length** popup menu item and 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

When they are no longer needed, you can delete a dimension element.

Procedure

1. Select the dimension element to delete using one of the following methods:
   - Select one of the dimension's elements and right-click and click the “Select Parent” on page 926 popup menu item.
   - Use multiple selection on page 115 to sequentially select each element belonging to the dimension.
   - Select the entire dimension with a selection rectangle.
2. Press the Delete key.

Resetting Dimension Measurements

If you need to edit a dimension, you can reset the measurement text.

Procedure

1. Click the dimensioning text to select it.

   If you cannot select the text, check the selection filter settings.
2. Right-click and click the **Reset Measurement** popup menu item.
3. Type the new measurement in the Text Value dialog box and click **OK**.

Chapter 41
CAM Output

The following topics describe setup, preview and output for a printer, plotter, photoplotter, NC drill device and output files that are compatible with a variety of automatic assembly and pick-and-place machines.

You use the CAM command on the File menu to produce laser copy, plots, Gerber files, drill drawings, and other manufacturing outputs.

CAM Documents

CAM is an acronym for Computer-Aided Manufacturing. Using the CAM tools, you can produce not only Gerber-formatted photoplot or manufacturing outputs, but printouts and plots as well.

A plot type, drill, silkscreen, or routing, together with an associated output device setup is called a CAM document.

There is no default set of output configurations 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.

You can use Import and Export to move a CAM documents list, including the aperture list and drill feed and speed table, between *.pcb* files.

Note:
You can create a maximum of 250 CAM Document configurations.

Associated Copper and CAM

CAM interprets pads and other objects differently when they are associated with copper in the PCB Decal Editor.

For more information, see Associating Copper with Terminals. Associating copper shapes and open copper is one way to create hard breakouts in decals.

- Terminals are interpreted as vias.
- Closed copper shapes are interpreted as pads.
- Open copper (a path drawn with copper) is interpreted as a trace.

RS-274-X Format

The RS-274-X format is based on the Gerber Format Guide (Document Number: 0000-00-RM-000, Part Number: 414-100-002) by Gerber Systems Corporation. RS-274-X in SailWind Layout is a data format created in CAM for photoplotters.

Clicking the RS-274-X output format from the Photo Plotter Advanced Setup Dialog Box creates a Gerber file with the information shown in Table 133.

Table 133. RS-274-X Fields

Field	Description
AM	Aperture macro
AD	Aperture description
FS	Format statement
MO	Units mode
IN	Image name
LN	Layer name
LP	Layer polarity
G36,G37	Fill area controls

These features are allowed for the 9500, 9800, 9900, GPC, and Insight/2020 photoplotters that also support the G74, G75 multiquadrant circular interpolation function codes. Some of these parameters appear in the Gerber file depending on photo plotter settings; however, the parameters in Table 134 are always present in the Gerber file.

Table 134. Gerber File Fields

Field	Description
The name of the design	%IN job name *% where job name is the name of the design.
The mode parameter	Indicates the units, for example, %MOIN*% or %MOMM*%.
A format statement parameter	Describes the format options selected in the Photo Plotter Advanced Setup Dialog Box, for example %FSLAX45Y45*%.

Aperture Table

All simple apertures; such as round, oval, or rectangular; are described in the output Gerber file as a %ADDxx*% parameter. The Photoplotter Aperture Report file is also created.

Aperture Macro

Copper areas associated with pins are output as unique aperture flashes.

Aperture support is as follows:

- Line-shaped polygonal copper with 50 vertexes or less and circular copper are supported.
- Line-shaped polygonal solid copper with 50 vertexes or less and circular solid copper are supported.
- Arc-shaped copper outlines are aperture macros in Gerber using polygonal approximation with up to 50 vertices. The %AM*% parameter limits the number of vertices to 50.
- Thermal reliefs for CAM planes are output as macro flash.

SailWind Layout does not support copper cutouts for coppers associated with component pins.

SailWind Layout does not support hatched coppers associated with component pins. All associated coppers appear and output in CAM as solid, regardless of the grid spacing and line width.

Fill Area

Solid, not hatched, copper and copper planes are output in fill area mode (G36,G37 brackets). Hatched areas are output in vector format. Circular copper areas, arc-shaped polygons, and circular and polygonal copper cutouts are all supported.

Verify Photoplots

Verify Photoplots Document Type supports the macros, aperture selection, and fill area commands of SailWind Layout Gerber outputs. Verify Photoplots can only process RS-274-X Gerber files created by SailWind Layout.

Creating CAM Outputs to Manufacture Your PCB

There are 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.

Procedure

1. Generate the outputs for the top layer:
    a. Create the Conductive Elements gerber-format file for the photo plotter. Choose between the following layer types:
        - Routing/Split Plane (gerber-format file) on page 948
        - CAM Plane (gerber-format file) on page 951

    b. Create the Silkscreen gerber-format file for the photo plotter (gerber-format file) on page 938
    c. Create the Solder mask gerber-format file for the photo plotter (gerber-format file) on page 941
    d. Create the Paste mask gerber-format file for the photo plotter (gerber-format file) on page 943
    e. Create the Assembly Drawing (non-gerber) on page 946
2. Generate the photo plotter Conductive Elements gerber-format file outputs for each internal layer. Choose between the following layer types:
    - Routing/Split Plane (gerber-format file) on page 948
    - CAM Plane (gerber-format file) on page 951
3. Generate the outputs for the bottom layer:
    a. Create the Conductive Elements gerber-format file for the photo plotter. Choose between the following layer types:
        - Routing/Split Plane (gerber-format file) on page 948
        - CAM Plane (gerber-format file) on page 951

    b. Create the Silkscreen gerber-format file for the photo plotter (gerber-format file) on page 938
    c. Create the Solder mask gerber-format file for a photo plotter (gerber-format file) on page 941
    d. Create the Paste mask gerber-format file for a photo plotter (gerber-format file) on page 943
    e. Create the Assembly Drawing (non-gerber) on page 946
4. Generate the outputs for each drill pair:

a. Create the Drill Drawing with Drill Table (non-gerber) on page 954

b. Create the NC Drill file for the drilling machine (drill file) on page 953

Related Topics

Verifying a Gerber File

CAM Output Creation

The following output types can be created.

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.

Tip
If you use the TrueLayer option, see "TrueLayer Associations" for information on how TrueLayer affects layers and CAM documents.

Procedure

1. Click the **File > CAM** menu item.
2. In the Define CAM Documents Dialog Box, click **Add**.
3. In the Add Document dialog box on page 1044, type a Document Name.

   It is beneficial to add a description of your custom output in the document name. 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 in a later step.
4. In the Document Type list, click Custom.

   When you select a specific document type, the program automatically generates a pre-configured 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. 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.
5. In the Output File box, type a name for the file you are creating.

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 and then 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**.

    If you close the .pcb 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:\SailWind Projects\Cam* directory (or a subdirectory if you created one) if you are not sending the output to your printer.

Creating a Set of CAM Documents Using Auto Define

You can quickly generate a set of CAM documents for your design using Auto Define. This feature uses a predefined group of default parameters to generate a “starter” set of CAM documents that you can further refine to meet your specific design requirements.

After using Auto Define to create your baseline CAM documents, you can edit each individual CAM document configuration to change the filename and modify the design object selections to meet your manufacturing output requirements. After the edits are complete, you can save and export the configurations for use in other designs.

Restrictions and Limitations

- Auto Define only creates photoplot (*.pho*) files. You must manually create any additional types of CAM documents required by your design flow for printing or pen plotting.
- Auto Define uses a generic naming convention to identify each of the different CAM document file types (solder mask, paste mask, routing/plane layers, assembly drawing, drill file, and others). You can rename the files to reflect your specific naming preferences or corporate standards.
- Auto Define analyzes the design and creates separate drill files for plated-through holes, non-plated holes, and individual partial via drill pairs (if present in the design).
- Auto Define attempts to create a complete set of CAM documents based upon your specific design setup. It does not overwrite any existing CAM document configurations that you may have already created for the design. Auto Define detects and preserves any existing CAM document configurations and only appends the remaining missing files to the design.
- You cannot modify the default parameters used to auto-generate the CAM Document configurations; however, once created, you can edit and save your modified CAM document configurations, export them, and use them in other designs.

Procedure

1. Click the **File > CAM** menu item.
2. In the Define CAM Documents dialog box, click the **Auto Define** button.

Results

The newly created CAM document configurations will appear in the CAM Documents area.

Note:
Auto Define always attempts to generate a full set of CAM document configurations based upon your specific design setup. If you want to use Auto Define to create a subset of the CAM document configurations, run Auto Define to create the full set of CAM document configurations and then manually delete any unwanted CAM document configurations.

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. Click the **File > CAM** menu item.
2. In the Define CAM Documents Dialog Box, click **Add**.
3. In the Add Document dialog box on page 1044, 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:
   - 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 is 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 during testing or troubleshooting. If you have 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 is 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 have chosen to justify the layers by the Top Left, and you have 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 is possible that your silkscreen layer will not be aligned to the other layers you have 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** and in the Photo Plotter Setup Dialog Box, make any necessary changes to the settings.
15. Click the **Advanced** button and 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:\SailWind Projects\Cam* directory or a subdirectory if you created one.

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 search-able pattern.

You can use two sets of reference designators, one set on the silkscreen layers for the silkscreen artwork positioned outside the components where they will not 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 have not built this second assembly drawing set of reference designators into your library decals, you can quickly add them to your design. For more information, see "Generating a Second Set of Reference Designators for Assembly Drawings".

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Changing the Reference Designators of Multiple Components in ECO Mode (Autorenumbering)

Creating a Set of CAM Documents Using Auto Define

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.

Note:
Virtual pins in the design are output as vias.

Procedure

1. Click the **File > CAM** menu item.
2. In the Define CAM Documents Dialog Box, click **Add**.
3. In the Add Document dialog box on page 1044, 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 theLayers Setup Dialog Box.

   ---

5. In the Output File box, type a name for the file you are creating.

   Tip
   An auto-generated 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:

- You need solder mask openings for each pad of each component and (optionally) each via.
  - If you have 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 have not 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 will 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 is 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 have chosen to justify the layers by the Top Left, and you have 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 is possible that your silkscreen layer will not be aligned to the other layers you have 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 are using the pad shapes of an outer layer to create the solder mask openings of the solder mask layer, you will 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:\SailWind Projects\Cam* directory or a subdirectory if you created one.

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 CAM Outputs to Manufacture Your PCB

Control of Solder Mask and Paste Mask

Creating a Set of CAM Documents Using Auto Define

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. Click the **File > CAM** menu item.
2. In the Define CAM Documents Dialog Box, click **Add**.
3. In the Add Document dialog box on page 1044, 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 theLayers Setup Dialog Box.

---

5. In the Output File box, type a name for the file you are creating.

Tip
An auto-generated 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:

   - You need paste mask openings for each pad of each surface mount component.

     - If you have 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 have not 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 is 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 have chosen to justify the layers by the Top Left, and you have 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 have 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 do not want to add paste mask openings over its pads. If you are using the pad shapes of an outer layer to create the paste mask openings of the paste mask layer, you will 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** and in the Photo Plotter Setup Dialog Box, make any necessary changes to the settings.
16. Click the **Advanced** button and 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:\SailWind Projects\Cam* directory or a subdirectory if you created one.

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 stencil 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 CAM Outputs to Manufacture Your PCB

Control of Solder Mask and Paste Mask

Creating a Set of CAM Documents Using Auto Define

Creating an Assembly Drawing

You create the assembly drawing for your manufacturer to reference when assembling the components on the board.

Procedure

1. Click the **File > CAM** menu item.
2. In the Define CAM Documents Dialog Box, click **Add**.
3. In the Add Document dialog box on page 1044, 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.

   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.

   An auto-generated 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. If you need to create an assembly drawing of an assembly variant, do the following:

a. In the Customize Document area, click **Assembly**.

b. 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 on page 1853.

c. In the Name list, select a variant and click **OK**.

7. 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, plotter or PDF file, and not output them to a gerber file. 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.

8. In the Customize Document area, click the **Layers** button.

9. In the Select Items Dialog Box, choose which layers and layer items should appear in the paste mask data:

   - 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 have 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.

10. Click **OK** to accept the changes and close the Select Items dialog box.

11. In the Customize Document area, click **Options**.

12. In the Plot Options Dialog Box, in the Positioning area, set the positioning options.

13. In the Suppress area, you can suppress the display of some Reference Designators.

    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 are not a true component and you do not need the assemblers to think it needs to be populated with a component.

14. For the assembly bottom drawing, you may want to select the Mirror Image check box to mirror the image.

15. Click **OK** to close the Plot Options dialog box.
16. 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.

17. Repeat these steps again to create the Assembly drawing for the opposite side of the board (if applicable).
18. 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.
19. Select the document(s) you want to output in the Document name list, and then click **Run**.
20. Click **Save**.

    If you close the *.pcb* 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:\SailWind Projects\Cam* directory (or a subdirectory if you created one) if you are not sending the output to your printer.

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 will not be hidden after the component placement. If you have not built this second assembly drawing set of reference designators into your library decals, you can quickly add them to your design. For more information, see "Generating a Second Set of Reference Designators for Assembly Drawings" on page 808.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Changing the Reference Designators of Multiple Components in ECO Mode (Autorenumbering)

Creating a Set of CAM Documents Using Auto Define

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.

Prerequisites

- Before you actually create the gerber-format file for this conductive layer, you must ensure that all copper planes are flooded and filled with copper.
- Virtual pins in the design are output as vias.

Procedure

1. Click the **File > CAM** menu item.
2. In the Define CAM Documents Dialog Box, click **Add**.
3. In the Add Document dialog box on page 1044, type a Document Name.

   It is 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.

   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:

   - 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 is important that each gerber-format file that you output for your design lines 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 have chosen to justify the layers by the Top Left, and you have 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 is possible that your silkscreen layer will not be aligned to the other layers you have 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** and in the Photo Plotter Setup Dialog Box, make any necessary changes to the settings.
14. Click the **Advanced** button and 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**.

CAUTION:
If your design has copper planes 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**. If you close the *.pcb* 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:\SailWind Projects\Cam* directory or a subdirectory if you created one.

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 planes. If this is a component layer, you will also see all the component pads and associated pin copper if it exists. The white areas are areas of no copper.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Creating a Set of CAM Documents Using Auto Define

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.

Note:
Virtual pins in the design are output as vias.

Procedure

1. Click the **File > CAM** menu item.
2. In the Define CAM Documents Dialog Box, click **Add**.
3. In the Add Document dialog box on page 1044, type a Document Name.

   It is 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.

   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:

   - 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 cutout 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 (negative) 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 is 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 have chosen to justify the layers by the Top Left, and you have 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 is possible that your silkscreen layer will not be aligned to the other layers you have 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** and in the Photo Plotter Setup Dialog Box, make any necessary changes to the settings.
15. Click the **Advanced** button and in the Photo Plotter Advanced Setup Dialog Box, make any necessary changes to the settings. 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**. If you close the *.pcb* 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:\SailWind Projects\Cam* directory or a subdirectory if you created one.

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 have used them.

CAM plane thermals are displayed in the CAM Preview dialog box (and in the Layout editor). The following calculations are also used for resulting output during printing, pen plotting, and RS-274-X photoplotting operations:

- The outer diameter of the thermal matches the width of the aperture set in the Photo Plotter Setup Dialog Box.
- The inner diameter is 75% of the outer diameter.
- The number of spokes is always four, arranged diagonally.
- The spoke width is 1/6 of the outer diameter.

Tip
The inner width for custom CAM plane thermals is set as the pad size defined in the Pad Stacks Properties dialog box. The outer width is set as the default same-net pad to corner rule.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Creating a Set of CAM Documents Using Auto Define

Creating an NC Drill File

You can create an NC drill file for each drill pair in your design. This allows the board fabricator to use separate drilling operations for specific layer pairs during the board fabrication process.

Procedure

1. Click the **File > CAM** menu item.
2. In the Define CAM Documents Dialog Box, click **Add**.
3. In the Add Document dialog box on page 1044, type a Document Name.

   It is 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.

   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
You can click **Run** and create the output file from the Add Document dialog box; however, it is better instead to add this configuration to the Define CAM Documents dialog box by clicking **OK** in the event 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**. If you close the *.pcb* 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:\SailWind Projects\Cam* directory or a subdirectory if you created one.

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. To examine the drill symbols and their locations, view the Drill Drawing on page 954 instead.

Related Topics

Creating a Set of CAM Documents Using Auto Define

Creating a Drill Drawing with Drill Table

You can create a drill drawing for one or more drill pairs.

Restrictions and Limitations

The drill chart is only output in the drill drawing CAM document. It is not embedded in the design.

Procedure

1. Choose the **File > CAM** menu item.
2. In the Define CAM Documents dialog box, click **Drill Symbols** to open the "Global Drill Symbols dialog box" on page 1387 to define your drill symbols.
3. If using partial vias, select the **Unique Through/Partial column** check box to allow filtering of the drill holes.

   Selecting this check box displays the Through/Partial column in the table to differentiate partial from through-hole drills.
4. If necessary, click **Regenerate** to update the table data.

   Populate data in the drill drawing table by manually adding entries or by gathering information from the design database by clicking **Augment** or **Regenerate**. If you have not saved any new default settings and if no data previously existed in the table, clicking **Augment** or **Regenerate** populates the table in ascending order of size and then plated holes before non-plated holes.
5. Customize the settings of the Drill data as needed.

   - Customize the text size.
   - Modify the symbols used and Tolerance. For more information, see "Modifying Drill Table Entries" on page 963.
   - Sort the data in the drill drawing table by clicking the column header of drill size, quantity of the holes, or plating type. For more information, see "Sorting the Data in the Drill Drawing Table" on page 962.
6. Click **OK** to close the Global Drill Symbols dialog box.
7. In the Define CAM Documents dialog box, click **Add**.
8. In the Add Document dialog box on page 1044, type a name in the Document Name box.

   It is 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.
9. From the Document Type dropdown list, select "Drill Drawing" and then, in the "Layer Association dialog box" on page 1437 that appears, choose one of the layers in your drill pair and click **OK**.

   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.
10. In the Output File box, type a name for the file you are creating.

    (An auto-generated name appears in the Output File box but you can overwrite it.) 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.
11. In the Customize Document area, click the **Layers** button.

12. In the "Select Items dialog box" on page 954, choose which layers to include, and check for these conditions:

    CAUTION:
    The objects that require drilling must be added to the drill drawing in order for the drill chart to be created. The chart only displays the drill data of items that are added to the drill drawing in the Select Items dialog box on page 1699. For example, if no pads or vias are added from the electrical layers, the chart will not display drill information for those locations.

    - If you have added dimensions to the Drill Drawing layer, you must enable Lines and Text.
    - For designs that only use through holes, you only need to include one electrical layer. For designs that use partial vias, you 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 see what will be included in the output.

13. Click **OK** to accept the changes and close the Select Items dialog box.
14. In the Customize Document area, click **Options**.
15. In the "Plot Options dialog box" on page 1641, in the Positioning area, set the positioning options.
16. In the Drill Chart area, type new coordinates for the Location of the chart if necessary.

    In the Preview window, notice the position of the magenta-colored rectangle with 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.

17. Click the **Drill Symbols** button and in the "Drill Symbols dialog box" on page 1334, select the "Use" check box for each drill symbol that you want to include in the drawing; then click **OK**.
18. Check your location of the Drill Chart in the Preview window and if needed, edit the location of the drill chart.
19. Click **OK** to close the Plot Options dialog box.
20. In the Output Device area, choose the device and update the device setup as needed.
21. Click **OK** to add the new file configuration to the "Define CAM Documents dialog box" on page 1261.

    Tip
    Although you can click **Run** and create the output file from the Add Document dialog box, the recommended course is to click **OK** to add this configuration to the Define CAM Documents dialog box in the event you need to generate the file again.

22. 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.

23. Select the document(s) you want to output in the Document name list, and then click **Run**.

24. Click **Save**. 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:\SailWind Projects\Cam* directory (or a subdirectory if you created one) if you are not sending the output to your printer. In addition, the Drill Drawing preview displays drill locations with markers and a drill chart listing all the drill sizes, based on your assignments in the Global Drill Symbols dialog box and the set design unit options (mils, metric, or inches). Dimensioning measurements for the board may also display on the drawing.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Creating a Set of CAM Documents Using Auto Define

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. Click the **File > CAM** menu item.
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**. 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 All Outputs

You can run multiple CAM document configurations against your current design.

Procedure

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**.

Results

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.

> **Note:**
> CAM interprets pads and other objects differently when they are associated with copper in the PCB Decal Editor. For more information, see the Pins with Associated Copper area of the “Select Items Dialog Box”.

> **Tip**
> 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.

> **Note:**
> Virtual pins in the design are output as vias.

Creating Reusable Fabrication Notes

You can save your fabrications notes to the library for reuse.

Procedure

1. Create the individual line items of text.

   For more information, see “Adding Free Text”.

2. Since text alone cannot be saved to the library, add an underline to the heading of your fab notes using a 2D Line.

For more information, see “Creating a Drafting Object”.

3. Combine all the text with the line item.

   For more information, see “Combining Line and Text Objects”.

4. Save your notes to a library.

   For more information, see “Saving a Drafting Item to a Library”.

Reporting Apertures of a Photo-Plot File(s)

You can produce a report of the apertures used in a CAM Document.

Prerequisites

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.

Procedure

1. Click the **File > CAM** menu item.
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**.

Results

The aperture report opens in the default text editor.

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.

For more information, see “Software Launch Options”.

Colors in CAM Documents

You can apply colors to CAM document objects if you are outputting to a printer or a pen plotter.

Restriction:
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. For monochrome output devices (printers/plotters), the only color available is black.

Colors of Objects

You can assign a color to the object types for printing.

- In the Select Items Dialog Box, in the Selected Color area, click a color in the palette and click the box beside a design object. You must select the check box of a design object before the object color swatch appears.

Colors of Individual Nets

You can use the colors you have assigned to nets in the View Nets dialog box when you print.

- In the Select Items Dialog Box, select the Color By Net check box to use the View Nets colors in the output.

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 the **Attribute** popup menu item.
2. In the “Attributes for” list, click 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.

Results

- 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.

Drill Drawing Options

Use the Drill Drawing Options dialog box to set drill drawing legend and marker parameters.

Sorting the Data in the Drill Drawing 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.

Procedure

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.

Results

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.

Examples

The sorting results are formatted to correspond with the filters you have chosen.

Table 135 shows the result of a sort by size (ascending order) and then by plated type.

Table 135. 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

Table 135. Example Sort Result (continued)

Symbol	Size	Quantity	Plated	Tolerance
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

Modifying Drill Table Entries

You can modify the drill table entries shown in the Drill Data area.

Restrictions and Limitations

- Symbol usage is exclusive - you cannot save your changes if you create duplicate symbol assignments.
- You cannot edit data in the Size, Quantity, or Plated columns.

Procedure

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.

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.

Assembly Variants

Assembly Variants allow you to create multiple versions of your assembly containing component substitutions and/or omissions. This allows one base design to support multiple product configurations at assembly time.

Substitute a Component for Assembly Variants
Creation of 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

You can use Assembly Variants to substitute components in your assemblies.

Procedure

1. Click the **Tools > Assembly Variants** menu item.
2. Choose any of the following 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.
   - Select a component in the multicolumn list, and click **Substitute** in the Status area.
   - Click **Substitute** from the Verb Mode list, and select a component in the Layout Editor.
3. In the Variant/Substitute dialog box, in the multicolumn list, double-click a Part Type in the Active column.

   The multicolumn list displays the attributes (Value) of the component you are substituting, its Default on page 1820 value, and its Active on page 1808 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.
7. 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.

Component Status Interpretation

The Status area explains 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.

Procedure

Select the Show Difference check box and view the updated list with only the differences.

Previewing a Variant

You can preview your substitutions.

Procedure

Click the **Preview** button and view the substitutions.

For more information, see “Previewing Assembly Variants”.

Creation of Assembly Variants

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.

Creating Assembly Variants

You can create Assembly Variants to represent different variations of your final assembly.

Procedure

1. Click the **Tools > Assembly Variants** menu item.
2. In the New variant's name box, type the name of the new variant, up to 26 characters,
3. Click **Create** and a new variant is created that contains all the items in the Base Option on page .
4. Use the **Display** list to filter the items to view in the multicolumn list.
5. Uninstall, or substitute on page 968 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

The following topics describe two different methods you can use to install, uninstall or substitute variant design components.

Using the Multicolumn List

You can modify the component status using the multicolumn list and the Status area.

Restrictions and Limitations

Since you cannot directly modify the Status of the Base Option on page which is based on the other variants, modifying the Status is unavailable when you view the Base Option in the multicolumn list.

Procedure

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 on page 964 appears for each item.

4. Make any substitutions you want and click **OK** to 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).

Procedure

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 on page 964 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.

Restriction:
You cannot modify the BaseOption on page .

Modifying Assembly Variants

By modifying an Assembly Variant, you can, for example, choose to uninstall or substitute a component that is Installed in the variant you click in the Name list.

Procedure

1. Click the **Tools > Assembly Variants** menu item.
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 on page 964 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 on page 1820.

   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**.

Modifying Assembly Variants by Component

You can modify 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.

Procedure

1. Click the **Tools > Assembly Variants** menu item.
2. In the Type list, click Components.
3. In the Name list, click the component whose status you want to view.
4. In the Display list, click **All** 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 on page 964 appears.

   If you change the status of a component from Substituted to Installed, the variant uses the Default component on page 1820.

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

You can delete existing assembly variants in the Assembly Variants dialog box.

Procedure

1. Click the **Tools > Assembly Variants** menu item.
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 choose to display, to not display, or to display components in a different color.

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.

Procedure

1. Click the **Tools > Assembly Variants** menu item.
2. Select the variant to preview in the **Name** list.
3. Click **Preview** and 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** to 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 using a feature of the Reports dialog box.

Procedure

Verify your assembly variant selections. and click the **Report** button.

For more information, see “Running a Report Using an Assembly Variant”.

Creation of Assembly Variant Assembly Drawings

You can create assembly drawings from assembly variants.

For more information, see “Creating an Assembly Drawing” on page 946.

CAM Preview

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.

You can also use the CAM Preview dialog box after you use Verify the Design to perform fabrication checking and you want to preview the CAM document associated with the layer for a selected error.

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. You must have at least one CAM document defined to use preview setup.

For more information, see "CAM Preview Dialog Box" on page 1150 and "CAM Preview Setup Dialog Box" on page 1152

Printing

The following topics describe how you can choose to print your CAM documents directly to a Windows printer, or to a PostScript file that you can then review or send to your fabricator.

Printing to a Windows Printer

One of the output choices for CAM documents is your printer.

Tip
Test the link between Windows and the printer by printing this help topic.

Procedure

1. Click the **File > CAM** menu item.
2. In the Define CAM Documents dialog box, select a document and click Edit.
3. In the Edit Document dialog box on page 1044, under Output Device, click the **Print** button.

   This indicates the output device. All printing is controlled using the Windows printer properties.
4. Click **Device Setup**.
5. In the Print Setup dialog box set printer and document properties and click **OK**.
6. Under Document Name, type a meaningful name for the document and click **OK**.

   Note:
   A meaningful name is one that indicates the document type, the type of output, and the layer.

   Your document is added to the CAM Documents list in the Define CAM Documents dialog box.
7. Click the **Layers** button to display the Select Items dialog box.
8. Set the layers to output information on and click **Add**.
9. Establish any color changes:
   a. 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.
   b. Continue to assign/redefine the colors by clicking the color and the check box of the objects.
10. When you finish setting options, click **OK** to return to the Edit Document dialog box.

11. To print, click **Run**. The message “Do you wish to generate the following outputs?” appears.
12. Click **Yes** to print.

Related Topics

Print PostScript to a File

Print PostScript to a File

SailWind 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.

You must set up your printer to print to a file before you create a PostScript file.

Setting the Printer to Print to a File

Configure your system to print to a file instead of directly to your printer.

Procedure

1. Locate printer information based on your platform:

    - Using your Windows Start menu, locate Printers and Faxes using the Windows Control Panel.

    - If you are having difficulty locating the printer setup controls, consult your Windows documentation.

2. Right-click the PostScript printer you want to use to print to a file and then click the **Properties** popup menu item.

3. In the Properties dialog box, 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

After setting up your system to print to a file, you can create your CAM documents as print files.

Procedure

1. Complete the steps above to set up the printer to print to a file.

2. Click the **File > CAM** menu item. The Define CAM Documents dialog box on page 931 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 *\SailWind Projects\Cam \default*.

4. Click **Add** in the CAM Documents area. The appears.

5. In the Add Document dialog box on page 1044 Document Type list 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.

9. In the Print Setup dialog box choose a printer and set printer properties and then click **OK**.

   For more information on this dialog box, see the Microsoft Windows Help.

10. 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. For more information, see “Printing”.

    Tip
    For best results, make sure that you have a design loaded before generating the output file.

11. In the Define CAM Documents dialog box, select your document and click **Run**.

    The message “Do you wish to generate the following outputs?” appears.

12. 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).

13. Use the dialog box to specify a name (and, optionally, a path) for the output file. The default path is *\My Documents\SailWind Projects\Cam\default*, but you may have changed this in step 3 above.

14. Click **OK**. The printer file is created.

DFM Analysis of CAM Documents

Once you create your CAM documents, you can analyze them using DFM Analysis.

Setting up DFM Analysis

If you have not previously set up your DFM Preferences, you can use Setup to specify the options you want to use while analyzing your CAM documents.

Procedure

1. Click the **Tools > DFM Analysis > Setup** menu item.
2. Select the Preferences you want and then run the analysis.

> **Tip**
> To view help on DFM Analysis, press the F1 key with the dialog box open.

Related Topics

DFM Analyses User Guide

Running the DFM Analysis

If you have already created your DFM Preferences, you can proceed directly to running you analysis.

Procedure

1. Click the **Tools > DFM Analysis > Start Analysis**.
2. The analysis will run and generate a report.

> **Tip**
> To view help on DFM Analysis, press the F1 key with the dialog box open.

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. Supported formats include Dynapert, Siemens, Universal, and Quad.

Batch Mode and Masked Mode

CAM Plus uses the *part.def* file differently depending on whether you click Masked in the Parts list in conjunction with Batch Part Definition File.

- With Masked selected and the Batch Part Def File check box selected, CAM Plus sorts parts to all machines as defined in the *part.def*file, ignoring non-defined parts, Part D.
- With Masked selected, the Batch Part Def File check box cleared, and a specific machine selected, CAM Plus sorts only parts defined in *part.def* as assigned to that machine.
- With something other than Masked selected and the Batch Part Def File check box selected, CAM Plus sorts parts assigned in **part.def** to their respective machines, but includes undefined parts in files for all machines.
- With something other than Masked selected, the Batch Part Def File check box cleared, and a specific machine selected, CAM Plus sends all parts to the selected machine.

Running CAM Plus

You can set up CAM Plus to generate machine-specific output files.

Prerequisites

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. This may require assistance from manufacturing engineering to determine which parts to insert by which machines, and to set standard bin assignments. Also, place all design components at their final locations before you start CAM Plus. By default, the part definition file is read from the *\Libraries* folder. For more information, see Part Definition File Format.

Procedure

1. Click the **File > CAM Plus** menu item.
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 SailWind 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 *\SailWind 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 SailWind 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 *6241bb.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 the table 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.

Table 136. 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.

   Note:
   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. CAM Plus produces and prints error messages in the file *padscim.err*.

CAM Plus Report File Names

The file names created by CAM Plus for various insertion machines are formatted with a prefix name of the machine and a suffix of the components selected.

The prefix for each machine type is shown under the heading for each machine in the CAM Plus Supported Machine Formats. The last two characters in the output file name (shown as xx) are determined by the components selected.

Table 137 shows the first letter in the pair of ending characters.

Table 137. Output File Name Ending Characters (First Letter)

Letter	Description
M	If the output selected is mask
S	If surface mount is selected

Table 137. Output File Name Ending Characters (First Letter) (continued)

Letter	Description
T	If through hole is selected
A	If all parts are selected

Table 138 shows the second letter is the pair of ending characters.

Table 138. Output File Name Ending Characters (Second Letter)

Letter	Description
T	If components on the top layer are selected
B	If components on the bottom layer are selected

CAM350

CAM350 is a pre-production CAM system that combines DFM analysis, DRC checking, test fixturing, planning, and tooling. The CAM350 products range from Gerber viewers to full-featured CAM editors that process PCB databases into usable fabrication and panel data.

CAM350 lets you analyze your design for manufacturing issues prior to fabrication, drastically reducing cycle time and cost. The tools are based on a fully-intelligent CAD database and can input and output virtually any Gerber format, IPC-D-350/IPC-D-356 data, CAD database, or netlist.

CAM350 Link

CAM350 Link is a SailWind Layout option that automatically translates a design database into a CAM350 (version 6.0 and greater) database. You no longer need to generate an ASCII file for translation; CAM350 Link uses the native SailWind Layout file format and adds a *.cam* extension. In addition to converting designs for translation, you can launch CAM350 and load the current database directly from the CAM350 dialog box.

CAM350 Link also supports backward annotation of Design For Manufacture (DFM) errors to a SailWind Layout database so that you can identify and correct DFM errors in SailWind Layout instead of making the corrections in CAM350.

See the “Creating CAM Outputs to Manufacture Your PCB” topic for information on outputting CAM documents from SailWind Layout. See the DFF Audit Process Flow using CAM350 Link topic for information on CAM350 Link procedures.

CAM350 Link Non-Supported Objects

Some design objects are not passed from SailWind Layout to CAM350 during CAM350 Link translation when translating a CAM document with nets and when translating CAD layers to CAM layers.

These specific design objects are:

- Associated pin copper is merged into a single layer custom aperture. Associated pin copper is always merged to the pad layer during translation to CAM350. The result is that associated copper is not handled on layers other than the pad layer. In addition, associated copper is not distinguished from pad flashes.

- Combined text is not output with owning lines in CAM documents. When combined text and lines are translated to CAM350, combined text visibility is controlled by text visibility in the CAM document itself, not by the line visibility in the CAM document settings.

- The pad oversize option in the CAM documents is not supported when translating to CAM350.

- Hatched copper is not supported for PADS-format ASCII import. Hatched copper is converted to solid fill copper rather than hatched copper when translating to CAM350.

- Decal lines and text on the Top or Bottom electrical layers are not distinguished from lines and text for Top or Bottom silkscreen layers. Decal lines and text from the Top and Bottom electrical layers are moved to silkscreen layers during CAM350 translation. Visibility control for Top and Bottom lines and text can not be distinguished from visibility control for silkscreen lines and text.

- Jumper reference designators and outlines are moved to silkscreen layer. Jumper reference designators are converted to free text and jumper outlines are converted to lines in CAM350. Jumper reference designators and outlines are also moved to the silkscreen layer during translation.

The CAM350 Analysis feature reports jumpers as net connectivity errors. CAM350 does not support CAM negative planes on outer mounting layers. CAM350 does not support a layer type of negative plane for outer mounting layers.

Test Points in CAM350

Test points in CAM350 are used for Bed of Nails testing or Flying Probe in-circuit testing. Test points in SailWind Layout are used for in-circuit testing using fixtures. Although the test configurations and rule data are similar, there is no direct mapping between SailWind Layout test point configuration data and CAM350 configuration data.

However, test point status is passed to CAM350 for information purposes. The test point status and the access side are translated from the ASCII file to the CAM350 database.

Test point configuration data and clearance rules are not passed to CAM350 from SailWind Layout.

Probe sizes are not passed to CAM350. Only the test point location and probe side are passed.

CAM350 Link Document Conversion

SailWind Layout CAM documents combine layer and data type specifications for output in a CAM photoplot file. A new layer is created in the CAM350 file for each SailWind Layout CAM document.

For more information on CAM documents see the “Creating CAM Outputs to Manufacture Your PCB” topic.

CAM documents can be arbitrarily named. These arbitrary names are used for layer names in the CAM350 database to hold the CAM document content. The CAM350 Link extracts the SailWind Layout data types from the PADS-format ASCII file for the layers specified in the MISC CAM section of the ASCII file. The resulting layers in the CAM350 database match the content of the Gerber photoplot file generated by SailWind Layout CAM operations.

Tip
CAM document names are truncated to 16 characters due to the maximum file name length in CAM350.

Supported CAM documents for CAM350 Link include the following five types:

- Plane: ground plane (pads, vias, copper, lines, text)
- Routing: top (pads, vias, tracks, copper, lines, text)
- Silkscreen: (outline top)
    - top: (ref. des., part type)
    - silkscreen top: (lines, text, outlines)
- Paste mask:
    - top: (pads)
    - paste mask top: (copper, lines, text)
- Solder mask:
    - top: (pads, test points)
    - solder mask top: (copper, lines, text, test points)

Chapter 42
Object Linking and Embedding

The following topics discuss the embedding of OLE objects in SailWind Layout, the characteristics of embedded objects, and the operations you can perform on them.

OLE in SailWind Layout

With SailWind Layout object embedding capabilities you can insert other files or other applications as linked or embedded objects within a SailWind Layout design. Linked objects automatically update from the source each time you open the SailWind Layout database.

For example you can insert a Microsoft Word document containing manufacturing information or a Microsoft Excel spreadsheet containing a Bill of Materials. SailWind Layout does not need to understand the format of the inserted object; SailWind Layout communicates with the application that created the file and that source application tells SailWind Layout what information to display and how to display it.

Insertion of SailWind Logic, Layout or Router files as OLE objects in other files (including other PADS files) is not supported. Any SailWind Logic, Layout or Router file inserted in another file will not behave properly and cannot be edited within the “container” application.

Related Topics

Exporting OLE Files

Importing an OLE File

Inserting OLE Objects in SailWind Layout

You can insert a linked or embedded object into your design.

Note:
If you have multiple OLE objects in a SailWind Layout design, you can export them to an .ole file on page 293 and import them into other designs.

Tip
If you insert an object whose source application is an OLE linking and embedding server, that application opens inside SailWind Layout, but runs in the background. The source application's toolbar takes over the SailWind Layout toolbar. You can then work with the source application in the same way as you would if you started it outside of SailWind Layout. This is called visual editing. When you click outside of the object, the SailWind Layout toolbar takes over again, and you can continue to design in SailWind Layout. Because the source application continues to run in the background you can click on the object and work in the source application at any time.

Restrictions and Limitations

- Insertion of SailWind Logic, Layout or Router files as OLE objects in other files (including other PADS files) is not supported.
- You cannot insert OLE linked or embedded objects in the PCB Decal Editor.

Procedure

1. Click the **Edit > Insert New Object** menu item. 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 make the inserted object a link to the original file, select the Link check box. If you choose not to link the object, it is an embedded object.
5. If you want to display the object as an icon, select the Display as Icon check box.
6. Click **OK** to insert the object in SailWind Layout.

Related Topics

Object Linking and Embedding

Embedded Text Documents

You can embed a multi-line text document in your design instead of using the Text tool on the Drafting Toolbar to add multiple single lines of text. After the file is embedded, you can resize the object appropriately.

For instructions, see "Inserting OLE Objects in SailWind Layout".

For this purpose, embedding is recommended over linking since the embedded document will reside inside the *.pcb* file and cannot get lost or accidentally deleted, as an external file can. You can see a sample of an embedded text file in the *preview.pcb* sample design. See the Notes section below the board outline. Double-click the text to activate the Microsoft Word document.

Please note the following restrictions:

- OLE objects cannot be plotted by a pen or photo plotter. They can only be printed.
- 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 be printed only by using a zero plot orientation.

Related Topics

Adding Free Text

OLE Object Selection

Selection of OLE objects in SailWind Layout operates differently than selection of other objects such as pads, nets, components and so forth.

The differences are:

- 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 SailWind Layout objects. OLE objects have selection priority over SailWind Layout components.
- OLE objects are always on top; to select SailWind Layout items under an OLE object, you must move the OLE object.

When you click on an OLE object in SailWind Layout, it behaves like a nontext item in a Word file. that is, it becomes 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.)

Right-click a selected OLE object to access a shortcut menu that lists all commands that you can apply to the OLE object.

OLE Object Management

Managing OLE objects is similar to managing SailWind Layout objects. You can cut, copy, and paste OLE objects using the Cut, Copy, and Paste commands from the Edit menu.

Undo and Redo do not affect OLE objects, so use care when managing them.

Cutting, Copying, and Pasting an OLE Object

You can cut or copy a linked or embedded object, and paste it.

Procedure

1. Copy or cut the OLE object as follows:

    a. Select the OLE object.

    b. Click **Edit menu > Copy** or **Edit menu > Cut**.

2. Click the **Edit > Paste** menu item.

3. Relocate the pasted object as necessary.

Toggling the Background Color of an OLE Object

You can toggle the background color of an OLE object between the design background color and white.

Procedure

1. Select the OLE object.

2. Right-click, and click the **White Background** popup menu item.

Toggling Display of OLE Objects

If you have many linked or embedded objects in your design, you can increase redraw speed by turning off OLE object display.

Procedure

1. Click the **Tools > Options** menu item > .**Global** category > **General** subcategory.
2. In the OLE Document Server area, select the "Display OLE Objects" check box.

Toggling an OLE Object's Display Type

You can toggle an object's display type between display as icon and display as the actual object.

Procedure

1. Select the OLE object.
2. Right-click and click the **(Object_Type) > Convert** popup menu item.
3. In the Convert Dialog Box, click the Display as Icon check box.

Moving an OLE Object

Move OLE objects just as you move non-text objects in a Word document.

Procedure

1. Click and hold the left mouse button on the object.
2. Drag the object to the new location, and release the mouse button.

Resizing an OLE Object

You can resize OLE objects by dragging any of the sizing handles.

Procedure

1. Select the object.
2. Click and drag one of the sizing handles to resize the object. Sizing handles are small, black squares that appear at the corners and along the sides of a rectangular area surrounding a selected object.
3. Release the mouse button when the object is sized correctly.

Converting an OLE Object

You can convert an embedded OLE object to another object type. The types of object you can convert the OLE object to depends on the object's source application.

Tip
Tip: You can also specify that an object remain as it is, but be activated as an object of a different type. See Specifying an OLE Object's Activation Type.

Procedure

1. Select the object.
2. Right-click and click the **(Object_Type) > Convert** popup menu item.
3. In the Convert Dialog Box, click **Convert to**, and select the type to convert to from the Object Type list.
4. Click **OK**.

Specifying an OLE Object's Activation Type

You can specify that an embedded object remain as it is, but be activated as an object of a different type. The types of object you can activate the OLE object as depends on the object's source application.

Tip
Tip: You can also convert an object to a different type. See Converting an OLE Object.

Procedure

1. Select the object.
2. Right-click and click the **(Object_Type) > Convert** popup menu item.
3. In the Convert Dialog Box, click **Activate as**, and select the type to activate the object as from the Object Type list.
4. Click **OK**.

Deleting OLE Objects

You can delete a single OLE object with the Delete key, or all of them using a special menu item.

Restrictions and Limitations

Undo and Redo do not affect OLE objects, so use care when managing them.

Procedure

1. If you want to delete all OLE objects in the design, click the **Edit > Delete All OLE Objects** menu item.
2. Otherwise, select the single OLE object to delete.
3. Press the Delete key.

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.

Procedure

1. Click the **Edit > Links** menu item. 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.

Open an OLE Object for Viewing or Editing

You can edit an OLE object's content within SailWind Layout (known as in-place 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.

See the documentation for the source application for more information on displaying, selecting, deleting, and saving SailWind Layout objects in container applications.

Viewing or Editing In Place in SailWind Layout

You can use an OLE object's source application within SailWind Layout to view or edit the object.

Restrictions and Limitations

- Linked objects cannot be edited in place: they open in a separate window for viewing or editing.
- If the source application does not support viewing/editing within SailWind Layout, the object opens in a separate window.

Procedure

1. Double-click on the object.
2. Make the appropriate edits.
3. When you are finished editing, click outside of the object to end the edit session.

Results

Updates are automatically reflected in the object.

Viewing or Editing in a Separate Window

You can edit an OLE object outside SailWind Layout, in the source application.

Procedure

1. Ensure that the Update on Redraw check box in the Global tab on page 1531 of the Options dialog box is checked.
2. Ctrl+double-click on the object.
3. Make the appropriate edits in the source application window.
4. When you are finished editing, click **File > Close and Return to** <design_file_name>. (This is the form of the command in MS Word. In other applications the command may be slightly different.)

Results

Updates are automatically reflected in the object.

Saving OLE Objects

OLE objects are automatically saved as part of your design when you save a SailWind Layout design.

If you want to save OLE objects separately, use Export on page 293 to save all the OLE objects in an *.ole* file. You can then use Import on page 287 to import them into other designs.

Procedure

1. Click the **File > Export** menu item.
2. Click OLE Files (*.ole) from the Save as Type list.
3. Browse to the location for the new OLE file.
4. Type a name for the file.
5. Click **Save**.

Chapter 43
Troubleshooting

The following topic explain how to deal with fatal design errors, *powerpcb.reg* problems, and locked test point difficulties.

Repairing a Design with Fatal Errors

When SailWind Layout encounters an unrecoverable problem, a fatal error warning with an error code appears.

The topics below explain what to do when you encounter a fatal database error, and steps you can take to recover lost data.

For related information, see also "Error Detection, BMW and BLT, Scripting and Macros".

Recovering from a Fatal Error During File Open

If SailWind Layout crashes or displays a fatal error when you open a design, the design file may be corrupted. Before opening a Support Center service request, try to repair the file using the SailWind Router database integrity checking.

Procedure

1. Immediately after the fatal error occurs, create a new folder named *<design_name>_save,* and copy the damaged design file and its backups (*Layout.pcb*, *Layou1.pcb*, *Layou2.pcb* and *Layou3.pcb*) to the new folder. (You may need to include these files as part of a Service Request.)

2. Open a standalone SailWind Router session, and open the damaged file. If the file does not open, Customer Support will need to review the design; exit this procedure and open a new Support Center service request including the files you saved in *<design_name>_save*. If the file opens, continue with Step 3.

3. Click **File** menu > **Save As**, and save the file under a different name, with the *.pcb* extension.

4. Close SailWind Router.

5. Open the new *.pcb* file in SailWind Layout. If the file does not open, Customer Support will need to review the design; exit this procedure and open a new Support Center service request including the files you saved in *<design_name>_save*. If the file opens, continue with the following step.

6. Export and re-import the design in ASCII format, as follows:

    a. Click **File menu > Export**.

    b. In the File Export dialog box, browse to an appropriate folder, name the new *.asc* file *<design_name>_ascii.asc*, and click **Save**.

    c. In the ASCII Output dialog box,

        i. Click **Select All**.

        ii. Leave Expand Attributes unchecked.

iii. From the Units list, select Basic.

iv. Click OK.

d. When the ASCII file is saved, click **File > New**. (If prompted, accept the default start-up file, and click No to the “Save old file” prompt.)

e. Click **File > Import**.

f. Browse to the *<design_name>_ascii.asc* file you just exported, select it, and click **Open**. If an *ascii.err* file is displayed Customer Support will need to review the design; open a new Support Center service request including the files you saved in *<design_name>_save*. If no *ascii.err*file is displayed, continue with Step 7.

7. Use one of the procedures described in Forward-Annotation of Design Changes from SailWind Logic to compare the netlists of this *.pcb* design and its associated schematic, and if necessary, update the *.pcb*. This restores any lost part or net connections to the *.pcb*.

8. The design is fixed; save the file as *<design_name>_fixed.pcb*.

Recovering from a Fatal Error During Normal Operation

If SailWind Layout crashes or displays a fatal error during normal operation, the *.pcb* file may be corrupted. Before opening a Support Center service request, create copies of all files and run through some simple methods that may repair the design database.

Restrictions and Limitations

You cannot use this procedure to repair a design that fails to open. In that case, try to recover the file using the procedure in Recovering from a Fatal Error During File Open.

Tip
Several files are created in the following procedure; the procedure specifies locations and names for these files in order to ensure that you work with the proper file in each step.

Procedure

1. Exit from the fatal error message. The design is saved to the backup file and closed.

2. Immediately create a new folder named *<design_name>_save,* and copy the damaged design file and its backups (*Layout.pcb*, *Layou1.pcb*,*Layou2.pcb* and *Layou3.pcb*) to the new folder. (You may need to include these files as part of a Service Request.)

3. Create another new folder named *<design_name>_repair* for the repair process files, and copy the damaged design file to the new folder as *<design_name>_bad.pcb*.

4. Restart SailWind Layout, and try to open *<design_name>_bad.pcb*. (Depending on the error, you may be able to open the file and continue.) If the file does not open, exit this procedure and use the procedure in Recovering from a Fatal Error During File Open. If the file opens, continue with Step 5.

5. Type the modeless command I and press the Enter key to run a database integrity check as described in Database Integrity Check During Normal Use.

6. Repeat the action that caused the error. If no errors/warnings appear, skip to Step 13. If errors/warnings do appear, continue with Step 7.

7. Export and re-import the design in ASCII format, as follows:

   a. Click **File menu > Export**.

   b. In the File Export dialog box, browse to the *<design_name>_repair* folder, name the new *.asc* file *ascii_1.asc*, and click **Save**.

   c. In the ASCII Output dialog box,

      i. Click **Select All**.

      ii. Leave Expand Attributes unchecked.

      iii. From the Units list, select Basic.

      iv. Click **OK**.

   d. When the ASCII file is saved, click **File > New**. (If prompted, accept the default start-up file, and click No to the “Save old file” prompt.)

   e. Click **File > Import**.

   f. Browse to the *ascii_1.asc* file you just exported, select it, and click **Open**. If an *ascii.err* file is displayed, exit this procedure and open a new Support Center service request including the files you saved in *<design_name>_save*. If no *ascii.err* file is displayed, continue with Step 8.

8. Repeat the action that caused the error. If no errors/warnings appear, skip to Step 14. Otherwise, continue with Step 9.

9. Save the design as *<design_name>_1.pcb*, and close SailWind Layout.

10. Open and save the design in SailWind Router, as follows:

    a. Start a standalone SailWind Router session, and open *<design_name>_1.pcb*. If the file does not open, Customer Support will need to review the design; exit this procedure and open a new Support Center service request including the files you saved in *<design_name>_save*. If the file opens, continue with step 10b.

    b. Click **File menu > Save As, and save the file as <design_name>_2.pcb.**

    c. Close SailWind Router.

11. Open *<design_name>_2.pcb* in SailWind Layout. If the file does not open, Customer Support will need to review the design; exit this procedure and open a new Support Center service request including the files you saved in *<design_name>_save*. If the file opens, continue with Step 12.

12. Repeat the action that caused the error. If errors/warnings still appear, Customer Support will need to review the design; exit this procedure and open a new Support Center service request including the files you saved in *<design_name>_save*. If no errors/warnings appear, continue with Step 13.

13. Export and re-import the design in ASCII format, as follows:

a. Click **File menu > Export**.

b. In the File Export dialog box, browse to an appropriate folder, name the new *.asc* file *<design_name>_ascii.asc*, and click **Save**.

c. In the ASCII Output dialog box,

i. Click **Select All**.

ii. Leave Expand Attributes unchecked.

iii. From the Units list, select Basic.

iv. Click OK.

d. When the ASCII file is saved, click **File > New**. (If prompted, accept the default start-up file, and click No to the “Save old file” prompt.)

e. Click **File > Import**.

f. Browse to the *<design_name>_ascii.asc* file you just exported, select it, and click **Open**. If an *ascii.err* file is displayed, Customer Support will need to review the design; exit this procedure and open a new Support Center service request including the files you saved in *<design_name>_save*. If no *ascii.err* file is displayed, continue with Step 14.

14. Use one of the procedures described in “Forward-Annotation of Design Changes from SailWind Logic” to compare the netlists of this *.pcb* design and its associated schematic, and if necessary, update the *.pcb*. This restores any lost part or net connections to the *.pcb*.

15. The design is fixed; save the file as *<design_name>_fixed.pcb*.

Database Integrity Check During Normal Use

When you run the database integrity check with the modeless command I, SailWind Layout verifies that the values in the design database are within an acceptable range. If problems are encountered, you are prompted to confirm an automatic database correction.

After you use the automatic correction routine to fix the errors, you should run a set of interactive checks on the new database. These interactive checks include, but are not limited to:

- Clearance checks
- Comparing netlists
- Continuity checks
- Tie plane check

For instructions on how to run these checks see “Verify the Design”.

Warning: Test Point Locked Dialog Box

A Warning dialog box appears when you modify vias, pins, or jumper pins that are locked test points, or clusters that contain test points, or routes that are connected to locked test points. The functions this warning dialog box performs vary, 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 Route Attached to a Locked Test Point

Modification of a Via or Virtual Pin That is a Locked Test Point

Move Sequential to Move Components, Unions, or Clusters with a Locked Test Point

Moving, Dispersing, or Aligning a Component, Cluster, or Union with a Locked Test Point

Chapter 44 SPECCTRA Link

The SPECCTRA Link sends PCB design file data to a SPECCTRA design file, and then imports the SPECCTRA results back to a PCB design file. The Link features an automatic DO File editor to edit existing or new DO files. Parameters are automatically saved between sessions.

SPECCTRA translates route protection status, physical design reuses, test point settings on both vias and component pins, decal keepouts, board keepouts, and board cutouts. Route protection status is passed to and from SPECCTRA for routed traces and vias using SPECCTRA's fix attribute for wires and route attribute for vias. Protection for unrouted traces is passed to SPECCTRA only for whole connections, such as routed links that start and end on component pins. For more information, see Route Protection Status to SPECCTRA.

Unused Pins Net

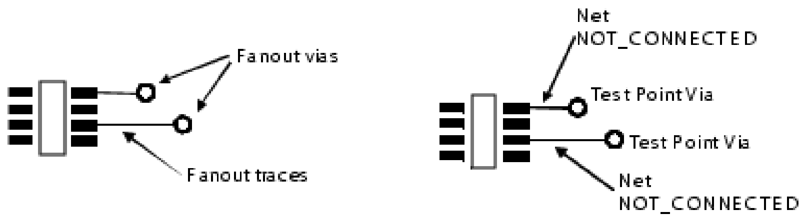
SailWind Layout passes unused pins, or pins that are not connected to a net, to SPECCTRA. These pins and their fanouts are added to a net of unused pins called +UNUSED_PINS+ in SPECCTRA (this was formerly called *UNUSED_PINS*). When you return the routed design to SailWind Layout, you can also pass the unused pins and fanout information. The +UNUSED_PINS+ net information is translated into the NOT_CONNECTED net in SailWind Layout. You can change the name of the NOT_CONNECTED net in the SPECCTRA Options dialog box, but the net should have a unique name.

The NOT_CONNECTED net is a normal net inside SailWind Layout that contains the fanout via, the trace, and the unused pin. Because this net is a normal net in SailWind Layout, pins in it are no longer treated as unused pins in subsequent SPECCTRA sessions. Use the existing DO file to protect this net from being routed as a normal net in subsequent sessions. To recreate this net in SPECCTRA, delete it in SailWind Layout before routing the design.

Tip
SailWind Layout cannot define a connection for a single pin net; therefore, if the +UNUSED_PINS + net contains only a single component pin and fanout, SailWind Layout cannot interpret it.

The figure below demonstrates fanouts on unused pins in SPECCTRA. It also demonstrates how SailWind Layout interprets those fanouts and unused pins. Because SPECCTRA now translates test point attributes on component pins as well as on vias, the following figure demonstrates how SPECCTRA passes test points assigned to the fanouts of the unused pins.

Figure 126. Fanouts on Unused Pins



Data Passed to SPECCTRA

You can pass DFT audit settings and keepouts to SPECCTRA.

DFT Audit Settings to SPECCTRA

You can pass DFT Audit test point placement options to SPECCTRA for its test point placement routine. SPECCTRA generally uses these options when applicable. Options such as test point clearances are passed to SPECCTRA. But options such as nail diameters and fixture drill sizes are used only by DFT Audit. Because they do not affect SPECCTRA's test point insertion, this type of information is not passed.

SPECCTRA bases its clearance calculations on DFT Audit options. SPECCTRA's clearance options are set in the Setup Test Point Rule area of the SPECCTRA DO File dialog box. For more information about DFT Audit, see “Design for Test”.

Tip
SPECCTRA does not allow assigning multiple test points to net. This setting is not passed to SPECCTRA.

Keepouts to SPECCTRA

The SailWind Layout to SPECCTRA Link supports via keepouts and passing other types of keepouts (for wires, bends, components, and pins) to SPECCTRA. Copper shapes on any level are passed in this way. By manipulating this existing keepout functionality, you can pass other specific keepouts to SPECCTRA.

SailWind Layout supports keepouts in the Layout Editor.

Slotted Holes to SPECCTRA

Non plated slotted holes are converted to keepouts on all layers. Plated slotted holes use a circular drill at the electrical center.

Route Protection Status to SPECCTRA

You can set options in SailWind Layout that prevent modifying routed traces and vias in SPECCTRA. This feature protects critically placed routes during interactive routing and in batch routing in automatic routers. Protection is passed to and from SPECCTRA. Components, vias, test points, and other design items associated with protected routes or unroutes are also protected.

Protected Traces and Vias to SPECCTRA

Routed traces with protected status are passed to SPECCTRA with a fix attribute. You cannot modify fixed traces, and the router cannot route to this trace.

Vias attached to at least one protected trace are passed to SPECCTRA with a route attribute. Although you cannot modify these vias, you can route to them to complete a connection.

Table 139 lists the rules used by the SPECCTRA Link to determine via attributes.

Table 139. SPECCTRA Link Rules

Via has protected traces	Locked test point status	Glue status	SPECCTRA via status
Yes	Any	Any	Route
No	Yes	Any	Route
No	No	Yes	Protect
No	No	No	Normal (by default)

Protected, Unrouted Traces to SPECCTRA

Route protection status is passed to SPECCTRA only for those SailWind Layout unrouted traces that are whole connections, which start and end on component pins. Unrouted connections are passed to SPECCTRA as a fromto with a fix attribute.

Protected Components with Routed Traces to SPECCTRA

A component is passed to SPECCTRA with lock status as long as the component has at least one pin attached to a trace with route protection.

Receiving Protection Status for Routed Traces from SPECCTRA

- SPECCTRA wires that have a fix attribute are returned to SailWind Layout as traces with route protection.
- Vias with a protect attribute in SPECCTRA are transformed to glued status in SailWind Layout.
- Vias that have a route attribute in SPECCTRA are not given special protection in SailWind Layout.

Table 140 through Table 142 provide lists of the passing and returning protection status for SailWind Layout traces, vias, and unroutes.

Table 140. Protection Status for Traces

Status in SailWind Layout	Status passed to SPECCTRA	Protection in SPECCTRA	Status passed from SPECCTRA	Status returned to SailWind Layout
Regular trace	Normal type wire	---	Normal type	Regular trace
Regular trace	Normal type	Protect	Protect type	Regular trace
Regular trace	Normal type	Fix	Protect type	Regular trace
Protected trace	Fix type	---	Fix type	Protected trace
Protected trace	Fix type	Protect/Unprotect	Fix type	Protected trace
Protected trace	Fix type	Fix/unfix	Fix type	Protected trace

Table 141. Protection Status for Vias

Status of trace in SailWind Layout						
Has Protected Traces	Locked TP status	Glued Status	Status passed to SPECCTRA	Protection in SPECCTRA	Status passed from SPECCTRA	Status returned to SailWind Layout
Yes	Any	Any	Route	Any	Route	Normal*
No	Yes	Any	Route	Any	Route	Normal*
No	No	Yes	Protect	---	Protect	Glued
No	No	Yes	Protect	Unprotect	Normal	Normal**
No	No	Yes	Protect	Unfix	Protect	Glued
No	No	Yes	Protect	Fix, protect	Protect	Glued
No	No	No	Normal	---	Normal	Normal
No	No	No	Normal	Fix, protect	Protect	Glued

* If defined in the original SailWind Layout design, the glue status of the via is lost. The test point is preserved by SPECCTRA and is returned to the SailWind Layout design.

** You can delete the via in SPECCTRA if the route, edit, or clean commands are performed in SPECCTRA after the Unprotect command.

Table 142. Protection Status for Unroutes

Type of Unroute in SailWind Layout	Passed to SPECCTRA as	Returned from SPECCTRA as
Not protected unroute	No special handling	No special handling

Table 142. Protection Status for Unroutes (continued)

Type of Unroute in SailWind Layout	Passed to SPECCTRA as	Returned from SPECCTRA as
Protected totally unrouted connection	Fromto type, Fix	No special handling—not protected in SailWind Layout
Protected unroutes of partially routed connections	No special handling	No special handling

Physical Design Reuses to SPECCTRA

Routes that are elements of a physical design reuse are passed to SPECCTRA with a fix attribute. Therefore, you cannot modify or route them. SPECCTRA only connects to vias and coppers in a physical design reuse if they are passed with a route attribute. Table 143 provides a summary of Physical Design Reuse Processing.

Table 143. Physical Reuse Processing

Reuse Element	Passed to SPECCTRA as	Returned from SPECCTRA
Component	lock_type fix	No(Warning issued; whether it has changed, or not, the new placement is not returned to SailWind Layout.)
Via	type route	No
Trace	type fix	No
Jumper	lock_type fix	No
Copper with netname	type route	No

Jumpers to SPECCTRA

You can pass jumpers to SPECCTRA. SPECCTRA will not attempt to route SailWind Layout jumpers.

Tip
Do not use SPECCTRA jumpers; they are not backward compatible.

SailWind Layout to SPECCTRA Rules Conversion

Design Rules are converted into one of two types of SPECCTRA rules using the keywords RULE and CIRCUIT. The type used in the conversion is indicated for each SPECCTRA rule.

Routing, High-Speed, and Clearance Rules

Routing, high-speed, and clearance rules are supported for classes, nets, groups, and pin pairs. Additionally, as Table 144 shows, SailWind Layout default rules are passed when possible.

Table 144. Routing Rules

SailWind Layout Routing Rules	SPECCTRA Routing Rules
Copper Sharing	(rule (tjunction on/off))
Priority	(circuit (priority #))
Selected Layers*	(circuit (use_layer # # #))
Selected Vias*	(circuit (use_via a b c))
*Selected Layer and Selected Via rules require the Advanced Rules option in SPECCTRA.	

The Link passes default routing rules to SPECCTRA. Copper Sharing is passed at the PCB rule level.

The Link also passes Priority, Selected Layers, and Selected Vias by creating an artificial class containing all nets you did not assign to a class. This artificial class is named CLASS_nnnnnnn, where nnnnnnn is a number from 0 to 9999999. The default routing rules are then passed to SPECCTRA in this artificial class, as shown in Table 145.

Table 145. High-Speed Rules

SailWind Layout High-Speed Rules	SPECCTRA High-Speed Rules
Min/Max Length	(circuit (length max min) (type actual))
Stub Length	(rule (max_stub #))
Match Lengths	(circuit (match_fromto_length on/off) (tolerance dist))
Shielding w/Gap	(circuit (shield on (use_net net)))(rule (shield_gap dist))Net must be a power net or SPECCTRA will fail on input.
Parallelism Length and Gap	(rule (parallel_segment (gap dist) (limit dist)

The Link passes the default high-speed rules to SPECCTRA. It passes Parallel Length, Tandem Length and Gap, Stub Length, and Min/Max Length by creating an artificial class containing all nets you did not assign to a class. This artificial class is named CLASS_nnnnnnn, where nnnnnnn is a number from 0 to 9999999. The high-speed rules are then passed to SPECCTRA in this artificial class.

Clearance Rules

As shown in Table 146, SPECCTRA accepts only one trace width per rule so the recommended width is passed.

Table 146. Clearance Rules

SailWind Layout	SPECCTRA
	(rule(width dist)
Trace to Trace spacing	(clearance dist (type wire_wire))
Via to Trace spacing	(clearance dist (type via_wire))
Via to Via spacing	(clearance dist (type via_via))
Pad to Trace spacing	(clearance dist (type pin_wire))
Pad to Via spacing	(clearance dist (type pin_via))
Pad to Pad spacing	(clearance dist (type pin_pin))
Smd to Trace spacing	(clearance dist (type smd_wire))
Smd to Via spacing	(clearance dist (type smd_via))
Smd to Pad spacing	(clearance dist (type smd_pin))
Smd to SMD spacing	(clearance dist (type smd_smd))
Board to Trace spacing	(clearance dist (type area_wire))
Board to Via spacing	(clearance dist (type area_via))
Board to Pad spacing	(clearance dist (type area_pin))
Board to SMD spacing	(clearance dist (type area_smd))
Smd to Via same net	(clearance dist (type smd_via_same_net))
Smd to Crn same net	(clearance dist (type smd_to_turn_gap))
Pad to Crn same net	(clearance dist (type pad_to_turn_gap))
Via to Via same net	(clearance dist (type via_to_via_same_net))
Drill to Drill spacing	(clearance dist (type drill_gap))
Body to Body spacing	Unsupported

The Link passes default clearance rules to SPECCTRA at the PCB rule level.

Tip
Copper is translated to a SPECCTRA keepout. In addition, a board outline is translated as a boundary. Both keepouts and boundaries use the area clearance mentioned above.

Specctra Text Translation

Text translates as a keepout. During generation of the SPECCTRA keepout, the maximum text clearance defined in the following clearance rules is used to expand the SPECCTRA keepout area to allow for the required clearance:

- Text-to-Trace spacing
- Text-to-Via spacing
- Text-to-Drill spacing
- Text-to-SMD spacing

Differential Pairs

SPECCTRA does not support minimum/maximum length directly for differential pairs. However, you can add these restrictions using routing rules applied to the pair of nets.

The following SPECCTRA syntax supports net pairs:

```
(pair (nets nm1 nm2 (gap dist)))
```

```
Pin pairs use the syntax:
```

```
(pair (wires nm1 nm2 (gap dist)))
```

Conditional Rules

Conditional rules are separated into two categories: high-speed and clearance. SPECCTRA can accept conditional rules for layers. However, any other rules can only be conditional if they use net classes. A class can contain only nets, not pin pairs or groups. Therefore, any conditional rule that includes a group or pin pair, and is a high-speed rule, cannot be passed to SPECCTRA.

Nets in conditional rules are automatically placed in dummy SPECCTRA classes using the naming convention NEW_CLASS_# as follows:

```
(class NEW_CLASS1 $$$2016)
```

High-Speed Conditional Rules

A high-speed conditional rule is converted like a non conditional high-speed rule, except that it must be of the form class vs. class, as follows:

```
### PADS Layout Conditional Rule ###
```

```
(class_class
```

```
(classes CLASS_0 CLASS_1)
```

```
(rule (parallel_segment (gap dist) (limit dist)))
```

```
(rule (tandem_segment (gap dist) (limit dist)))
```

```
)
```

```
###############################
```

Clearance Conditional Rules

Conditional clearance rules must be of the form object vs. layer, as follows:

```
###########################
```

```
### PADS Layout Group Rule ###
```

```
(group GROUP_0
```

```
(fromto U2-2 U1-2
```

```
)
```

```
(fromto U4-9 U1-9
```

```
)
```

```
(fromto U3-10 U4-10
```

```
)
```

```
(layer_rule 1
```

```
(rule
```

```
(width dist)
```

```
(clearance dist (type wire_wire))
```

```
(clearance dist (type via_wire))
```

```
etc.
```

Tip
Conditional rules vs. text are ignored. SPECCTRA does not recognize text.

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.

Procedure

1. Click the **File > Export** menu item.
2. In the File Export dialog box, in the Save as type list, click SPECCTRA Files (*.dsn).
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. (For more information, see "Data Passed to SPECCTRA", and the "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 the Delete key on your keyboard.

Loading In and Out of SPECCTRA Automatically

When you start SPECCTRA from within SailWind Layout, the SPECCTRA Link dialog box enables you to load in and out of SPECCTRA automatically.

If you set up the maximum number of vias in SailWind Layout, the SPECCTRA Link automatically passes them to SPECCTRA.

CAUTION:
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.

The SPECCTRA Link passes various types of keepouts in SailWind Layout—such as placement, trace, and via keepouts— to SPECCTRA automatically: placement keepout (as place_keepout), trace keepout (as wire_keepout), and via keepout (as via_keepout).

Tip
If you want to have finer control of SPECCTRA, use the Stand-alone SPECCTRA Link dialog box to load in and out of SPECCTRA manually on page 1012.

Procedure

1. On the File menu, click **Export**.
2. In the File Export dialog box, in the Save as type list, click **SPECCTRA Files (*.dsn)**.
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. For more information, see "Creating or Editing a .do File".
6. To set the SPECCTRA automatic startup information, click **Setup**. (For more information, see "SPECCTRA Setup Dialog Box" on page 1729.)
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. (For more information, see "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**. (For more information, see "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 SailWind Layout session (unless "Launch SailWind Layout 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 SailWind Layout session to save memory.

Loading In and Out of SPECCTRA Manually

If you start SPECCTRA independently of SailWind 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).

Tip
If you want to start SPECCTRA from within SailWind Layout, you use the SPECCTRA Link dialog box to load in and out of SPECCTRA automatically on page 1012.

Procedure

1. If you are in SailWind Layout, save the *.pcb* file.

2. Use Windows Explorer to navigate to your *C:\<install_folder>\SailWind<version>\Programs* directory, and double-click *pads2sp.exe*.

3. In the SPECCTRA Link dialog box, to set up the SPECCTRA automatic startup information, click **Setup**. (For more information, see "Setting the SPECCTRA Automatic Startup Information".)

4. To translate the *.pcb* file to SPECCTRA format click **To SPECCTRA**. The To SPECCTRA dialog box appears. (For more information, see "Translating Design Data from SailWind Layout to SPECCTRA".)

5. Start 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 are finished with SPECCTRA, to run the Link again to translate the output back to a *.pcb* design file, click **From SPECCTRA**. (For more information, see "Translating Design Data from SPECCTRA to SailWind Layout".)

8. Load the new *.pcb* file into SailWind Layout.

Translating Design Data from SailWind Layout to SPECCTRA

Use the To SPECCTRA dialog box to translate a *.pcb* design file into a SPECCTRA design file.

Procedure

1. Use Windows Explorer to navigate to your *C:\<install_folder>\SailWind<version>\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**. (For more information, see "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**. (For more information, see "Setting SPECCTRA Options".)

Translating Design Data from SPECCTRA to SailWind Layout

Use the From SPECCTRA dialog box to translate design data modified by SPECCTRA back into a *.pcb* design file.

Procedure

1. Use Windows Explorer to navigate to your *C:\<install_folder>\SailWind<version>\Programs* 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**. (For more information, see "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. (For more information, see "Unused Pins Net".)

Procedure

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
SailWind 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. (For more information, see "SailWind Layout to SPECCTRA Rules Conversion".)

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 SailWind Layout, click Return UNUSED_PINS routing to SailWind Layout. Type the name of the net in the SailWind Layout design that will contain the unused pins. Provide a new name if you do not want to use the default.

   The maximum netname length in SailWind Layout is 47 characters. You can use any alphanumeric characters except for brackets{ }, asterisks *, or spaces.

   Clear this option to ignore unused pin and fanout information when returning to SailWind Layout.

   Tip
   SPECCTRA names the unused pins net +UNUSED_PINS+ while previous versions named it *UNUSED_PINS*. The SPECCTRA Link interprets both names.

Setting the SPECCTRA Automatic Startup Information

Use the SPECCTRA Setup dialog box to set SPECCTRA automatic startup information.

Procedure

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 SailWind Layout after it processes the *.do* file commands, click **Launch SailWind Layout session**.

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.

Procedure

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. (For more information, see “Order” and “Choosing Starburst or Daisy-Chain Wiring” topics in the *SPECCTRA Help*.)

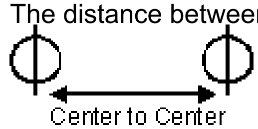
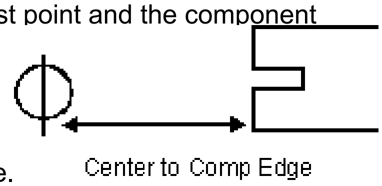
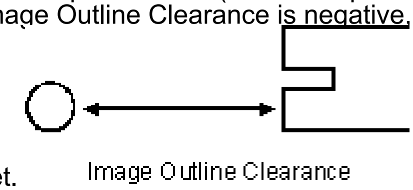
    - Action - control wire rerouting, net routing, and the availability of connections, vias, and layers for autorouting. (For more information, see the “Protect/Unprotect”,” “Fix/Unfix,” and “Select/ Unselect” topics in the *SPECCTRA Help*.)

    - Cost - control routing costs and override the autorouter internal cost table. (For more information, see the “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. (For more information, see the “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. (For more information, see “Data Passed to SPECCTRA”, and the “Testpoint” and “Testpoint Antennas” topics in the *SPECCTRA Help*.)

Table 147. 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.
Grid X, Y	The test point grid. For more information, see “Grid Options on page 1537”.

- 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. (For more information, see "Setting the SPECCTRA Automatic Startup Information".)
10. Click **Finish** and set the parameters in the Setup SPECCTRA Finish dialog box. (For more information, see "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.

Procedure

1. Click the **File > Export** menu item.
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**.

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 SailWind Layout.

During the development of the split/mixed plane features in SailWind 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 the "Defining Split Planes After Routing in SPECCTRA" topic.

- If you typically define split planes before you route your designs in SPECCTRA, see the "Defining Split Planes Before Routing in SPECCTRA" topic.

SPECCTRA translates split/mixed plane layers without routing, named copper, or plane polygons as power layers. SPECCTRA does not consider power layers as routing layers and, therefore, 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. If design rules are present the design will not open in SPECCTRA. The SPECCTRA Link automatically removes or ignores design rules associated with split/mixed plane layers.

Routed Traces on SailWind Layout Split/Mixed Plane Layers

With SailWind Layout you can route traces on split/mixed plane layers. This is also possible in SPECCTRA, but the file is automatically adjusted to achieve the proper results in SPECCTRA. These changes result in behavior modifications of the routing commands in SPECCTRA, possibly causing unexpected routing patterns.

Adjustments to the Design

SPECCTRA translates SailWind Layout split/mixed plane layers with routing or named copper as mixed, rather than power layers. Therefore, SPECCTRA routes on these layers, if necessary, to complete the design.

Behavior Changes in SPECCTRA

Mixed layers are translated as routing layers, increasing the layer count passed to SPECCTRA. For example, a four-layer design with two routing layers, one power layer, and one mixed layer is considered a three routing layer design and cannot be opened in a SPECCTRA configuration that is licensed for only two routing layers.

SPECCTRA considers plane polygons on mixed layers as areas available for connecting component pins to the plane net, but does not consider them obstacles to routing. Therefore, the fanout and route commands can add routes that pass through the plane polygons on SPECCTRA mixed layers.

Isolated instances of routing failure may occur. Failures may include the failure of the fanout command to connect pins to the plane polygons, or when the route command moves a connected plane net pin outside the area defined by the plane polygon, thus isolating the pin from the plane net.

To avoid the behavior changes, use one of the following procedures in your design process:

- Split the planes after routing. For more information, see “SPECCTRA and Split/Mixed Planes”.
- Unroute the offending traces before proceeding. Prior to passing the design to SPECCTRA, unroute any traces in the split/mixed layers. Remove any named copper. After routing in SPECCTRA completes and the design returns to SailWind Layout, reroute the traces you previously unrouted and restore the named copper shapes to the proper split/mixed layer.

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 SailWind Layout and SPECCTRA to quickly route your split plane designs.

Procedure

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. (For more information, see “Layers Setup dialog box and Plane Layer Nets Dialog Box on page 1640”.)

   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 SailWind Layout you can define overlapping plane polygons, but plane flooding 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. (For more information, see "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 SailWind 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. For information on how to do this, see "Verify the Design", "Plane Checking Setup", "Creating a Copper Plane Manually", "Creating a Copper Plane Automatically" and "Plane Checking Setup."

Defining Split Planes After Routing in SPECCTRA

When your design process involves splitting planes after you route the design in SPECCTRA, you do this in SailWind Layout.

Procedure

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. (For more information, see "Layers Setup Dialog Box" and 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. (For more information, see "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 SailWind Layout Split/Mixed Plane Layers.

With SailWind Layout you can define overlapping plane polygons, but plane flooding 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 SailWind 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. For information on how to do this, see “Verify the Design”, “Plane Checking Setup”, “Creating a Copper Plane Manually”, “Creating a Copper Plane Automatically” and “Plane Checking Setup.”

Chapter 45
Error Detection, BMW and BLT, Scripting and Macros

The following topics discuss crash detection and the use of the Basic Media Wizard and Basic Log Test tools.

Crash Detection

If crash detection is enabled, the Error Detected dialog box opens at a crash and allows you to save a report of the SailWind environment as well as pertinent files in a compressed Dump File. You can then submit this file to our Support Center for troubleshooting. You can attach feedback to this report, and optionally, the BMW media and project files.

The Error Detected Dialog Box is inaccessible unless the software crashes and crash detection is enabled in the software *.ini* file.

Crash detection is controlled by the CrashDetection switch in the *.ini* file; it is turned off by default.

- If no CrashDetection switch exists in the [General] section of the *.ini* file, or if 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 the CrashDetection switch exists in the [General] section of the*.ini* file, 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 SailWind Logic, SailWind Layout and SailWind Router sessions. They are particularly useful as a means of supplying information to Technical Support engineers trying to identify and resolve any problematical behavior you may encounter.

If you report problematical behavior for one of the SailWind tools to Technical Support, Tech Support engineers may ask you to use BMW to record session playback media documenting the actions that caused the problem. The tech support engineers can then replay the session with BLT to help them identify and resolve the problem.

Creation of Session Playback Media With BMW

A choice table can help to choose which method to use to create the session playback media.

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 SailWind tool to crash, you can create session playback media based on either the current or the immediately previous SailWind 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 SailWind 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

You can record a SailWind Layout session even though the tool did not crash.

Procedure

1. Start the SailWind tool, but do not open the file in which you encountered the problematical behavior. (You must enable session logging before you open the file.)
2. Type the modeless command BMW ON and press the Enter key to enable session logging. Logging remains enabled for this and all future sessions until you disable it with the BMW OFF command.
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 the Enter key.
6. In the Media Wizard dialog box, click Create Media from Current Session.
7. Type your initials in the User Initials text 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

In certain conditions, you can automatically record a SailWind Layout session that caused the software to crash.

If you are affected by the restrictions listed below, see "Manually Creating Session Playback Media For a Crashed Session" on page 1026.

Restrictions and Limitations

- This procedure works only if the previous (crashed) session started with BMW logging already enabled and logging remained enabled throughout the session.
- This procedure does not give useful results if any additional instance of the SailWind tool ran concurrently (for any period) with the previous (crashed) session.

Procedure

1. Start the SailWind tool, but do not open the file in which you encountered the problematical behavior. (You must enable session logging before you open the file.)
2. Type the modeless command BMW ON and press the Enter key to enable session logging. Logging remains enabled for this and all future sessions until you disable it with the BMW OFF command.

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 SailWind 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

You can manually create session playback media for a crashed session.

Use this procedure when the session you are recreating caused the SailWind 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 listed restrictions.

Procedure

1. Start the SailWind tool, but do not open the file in which you encountered the problematical behavior. (You must enable session logging before you open the file.)
2. Type the modeless command BMW ON and press the Enter key to enable session logging. Logging remains enabled for this and all future sessions until you disable it with the BMW OFF command.
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 SailWind tool.
6. Type the **BMW** modeless command and press the Enter key.
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 *\SailWind 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

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.

Tip
You may see a log file named *<pads_tool>_Session.log* listed in the *\SailWind 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 *\SailWind Projects* folder, and copies into it the *.pcb* file and the Session Log File 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: *\SailWind 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:

Related Topics

Session Log Files

Replaying Session Playback Media with BLT

You can use BLT to replay session playback media created by BMW.

Procedure

1. Type the BLT modeless command, and press the Enter key.
2. Select the session playback media from the Media Directories list and click **OK**.

Results

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

You can use a command line switch in the startup options of the software if you want to automatically record every SailWind Layout session.

If you want BMW to automatically prompt you to create media from the previous session each time you start a SailWind tool, open the SailWind 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).

When you use BMW as a command line option, it creates media of the previous session; use the BMW modeless command to create media of your current session.

Scripting and Macros

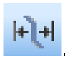
All scripting and macro documentation has been moved.

It can be found in the SailWind Layout Command Reference Manual.

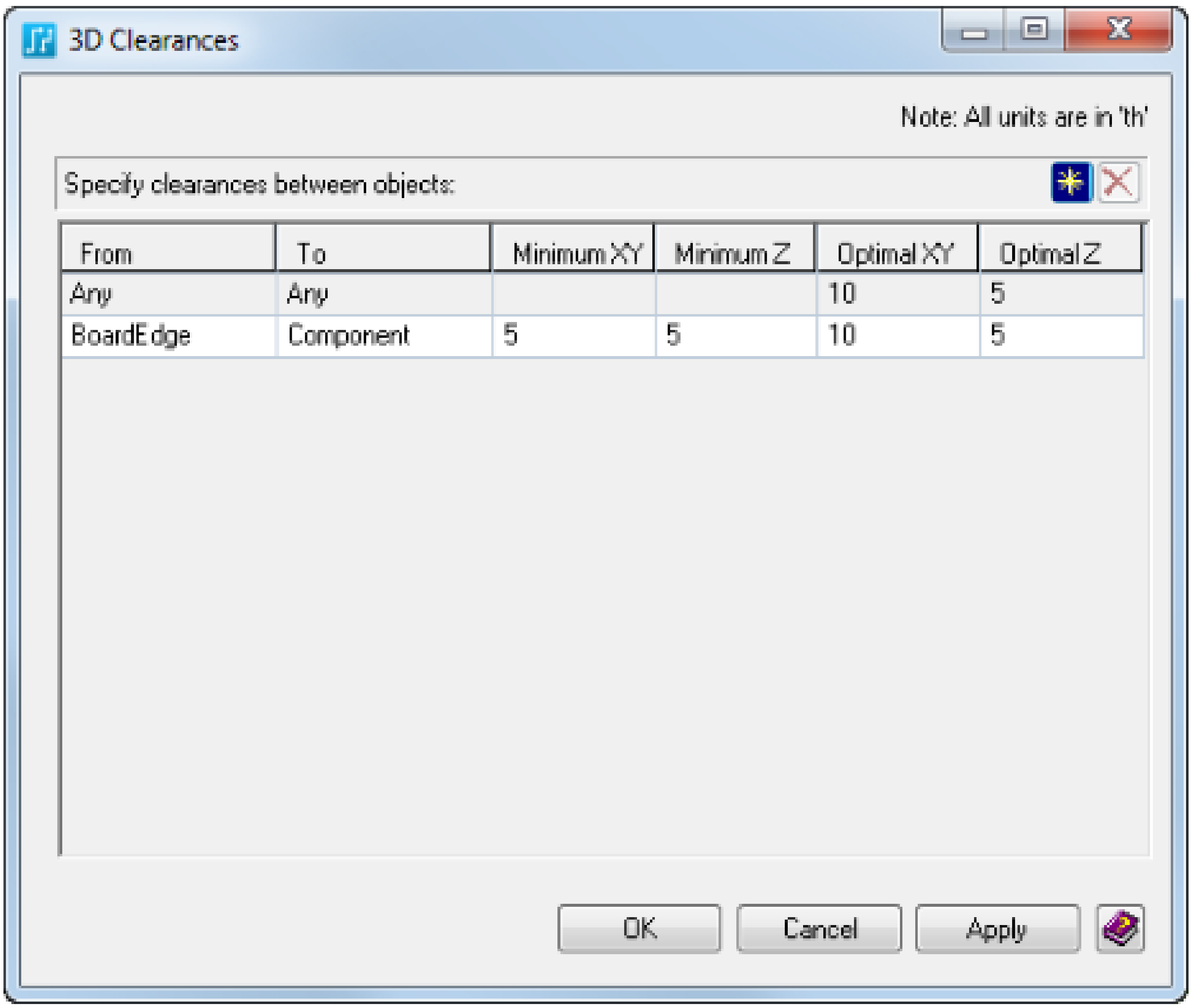
Chapter 46
GUI Reference Elements A

Read the sections that follow to learn more about dialog box elements in SailWind Layout.

3D Clearances Dialog Box

To access: Open the SailWind 3D window (**View > SailWind 3D**), then click the 3D clearances button .

Use the 3D Clearances dialog box to define additional clearances for the 3D models in your design. SailWind Layout performs a clearance check using the additional values when you click the SailWind 3D window Design Rule Check button.



Objects

Field	Description
Specify clearances between objects table	Defines the optimal and minimum allowable clearances in the X, Y, and Z axes between two design objects in the 3D View.

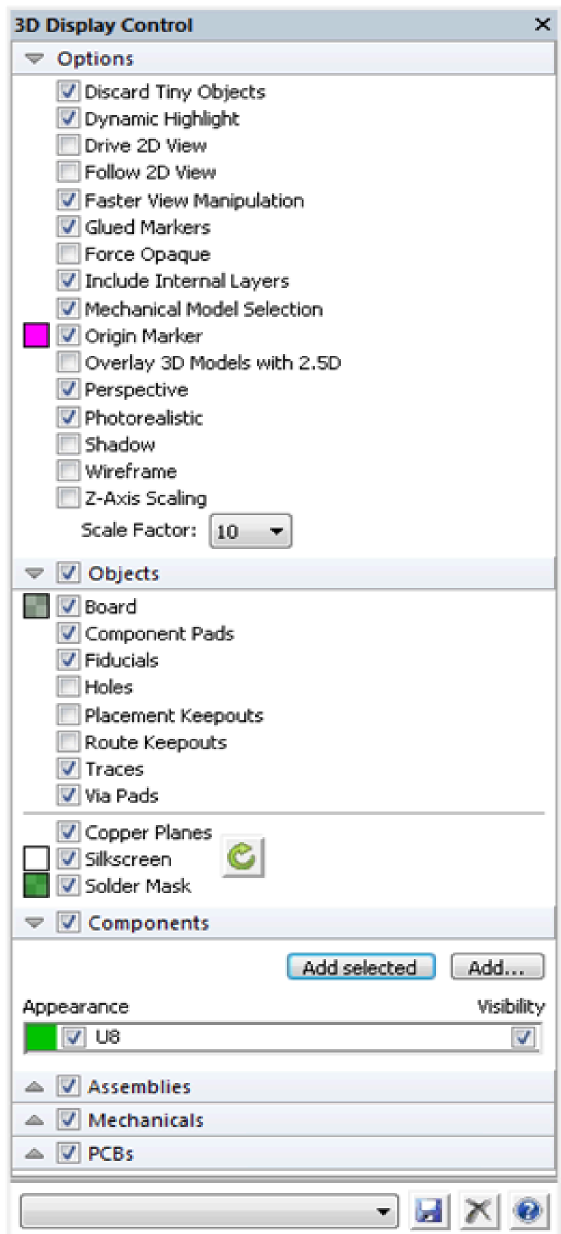
Field	Description
	Use the buttons to create  or delete  additional object clearance definitions. You can add a new general rule to the 3D clearances dialog box; for instance, you can specify a 5 thousandths minimum clearance between all parts (components) and the board edge. Alternatively, you can specify a specific clearance for an individual part, board, mechanical component, or assembly to allow it to exceed a general clearance rule. For example, if you have connector P2 that extends over the edge of the board (thereby violating a previously defined component-to-board-edge rule of 5 thousandths), select the connector in the SailWind 3D window. In the 3D Clearances dialog box, find the part listed as “Component: P2” in the From/To dropdown lists. You can then define a new 3D XY clearance of “0” (effectively removing the clearance constraint) to allow it to extend beyond the edge of the board.
From/To:	Select the desired object from the dropdown lists to define the associated pairing clearance: • **Assembly** — The physical packaging or enclosure in which the active board (PCB) resides. • **Board** — Any part of the board surface, including the board edge. • **Board Edge** — Any edge of the board; includes the edge from the board outline as well as an edge formed by through-cutouts, contours, mounting holes, and cavities. • **Component** — Any imported mechanical model that is designated a “Component” type through the Mechanical Model Properties Dialog Box. • **Mechanical** — Any imported mechanical model that is designated a “Mechanical” type through the Mechanical Model Properites dialog box. • **PCB** — Any imported mechanical model that is designated a “PCB” type through the Mechanical Model Properties dialog box.
Minimum XY **Note:** If online DRC is enabled, when the clearance of an object is less than the minimum, the object turns red.	Defines the minimum allowable clearance in both the X and Y direction (in thousandths) from one object to another. This value cannot exceed the Optimal XY clearance value. If the Minimum XY value is set greater, the Optimal XY value automatically updates to match it.
Minimum Z **Note:** If online DRC is enabled, when the clearance of an object is less than the minimum, the object turns red.	Defines the minimum allowable clearance in the Z direction (in thousandths) from one object to another. This value cannot exceed the Optimal Z clearance value. If the Minimum Z value is set greater, the Optimal Z value automatically updates to match it.

Field	Description
Optimal XY **Note:** If online DRC is enabled, when the clearance of an object is less than the optimal, the object turns yellow.	Defines the ideal clearance in both the X and Y direction (in thousandths) from one object to another.
Optimal Z **Note:** If online DRC is enabled, when the clearance of an object is less than the optimal, the object turns yellow.	Defines the ideal clearance in the Z direction from one object to another.

3D Display Control Window

To access: **SailWind 3D window > 3D General Toolbar > 3D Display Control button**

Manage the view options and change the color and transparency of component and mechanical models.



Objects

Area	Field	Description
Options		
	Discard Tiny Objects	Hides small objects when zoomed out.
	Dynamic Highlight	Highlights components and mechanical models as the cursor hovers over top.
	Drive 2D View	Changing the view and zoom in the 3D view drives the 2D view also.
	Follow 2D View	The 3D view follows changes of the view and zoom in the 2D view.
	Faster View Manipulation	Temporarily hides small models and objects like vias and traces during movements (such as panning or zooming) to make the 3D manipulation faster.
	Glued Markers	A purple pyramid-shaped marker displays over glued components at all times. Letter indicators (G) also appear on each facet of the marker (the L is short for locked).
	Force Opaque	Overrides the transparency settings of everything to which a color is assigned.
	Include Internal Layers	Displays internal layers in the 3D view.
	Mechanical Model Selection	Allows selection of mechanical models. For example, allows selection of a semi-transparent enclosure placed atop components on the PCB.
	Origin Marker	Displays the origin as a double cone shape - helps to locate the origin on the x,y, and z axes.
	Overlay 3D Models with 2.5D	Displays 3D step models and 2.5D (extruded models created from the placement outline and the Geometry.Height attribute). Provides a quick check if the 3D step model matches what's expected.
	Perspective	Helps with depth perception by adding a vanishing point behind the 3D image. Close objects appear larger and farther objects appear smaller when you tilt away from Top view.
	Photorealistic	Shows a lifelike color of copper.
	Shadow	Adds a drop shadow.
	Wireframe	Displays everything with outlines, similar to outline mode in the 2D view.
	Z-Axis Scaling	Adds an exaggerated spacing between layers of the board stackup to review internal structures such as buried vias. Does not affect the measurement tools which report true values. Hides Assemblies, Mechanicals, and PCBs to prevent visual collisions of 3D structures for better viewing of the board. Lists of those objects in this dialog are also unavailable.

Area	Field	Description
		Scale Factor — Specifies the multiplication factor for visual extrusion of the board stackup. **Note:** To view the barrels of your vias, you must also select the “Holes” and “Via Pads” check boxes.
Objects		
	Board	Displays board outlines and cut outs. You can set the board color and transparency level by clicking the color tile next to the check box.
	Component Pads	Displays pads and drills of through hole pads.
	Fiducials	Displays global fiducials. Fiducials are defined as parts with a single surface mount pad (zero drill size) and one or more of the following is true: • It is not an ECO registered part. • It has an attribute named Fiducial with no value. If opposite side fiducials are required, you will need to add a second fiducial on that other side of the board.
	Holes	Displays the holes in the PCB such as the holes in vias and holes in through-hole pins. **Note:** To display the vias, you must also select the “Via Pads” check box.
	Placement Keepouts	Displays the Placement keepout as an obstruction with height to represent the keepout area. Takes into account any allowed height value underneath it. Placement keepouts set to <All Layers> are not displayed. If a keepout is both a Placement and “Routing” keepout, it takes priority as a placement keepout and is its visibility is controlled by the Placement Keepout check box.
	Route Keepouts	Displays Trace and Copper, Copper plane, Via and jumper, and Test point keepouts. Internal layer keepouts are not displayed. If a keepout is both a Placement and “Routing” keepout, it takes priority as a placement keepout and is its visibility is controlled by the Placement Keepout check box.
	Traces	Displays traces and copper.
	Via Pads	Displays via pads. (To display the holes in your via pads, you must also select the “Holes” check box.)
	Copper Planes	Displays copper planes. Changes to this setting do not appear automatically. You must use the adjacent Refresh button or the Refresh button on the 3D View toolbar.
	Silkscreen	Toggles the display of objects on the silkscreen layer. The objects shown for silkscreen in 3D are based on the settings in your silkscreen CAM documents. If you have not yet defined silkscreen CAM files, the default settings for silkscreen are used. Objects located outside the board outline are not shown in SailWind 3D unless they are partially inside the board outline. Changes to this setting do not appear

Area	Field	Description
		automatically. You must use the adjacent Refresh button or the Refresh button on the 3D View toolbar.
	Solder Mask	Toggles the display of the solder mask layer. The objects shown for silkscreen in 3D are based on the settings in your silkscreen CAM documents. If you have not yet defined solder mask CAM files, the default settings for solder mask are used. Text on solder mask layers is shown as cut outs in the solder mask. Objects located outside the board outline are not shown in SailWind 3D unless they are partially inside the board outline Changes to this setting do not appear automatically. You must use the adjacent Refresh button or the Refresh button on the 3D View toolbar.
Components Defines the appearance of design components added to the list. Select the Components check box to make all components visible (if you clear the check box, you cannot make components in the list visible individually).		
	Add Selected	Adds the selected components to the list so you can customize their appearance. You can also use the Add button to choose from a list of design components.
	Add	Opens the Components dialog box where you can choose components to add from a list, by reference designator, instead of by selecting them in the view. Deletions can also be performed in this dialog.
	Appearance	• **Color tile** — Click to assign a custom color, and/or transparency to the photo-realistic or custom color. A letter P appears in the color tile until you decide to override the photo-realistic color with a custom color. • **Check box** — Enables the custom appearance assigned in the adjacent color tile.
	Visibility check box	Displays the component in the 3D view.
Assemblies Defines the appearance of assembly objects. After importing the model, you assign the model to this type in the Mechanical Model Properties. You must then add the model to the list. Right-click an item to remove it or remove all items from the list. This section is unavailable when the Z-Axis Scaling check box is selected. Select the Assemblies check box to make all assemblies visible (if you clear the check box, you cannot make assemblies in the list visible individually).		
	Add Selected	Adds the selected assembly type models to the list so you can customize their appearance. You can also use the Add button to choose from a list of imported models that are set to the Assembly type.
	Add	Opens the Assemblies dialog box where you can choose models to add from a list instead of by selecting them in the view. Deletion is also performed in this dialog.

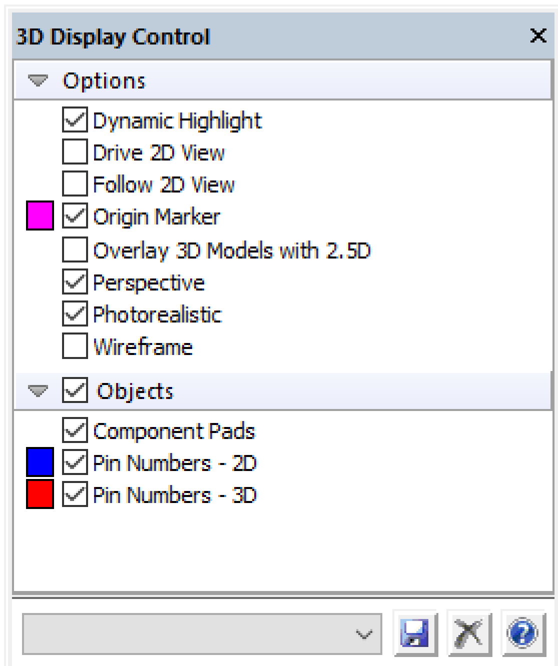
Area	Field	Description
	Appearance	• **Color tile** — Click to assign a custom color, and/or transparency to the photo-realistic or custom color. A letter P appears in the color tile until you decide to override the photo-realistic color with a custom color. • **Check box** — Enables the custom appearance assigned in the adjacent color tile.
	Visibility	Displays the assembly in the 3D view.
Mechanicals Defines the appearance of mechanical objects. After importing the model, you assign the model to this type in the Mechanical Model Properties. You must then add the model to the list. Right-click an item to remove it or remove all items from the list. This section is unavailable when the Z-Axis Scaling check box is selected. Select the Mechanicals check box to make all mechanical objects visible (if you clear the check box, you cannot make mechanical objects in the list visible individually).		
	Add Selected	Adds the selected mechanical type models to the list so you can customize their appearance. You can also use the Add button to choose from a list of imported models that are set to the Mechanical type.
	Add	Opens the Mechanicals dialog box where you can choose models to add from a list instead of by selecting them in the view. Deletion is also performed in this dialog.
	Appearance	• **Color tile** — Click to assign a custom color, and/or transparency to the photo-realistic or custom color. A letter P appears in the color tile until you decide to override the photo-realistic color with a custom color. • **Check box** — Enables the custom appearance assigned in the adjacent color tile.
	Visibility	Displays the mechanical object in the 3D view.
PCBs Defines the appearance of PCB models. After importing the model, you assign the model to this type in the Mechanical Model Properties. You must then add the model to the list. Right-click an item to remove it or remove all items from the list. This section is unavailable when the Z-Axis Scaling check box is selected. Select the PCBs check box to make PCB models visible (if you clear the check box, you cannot make PCB models in the list visible individually).		
	Add Selected	Adds the selected PCB type models to the list so you can customize their appearance. You can also use the Add button to choose from a list of imported models that are set to the PCB type.
	Add	Opens the PCBs dialog box where you can choose models to add from a list instead of by selecting them in the view. Deletions can also be performed in this dialog.

Area	Field	Description
	Appearance	• **Color tile** — Click to assign a custom color, and/or transparency to the photo-realistic or custom color. A letter P appears in the color tile until you decide to override the photo-realistic color with a custom color. • **Check box** — Enables the custom appearance assigned in the adjacent color tile.
	Visibility	Displays the PCB in the 3D view.
3D Display Configurations You can save 3D display settings. Press the Save button at the bottom of the window and provide a name for the saved settings. The configuration is saved and the name is added to the list beside the Save button for recall when needed. All settings are saved except the design items listed under Components, Assemblies, Mechanicals, PCBs. Choose a configuration and click the Delete button to delete the configuration from the list.		

3D Display Control Window in the Decal Editor

To access: In the Decal Editor, **SailWind 3D window > 3D General Toolbar > 3D Display Control button**

Set various options to manage the view in the SailWind 3D window and control what it displays.



Objects

Object	Description
Options	• **Dynamic Highlight** — Highlights components as the cursor hovers over top. • **Drive 2D View** — Changing the view and zoom in the 3D view drives the 2D view also. • **Follow 2D View** — The 3D view follows changes of the view and zoom in the 2D view. • **Origin Marker** — Display the origin as a double cone shape. Helps to locate the origin on the x,y, and z axes. • **Overlay 3D Models with 2.5D** — Displays 3D step models and 2.5D (extruded models created from the placement outline and the geometry.height attribute). Provides a quick check if the 3D step model matches what’s expected. • **Perspective** — Helps with depth perception by adding a vanishing point behind the 3D image. Close objects appear larger and farther objects appear smaller when you tilt away from Top view.

Object	Description
	• **Photorealistic** — Shows true color of copper. • **Wireframe** — Displays everything with outlines, like outline mode in the 2D view.
Objects	• **Component Pads** — Enables the display of pads. • **Pin Numbers - 2D** — Enables the display and custom coloring of decal pin numbers. • **Pin Numbers - 3D** — Enables the display and custom coloring of 3D model pin numbers, if they exist.
3D Display Configurations	You can save 3D display settings. Press the Save button at the bottom of the window and provide a name for the saved settings. The configuration is saved and the name is added to the list beside the Save button for recall when needed. Choose a configuration and click the Delete button to delete the configuration from the list.

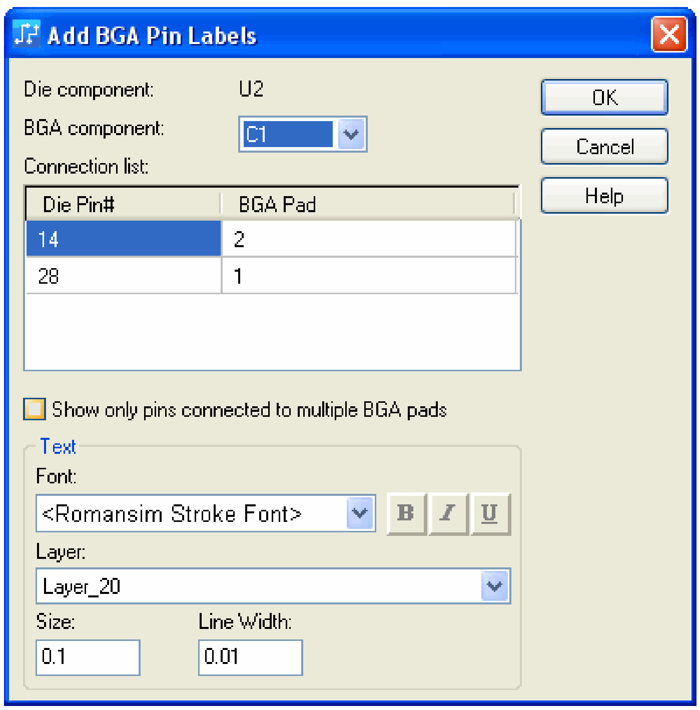
Add BGA Pin Labels Dialog Box

To access: **BGA Toolbar button > Wire Bond Diagram button >** select a BGA pin

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.

Description

- 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.



Objects

Field	Description
Die Component	Displays the component name.
BGA Component	Lists the BGA components available.

Field	Description
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.
	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. **Tip** • 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

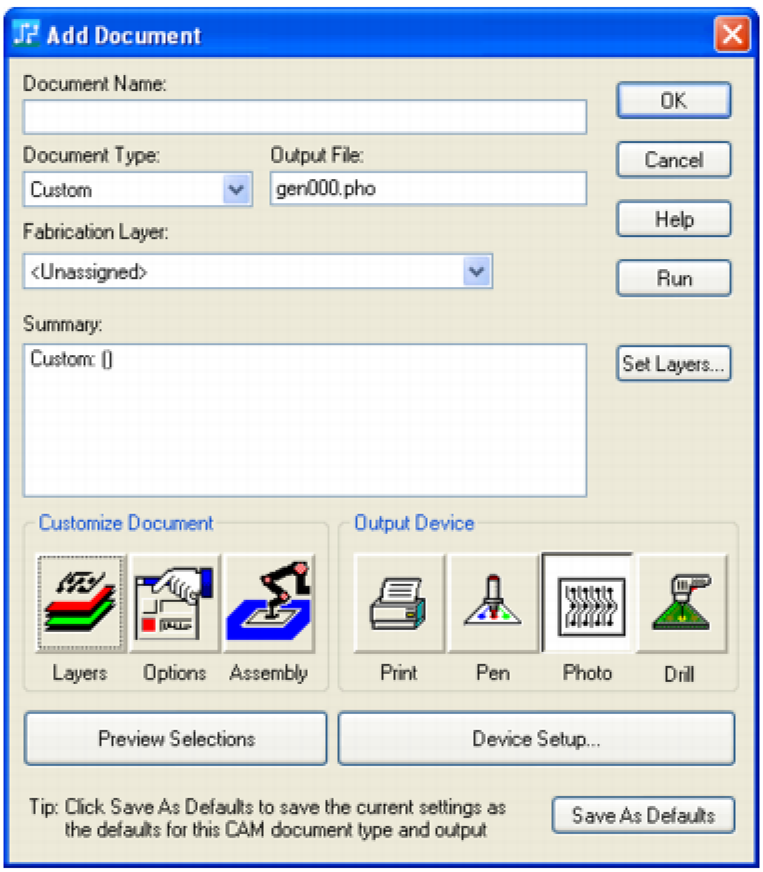
Adding BGA Pin Labels

Add/Edit Document Dialog Box

To access:

- **File > CAM** menu item > **Add**
- **File > CAM** menu item > select a document name > **Edit** button

Use the Add Document or the Edit Document dialog box to define a CAM document. The Add Document and Edit Document dialog boxes are identical except the Edit CAM Document dialog box opens populated with the information from the selected document, while the ADD CAM Document dialog box opens empty.



Objects

Field	Description
Document Name	The name of the CAM document.

Field	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 SailWind Layout. To improve visibility in the work area, CAM planes are negative images and take on the background color of SailWind 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.
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 Global Drill Symbols 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

Field	Description
	supports the macros, aperture selection, and fill area commands of SailWind Layout Gerber outputs. Verify Photo plots can only process RS-274-X Gerber files created by SailWind 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 are working: • Plot Options on page 1641 — Printer, Pen and photo plots • Drill Drawing Options on page 1387 — Drill drawings **Restriction:** Available from the Plot Options Dialog Box only when the Drill Drawing document type is selected. • NC Drill Options on page 1481 — NC drill output
Assembly button	Opens the Select Assembly Variant Dialog Box.
Output Device	You can print or plot your outputs on paper or you can send a photo plot file to manufacturing. **Tip** Print the file to make sure it's okay before then send the file to the manufacturer. • **Print** — Send the document to a printer in your workplace when you click Run. • **Pen** — Send the document to a pen plotter in your workplace when you click Run. • **Photo** — Create a photo plotter file when you click Run. This gerber file is sent to manufacturing for use in a photo plotter to create your PCB. • **Drill** — Create an NC Drill file when you click Run. This is a file sent to manufacturing for input into a computerized drilling machine to drill all the vias.
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 on page 1617, Photo plot Setup on page 1622, or NC Drill Setup on page 1483.

Field	Description
Save As Defaults button	Specifies to save your current settings as the defaults for this CAM document type and output device. The layer selections are not saved as defaults. The location of the Drill Chart is not saved. **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

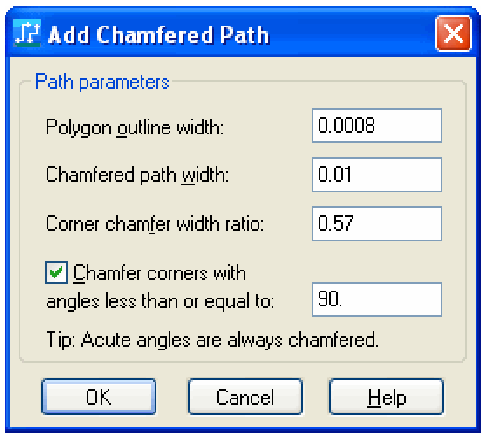
Creating CAM Outputs to Manufacture Your PCB

Add Chamfered Path Dialog Box

To access: Drafting Toolbar > **Copper** button > right-click > **Chamfered Path** popup menu item

The Add Chamfered Path dialog box opens when you select the Chamfered Path - copper shape. You set Chamfered Path parameters before you add the copper to the design.

Tip
You may need to click **OK** to switch off Design Rule Checking.



Objects

Field	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

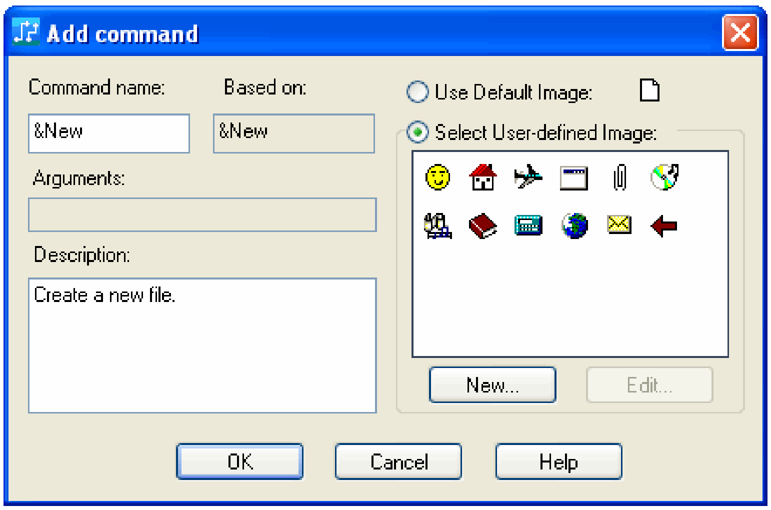
For information about the Add Class Tasks dialog box, refer to the documentation for the Add Net Tasks dialog box.

See: Add Net Tasks/Add Class Tasks Dialog Box

Add/Edit Command Dialog Box

To access: **Tools > Customize** menu item > **Commands** tab > **New** or **Edit** button

Use the Add Command dialog box to create commands that you can then use as selections on menus or as buttons on toolbars.



Objects

Field	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:** SailWind Router only.
Description	Lists what the new command does.
Use Default Image	Use the recommended image. This option is available only if an image was associated with the original command.
Select User-defined Image	Select or create your own image to associate with the new command. This option is available only if an image was associated with the original command.

Field	Description
New	Open the Edit Button Image Dialog Box.
Edit	Open button in the Edit Button Image Dialog Box.

Related Topics

Creating a Custom Command

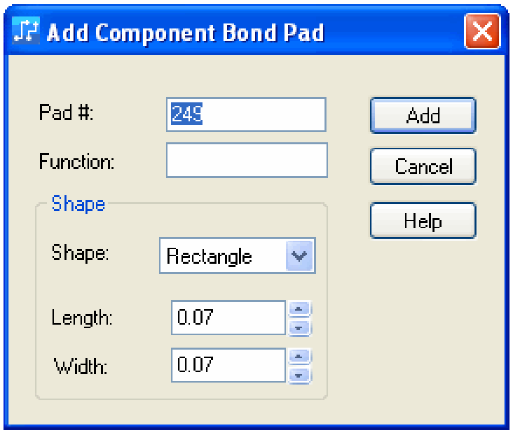
Creating a Custom Menu

Add Component Bond Pad Dialog Box

To access: **BGA Toolbar** > **Wire Bond Editor** button > click the BGA > right-click > **Add CBP** popup menu item

Use the Add Component Bond Pad dialog box to create new bond pads in your design.

Note:
This information applies only to the BGA toolkit.



Objects

Field	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.
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

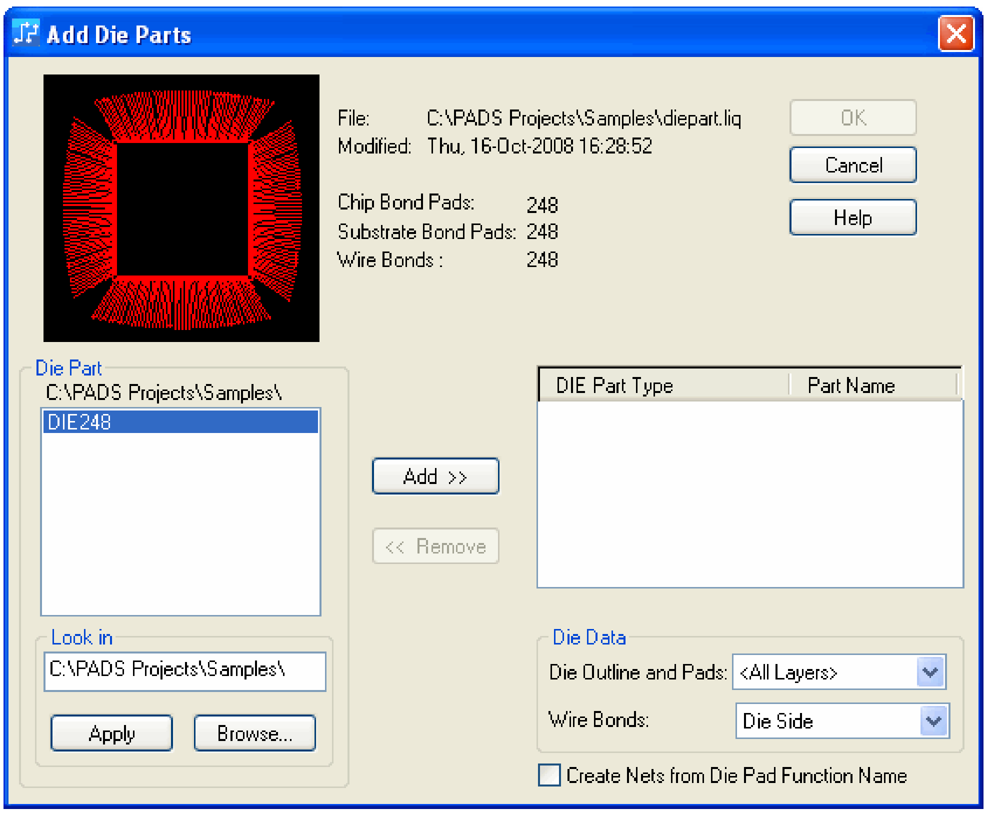
Adding Component Bond Pads Manually to the Die

Add Die Parts Dialog Box

To access: **BGA Toolbar** > **Add Die Parts** button

Use the Add Die Parts dialog box to import die parts from Library IQ into SailWind Layout.

Note:
This information applies only to the BGA toolkit.



Objects

Field	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.

Field	Description
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 SailWind Layout. The net name matches the pin function.

Add Drafting Dialog Box

For information on the Add Drafting Dialog box, refer to the documentation for the Drafting Properties dialog box.

See the “Drafting Properties Dialog Box” topic.

Add Free Text Dialog Box

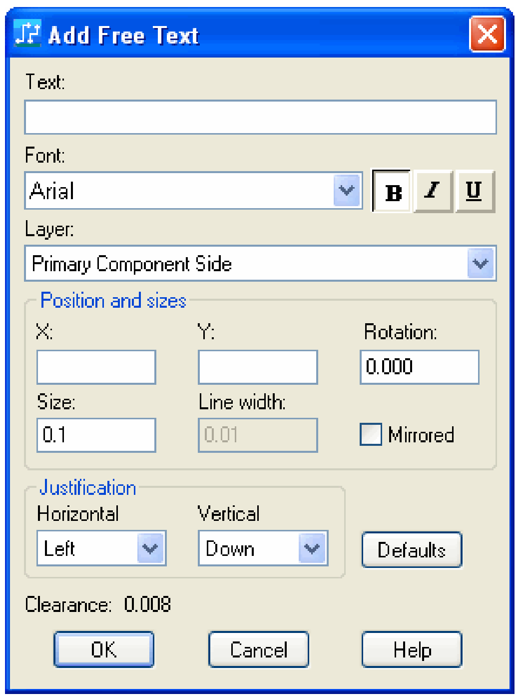
To access: **Drafting Toolbar** button > **Text** button

Use the Add Free Text dialog box to add text that does not belong to another object (free text).

Description

When you add text, there is an invisible bounding rectangle around the text itself.

Note:
Text can only be added one line at a time. See "Creating Reusable Fabrication Notes" for a tip on saving multiple lines of text to the library for reuse.



Objects

Field	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. 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.

Field	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. For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**. **Tip** You can set justification for existing text by selecting the text, then right-clicking and clicking the **Justify Horiz** or **Justify Vert** popup menu item.
Defaults	Restores the SailWind 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

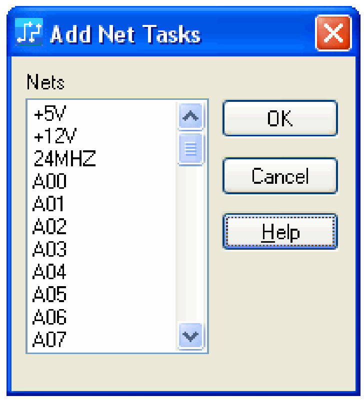
To access:

- **Tools > Verify Design** menu item > High Speed check > **Setup** button > **Add Nets** button

  or

- **Tools > Verify Design** menu item > High Speed check > **Setup** button > **Add Classes** button

Use to run specific electrodynamic checks on nets or classes.



Objects

Field	Description
Nets/Classes list	Lists the nets or classes available in the design.

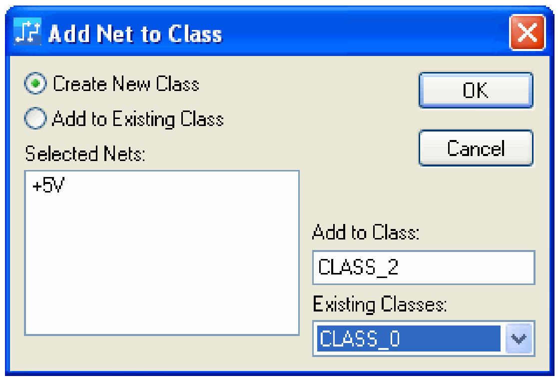
Related Topics

Adding Nets or Classes for Specific High-Speed Checks

Add Net to Class Dialog Box

To access: Select one or more nets > right-click > **Make Class** popup menu item

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.



Objects

Field	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.

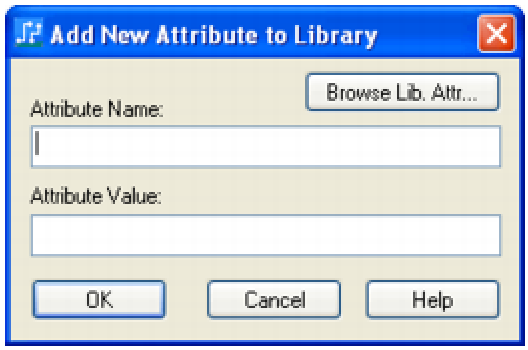
Related Topics

Creating Class Design Rules

Add New Attribute to Library Dialog Box

To access: **File > Library** menu item > Library Manager dialog box > **Attr Manager** button > **Add Attr** button

Use the Add New Attribute to Library dialog box to set name and value properties when adding new attributes to libraries.



Objects

Field	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

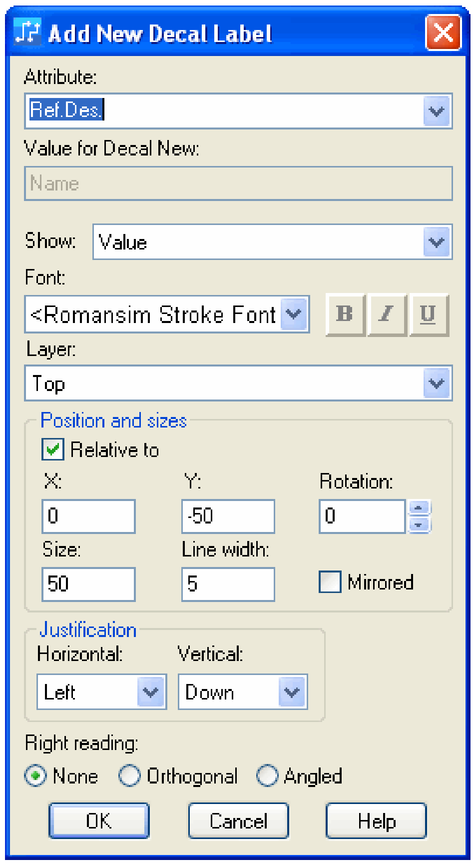
Library Manager Dialog Box

Part Information Dialog Box, Attributes Tab

Add New Decal Label Dialog Box

To access: **Tools > PCB Decal Editor** menu item > **Drafting** button > **Add New Label** button > select a decal

Use to create attribute labels for decals.



Objects

Field	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.
Value for	The value of the selected attribute. 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.

Field	Description
	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 SailWind 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 on page 1862, 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. 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.
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.

Field	Description
	
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. For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**. **Tip** You can set justification for existing text by selecting the text, then right-clicking and clicking the **Justify Horiz** or **Justify Vert** popup menu item.
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

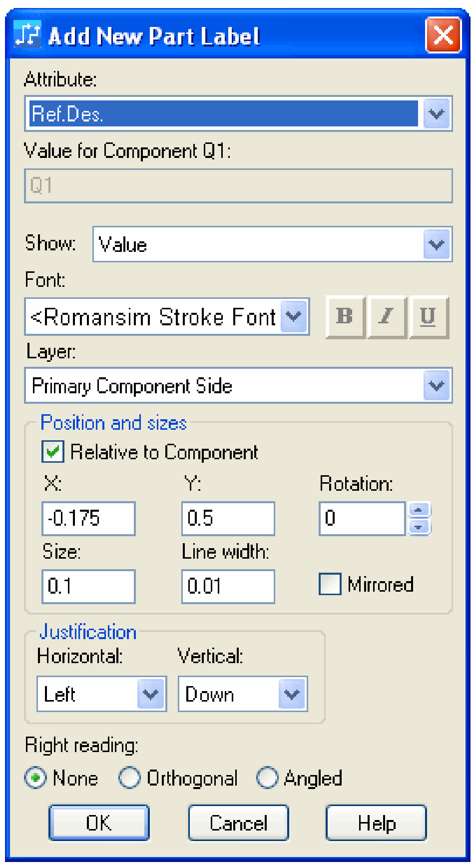
Add New Part Label Dialog Box

To access: Select a part > right-click > **Add New Label** popup menu item

Use the Add New Part Label dialog box to create attribute labels, part type labels, and reference designator labels for components or jumpers.

Description

- Reference designator is the only label available for use when you are creating labels for jumpers.
- If you do not 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 on page 1813 around the label itself.



Objects

Field	Description
Attribute	The attributes available to you. If you are creating labels for jumpers, Reference Designator is the only available attribute.

Field	Description
	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.
Value for	The value of the selected attribute. 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 SailWind 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 on page 1862, 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. 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.
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.

Field	Description
	
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. For vertical justification, click **Left**, **Center**, or **Right**. For horizontal justification, choose **Up**, **Center**, or **Down**. **Tip** You can set justification for existing text by selecting the text, then right-clicking and clicking the **Justify Horiz** or **Justify Vert** popup menu item.
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.

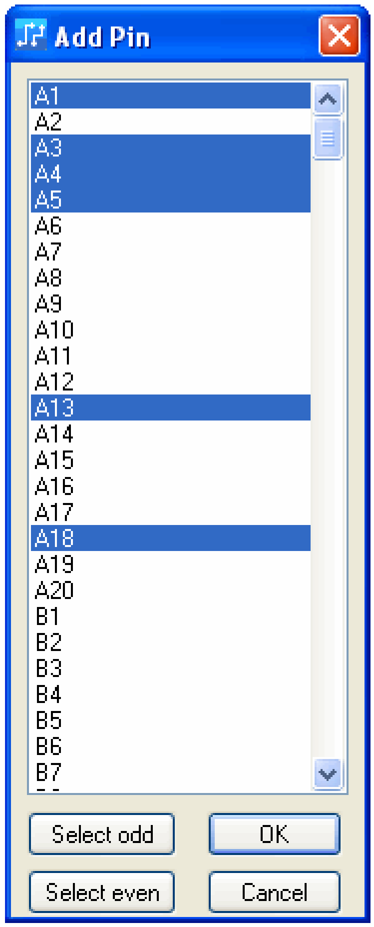
Related Topics

Adding a New Part Label

Add Pin Dialog Box

To access: **Setup > Pad Stacks** menu item > select a Decal or Via > under the Pin: Plated: list, click the **Add** button.

Use the Add Pin dialog box to select specific pins to add to the Pin: Plated: list in the Pad Stacks Properties dialog box.



Objects

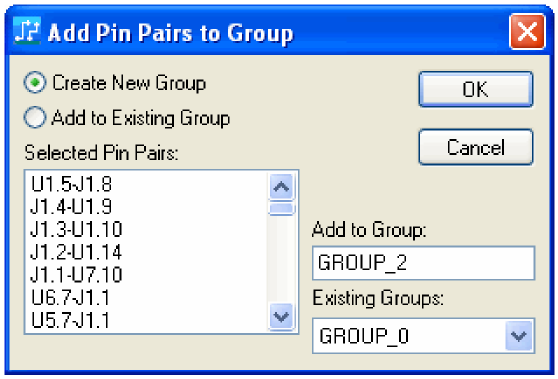
Field	Description
Pin list	Select the specific pin(s) you want to add to the Pin: Plated: list in the Pad Stacks Properties Dialog Box. The pins will be listed individually and can be deselected once added to the Pin: Plated: list if you choose not to give them the all the same pad stacks. • Ctrl+Click multiple items • Shift+click for a range of items • Click and drag for a range of items

Field	Description
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

To access: Select pin pairs > right-click > **Make Group** popup menu item

Use the Add Pin Pairs to Group dialog box to create a collection of pin pairs with a common set of design rules.



Objects

Field	Description
Create New Group	Specifies to make a new group for the selected pin pairs. This selection is unavailable if there are no other existing groups.
Add to Existing Group	Specifies to add the selected pin pairs to an existing group. This selection is unavailable if there are no other existing groups.
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

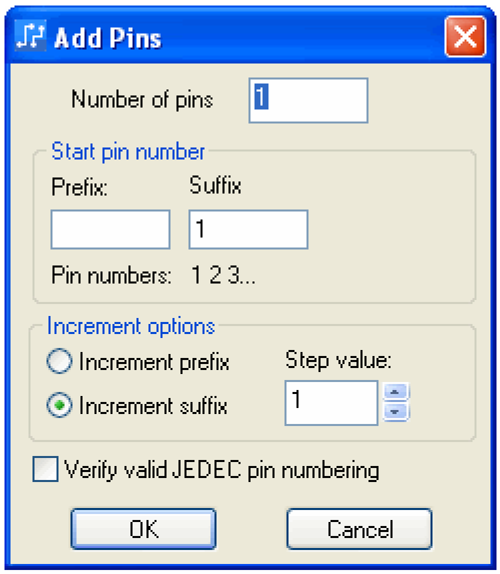
Creating Group Design Rules

Add Pins Dialog Box

To access:

- **File > Library** menu item > select a library > **Parts** button > **New** button > **Pins** tab > **Add Pins** button
- **File > Library** > select a library > **Parts** button > select a part > **Edit** button > **Pins** tab > **Add Pins** button

Use to add pins to a part type.



Objects

Field	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. Alphabetic and numeric values can be used. For example, A1 or 1A. For a single numeric, use either the Prefix or the Suffix box, and void the other box.
Suffix	The suffix you want for your pins. Alphabetic and numeric values can be used. For example, A1 or 1A. For a single numeric, use either the Prefix or the Suffix box, and void the other box.

Field	Description
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

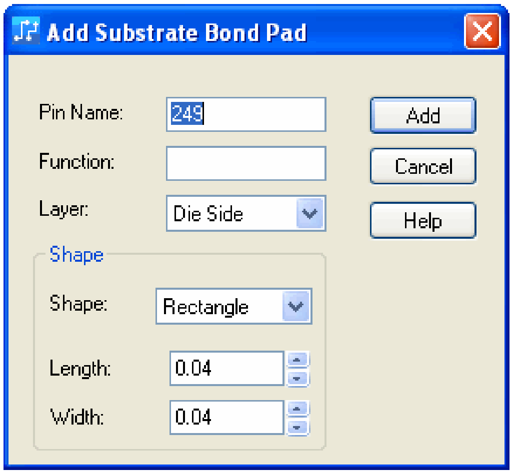
Adding a Series of Pins to the Pins Table

Add Substrate Bond Pad Dialog Box

To access: **BGA Toolbar** > **Wire Bond Editor** button > click the BGA > Right-click > **Add SBP** popup menu item

Use the Add Substrate Bond Pad dialog box to create new bond pads in your design.

Note:
This information applies only to the BGA toolkit.



Objects

Field	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

Adding Substrate Bond Pads

Add/Rename SBP Ring Dialog Box

To access: **Wire Bond Wizard Dialog Box > Add** button

Use the Add SBP Ring dialog box to add a ring for the substrate bond pads.

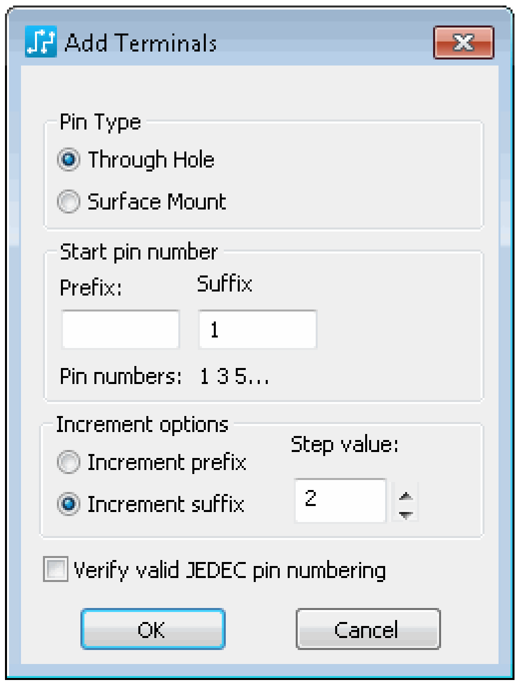
Objects

Field	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

To access: **Tools > PCB Decal Editor** menu item > **Drafting Toolbar** button > **Terminals** button

Use the Add Terminals dialog box to set the type and numbering prior to adding terminals (pads or lands) of the decal.



Objects

Field	Description
Pin Type	**Through Hole/Surface Mount** — Choose between placing a default through hole pin or surface mount pin. The default Through Hole pin is hard-coded as 60 mil round with 35 mil drill. The Surface Mount pin is hard-coded as 60 mil square. After placement, customize the shapes in the "Pad Stacks Properties Dialog Box" on page 1566.
Start Pin Number	**Prefix/Suffix** — The prefix and suffix you want for your pins. For a single numeric, use either the Prefix or the Suffix box, and void the other box. Alphabetic and numeric values can be used. For example, A1 or 1A. **Pin numbers** — A preview of pin numbers based on your input in the Prefix and Suffix boxes.
Increment options	**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.

Field	Description
Verify valid JEDEC pin numbering	Ensures that legal alphanumeric values are used.

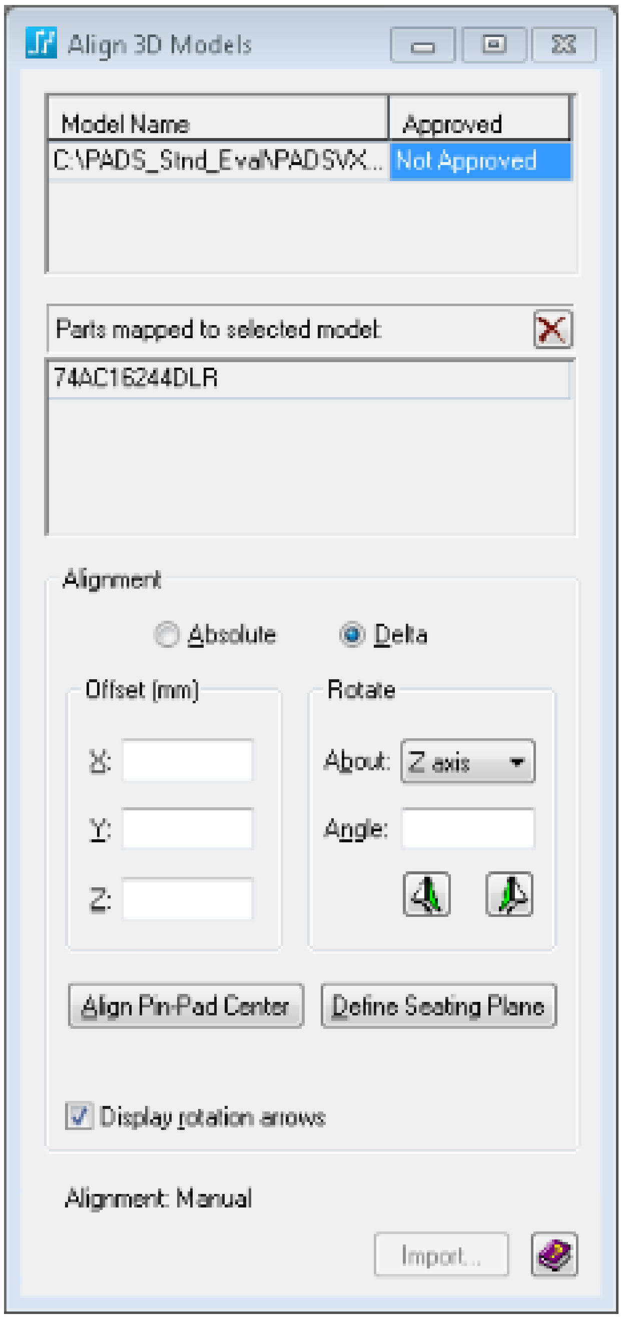
Related Topics

Adding Terminals

Align 3D Models Dialog Box

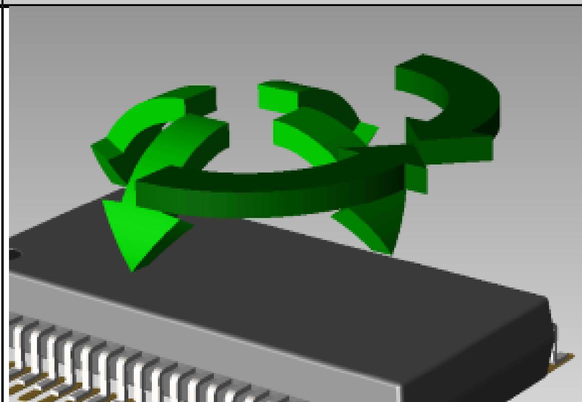
To access: In the SailWind 3D window, select a component in the design then right-click and choose the **Edit decal** popup menu item.

Use the Align 3D Models dialog box to assign and align a 3D model with the pads on the decal. The dialog box opens automatically whenever the Decal Editor opens for a component from the SailWind 3D window.



Fields

Field	Description
Model Name	Displays the name of the selected model.
Approved	Provides a means of documenting whether the 3D model mapping has been approved for record-keeping purposes. Select either “Approved” or “Not approved” from the list. The selected annotation appears in the dialog box whenever Decal Editor is opened for the corresponding part from the SailWind 3D window.
Parts mapped to selected model	Displays a list of parts that use the 3D model. In most cases, only one part appears in the list. However, if more than one part in the PCB design shares the 3D model (for instance, you have the same 3D model represent both capacitors and resistors because the parts appear identical in 3D), more than one part appears in the list.
Alignment mode	Select one of the following: • **Absolute** — Moves the model in relation to the fixed XYZ origin. • **Delta** — Moves the model in relation to its currently displayed position.
Offset	Type values in the X, Y, and Z boxes to move the 3D model. Type positive or negative values to move the 3D model up or down, forward or backward. **Tip** Use the Measure Distance tool in the SailWind 3D window to determine the distance to move the 3D model from its current position to the decal pad.
Rotate	From the About list, select an axis (X, Y, or Z) and type an angle value (in degrees) to rotate the 3D model. Click the “rotate left” or “rotate right” buttons to rotate the 3D model by 90 degrees around the selected axis. Alternatively, you can click any of the green “directional” arrows located immediately above the model to rotate it in 90 degree increments.

Field	Description
	
Display rotation arrows	Makes the rotational arrow controls appear.
Align Pin-Pad Center	Aligns the center of a model pin that you choose with the center of a specific decal pin that you choose. Clicking this button temporarily hides the pads from view. Click the pin surface that you want to center with the pad. The pads become visible again after clicking the pin surface. Click the pad surface next to complete the pin-to-pad center alignment. For more information, see Assigning a 3D Model to a Component.
Define Seating Plane	Click to select the face of the model that you want to seat against the PCB pad. For example, to ensure the pins of a DIP-14 package (or other through-hole component) seat properly in the pad holes, click this button then select the appropriate face on the model (such as the “flared” portion of a pin). The selected model face then seats against the pad.
Alignment:	Indicates whether the 3D model has been aligned manually or automatically.
Import	Click to map a 3D model to the part. Clicking this button opens the Browse dialog box to allow you to locate a 3D CAD file (such as a *.sat* or *.stp* file) for the desired 3D model. This button is available only after selecting the part in the “Parts mapped to selected model” list.

Related Topics

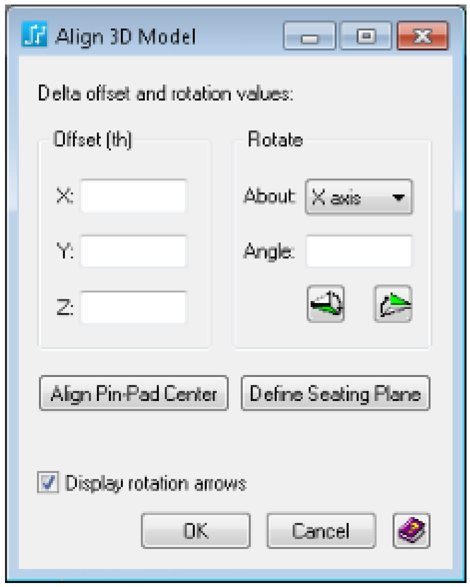
Assigning a 3D Model to a Component

3D Model Mapping

Align 3D Model Dialog Box

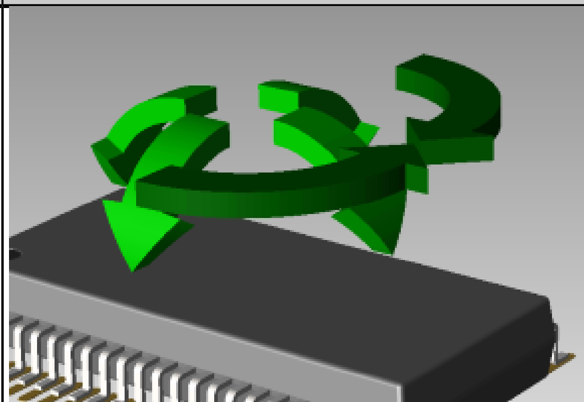
To access: **In the SailWind 3D window, select a component in the design, then click the Import Part Model button**.

Use the Align 3D Model dialog box to align a component's 3D model with the pads on the decal. After importing a 3D model, this dialog opens automatically for aligning the model with the pads on the decal.



Fields

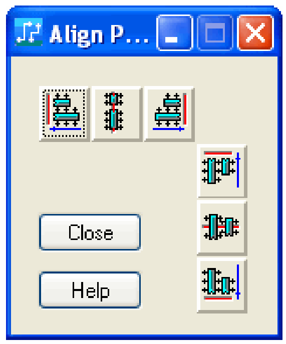
Field	Description
Offset	Type values in the X, Y, and Z boxes to move the 3D model. The values you type move the 3D model in relation to its currently displayed position. Type positive or negative values to move the 3D model up or down, forward or backward. **Tip** Use the Measure Distance tool in the SailWind 3D window to determine the distance to move the 3D model from its current position to the decal pad.
Rotate	From the About list, select an axis (X, Y, or Z) from the About list and type an angle value (in degrees) to rotate the 3D model. Click the "rotate left" or "rotate right" buttons to rotate the 3D model by 90 degrees around the selected axis. Alternatively, you can click any of the green "directional" arrows located immediately above the model to rotate it in 90 degree increments.

Field	Description
	
Display rotation arrows	Makes the rotational arrow controls appear.
Align Pin-Pad Center	Aligns the center of a model pin that you choose with the center of a specific decal pad that you choose. Clicking this button temporarily hides the pads from view. Click the pin surface that you want to align with the pad. The pads become visible again after clicking the pin surface. Click the pad surface next to complete the pin-to-pad alignment. For more information, see Assigning a 3D Model to a Component.
Define Seating Plane	Click to select the face of the model that you want to seat against the PCB pad. For example, to ensure the pins of a DIP-14 package (or other through-hole component) seat properly in the pad holes, click this button then select the appropriate face on the model (such as the “flared” portion of a pin). The selected model face then seats against the pad.

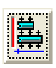
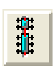
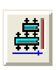
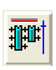
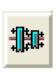
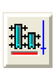
Align Parts Dialog Box

To access: Select the parts you want to align > right-click > **Align** popup menu item

Use the Align Parts dialog box to align selected objects along the left side, right side, vertical center, top, bottom, or horizontal center.



Objects

Field	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.

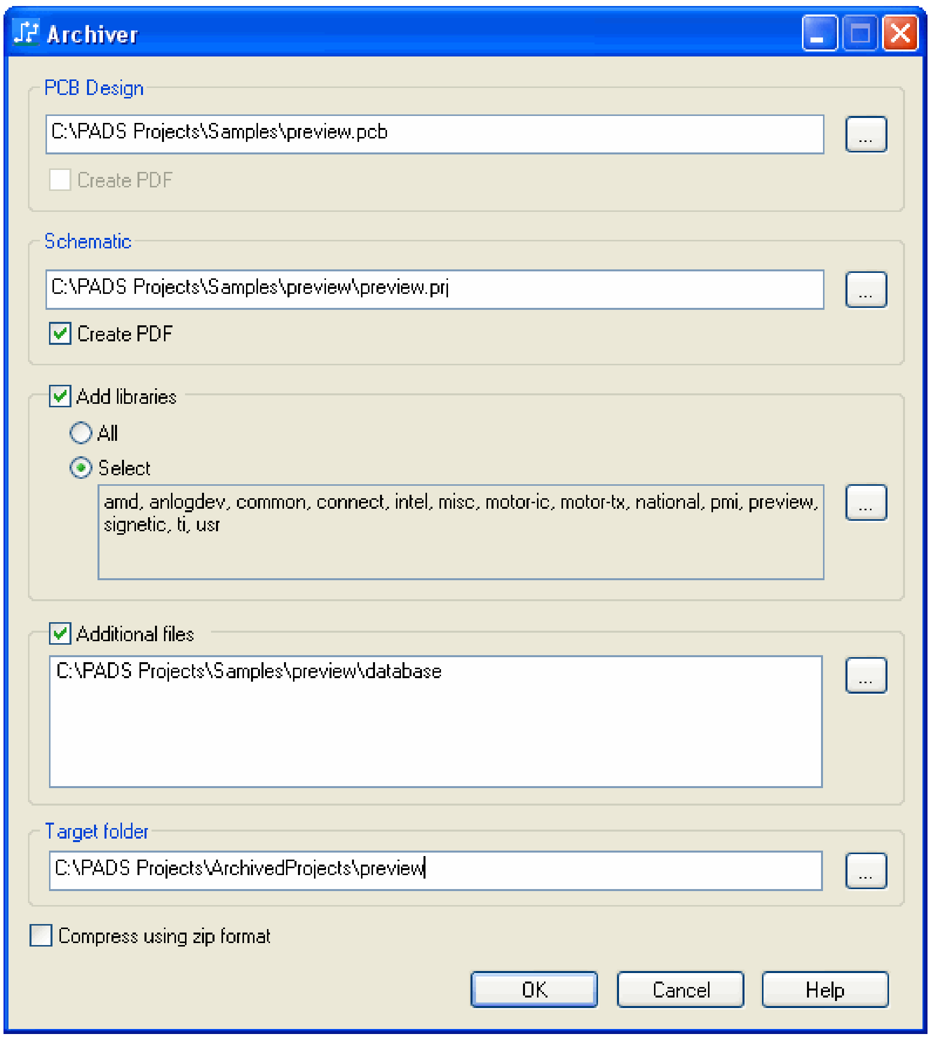
Related Topics

Aligning Objects

Archiver Dialog Box

To access: **File > Archive** menu item

Use the Archiver dialog box to create archives of your designs, projects, files and folders, and libraries.



Objects

Field	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 the Create PDF check box to create a PDF file of the PCB design.

Field	Description
	Restriction: This is unavailable if the file you chose is different from the current design.
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 the Create PDF check box to create a PDF file of the schematic file. **Restriction:** This is only available if you chose a PADS Designer file; it is unavailable if you chose a SailWind 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 check box 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.

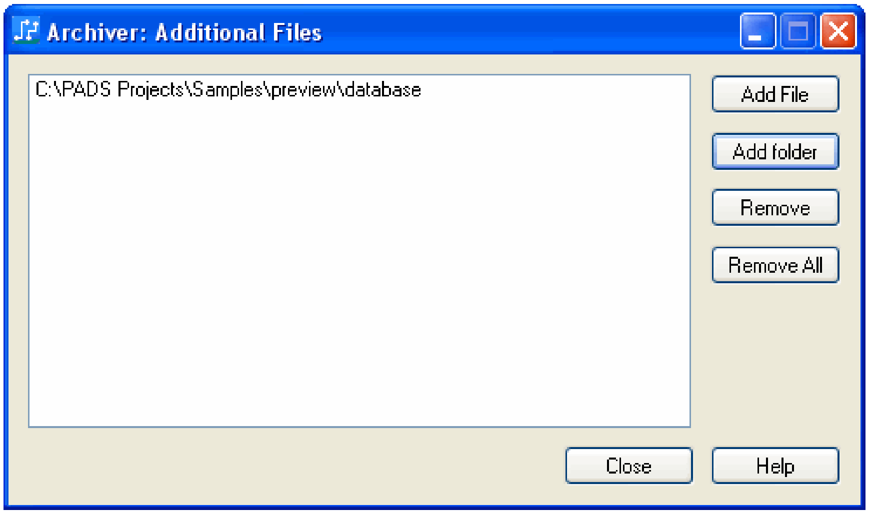
Related Topics

Archiving Your Design

Archiver - Additional Files Dialog Box

To access: **File > Archive** menu item > Additional Files check box > **Browse** button

Use the Archiver: Additional dialog box to add files and folders to the design you want to archive.



Objects

Field	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.

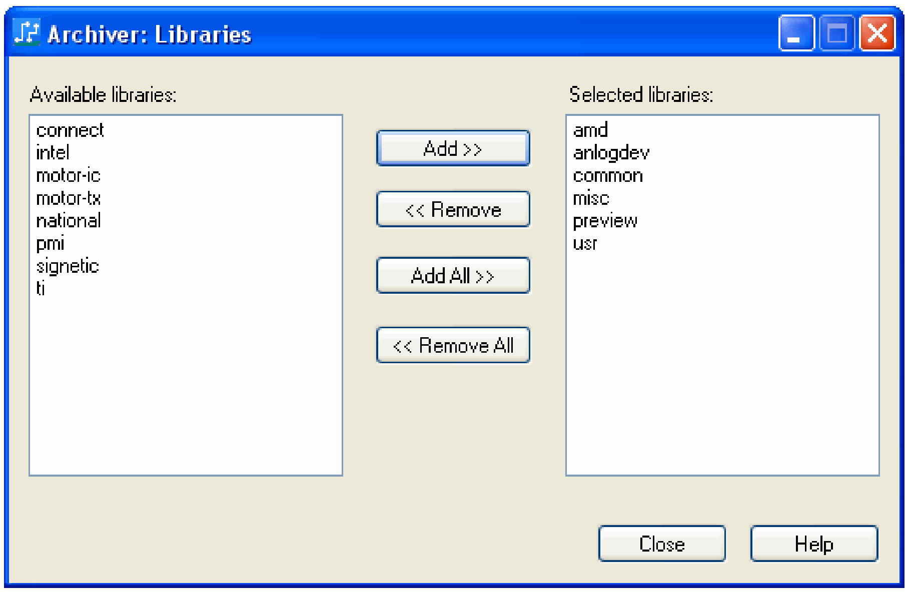
Related Topics

Archiving Your Design

Archiver - Libraries Dialog Box

To access: **File > Archive** menu item > Add libraries check box > click Select > **Browse** button

Use the Archiver: Libraries dialog box to add libraries to the design you want to archive.



Objects

Field	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.

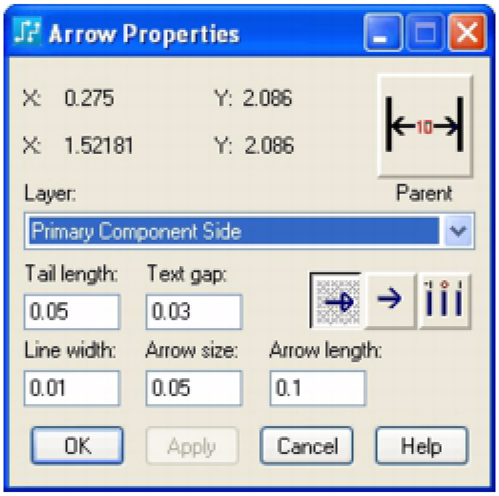
Related Topics

Archiving Your Design

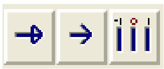
Arrow Properties Dialog Box

To access: Select an arrow > right-click > **Properties** popup menu item.

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.



Objects

Field	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.

Field	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.

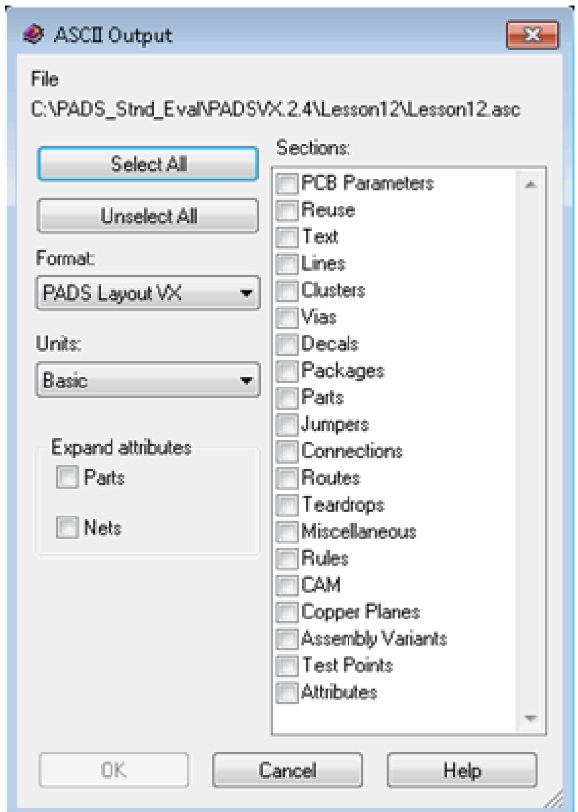
Related Topics

Moving Dimensions and Dimension Objects

ASCII Output Dialog Box

To access: Choose the **File > Export** menu item > choose **ASCII Files (*.asc)** as the export file type > click **Save**

You can use ASCII files to exchange design data between SailWind Layout and external translators or previous versions of SailWind Layout.



Objects

Field	Description
File	The name of the file you are exporting.
Sections	Specifies the sections of the ASCII file you want to export. • **PCB Parameters** — Global design information, such as units and colors • **Reuse** — Elements in, and the definition of, a physical design reuse • **Text** — Text • **Lines** — Two-dimensional (2D) lines • **Clusters** — Clusters and unions • **Vias** — Vias (including dangling vias), jumpers, and padstacks

Field	Description
	• **Decals** — Footprints • **Packages** — Electrical information • **Parts** — Component instances • **Jumpers** — Jumpers. If you plan to export to the PowerPCB V1.1 ASCII format, you cannot output complete jumper information. This is because SailWind 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** — 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 • **Copper Planes** — Copper plane information (applies only to copper planes; does not include information on 2D lines, keepouts, or other shapes). • **Assembly Variants** — 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). Previous versions do not support all of the default attributes. **Tip** Values through the attribute hierarchy are not exported.
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. Use PowerPCB V3.0 to export to both PowerPCB and PowerBGA V3.0 ASCII formats.
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.

Related Topics

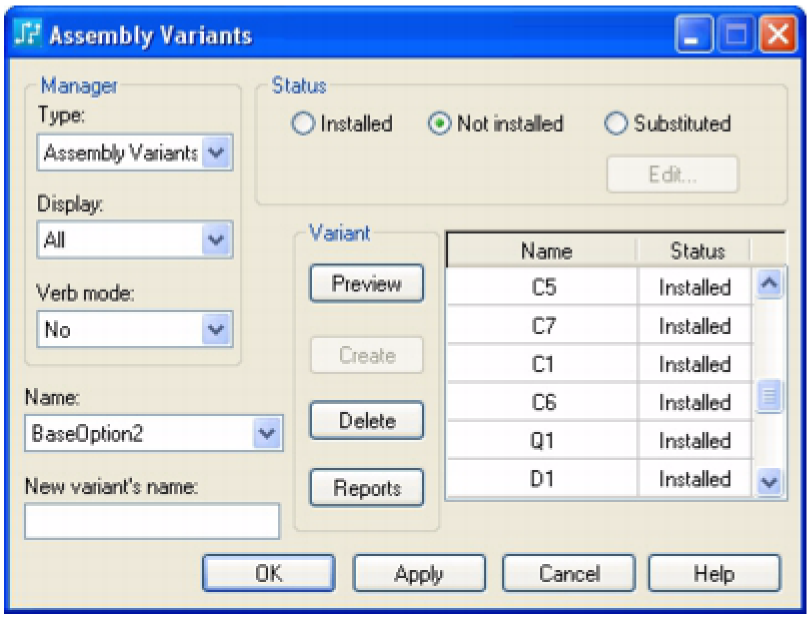
Exporting an ASCII File

Assembly Variants Dialog Box

To access: **Tools > Assembly Variants**

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.

Figure 127. Assembly Variants Dialog Box



Objects

Field	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, Not 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. 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.

Field	Description
Status area	Specifies the status for the selected variant: Installed, Not installed, Substituted. **Tip** If you click Substituted, the Variant Substitute Dialog Box on page 1773 opens.
Edit button	Opens the Variant/Substitute Dialog Box. 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. Changes depending on the selection made in the Type list.

Related Topics

Assembly Variants

Assign CBPs to Rings Dialog Box

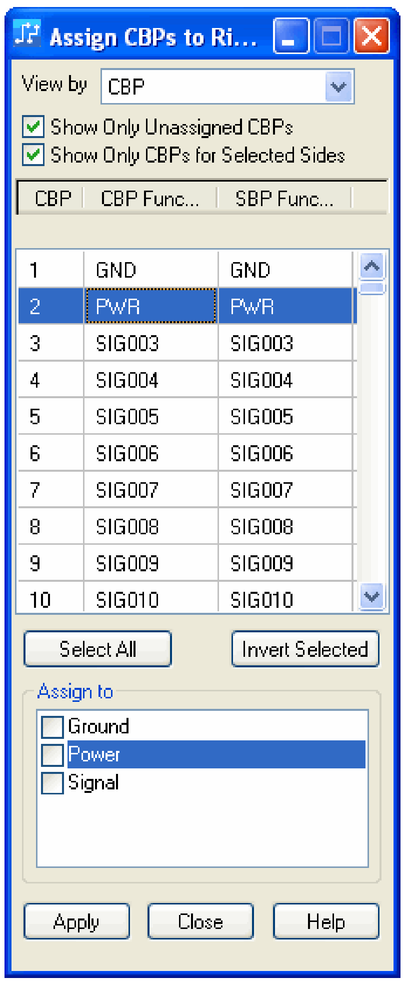
To access: **BGA Toolbar > Wire Bond Wizard > Assign CBPs** button

Use the Assign CBPs to Rings dialog box to specify which component bond pads to wire bond to which ring.

Description

You can preview this placement, see preview of CBP assignments.

Note:
This information applies only to the BGA toolkit.



Objects

Field	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.
CBP column	Displays the CBP number. Displayed only when CBP is selected from the View by list.
CBP Function column	Component bond pad function. Displayed only when CBP or CBP Function is selected from the View by list.
SBP Function column	Substrate bond pad function. 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. 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

Assigning CBPs to Rings

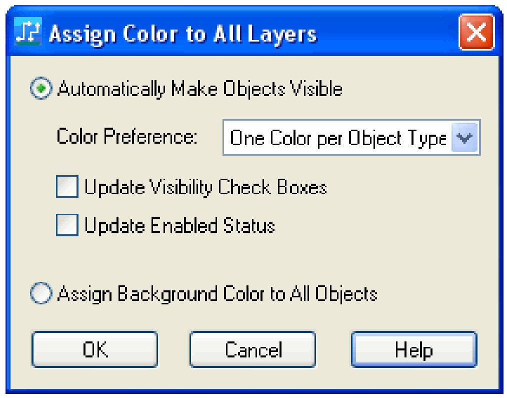
Assign Color to All Layers Dialog Box

To access: **Setup > Display Colors > Assign All** button

Use the Assign Color to All Layers dialog box to make objects visible or hide them in a design.

Description

For more information, see Making All Objects Visible, and Hiding Design Objects.



Objects

Field	Description
Automatically Make Objects Visible	Specifies to automatically find objects on any layer that have the current background color and assigns a color based on the Color Preference list setting.
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 on page 1365. When you select

Field	Description
	this option, nonelectrical layers that do not contain data are disabled.
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.

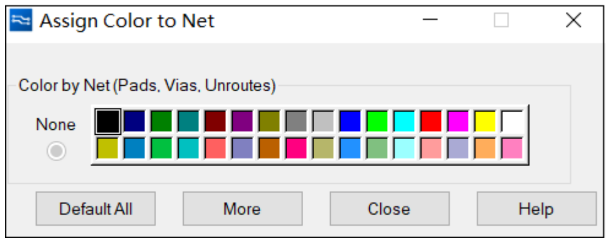
Related Topics

Setting Colors for the Design

Assign Color to Net Dialog Box

To access: Select a net > right-click > **Assign Color to Net** menu item

Use this dialog box to selectively assign display colors to individual nets.



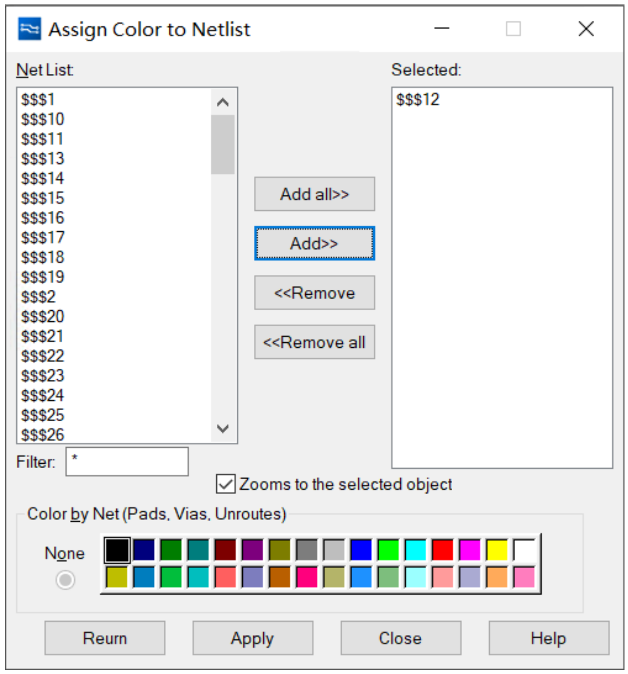
Objects

Field	Description
Color by Net area	Sets a color to the net(s) in selection.
Default All button	Returns net colors to their default assignments (as defined in the Display Colors Setup Dialog Box), except those nets locked in the View Nets Dialog Box.
More button	Opens the Assign Color to Netlist Dialog Box.

Assign Color to Netlist Dialog Box

To access: Click **More** in the Assign Color to Net Dialog Box

Use this dialog box to selectively assign display colors to a batch of nets simultaneously.



Objects

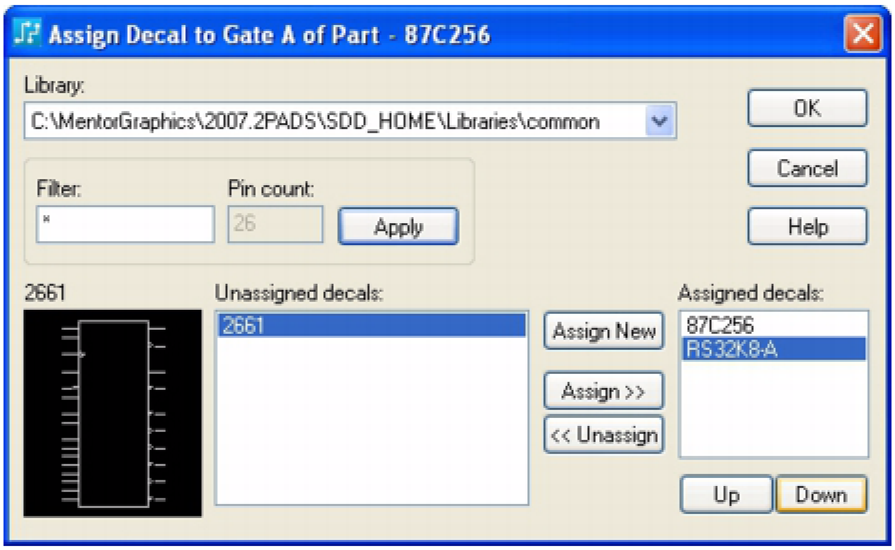
Field	Description
Net list	Lists all of the nets available in the design.
Selected list	Lists all of the nets for which you want to assign a display color.
Add/Remove buttons	Add moves selected net names into the Selected list box where you can highlight net names and set display color for them. Use Ctrl/Shift for multiple selections. Click **Remove** to move the net from the Selected list to the Net List.
Add all/Remove all buttons	• **Add all** moves all the nets from the Net Lsit to the Selected list • **Remove all** moves all the nets from the Selected list to the Net Lsit
Filter box	Filters the Net List by net name. • You can use wildcards or expressions for filter. • The search result will be displayed in real time.
Zooms to the selected object	Fits the object(s) you select into the workspace, which takes effect on selections made in both the Net List and Selected list.

Field	Description
Color by Net area	Sets a color to all nets in the Selected list.
Return button	Goes back to the Assign Color to Net Dialog Box.
Apply button	Makes the color assignment for the Selected list effective, except the nets locked in the View Nets Dialog Box.

Assign Decal to Gate Dialog Box

To access: **File > Library** menu item > select a library > **Parts** button > select part > **New** or **Edit** button > **Gates** tab > double-click a CAE Decal cell > **Browse** button

Use the Assign Decal to Gate dialog box to assign default and alternative CAE decals to gates.



Objects

Field	Description
Library list	Lists all libraries available to you.
Filter	Narrows down your Unassigned decals list. 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.

Field	Description
<< 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.

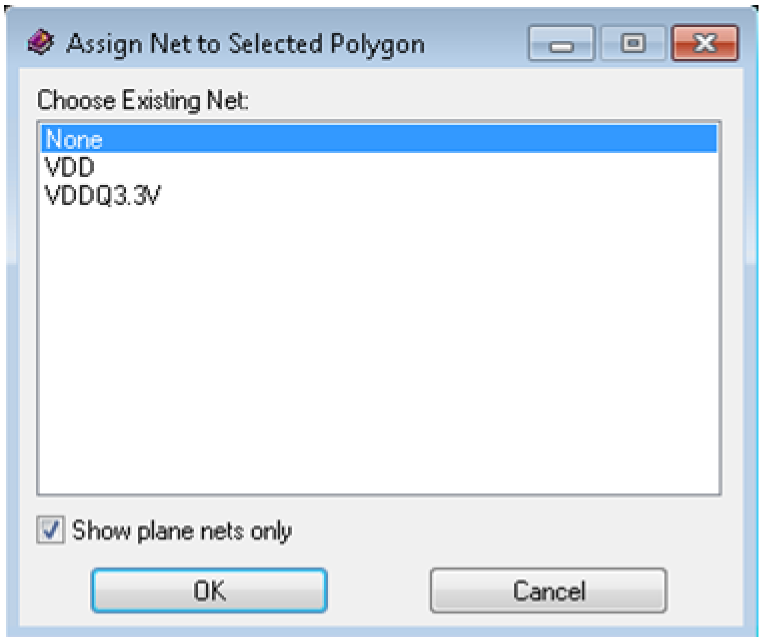
Related Topics

Assigning CAE Decals to Gates

Assign Net to Selected Polygon Dialog Box

To access: Create a split plane on a Split/Mixed plane layer

Choose a net to assign to the split planes you create on a Split-Mixed plane.



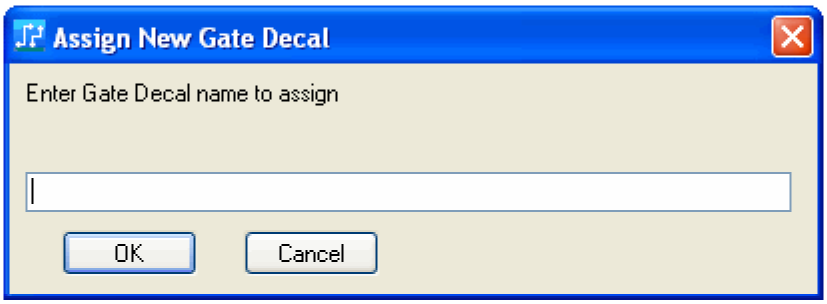
Objects

Field	Description
Choose Existing Net list	Choose a net from the list to assign to the currently selected plane. After making your selection, click **OK** to assign a net to the next split plane.
Show Plane Nets Only check box	Select the check box to limit the nets displayed in the list to only those set up as plane nets. Use this check box when you have an extensive list of available nets to choose from and you want to select a plane net only.

Assign New Gate Decal Dialog Box

To access: **File > Library** menu item > select a library > **Parts** button > select part > **New** or **Edit** button > **Gates** tab > double-click a CAE Decal cell > **Browse** button > **Assign New**

Use the Assign New Gate dialog box to assign a new gate decal when it does not yet exist in the library.



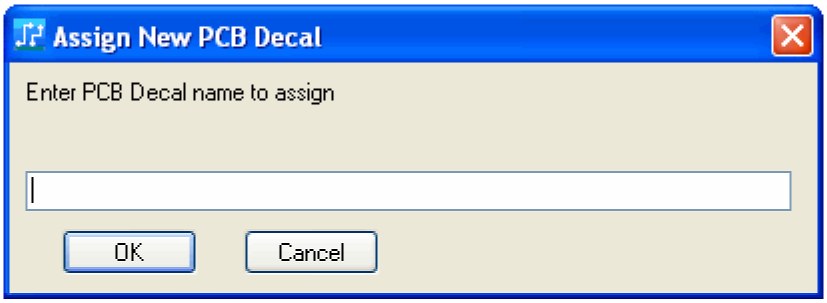
Objects

Field	Description
Text box	Enter the name of the new gate decal you intend to add to the library.

Assign New PCB Decal Dialog Box

To access: **File > Library** menu item > select a library > **Parts** button > select part > **New** or **Edit** button > **PCB Decals** tab > **Assign New**

Use the Assign New PCB Decal dialog box to assign a new PCB Decal when it does not yet exist in the library.



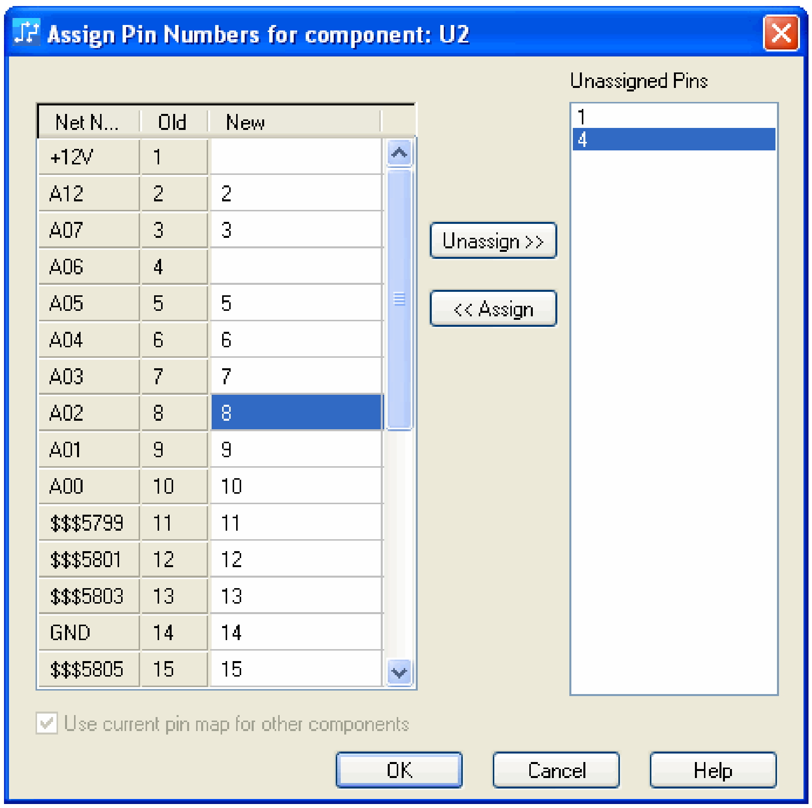
Objects

Field	Description
Text box	Enter the name of the new PCB decal you intend to add to the library.

Assign Pin Numbers Dialog Box

To access: This dialog box opens when you perform a task that can change the number of pins, such as changing a component on page 826 or updating a part type from the library on page 827.

If you update a part type from the library and the pin numbers have changed, 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.



Objects

Field	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.
New	Specifies the new pin number connected to the net. If a pin has a signal assigned to it in the part type, the signal is 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 is empty and/or a pin number is listed in the Unassigned Pins list.
Unassign	Moves the selected new pin(s) to the Unassigned Pins list.

Field	Description
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. If a pin has a signal assigned to it in the part type, the signal is 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

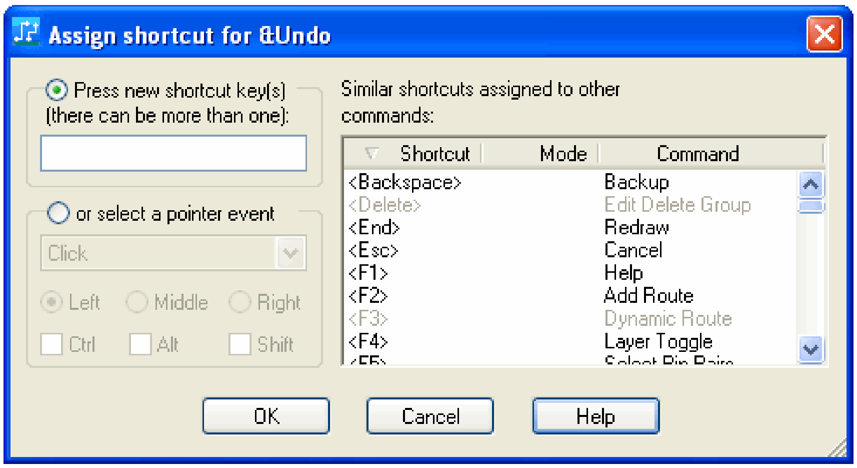
Updating a Part Type from the Library in ECO Mode

Changing a Component in ECO Mode

Assign Shortcut Dialog Box

To access: **Tools > Customize** menu item > **Keyboard and Mouse** tab > **New** button

Create a new shortcut key or pointer event using the Assign Shortcut dialog box.



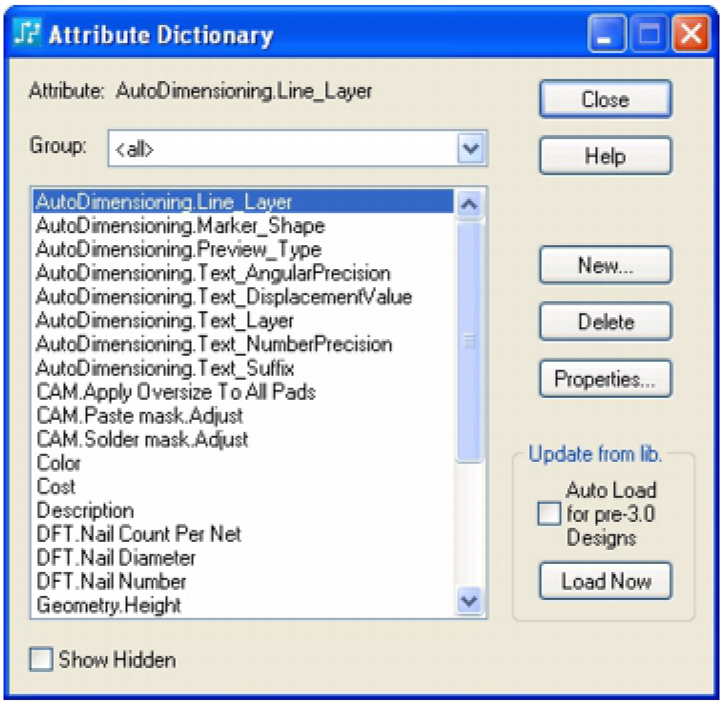
Objects

Field	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.

Attribute Dictionary Dialog Box

To access: **Edit > Attribute Dictionary** menu item

Although you can add new attributes to design objects using the Object Attributes dialog box (select object > right-click > **Attribute** popup menu item), 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 your design or to edit and delete its attributes. You can also use the Attribute Dictionary to assign attributes for the design, or remove attributes from objects.



Objects

Field	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 on page 1111 tab of the Attribute Properties dialog box.
New	Opens the Attribute Properties dialog box on page 1115.

Field	Description
Delete	Removes the selected attribute. 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 on page 1115. 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.
Auto Load for pre-3.0 Designs	Select this check box to automatically update attributes for part types and decals from the current libraries to the Attribute Dictionary as soon as the design is loaded. Then load a design. **Restriction:** Designs created in PADS software before version 3.0 will not list any provided Default Attributes.
Load Now	Automatically loads attributes for part types and decals from the current libraries to the Attribute Dictionary.

Related Topics

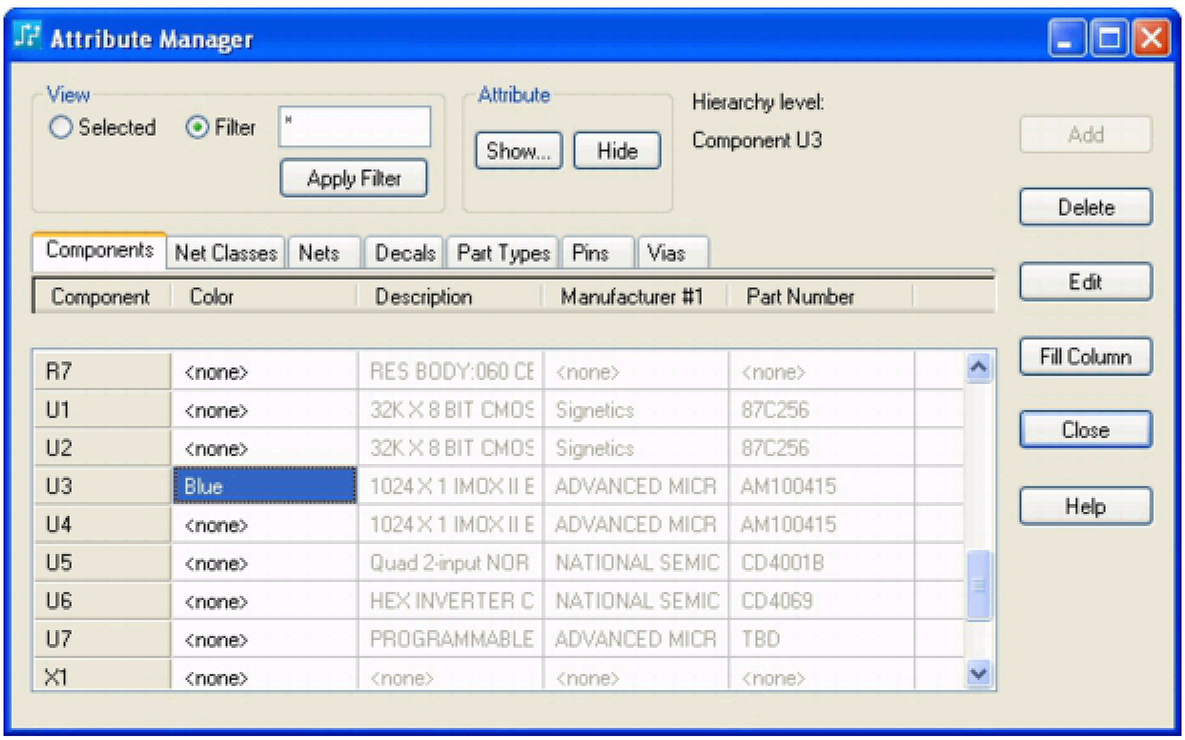
Controlling Attributes

Attribute Manager Dialog Box

To access: **Edit > Attribute Manager** menu item

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 a value summary of an attribute that is based on every value of the attribute assigned to objects of the same type.

CAUTION:
Hidden attributes are not available to the Attribute Manager and are not listed in the Show Attributes dialog box.



Objects

Field	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. You can use wildcards in this box.

Field	Description
	Tip Attributes are shown in the spreadsheet only after you have selected the ones to show from the Show Attributes Dialog Box.
Apply Filter	Executes the filter arguments.
Show	Opens the Show Attributes Dialog Box to select which attributes you want to view in the spreadsheet.
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 are on: Components, Net Classes, Nets, Decals, Part Types, Pins, and Vias. 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.
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 have added to one cell in that column.

Related Topics

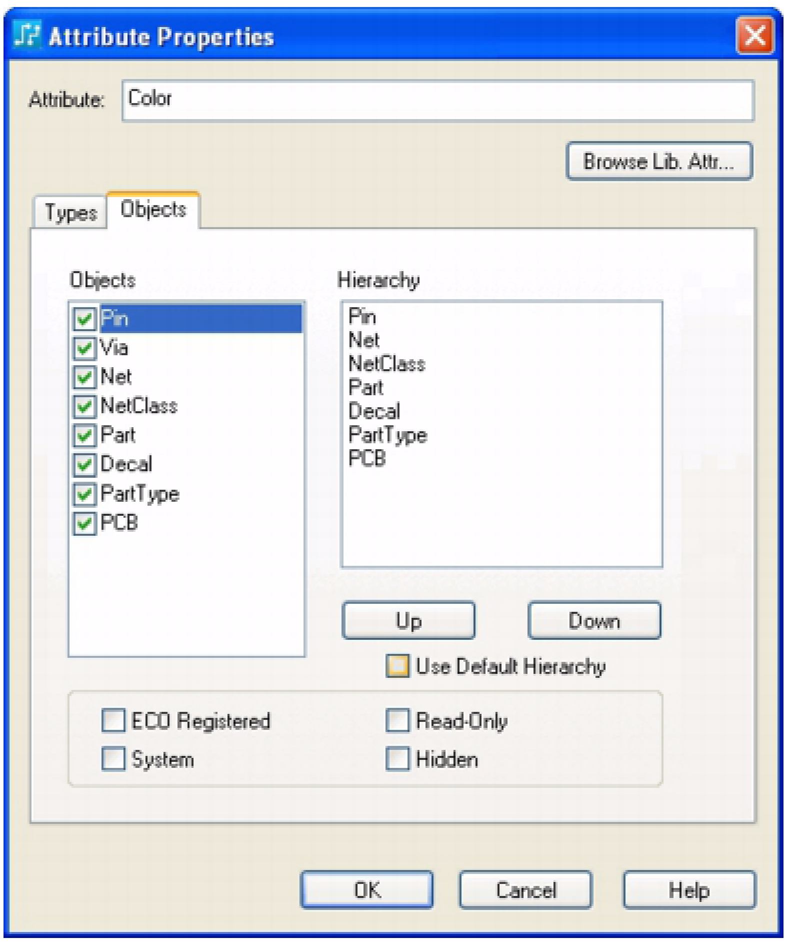
Attribute Manager

Attribute Properties Dialog Box, Objects Tab

To access:

- **Edit > Attribute Dictionary** menu item > select attribute > **Properties** button > **Objects** tab
- **Edit > Attribute Dictionary** menu item > **New** button > **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, SailWind Layout makes all of the options in this dialog box unavailable except the System attribute option (which you would use to turn off the system attribute flag).



Objects

Field	Description
Attribute	The name of the attribute.

Field	Description
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. If you select the Via check box, the ECO Registered check box is unavailable. See also: "Assigning Attributes to Design Objects With the Object Attributes Dialog".
Hierarchy	The order in which attribute values have priority. You must select an object in the Objects list to see the Hierarchy assigned to it. The object at the top of the list has the highest priority. Clear the Use Default Hierarchy check box to change the default hierarchy of attribute values. You must select enough objects in the Objects list to activate a hierarchy. The order of the objects in the Objects list shows the default hierarchy. For example, an attribute value can be assigned to a decal, part type, and to the part (component) in the design. A part-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 at the highest possible hierarchy level becomes the assumed attribute.
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. Hierarchy modification is restricted. SailWind 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. SailWind Layout automatically places Net above Net Class in the hierarchy.
ECO Registered	Allows you to specify if the attribute is ECO registered on page 1824. If so, changes to the attribute are registered in the ECO file. You can modify attributes only when the ECO Toolbar is open (ECO mode). 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 on page 355, 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.

Field	Description
	Restriction: • 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.
System	Shows whether the attribute is a system attribute. System attributes are used by SailWind 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, SailWind Layout operation. **Note:** 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. **CAUTION:** Do not modify the properties or Attribute Dictionary entry of a system attribute. Severe program or script errors can occur.
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. 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. **Note:** 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).
Hidden	Hides the attribute, so that it is not visible or editable. It will not appear in any dialog boxes. 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).

Field	Description
	For more information, see "Attribute Hierarchy". **Note:** 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).

Related Topics

Attribute Properties Dialog Box, Types Tab

Attribute Types

Attribute Properties Dialog Box, Types Tab

To access:

- **Edit > Attribute Dictionary** menu item > select attribute > **Properties** button
- **Edit > Attribute Dictionary** menu item > **New** button

Use the **Types** tab to set the attribute type.

Description

The **Types** tab controls change depending on what you have selected. The major differences are as shown below:

Figure 128. Types Tab - Number

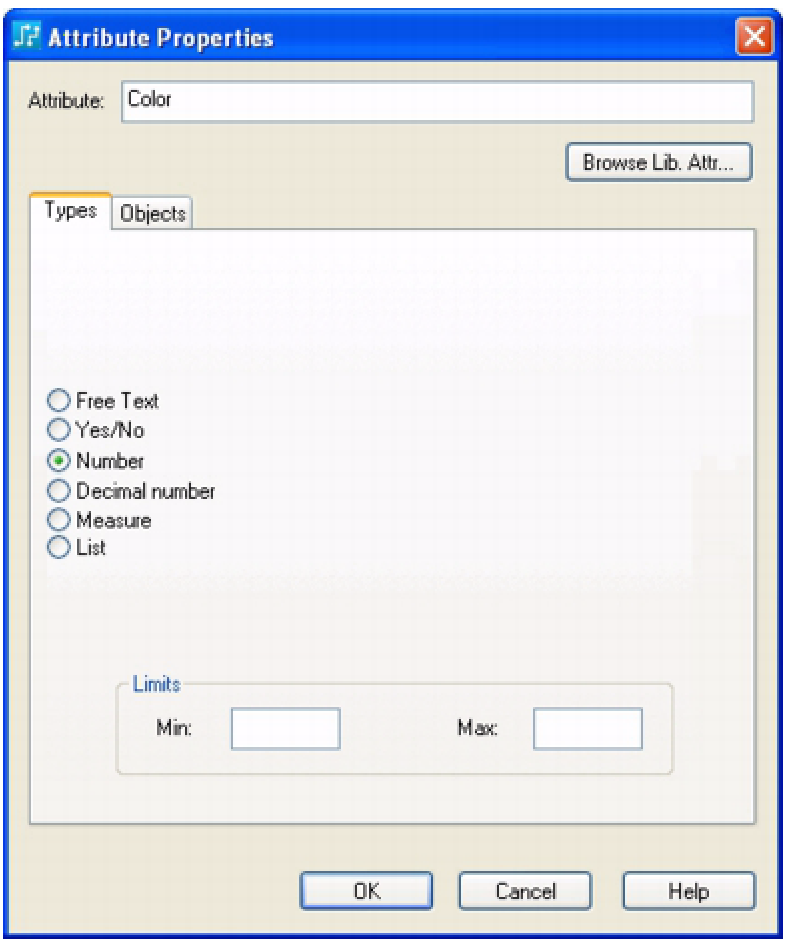


Figure 129. Types Tab - Measure

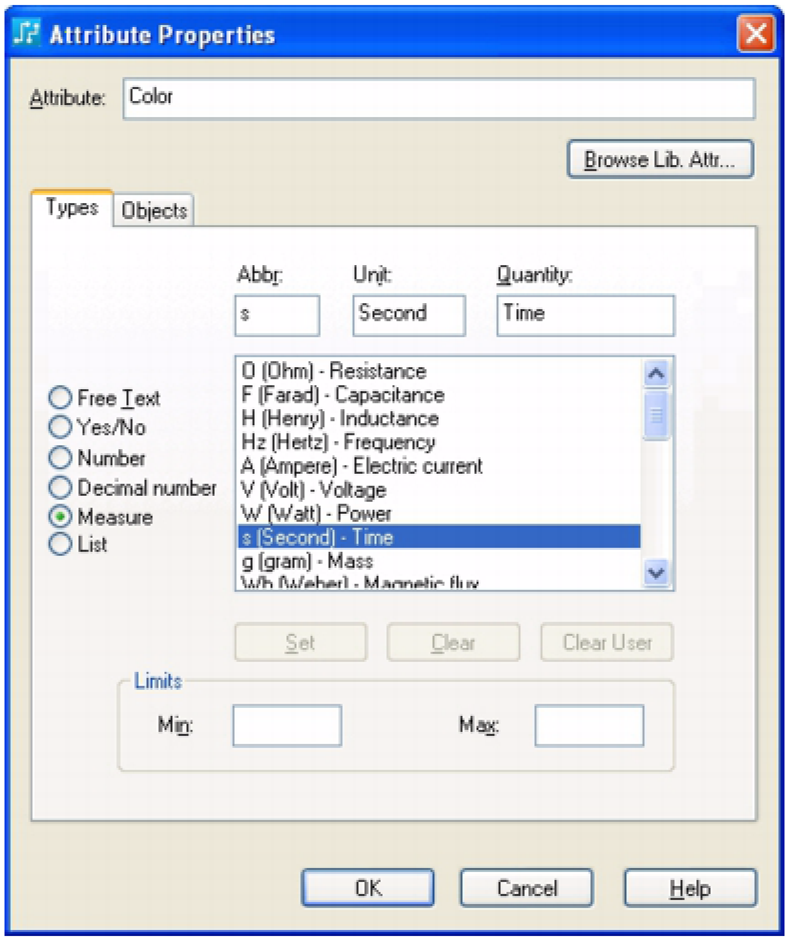
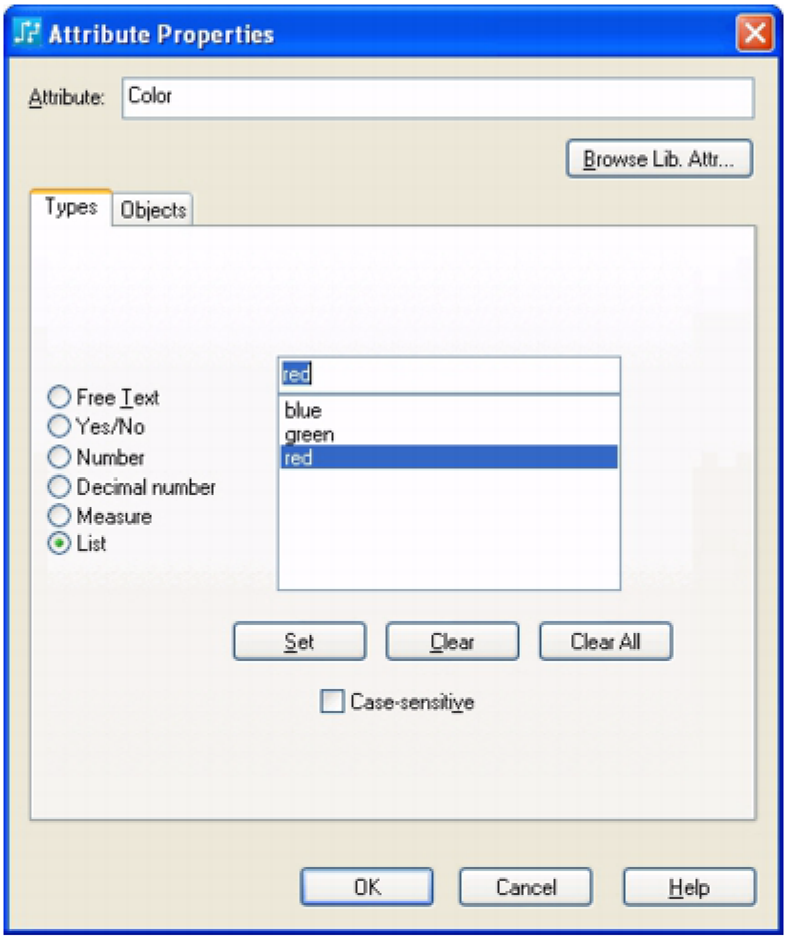


Figure 130. Types Tab - List



Objects

Field	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.

Field	Description
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.
Case-sensitive	Preserves the letter case of List entries. 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. SailWind 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

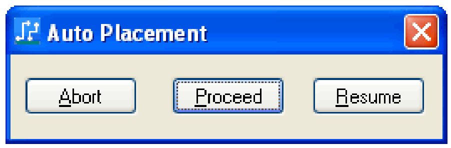
Attribute Properties Dialog Box, Objects Tab

Attribute Types

Auto Placement Prompt

To access: **Tools > Cluster Placement** menu item > **Place Clusters** button > **Run** button > **Interrupt** button

Use the Auto Placement prompt to control the interruption of the cluster placement process.



Objects

Field	Description
Abort	Quits the Cluster Placement process. You are prompted 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

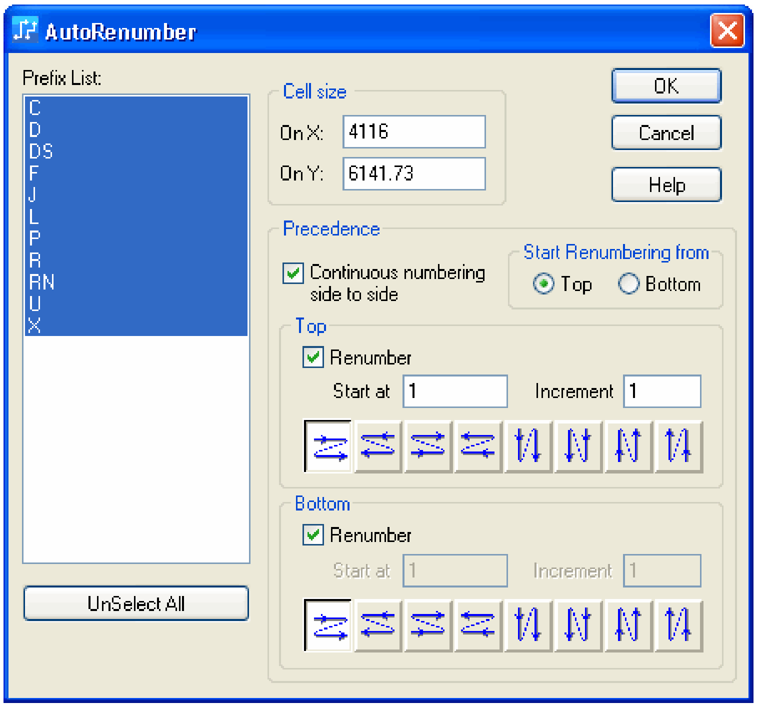
Cluster Placement Status Dialog Box

Cluster Placement Dialog Box

AutoRenumber Dialog Box

To access: On the ECO Toolbar, click Auto Renumber.

After 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.



Objects

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.

Field or Button	Description
	See also: “AutoRenumber Sweeps”.
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. 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. 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. 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

Chapter 47
GUI Reference Elements B Through C

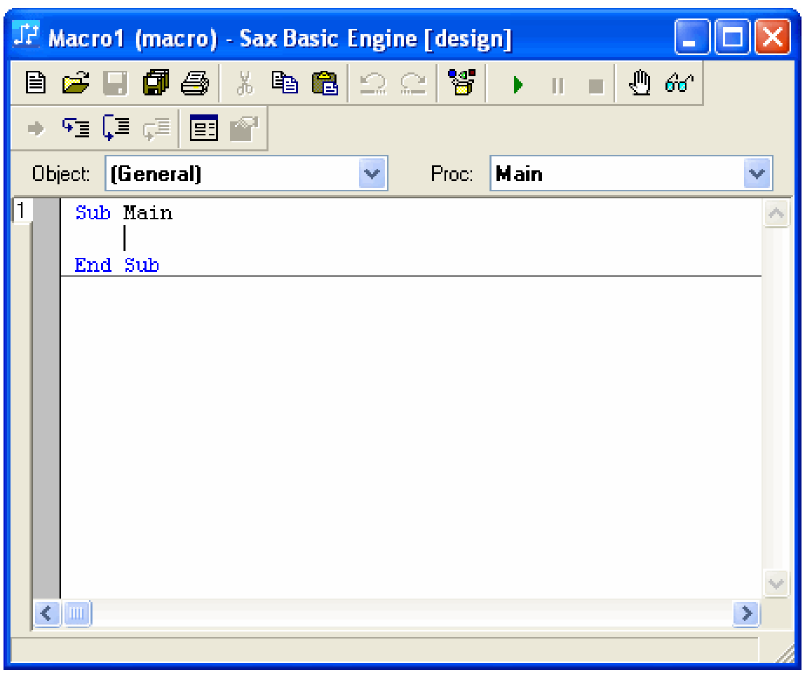
Read the sections that follow to learn more about the dialog box elements in SailWind Layout.

Basic Script Editor Dialog Box

To access: **Tools > Basic Scripts > Basic Script Editor** menu item

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.



Related Topics

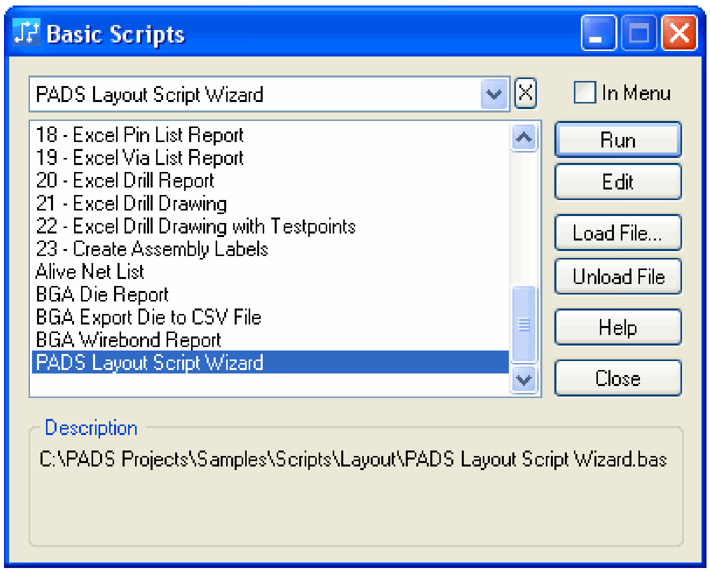
Scripting and the Automation Server [SailWind Layout Command Reference Manual]

The Macro Language [SailWind Layout Command Reference Manual]

Basic Scripts Dialog Box

To access: **Scripts > Script Manager**

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 do not use often. This dialog box runs only existing sample scripts.



Objects

Field	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 the Enter key, 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.

Field	Description
	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

To access: **BGA Toolbar > Route Wizard** button

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. When you click either Run or OK, SailWind Layout stores in the design the selections you make in the BGA Route Wizard dialog box.

Note:
This information applies only to the BGA toolkit.

Description

The Die Wizard has 4 tabs:

- "Figure 132" on page 1129
- "Figure 133" on page 1129
- "Figure 134" on page 1130
- "Figure 135" on page 1130

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.

Figure 131. BGA Route Wizard Dialog Box

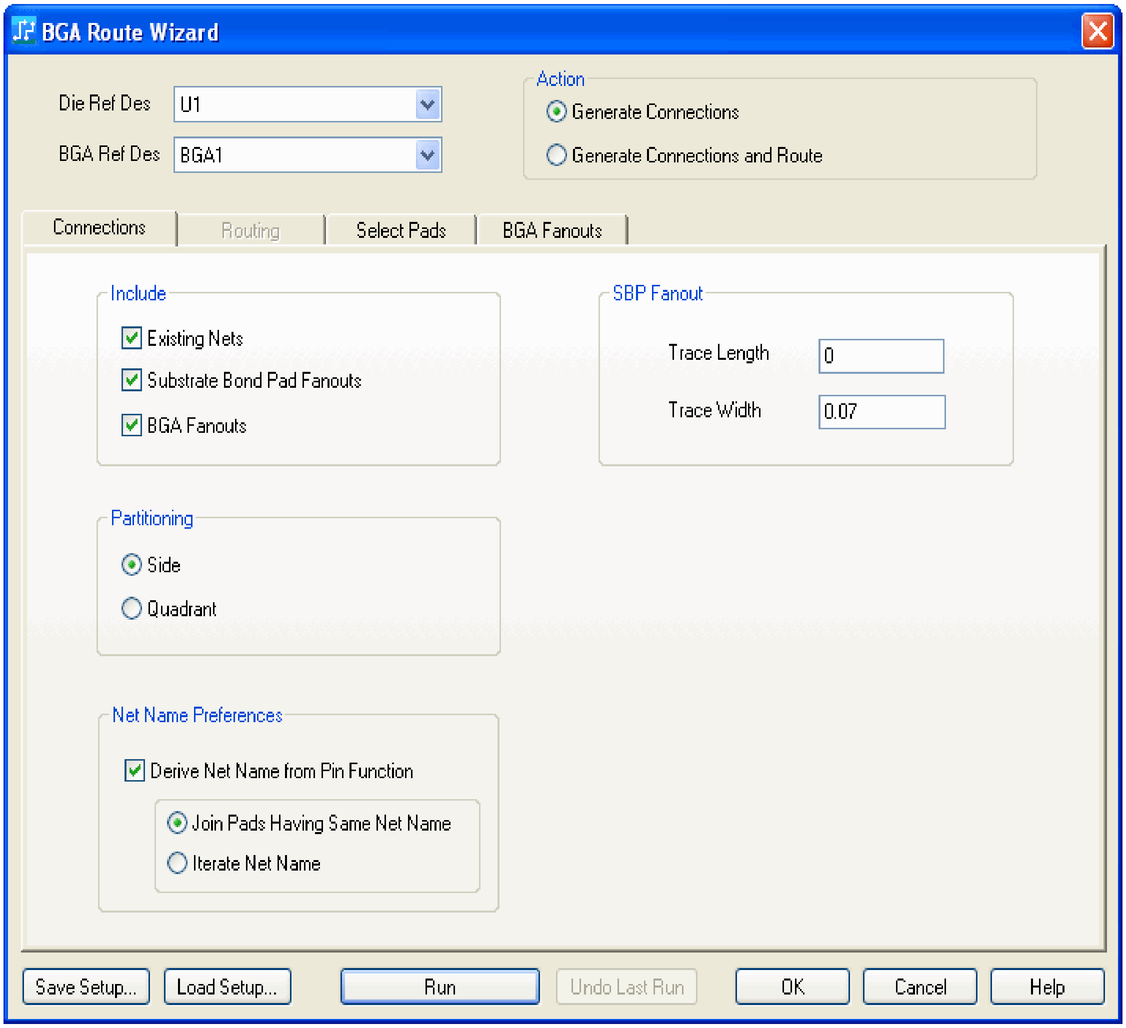


Figure 132. Connections tab

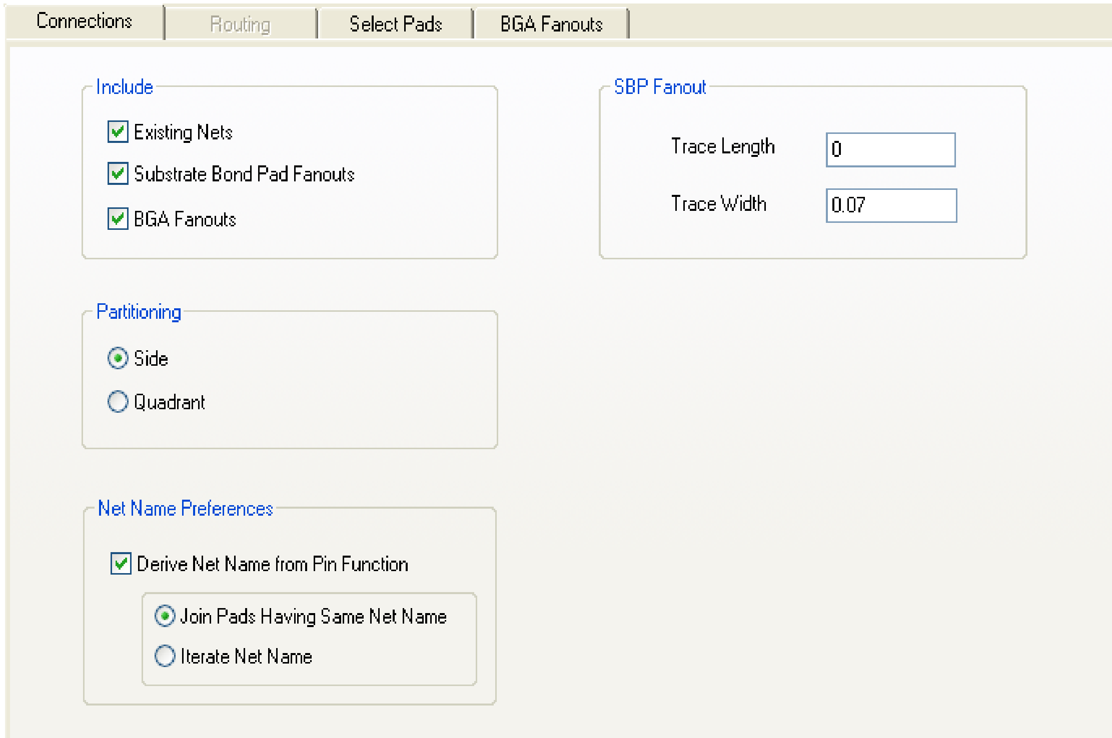


Figure 133. Routing tab

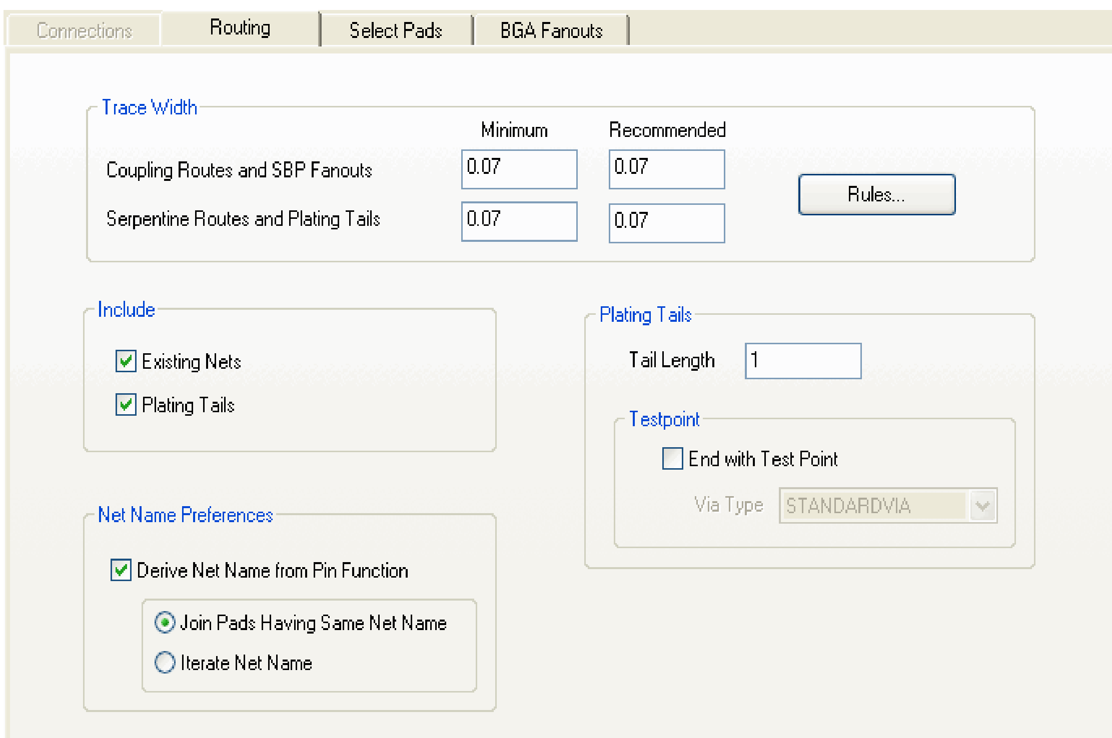


Figure 134. Select Pads tab

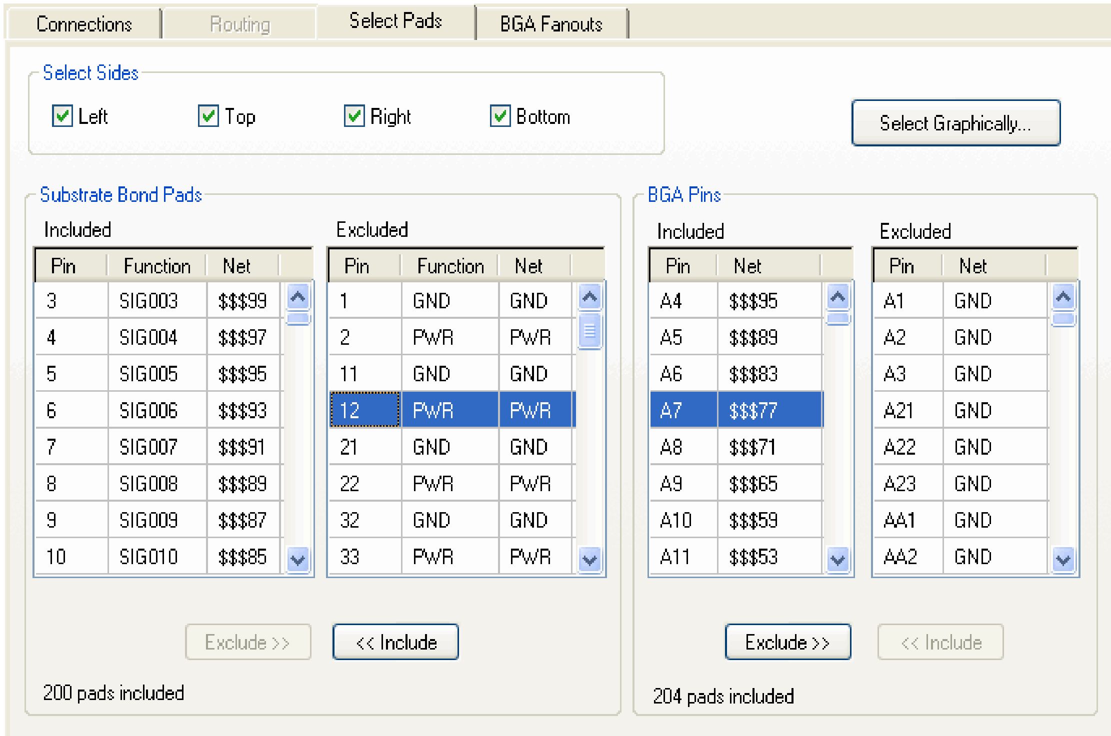
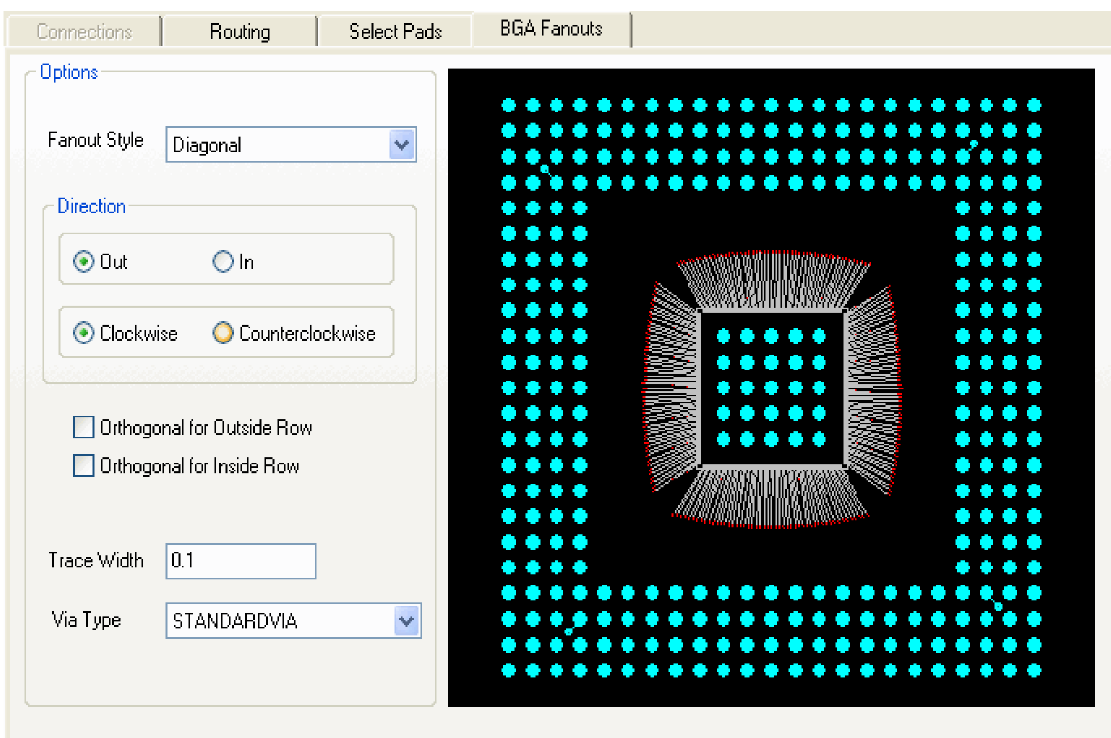


Figure 135. BGA Fanouts tab



Objects

Table 148. BGA Route Wizard Dialog Box Fields

Field	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.
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 on page 1812 and SBP fanouts on page 1857 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 on page 1823 designs only.
	  • Serpentine routes on page 1858 and plating tails on page 1849 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 on page 1809 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 on page 1127 • Routing on page 1127 • Select Pads on page 1127 • BGA Fanouts on page 1127
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 *\SailWind Projects* folder.

Table 148. BGA Route Wizard Dialog Box Fields (continued)

Field	Description
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.

Table 149. Connections tab contents

Field	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: "Die Partitioning".
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.

Table 149. Connections tab contents (continued)

Field	Description
	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”.

Table 150. Routing tab contents

Field	Description
Coupling Routes and SBP Fanouts	Sets the trace width rules during route pattern creation for SBP fanouts on page 1857 and any-angle coupling routes on page 1809. 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 on page 1858 and plating tails on page 1849. 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.
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

Table 150. Routing tab contents (continued)

Field	Description
	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 on page 1849 for serpentine traces on page 1858.
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.
Tail Length	Defines the length of the plating tail wire on page 1849 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 that displays after you run the route wizard. 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.

Table 151. 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.
Select Graphically button	Opens the Select Graphically Dialog Box.
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.
BGS Pins list	Lists the BGA pins 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.
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.

Table 152. BGA Fanout tab contents

Field	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.

Table 152. BGA Fanout tab contents (continued)

Field	Description
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 for this design using **Setup > Pad Stacks**.
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

The BGA Route Wizard

Creating a Wire Bond Fanout

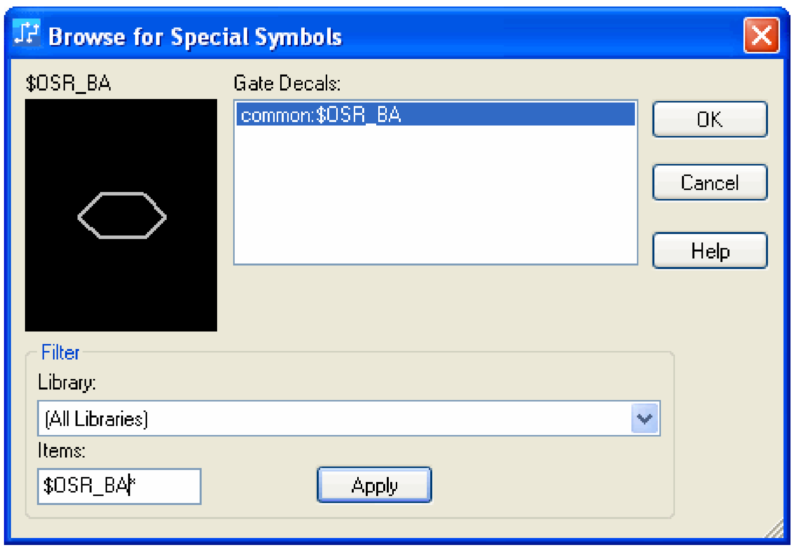
Browse for Special Symbols Dialog Box

To access:

- **File > Library** menu item > select a Library > **Parts** button > **New**
- **File > Library** menu item > select a Library > **Parts** button > select a connector **Edit** button > **Connector** tab > double-click Special Symbol cell > **Browse** button

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 SailWind Logic.

Figure 136. Browse for Special Symbols Dialog Box



Objects

Field	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. 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

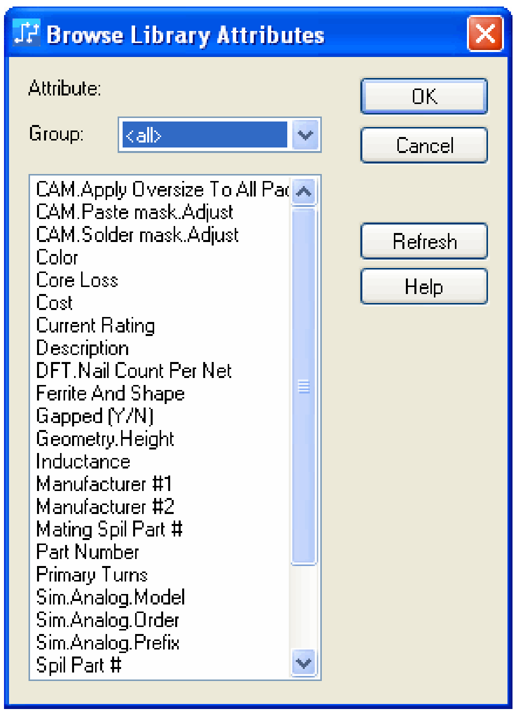
Assigning Special Symbols to a Connector

Browse Library Attributes Dialog Box

To access:

- **File > Library** menu item > Library Manager dialog box > **Attr Manager** button > **Browse Lib. Attr** button
- **File > Library** menu item > Library Manager dialog box > **Attr Manager** button > **Add Attr** button > **Browse Lib. Attr** button
- Attribute Properties dialog box > **Browse Lib. Attr.** button

Use the Browse Library Attributes dialog box to select an attribute from a list of all part and decal attributes available to the design.



Objects

Field	Description
Attribute	Displays the selected attribute.
Group	Filters the attribute list. (Includes structured attributes on page 1862.)
Refresh	Manually updates the attribute list if you change the list of libraries in the Library Manager.

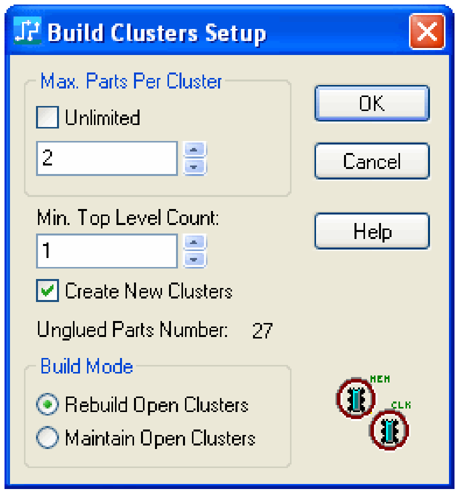
Related Topics

Renaming Attributes of Library Items

Build Clusters Setup Dialog Box

To access: **Tools > Cluster Placement** menu item > **Build Clusters** button**> Setup** button

Use the Build Clusters Setup dialog box to create new clusters from parts and unions. Creation is based on connectivity.



Objects

Field	Description
Unlimited	Specifies unrestricted parts per cluster.
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.

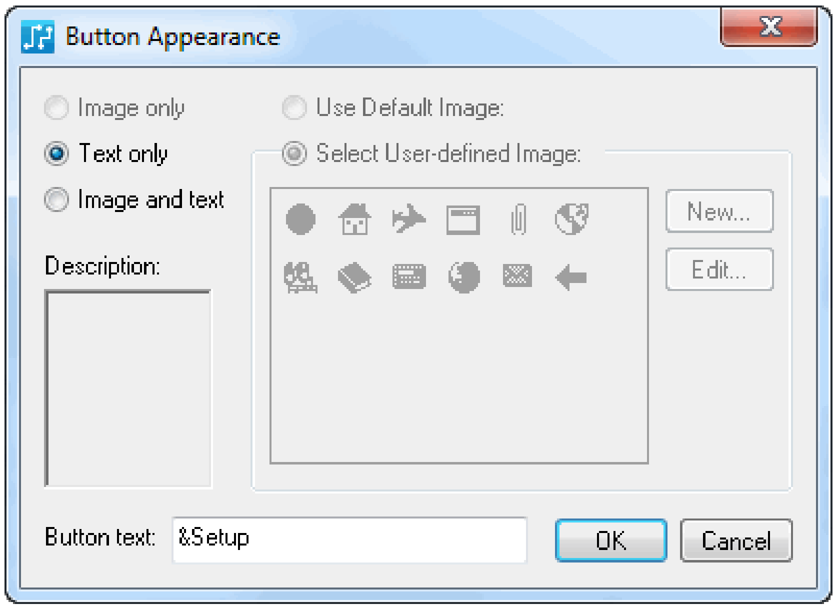
Related Topics

Automatic Placement

Button Appearance Dialog Box

To access: **Tools > Customize** menu item. In the Customize dialog box, right-click on a menu or menu item in SailWind Layout and click the **Button Appearance** popup menu item.

Use the Button Appearance dialog box to customize the appearance of menu items.



Objects

Field	Description
Image only, Text only, Image and text	For menu items, choose what to display in the menu. Images appear in the gutter to the left of the command name. Applying images for menu items does not function for the menu name but only menu items.
Description	This field is not available.
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.
Button Text	Type the name for the menu item. Use the “&” character to the left of the letter you want to use with the Alt button to select menu items without the mouse and cursor.

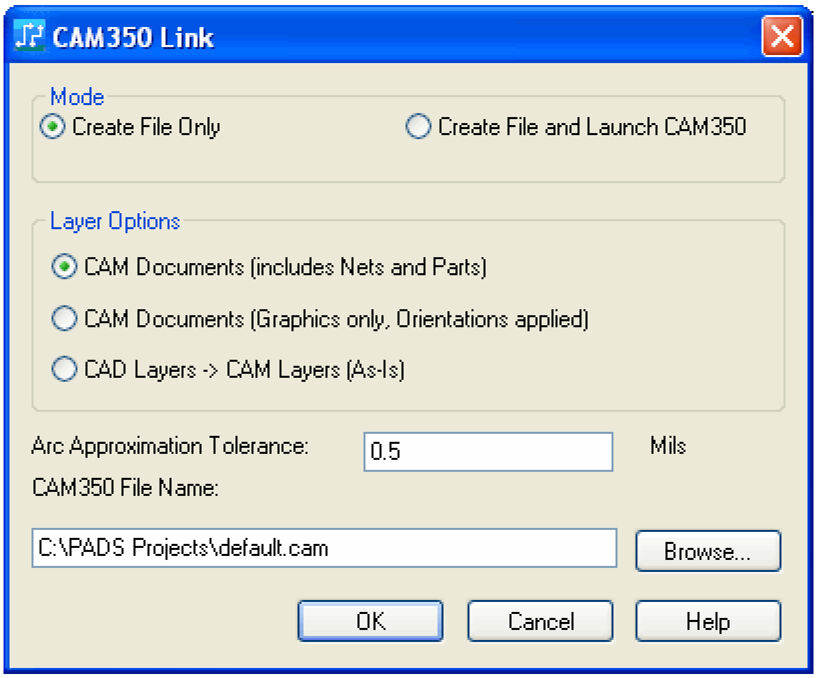
Related Topics

Customize Dialog Box, Commands Tab

CAM350 Link Dialog Box

To access: **File > Export** menu item > select CAM350 Files > **Save**

Use the CAM350 dialog box to set options for, and to generate, a *.cam* output file.



Objects

Field	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 SailWind 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.

Field	Description
	Note: **Requirement:** Before using this option, flood all copper planes. Also, on the **Tools > Options** menu item > **Copper Planes** category > **Hatch and Flood** subcategory, in the Save to PCB File area, click “All copper 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 SailWind 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 SailWind Layout and the precision for SailWind 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 SailWind Layout will be interpreted by CAM350. • **CAD Layers -> CAM Layers (As-Is)** — Translates no CAM documents. Translates CAD layers as defined in the SailWind 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 planes. Also, on the **Tools > Options** menu item > **Copper Planes** category > **Hatch and Flood** subcategory, in the Save to PCB File area, click “All copper 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

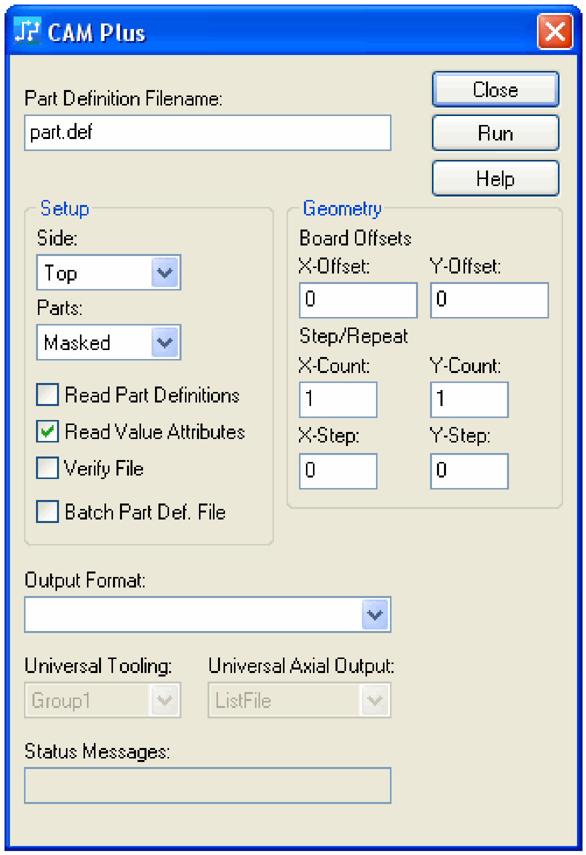
Exporting to CAM350

Annotating DFF Errors

CAM Plus Dialog Box

To access: **File > CAM Plus**

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*.



Objects

Field	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. For more information, see “Batch Mode and Masked Mode” on page 978. **Tip:** Masked parts are those that are assigned to the machine selected format as an insert class.

Field	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 SailWind 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 \SailWind 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 SailWind 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 more information, see "Batch Mode and Masked Mode" on page 978. 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. The default is 0 offset, equivalent to treating the CAD system origin as the origin of the machine. Another way to think of this is that the X and Y offset are the distance from the location dowel of the machine to the SailWind Layout system origin. If the location dowel is in the lower left corner, this will be a positive value.
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.

Field	Description
	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 whether or not their insert class is defined as belonging to the specific machine. When an output is performed, the resulting file or files are stored in a program-created folder under *\SailWind Projects\cam*. The subfolder matches the design's file name. For example, if the design loaded when CAM Plus is started is test.pcb, the results are stored in *\SailWind Projects\cam\test*.
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

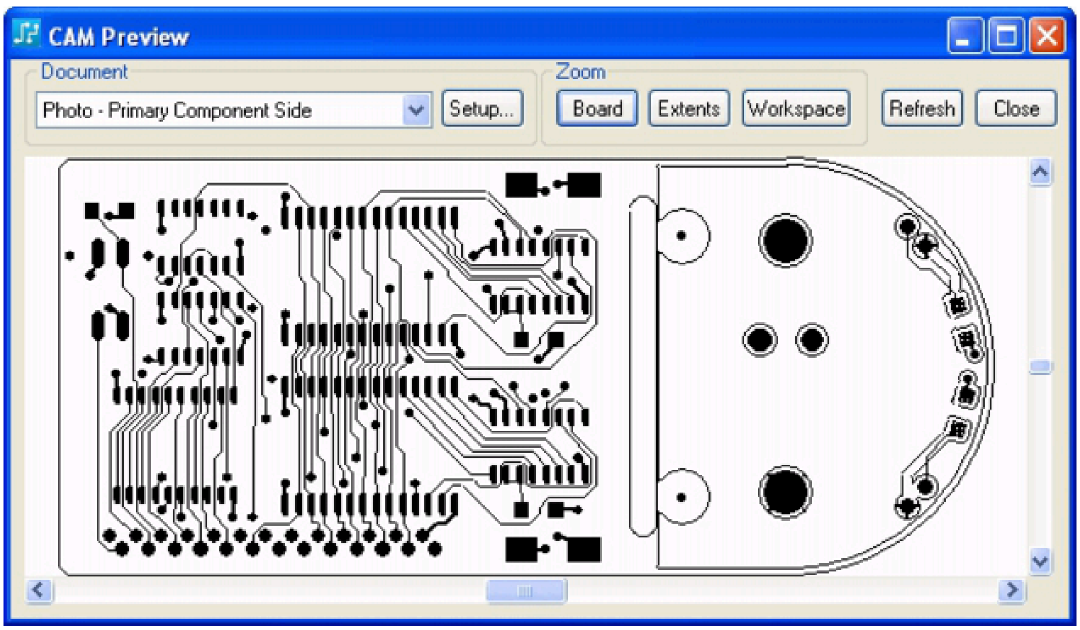
CAM Plus Assembly Machine Interface

CAM Preview Dialog Box

To access:

- **File > CAM** menu item > **Preview** button
- **File > CAM** menu item > **Add** button > **Preview Selections**
- **File > CAM** menu item > **Add** button > **Layers** > **Preview** button

Use the CAM Preview dialog box to preview your CAM Documents.



Objects

Field	Description
Document list	Lists the available documents to preview.
Setup	Opens the “CAM Preview Setup Dialog Box” on page 1152.
Zoom area — In addition to the buttons of this area, you can zoom using 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.	

Field	Description
Board	Fits the contents of the selected document into the preview window.
Extents	Fits the world, or the whole coordinate system, into the preview window.
Workspace	Fits the current screen view into the preview window.
Refresh	Refreshes the view from changes in the design or 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.

Related Topics

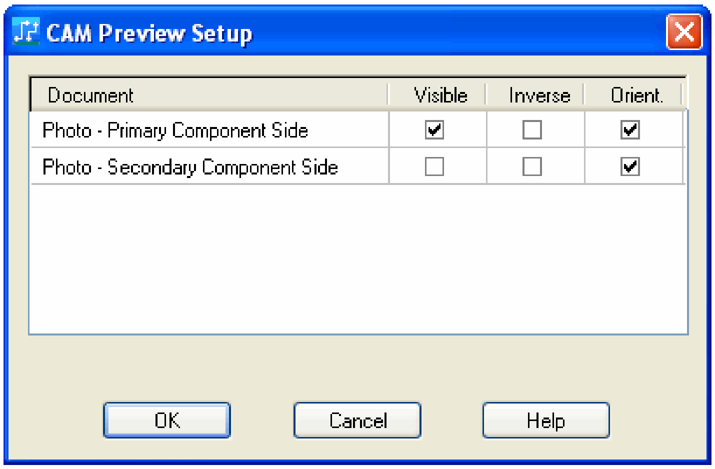
Creating CAM Outputs to Manufacture Your PCB

CAM Preview Setup Dialog Box

To access: **File > CAM** menu item > select a document > **Preview** button > **Setup** button

Use the CAM Preview Setup dialog box to invert, show plot orientation, or overlay multiple CAM Documents. If desired, you can change the preview attributes of all documents.

Tip
You must have at least one CAM document defined to use preview setup.



Objects

Field	Description
Document	Displays the document name.
Visible	Specifies the documents you want to see; you can overlay multiple CAM documents. This requires that you have already set up other 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

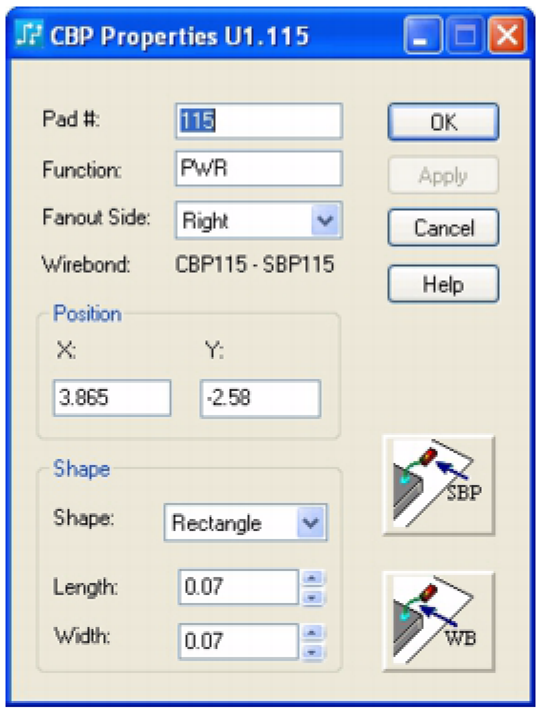
CAM Preview

CBP Properties Dialog Box

To access: **BGA Toolbar** button > **Wire Bond Editor** button > Select a CBP > right-click > **Properties** popup menu item

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.

Note:
This information applies only to the BGA toolkit.



Objects

Field	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.

Field	Description
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.

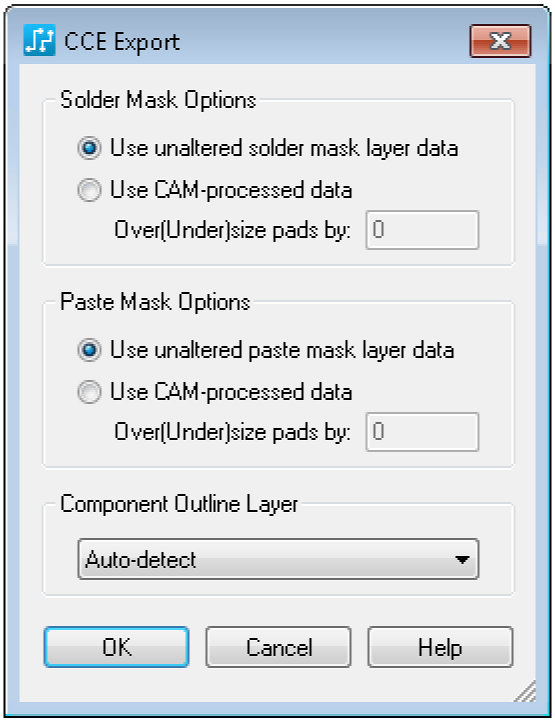
Related Topics

Editing Component Bond Pad Properties

CCE Export Dialog Box

To access: **File > Export** menu item > select CCE Files > **Save**

Using the CCE Export dialog box, you can export design elements to CAMCAD, and visECAD.



Objects

Field	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, including any 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. • **Over(Under)size pads by** — Type a value to apply a global oversize or undersize to all solder mask pads. This global setting is the lowest of a hierarchy of possible values. For more information, see "Control of Solder Mask and Paste Mask".

Field	Description
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. • **Use CAM-processed data** — Select this option to accept automatically-generated pads copied from the associated top or bottom layer, including any 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. • **Over(Under)size pads by** — Type a value to apply a global oversize or undersize to all paste mask pads. This global setting is the lowest of a hierarchy of possible values. For more information, see "Control of Solder Mask and Paste Mask".
Component Outline Layer	Choose a layer to use as the highest priority layer when searching for component outline shapes for each decal to export. Only layers marked as Enabled in the "Enable/Disable Layers Dialog Box" on page 1365 are available. If you choose the Auto-detect item, the following search order is applied to locate a component outline: 1. Checks if the *SailWindpcb.ini* option for the placement outline selection exists ([Graphics] 3D_Shape_Layer) and tries that layer. 2. Tries PADS layer 25 (recommended layer for 3D outline). 3. Tries PADS assembly top layer. 4. Tries PADS top layer. 5. Tries PADS layer 0 ("All Layers" layer setting). 6. Tries PADS layer 20 (recommended layer for placement outlines). If no closed shapes are found, goes to next search. 7. If no closed shapes are found, uses an artificial extents box around all component pins. 8. If no pins exist, uses an artificial extents box around all footprint elements (coppers, lines, and text). 9. If nothing is found, uses an artificial tiny box for empty footprints. Found or generated outline shapes for a specific component are placed on PLACEMENT_OUTLINE_TOP or PLACEMENT_OUTLINE_BOTTOM (special layers in the CCE file) depending on the side of the board where the component is placed. The CCE Export report file lists the layer used for each decal when using Auto-Detect. When a specific layer is selected on the export dialog, the report only lists decals and source layers that do not match the requested layer.

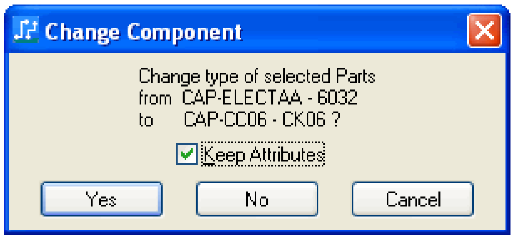
Related Topics

Exporting CCE Files

Change Component Dialog Box

To access: select a component > ECO Toolbar > **Change Component** button > right-click > **Library Browse** popup menu item > select a part type > **Replace**

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.



Objects

Field	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

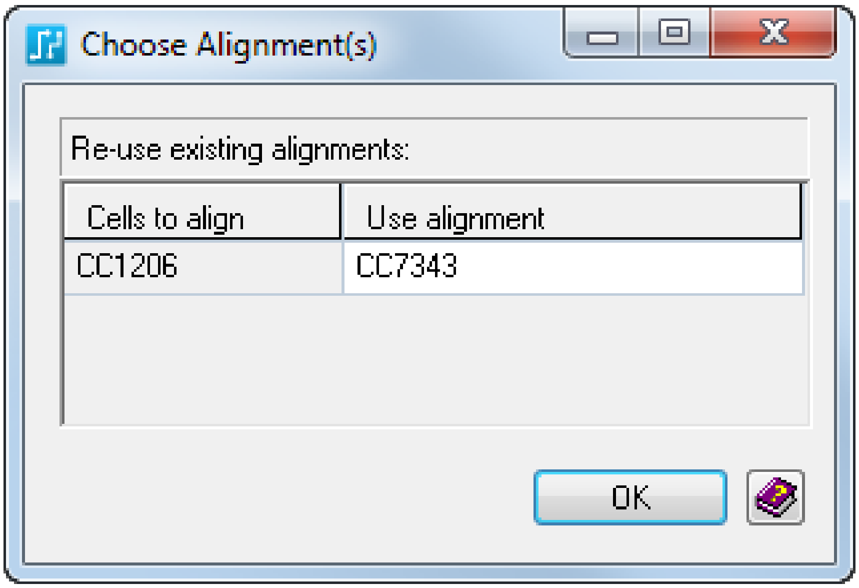
Updating a Part Type from the Library in ECO Mode

Choose Alignments Dialog Box

To access: From the Manage Mappings dialog box, select a part from the “Parts without model” list and assign it to a 3D model in the “Parts mapped to models” list; then click **Yes** at the prompt.

Use the Choose Alignments dialog box to select an existing alignment pattern for a 3D model that you want to reuse with another part.

A part selected for a model assignment will initially align the 3D model in accordance with the alignment information in the part’s decal. You can use the Choose Alignments dialog box to accept or reject the pre-established decal alignment.



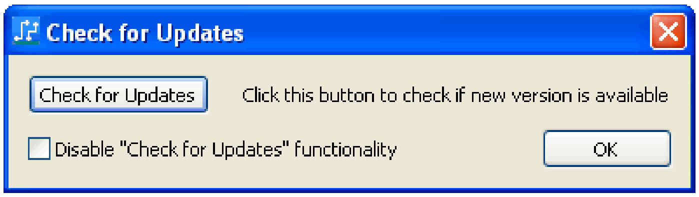
Objects

Field	Description
Cells to align	Displays the name of the part decal that can reuse a model alignment.
Use alignment	Displays a list of decal model alignments that can be reused. For example, if you want to assign the 3D model for a capacitor to a resistor (because the 3D models appear the same), and the 3D model of the capacitor is currently used with decal CC0805, the decal name of “CC0805” appears in this field. You can use the alignment of the current decal or select **None** from the dropdown list to align the 3D model manually.

Check for Updates Dialog Box

To access: **Help > Check for Updates** menu item

Use the Check for Updates dialog box to manually check for a new version of PADS, and to disable or enable automatic checks.



Objects

Field	Description
Check for Updates button	Manually checks for a new version of the SailWind software.
Disable “Check for Updates” functionality	Determines if SailWind automatically checks for a new version of the software. Click to stop SailWind from automatically checking for a new version of the SailWind products; click to clear to have SailWind automatically check for a new version.

Related Topics

Check for (Software) Updates

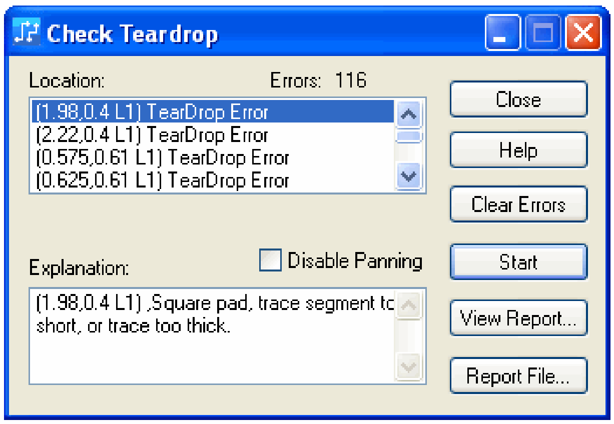
Check Teardrop Dialog Box

To access: **Tools > Options** menu item > **Teardrops** tab > **Check** button

Use the Check Teardrops dialog box to check for and view teardrop errors. SailWind Layout reports the error location, layer, and a short explanation of the error.

Description

You must enable teardrops before you can check them. **Tools > Options** menu item > **Routing** tab > select Generate Teardrops check box > click **OK**.



Objects

Field	Description
Location list	The location of the teardrop error. Click an error to pan to it in the design.
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 only the error markers in the design. Does not fix the errors.
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. This changes the default name of the report file until you change it.

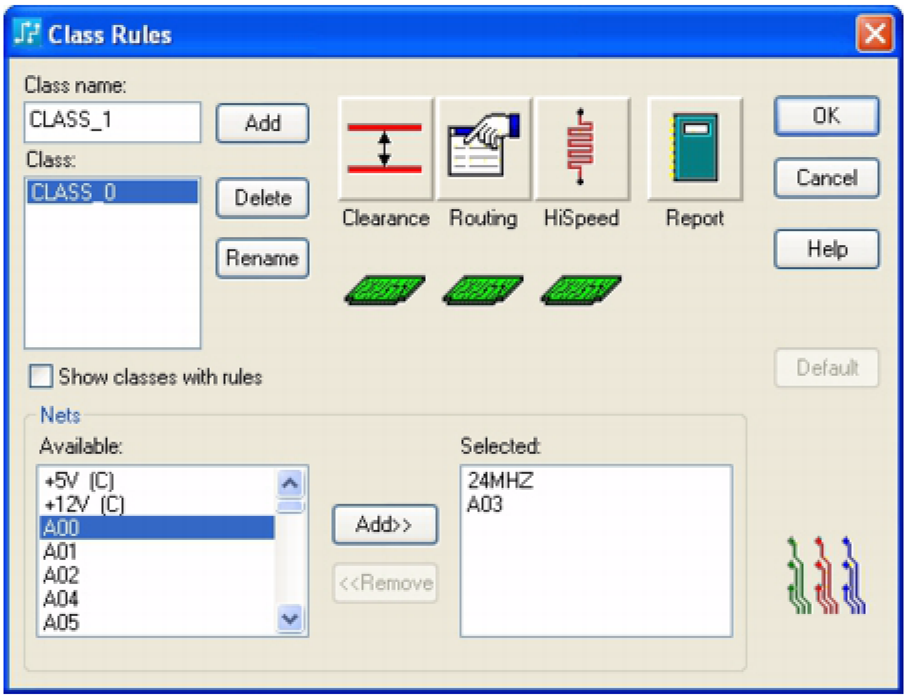
Related Topics

Modifying Net Properties

Class Rules Dialog Box

To access: **Setup > Design Rules** menu item > **Class** button

Use the Class Rules dialog box to create and manage classes of nets and to define design rules that apply to them.



Objects

Field	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.

Field	Description
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.
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

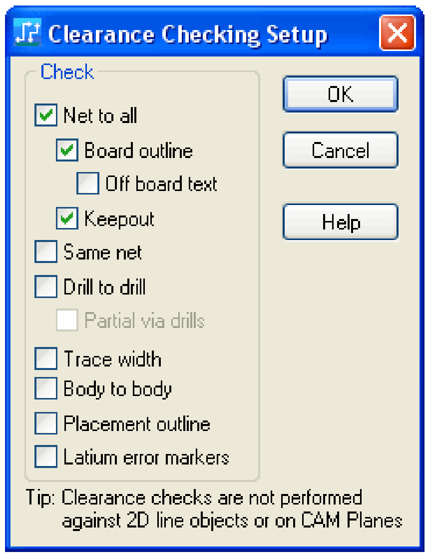
Displaying the Nets of a Class Design Rule

Design Rule Hierarchy

Clearance Checking Setup Dialog Box

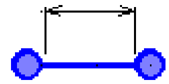
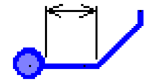
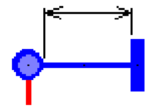
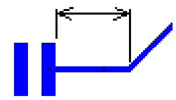
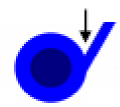
To access: **Tools > Verify Design** menu item > Clearance check > **Setup** button

Use the Clearance Checking Setup dialog box to specify which clearances to check during a Clearance verification.



Objects

Field	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. **Note:** **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.
Same Net	Checks clearances between objects along the same net. Object to object checking includes the checks listed below:

Field	Description
	• 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.
Body to body	Checks component body outline against component body outline. **CAUTION:** The Verify Design “Body to body” check does not check component outlines against TH or SMD pads. **Tip** When used in combination with the Board Outline check, you can check component outlines against the board outline. The component body outline is the furthest extent of any 2D line object on the following layers.

Field	Description
	• Component layer (top or bottom) • Associated silkscreen layer for component layer (silkscreen top or bottom) • All Layers/Layer 0 (zero) 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: "Nudging Overlapping Parts on page 500".
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 on page 1775 dialog box.

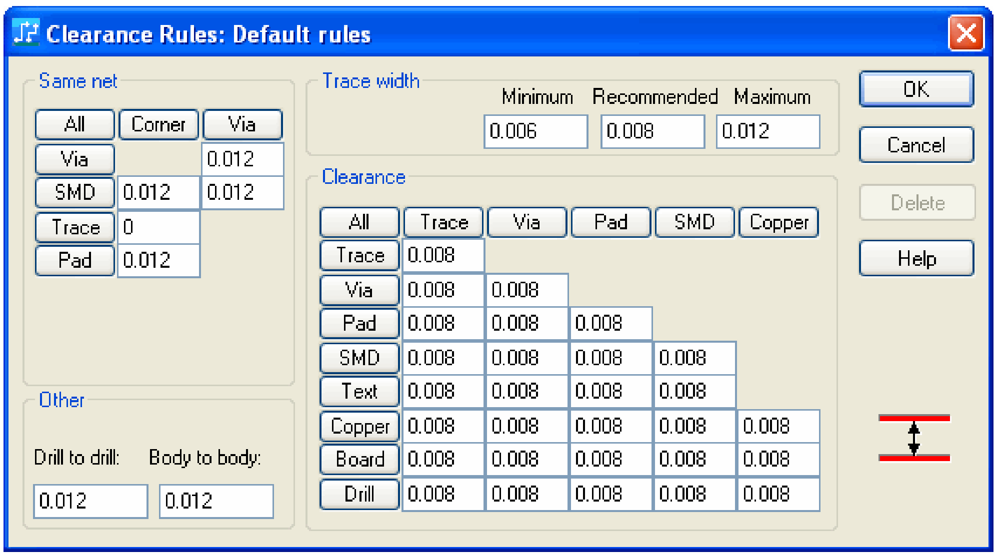
Related Topics

Setting Up Clearance Checking

Clearance Rules Dialog Box

To access: **Setup > Design Rules** menu item > choose a hierarchy level > **Clearance** button

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 verifications check and report any clearance violations. SailWind Layout checks object types against other object types as well as against themselves (such as traces). Values display in current design units.



Objects

Area	Description
Same net	Specifies the space that must be maintained between items that are of the same net. • **SMD** — surface mount pad • **SMD-to-via** — value is also used for SMD-to-SMD and SMD-to-Pad rules • **Via-to-via** — value is also used for VIA-to-PAD and PAD-to-PAD • **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 SailWind 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>
<tr><th>Area</th><th>Description</th></tr>
<tr><td>Trace width</td><td>Specifies the allowed range of trace widths.
• Minimum — Minimum width for interactive routing
• Recommended — Width to use when routing begins; except in SailWind Router where the width is automatically adjusted if the pin pitch is too small and using the recommended width would violate the trace to pad clearance.
• 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.</td></tr>
<tr><td>Clearance</td><td>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
• Via — also applies to jumper pins and virtual pins
• The Copper to Via, Pad, SMD or Drill values are also used to create the default thermals and anti-pads.
See also: “Design Rule Versus Pad Stack - Thermals and Antipads” on page 797.
Note:
• 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.</td></tr>
<tr><td>Other</td><td>Specifies the clearances between additional design objects:
• Drill to drill — Specifies the spacing required between drills
• Body to body — Specifies the spacing required between the component body outlines.
Note:
The spacing between component bodies is measured from the edges of the lines used to define the component body outlines.</td></tr>
<tr><td>Delete button</td><td>Removes non-default clearance rules at the current level of the rules hierarchy.
Restriction:
You cannot delete the Default clearance rules.</td></tr>
<tr><td>OK button</td><td>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</td></tr>
</table>

Area	Description
	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".

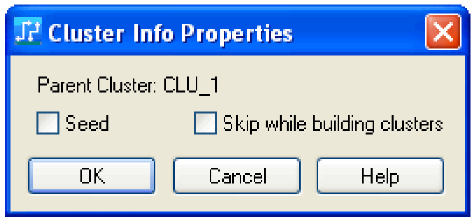
Related Topics

Design Rule Hierarchy

Cluster Information Properties Dialog Box

To access: Select a union within a cluster > right-click > **Properties** popup menu item > **Cluster Info** button

The Cluster Information Properties dialog box modifies cluster attributes for the selected component or union that belongs to a cluster.



Objects

Field	Description
Seed	Identifies a cluster as a base, or top level cluster, during build and grow operations. SailWind 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.

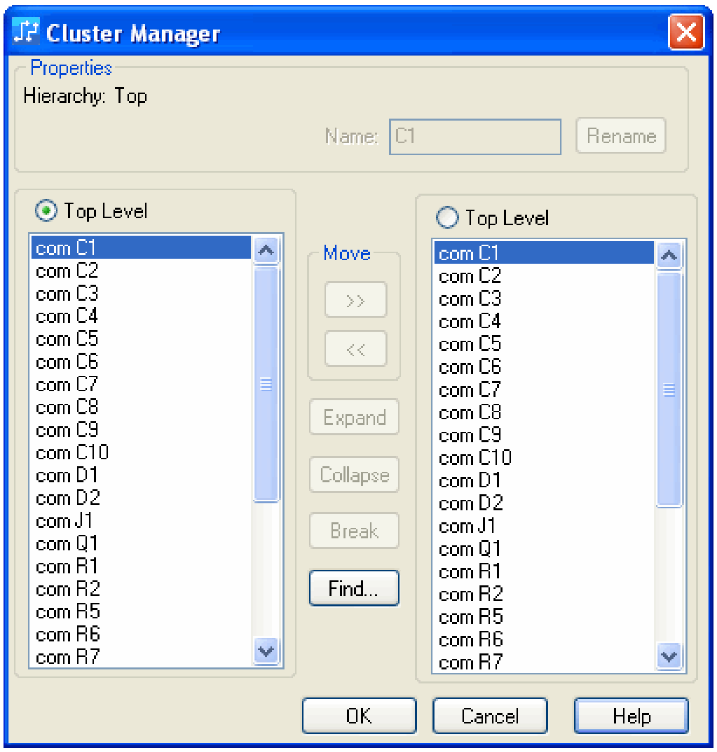
Related Topics

Union Properties Dialog Box

Cluster Manager Dialog Box

To access: **Tools > Cluster Manager** menu item

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.



Objects

Field	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:

Field	Description
	• Clusters — CLU • Unions — UNI • Components — com
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

Automatic Placement

Cluster Placement

Cluster Placement Dialog Box

To access: **Tools > Cluster Placement** menu item

Use the Cluster Placement dialog box to automatically build new clusters, place clusters within the board outline, and place parts within the board outline.

Objects

Field	Description
Build Clusters	Select to create new clusters outside the board outline. Created clusters default to open. Click the Build Clusters Setup button to customize the clusters in the Build Clusters Setup Dialog Box. (Unavailable till you click the Build Clusters button.)
Place Clusters	Select to position clusters within the board outline. Cluster members are also moved during this process, whose placement may differ as follows. You can see them if you use Disperse in Cluster View Mode or exit Cluster View Mode. • If any cluster member is glued, the unglued is placed around the glued one, maintaining their relative layout. A random one will be selected if more than one glued object is in the cluster. • If no glued component is in the cluster, all cluster members are placed where the cluster locates, maintaining their relative layout. Click the Place Clusters Setup button to customize the placement of clusters in the Place Clusters Setup Dialog Box. (Unavailable till you click the Place Clusters button.)
Place Parts	Select to perform automatic placement operations.

Field	Description
	Click the Place Parts Setup button to customize placing parts in the Place Parts Setup Dialog Box. (Unavailable till you click the Place Parts button.)
Run	Click to initiate the operations for all selected function buttons; for example, you can select all three buttons to build clusters, place clusters, and place parts in one pass. Also displays the Cluster Placement Status Dialog Box, in which you can interrupt the ongoing operations. (This button is unavailable until you select at least one of the three function buttons.)
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.
Nudge	Runs a Nudge pass in automatic mode, automatically adjusting parts that fail Clearance Checking. **Tip** • 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

Automatic Placement

Cluster Placement

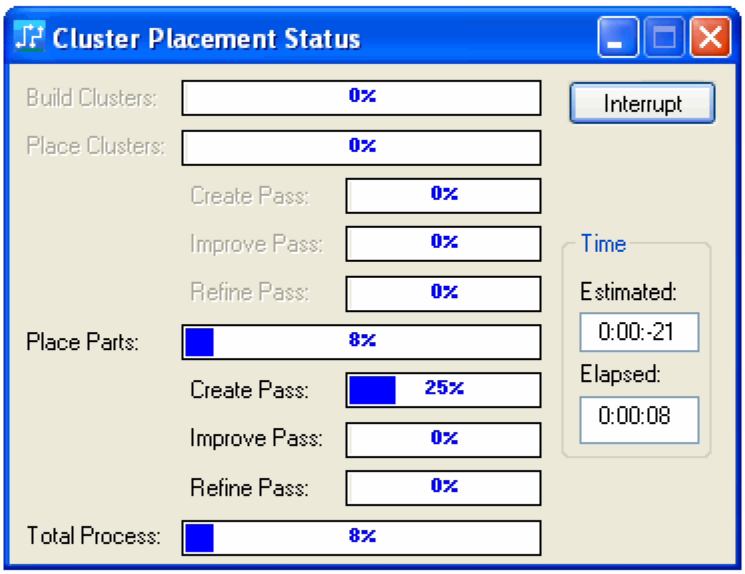
Cluster Placement Status Dialog Box

To access: This dialog box is displayed when you use the Cluster Placement dialog box to automatically build new clusters, place clusters within the board outline, and place parts within the board outline.

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.

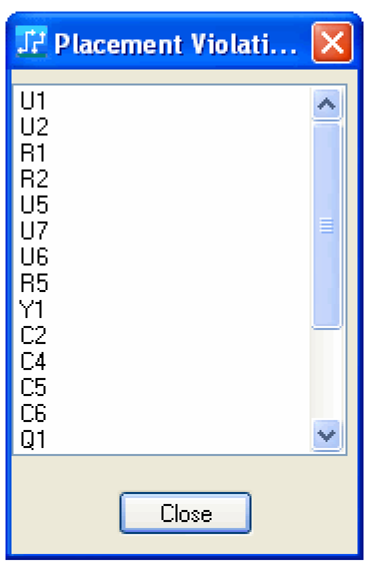
Description

Figure 137. Cluster Placement Status Dialog Box



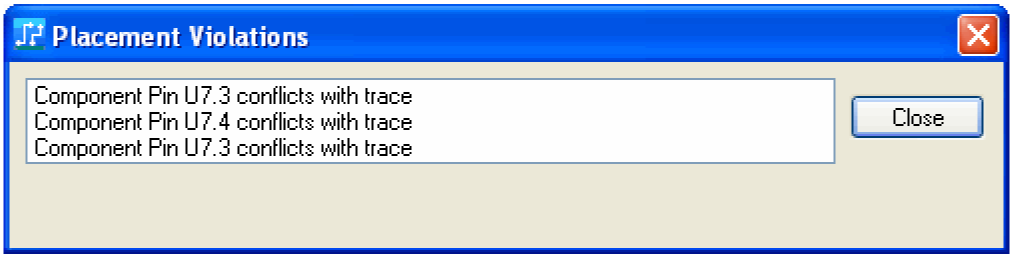
After completion of placement operations, errors appear in the Placement Violations dialog box.

Figure 138. Placement Violations Dialog Box



Double-click an item in the list for information.

Figure 139. Placement Violations Info Dialog Box



Objects

Table 153. Cluster Placement Status Dialog Box Fields

Field	Description
Interrupt	Pauses the placement process and opens the Auto Placement Prompt on page 1119.

Related Topics

Automatic Placement

Cluster Placement

Cluster Properties Dialog Box

To access: Select a cluster > right-click > **Properties**

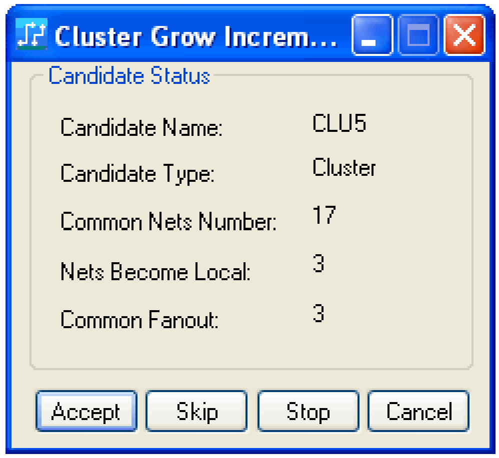
Use the Cluster Properties dialog box for information on a selected cluster and to make modifications.

Description

Figure 140. Cluster Properties Dialog Box

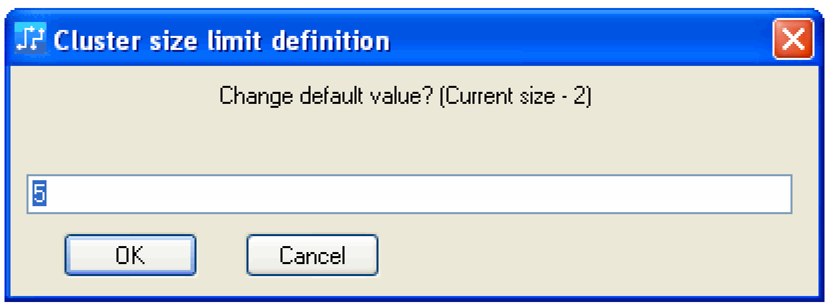
Press the Incremental button to open the Cluster Grow Incremental dialog box. Use it to cycle through other clusters in the design and select which ones to absorb into the current cluster.

Figure 141. Cluster Grow Incremental Dialog Box



Press the Automatic button to open the Cluster Size Limit Definition dialog box. Use it to automatically add new members to the cluster based on a max number.

Figure 142. Cluster Size Limit Definition Dialog Box



Objects

Field	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.

Field	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 on page 1177 to incrementally add new members to the cluster. See also: “Modifying Existing Clusters”.
Automatic	Opens the Cluster Size Limit Definition dialog box on page 1177 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. SailWind 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.

Related Topics

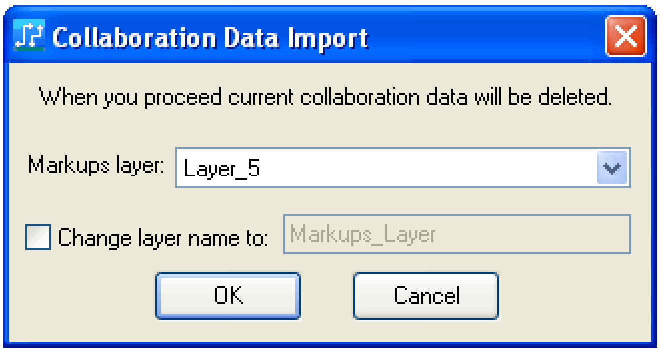
Automatic Placement

Cluster Placement

Collaboration Data Import Dialog Box

To access: In the Markups Dialog Box, click the **Import** button, browse to and select a file, then click **Open**

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.



Objects

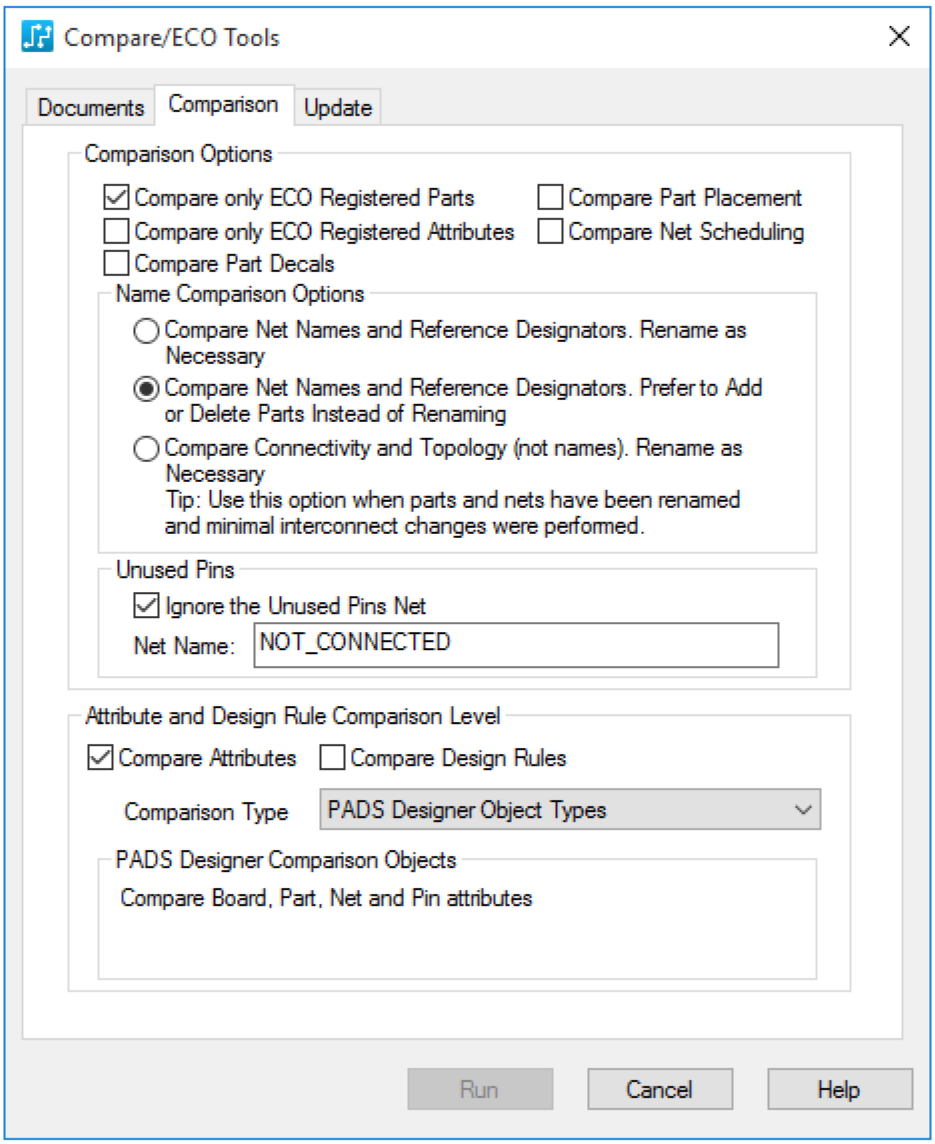
Field	Description
Markups Layer	Select a documentation layer from the list, as the location for any 2D line markups that might be included in the collaboration data.
Change layer name to	Select the check box to automatically change the name of the layer as listed in the Layers Setup. You can accept the default layer name or type a new one.

Compare/ECO Tools Dialog Box, Comparison Tab

To access: **Tools > Compare/ECO Tools** menu item > **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, SailWind Layout ignores the reuse definition on page 1854 and uses the actual elements in the physical design reuse.



Objects

Field	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.	
Compare only ECO Registered Parts	Comparison excludes non-ECO-registered parts on page 1824. 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 on page 1824. **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 SailWind Router. For more information, see "Rescheduling Nets" in the SailWind 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. • **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.
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. 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.

Field	Description
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 Table 154 table.
Comparison Objects area	Lists what you have selected in the Attribute and Design Rule Comparison Level area
Run	Compares the designs. Available when you select an option in the Output Options area on the **Documents** tab.

Table 154.

Select	With	To Compare
Compare Attributes	SailWind Logic Object Types Select this option to compare SailWind 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.
	PADS Designer objects Select this option to compare PADS Designer 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 Select this option when you are comparing different versions of a SailWind 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	SailWind Logic Object Types	Net and net class rules
	PADS Designer 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	SailWind Logic Object Types	• Part and net attributes • Net and Net Class rules • General and Conditional rules • Differential Pairs rules

Table 154. (continued)

Select	With	To Compare
	PADS Designer 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

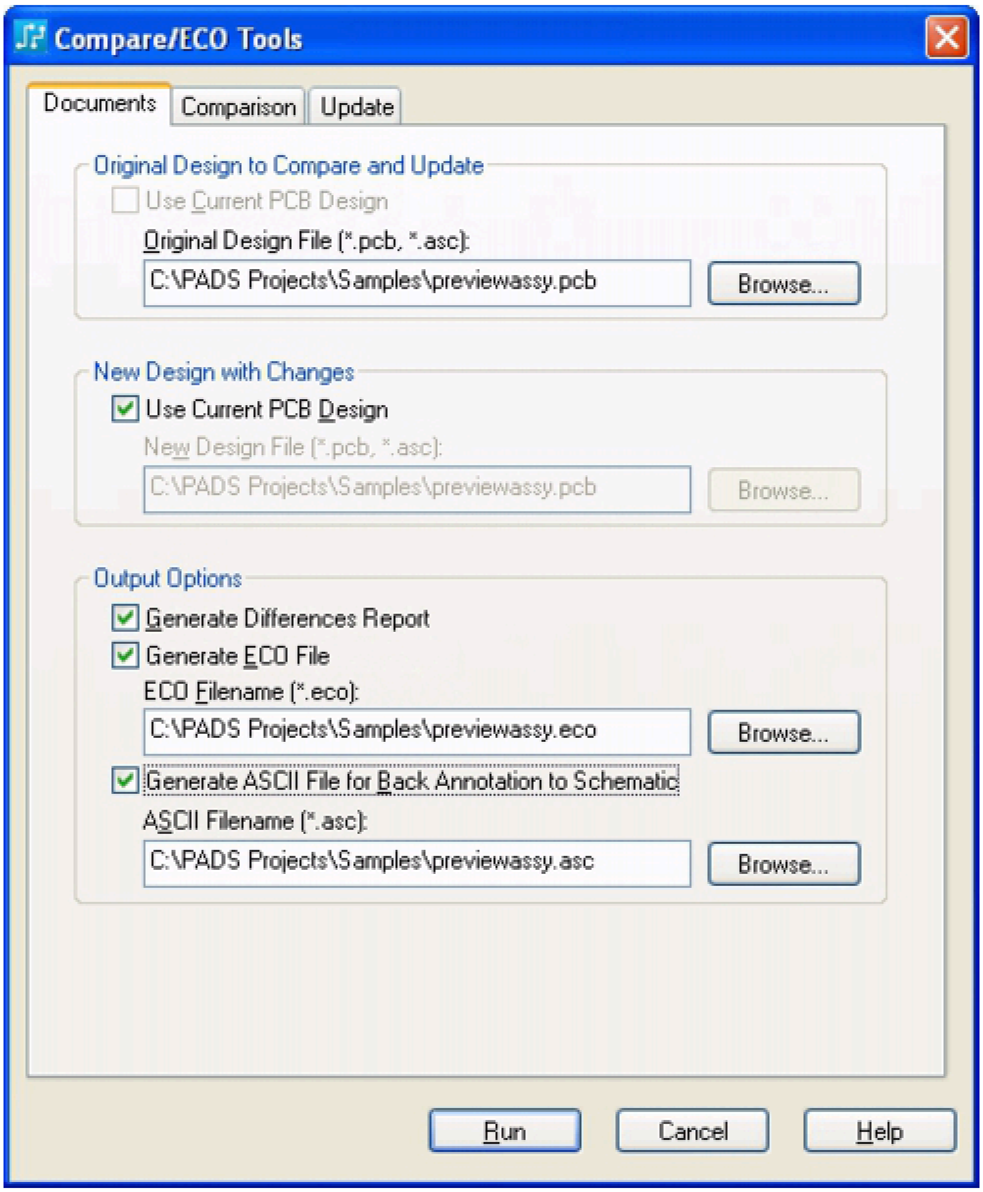
Comparing Designs

Working with SailWind Logic

Compare/ECO Tools Dialog Box, Documents Tab

To access: **Tools > Compare/ECO** menu item

Use the **Documents** tab on the Compare/ECO Tools dialog box to specify the designs to compare and the files to create.



Objects

Field	Description
Original Design to Compare and Update	• **Use Current PCB Design** — Specifies to use the current design as the original to compare and update. Clear Use Current PCB Design to browse for another one. • **Original Design File** — Specifies to browse for another design to use as the original to compare and update.
New Design with Changes	• **Use Current PCB Design** — Specifies to use the current design as the new design with changes. Clear Use Current PCB Design to browse for another one. • **New Design File** — Specifies to browse for another design to use as the new design with changes.
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 *\SailWind Projects* folder.
Generate ECO File	Specifies to create an ECO on page 1824 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. Available when you select an option in the Output Options area.

Related Topics

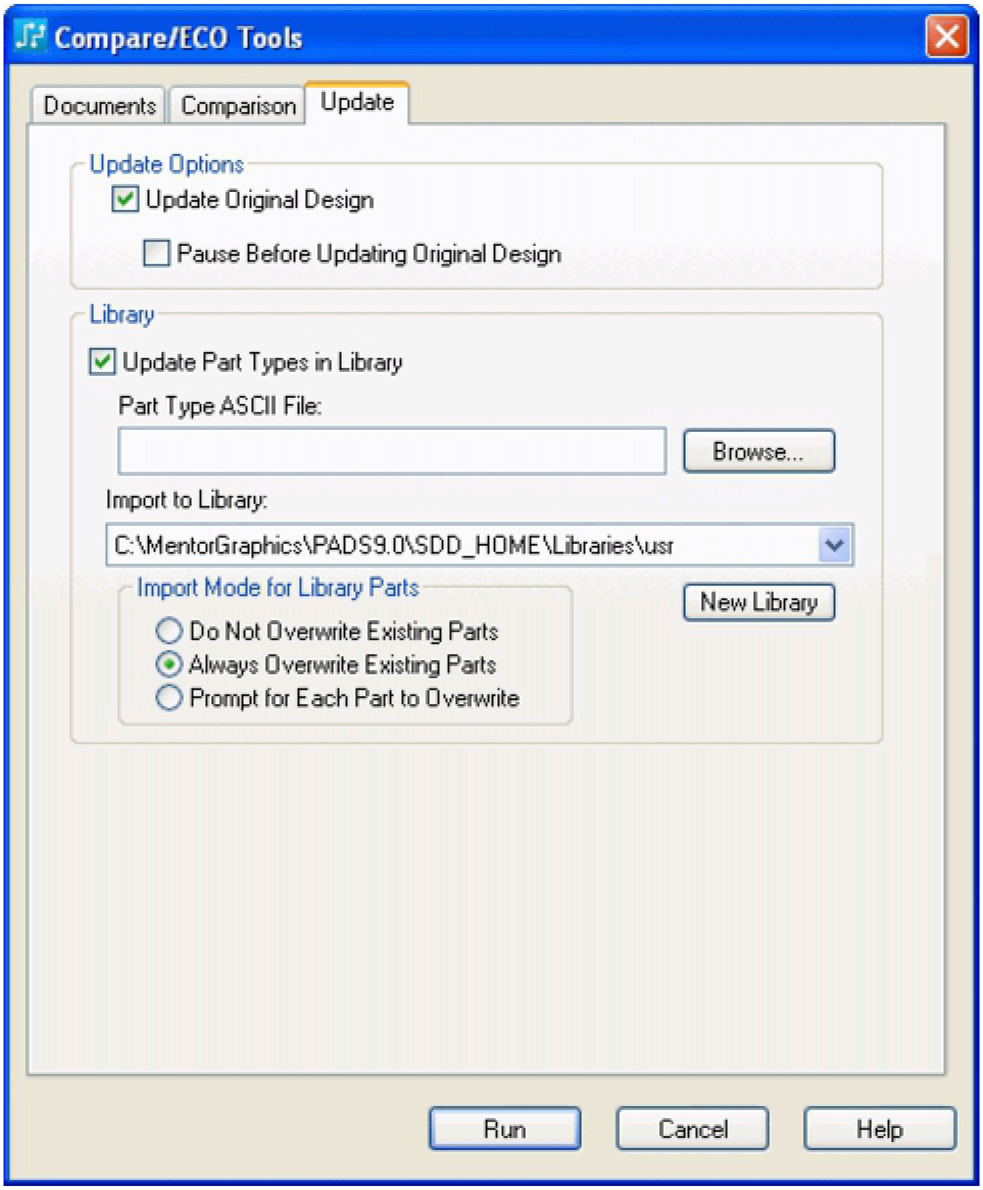
Comparing Designs

Working with SailWind Logic

Compare/ECO Tools Dialog Box, Update Tab

To access: **Tools > Compare/ECO** menu item > **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.



Objects

Field	Description
Update Original Design	Specifies to update the PCB layout in memory to match the new design by automatically importing into SailWind Layout the ECO file resulting from design comparison. This check box 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 SailWind Layout design in memory. 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 SailWind Layout part type library by automatically importing the PADS-format ASCII part type file (*.p*) generated with ViewPCB in PADS Designer.
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. Available when you select an option in the Output Options area on the **Documents** tab.

Related Topics

Comparing Designs

Working with SailWind Logic

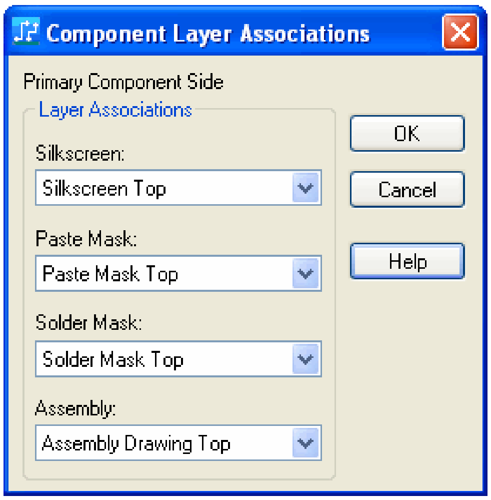
Component Layer Associations Dialog Box

To access: **Setup > Layer Definition** menu item > **Associations** button

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.

Description

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.



Objects

Field	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
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:

Field	Description
	• 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

To access:

- Select a component > right-click > **Properties**
- Select a component > Standard Toolbar > **Properties** button
- Select a component > press the Ctrl-Q (or Alt-Enter) keys

Use the Component Properties dialog box to modify component placement, decal, cluster, label and electrical net information.

Description

There are three tabs that change the content displayed in the dialog box:

- Use the **Part** tab in the Component Properties dialog box to view all information about the part. This is not editable.
- Use the **Cluster** tab in the Component Properties dialog box to exclude clusters or set one as a base cluster.
- Use the **Labels** tab in the Component Properties dialog box to specify the label you want to create or edit, and to open the Part Label Properties dialog box, where you can set the label properties.

Use the Electrical Nets area to make electrical nets settings.

Note:
This area only appears if you have the Advanced Rules license feature. For more information, see "Electrical Nets" on page 337.

Figure 143. Component Properties Dialog Box

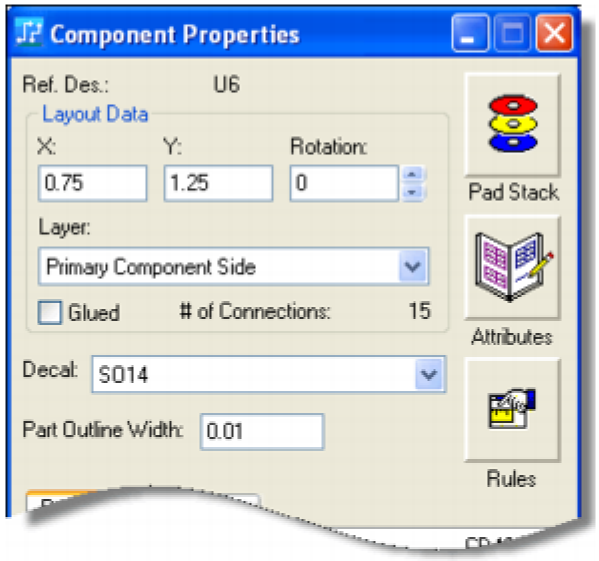


Figure 144. Part Tab

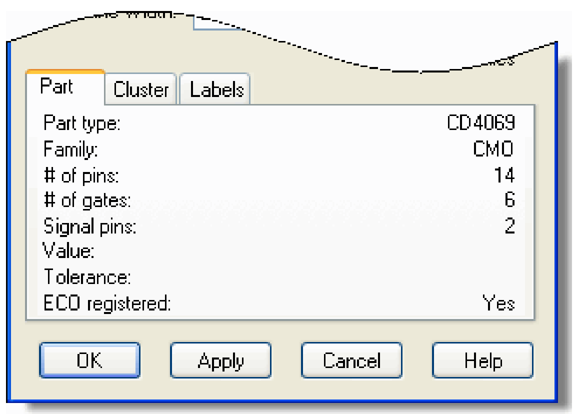


Figure 145. Cluster Tab

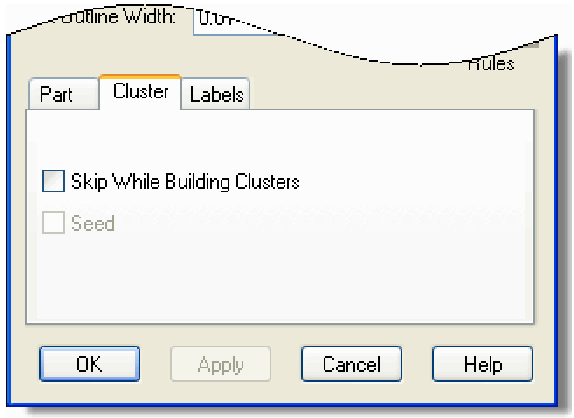


Figure 146. Labels Tab

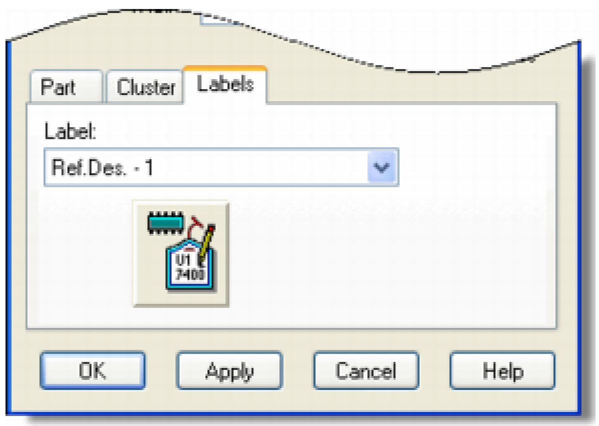
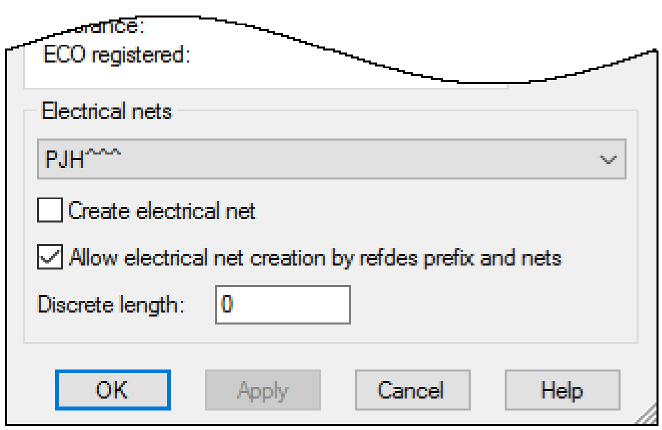


Figure 147. Electrical Nets Area



Objects

Table 155. Component Properties Dialog Box

Field	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 on page 1824. This allows attribute changes even if the attribute is ECO-registered on page 1824. 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. SailWind Layout only changes the part outline widths on page 1846 of selected components' decals, not the widths of all decals in the design. See also: "Changing the Part Outline Width".
Pad Stack button	Opens the component in the Pad Stack Properties dialog box on page 1566.
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".

Table 156. Cluster Tab Contents

Field	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. SailWind Layout searches outward from seed clusters to identify other clusters that you can add to the seed cluster.

Table 157. Labels Tab Contents

Field	Description
Label list	Specifies the label you want to create or edit.
Part Label Properties	Opens the Part Label Properties Dialog Box.

Table 158. Electrical Nets Area Contents

Field	Description
Electrical nets list	Names of the electrical nets attached to or going through the component. An electrical net name has a suffix of three caret symbols.

Create electrical net	Select the check box to create an electrical net with the component's nets. This check box is also selected by the Create Electrical Net command for selected components.
	Restriction:
	• If a net of a selected component is excluded from electrical nets, it cannot be included in an electrical net.
	(A component or net is excluded from electrical nets if its Create electrical net and Allow electrical net creation by components check boxes are both cleared.)
	• If a selected component has more than two pins, the following conditions must apply, or the component cannot create an electrical net, that is, the electrical net cannot go through the component:
	  • All pins must connect to a gate.
	  • Each gate must have exactly two pins.
	Clear the check box to remove the component from an electrical net, that is, to prevent an electrical net from going through it.
	This check box is also cleared by the Disable Electrical Net Creation popup command for selected components.

Table 158. Electrical Nets Area Contents (continued)

Field	Description
	Restriction: If the Create electrical net check box in the Net Properties dialog box is selected for the component's nets, you must also clear the Allow electrical net creation by refdes prefix and nets check box. Clear both check boxes to prevent the selected components from creating electrical nets (that is, to prevent any electrical net from going through them).
Allow electrical net creation by refdes prefix and nets	Select the check box to allow the component's nets to be included in electrical nets automatically by specifying the component's refdes prefix in the Electrical Nets dialog box, or by selecting the Create electrical net check box in the Net Properties dialog box. Clear the check box to prevent electrical net creation of the selected components' nets through those components by net or by refdes prefix. The nets can still be made into electrical nets by the other components they are attached to. This check box is also cleared by the Disable Electrical Net Creation popup command for selected components. **Restriction:** Components that have the Create electrical net check box selected are not prevented from being added to an electrical net by clearing this check box. Clear both check boxes to prevent the selected components from being added to an electrical net (that is, to prevent any electrical net from going through them).
Discrete length	Enter the length to be used for this component when calculating pin pair and net lengths. Half the Discrete length value is added to each connected pin pair, and net length measurement. You can add a value to represent the length of the component pin that will solder to the pad. For electrical nets, it gives a more accurate total length measurement where physical nets are chained together through discrete components. Without a Discrete length value, lengths of connected pin pairs and nets are measured only up to the origin of the pad. This may not be the geometric center of the pad in offset pads.

Related Topics

Modifying Component Properties

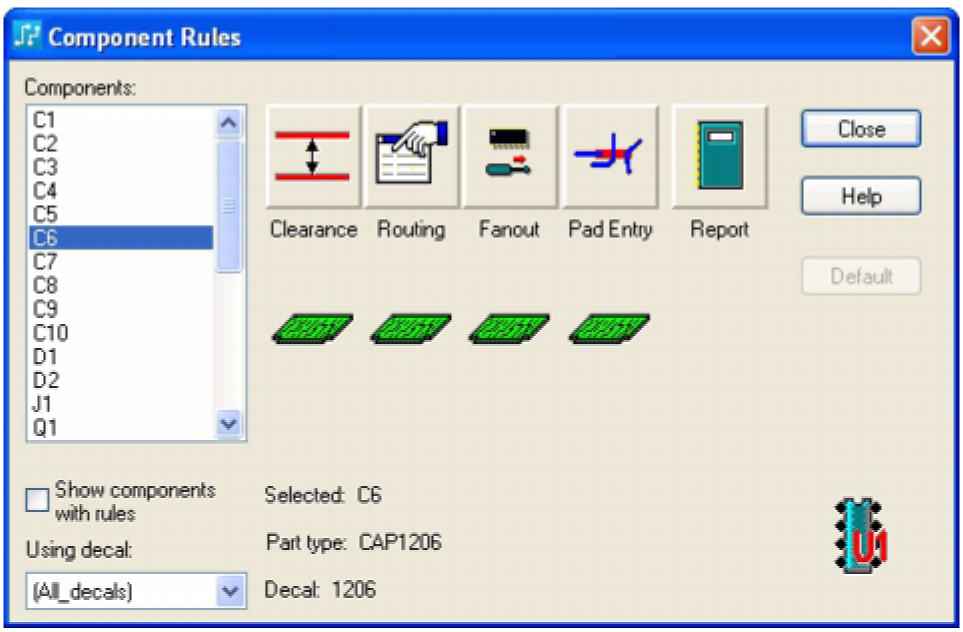
Component Rules Dialog Box

To access:

- **Setup > Design Rules** menu item > **Component** button
- Select a component > right-click > **Show Rules**

Use the Component Rules dialog box to define design rules that apply to components.

Tip
You can define Component Rules in SailWind Layout; however, these rules are used in SailWind Router only.



Objects

Field	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. Select (All_decals) to display all decals.

Field	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 components selected in the Components list.
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

Resetting Component Rules to Default Rules

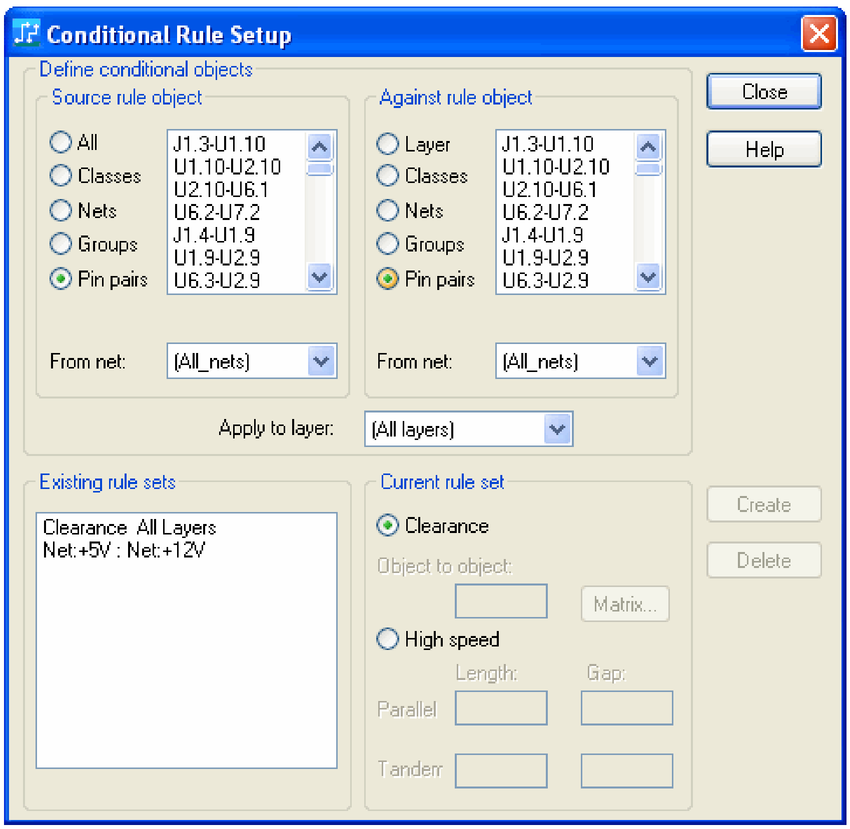
Design Rule Hierarchy

Conditional Rule Setup Dialog Box

To access: **Setup > Design Rules** menu item > **Conditional Rules** button

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.

Note:
The Advanced Rules option is required to setup conditional design rules.



Objects

Field	Description
Source rule object	Specifies the object for which to set the rule.
From net	Specifies to display pin pairs for a specific net. Select (All_nets) to display all pin pairs. 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.

Field	Description
	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. Select (All_nets) to display all pin pairs. Available only if Pin pairs is selected in the Against rule object area.
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. For reporting purposes, nets and pin pairs in the Source rule object list are identified as aggressors on page 1808. 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

Creation of Rules for Your Design

Confirm Pin Swap Dialog Box

To access: **ECO Toolbar > Swap Pin** button > select pins without a swap ID or with different swap IDs

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.

Objects

Field	Description
Don’t display again	Prevents this prompt from appearing the next time you swap pins with different or undefined swap IDs. This setting is reset when you close and then return to the ECO Toolbar.
OK	Proceeds with the swap of the pins. 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 on page 1590.

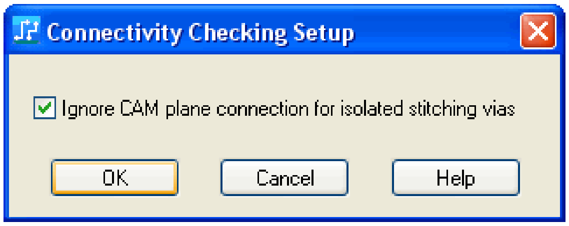
Related Topics

Swapping a Pin Manually in ECO Mode

Connectivity Checking Setup Dialog Box

To access: **Tools > Verify Design** menu item > Connectivity check > **Setup** button

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.



Objects

Field	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

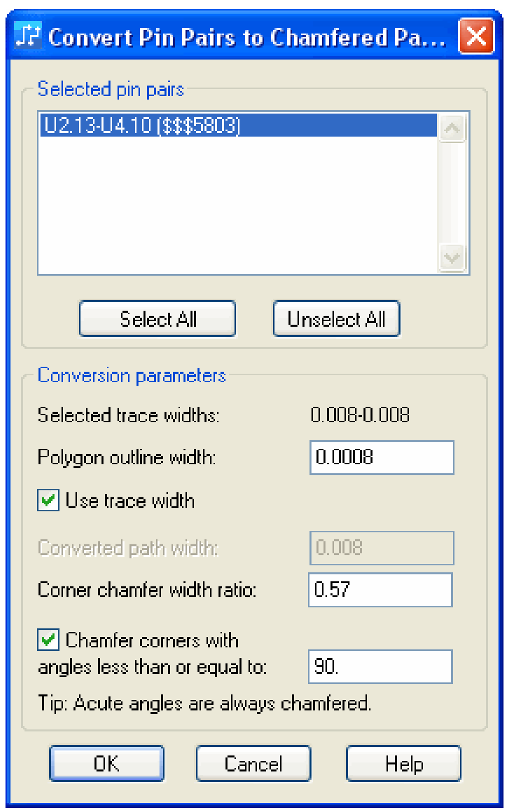
Setting Up Checking for Isolated Stitching Vias

Convert Pin Pairs to Chamfered Paths Dialog Box

To access: Select a pin pair, multiple pin pairs, or a net > right-click > **Convert to Chamfered Paths** popup menu item

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. When you convert the trace to a chamfered path, SailWind Layout automatically makes the net assignment.

Tip
Unrouted or partially routed pin pairs, or pin pairs belonging to reuse blocks, are excluded from selection.



Objects

Field	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.

Field	Description
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. 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. **Tip** Since copper is created with an outline and a fill, you can specify a very narrow outline width to achieve very sharp corners.
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. 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 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.

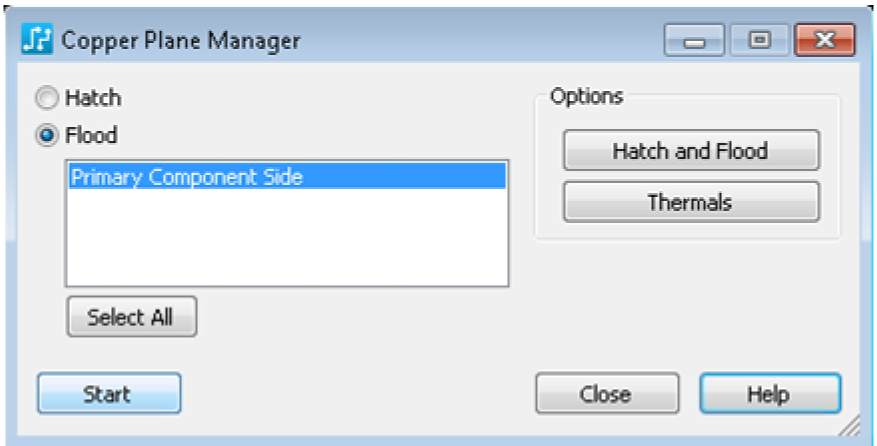
Related Topics

Converting a Trace to a Copper Chamfered Path

Copper Plane Manager Dialog Box

To access: Choose the **Tools > Copper Plane Manager** menu item

Flood copper plane outlines, and refill hatched copper plane areas.



Objects

Object	Description
Hatch	Fills in the hatch lines of previously flooded areas. Unlike Flood, Hatch does not create new isolation areas or thermal connections. If objects within the flooded area were moved, or changed thermal settings since the original flooding, you need to change the hatch outline to a pour or plane outline and use Flood to generate the new hatch in the area. Because the hatch lines are not stored, you should also perform hatch after opening a file, unless you have the “Autohatch on file load” setting enabled in the “Hatch and Flood” on page 1511 options.
Flood	Floods all copper planes on selected layers. • Flooding creates isolation areas around traces and pads that are inside the copper plane outline but are not part of the same net. It also creates thermal relief connections around pins that belong to the same net (provided the “Plane Thermal” check box is selected for the pins in the Pin Properties Dialog Box), and performs the process on all copper planes on the selected layer or layers. To flood individual copper planes, select a copper plane shape in the design, right-click, and choose the **Flood** popup menu item. • It 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, but can be detected with a Connectivity design verification check.

Object	Description
	• Click **Select All** to select all layers listed in the Layers list. • Alternatively, you can flood copper planes on all layers at one time by clicking the Flood All button in the main toolbar. 
Options	Provides quick access to related subcategories of the Tools menu > Options dialog box. Select the option type and click **Setup**. • Thermals — Set the size and shape of default thermals. • Hatch and Flood — Set hatching and flooding options.

Related Topics

Flooding Copper Planes

Hatching Copper Planes

Verify the Design

Create Array Dialog Box

To access: Select the components to include in the array > right-click > **Create Array** popup menu item

Use the Create Array dialog box to set options for creating an array.

Description

Figure 148. Planar tab

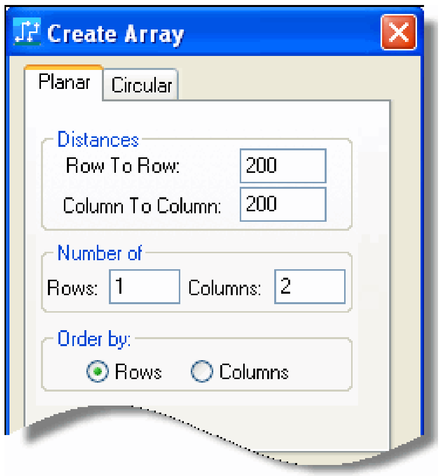


Figure 149. Circular tab

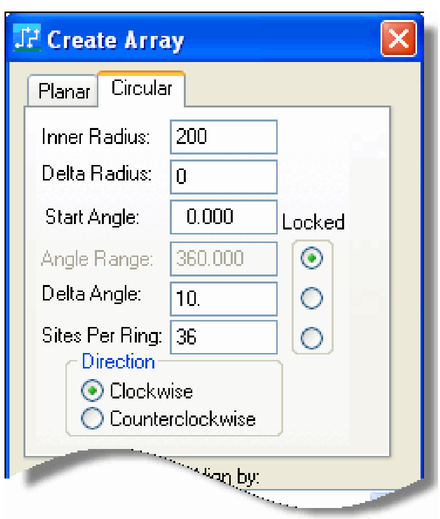
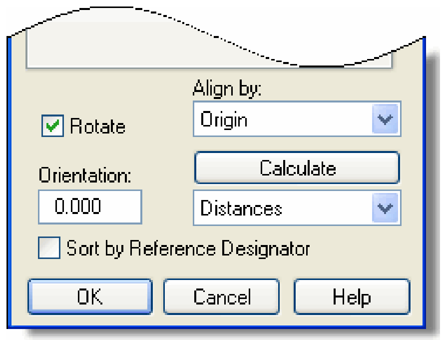


Figure 150. Create Array Dialog Box



Objects

Table 159. Create Array Dialog Box Fields

Field	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.
Order by area	• 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. ·6¤ ·7¤ ·8¤ ·9¤ 10¤ ·1¤ ·2¤ ·3¤ ·4¤ ·5¤ • 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. ·4¤ ·8¤ ·3¤ ·7¤ ·2¤ ·6¤ ·1¤ ·5¤
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.

Table 159. Create Array Dialog Box Fields (continued)

Field	Description
Delta Radius	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.
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.
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 SailWind 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:

Table 159. Create Array Dialog Box Fields (continued)

Field	Description
	• 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

Setting Up a Polar Grid

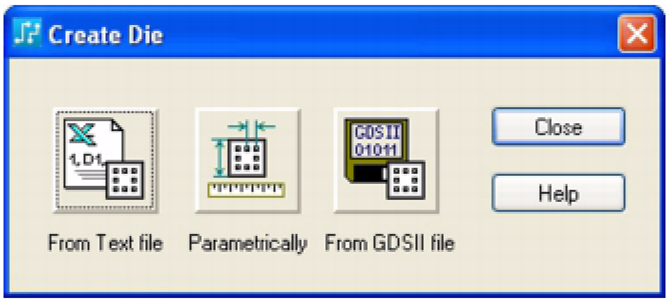
Component Arrays

Create Die Dialog Box

To access: **BGA Toolbar > Die Wizard** button

Use the Create Die dialog box to select the method you want to use to create a new die.

Note:
This information applies only to the BGA toolkit.



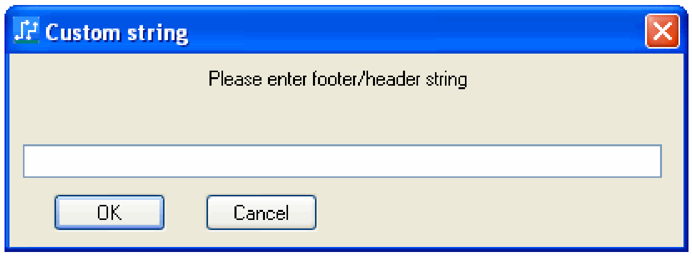
Objects

Field	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

Custom String Dialog Box

To access: **File > Create PDF** menu item > in a header or footer position list, select **Custom**

Use the Custom String dialog box to create custom header or footer text for PDF outputs.



Objects

Name	Description
text box	Type your custom text for the selected header or footer position.

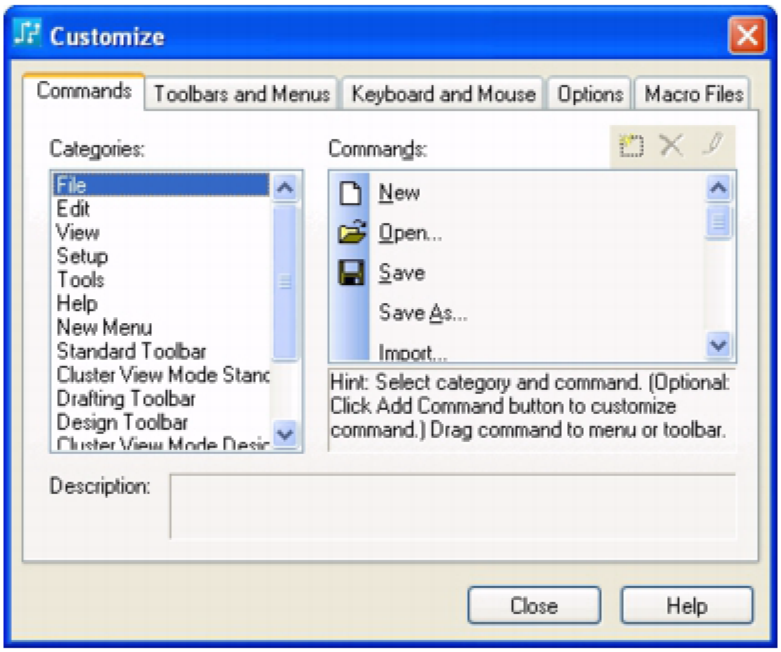
Related Topics

PDF Configuration Dialog Box

Customize Dialog Box, Commands Tab

To access: **Tools > Customize** menu item > **Commands** tab

Use the **Commands** tab to add commands to menus or toolbars, or to create custom menus.



Objects

Field	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 have added, or rename a command you have added.

Related Topics

Creating a Custom Command

Editing a Custom Command

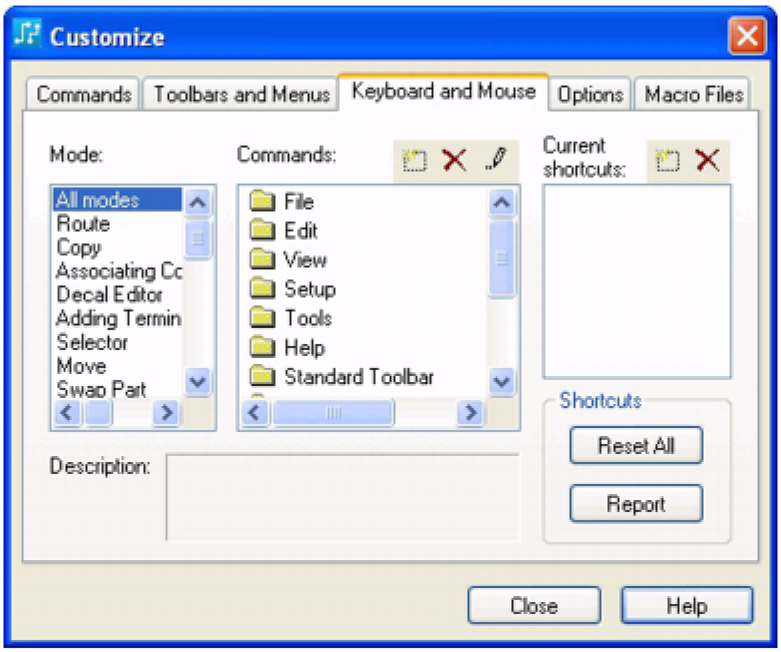
Creating a Custom Menu

Adding Items to Toolbars and Menus

Customize Dialog Box, Keyboard and Mouse Tab

To access: **Tools > Customize** menu item > **Keyboard and Mouse** tab

Create and customize shortcut keys using the **Keyboard and Mouse** tab of the Customize dialog box.



Objects

Field	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 on page 1050), delete a command you have added, or rename a command you have added (opens the Edit Command dialog box on page 1050).
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 have added.
Description	Lists what the selected command does.

Field	Description
Reset	Sets the selected toolbar or shortcut menu to the default settings.
Report	Saves a report of all current shortcut commands.

Related Topics

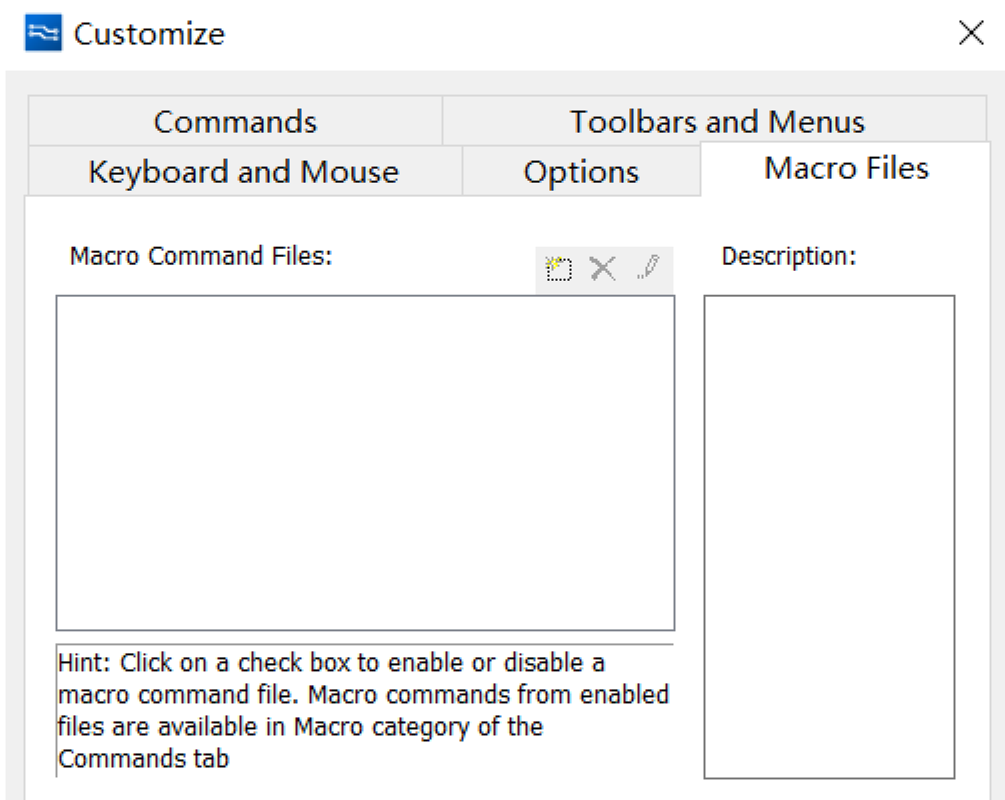
Shortcut Key Customization

Customize Dialog Box, Macro Files Tab

To access: **Tools > Customize** menu item > **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 SailWind tool or scripting it in Macro language. For more information, see "Recording Design Work in Macro Files".



Objects

Field	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 have added.
Description	Lists what the selected macro does.

Related Topics

Recording Design Work in Macro Files [SailWind Layout Command Reference Manual]

Screen Appearance

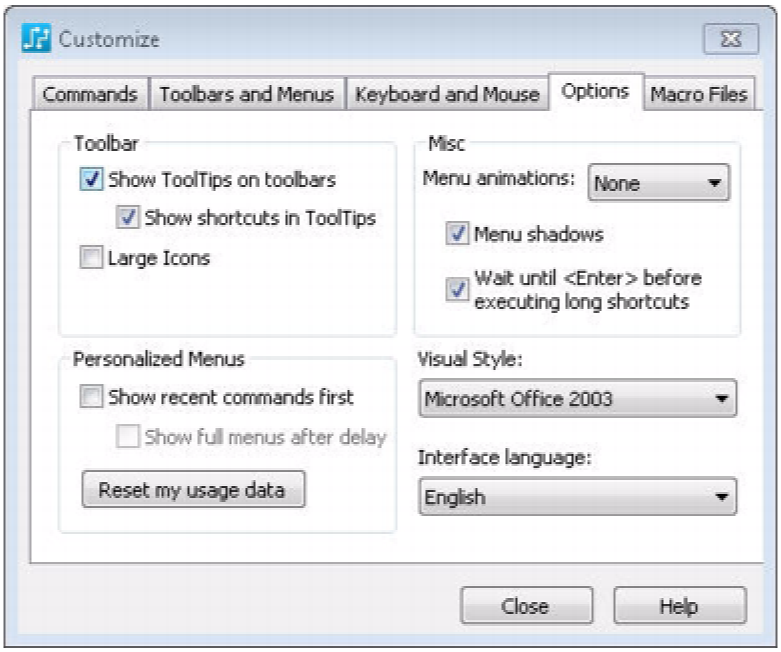
Creating a Custom Menu

Software Interface Customization

Customize Dialog Box, Options Tab

To access: **Tools > Customize** > **Options** tab

Customize the SailWind interface by changing the appearance of menus and toolbars using the **Options** tab of the Customize dialog box.



Objects

Field	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.
Menu shadows	Displays a shadow behind the menu.
Wait until <Enter> before executing long shortcuts	Delays the execution of shortcut keys until you press the Enter key.

Field	Description
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. 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, Chinese Simplified.

Related Topics

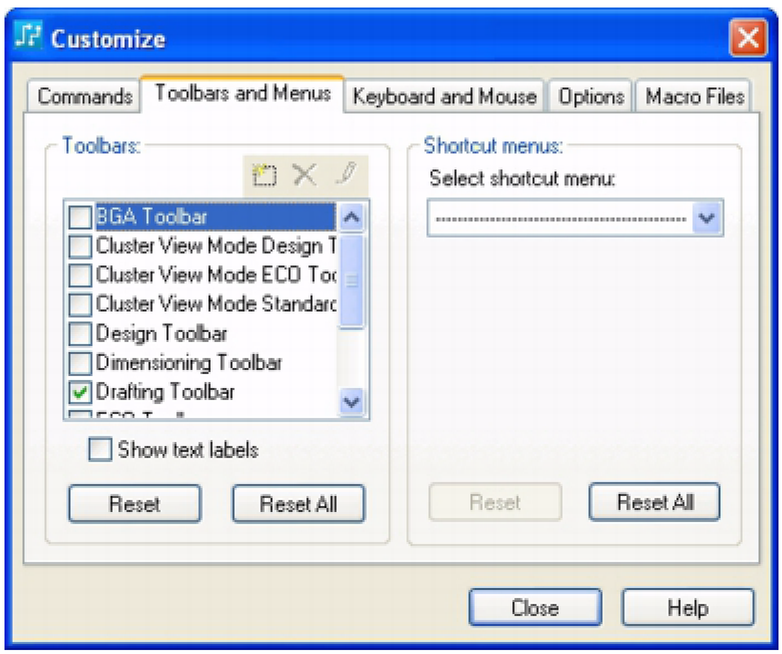
Screen Appearance

Customize Dialog Box, Toolbars and Menus Tab

To access: **Tools > Customize** menu item > **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".



Objects

Field	Description
Toolbars list	Specify which toolbars to display in the main window.
	Add a new toolbar, delete a toolbar you have added, or rename a toolbar you have 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.

Field	Description
	Restriction: SailWind 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

Software Interface Customization

Chapter 48
GUI Reference Elements D

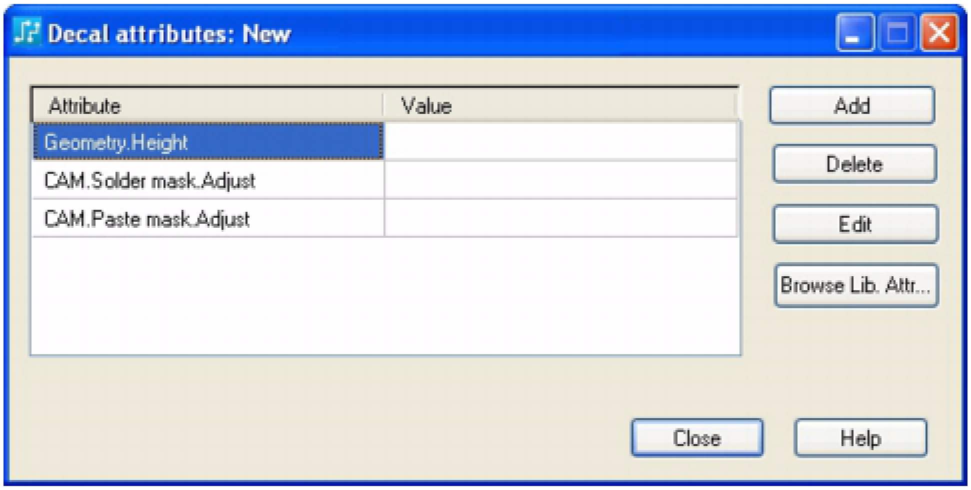
Read the sections that follow to learn more about dialog box elements in SailWind Layout.

DXF Import Dialog Box
DXF Import Dialog Box

Decal Attributes Dialog Box

To access: **Tools > PCB Decal Editor** menu item > **Edit > Attribute Manager** menu item

Use this dialog box to assign decal attributes, such as Geometry.Height. The information travels with the decal.



Objects

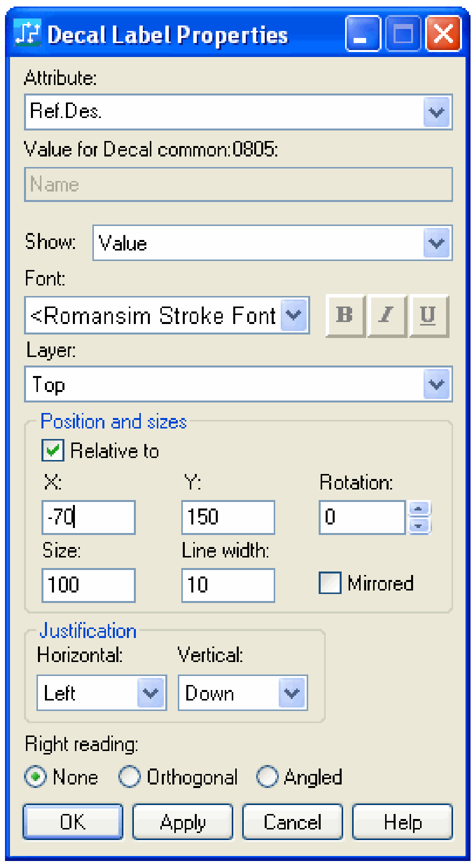
Field	Description
Attribute table	• Attribute column — Lists the attributes assigned to the decal. • Value column — Lists the value of the attributes assigned to the decal. You can specify the units for the value.
Add	Adds a new row to the end of the Attribute table.
Delete	Removes the selected row from 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

To access: Select a decal label > right-click > **Properties** popup menu item

Use this dialog box to modify a decal label or to change the attribute the label displays.

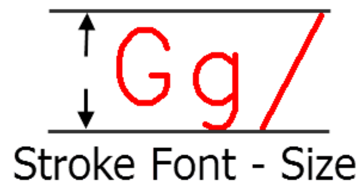
Tip
If you select multiple labels, settings in this dialog box apply to all selected labels.



Objects

Field	Description
Attribute	The attributes available to you. 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.
Value for	The value of the selected attribute.

Field	Description
	Tip • 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 SailWind 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 on page 1862, 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. **Tip** • 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.
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.

Field	Description
	
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. **Tip** • 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 None, Orthogonal, or Angled to indicate the direction of reading you want.

Related Topics

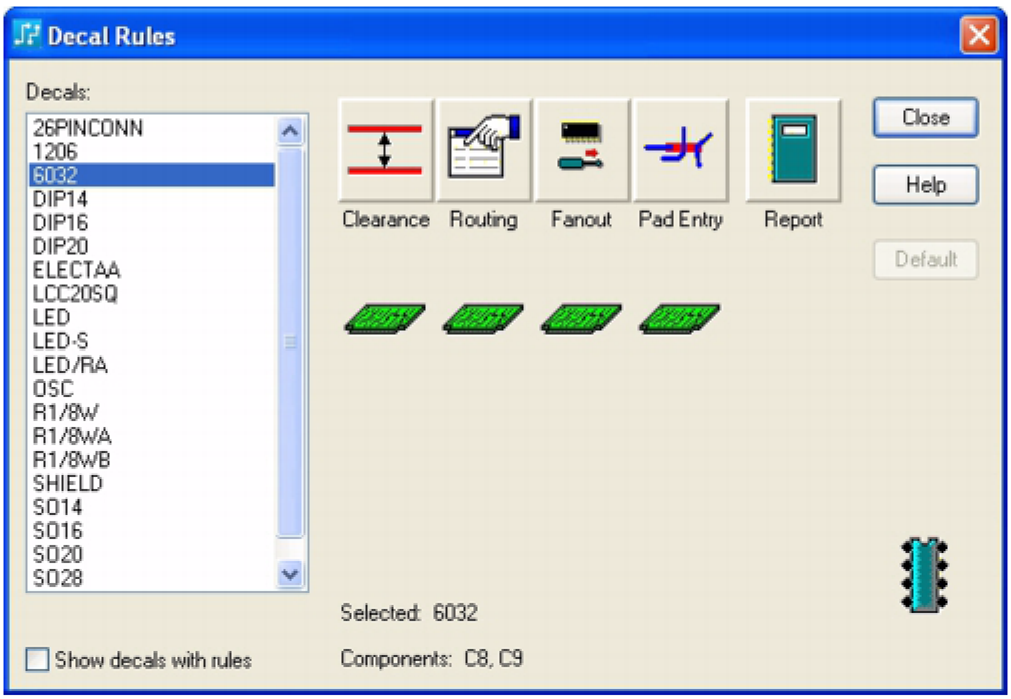
Modifying Decal Label Properties

Decal Rules Dialog Box

To access: **Setup > Design Rules** menu item > **Decal** button

Use this dialog box to define design rules that apply to decals.

Tip
You can define Decal Rules in SailWind Layout; however, these rules are used in SailWind Router only.



Objects

Field	Description
Decals list	Lists all decals in the design.
Show decals with rules	Specifies to show only decals that have rules.
Clearance	Opens the Clearance Rules Dialog Box.
Routing	Opens the Routing Rules Dialog Box.
Fanout	Opens the Fanout Rules Dialog Box.

Field	Description
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

Resetting Decal Rules to Default Rules

Creating Decal Design Rules in the PCB Decal Editor

Design Rule Hierarchy

Decal Rules Dialog Box (Decal Editor)

To access: **Tools > PCB Decal Editor** menu item > **Setup > Decal Rules** menu item

Use this dialog box to define design rules that apply to decals in the Decal Editor.

Tip
You can define Decal Rules in SailWind Layout; however, these rules are used in SailWind Router only.

Objects

Field	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

To access: **Tools > PCB Decal Editor** menu item > **Drafting Toolbar** button > **Wizard** button > **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.

Description

The **BGA/PGA** tab controls change depending on your Device type selection. The two differences are shown in the figures below:

Figure 151. BGA/PGA Tab - Through hole Controls

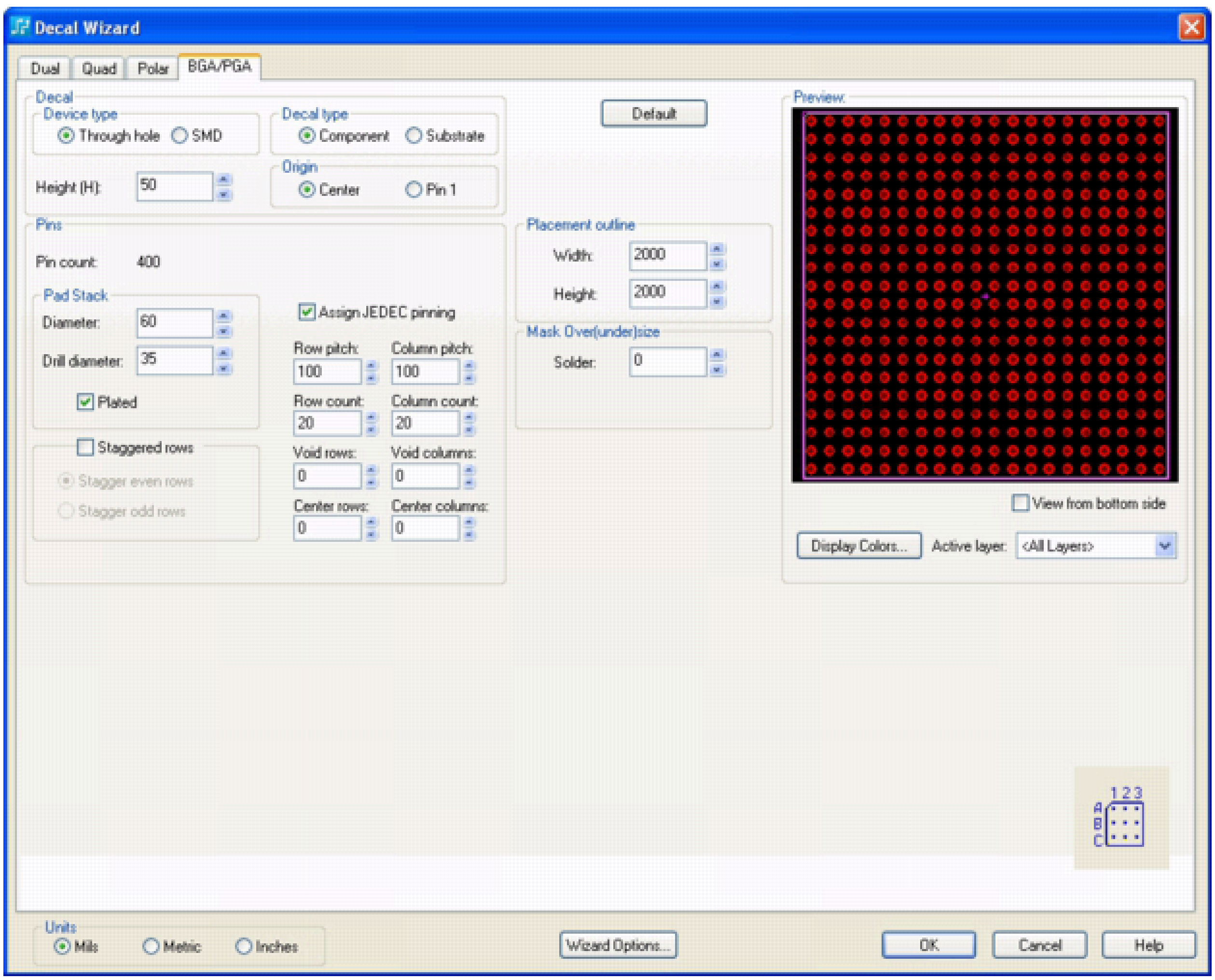
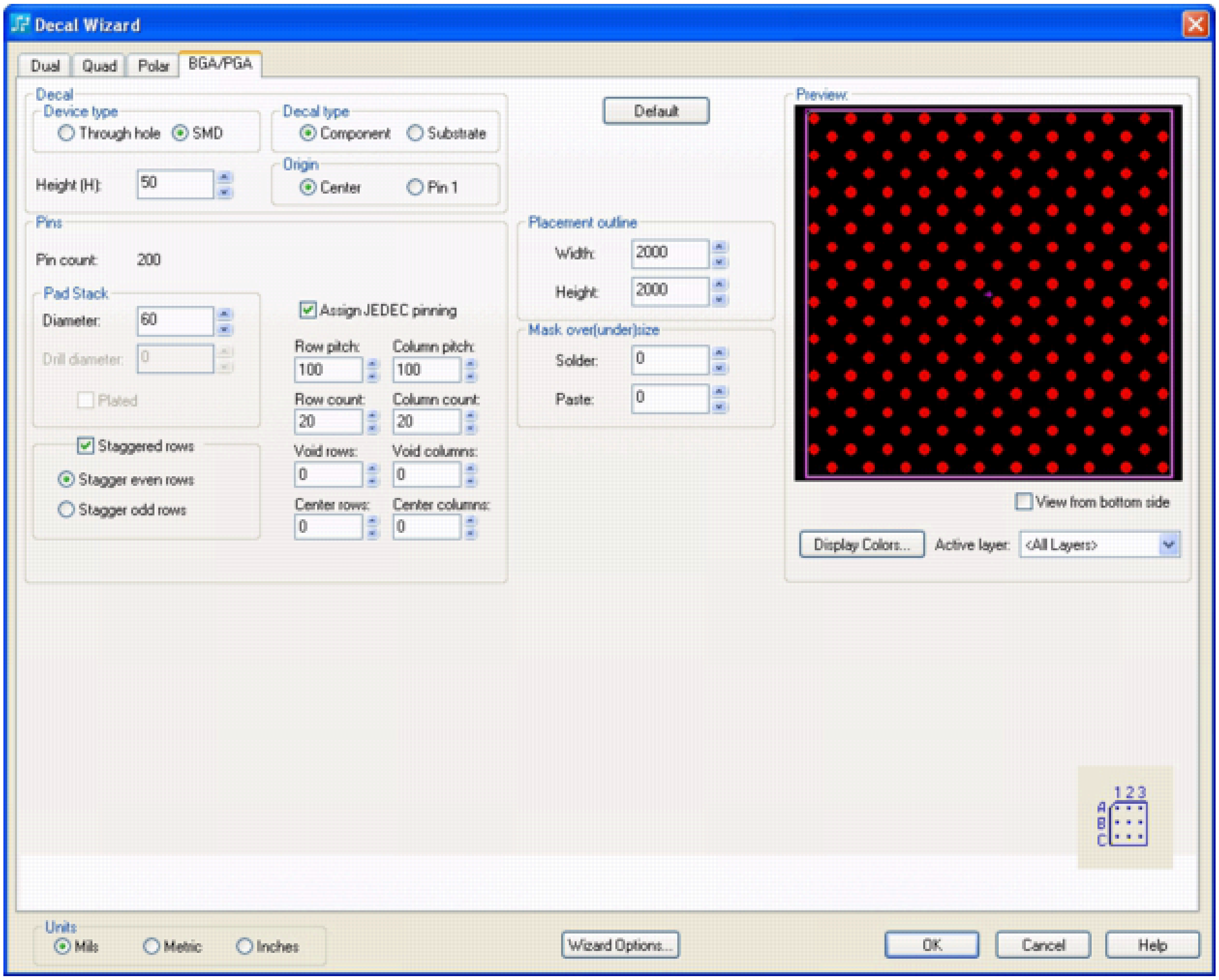


Figure 152. BGA/PGA Tab - SMD Controls



Objects

Table 160. BGA/PGA Tab Contents

Field	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.

Table 160. BGA/PGA Tab Contents (continued)

Field	Description
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	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. This control is only available when the Device type is set to Through hole.
Plated	Flags the through holes for plating. 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.
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.

Table 160. BGA/PGA Tab Contents (continued)

Field	Description
	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 have chosen not to create the outline in the Decal Wizard Options on page 1252.	
Width	Specifies the width of the placement outline. The placement outline options are set in the Decal Wizard Options on page 1252.
Height	Specifies the height of the placement outline. The placement outline options are set in the Decal Wizard Options on page 1252.
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 on page 1252. This area is unavailable if you have chosen not to create the solder mask outline in the Decal Wizard Options on page 1252.
Paste	Specifies the oversize or undersize of the paste mask. The paste mask options are set in the Decal Wizard Options on page 1252.

Table 160. BGA/PGA Tab Contents (continued)

Field	Description
	Restriction: • This control is only available when the Device type is set to SMD. • This area is unavailable if you have chosen not to create the paste mask outline in the Decal Wizard Options on page 1252.
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 on page 1252.

Related Topics

Creating a Basic Decal Automatically

Decal Wizard Dialog Box, Dual Tab

To access: **Tools > PCB Decal Editor** menu item > **Drafting Toolbar** button > **Wizard** button > **Dual** tab

Use the Dual wizard to create both DIP and SMD decals.

Description

The **Dual** tab controls change depending on your Device type selection. The differences are shown in the following two figures.

Figure 153. Dual Tab - Through hole Controls

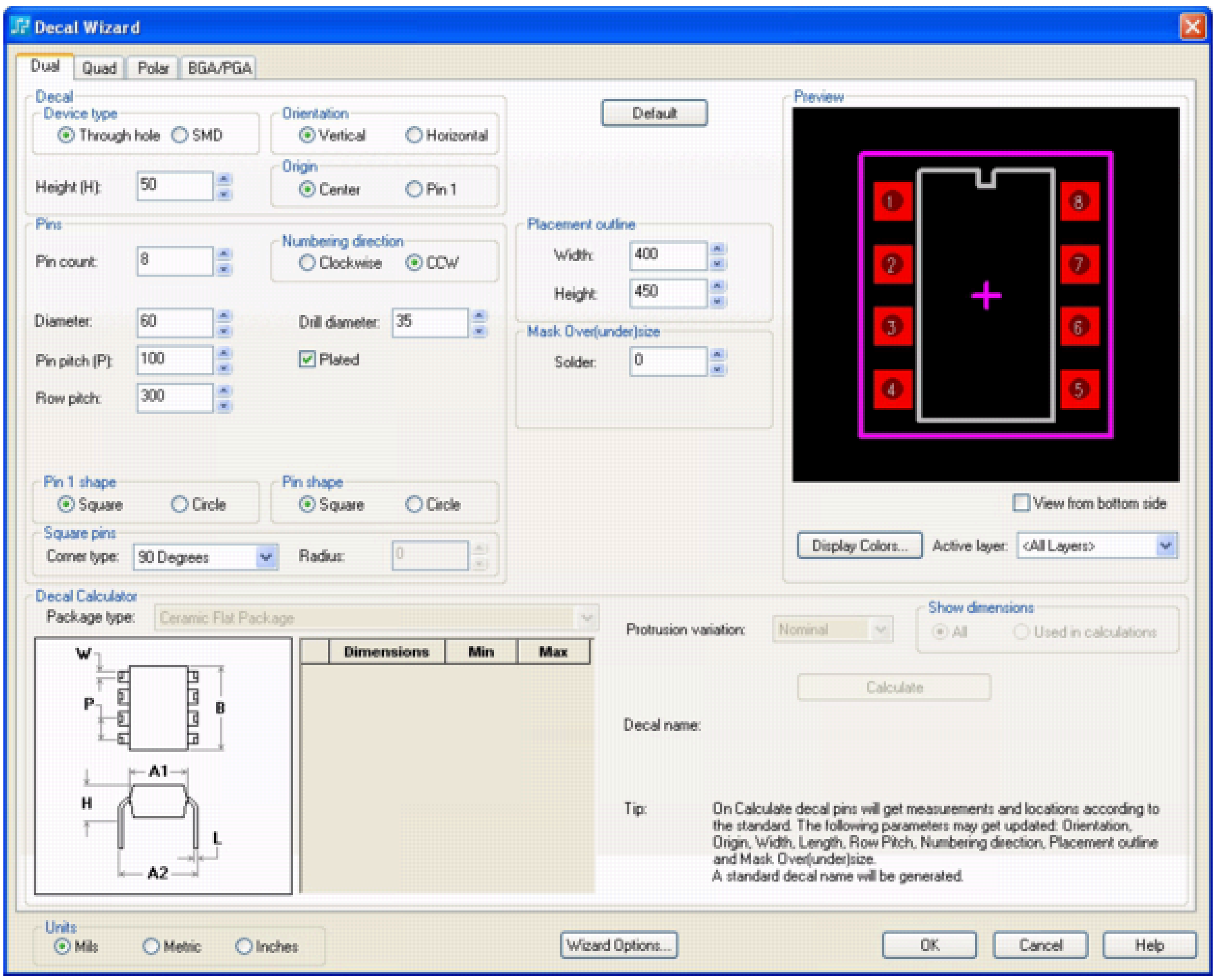
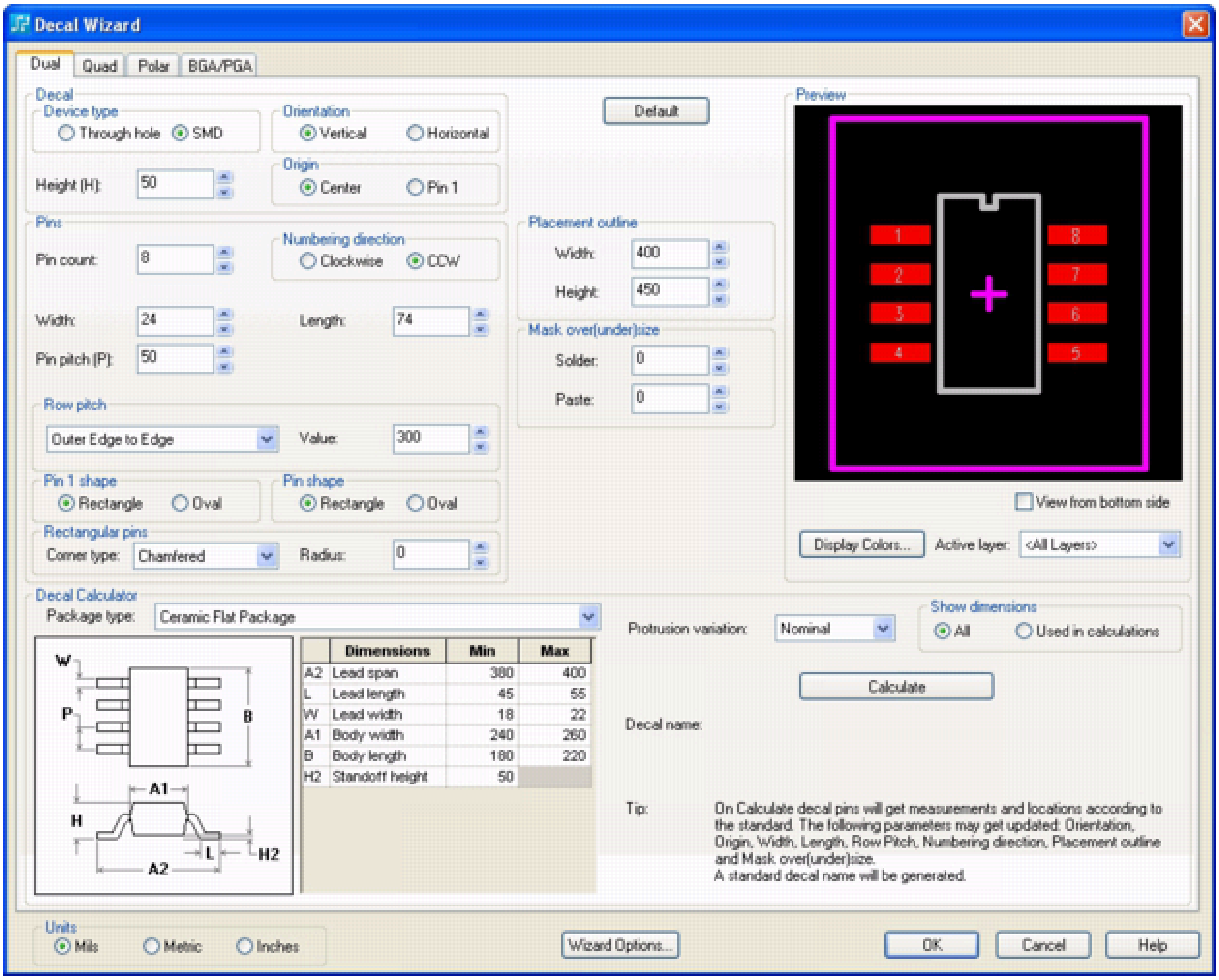


Figure 154. Dual Tab - SMD Controls



Objects

Table 161. Quad Tab Contents

Field	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.

Table 161. Quad Tab Contents (continued)

Field	Description
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.
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.

Table 161. Quad Tab Contents (continued)

Field	Description
	• 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	
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 have chosen not to create the outline in the Decal Wizard Options on page 1252.	
Width	Specifies the width of the placement outline. The placement outline options are set in the Decal Wizard Options on page 1252.
Height	Specifies the height of the placement outline. The placement outline options are set in the Decal Wizard Options on page 1252.
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 on page 1252. **Restriction:** This area is unavailable if you have chosen not to create the solder mask outline in the Decal Wizard Options on page 1252.

Table 161. Quad Tab Contents (continued)

Field	Description
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 on page 1252. **Restriction:** • This control is only available when the Device type is set to SMD. • This area is unavailable if you have chosen not to create the paste mask outline in the Decal Wizard Options on page 1252.
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 - on page 1257**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 on page 1838: minimum, nominal, and maximum. See "Decal Wizard Options - on page 1257**Package types** tab" for settings of these material conditions.

Table 161. Quad Tab Contents (continued)

Field	Description
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 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 on page 1252.

Related Topics

Creating a Basic Decal Automatically

Decal Wizard Dialog Box, Polar Tab

To access: **Tools > PCB Decal Editor** menu item > **Drafting Toolbar** button > **Wizard** button > **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.

Description

The **Polar** tab controls change depending on your Device type selection. The two differences are shown in the figures below:

Figure 155. Polar Tab - Through hole Controls

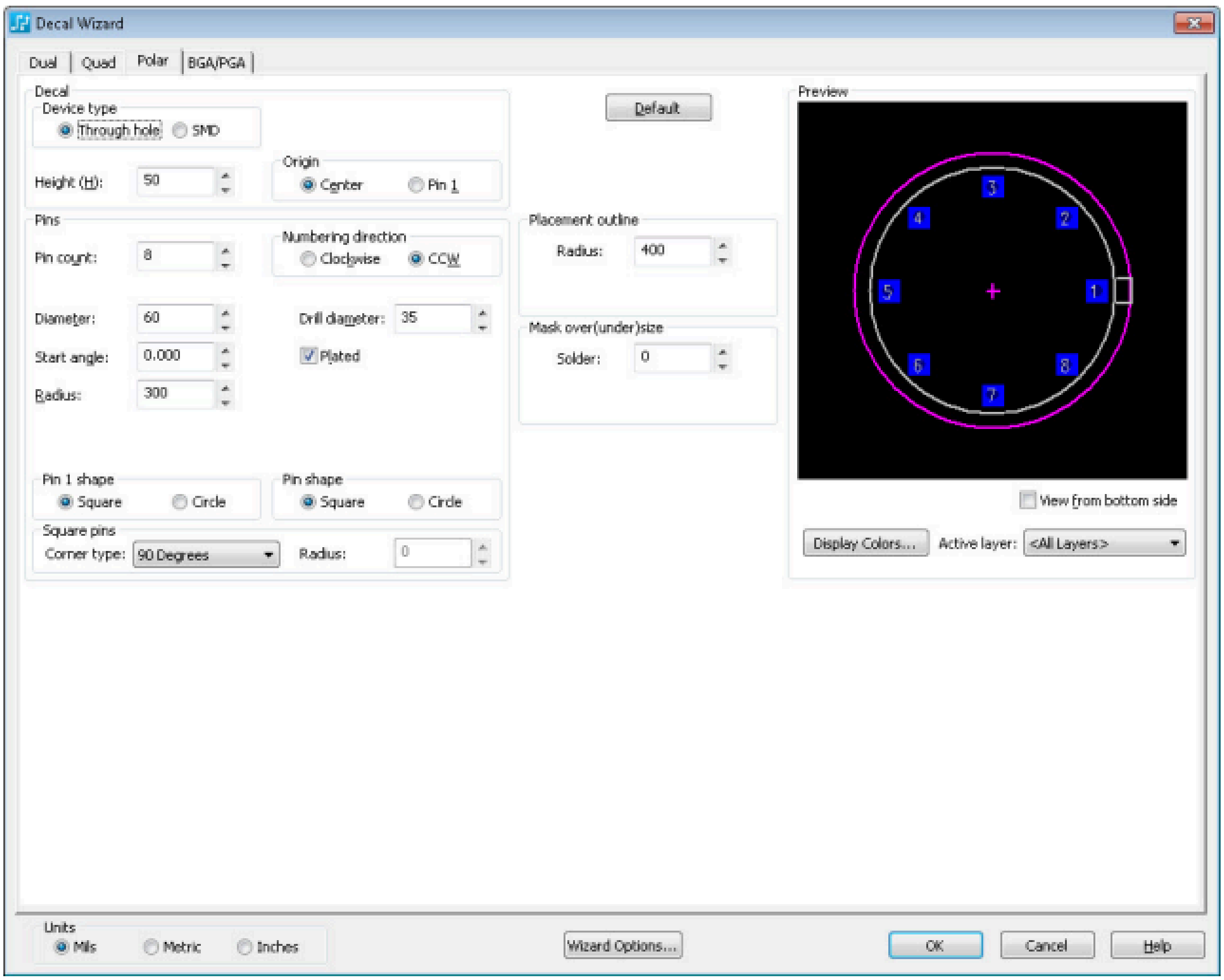
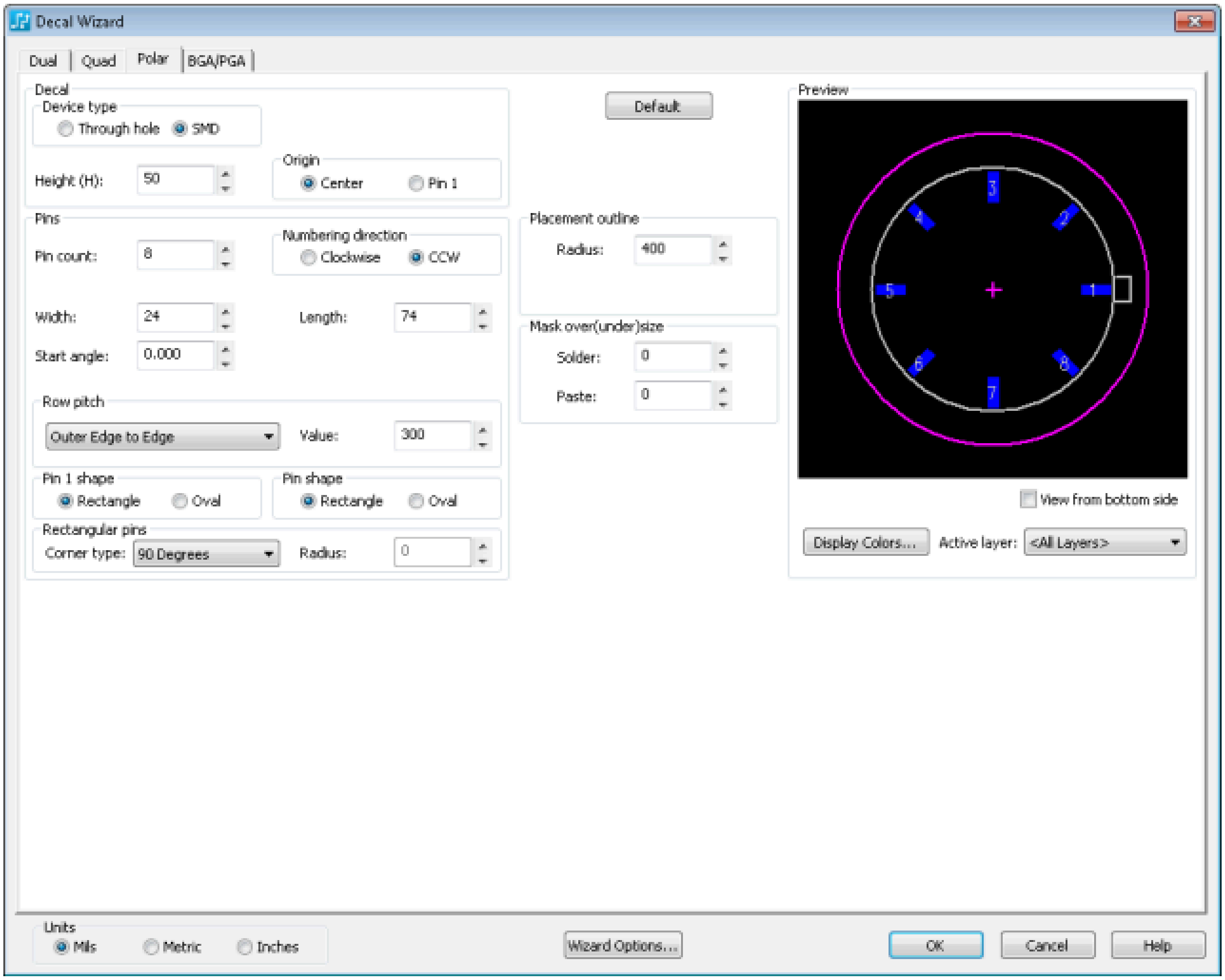


Figure 156. Polar Tab - SMD Controls



Objects

Table 162. Polar Tab Contents

Field	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.

Table 162. Polar Tab Contents (continued)

Field	Description
Pins area	
Pin count	Sets the number of pins in the decal.
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.

Table 162. Polar Tab Contents (continued)

Field	Description
	• 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	
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 have chosen not to create the outline in the Decal Wizard Options on page 1252.	
Radius	Specifies the radius of the placement outline. The placement outline options are set in the Decal Wizard Options on page 1252.
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 on page 1252. **Restriction:** This area is unavailable if you have chosen not to create the solder mask outline in the Decal Wizard Options on page 1252.
Paste	Specifies the oversize or undersize of the paste mask. The paste mask options are set in the Decal Wizard Options on page 1252. **Restriction:**

Table 162. Polar Tab Contents (continued)

Field	Description
	• This control is only available when the Device type is set to SMD. • This area is unavailable if you have chosen not to create the paste mask outline in the Decal Wizard Options on page 1252.
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 on page 1252.

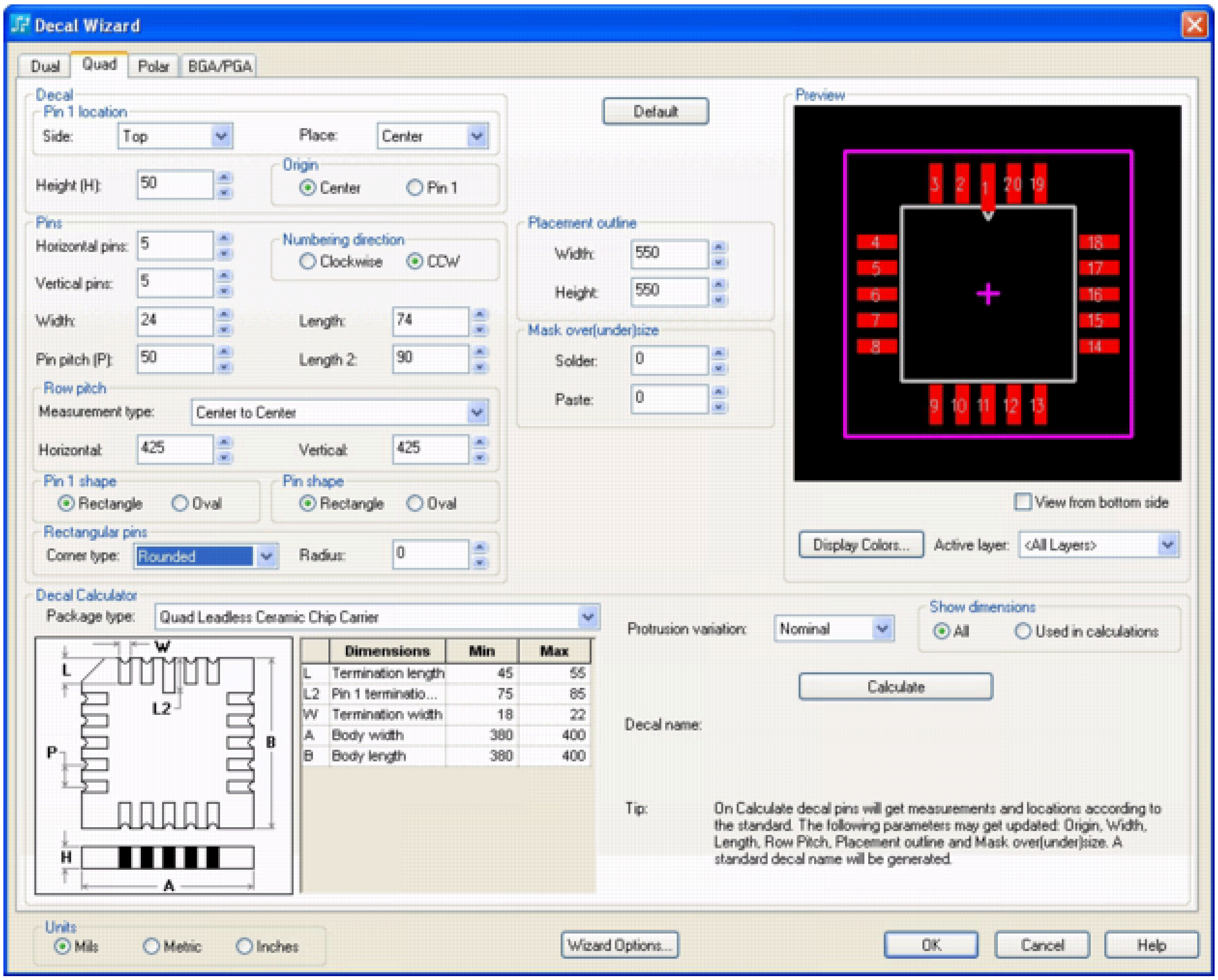
Related Topics

Creating a Basic Decal Automatically

Decal Wizard Dialog Box, Quad Tab

To access: **Tools > PCB Decal Editor** menu item > **Drafting Toolbar** button > **Wizard** button > **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.



Objects

Table 163. Quad Tab Controls

Field	Description
Decal area	
Pin 1 location	Specify the side and place to locate pin 1.

Table 163. Quad Tab Controls (continued)

Field	Description
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.
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.

Table 163. Quad Tab Controls (continued)

Field	Description
	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 have chosen not to create the outline in the Decal Wizard Options on page 1252.	
Width	Specifies the width of the placement outline. The placement outline options are set in the Decal Wizard Options on page 1252.
Height	Specifies the height of the placement outline. The placement outline options are set in the Decal Wizard Options on page 1252.
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 on page 1252. **Restriction:** This area is unavailable if you have chosen not to create the solder mask outline in the Decal Wizard Options on page 1252.
Paste	Specifies the oversize or undersize of the paste mask. The paste mask options are set in the Decal Wizard Options on page 1252. **Restriction:** This area is unavailable if you have chosen not to create the paste mask outline in the Decal Wizard Options on page 1252.
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 - on page 1257**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

Table 163. Quad Tab Controls (continued)

Field	Description
	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 on page 1838: minimum, nominal, and maximum. See "Decal Wizard Options - on page 1257**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 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 on page 1252.

Related Topics

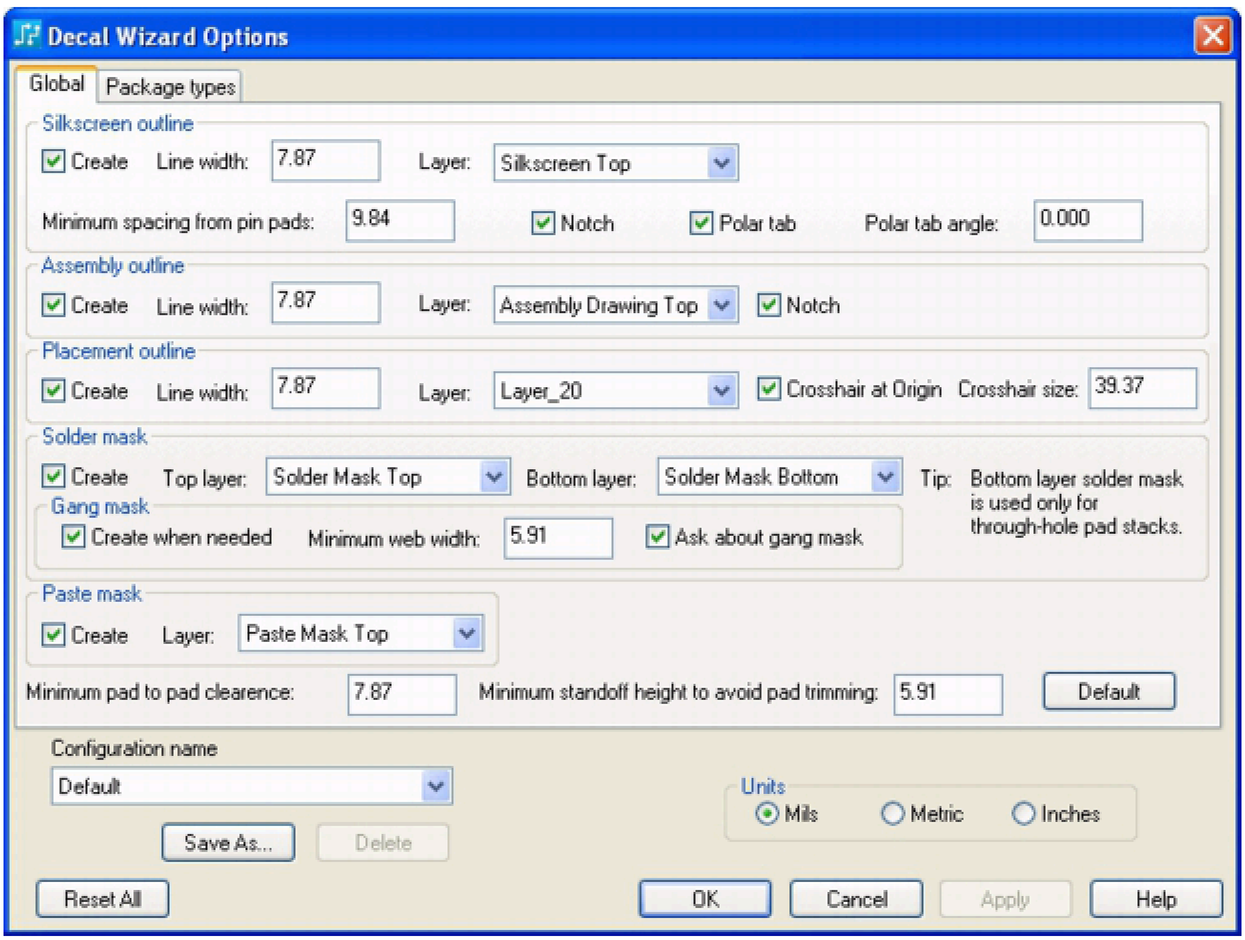
Creating a Basic Decal Automatically

Decal Wizard Options Dialog Box, Global Tab

To access:

- PCB Decal Editor > **Tools > Wizard Options** menu item > 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

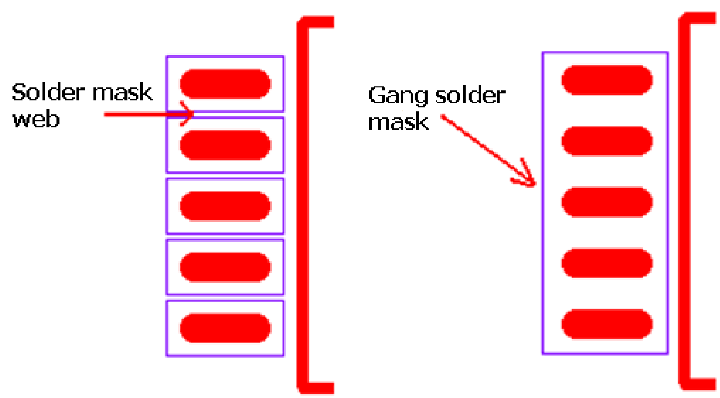
Use the **Global** tab to enable or disable supplemental documentation layers and settings.



Objects

Field	Description
Silkscreen outline area	

<table>
<tr><th>Field</th><th>Description</th></tr>
<tr><td>Create</td><td>Specifies to create the decal silkscreen.</td></tr>
<tr><td>Line width</td><td>Specifies the width of the line used to draw the silkscreen.</td></tr>
<tr><td>Layer</td><td>Specifies the layer on which to draw the silkscreen.</td></tr>
<tr><td>Minimum spacing from pin pads</td><td>Specifies the gap between pad edges and the silkscreen outline edge.</td></tr>
<tr><td>Notch</td><td>Specifies to create a notch in rectangular silkscreen shapes.</td></tr>
<tr><td>Polar tab</td><td>Specifies to add a tab to polar (circular) silkscreen shapes.</td></tr>
<tr><td>Polar tab angle</td><td>Specifies the location of the polar tab in degrees away from pin 1.</td></tr>
<tr><td colspan="2">Assembly outline area</td></tr>
<tr><td>Create</td><td>Specifies to create the assembly outline.
Restriction:
• 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 will not match the body of the component unless you click the Calculate button to generate a new assembly drawing with the new parameters.</td></tr>
<tr><td>Line width</td><td>Specifies the width of the line used to draw the assembly outline.</td></tr>
<tr><td>Layer</td><td>Specifies the layer on which to draw the assembly outline.</td></tr>
<tr><td>Notch</td><td>Specifies to create a notch in the assembly outline shape.</td></tr>
<tr><td colspan="2">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 on page 1257.) The clearance is calculated from the edges of pads or the component body, to the centerline of the placement outline.</td></tr>
<tr><td>Create</td><td>Specifies to create the placement outline.
Clearing this checkbox makes the Placement outline area unavailable in the Decal Wizard.</td></tr>
<tr><td>Line width</td><td>Specifies the width of the line used to draw the placement outline.</td></tr>
<tr><td>Layer</td><td>Specifies the layer on which to draw the placement outline.</td></tr>
</table>

Field	Description
Crosshair at Origin	Specifies to create a crosshair geometry at the origin of the decal. The crosshair is created with the same Line width as the placement outline.
Crosshair size	Specifies the size of the crosshair. 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. 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 val  ue. **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.

Field	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. Clearing this check box 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. 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. For more information, see “The Decal Wizard Options Configuration File”.
Save As	Allows you to save the current settings in a configuration for reuse. Opens the Save Configuration dialog box on page 1684. **Restriction:** Configuration names can be a maximum of 30 characters. Configuration files are stored in the *UserDir* folder as set in the *SailWindpcb.ini* file and has the extension *.dwc*. For more information, see “The Decal Wizard Options Configuration File”.
Units	Specify whether the values are Mils, Metric, or Inches.

Field	Description
Reset All	Resets both **Global** and **Package types** tabs (including all package types) settings to their default values.

Related Topics

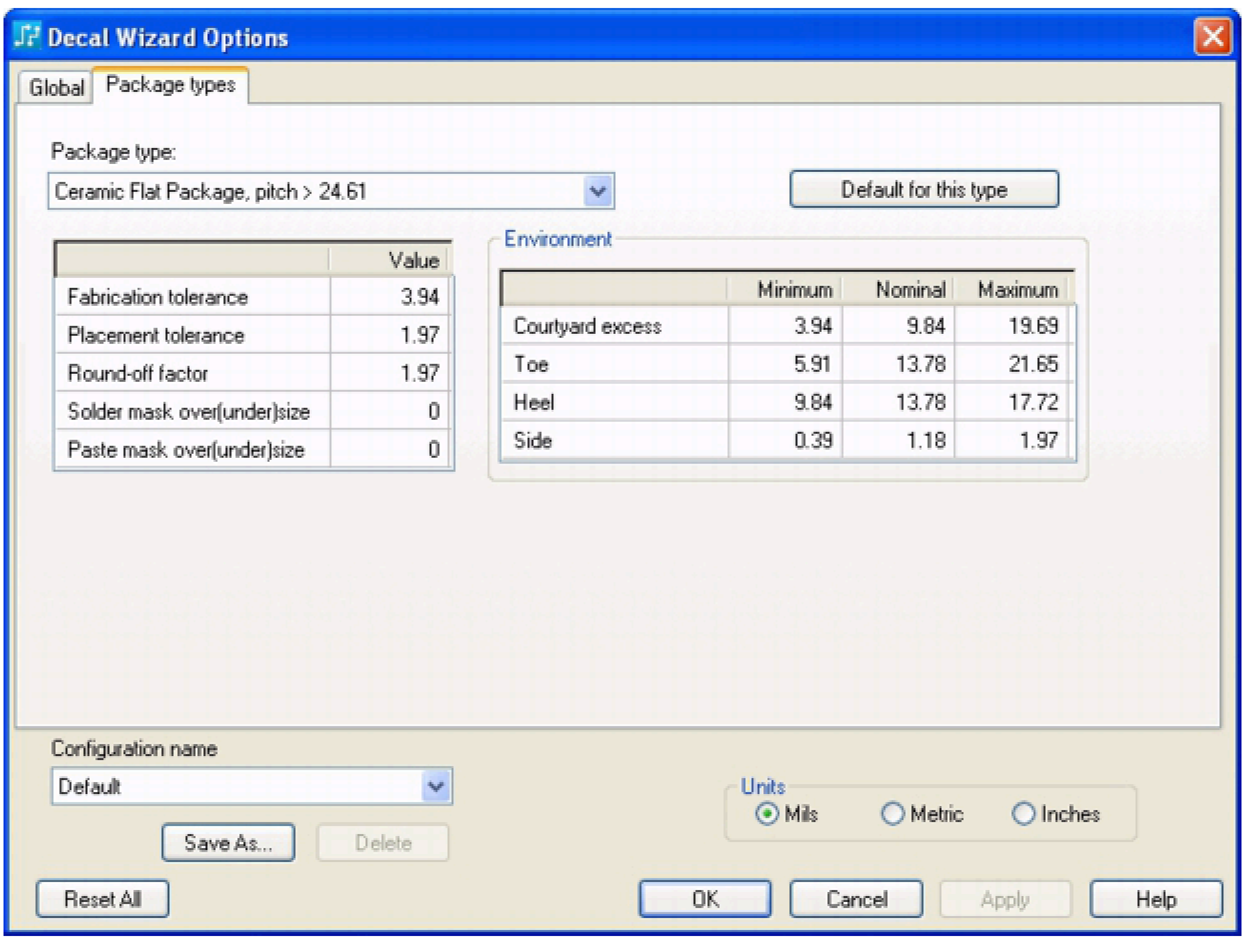
Decal Wizard Options Dialog Box, Package Types Tab

Decal Wizard Options Dialog Box, Package Types Tab

To access:

- PCB Decal Editor >**Tools > Wizard Options** menu item > **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

Use this tab to set the defaults for each of the Package types used by the Decal Calculator.



Objects

Field	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. If you open this dialog box from the Decal Wizard, this list item will correspond to the package type and the pitch value you have 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.
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.

Field	Description
	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.
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. For more information, see "The Decal Wizard Options Configuration File".
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 *SailWindpcb.ini* file and has the extension *.dwc*. For more information, see "The Decal Wizard Options Configuration File".
Units	Specify whether the values are Mils, Metric, or Inches.
Reset All	Resets both **Global** and **Package types** tab settings to their default values.

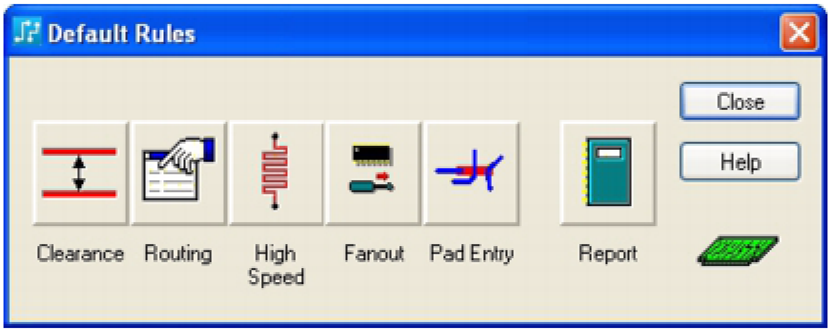
Related Topics

Decal Wizard Options Dialog Box, Global Tab

Default Rules Dialog Box

To access: **Setup > Design Rules** menu item > **Default** button

Use this 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.



Objects

Field	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

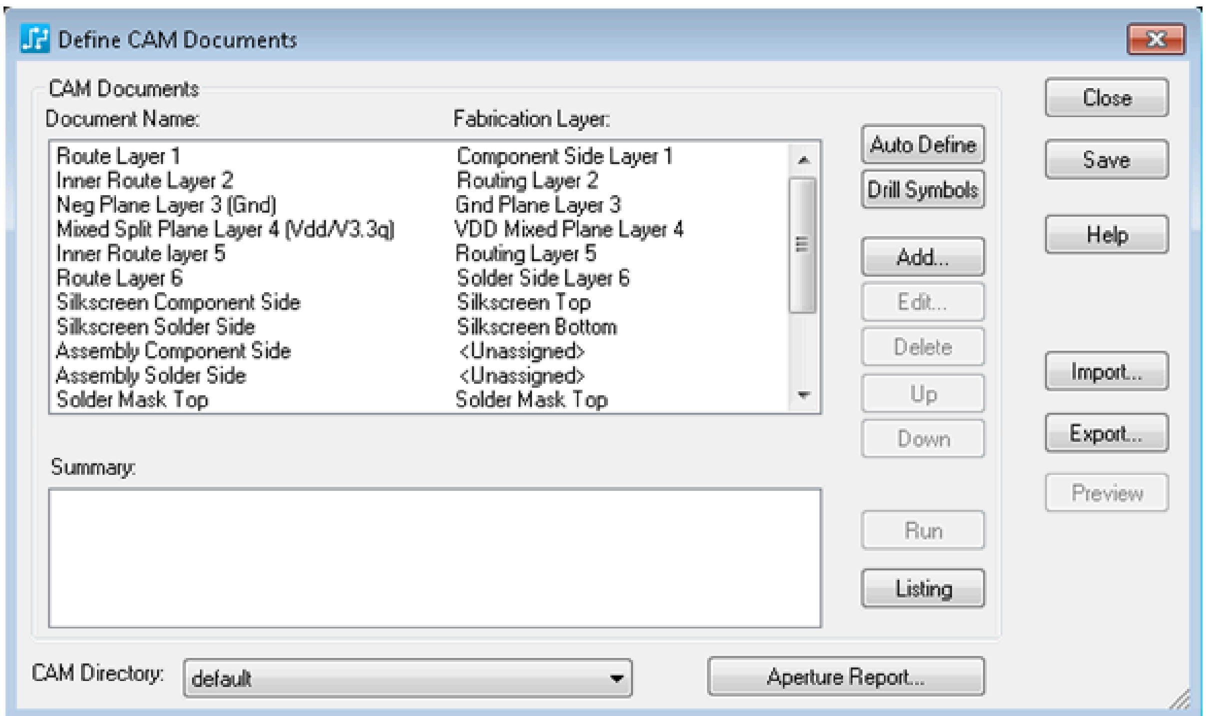
Design Rule Hierarchy

Define CAM Documents Dialog Box

To access: Choose the **File > CAM** menu item

Use this 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.



Objects

Field	Description
Document Name	The name of the CAM document.
Fabrication Layer	The layer on which you will be using CAM350 for post processing.
Auto Define button	Creates a complete set of CAM document definitions for the design using predefined default settings. Once created, the individual CAM document definitions can be edited or deleted as desired. Any existing CAM document definitions will not be overwritten.
Drill Symbols button	Opens the “Global Drill Symbols dialog box” on page 1387, enabling you to define the drill symbols that you want to be available for use in drill drawings.

Field	Description
	You enable each drill symbol you want to use in a specific drawing separately through the "Drill Symbols dialog box" on page 1334.
Add button	Opens the Add Document dialog box on page 1044. 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 on page 1044 where you can edit the document settings.
Delete button	Removes the selected CAM document.
Up button	Moves the selected CAM document up one space for list organization.
Down button	Moves the selected CAM document down one space for list organization.
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.
CAM Directory	Specifies the location of the CAM output file. **Tip** • The default location is *\SailWind Projects\Cam\default* which is called from the CAMDir entry of the *SailWindpcb.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.
Aperture Report	Produces a report of the apertures used in a CAM Document. 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.
Save	Saves the document configurations added to the CAM Documents list. Configurations are saved into the current software session but not yet saved into the *.pcb* file. You must save the *.pcb* design file for the changes made to the CAM documents to become part of the design file.
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. You can reuse your CAM document configurations with other similar designs by exporting them and importing them into another design.
Export	Saves the configuration to a separate file that you can later import to use for similar *.pcb* files.

Field	Description
	Tip If you name the file *default.cam*, it is used as the default CAM configuration for each *.pcb* file. You can reuse your CAM document configurations with other similar designs by exporting them and importing them into another design.
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

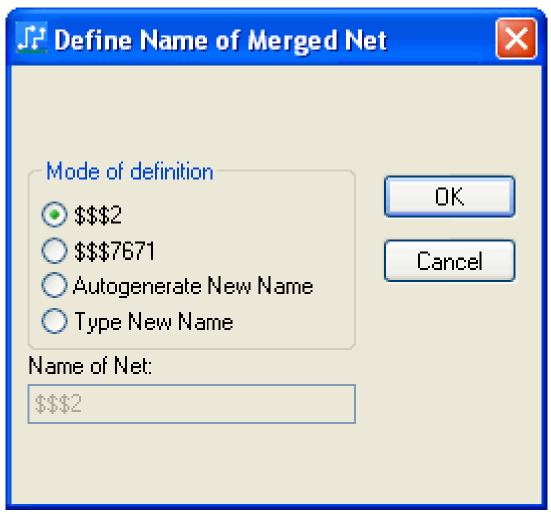
Creating CAM Outputs to Manufacture Your PCB

Define Name of Merged Net Dialog Box

To access:

- **ECO Toolbar > Add Connection** button > click two pins with different netnames
- **ECO Toolbar** button >**Add Route** button > create a route between two pins with different netnames
- BGA Toolbar > **Add Connection** and connect two BGA pins that have distinct nets

You use this dialog box to rename the net of two pins connected using the Add Connection or Add Route ECO tools.



Objects

Field	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

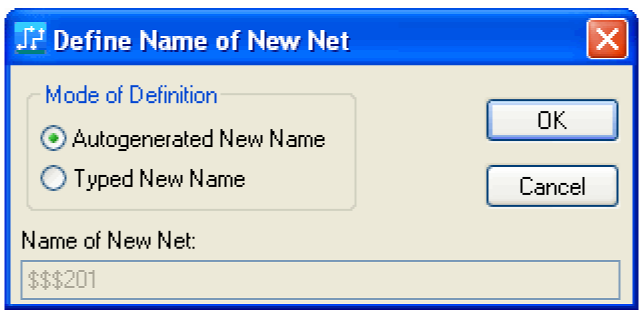
Define Name of New Net Dialog Box

To access:

- **ECO Toolbar** button >**Delete Connection** button > click a connection bridging two pin pairs.
- In the Die Flag Wizard Dialog Box, click **New Net**.

You use this 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.



Objects

Field	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

To access:

- **ECO Toolbar** button >**Delete Component** button > select one or multiple components
- **ECO Toolbar** button > select one or multiple components > press the Delete key

In ECO mode, use this dialog box to delete parts, resulting single-pin nets, and optionally any attached traces.

Description

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.

Figure 157. Delete Part Dialog Box 1

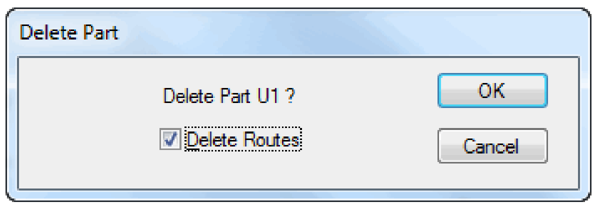
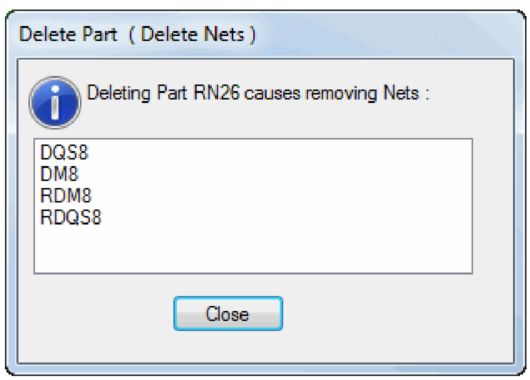


Figure 158. Delete Part Dialog Box 2



Objects

Table 164. Delete Part Dialog Box 1 Contents

Field	Description
Delete Part/Selected Parts	Prompts to confirm the deletion of the selected part(s). Specifies the reference designator of the part if you have only selected a single part.
Delete Routes	Select the check box to also delete all traces routed to and from this component.

Table 164. Delete Part Dialog Box 1 Contents (continued)

Field	Description
	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.

Table 165. Delete Part Dialog Box 2 Contents

Field	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

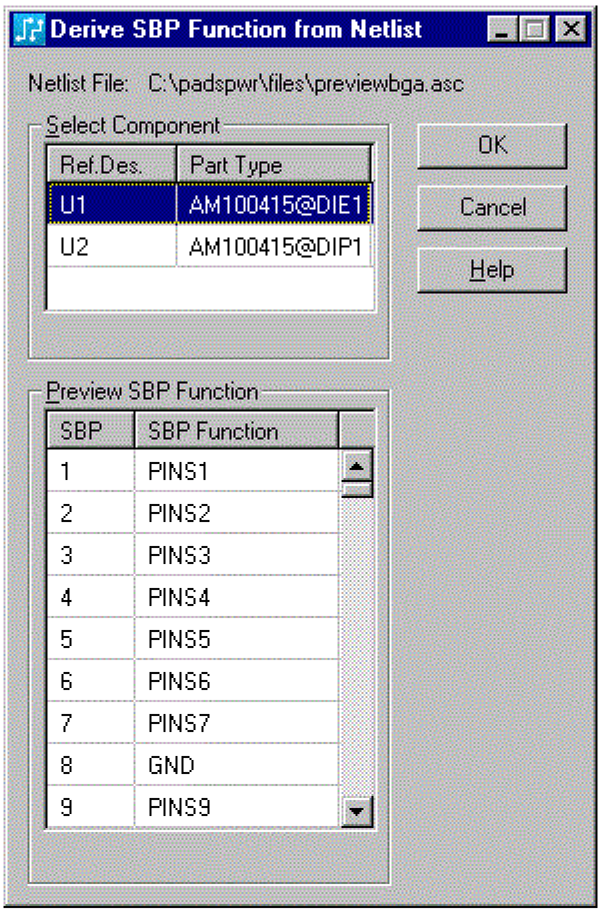
To access: **BGA Toolbar** button > **Wire Bond Wizard** button > **SBP Naming** button > **Derive from Netlist** button > open a file

Use the Derive SBP Function from Netlist dialog box to import the SBP functions from a BGA, SailWind Layout, or SailWind Logic netlist ASCII file. SBP numbers are used as they are currently defined in the SBP Properties dialog box.

Note:
This information applies only to the BGA toolkit.

Description

See also: “Exporting ASCII Files on page 291”.



Objects

Field	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

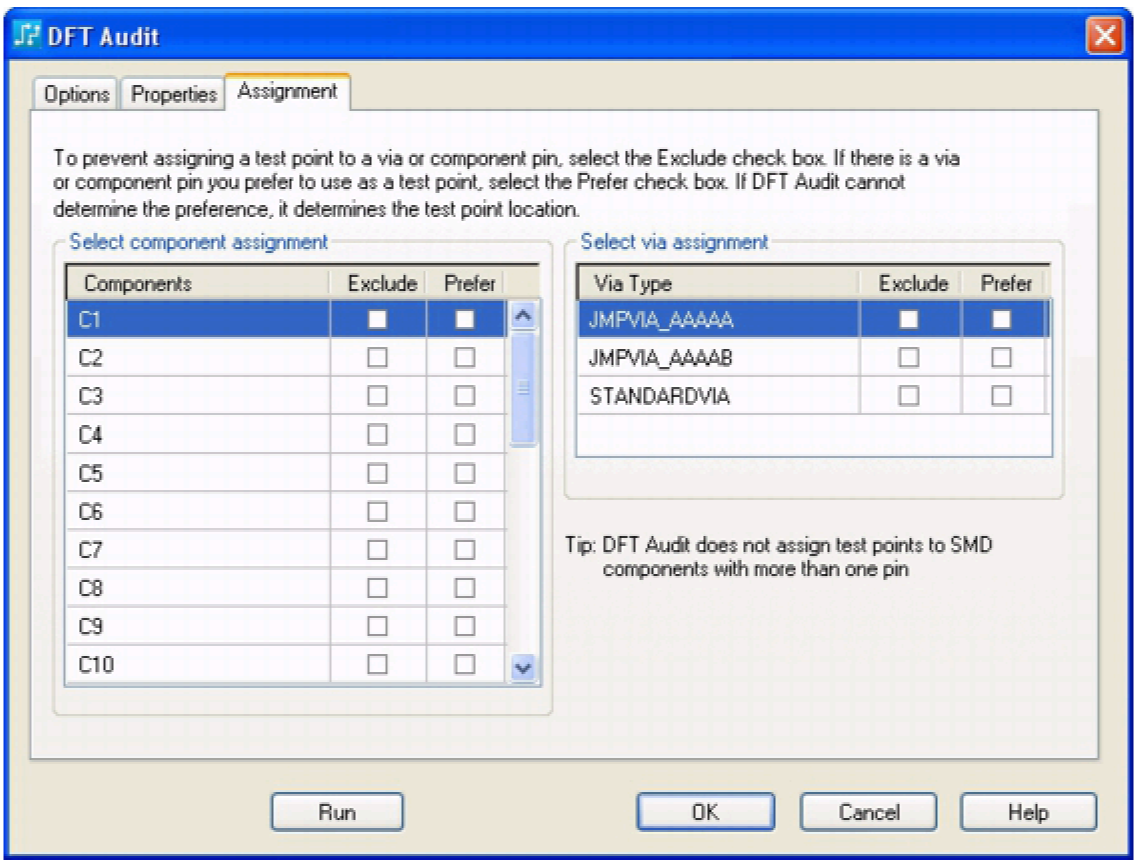
Importing the SBP Functions

DFT Audit Dialog Box, Assignment Tab

To access: **Tools > DFT Audit** menu item > **Assignment** tab

Use this 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.



Objects

Field	Description
Components column	Lists the components available to you.
Exclude column	Specifies to prevent the use of the component as a test point.

Field	Description
	To apply even weighting, clear the Exclude and Prefer check boxes.
Prefer column	Specifies to favor the use of the component as a test point. To 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. To apply even weighting, clear the Exclude and Prefer check boxes.
Prefer column	Specifies to favor the use of the via as a test point. To apply even weighting, clear the Exclude and Prefer check boxes.
Run button	Starts the automatic audit process.

Related Topics

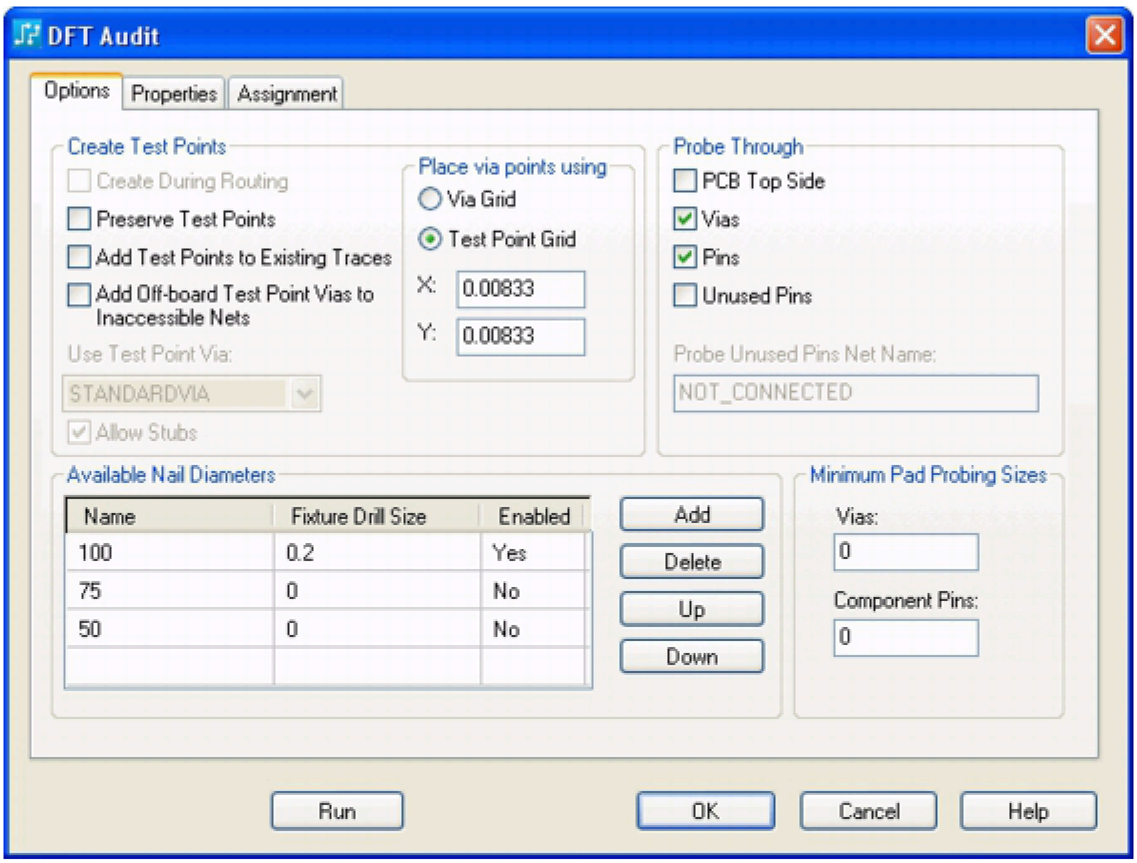
Setting Test Point Assignment Eligibility

DFT Audit Dialog Box, Options Tab

To access: **Tools > DFT Audit** menu item > **Options** tab

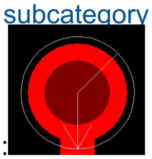
You have several options for placing test points on the **Options** tab of the DFT Audit dialog box.

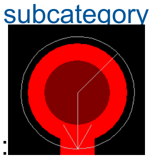
Tip
To reduce design iterations, consult with your automated test engineers when setting DFT Audit options.



Objects

Field	Description
Create During Routing	When using SailWind Router Link, select the check box to enable SailWind Router to create test points when routing nets. When using SailWind Layout, this check box has no function.

Field	Description
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 SailWind 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.
Add Off-board Test Points to Inaccessible Nets	Adds test point vias to nets that SailWind Router cannot make accessible. SailWind 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 — uses the via grid that you set in the Grids options on page 1537. • Test Point Grid — specify the grid spacing in the X and Y boxes.
Probe Through area	
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 the Show Test Points check box is selected in the Options dialog box > **Routing** category > General subcategory on page 1542, an arrow is drawn on it in the design: 
Pins	Specifies to use pins as test points.

Field	Description
	Tip When the pin is flagged as a test point, and the Show Test Points check box is selected in the Options dialog box > **Routing** category > General subcategory on page 1542, an arrow is drawn on it in the design: 
Unused Pins and Probe Unused Pins Net Name	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** — adds a row to the bottom of the table. • **Delete** — removes the selected row from the table. • **Up** and **Down** — moves the selected row up or down.
Minimum 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

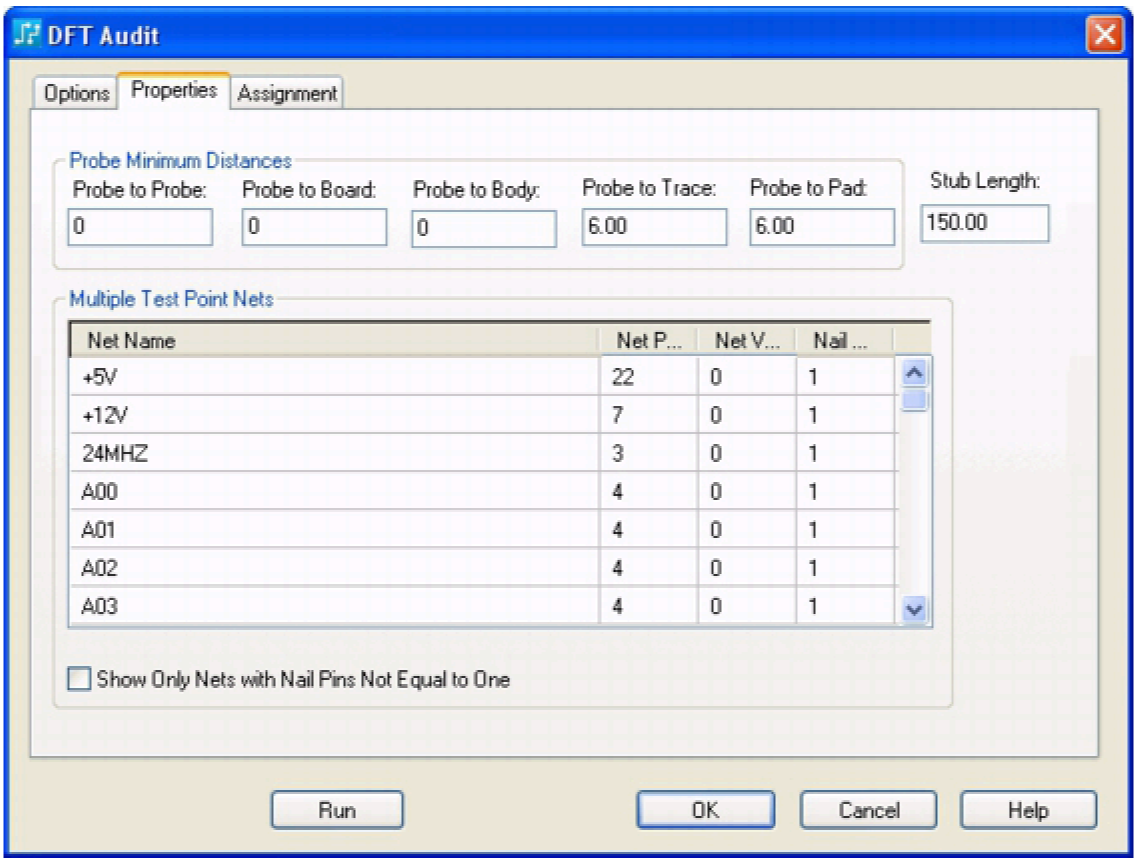
Placing Test Points

DFT Audit Dialog Box, Properties Tab

To access: **Tools > DFT Audit** menu item > **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.



Objects

Field	Description
Probe Minimum Distances area	Specifies the minimum distances between the probe and other design objects using options in the Probe Minimum Distances area.

Field	Description
	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. **Tip** • 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

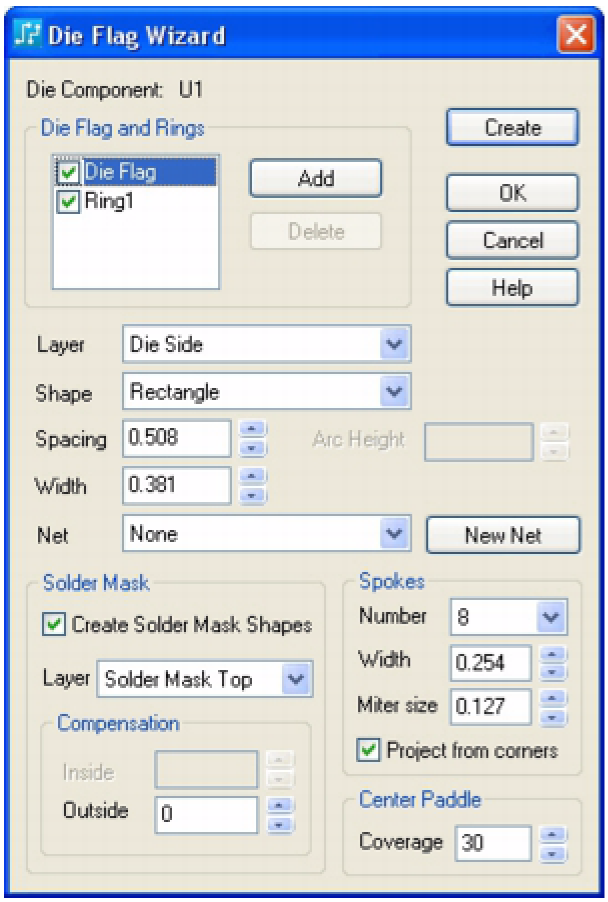
Setting Test Point Properties

Die Flag Wizard Dialog Box

To access: **BGA Toolbar** button > **Die Flag Wizard** button > select die

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.



Objects

Field	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.

Field	Description
	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.
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

Field	Description
	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.
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

Creating a Die Flag and Rings

The Die Flag Wizard

Die Wizard - Create from GDSII File Dialog Box

To access: **BGA Toolbar** button > **Die Wizard** button > **From GDSII File** button

Use this dialog box to add a new die to the current design or library based on an imported GDSII file.

Description

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

Restriction:
This information applies only to the BGA toolkit.

The Die Wizard has 5 tabs:

- "Figure 160" on page 1282
- "Figure 161" on page 1283
- "Figure 162" on page 1283
- "Figure 163" on page 1284
- "Figure 164" on page 1284

Figure 159. Die Wizard - Create from GDSII File Dialog Box

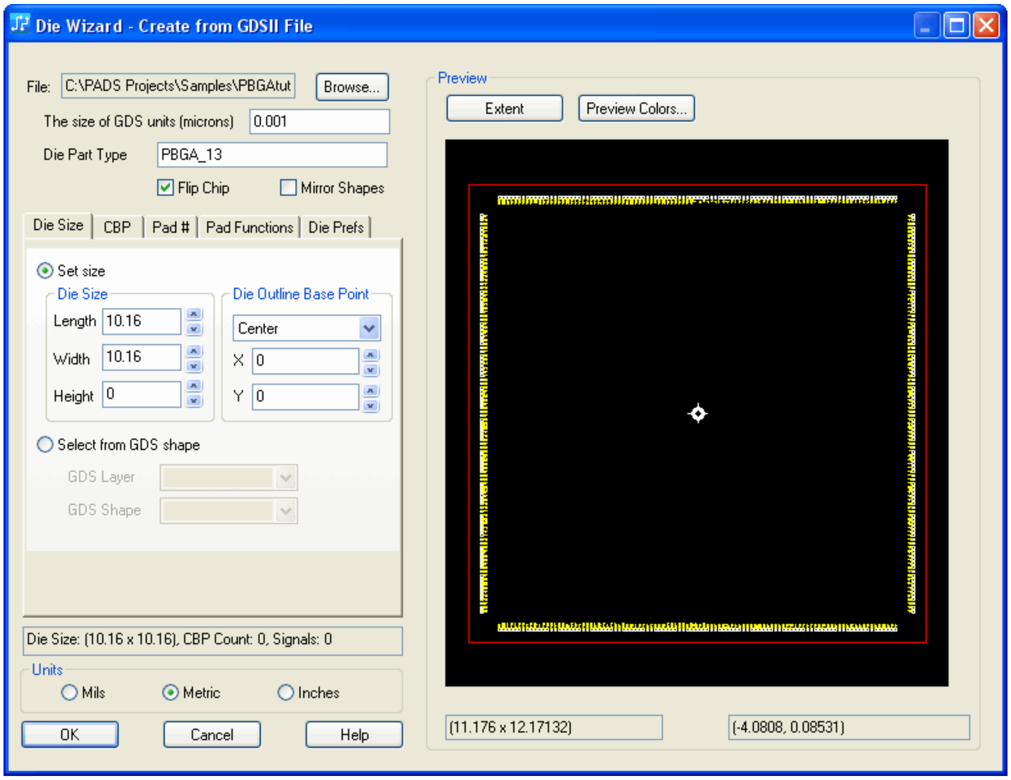


Figure 160. From GDSII Die Size tab

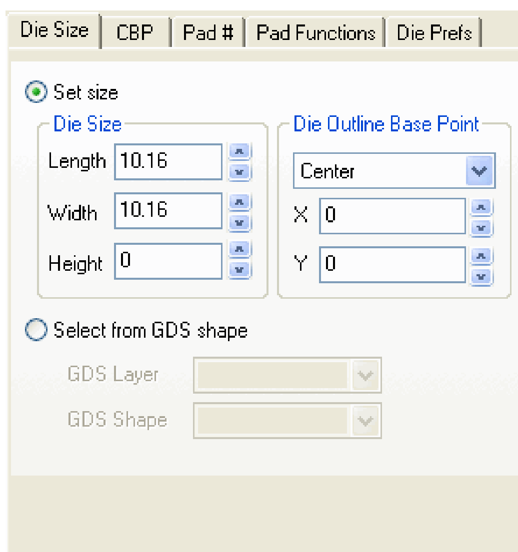


Figure 161. From GDSII CBP tab

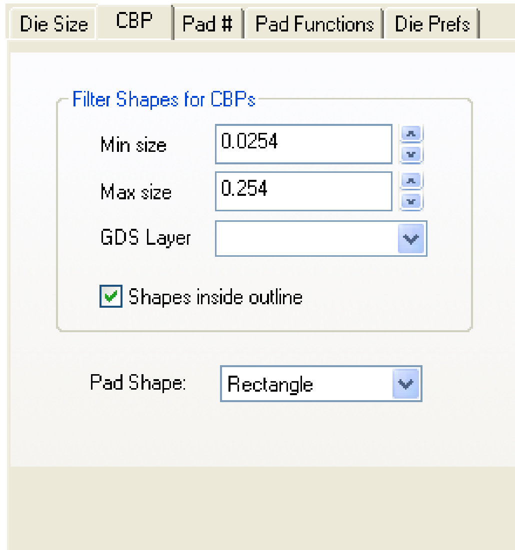


Figure 162. From GDSII Pad # tab

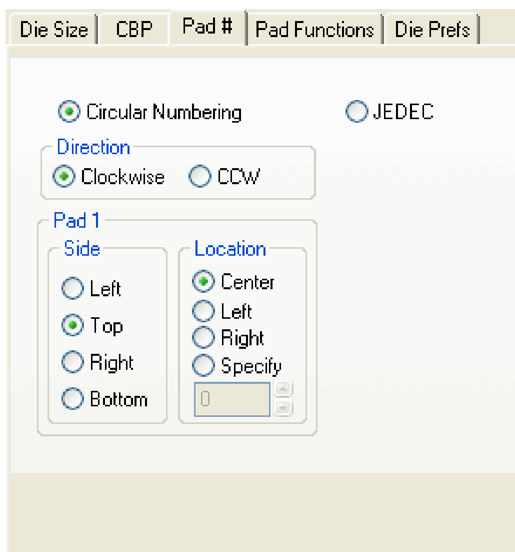


Figure 163. From GDSII Pad Functions tab

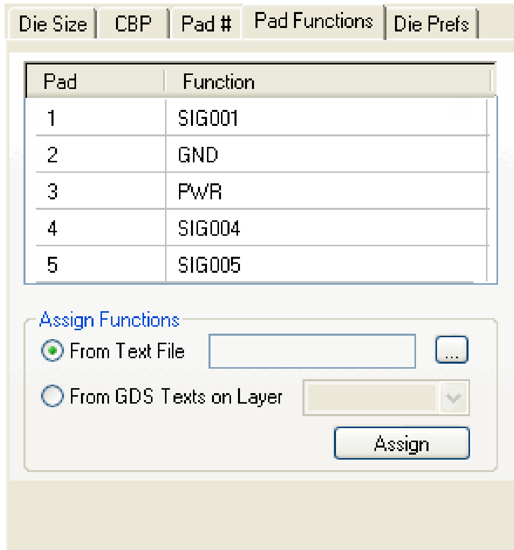
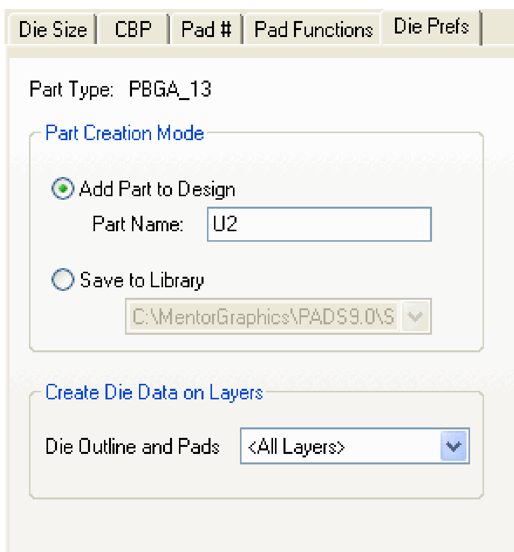


Figure 164. From GDSII Die Prefs tab



Objects

Table 166. Die Wizard - Create from GDSII File Dialog Box Fields

Field	Description
File	Displays the name of the text file you want to use to create the die. 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.

Table 166. Die Wizard - Create from GDSII File Dialog Box Fields (continued)

Field	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 on page 1281 • CBP on page 1281 • Pad # on page 1297 • Pad Functions on page 1297 • Die Prefs on page 1281
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. **Restriction:** 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 on page 1305 so you can select colors that help you preview your design.
Show All check box	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.

Table 167. Die Size tab contents

Field	Description
Set Size area — Select to set the die size and die outline position manually.	
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	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	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.
Select from GDS Shape area — Select to choose a GDS shape for the die outline. You can modify the Height control, but you cannot modify the other parameters in the Set Size area.	
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 Layers color. The die display adjusts dynamically as you choose values, to help you select the proper shape.

Table 168. CBP contents

Field	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.

Table 169. Pad # tab contents

Field	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 — On the bottom of the design.
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.

Table 170. Pad Functions tab contents

Field	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.
Assign Functions area	Specifies how you want to use to assign functions:

Table 170. Pad Functions tab contents (continued)

Field	Description
	• 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.

Table 171. Die Prefs tab contents

Field	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

Creating a Die from a GDSII File

Die Wizard - Create from Text File Dialog Box

To access: **BGA Toolbar** button > **Die Wizard** button > **From Text File** button

Use this dialog box to add a new die to the current design or library based on an imported text file.

Description

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.

Restriction:
This information applies only to the BGA toolkit.

The Die Wizard has 5 tabs:

- “Figure 166” on page 1290
- “Figure 167” on page 1291
- “Figure 168” on page 1291
- “Figure 169” on page 1292
- “Figure 170” on page 1292

Figure 165. Die Wizard - Create from Text File Dialog Box

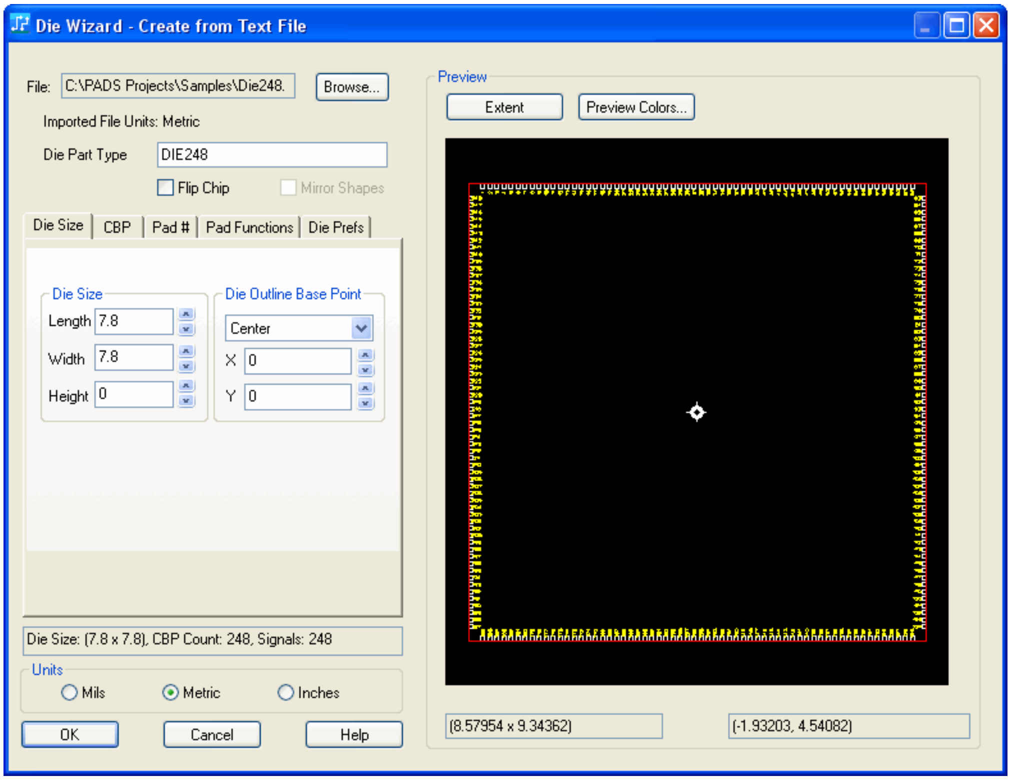


Figure 166. From Text File Die Size tab

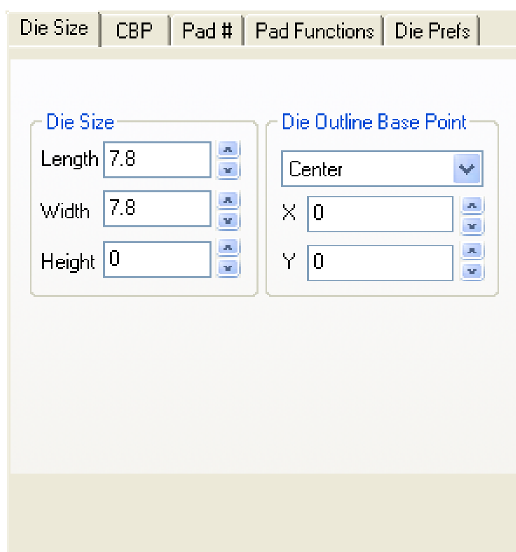


Figure 167. From Text File CBP tab

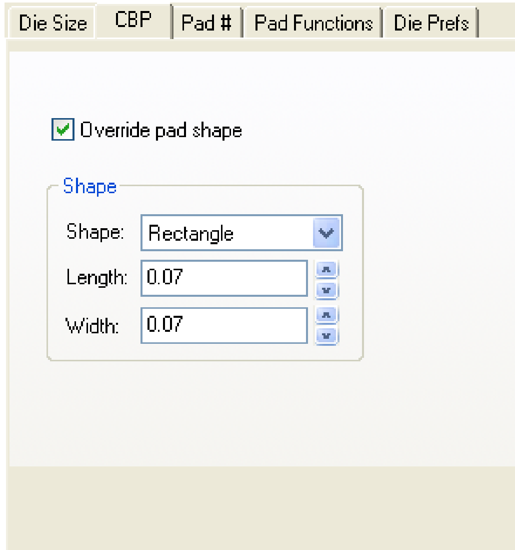


Figure 168. From Text File Pad # tab

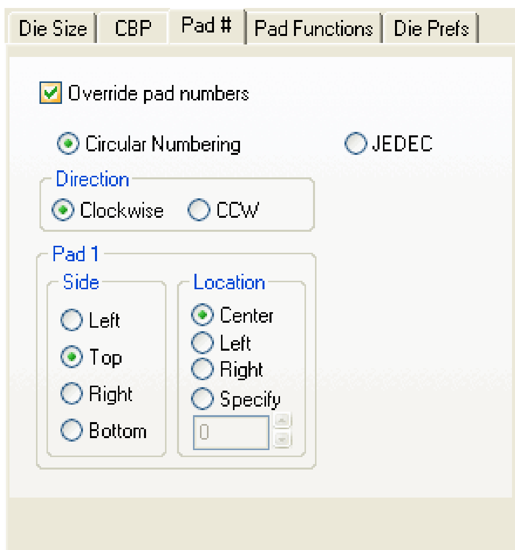


Figure 169. From Text File Pad Functions tab

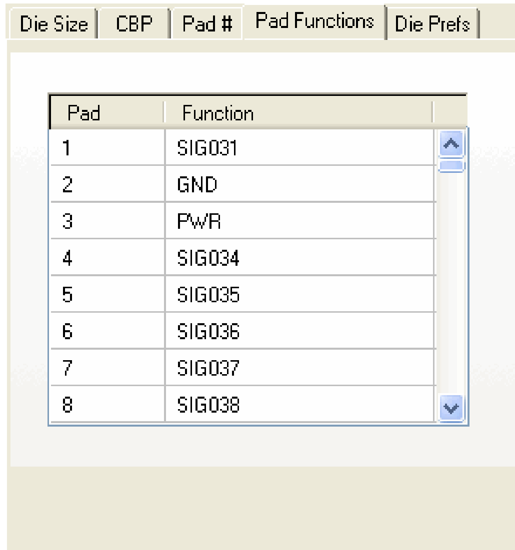
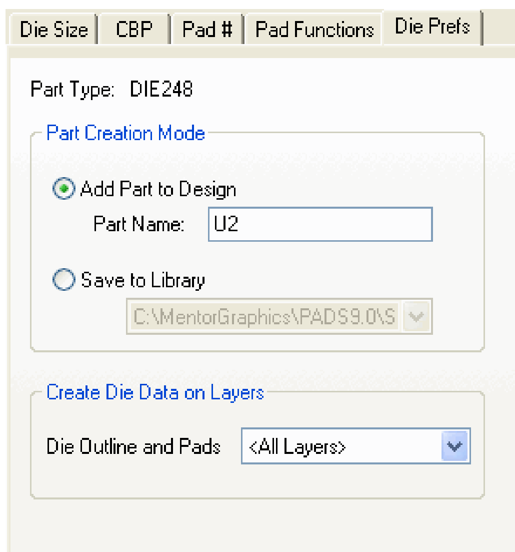


Figure 170. From Text File Die Prefs tab



Objects

Table 172. Die Wizard - Create from Text File Dialog Box Fields

Field	Description
File	Displays the name of the text file you want to use to create the die. 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.

Table 172. Die Wizard - Create from Text File Dialog Box Fields (continued)

Field	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 on page 1289 • CBP on page 1289 • Pad # on page 1289 • Pad Functions on page 1289 • Die Prefs on page 1289
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. **Restriction:** 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 on page 1305 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.

Table 173. Die Size tab contents

Field	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.

Table 173. Die Size tab contents (continued)

Field	Description
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.

Table 174. CBP tab contents

Field	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.

Table 175. Pad # tab contents

Field	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.

Table 175. Pad # tab contents (continued)

Field	Description
	Top — On the top of the design. Right — On the right side of the design. Bottom — On the bottom of the design.
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.

Table 176. Pad Functions tab contents

Field	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 177. Die Prefs tab contents

Field	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

Creating a Die from a Text File

Die Wizard - Create Parametrically Dialog Box

To access: **BGA Toolbar** button > **Die Wizard** button > **Parametrically** button

Use this dialog box to set up parameters for adding a new die to the current design or library.

Description

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

Restriction:
This information applies only to the BGA toolkit.

The Die Wizard has 5 tabs:

- “Figure 172” on page 1298
- “Figure 173” on page 1299
- “Figure 174” on page 1299
- “Figure 175” on page 1300
- “Figure 176” on page 1300

Figure 171. Die Wizard - Create Parametrically Dialog Box

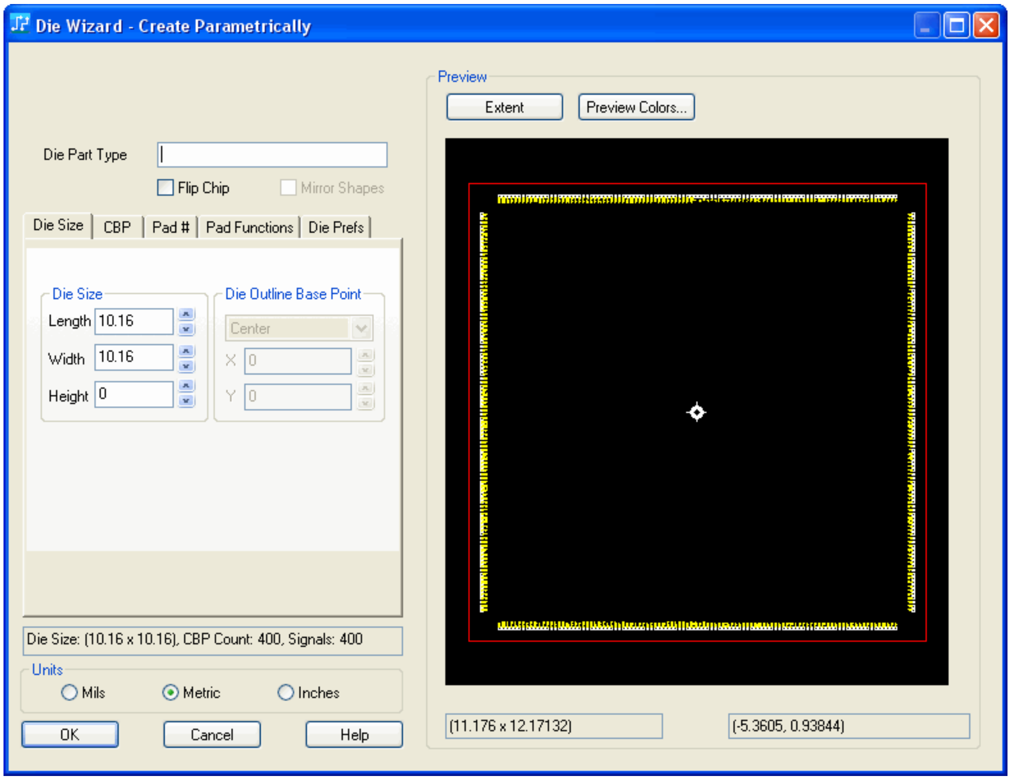


Figure 172. Die Size tab

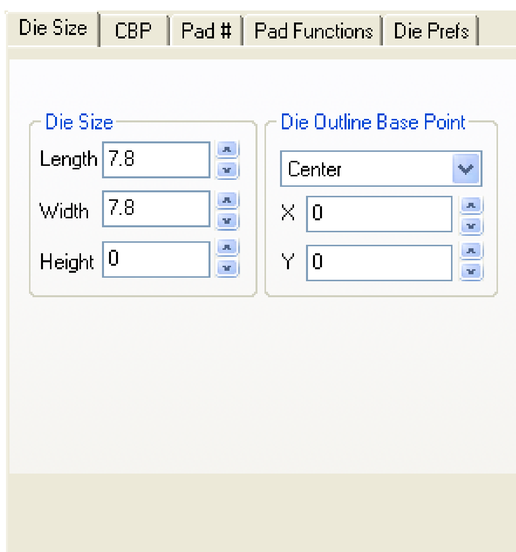


Figure 173. CBP tab

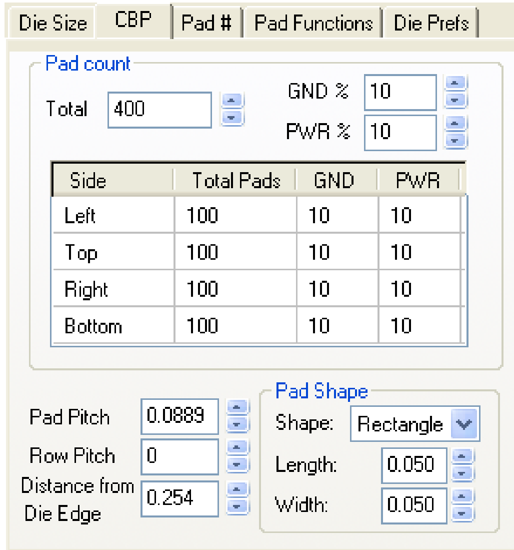


Figure 174. Pad # tab

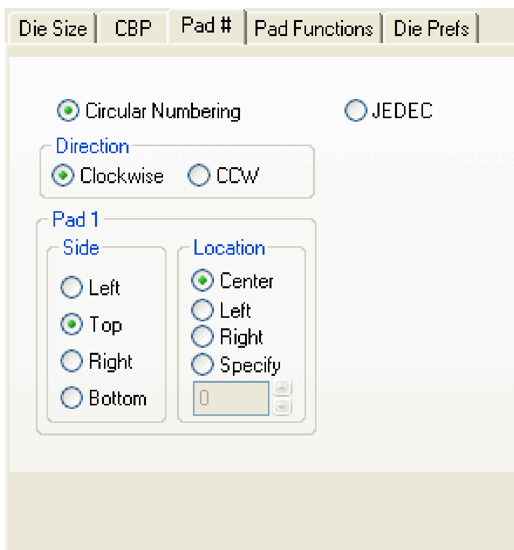


Figure 175. Pad Functions tab

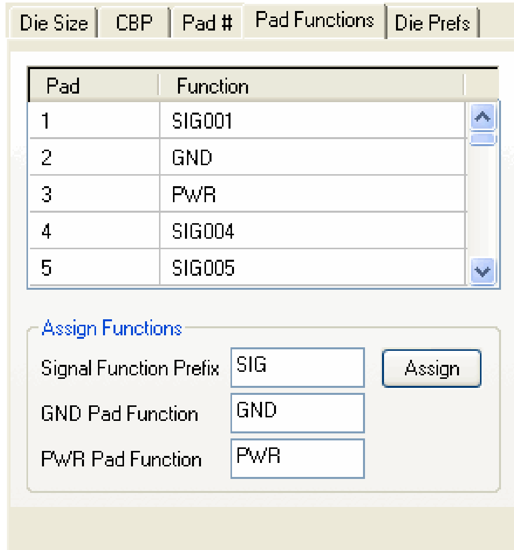
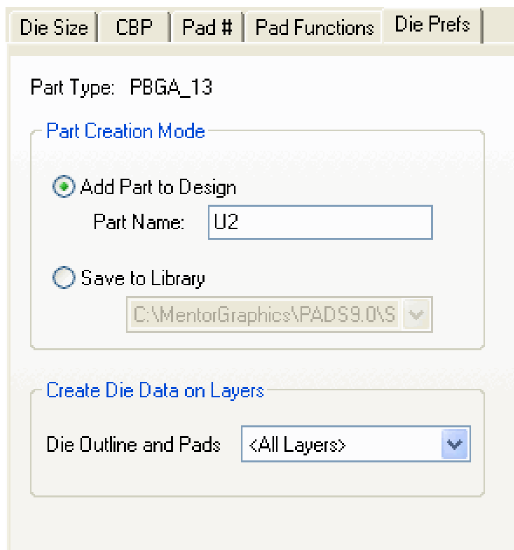


Figure 176. Die Prefs tab



Objects

Table 178. Die Wizard - Create Parametrically Dialog Box Fields

Field	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.

Table 178. Die Wizard - Create Parametrically Dialog Box Fields (continued)

Field	Description
tabs	• Die Size on page 1297 • CBP on page 1297 • Pad # on page 1297 • Pad Functions on page 1297 • Die Prefs on page 1297
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. **Restriction:** 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 on page 1305 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.

Table 179. Die Size tab contents

Field	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.

Table 179. Die Size tab contents (continued)

Field	Description
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.

Table 180. CBP contents

Field	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.
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 on page 1866: 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.

Table 181. Pad # tab contents

Field	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 — On the bottom of the design.
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.

Table 182. Pad Functions tab contents

Field	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.
Signal Function Prefix	Type or view the function prefix you want to use to derive the signal pad names.

Table 182. Pad Functions tab contents (continued)

Field	Description
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.

Table 183. Die Prefs tab contents

Field	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

Creating a Die Parametrically

Die Wizard Preview Colors Dialog Box

To access: **BGA Toolbar** button > **Die Wizard** button > any button > **Preview Colors** button

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.

Description

Setting colors on this dialog box does not change the colors you set in the Display Colors Setup Dialog Box.

Objects

Field	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.

Field	Description
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

Setting Die Preview Colors in the Die Wizard

Differential Pairs Dialog Box

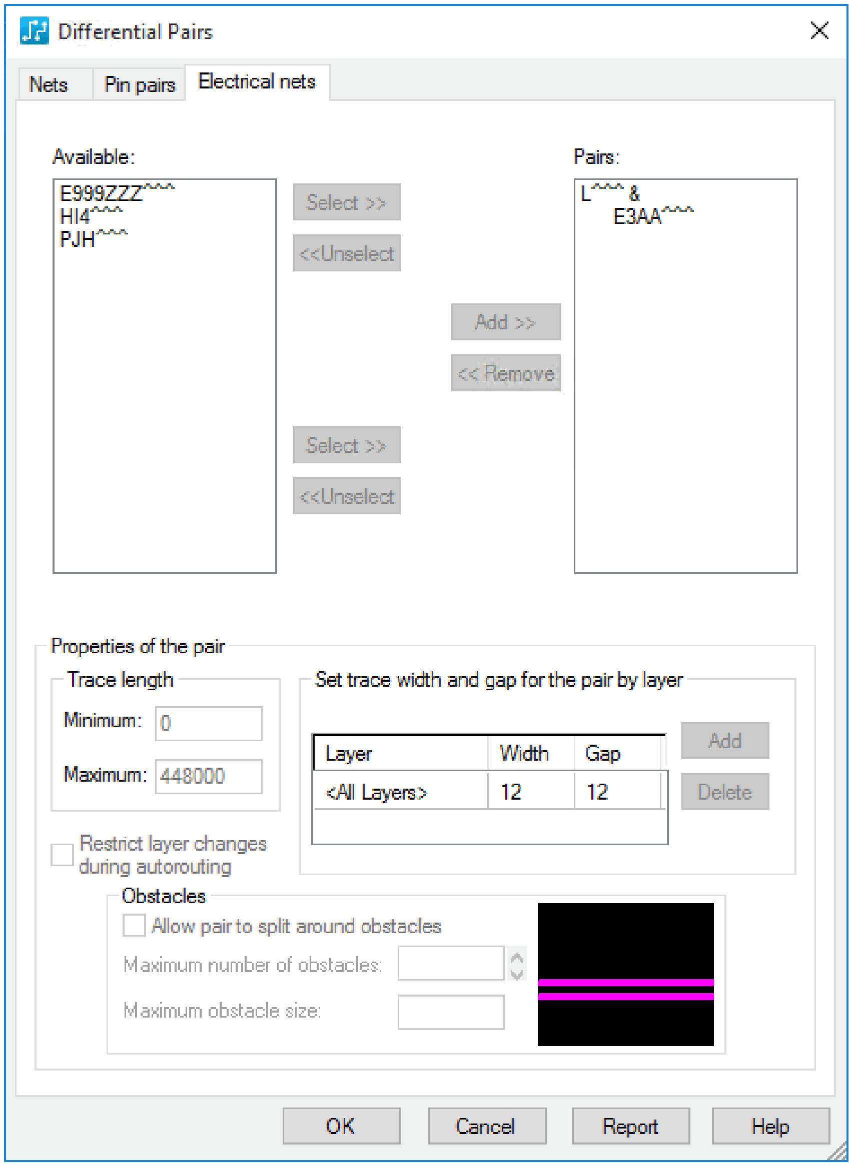
To access: **Setup > Design Rules** menu item > **Differential Pairs** button

Use the Differential Pairs dialog box and its tabs to identify nets, electrical nets, or pin pairs that behave electrically as differential pairs, and to define differential pair design rules.

Description

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 on page 1818, the minimum and maximum trace lengths and widths, and how to respond to obstacles on page 1843 in the controlled gap area.

Tip
You can define Differential Pairs Rules in SailWind Layout, but these rules are used in SailWind Router only. Layout does, however, respect the diff pair gap as a trace to trace clearance.



Objects

Field	Description
Available list	Lists the nets, pin pairs, or electrical nets available for differential pair creation. **Tip** Nets, electrical 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.
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.

Field	Description
<<Unselect	Moves the first and second items back to the Available list.
Add>>	Moves the selected items to the Pairs list. 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. 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. **Tip** Setting the differential pair width and gap per layer enables you to better control impedance. • 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 on page 404. • **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.
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 on page 1862 or end zone on page 1825 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.

Field	Description
Preview area	Shows the way in which the differential pairs will split around objects based on your selections.

Related Topics

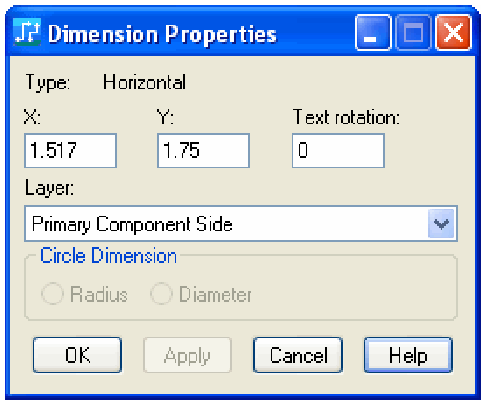
Creating Differential Pair Design Rules

Deleting a Differential Pair Design Rule

Dimension Properties Dialog Box

To access: Select a dimension > right-click > **Properties** popup menu item.

The Dimension Properties dialog box reflects the type of dimension (Vertical, Horizontal, or Aligned), that you have currently selected.



Objects

Field	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 Options dialog box > **Dimensioning** category > General subcategory on page 1520of 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

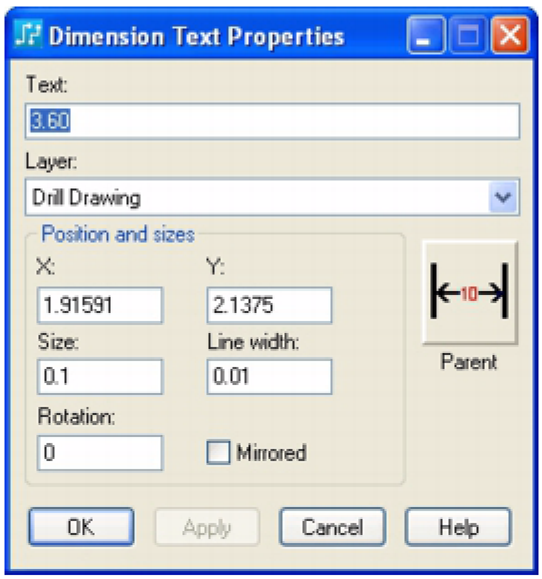
To access: Select dimension text > right-click > **Properties** popup menu item

The Dimension Text Properties dialog box displays information about the selected text string and provides several areas for modifying the string.

Description

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.



Objects

Field	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.

Field	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.

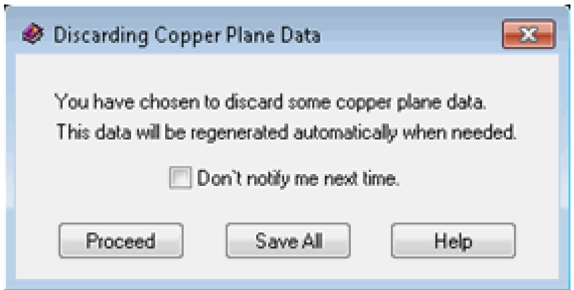
Related Topics

Dimensioning Process

Discarding Copper Plane Data Dialog Box

To access: The Discarding Copper Plane Data dialog box appears automatically when you save a file in which you have copper plane data, and have set Prompt to discard copper plane data on the Options dialog box.

Use the Discarding Plane Data dialog box to control which plane data is saved.



Objects

Field	Description
Don’t notify me next time	Select the check box to suppress this message for subsequent file saves.
Proceed	Saves only the plane polygons.
Save All	Saves all copper plane data and changes the option for future saves.

Disconnect Pin Dialog Box

To access: select a pin > ECO Toolbar > **Delete Connection** button

Use the Disconnect Pin dialog box to confirm the disconnection of the pin from the net to which it is attached and to delete the route segment(s) that are attached to the pin.

CAUTION:
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.

Objects

Field	Description
Delete Routes	Select to delete the route segment(s) that are connected to the pin. If you do not delete the routes, the pin will still appear as though it is 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.

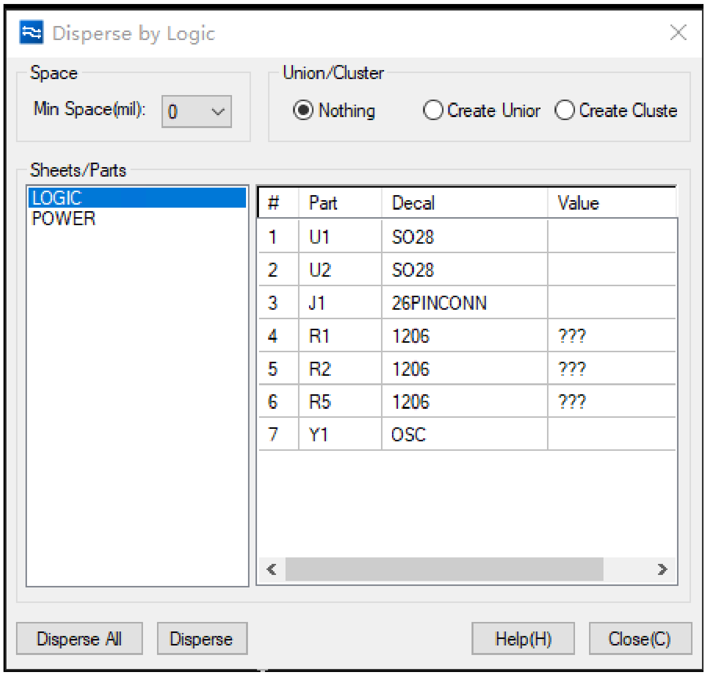
Related Topics

Deleting a Connection in ECO Mode

Disperse by Logic Dialog Box

To access: With nothing selected, right-click and click the **Disperse by Logic** popup menu item.

Use this dialog box to disperse parts while retaining their relative positions almost as indicated in the schematic sheet. You can also specify what action to take and perform on the parts to disperse.



Objects

Table 184. Parameter Description

Field	Description
Min Space	Specifies the minimum allowable spacing between parts to disperse in mils.
Union/Cluster area	Specifies what action to take and perform on the parts to disperse by clicking **Disperse All**/**Disperse** button, with options below: • Nothing (default): Nothing else • Create Union: Use the parts to create a union. • Create Cluster: Use the parts to create a cluster.
Sheets/Parts area	• Sheets list on the left: Lists all the schematic sheets. • Parts table on the right: Displays information of parts in the selected schematic sheet, including reference designator, PCB decal, and value. Changes depending on the selection made in the Sheets list.

Table 184. Parameter Description (continued)

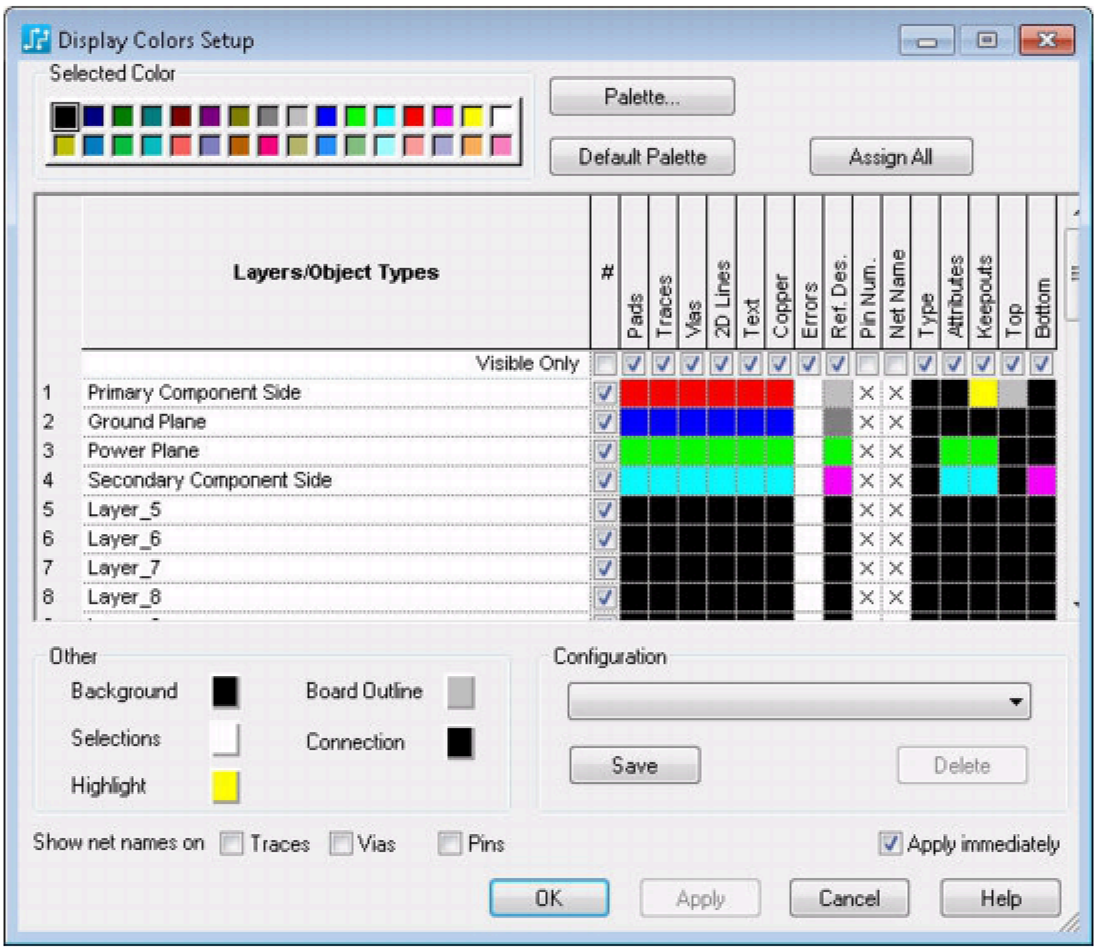
Field	Description
Disperse All	Disperses parts in all schematic sheets, except those glued or in unions/clusters.
Disperse	Disperses parts in the selected schematic sheet, except those glued or in unions/clusters. **Note:** If no space is available for placement, the parts attach to the pointer for manual placement in the design.

Display Colors Setup Dialog Box

To access: **Setup > Display Colors** menu item

Use the Display Colors Setup dialog box to Set display colors, save them, and restore them. Use this dialog to also change the color palette and make objects visible or invisible.

Tip
Changes you make to the color configuration in the Display Colors Setup dialog box do not apply to disabled layers.



Objects

Field	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: “Changing 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: “Changing 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.
Layers/Object Types matrix — Use this area to assign various colors to different objects on different layers. See also: “Setting Colors for the Design”.	
Layers (rows)	The rows of layers lists the layers as you have named them in the Layers Setup Dialog Box. **Tip** • 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.
Object Types (columns)	The columns list object types in the design. • Pin Numbers and Net Names — Sizing is controlled by the settings in the Display Options on page 1525. • Top/Bottom Objects—Used to control the visibility of component body outlines and placement outlines. Tiles in these columns associated with the Top and Bottom Component Side or Top/Bottom Silkscreen layers control the visibility of component body outlines. Tiles in these columns associated with Layer 20 control the visibility of Layer 20 (Placement) outline objects. **Tip** • 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.

Field	Description
	Note: 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.
Visible only	Lists only visible layers. A layer is visible if at least one tile is assigned a non-background color.
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. • Board outline — Applies to the board outline and board cut outs. • Connection — Unrouted pin pairs or the “ratsnest.”
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. **Tip** • 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 on page 1525.
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

Hiding Layers

Display Colors Setup Dialog Box in the Decal Editor

Setting Colors for the Design

Saving Color Assignments to a File

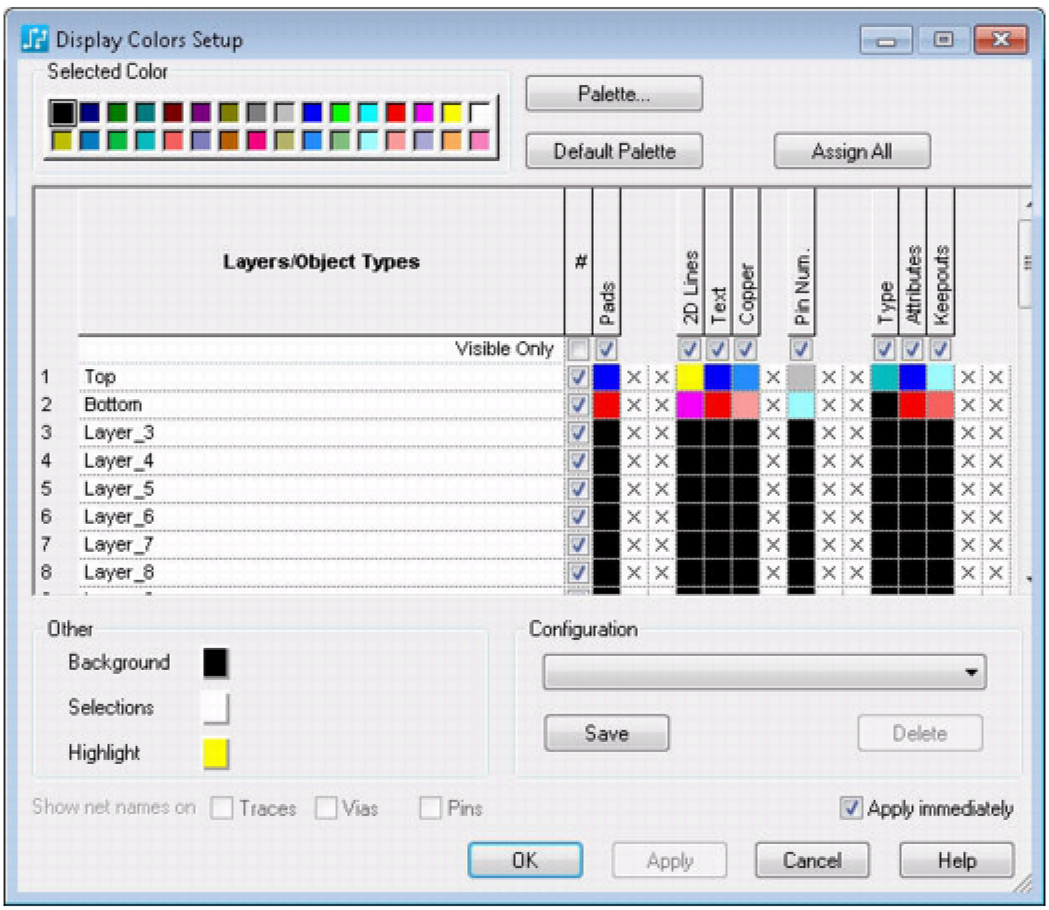
Display Colors Setup Dialog Box in the Decal Editor

To access: **Tools > PCB Decal Editor** menu item > **Setup > Display Colors** menu item

Use the Display Colors Setup dialog box to set display colors, save them, and restore them. Also use this dialog box to change the color palette and make objects visible and invisible.

Description

- 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.



Objects

Field	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: "Changing 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: "Changing 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.
Layers/Object Types matrix — Use this area to assign various colors to different objects on different layers.See also: "Setting Colors for the Design".	
Layers (rows)	The rows of layers lists the layers as you have named them in the Layers Setup Dialog Box. **Tip** • 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. **Tip** • 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.

Field	Description
	Tip The layer names that you assigned in the Layers Setup Dialog Box appear along the left side of this area. See also: “Setting Colors for the Design”.
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.
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

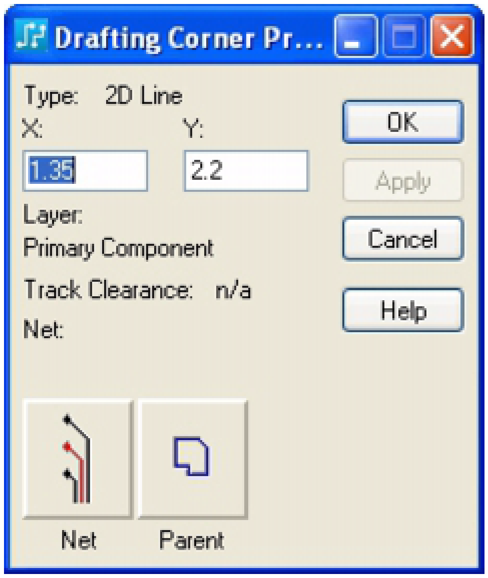
Setting Colors for the Design

Saving Color Assignments to a File

Drafting Corner Properties

To access: Select a drafting edge > right-click > **Properties** popup menu item

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.



Objects

Field	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 on page 1487.
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

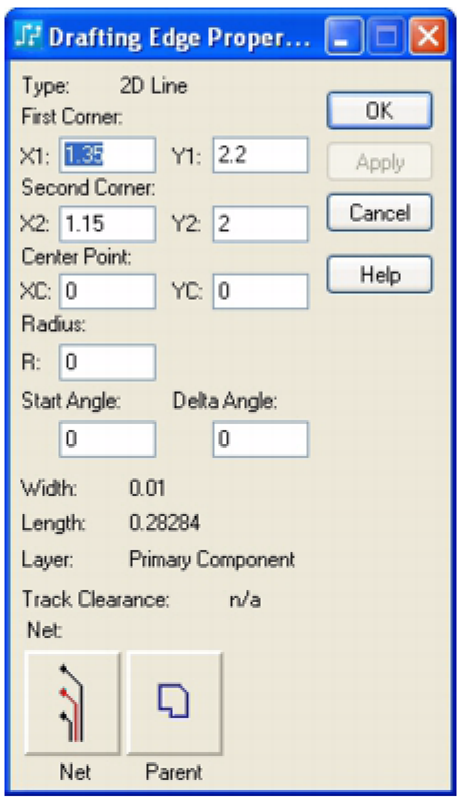
To access: Select a drafting edge > right-click > **Properties** popup menu item

Use the Drafting Edge Properties dialog box to view and edit an object type, layer assignment, trace clearance, or assigned net.

Description

You can move an edge by coordinates or an arc by radius or start/end angle, access net information, and select the parent shape. The following are 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 canceled.



Objects

Field	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. **Tip** • 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 cannot be interpreted, the command is canceled.
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 on page 1487.
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

To access: Choose a drafting shape, right-click and choose **Properties**

You can select and edit an entire drafting shape or a part of the shape. The properties you can edit depend on whether you select a corner of the drafting object, an edge of the drafting object, the entire drafting object, or the parent object.

Restriction:
Several of the options in this dialog box are unavailable if the object is part of a physical design reuse.

Description

Figure 177. Drafting Properties Dialog Box

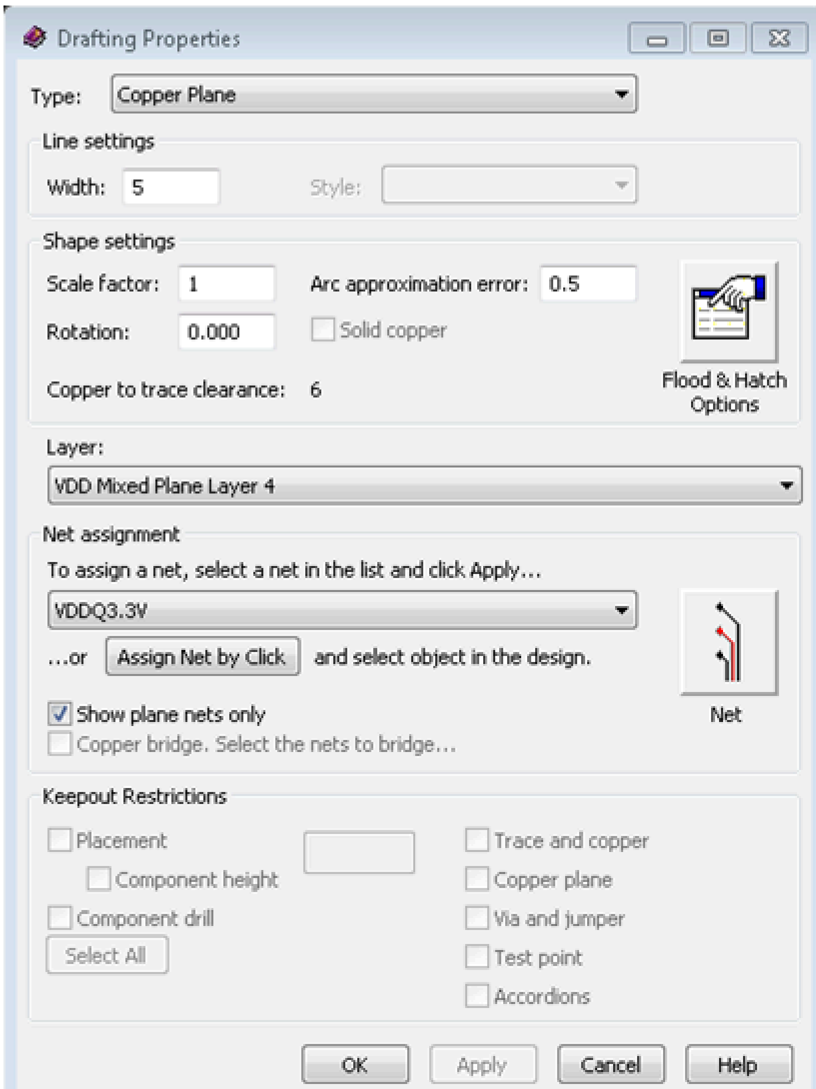
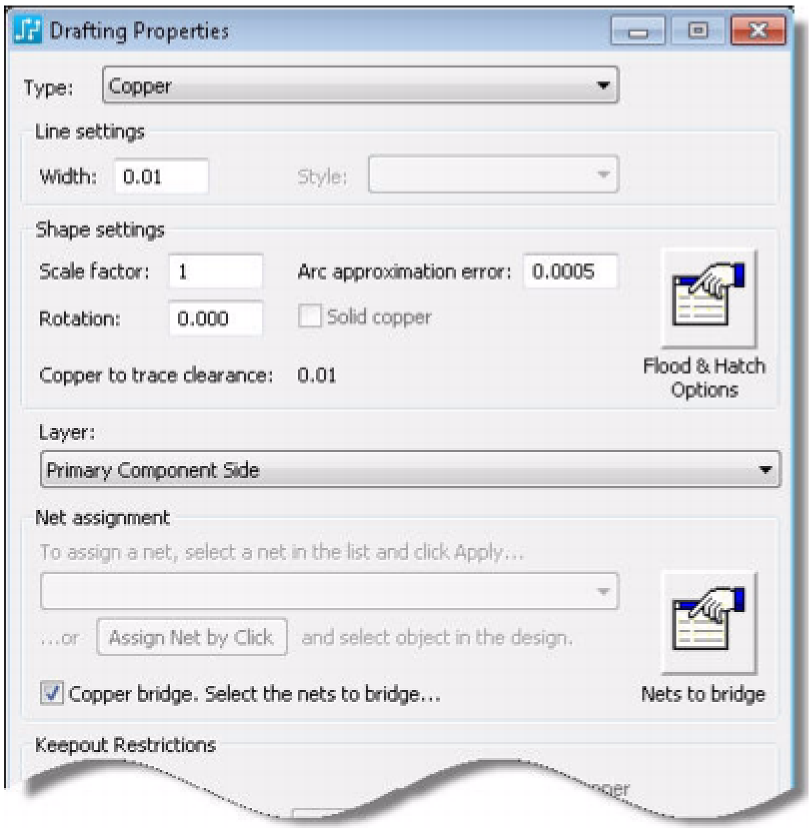


Figure 178. Showing the Nets to Bridge Button



Objects

Table 185. Drafting Properties Dialog Box Fields

Field	Description
Type	Converts the selected shape to a new one: Copper Plane (flooded), 2D Line, Board Outline, Board Cut Out, Copper Cut Out (also used for cut outs in Copper Planes), Keepout, or Copper. **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. **Restriction:** • 2D lines that have been combined cannot be changed into other shapes. You must first explode the 2D line combination and then try to change it to another object. • 2D lines that are continuous but are not a closed polygon can be changed only into a copper path.

Table 185. Drafting Properties Dialog Box Fields (continued)

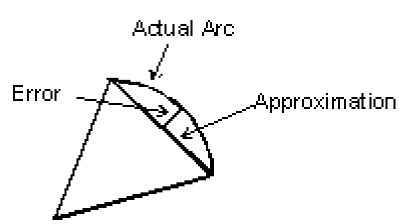
Field	Description
	• If there is no board outline in the design, the Board Cut Out type does not appear in the Drafting Properties dialog box Type list. • If there is a board outline in the design, the Board Outline option does not appear in the dialog box.
Line settings area	
Width	Specifies the width of the lines used for the outline and fill of the object if it applies.
Style	Specifies the style of the 2D line object: Solid, Dash, Dot, Dash dot, Dash double-dot. You can also set the styles before creating the 2D line using the LS Modeless Commands. **Restriction:** Available for 2D Line types only.
Shape settings area	
Scale factor	Change the Scale Factor to resize the selected shape. Values greater than 1 will increase the size while values less than 1 will shrink the object. A scale value of 2 will double the shape's size, a scale value of 0.5 will halve the shape's size. The object will scale up or down from the origin of the shape. The line width does not scale. An arc is too large if its radius is greater than 14 inches or its center is outside the database coordinate area. The database coordinate area is (-28, -28) to (28, 28) inches.
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	Fills the shape with solid copper despite the Width and Copper Hatch grid values which normally dictate the fill pattern of copper objects.
Flood & Hatch Options button	Opens the Flood and Hatch Options on page 1381 dialog box. **Restriction:** This button is available only for Copper Plane drafting objects.

Table 185. Drafting Properties Dialog Box Fields (continued)

Field	Description
Layer list	Specifies the layer on which this object is located. **Restriction:** Copper and copper planes 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 cannot do this in the Layout Editor.
Net assignment area	
Net assignment list	Specifies the net to assign to the drafting object. You can also use the **Assign Net by Click** button also found in this dialog box. **Restriction:** • 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.
Assign Net by Click button	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. 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.
Show plane nets only	When selected, limits the display of nets in the “Net assignment” list to only plane nets. Use this feature when you want to assign nets to plane areas on a split/mixed plane, and you want to limit the list to display only those nets assigned for use on the plane layer. This check box appears only when you select “Copper Plane” as the object type and you select a split/mixed plane layer from the Layer list. You can define the list of plane nets in the Layers Setup dialog box by clicking the **Assign Nets** button for a Split/Mixed layer (choose the **Setup > Layer Definition** menu item to open the Layers Setup dialog box).
Copper bridge, Select the nets to Bridge...	When selected, designates the drafting object as 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 is not flagged as a violation when it is physically bridging

Table 185. Drafting Properties Dialog Box Fields (continued)

Field	Description
	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 option available only to Copper drafting objects. See also: "Bridging Nets with Copper".
Nets to bridge button	Opens the Net Association dialog box on page 1486 in order to select the nets to bridge using the copper. Restriction: This is only available when Copper bridge, Select the nets to Bridge check box is selected. See also: "Bridging Nets with Copper".
Net button	Opens the Net Properties dialog box on page 1487 in order to view and change the properties of the net assigned to the drafting object. **Restriction:** The net does not function if there is no net assigned to the drafting object.
Keepout Restrictions area This area is only available to Keepout drafting objects. If you set a layer before assigning restrictions, there are restrictions that are not available.	
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. **Tip** When set on a decal-level keepout, this prevents placement of other components, not itself.
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. **Tip** The height restriction does not apply to the component that has the keepout but other components in the design. See also: "Restricting Heights in Areas of Component Layers". **Restriction:** Unavailable unless you select the Placement check box.
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.) **Restriction:**

Table 185. Drafting Properties Dialog Box Fields (continued)

Field	Description
	• 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 is not prevented but the Verify Design utility reports any copper located within the keepout area.
Copper plane	Prevents copper 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.
Accordions	Restricts the placement of accordions within the keepout area, when design rule checking is enabled. **Tip** • While you can set this option in SailWind Layout, it applies to SailWind 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.

Related Topics

Drafting Object Properties

Drill Symbols Dialog Box

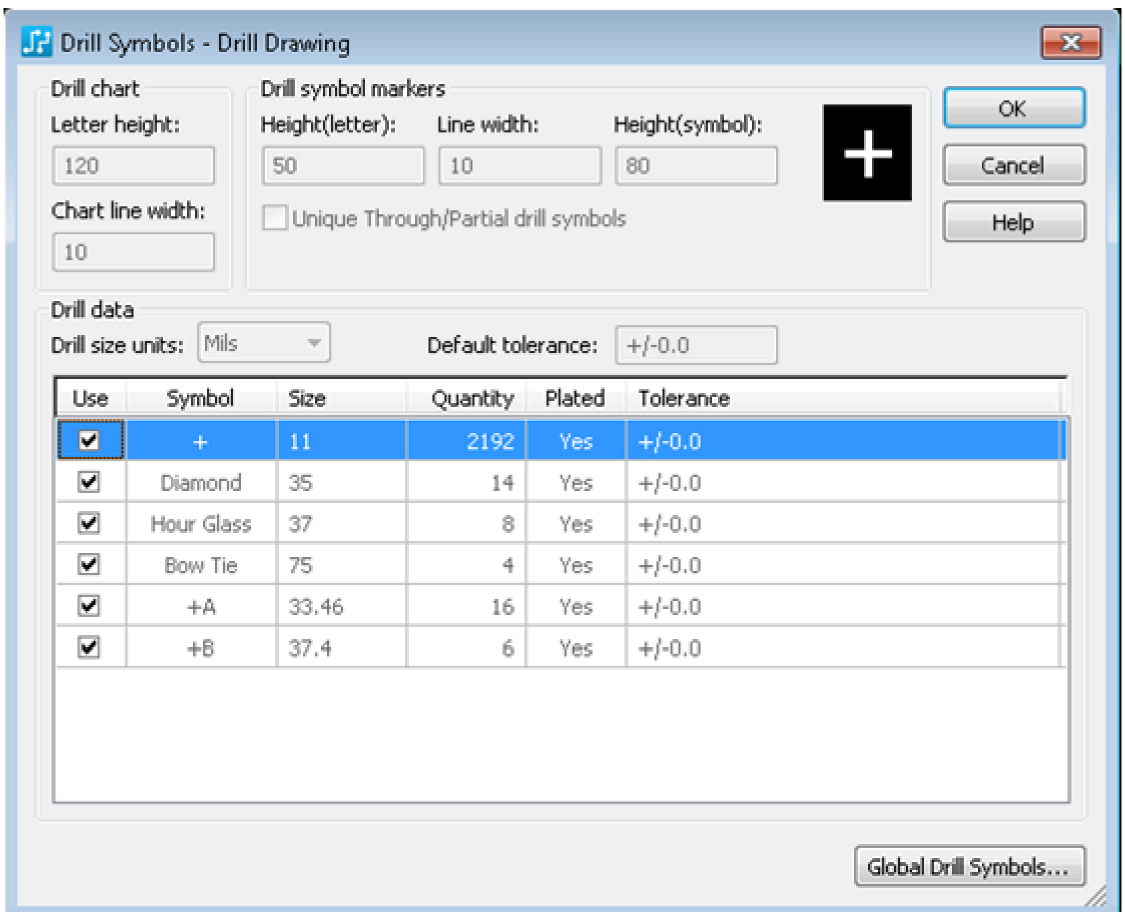
To access:

- Click the **File > CAM** menu item > click the **Add** button > select “Drill Drawing” from the Document Type dropdown list > select a layer > click the **Options** button > click the **Drill Symbols** button

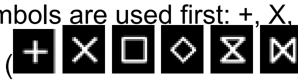
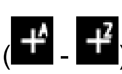
- Click the **File > CAM** menu item > select a drill drawing from the Document Name list > Click **Edit** > click the **Options** button > click the **Drill Symbols** button

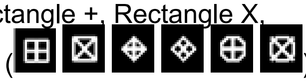
Use the Drill Symbols dialog box to include or exclude the use of certain drill symbols for a selected document.

Note:
To change any settings in the dialog box, click **Global Drill Symbols**.



Objects

Field	Description
Global Drill Symbols	Opens the “Global Drill Symbols dialog box” on page 1387, enabling you to edit drill drawing legend and marker parameters that are read-only in this dialog box.
Drill Chart area	
Letter Height	Displays the global height setting of letters in the drill drawing in design units.
Chart Line Width	Displays the width of the chart lines in design units.
Drill symbol markers area	
Height (letter)	Displays the letter height of the drill symbol marker.
Line Width	Displays the line width for the drill marker symbol and for all text.
Height (symbol)	Displays the line width for the drill marker symbol.
Unique Through/Partial drill symbols	When selected (in the Global Drill Symbols dialog box), the Through/Partial column displays in the drill data table of this dialog box. This column enables you to differentiate between partial and through-hole drills.
Drill Data area	
Drill size units	Displays the units to use for the drill size.
Default tolerance	Displays the tolerance that appears by default in the Tolerance column of the table.
Table Contents — Displays the drill data of symbols that are defined in the Global Drill Symbols dialog box and assigned to items in the drill drawing through the Select Items Dialog Box. For example, if no pads or vias are added from the electrical layers, the chart does not display drill information pertaining to those locations.	
“Use” check box	Select the check box for each symbol that you want to enable for the selected drill drawing. You can edit the symbols by clicking **Global Drill Symbols** to open the “Global Drill Symbols dialog box.” on page 1387.
Symbol and Size columns	Displays the symbols that 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 ()

Field	Description
	• Six more graphical symbols are assigned: Rectangle +, Rectangle X, Diamond +, Diamond X, Circle +, and Circle X () • Rectangle +A through Rectangle +Z are assigned ( - )
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.
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.
Through/Partial column	Indicates whether the drill is for a partial or through-type hole. **Tip** • This column displays only if the Unique Through/Partial column check box in the “Drill symbol markers” area is selected. • When this column displays, if both through and partial drills of a given drill size exist, they are counted separately and appear in separate rows. When this column does not display, through and partial drills of a given size are totaled together and appear in a single row.
Tolerance	Displays the tolerance of the drill size.

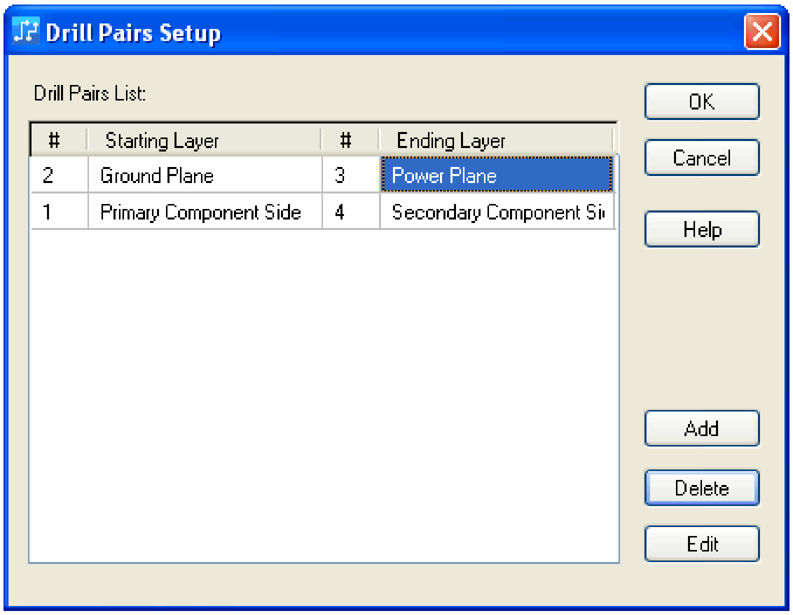
Related Topics

Global Drill Symbols Dialog Box

Drill Pairs Setup Dialog Box

To access: **Setup > Drill Pairs**

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 do not drill together.



Objects

Field	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.

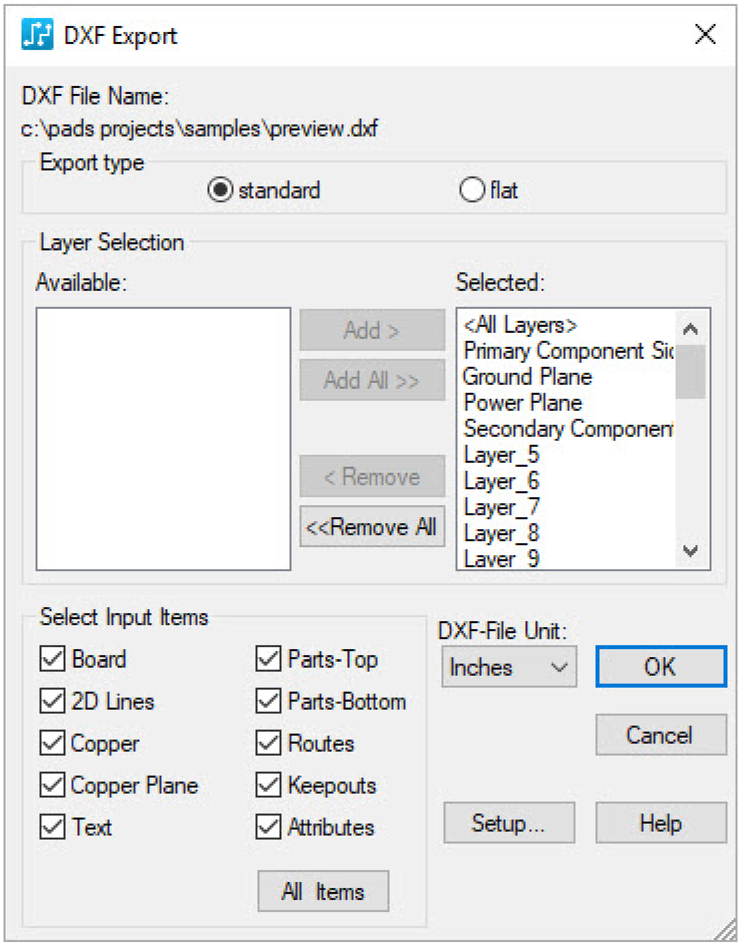
Related Topics

Defining Drill Pairs

DXF Export Dialog Box

To access: **File > Export** menu item > select DXF Files > **Save**

Use the DXF Export dialog box to export DXF formatted files.



Description

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.

Objects are exported as follows:

- 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.)

The *.dxf* file format is revision 12.

Objects

Field	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 SailWind Layout design. Items exported from the layer have the SailWind 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 SailWind 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.
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).

Field	Description
	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 Plane	Select this check box to export copper planes. This exports either hatch outlines or pour outlines depending on the Display mode setting in the "Hatch and Flood options" on page 1511. • 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" on page 1511: Minimum Hatch Area, Smoothing Radius, View, and Display Mode. • Individual custom Flood on page 1381 settings if they exist as Copper Plane 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 Plane Cut Outs Either Pour outline cut outs or Hatch outline cut outs are exported depending on the Display mode setting in the "Hatch and Flood options" on page 1511. • 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).
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)

Field	Description
	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
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).

Field	Description
	• **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).
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” on page 1511. The translator uses the Keepout hatch grid setting on page 1537.
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.

Field	Description
	Attribute values are exported, under Standard export, to the LABEL_ATTRIBUTE layer. **Tip** • 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.

Related Topics

Exporting DXF Files

DXF Format

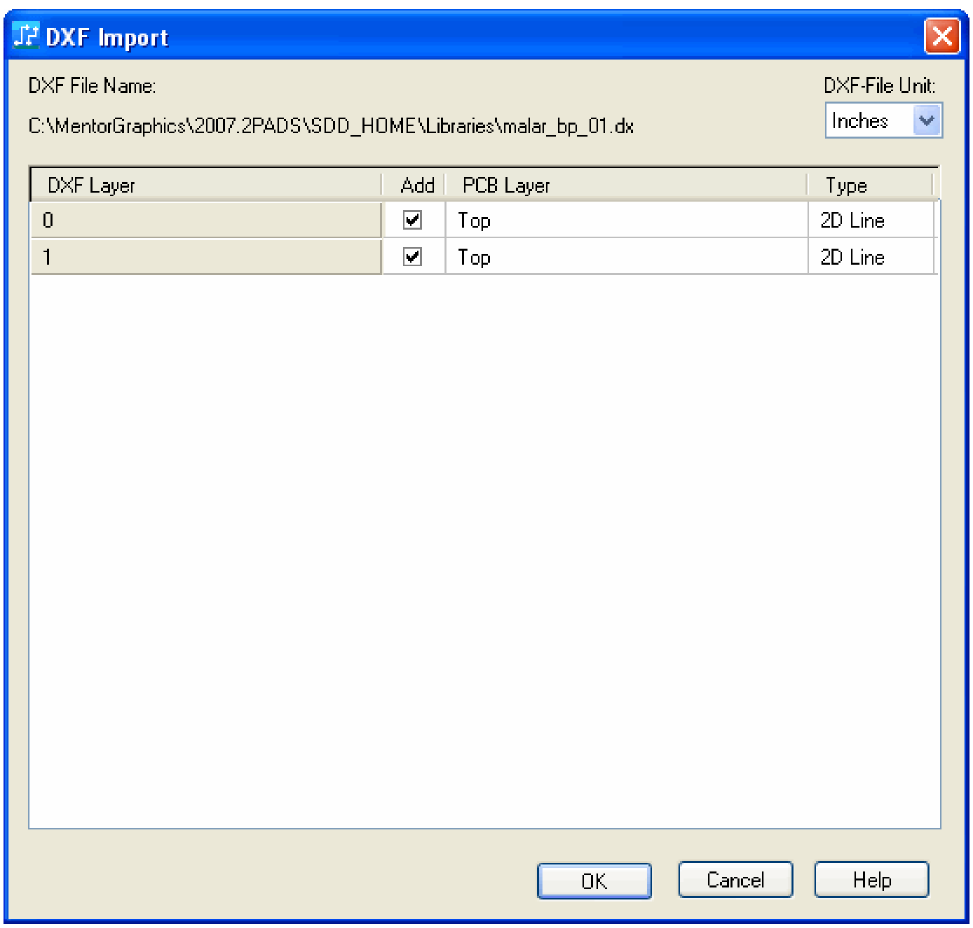
DXF Import Dialog Box

To access: **Drafting Toolbar** button > **Import DXF File** button > Open a *.dxf* file

You can import specialized shapes of DXF format into your decal or into the design using the AutoCAD 2004 DXF format.

Tip
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 SailWind 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 cannot 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.



Objects

Field	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

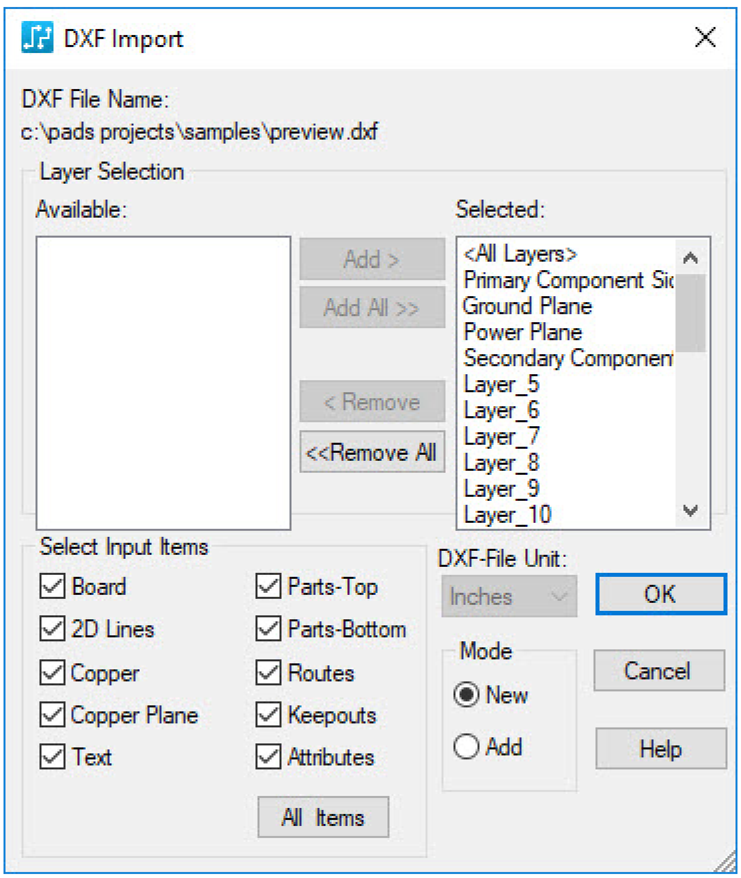
DXF Import Dialog Box

To access: **File > Import** menu item > set file type to DXF Files (*.dxf) > select a DXF file > **Open**

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.

Tip
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 cannot be changed. This allows a whole design, exported from SailWind Layout, to be re-imported 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 SailWind Layout layers.



Objects

Field	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 SailWind 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 SailWind 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.
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 Plane	Select this check box to import copper planes. • 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:

Field	Description
	• "Copper Planes / Hatch and Flood options" on page 1511: Minimum Hatch Area, Smoothing Radius, View, and Display Mode. • Individual custom Flood on page 1381 settings if they exist as Copper Pour attributes: Hatch Grid, Smoothing Radius, Hatch Direction, and Flood over Vias.
	Copper Plane Cut Outs Either Pour outline cut outs or Hatch outline cut outs are imported. • Plane 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).
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
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.

Field	Description
	• **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).
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.

Field	Description
	The translator uses the Keepout hatch grid setting on page 1537.
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. **Tip** • 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 SailWind Layout. Reads SailWind Layout layer and via information in the file.
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 SailWind Layout.

Related Topics

Importing DXF Files

DXF Format

Chapter 49
GUI Reference Elements E Through G

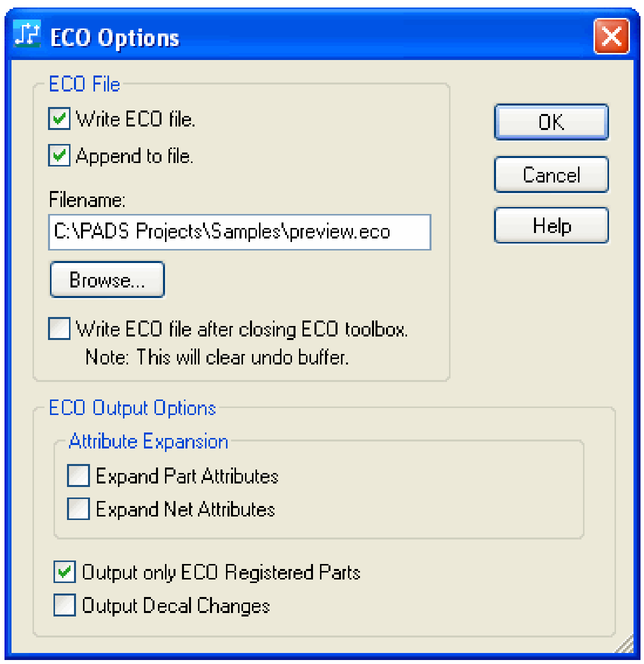
Read the sections that follow to learn about dialog box elements in SailWind Layout.

ECO Options Dialog Box

To access:

- **Tools > ECO Options** menu item
- **ECO Toolbar** button (when you select it the first time in a session)
- **ECO Toolbar** button > **ECO Options** button

Use the ECO Options dialog box to specify whether and how an ECO file is written when you make engineering (netlist) changes to a design using the ECO Toolbar.



Objects

Table 186. ECO Options Dialog Box Fields

Field	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: "Table 187".
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: "Table 187".

Table 186. ECO Options Dialog Box Fields (continued)

Field	Description
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: "Table 187".
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-Registered Parts"
Output Decal Changes	Select to record decal changes to the ECO file. If this option is selected and if SailWind Layout is not in ECO mode: • When you try to right-click and click the **Edit Decal** popup menu item, 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 on page 1632, 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. • When you try to change the decal of a component to an alternate decal using the Component Properties dialog box, on page 1191 you get the prompt message "You can modify decals only in ECO mode or if ECO option Output Decal Changes is off." You must simply open the ECO Toolbar to edit the decal.

Table 187. ECO Options - Check Box Combinations

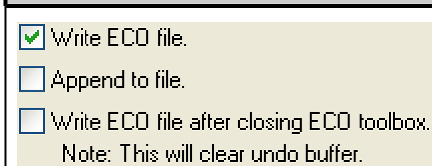
Check Box Combination	Result
	Creates a new *.eco* file according to the pathname in the Filename box. Overwrites any existing *.eco* file with the same name at that location with the start of a new software session. During the software session, retains appended additional ECO changes in memory, until the changes are saved to the *.eco* file whenever you: • Close the ECO Toolbar and you Save the *.pcb* file. • Exit the software or open a new design.

Table 187. ECO Options - Check Box Combinations (continued)

Check Box Combination	Result
☑ Write ECO file. ☑ Append to file. ☐ Write ECO file after closing ECO toolbox. Note: This will clear undo buffer.	Creates a new *.eco* file according to the pathname in the Filename box, or if it already exists, appends to the *.eco* file listed in the Filename box. Appends additional ECO changes to the *.eco* file, even when you start a new software session. Writes changes to the *.eco* file whenever you: • Close the ECO Toolbar and you Save the *.pcb* file. • Exit the software or open a new design.
☑ Write ECO file. ☐ Append to file. ☑ Write ECO file after closing ECO toolbox. Note: This will clear undo buffer.	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 with the start of a new software session. During the software session, retains appended additional ECO changes in memory, until the changes are saved to the *.eco* file whenever you: • Close the ECO Toolbar. • Exit the software or open a new design.
☑ Write ECO file. ☑ Append to file. ☑ Write ECO file after closing ECO toolbox. Note: This will clear undo buffer.	Creates a new *.eco* file according to the pathname in the Filename box, or if it already exists, appends to the *.eco* file listed in the Filename box. Appends additional ECO changes to the *.eco* file, even when you start a new software session. Writes changes to the *.eco* file whenever you: • close the ECO Toolbar. • exit the software or open a new design.

Related Topics

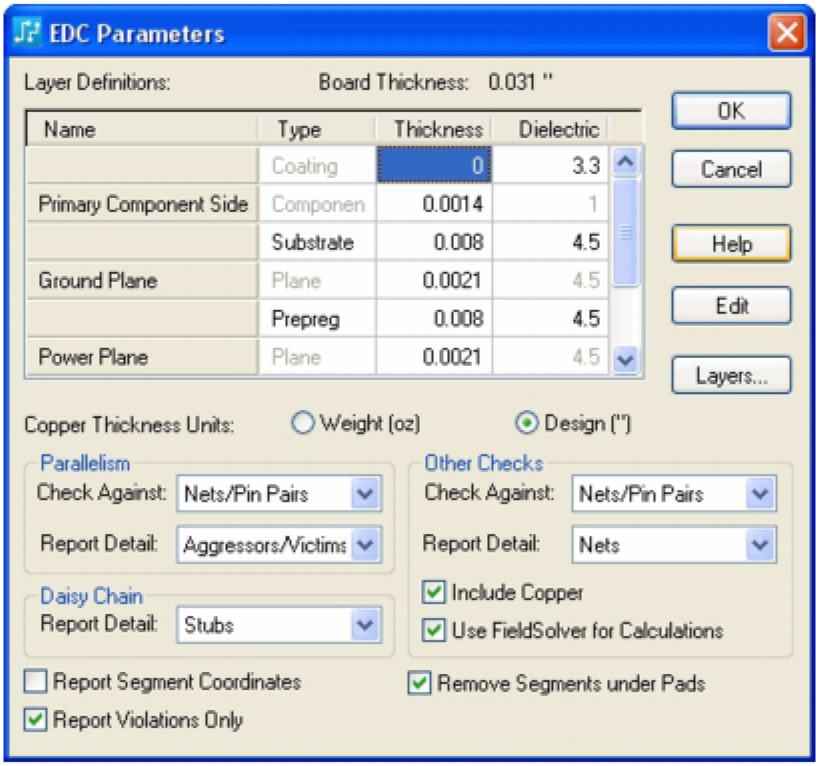
Comparing Designs

Recording ECO Changes

EDC Parameters Dialog Box

To access: **Tools > Verify Design** menu item > High Speed check > **Setup** button > **Parameters** button

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.



Objects

Field	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.

Field	Description
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
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.

Field	Description
	• **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.
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: “PADS HyperLynx SI/PI Guide”.
Report Segment Coordinates	Specifies to include segment coordinates in reports where “Segments” has been selected in the Report Detail in any one of the 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

Setup of EDC Parameters

Edit CAM Document Dialog Box

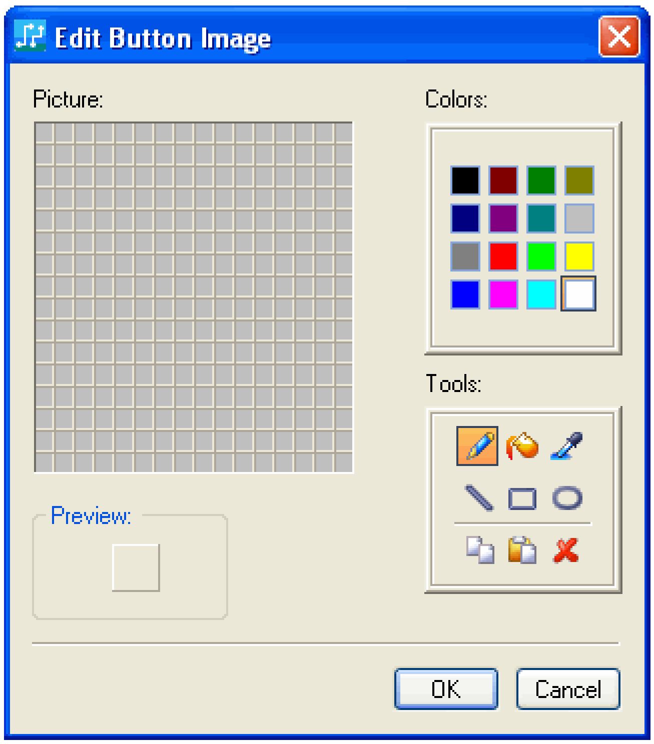
This dialog box is documented in a combined topic (Add/Edit Document dialog box).

See: Add/Edit Document Dialog Box

Edit Button Image Dialog Box

To access: **Tools > Customize** menu item > **Commands** tab > **New** button > Select User-Defined Image > **New** or **Edit**

Use the Edit Button Image dialog box to create or edit button icons.



Objects

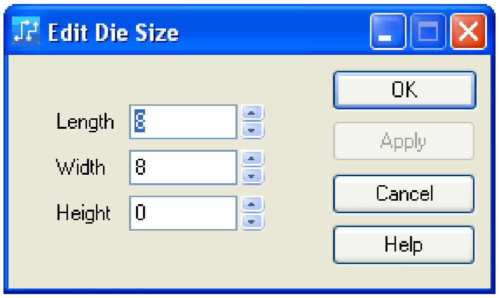
Field	Description
Colors area	Select a color to use with the tools
Tools area	Select a tool to draw/edit the picture or icon of the button

Edit Die Size Dialog Box

To access: **BGA Toolbar** button > **Wire Bond Editor** button > click the BGA > right-click > **Edit Die Size** popup menu item

Use the Edit Die Size dialog box to change the size of a die.

Note:
This information applies only to the BGA toolkit.



Objects

Field	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

Editing the Die Size

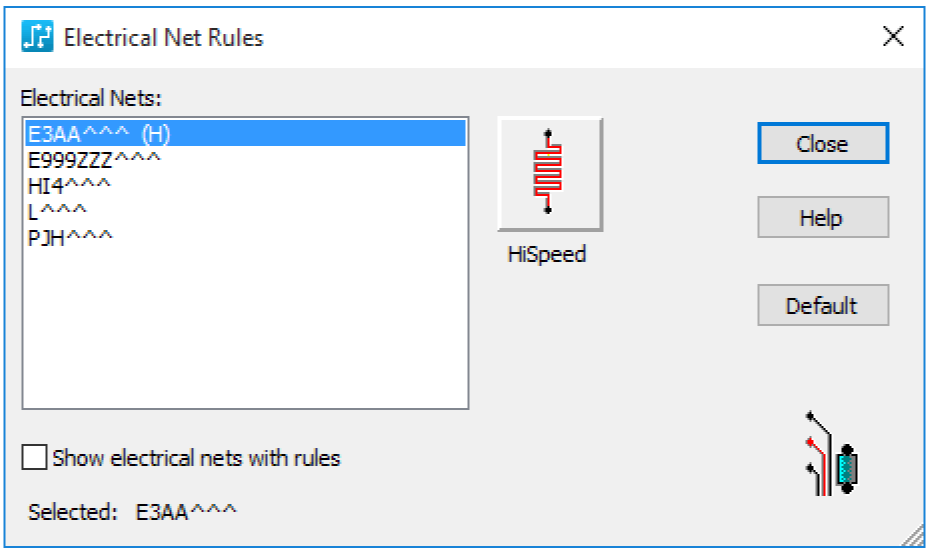
Electrical Net Rules Dialog Box

To access:

- **Setup > Design Rules** menu item > **Electrical Nets** button
- Select an electrical net > right-click > **Show Rules** popup menu item

Use the Electrical Net Rules dialog box to select a list of electrical nets. You can also assign HiSpeed rules or clear all rules defined for them.

Description



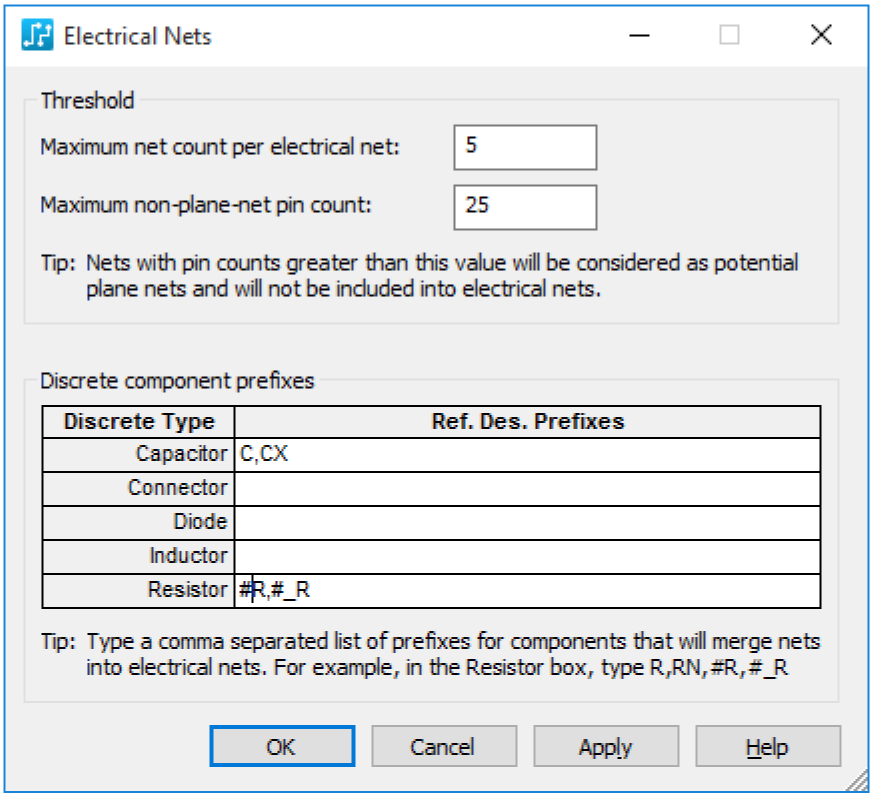
Objects

Field	Description
Electrical Nets list	Lists the electrical nets in the design. **Tip** (H) identifies electrical 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 electrical nets.
Show electrical nets with rules	Lists only the electrical nets that have rules.
Default button	Removes all rules from the selected electrical nets.

Electrical Nets Dialog Box

To access: **Setup > Electrical Nets** menu item

Set options for electrical nets and create electrical nets based on component reference designators. The options set limits on the number of nets allowed in an electrical net, and on the number of pins allowed in a single net included in an electrical net.



Objects

Field	Description
Maximum net count per electrical net	Specify the maximum number of nets to allow in an electrical net to prevent creating electrical nets with an unreasonable number of nets. This limit applies to all methods of creating electrical nets.
Maximum non-plane-net pin count	Specify the maximum pin count for nets included in electrical nets to prevent potential plane nets from being included in an electrical net. The electrical net can be split, truncated, or not created at all, depending on the location of the faulty net. (Plane nets are not allowed in electrical nets.) Virtual pins are not included in a net's pin count. This limit applies to all methods of creating electrical nets.

Field	Description
Discrete component prefixes	Specify the refdes prefixes of the associating components to create new electrical 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. Discrete Type column — the 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 electrical net cannot 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 electrical nets.

Related Topics

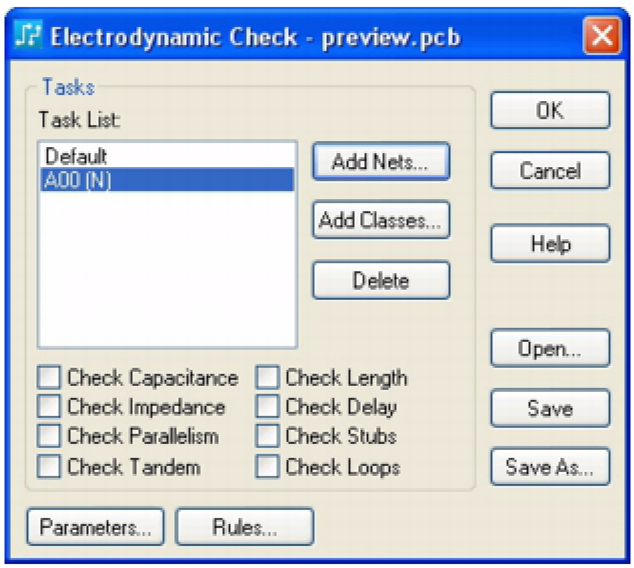
Conditions Governing Electrical Net Creation

Electrodynamic Check Dialog Box

To access: To access: **Tools > Verify Design** menu item > High Speed check > **Setup** button

Use the Electrodynamic Check dialog box to enable high-speed checks for individual nets and classes or for the whole design.

Restriction:
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.



Objects

Field	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 on page 1057.
Add Classes button	Opens the Add Class Tasks dialog box on page 1057.
Delete button	Removes the task from the Task List.

Field	Description
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.
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 on page 1402 or Conditional Rule Setup on page 1199 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 *\SailWind 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

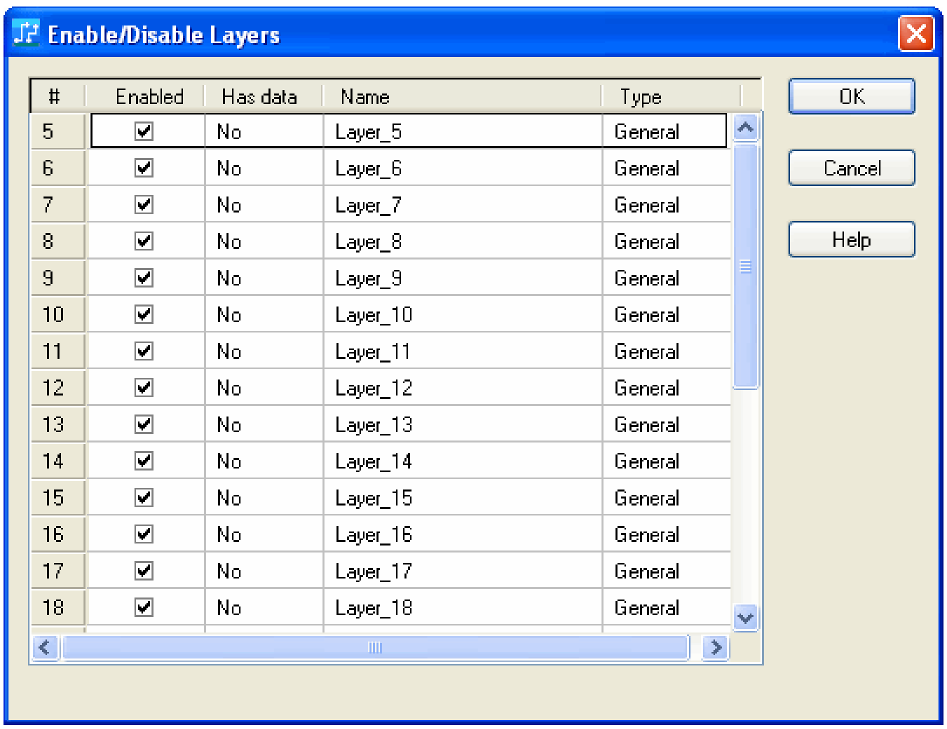
Setup of High Speed (Electrodynamic) Checking

Enable/Disable Layers Dialog Box

To access: **Setup > Layer Definition** menu item >**Enable/Disable** button

Use the Enable/Disable Layers dialog box to specify which nonelectrical layers to enable or disable. This shows or hides layers in layer lists, such as 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.



Objects

Field	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

Hiding or Displaying Non-electrical Layers in Layer Lists

Hiding Layers

Error Detected Dialog Box

To access: This dialog box is inaccessible unless the software crashes and crash detection is enabled in the software *.ini* file.

The Crash Detected dialog box opens at a crash and allows you to save a report of the SailWind environment as well as pertinent files into a compressed Dump File. You can then submit this file to SailWind Software Customer Support. You can attach feedback to this report, and optionally, the BMW media and project files.

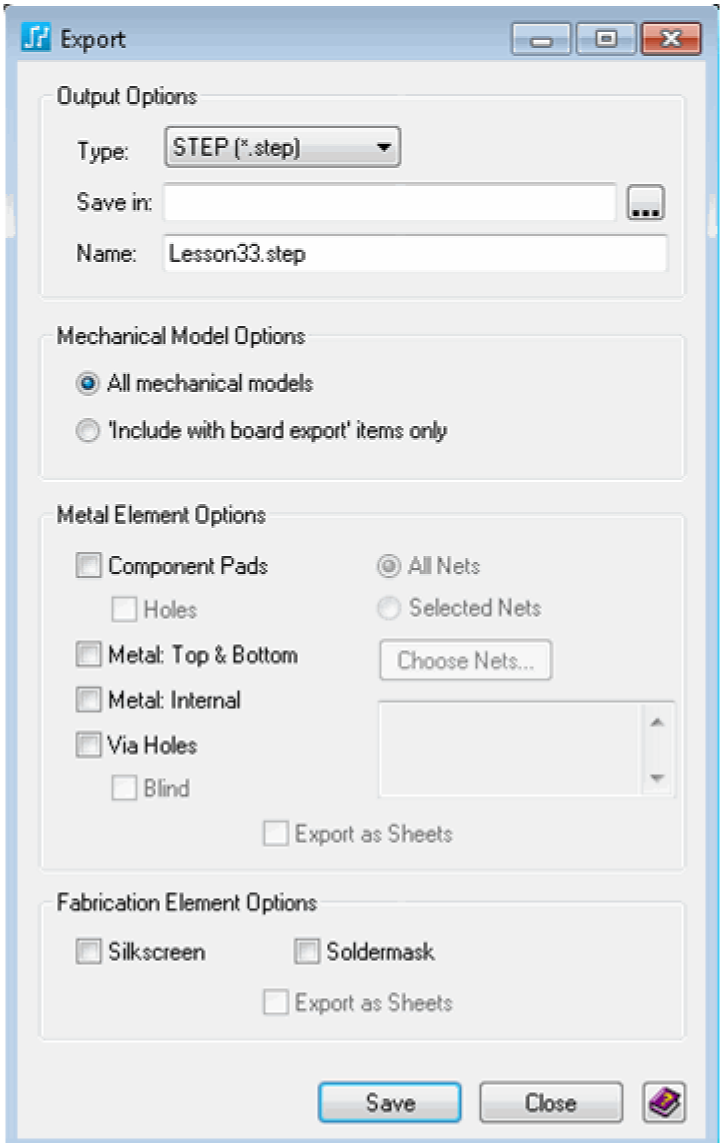
Objects

Field	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”.
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.

Export Dialog Box

To access: **SailWind 3D window > 3D General Toolbar > Export button**

Specify the content and file format to export from the 3D view.



Objects

Object	Description
Type	The following file types to output: STEP (*.step*), 3D PDF (*.pdf*), BMP (*.bmp*), GIF (*.gif*), JPG (*.jpg*), PNG (*.png*), SAT (*.sat*), STEP (*.step*), TIF (*.tif*).

Object	Description
	Note: The Options fields are available for SAT and STEP file types only.
Save in	The location to save the file. If there's an attempt to output files to a directory where some files of the same name already exist, those files and all others with the same root name will be deleted in order not to have files with potentially stale data.
Name	The filename to save. Separate files are created for the board and components, each copper layer, the Silkscreen and Soldermask. All the files have the same root name as entered in the Name field but with different suffixes appended, for example, _Silkscreen, _Soldermask, _CopperTop.
Mechanical Model Options	Choose whether to include all mechanical models that were imported into the 3D view, or to export only models for which the "Include with board export" check box is selected (in the model's Properties dialog box). These options are only available for SAT and STEP file types.
Metal Element Options	These items can be included with or omitted from the general items that are exported. Only available for SAT and STEP file types. • Component Pads — Controls the output of component pads only. If the Holes check box is checked, the holes in those pads will also be generated in the board STEP file. • Metal: Top & Bottom — All the other copper data on the outer board layers. • Metal: Internal — Data on any internal copper layers will be exported. This option needs to be selected if you want the pads on internal layers. Copper on internal layers is generally only consumed by tools like Simcenter Flotherm XT where they want to do a thermal analysis. • Via Holes — Outputs the via holes in the board STEP file with the option to include the blind via holes too. Holes take a lot of time to process. If there are many blind via holes you can save a lot of time by not exporting them. If you export Soldermask it will cover up the via holes unless you have a Soldermask pad on the Soldermask layer which is not usually the case for vias. • Export as sheets — If selected, exports copper data as an infinitely thin sheet without any thickness. Depending upon the use-case, this may be fine for the target system. If cleared, the copper data is output with the same thickness as seen in the 3D view. The advantage of using this option is that you can typically see a reduction in file size and a reduction in load time in the target system. If you simply want this data for visualization purposes then consider using this option • All Nets/Selected Nets — Of the object types selected to export (pads, planes, traces), select all nets to export all those objects or limit the objects by choosing nets in the Select Nets Dialog Box.
Fabrication Element Options	For STEP file types only. Select the check box of the fabrication element type (Silkscreen or Soldermask) that you want to export. Export as sheets — If selected, exports the silkscreen or soldermask data as an infinitely thin sheet without any thickness. Depending upon

Object	Description
	the use-case, this may be fine for the target system. If cleared, the silkscreen or soldermask data is output with the same thickness as seen in the 3D view. The advantage of using this option is that you can typically see a reduction in file size and a reduction in load time in the target system. If you simply want this data for visualization purposes then consider using this option

Related Topics

Exporting the 3D Image to a 3D PDF File

Exporting the 3D Image as a Mechanical Model

Exporting the 3D Image to a Graphics File Type

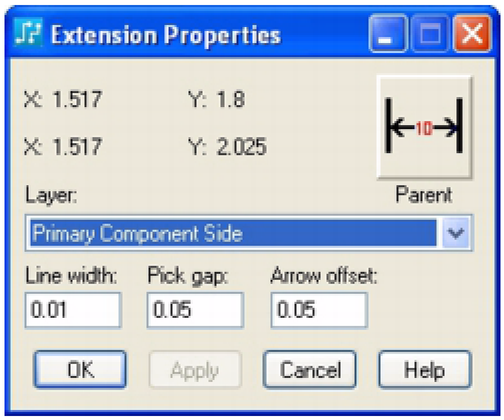
Extension Properties Dialog Box

To access: Select an extension > right-click > **Properties** popup menu item

The Extension Properties dialog box displays coordinate information for the selected extension line and provides several areas for modifications.

Description

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.



Objects

Field	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

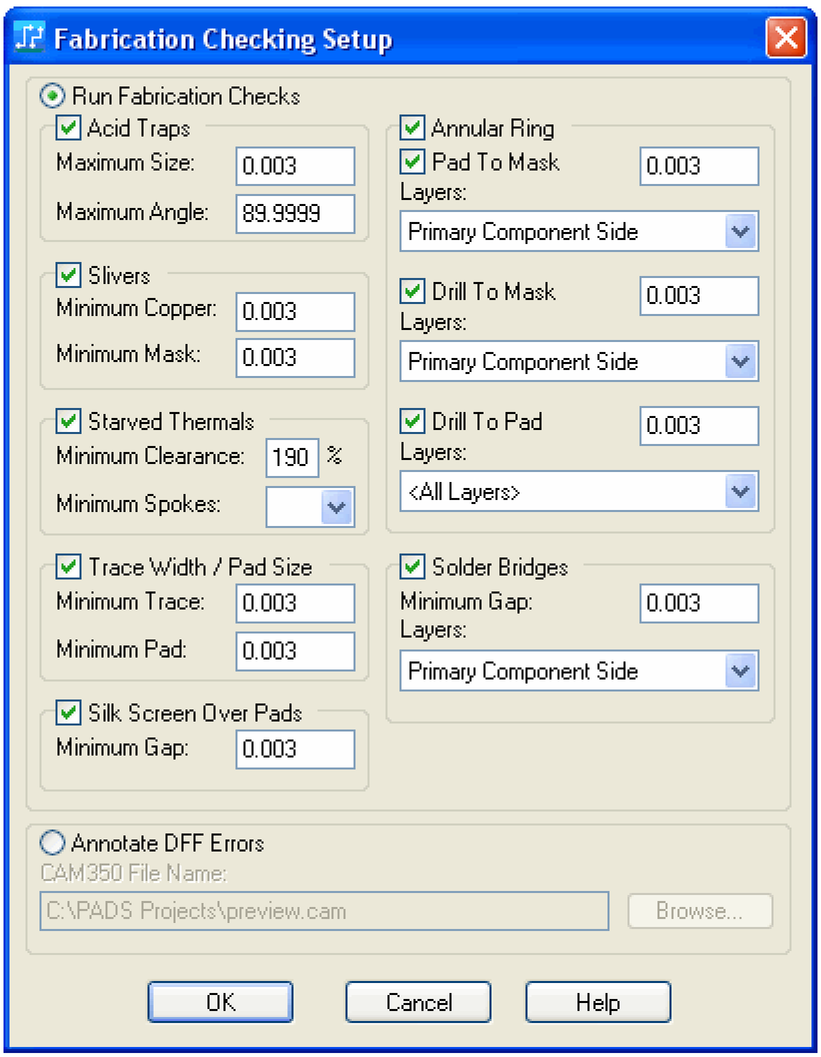
Dimensioning Process

Fabrication Checking Setup Dialog Box

To access: **Tools > Verify Design** menu item > Fabrication check > **Setup** button

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.

Restriction:
You need the CAM350 Link license option to use this.



Objects

Field	Description
Run Fabrication Checks	Runs the fabrication checks in SailWind Layout.

Field	Description
Acid Traps	Flags 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	Flags 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	Flags 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	Flags 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. • **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	Flags 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	Flags minimum annular rings on top and bottom layers, comparing electrical, drill, and mask layers.
Pad to Mask	Flags 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

Field	Description
	top solder mask layer or the bottom electrical layer against the bottom solder mask layer. • **Layers list** — Specifies the layer to use for checking. **Restriction:** You must select the Annular Ring check box to enable this fabrication check.
Drill to Mask	Flags 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. **Restriction:** You must select the Annular Ring check box to enable this fabrication check.
Drill to Pad	Flags minimum clearance distances between a drill and its associated pad. Checks the drill to pad value against the <pad size - drill size> values in your design. The offset and annular ring is checked against the specified clearance value. This check can be run on each electrical layer. • **Layers list** — Specifies the layer to use for checking. **Tip** You must select the Annular Ring check box to enable this fabrication check.
Solder Bridges	Flags 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.
Annotate DFF Errors	Loads DFF errors from a CAM350 file. Select this if you use CAM350 for checking your fabrication errors.
CAM350 File Name	Specifies the *.cam* path and file name of a file to back-annotate DFF errors into SailWind Layout for design verification. Click **Browse** to navigate to the location.

Related Topics

Fabrication Checking

Fanout Rules Dialog Box

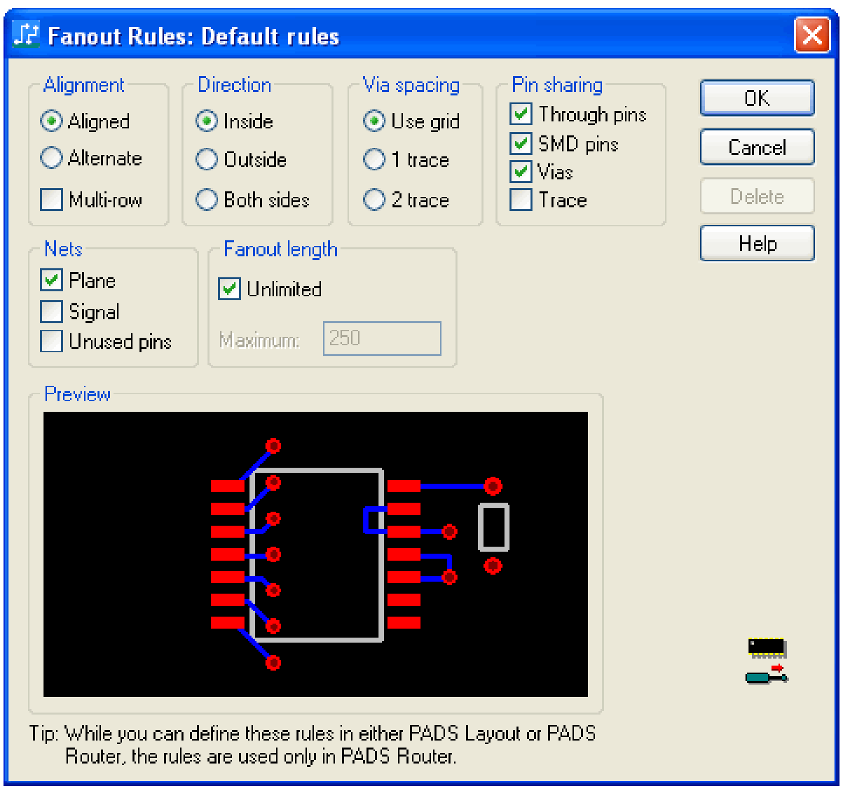
To access:

- **Setup > Design Rules** menu item > **Default** button > **Fanout** button

Use the Fanout Rules dialog box to specify fanout rules. Use fanouts to make routing easier and to ensure that connections are made. SailWind 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.

Tip
While you can define these rules in either SailWind Layout or SailWind Router, the rules are used only in SailWind Router.

Restriction:
In SailWind Layout, you can define fanout rules for only SOIC/QUAD type components. Use SailWind Router to also define fanout rules for BGA and staggered BGA fanout patterns.



Objects

Area	Description
Alignment	Specify the via placement: • **Aligned** — Aligns fanout vias on the SailWind 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.
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 SailWind Router to route one wire between adjacent vias. • **2 Trace** — Width of two traces. This sets a via grid that enables SailWind Router to route two wires between adjacent vias.
Pin Sharing	Specify all the connections on the same net that SailWind 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 on page 1860 pin if the cost is lower than the cost to use a via. If this is disabled, SailWind Router routes each SMD pad directly to a pin or via. • **Vias** — Connect to a shared via. If this is disabled, SailWind Router creates unique vias for each surface-mount pad. • **Trace** — Connect to a shared trace, which creates a T-Junction on page 1864.
Nets	Specify the types of nets where fanouts can be created: • **Plane** — Create fanouts for pins belonging to plane nets on page 1849. • **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:

Area	Description
	• **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

Find Dialog Box

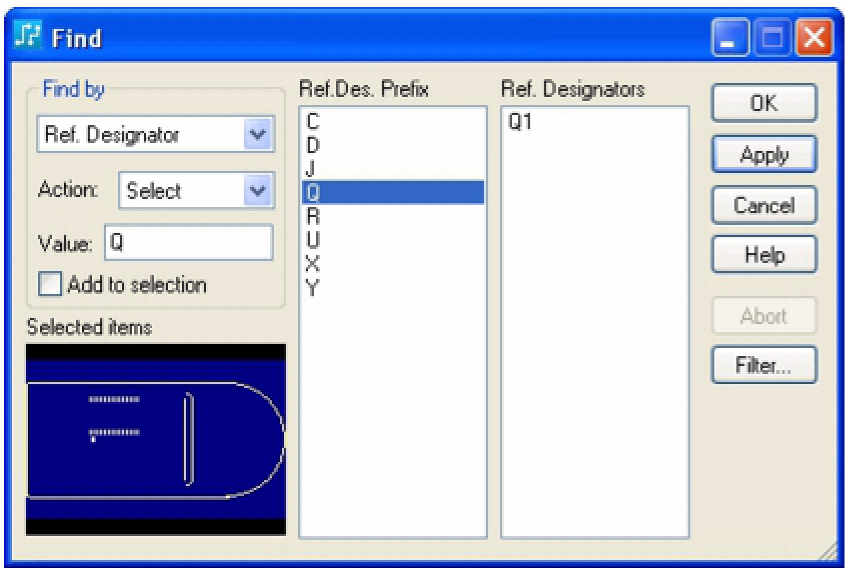
To access: **Edit > Find** menu item

Use Find to locate and select single or multiple objects by reference designator, part type, line width, or other attributes.

Description

Find works two ways, depending on your selection mode on page 115:

- **Select mode** — Find ignores the Selection Filter on page 114 settings and selects whatever you ask it to.
- **Verb mode** — Find only looks for items that are logical for verb mode.



Objects

Field	Description
Find by object type list	Specifies how you want to search. You can find by: • Reference designator • Nets • Line width • Part type • Pin pair • Via type

Field	Description
	• Decals • Net class • Groups • Reuse type (physical design reuse) • Clusters • Unions • Hatch outline • Isolated hatch outline • Pad size • Thermal attributes (custom pad stack thermals) • Jumper via • Keepouts • Attribute • 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 on page 155.
Add to Selection	Select the check box to increase the selection of objects in the design by any items you select with the Find dialog. Objects are not deselected when you select another object. This feature is essential if you want to select a variety of object types using the Find dialog.

Field	Description
	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 in the Selections color on page 1318.
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. If you select only the D prefix, all components with the D prefix will selected. 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 on page 114. Unless you are in a

Related Topics

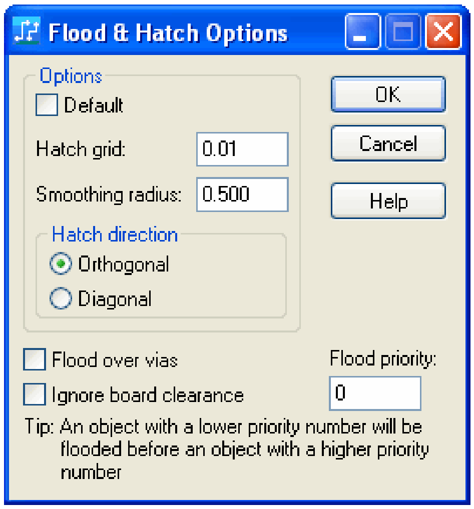
Object Finding

Use of the Find Dialog Box During Placement

Flood and Hatch Options Dialog Box

To access: Select edge of pour or plane area > right-click > **Select Shape** popup menu item > right-click > **Properties** popup menu item > **Options** button

Use the Flood and Hatch Options dialog box to establish unique display settings for copper planes.



Objects

Field	Description
Options area	**Default check box** — Specifies to use the default settings as found on the Grids on page 1537 and "Hatch and Flood" on page 1511 subcategories of the Options dialog box. Clear this check box to access the hatch settings. **Hatch grid** — Specifies the distance between hatch lines. **Smoothing radius** — Controls the radius of copper plane corners. <corner radius> = <shape outline width> * <smoothing radius> The smoothing radius parameter can be set between 0 and 5. **Hatch direction** — Specifies the hatch line orientation.
Flood over vias	Specifies to flood over any vias within the copper plane that are part of the same net. **Tip** • When this setting is changed and applied to the selected object, the changed setting becomes the default for all newly-created copper planes. 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.

Field	Description
Ignore board clearance	Specifies to ignore the board-to-copper clearance setting and have the copper plane flood outside the board area.
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 planes on different layers are processed independently. See also: “Setting Flooding Order of Overlapping Copper Planes” on page 788 and “Copper Plane Flood Priorities” on page 783. **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: “Embedded Copper Planes” on page 685.

Related Topics

Creating a Copper Plane Manually

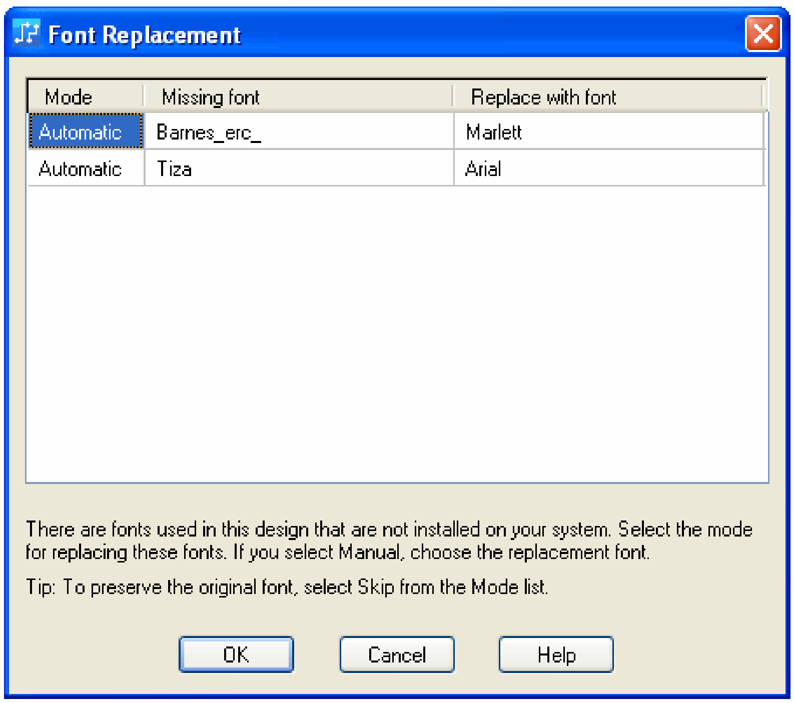
Flood Over Vias in a Copper Plane

Font Replacement Dialog Box

To access: When you open a design created with fonts that are not installed on your system, the Font Replacement dialog box opens automatically.

Use the Font Replacement dialog box to manage how missing fonts are replaced in your design.

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.



Objects

Field	Description
Mode	Specifies the mode to replace this font. • **Automatic** — Specifies to replace the font automatically with the one selected by SailWind 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.

Field	Description
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 SailWind 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

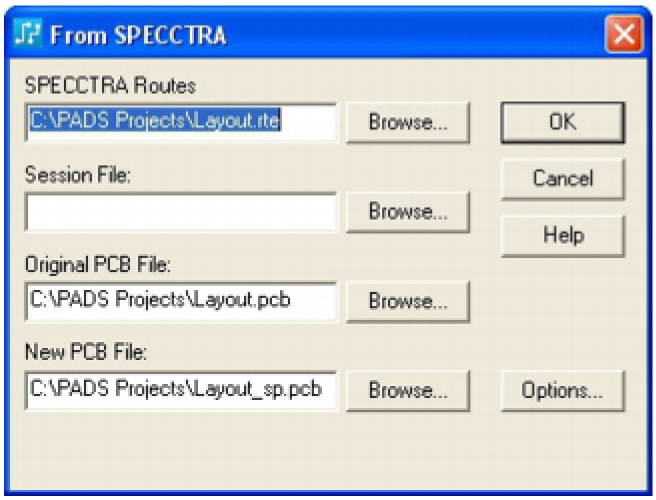
Replacing Missing Fonts

Finding Fonts

From SPECCTRA Dialog Box

To access: Use Windows Explorer to navigate to your *C:\<install_folder>\SailWind<version>\Programs* directory, and double-click *pads2sp.exe* and then click the **From SPECCTRA** button

Use the From SPECCTRA dialog box to translate design data modified by SPECCTRA back into a *.pcb* design file.



Objects

Field	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. 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. Click **Browse** to locate the file.
Original PCB File	Specifies the original (source) *.pcb* file. Click **Browse** to locate the file.
New PCB File	Specifies the file to be created from the SPECCTRA file. Click **Browse** to locate the file.
Options button	Opens the Options dialog box on page 1727.

Related Topics

Translating Design Data from SPECCTRA to SailWind Layout

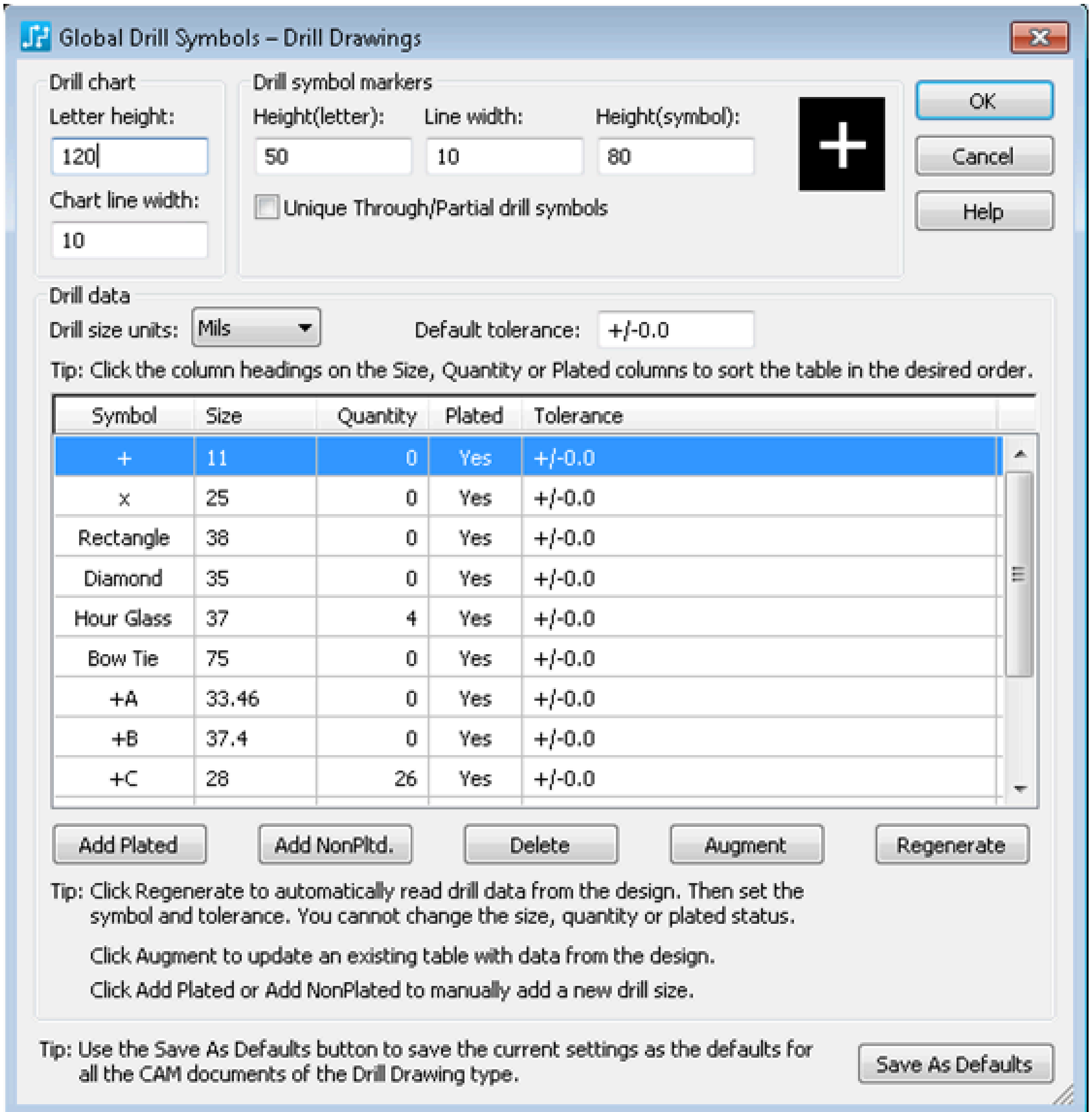
Global Drill Symbols Dialog Box

To access:

- Click the **File > CAM** menu item > **Drill Symbols** button
- Click the **File > CAM** menu item > click the **Add** button > select “Drill Drawing” from the Document Type list > select a layer > click the **Options** button > click the **Drill Symbols** button > click the **Global Drill Symbols** button

Use the Global Drill Drawing Options dialog box to set drill drawing legend and marker parameters, such as which symbol to use for a drill size and how tall to make text displayed on the drill drawing.

Note:
Use the Global Drill Symbols dialog box to set up global symbol options. You can set which symbols to include in a particular drill drawing through the “Drill Symbols dialog box” on page 1334.



Objects

Field	Description
Drill Chart area	
Letter Height	Specifies the height of letters in design units.
Chart Line Width	Specifies the width of the chart lines in design units.
Drill Symbol Markers area	

Field	Description
Drill Symbol Markers	• **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. • **Unique Through/Partial drill symbols** — Enables or disables the display of the Through/Partial column in the table.
Drill Data area	
Drill size units	Specifies the units to use for the drill size.
Default tolerance	Specifies the tolerance to use in the Tolerance column of the table.
Chart Contents — the chart only displays the drill data of items that are added to the drill drawing in the Select Items Dialog Box. For example, if no pads or vias are added from the electrical layers, the chart does not display drill information for those locations.	
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 are assigned: Rectangle +, Rectangle X, Diamond +, Diamond X, Circle +, and Circle X () • 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 description in this table 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.
Through/Partial column	Indicates whether the drill is for a partial or through-type hole. **Tip**

Field	Description
	• This column displays only if you first select the Unique Through/Partial column check box in the “Drill symbol markers” area. • When this column displays, if both through and partial drills of a given drill size exist, they are counted separately and appear in separate rows. When this column does not display, 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	Opens the Enter Drill Size dialog box so you can add 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	Opens the Enter Drill Size dialog box so you can 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 select multiple entries for removal, press the Ctrl or Shift key and click to select them.
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 pre-populated by a set of drill sizes that may or may not be in use in the current design. See the Save As Defaults button description in this table for more information. Using Augment retains 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. • 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.
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 Save As Defaults button description in this table for more information. Clicking **Regenerate** clears and creates a list of only the hole/drill sizes used in the design.

Field	Description
	Tip The drill table reflects entries for slotted holes if they exist in the design database after you click **Regenerate**. • 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.
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:\<install_folder>\<version>\Settings* **Restriction:** The drill chart location is not saved when you click **Save As Defaults**.

Related Topics

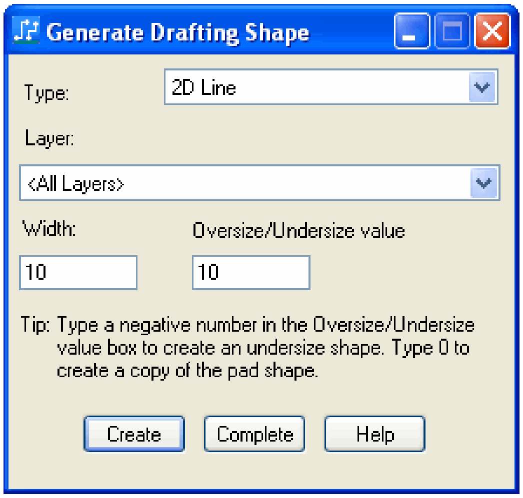
Drill Symbols Dialog Box

Creating CAM Outputs to Manufacture Your PCB

Generate Drafting Shape Dialog Box

To access: **Tools > PCB Decal Editor** menu item > select terminal > right-click and click **Generate Drafting Shape** popup menu item

You can use the outline of a terminal as the basis to create new drafting shapes.



Objects

Field	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.

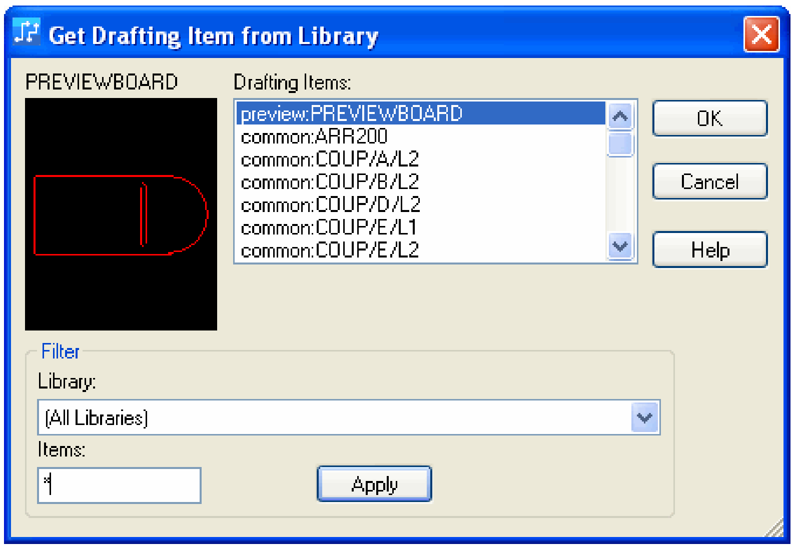
Related Topics

Generating Drafting Shapes from Terminals

Get Drafting Item from Library Dialog Box

To access: **Drafting Toolbar** button > **From Library** button

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.



Objects

Field	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 on page 155. 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

To access:

- **ECO Toolbar > Add Component** button
- **ECO Toolbar > Change Component** button

Use the Get Part Type from Library dialog box to Retrieve parts from a library when you add or update parts.

Description

The dialog box is slightly different depending on if you are adding or updating parts.

Figure 179. Add Component View

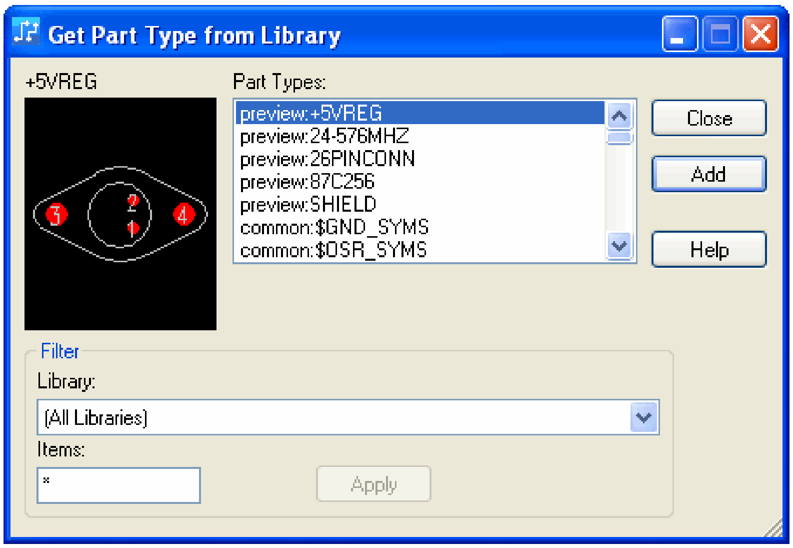
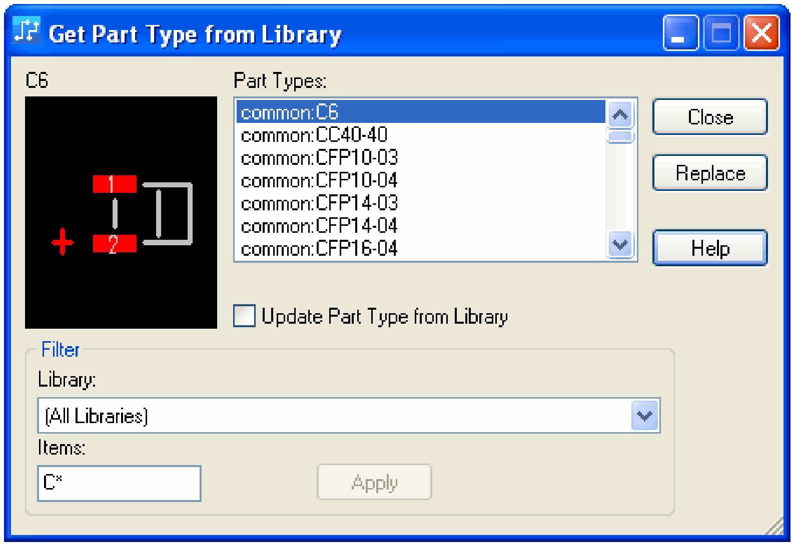


Figure 180. Change Component View



Objects

Field	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 on page 155. An asterisk (*) displays all parts in the list.
Apply button	Searches the library for the specified item.

Related Topics

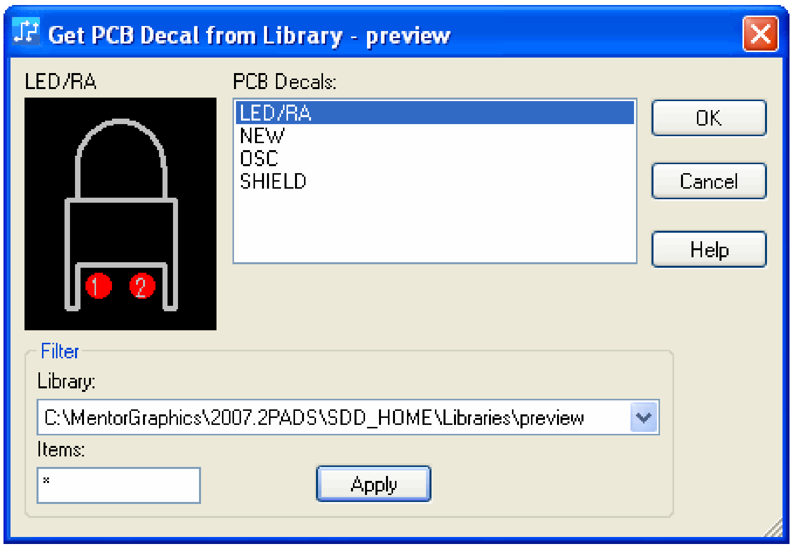
Adding a Component in ECO Mode

Changing a Component in ECO Mode

Get PCB Decal From Library Dialog Box

To access: **Tools > PCB Decal Editor** menu item > **Open** button

Use the Get PCB Decal from Library dialog box to open the decal you want to edit.



Objects

Field	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. You can use wildcards in this box.
Apply	Executes the filter arguments.

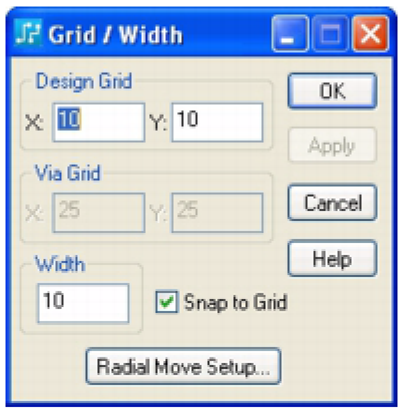
Related Topics

Editing a Library Decal

Grid/Width Dialog Box

To access: In the PCB Decal Editor, **Setup > Grid and Width** menu item. The dialog box can also be left open when you exit the PCB Decal Editor in order to use it within the Design Editor. It can no longer be opened from the Status Bar.

Use the Grid/Width dialog box to quickly change grid and width settings.



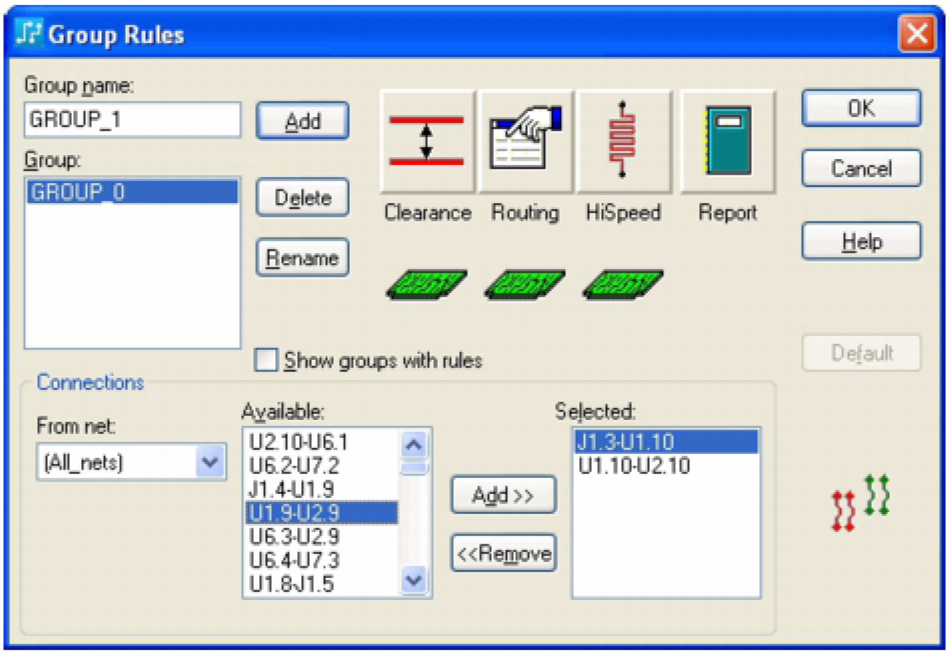
Objects

Field	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

To access: **Setup > Design Rules** menu item > **Group** button

Use the Group Rules dialog box to add and manage groups of pin pairs, and to define design rules that apply to them.



Objects

Field	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.
HiSpeed	Opens the HiSpeed Rules Dialog Box.

Field	Description
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

Chapter 50
GUI Reference Elements H Through J

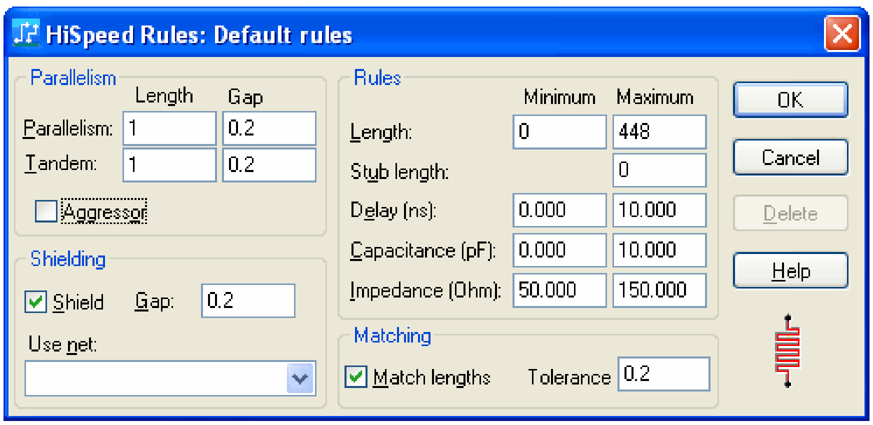
Read the sections that follow to learn more about dialog box elements in SailWind Layout.

HiSpeed Rules Dialog Box

To access: **Setup > Design Rules** menu item > choose a hierarchy level > **High Speed** button

Use this dialog box to set up high-speed rules.

Note:
The Electrodynamic Checking (EDC) licensed option is required to check high-speed rules.



Objects

Area	Description
Parallelism	• **Parallelism** — Specifies Length and Gap values for parallelism on page 1845. 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 on page 1864. 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 on page 1808.
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. **Restriction:** • Options in the Shielding area are unavailable if the design has no plane layers. • SailWind 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.

Area	Description
Rules area	
Length	Specifies the minimum and maximum length allowed. Includes half the Discrete length value of each connected pin of components that have a Discrete length assigned.
Stub length	Specifies the maximum distance from the T-Junction on page 1864 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 Parameter Setup 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 on page 1627 and Net Properties on page 1487 dialog boxes. **Restriction:** • This rule is only verified when you run Tools > Verify Design on page 1775 after you set up a High Speed check on page 1363 to check Delay. It is not verified by online DRC on page 1843.
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 Parameter Setup 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 on page 1487 dialog box. **Restriction:** • Capacitance cannot be set for Pin Pairs. • This rule is only verified when you run Tools > Verify Design on page 1775 after you set up a High Speed check on page 1363 to check Capacitance. It is not verified by online DRC on page 1843.
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 Parameter Setup 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 on page 1627 and Net Properties on page 1487 dialog boxes. **Restriction:** This rule is only verified when you run Tools > Verify Design on page 1775 after you set up a High Speed check on page 1363 to check Impedance. It is not verified by online DRC on page 1843.
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.

Area	Description
	• **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:** SailWind 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

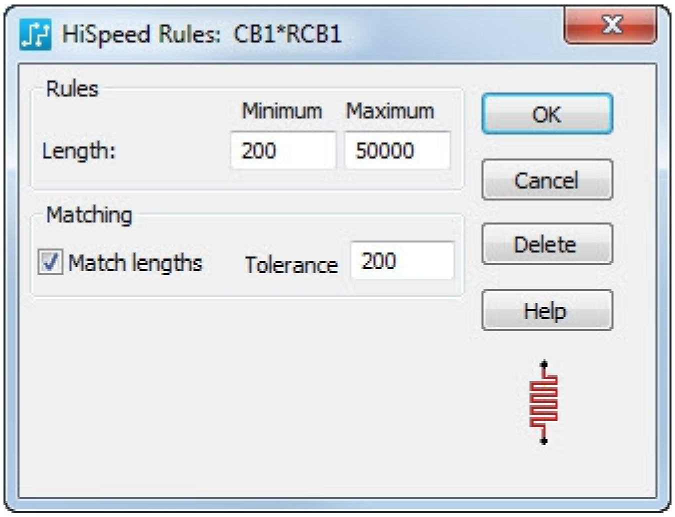
Design Rule Hierarchy

HiSpeed Rules Dialog Box, Electrical Nets

To access: Electrical Net Rules dialog box > **HiSpeed** button

Assign min/max length rules and create matched length groups for electrical nets.

Tip
The Electrodynamic Checking (EDC) licensed option is required to check high-speed rules.



Objects

Field	Description
Length	Set the minimum and maximum length for the electrical nets you selected in the "Electrical Net Rules Dialog Box" on page 1360. Includes half the Discrete length value of each connected pin of components that have a Discrete length assigned.
Match lengths	Select this check box to make the electrical nets you selected in the "Electrical Net Rules Dialog Box" on page 1360 members of a matched length group. Clear the check box to remove the selected electrical nets from an existing matched length group.
Tolerance	Set the tolerance value for length-matching.

Related Topics

Design Rule Hierarchy

HYP Export Dialog Box

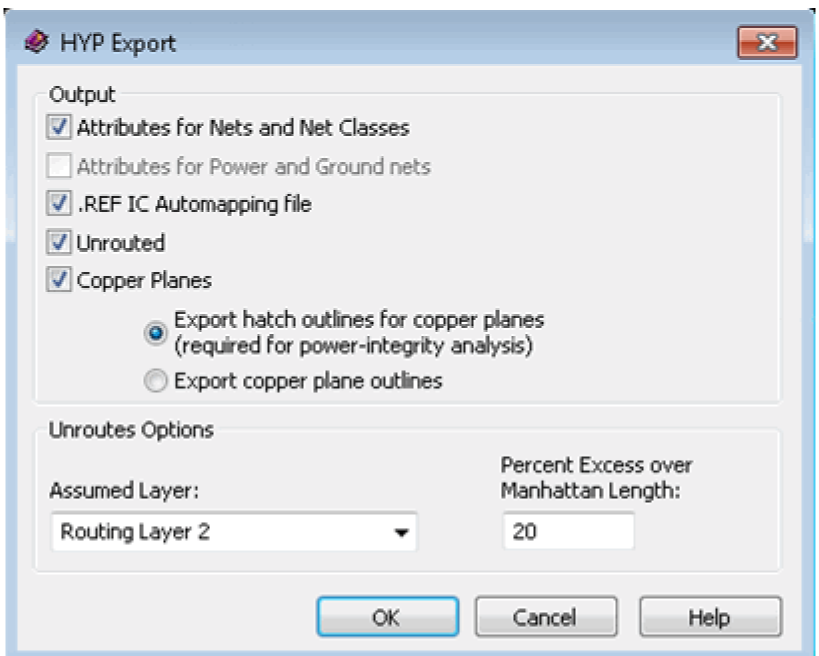
To access: Choose the **File > Export** menu item and select "Hyp Files (*.hyp)" from the "Save as type" dropdown list; then click **Save**

Use the HYP Export dialog box to export the design in the form of a HYP file.

Description

For information about the format of the HYP file and creating a "BoardSim-friendly" design, see the PADS HyperLynx online Help. SailWind Layout passes the Value, Tolerance, Voltage, PADS 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.

Note:
Only PADS 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.



Objects

Field	Description
Output area	
Attributes for Nets and Net Classes	Exports attributes for nets and net classes.

Field	Description
Attributes for Power and Ground nets	Exports attributes for power and ground nets.
.REF IC Automapping file	Creates a PADS 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	Exports unrouted nets.
Copper Planes	Exports copper plane information using the following options: • **Export hatch outlines for copper planes** — Exports 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 copper plane outlines** — Exports copper plane outlines only (no hatching).
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

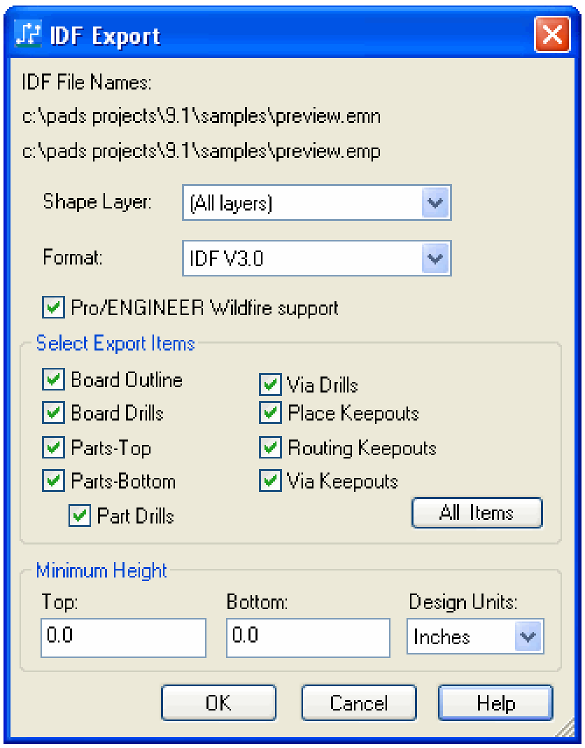
Exporting to PADS HyperLynx BoardSim

IDF Export Dialog Box

To access: **File > Export** menu item > Select IDF File > **Save**

Use the IDF Export dialog box to exchange design data between SailWind 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 on page 303 information, drilled hole on page 302 information, or part outline on page 302 information for a more accurate IDF export.



Objects

Field	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 on page 302 for the decals in your design that you want to send to the mechanical design system.
Format	Specifies the IDF version to use.

<table>
<tr><th>Field</th><th>Description</th></tr>
<tr><td>Pro/ENGINEER Wildfire support</td><td>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.</td></tr>
<tr><td>Select Export Items area</td><td>Use these check boxes to select the items you want to export to the layer in the Shape Layer list.

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. 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. 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 category Text and Lines subcategory on page 1553.
• 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. This option is unavailable for IDF 2.0.</td></tr>
<tr><td>Top/Bottom</td><td>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.</td></tr>
<tr><td>Design Units</td><td>Specifies the design units for this file.</td></tr>
</table>

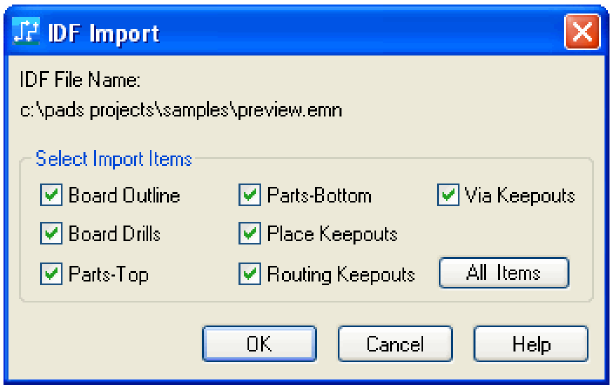
Related Topics

Exporting IDF Files

IDF Import Dialog Box

To access: **File > Import** > Select IDF File > **Open**

Use the IDF Import dialog box to import board outlines, keepouts, components, and holes from a mechanical design system. SailWind Layout cannot import the information in the *.emp* library file.



Objects

Field	Description
IDF File Name	The name of the file you are importing.
Select Import Items area	Use these check boxes to select the items you want to import. 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. 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. 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:

Field	Description
	• Board-level placement keepouts • Top and bottom component height restrictions defined for the whole board. Find this setting in the Options dialog box > Text and Lines subcategory on page 1553 • **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. 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. This option is unavailable for IDF 2.0.

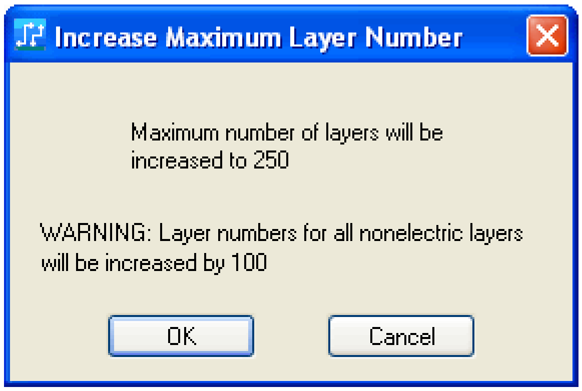
Related Topics

Importing IDF Files

Increase Maximum Layer Number Dialog Box

To access: **Setup > Layer Definition** menu item > **Max Layers** button

Use the Increase Maximum Layer Number dialog box to increase the number of available electrical and documentation layers from 30 to 250.



Objects

Field	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

Intelligent Layout Dialog Box

To access: **AI > Intelligent Layout** menu item

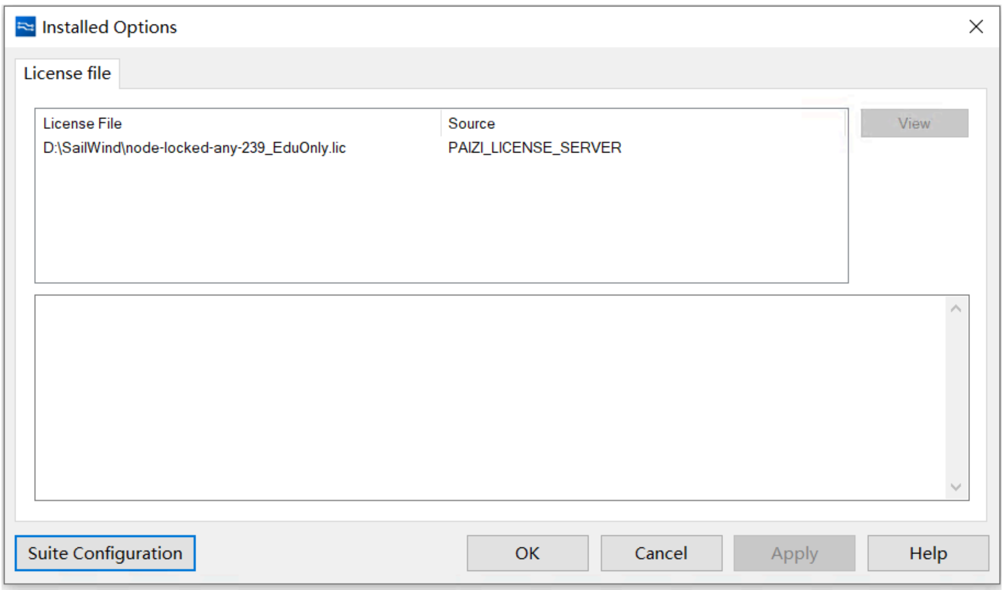
Objects

Field	Description
Table area: Lists the intelligent identified component groups along with their layout state and rule set correspondingly. **Note:** Rule set is not currently supported.	
Rule setup button	Opens the Rule Setup Dialog Box.
Module Analysis button	Initiates the module analysis.
Module Layout button	Initiates the layout process.
Cluster Create button	Creates clusters from the component groups.
Cluster Layout button	Opens the Cluster Placement Dialog Box.

Installed Options Dialog Box, License File Tab

To access: **Help > Installed Options** menu item > **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.



Objects

Field	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 SailWind Suite Configuration Dialog Box.

Field	Description
	Restriction: Available only with floating/server-based licenses, a mix of different SailWind Suites, or a mix of unbundled licenses and suites.

Related Topics

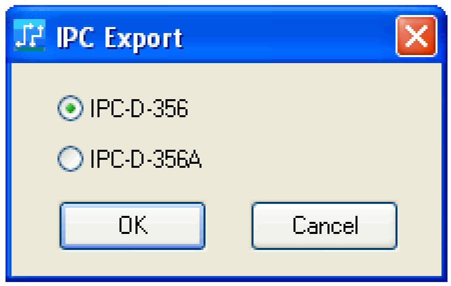
Viewing a Node Locked License File

Viewing the Floating License Status

IPC Export Dialog Box

To access: **File > Export** > Select IPC356 Files > **Save**

Use this dialog box to choose between IPC-D-356 netlist formats.



Objects

Field	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 Topics

The IPC-D-356 Netlist

Exporting an IPC-D-356 Netlist

JEDEC Array Pinning Dialog Box

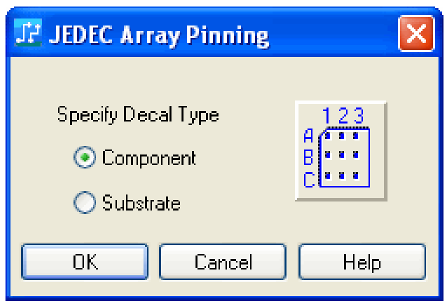
To access: **Tools > PCB Decal Editor** menu item > **Tools > Assign JEDEC Array Pinning**

Use the JEDEC Array Pinning dialog box to assign an alphanumeric string to each pin in an array following the JEDEC standard.

Description

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.



Objects

Field	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

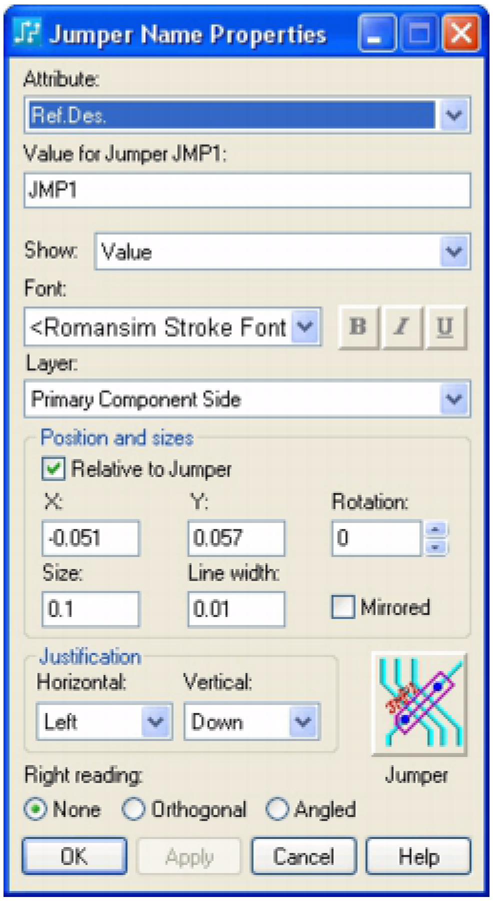
Assigning JEDEC Pinning

Jumper Name Properties Dialog Box

To access:

- Select a jumper name > right-click > **Properties**
- Select a jumper > right-click > **Properties** > **Label** button

Use the Jumper Name Properties dialog box to modify the jumper name and its attributes.



Objects

Field	Description
Attribute	The attributes available to you. If you are creating labels for jumpers, Reference Designator is the only available attribute.

Field	Description
	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.
Value for	The value of the selected attribute. **Tip** • 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 SailWind 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 on page 1862, 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. **Tip** • 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.
Size	Specifies the size of the font. Size (pts): This is font size in points and appears for system fonts

Field	Description
	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. **Tip** • 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

Jumpers

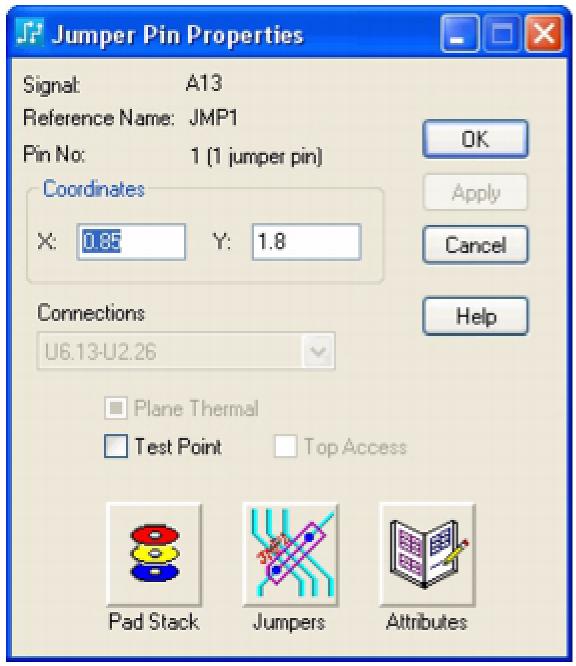
Jumper Pin Properties Dialog Box

Jumper Properties Dialog Box

Jumper Pin Properties Dialog Box

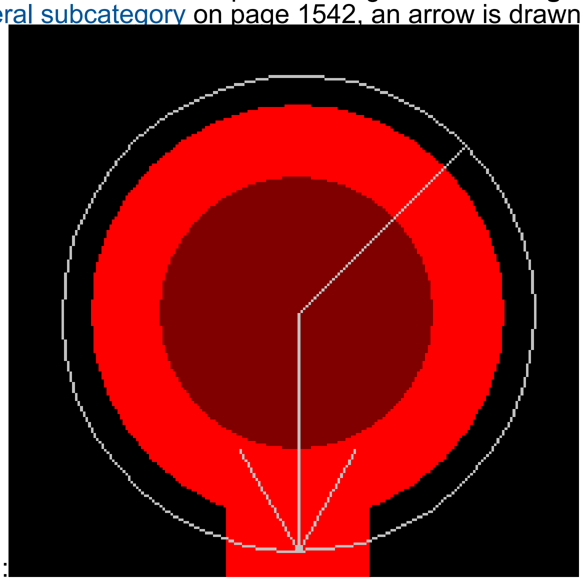
To access: Select a jumper pin > right-click > **Properties** popup menu item

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.



Objects

Field	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 on page 1866. The status of a thermal is set individually by pin and via, and the thermal indicators will appear on plane layers only. Click to

<table>
<tr><th>Field</th><th>Description</th></tr>
<tr><td></td><td>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.</td></tr>
<tr><td>Test Point</td><td>Makes the via or pin a test point. This is a three-state check box on page 1866.
See also: “Performing a Test Point Audit”.
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 in the Options dialog box > Routing category >General subcategory on page 1542, an arrow is drawn on it in the design: </td></tr>
<tr><td>Top Access</td><td>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).</td></tr>
<tr><td>Pad Stack button</td><td>Opens the jumper pin in the Jumper Parameters Properties Dialog Box on page 1427. You can change the pad stacks for individual pins in the jumpers.
See also: “Editing a Pad Stack in the PCB Decal Editor”.</td></tr>
</table>

Field	Description
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

Jumpers

Jumper Name Properties Dialog Box

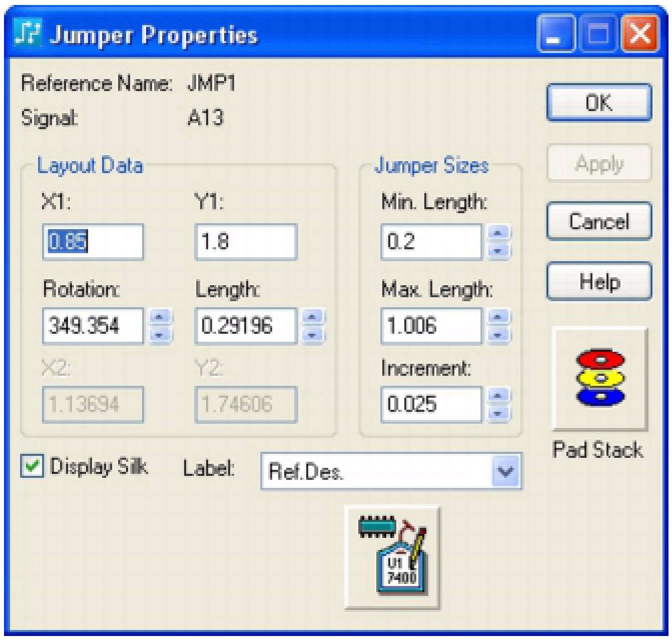
Jumper Properties Dialog Box

Jumper Properties Dialog Box

To access: Select a jumper > right-click > **Properties**

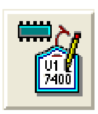
Use the Jumper Properties dialog box to modify jumper location, label, and size.

Figure 181. Jumper Properties Dialog Box



Objects

Field	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.

Field	Description
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.
Display Silk	Enables the display of a silkscreen outline for the jumper. **Tip** • 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 on page 1427. 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

Jumpers

Jumper Name Properties Dialog Box

Jumper Pin Properties Dialog Box

Jumpers Dialog Box

To access: **Setup > Jumpers** menu item (Jumpers Dialog Box)

- Select a jumper > right-click > **Properties** popup menu item > **Pad Stack** button (Jumper Parameters Properties 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.

Description

The Jumpers dialog box controls change depending on what you have selected for the Pad Style. The three major differences are shown in the figures below:

Figure 182. Jumpers Dialog Box - Pad

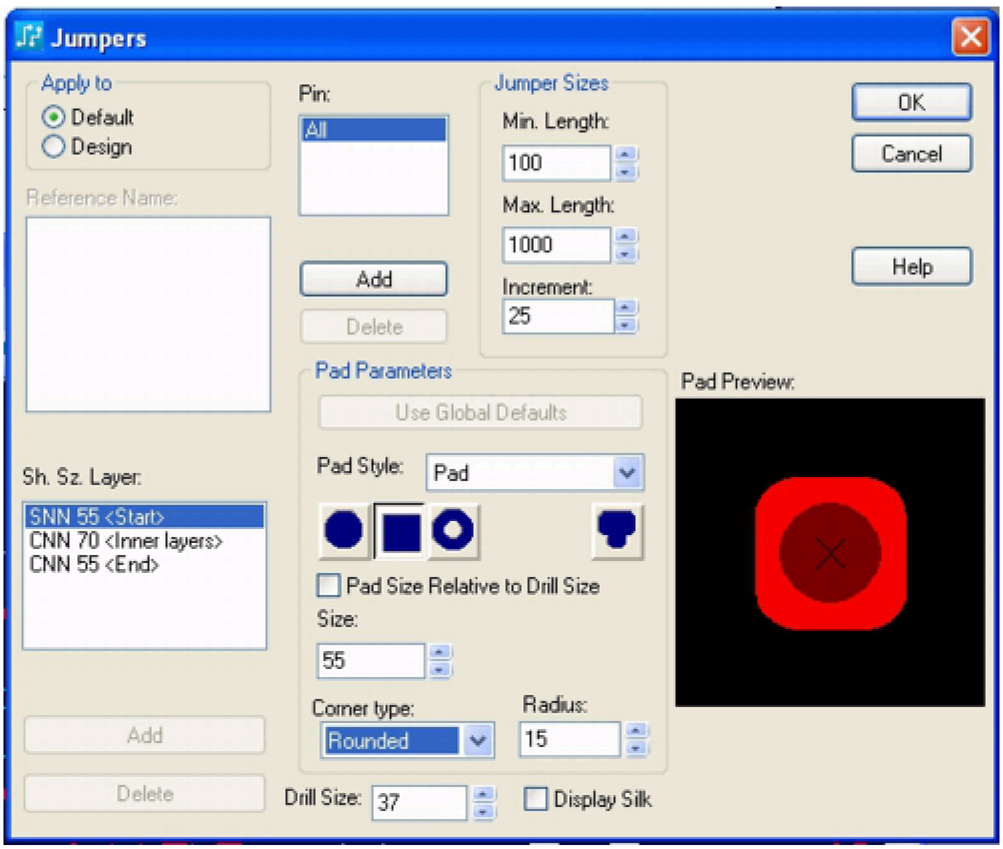


Figure 183. Jumpers Dialog Box - Thermal Settings

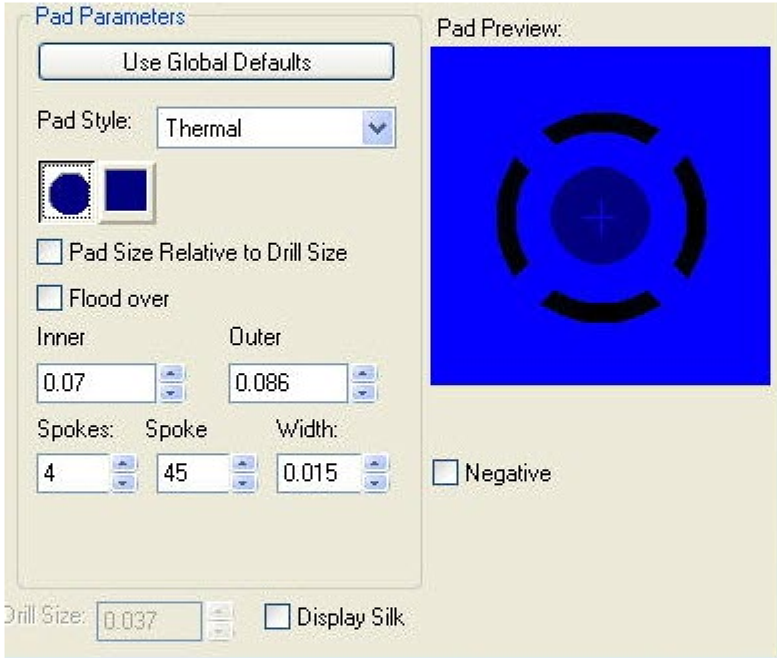
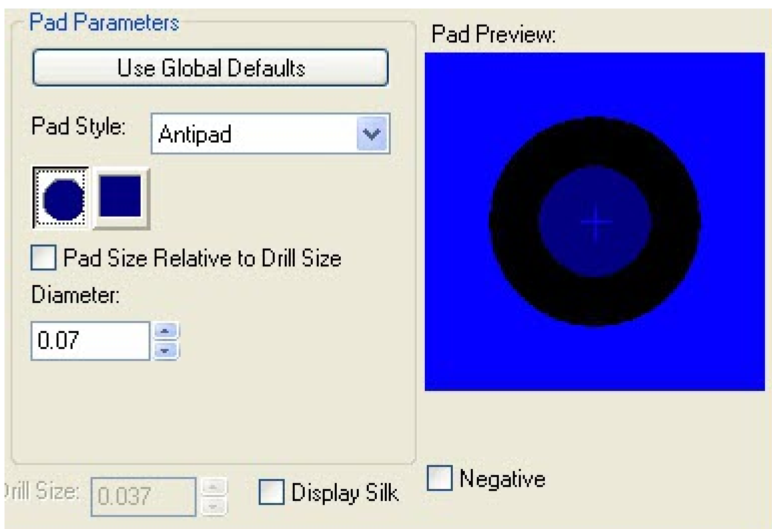


Figure 184. Jumpers Dialog Box - Antipad Settings



Objects

Table 188. Jumpers Dialog Box Fields

Field	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.

Table 188. Jumpers Dialog Box Fields (continued)

Field	Description
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.
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 Options dialog box > **Copper Planes** category > Thermals subcategory on page 1514. **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" on page 797.
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" on page 797. **Tip** • 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

Table 188. Jumpers Dialog Box Fields (continued)

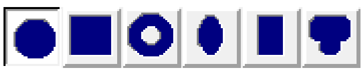
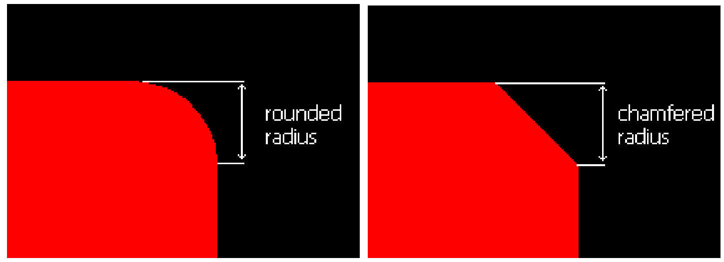
Field	Description
	antipads, diameters are initially set to follow the current default pad to copper design rule. If you select Use Design Rules for Thermals and Antipads in the Options dialog box **Copper Planes** category > Thermals subcategory on page 1514, the outer diameter is ignored and the clearance 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 “Creation of Custom Pad Shapes” 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.
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

Table 188. Jumpers Dialog Box Fields (continued)

Field	Description
	Figure 185. 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. **Tip** • 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

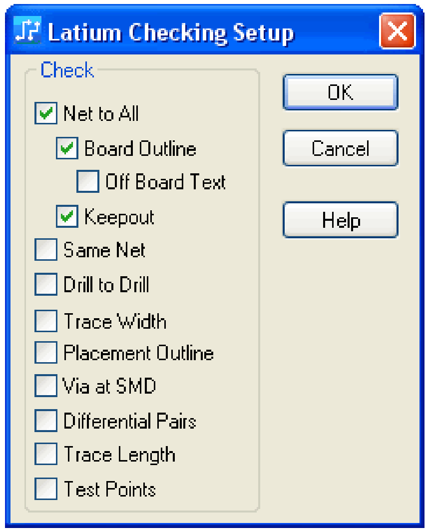
Chapter 51
GUI Reference Elements K Through O

Read the sections that follow to learn more about dialog box elements in SailWind Layout.

Latium Checking Setup Dialog Box

To access: **Tools > Verify Design** menu item > Latium Design Verification check > **Setup** button

Use the Latium Checking Setup dialog box to set the rules to check using SailWind Router.



Objects

Field	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.
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.

Field	Description
	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

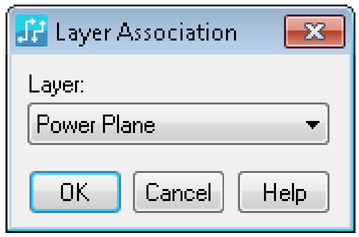
Setting Up Latium Checking

Cut Outs Absorbed by Copper

Layer Association Dialog Box

To access: Add/Edit CAM Document dialog box on page 1044 > select a document type from the Document Type list (not Custom)

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.



Objects

Field	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 Parameter Setup Dialog Box

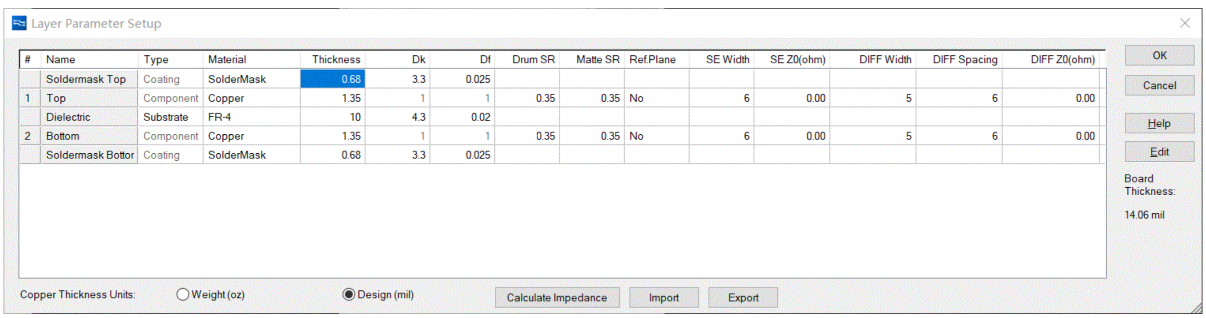
To access: **Setup > Layer Definition** menu item > **Advanced** button

Use the Layer Parameter Setup dialog box to define electrical layer and dielectric material layer thickness, dielectric constant and other information for impedance calculation. When you verify your design, the electrodynamic check also uses this information.

Tip
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.

Description

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.



Objects

Field	Description
#	Displays the electrical layer number, which corresponds to **Lev.** in the Layers Setup Dialog Box.
Name	Displays the name of the layer, which corresponds to that defined in the Layers Setup Dialog Box.
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.
Material	Specifies the material of each layer. You can choose one from the drop-down list.
Thickness	Specifies the thickness of the layer.

Field	Description
	The input range varies from the material type and current design units: • For dielectric material, its thickness must be within 5.08 mm/200 mil/0.2 inch. • For copper, its thickness must be within 0.25 mm/10 mil/0.2 inch. **Tip:** If no coating is required, set thickness to zero.
Dk	Specifies the dielectric constant value.
Df	Specifies the dielectric loss tangent. Range: 0—1
Drum SR	Specifies the roughness of the drum side. The input range varies from the current design units: • Mils: 0—1 • Metric: 0—0.02540 • Inches: 0—0.001
Matte SR	Specifies the roughness of the matte side. The input range is silmilar to that of the drum side.
Ref. Plane	Specifies the electrical layer as reference layer or not. **Note:** Plane layers specified in the Layers Setup Dialog Box are set to reference layers by default.
SE Width	Specifies the single-ended trace width. The input range varies from the current design units as follows and the same rule applies to that of DIFF Width as well as DIFF Space. • Mils: 0—100 • Metric: 0—2.54 • Inches: 0—0.001
SE Z0 (ohm)	Displays the single-ended impedance calculated by clicking **Calculate Impedance**.
DIFF Width	Specifies the trace width of differential pairs.
DIFF Space	Specifies the spacing between differential pairs.
DIFF Z0 (ohm)	Displays the differential impedance calculated by clicking **Calculate Impedance**.
Edit	Makes the selected cell available for editing. **Exception:** Cells that are grayed out or blank are unavailable for editing.

Field	Description
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.
Calculate Impedance	Clicks to start impedance calculation.
Import	Clicks to import a XML file.
Export	Clicks to export a XML file.

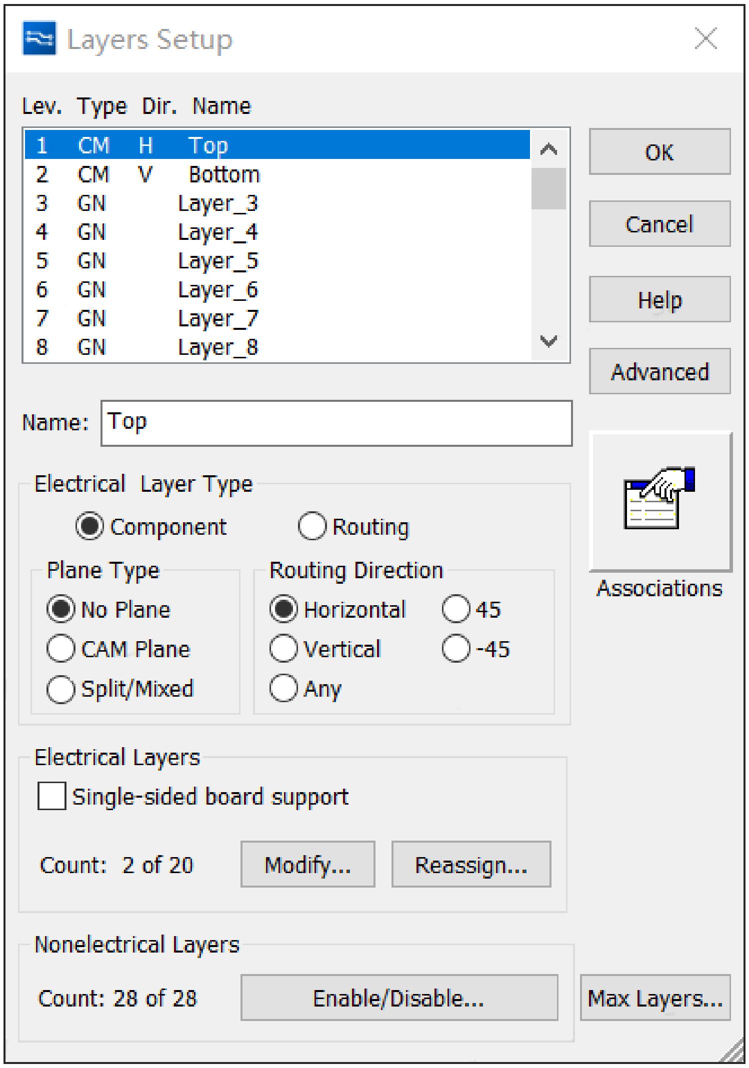
Related Topics

Setting Layer Parameters

Layers Setup Dialog Box

To access: **Setup > Layer Definition** menu item

Use the Layers Setup dialog box to define each layer in your design.



Objects

Field	Description
Layers list	• **Lev. 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".

Field	Description

	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
	• **Dir. column** — Identifies the specified routing direction, Horizontal (H), Vertical (V), Any (A), 45 (/), or -45 (\).
	• **Name column** — Specifies the name of the selected layer.

Name	The name for the layer which defines its function. You can type a unique name for each layer to identify it. This name appears in all layer listings.
Electrical Layer Type area	The Component and Routing options are available only when you have 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. 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. • **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.
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** — A signal routing layer allowing copper planes. If a No Plane layer is changed to a Split/Mixed or CAM Plane layer, nets belonging to the new plane are excluded from Electrical Nets. • **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 Planes cannot be created on CAM Plane layers. • **Split/Mixed Plane** — A dedicated copper plane layer allowing multiple copper planes and routing. Routes can be placed within or outside copper planes. Copper planes avoid traces within their outline by a clearance area defined in the design rules.

Field	Description
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.
Associations	Opens the Component Layer Associations Dialog Box. The Associations button becomes available when you select a component layer in the Layers list.
Assign Nets	Opens the Plane Layer Nets on page 1640 dialog box. The Assign Nets button becomes available when you select either a CAM plane or split/mixed 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. **Tip** • 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 on page 1480 dialog box. See also: “Modifying the Number of Electrical PCB Layers”.
Reassign	Opens the Reassign Electrical Layers on page 1655 dialog box. See also: “Reassigning Electrical Layers”.
Enable/Disable	Opens the Enable/Disable Layers on page 1365 dialog box. See also: “Hiding or Displaying Non-electrical Layers in Layer Lists”.
Max Layers	Opens the Increase Maximum Layer Number on page 1413 dialog box.

Field	Description
	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".
Advanced	Opens the Layer Parameter Setup dialog box.

Related Topics

Setting Up an Outer Layer

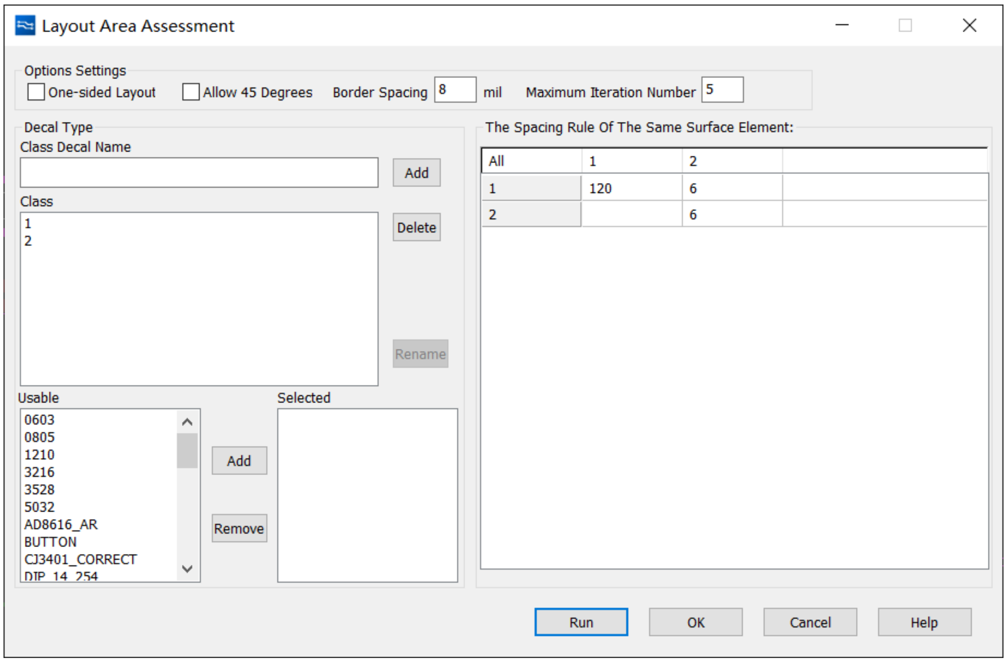
Setting Up an Inner Layer

Setting Up a Documentation Layer

Layer Modes

Layout Area Assessment Dialog Box

To access: **AI > Layout Area Assessment** menu item



Objects

Field	Description
Options Settings area	
One-sided Layout	Select this option to enable the single-sided layout.
Allow 45 Degrees	Select to allow rotating components by 45 degrees.
Border Spacing	Specify the minimum clearance from the component body to the board outline.
Max Iteration Number	Specify the number of algorithm iterations, with an upper limit of 100.
Decal Type area	
Class Decal Name	Specify the name of the class.
Class list	Lists all class names.
Add button	Add the class name to the Class list.
Delete button	Remove the selected class from theClass list.

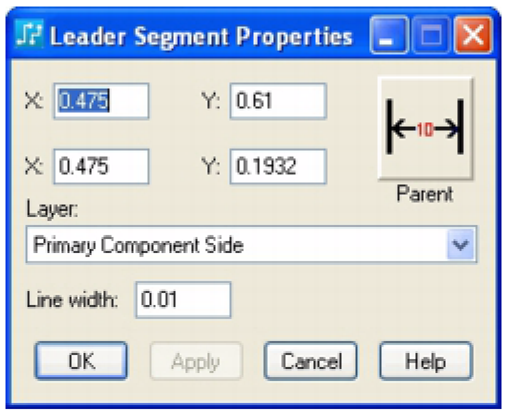
Field	Description
Rename button	Renames the class selected in the Class list with the text in the Class Decal Name box.
Usable list	Lists the PCB decals available for the class in selection.
Selected list	Lists the PCB decals of the class in selection.
Add button	Moves the selected items from the Usable list to the Selected list.
Remove button	Moves the selected items from the Selected list to the Usable list.
The Spacing Rule of the Same Surface Element area: Define the clearance rule between classes.	

Leader Segment Properties Dialog Box

To access: Select the first segment of a dimensioning leader segment, the segment with the arrow > right-click > **Properties** popup menu item

The Leader Segment Properties dialog box displays coordinate information for the selected dimensioning arrow and provides several areas for modifications.

Note:
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.



Objects

Field	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.

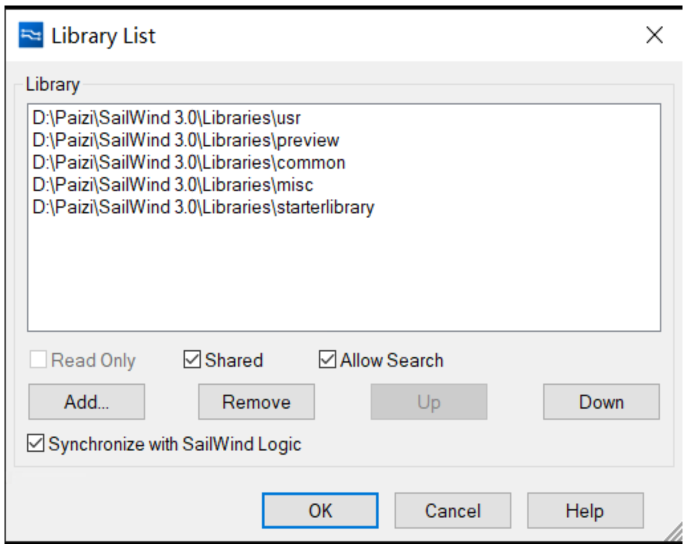
Related Topics

Dimensioning Process

Library List Dialog Box

To access: **File > Library** > **Manage Lib. List** button

Use the Library List dialog box to specify the libraries available to the design, library search order, and other search-related options.



Objects

Field	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.
Remove	Removes a library from the Library list.
Up/Down	Moves the order of the libraries in the Library list.
Synchronize with SailWind Logic	Specifies to push the library settings from SailWind Layout to SailWind Logic.

Related Topics

Library Availability and Search Options

Library Manager Dialog Box

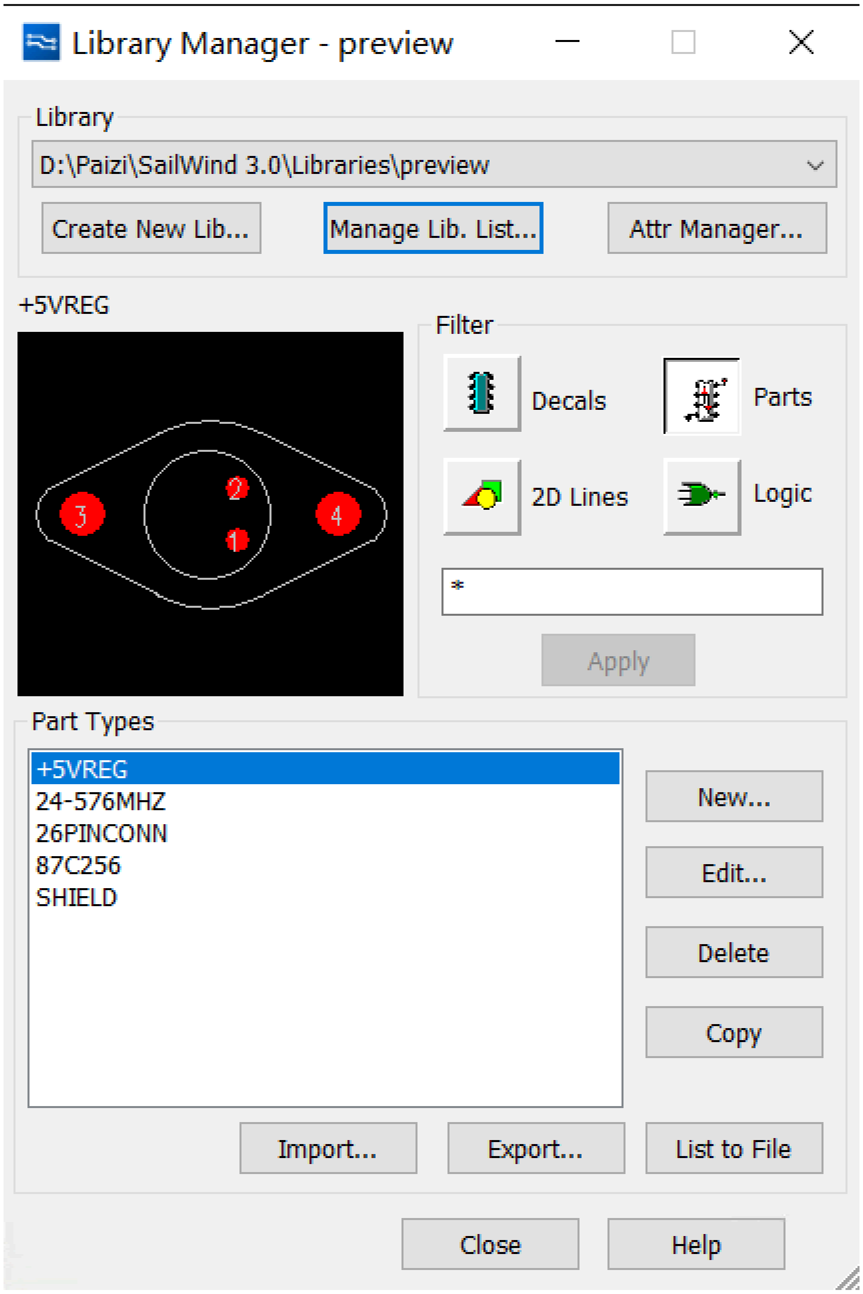
To access: **File > Library**

Use the Library Manager dialog box to create libraries, to display the contents of libraries, and to manage the contents of libraries.

Description

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.



Objects

Field	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.

Field	Description
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 on page 155 in the Filter box. **Tip** Add an asterisk (*) to the box to display all items.
Filter list	The results from your filter area selections.
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, on page 1588 **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. • **Logic** — Unavailable. Use SailWind 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, on page 1588 **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. • **Logic** — Unavailable. Use SailWind 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.

Field	Description
	Restriction: This button is unavailable when the Library is set to All Libraries. See also: "Importing Library Data".
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 New Library

Library Availability and Search Options

Deleting All Items in a Library

Library Attribute Management

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

To access: type BLT > press the Enter key

Use the Basic Log Test (BLT) to replay session playback media created by the Basic Media Wizard (BMW).

Tip
If nothing happens, close SailWind Layout and restart it.

Objects

Field	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.

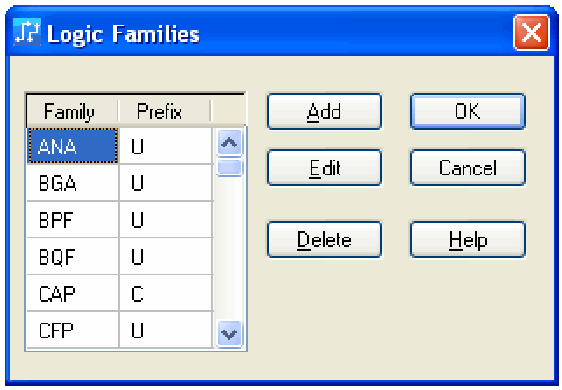
Related Topics

Replaying Session Playback Media with BLT

Logic Families Dialog Box

To access: **File > Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **General** tab > **Families** button

Use the Logic Families dialog box to add, delete, or edit logic family names or their reference designator prefix. Logic Families are abbreviated names for categories of parts. The abbreviated names are assigned a reference designator prefix. When creating a new part, you select a logic family to specify the reference designator prefix.



Objects

Field	Description
Family	The abbreviated name for the category of the part. The name is limited to 3 characters. For an explanation of the list of logic families that install with the software, see "Default Logic Families" below.
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. You can also double-click a cell to edit the contents.
Delete	Removes the selected row.

Default Logic Families

- The following is a list of the default logic family abbreviations along with their assigned reference designator prefixes and long description.
    - ANA (U) - Analog
    - BGA (U) - Ball Grid Array

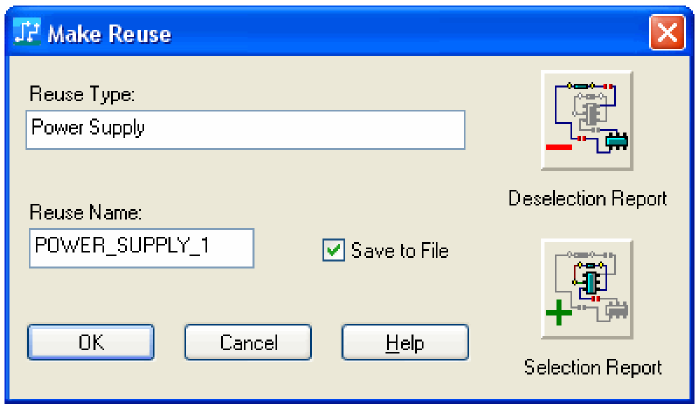
- BPF (U) - Band Pass Filter
- BQF (U) - PQFP (Bumpered Quad Flat Package)
- CAP (C) - Capacitor
- CFP (U) - Ceramic Flat Pack
- CLC (U) - CLCC (Ceramic Leadless Chip Carrier)
- CMO (U) - CMOS (Complimentary Metal Oxide Semiconducotor)
- CON (J) - Connector
- CQF (U) - CQFP (Ceramic Quad Flat Pack)
- DIO (D) - Diode
- DIP (U) - Dual Inline Package
- ECL (U) - Emitter Couple Logic
- EDG (P) - Edge Connector
- FUS (F) - Fuse
- HMO (U) - HMOS (High density, short channel Metal Oxide Semiconductor)
- HOL (X) - Hole
- IND (L) - Inductor
- LCC (U) - Leadless Chip Carrier
- MOS (U) - Metal Oxide Semiconductor
- OSC (Y) - Oscillator
- PFP (U) - Power Flat Package
- PGA (U) - Pin Grid Array
- PLC (U) - PLCC (Plastic Leaded Chip Carrier)
- POT (P) - Potentiometer

- PQF (U) - PQFP (Plastic Quad Flat Package)
- PSO (U) - PSOP (Plastic Small Outline Package)
- QFJ (U) - Quad Flat Pack with J-lead
- QFP (U) - Quad Flat Pack
- QSO (U) - QSOP (Quarter-Sized Outline Package)
- RES (R) - Resistor
- RLY (K) - Relay
- SCR (SC) - Thyristor
- SKT (U) - Socket
- SOI (U) - SOIC (Small Outline Integrated Circuit)
- SOJ (U) - Small Outline J-lead
- SOP (U) - Small Outline Package
- SSO (U) - SSOP (Shrink Small Outline Package)
- SWI (S) - Switch
- TQF (U) - TQFP (Thin Profile Quad Flat Pack)
- TRX (Q) - Transistor
- TSO (U) - TSOP (Thin Small Outline Plastic)
- TTL (U) - Transistor-Transistor Logic
- VSO (U) - Variable Speed Oscillator
- XFR (T) - Transformer
- ZEN (Z) - Zener Diode

Make Reuse Dialog Box

To access: Select the objects you want to include > right-click > **Make Reuse** popup menu item

Use the Make Reuse dialog box to specify a reuse type and reuse name for the physical design reuse you are creating.



Objects

Field	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 cannot 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.

Field	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 *\SailWind 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 SailWind Layout Reuse Files (*.reu).
Deselection Report	Creates the report file *report.rep* in the *C:\<install_folder>\<version>\Settings* 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. **Restriction:** 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:\<install_folder>\<version>\Settings* folder and opens the file in the default text editor. The file contains a list of items included in the physical design reuse. **Restriction:** 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.
OK	The **OK** button is unavailable until you enter a Reuse Type and a Reuse Name.

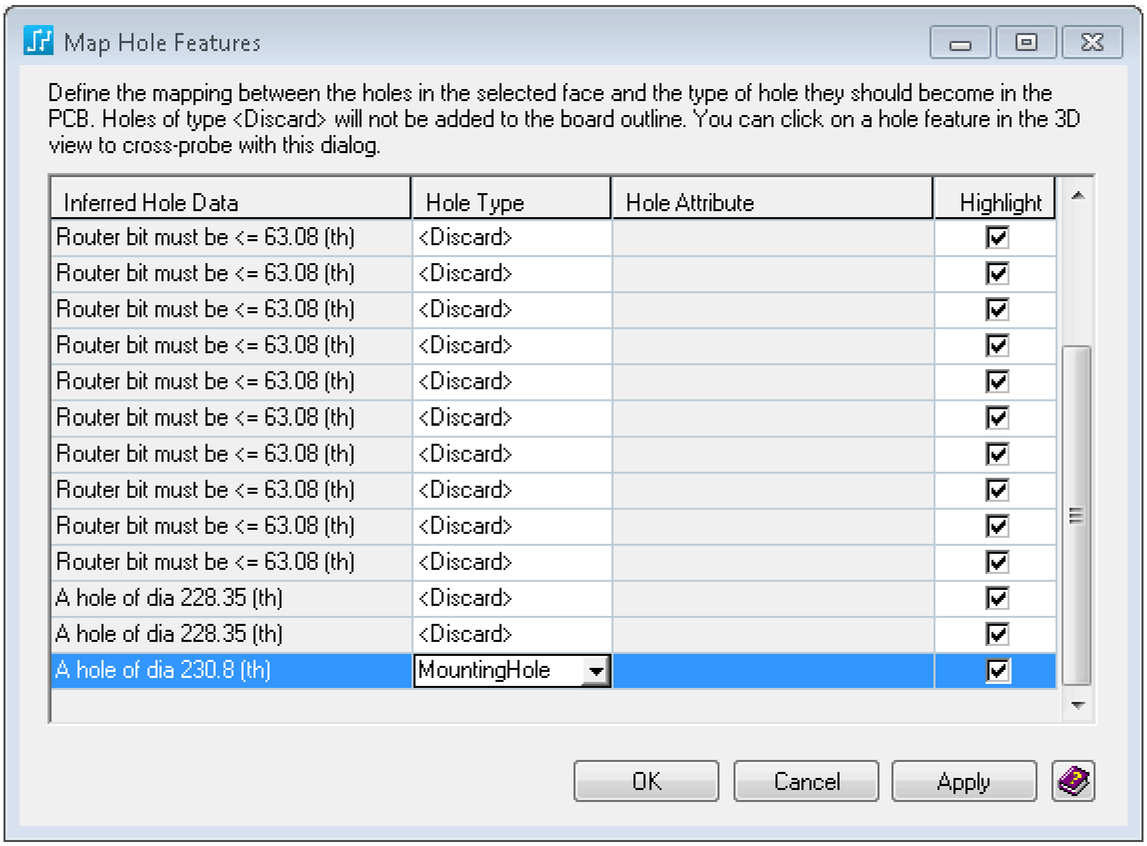
Related Topics

Reusing Designs or Parts of Designs

Map Hole Features Dialog Box

To access: Select a mechanical model in the 3D Window and click the **Create Board Outline** button; then click a point on the model to use as the board origin.

Use the Map Hole Features dialog box to map holes in an imported mechanical model to hole types (such as cutouts or mounting holes) if you want to use them in your board outline. Holes from an imported mechanical model that are not mapped to a specific hole type are discarded by default.



Objects

Field	Description
Inferred Hole Data	Displays an identifying description of the hole; for example, “A hole of dia 228 (th).” SailWind Layout generates the description based on data from the imported mechanical model.
Hole Type	Indicates how SailWind Layout should regard the hole. Select from the dropdown list by clicking in the table cell. Available choices include “Discard,” “Contour,” or “MountingHole.” SailWind Layout ignores any holes with the “Discard” selection. If you choose “MountingHole,” you must also select a corresponding attribute from the “Hole Attribute” column.

Field	Description
Hole Attribute	Select an appropriate decal or pad stack to use for the hole by clicking in the table cell and selecting from the dropdown list. SailWind Layout populates the list with all mounting holes currently defined in the design. **Restriction:** You must have at least one mounting hole already defined for your design before using this feature. The list becomes available only if you select "MountingHole" as the Hole Type.
Highlight	Select the check box if you want SailWind Layout to highlight the associated hole in the 3D Window. Highlighting the hole provides a means of quickly locating it.

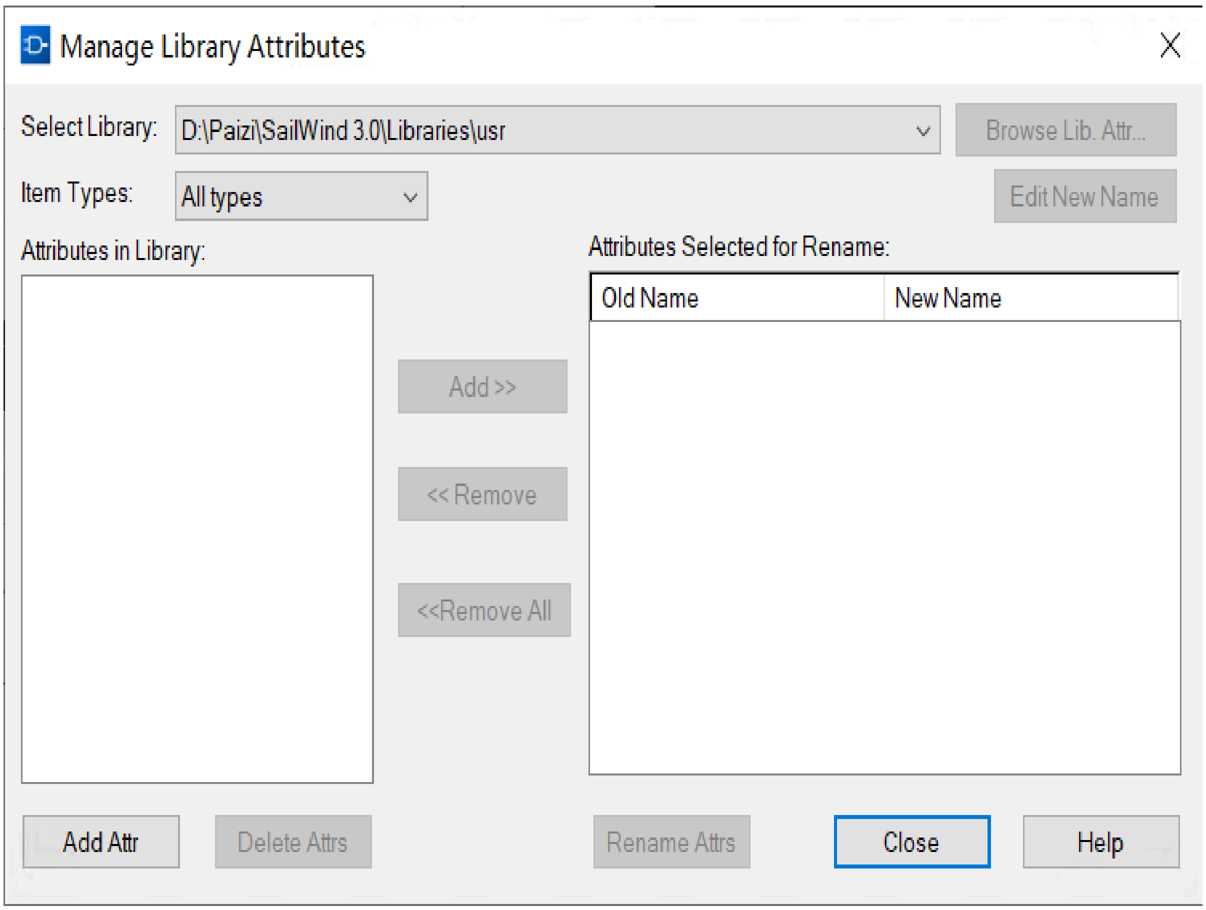
Related Topics

Creating a Board Outline From an Imported Mechanical Model

Manage Library Attributes Dialog Box

To access: **File > Library** > **Attr Manager** button

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.

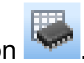


Objects

Field	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.

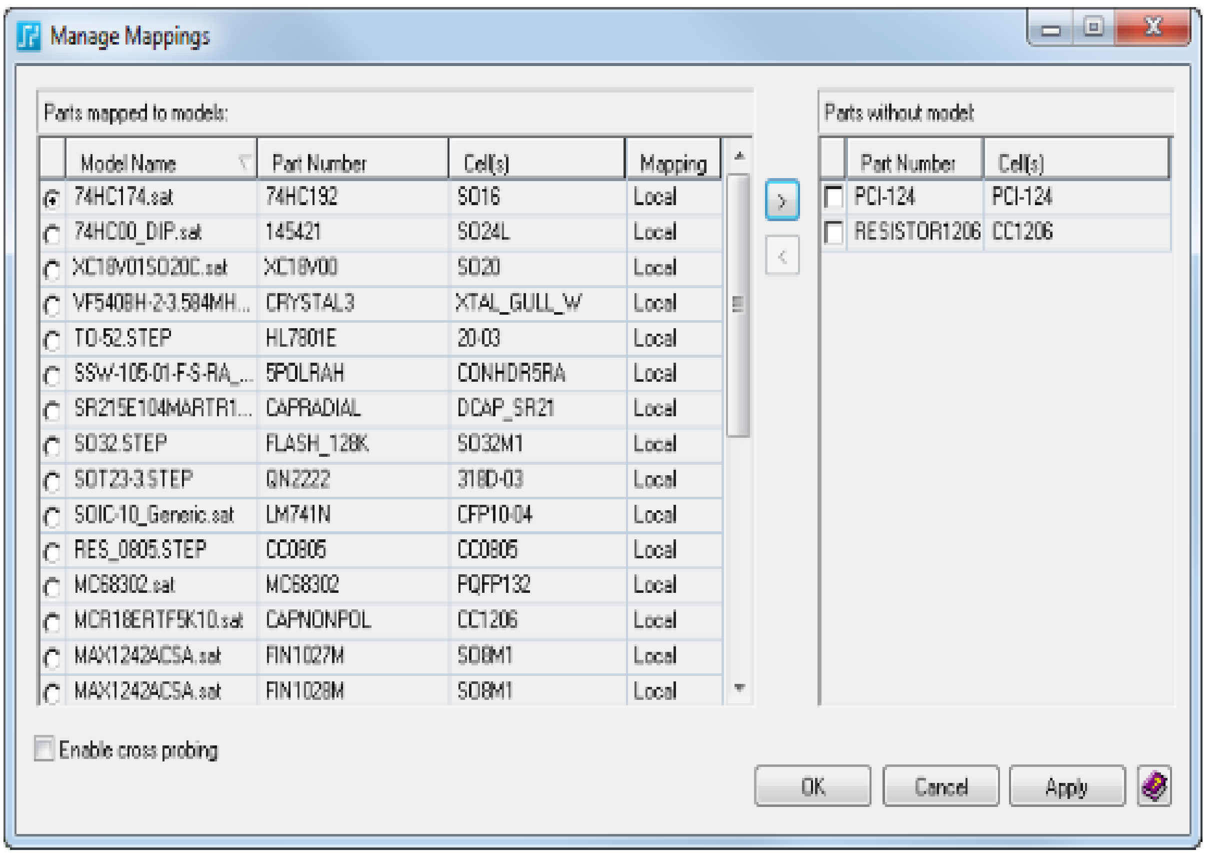
Field	Description
	Tip This is available only when an attribute in the New Name column in the Attributes Selected for Rename list is selected.
Attributes in Library list	The list of attributes in the selected library. **Tip** This is available only when an attribute in the New Name column in the Attributes 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 have selected to rename.
Add Attr	Opens the Add New Attribute to Library Dialog Box.
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.

Manage Mappings Dialog Box

To access: Open the 3D view (**View > SailWind 3D**), then click the **Manage Mappings** button .

Use this dialog box to reassign or reuse 3D models for parts on the rendered PCB design. For example, if you want to assign an SOIC 3D model to other SOIC devices on the PCB that appear physically identical, you can use this dialog box to make the assignment to the other devices.

Restriction:
You cannot reassign a 3D model to a part if the part is already saved in the library with a 3D model.



Objects

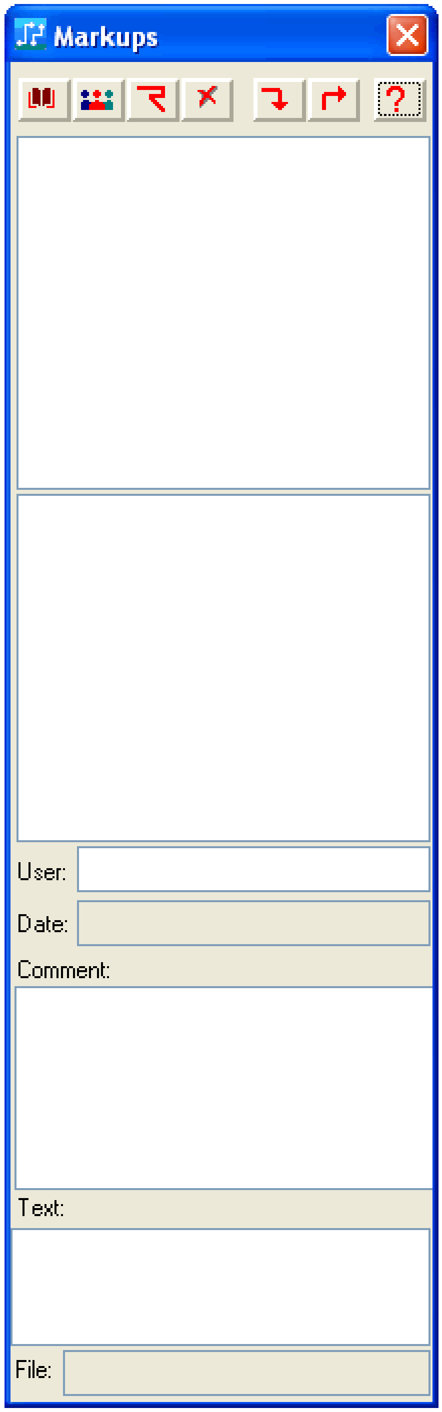
Field	Description
Parts mapped to models	Lists all 3D models and corresponding model information currently associated with parts in the PCB design. **Model Name** — Displays the model name associated with a part. If you map a model to more than one part in the design, the model name appears more than once in the list.

Field	Description
	Part Number — Displays the number of the part associated with the model. You can only assign one model to a part; therefore, part numbers appear only once in the list. **Cells** — Displays the decal name associated with the part. **Mapping** — Indicates whether the model assignment is mapped locally or in the part library. You can change model assignments for a part locally; however, if the part is saved with a model in the library, you cannot change the model assignment.
Parts without model	Lists information about all of the parts that are not mapped to 3D models. Parts not mapped to 3D models are represented by either 2D or 2.5D (extruded) models instead. **Part Number** — Displays the number of the unmapped part. Selecting the check box next to the part number allows you to map it to a model in the “Parts mapped to models” list. **Cells** — Displays the decal name associated with the part.
	Click the right arrow to remove the mapping information from the selected part model in the **Parts mapped to models** list, thereby moving it to the **Parts without models** list.
	Click the left arrow to map the part selected in the **Parts without models** list to the model selected in the **Parts mapped to models** list. This button is unavailable if you do not also select a model in the “Parts mapped to models” list. You can map more than one part to a model. If the alignment data for the 3D model is already associated with a different decal, the software enables you to choose to reuse the alignment data or perform a manual alignment.
Enable cross probing	Select this check box to enable you to interactively select a part in the design workspace and make it selected also in the **Parts mapped to models** list or the **Parts without models** list. This check box is cleared by default each time the dialog box closes.

Markups Dialog Box

To access: **Edit > Markups** menu item

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.



Objects

Field	Description
Add Topic	Click to add a new topic to the Collaboration Data tree.

Field	Description
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 SailWind 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

Exporting Markups Using the Markups Dialog Box

Exporting Markups Using File > Export

Importing Markups Using the Markups Dialog Box

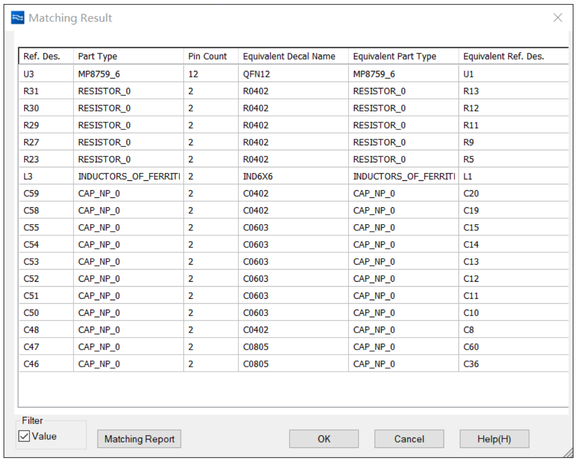
Importing Markups Using File > Import

Matching Result Dialog Box

To access:

- On the Design Toolbar click the Make Like Reuse button, and then select a reuse type/file in the "Select Reuse Module" dialog box.
- Select a Reuse in the design, and then right-click and click the **Make Like Reuse** popup menu item, or on the Design Toolbar click the Make Like Reuse button.

Use this dialog box to see whether a match to the selected reuse is found in the design. If yes, you can make a new physical reuse design with it; if not, you can view the matching report in detail.



Objects

Table 189. Parameter Description

Area	Object	Description
The first three colums	Ref. Des.	Displays part information of the selected reuse.
	Part Type	
	Pin Count	
The last three colums	Equivalent Decal Name	Displays information of parts equivalent to those in the selected reuse, which changes depending on the selection made in the Filter area. If no match is found in the Equivalent Ref. Des. list, you can clear the Value check box to have another try.
	Equivalent Part Type	
	Equivalent Ref. Des.	
Filter area	Value	Specifies whether to match parts by comparing Value.

Table 189. Parameter Description (continued)

Area	Object	Description
		Note: Value refers to part attributes Value and Tolerance.
-	Matching Report	Opens the report file *Layout.err* in your default text editor. The file contains operations and matching results from the Make Like Reuse command, which is created in your working folder.
-	**OK** button	Close the dialog box. Note that if a match is found, the new physical design reuse dynamically attaches to the pointer.

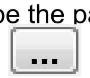
MCAD Collaborator

To access: Open the SailWind 3D window (**View > SailWind 3D**), then click the **MCAD Collaborator** button .

Use this dialog box to exchange mechanical design change information between the PCB design team and the mechanical design team. This dialog box allows you to create and annotate change proposals in a *.idx*, *.idxz*, or *.xml* file, which you can then exchange with mechanical designers.

Objects

Field	Description
Files section — Displays the file names of the change proposal files you have already processed.	
History section — Displays a reverse chronological log (the most recent first) of all of the change files for the current design.	
Clear	Deletes all of the change files displayed in the **History** pane. **Note:** If you clear the History, you cannot restore the change files later.
Communications section — Defines the file and communication parameters for the mechanical collaboration. Use this section to ensure change proposal information exchanged with the mechanical designers is coordinated. For example, define the schema units in this section so the scale of any proposed changes is understood between the mechanical designers and board layout designers.	
Write Options	Defines the units and file type for the change data files. • **Schema Version** — Allows the selection of Schema Version 2.0 or 3.0. See “Supported Design Objects for Data Exchange” on page 561. • **Schema units** — Defines the units for the change data files (millimeters, inches, microns, thousandths). • **File Type** — Defines the file type/extension for the change data files: .idx, idxz (zipped) or .xml. • **Data Transfer** — Defines the method for exchanging the change data files (email, file). • **Email Notification** check box — Enables or disable selection of Email as the method of communication. • **Email Server** — Specifies the type of email server protocol to use. Select either **SMTP** or **Outlook** (MAPI).
Receiver Info	Specifies the recipient information for communicating by email.

Field	Description
	• **Name** — Defines the name of the receiver. • **Company** — Defines the company name of the receiver. • **Email ID** — Defines the email address of the receiver.
Sender Info	Specifies the sender information for communicating by email. • **Name** — Defines the name of the sender. • **Company** — Defines the company name of the sender. • **Email ID** — Defines the email address of the sender.
Data Path	Defines the shared directory path for the change data files. You can type the path directly or browse for it by clicking the browse button. 
Collaboration Data section — Displays the file name of the change file you are processing and a tree list of all of the design elements that are impacted by the change request.	
Out of Sync Data Notification	Indicates whether the data is synchronized between PCB and MCAD. If the data is out of sync, a highlighted message appears. Click the button to synchronize the data.
File Type Selector	Defines the type of file for the change. Choose one of the following options from the dropdown list: • **Baseline** — Initial design data file. • **Proposal** — Proposed change data file. • **Response** — Response data file with acceptance or rejection of proposed change.
Tree List	Displays all of the change items in the selected change file. The items are grouped by the type of design element. The icon for each item indicates the type of change: added, deleted, modified. Checked, the item is accepted. Unchecked, the item is rejected.
Add 	Adds a new change request.
Delete 	Deletes an existing change request.
Modify 	Modifies an existing change request.
Highlight/Unhighlight Graphics	Enables 3D animation of the selected change in the 3D View. The design elements impacted by the change are highlighted.

Field	Description
	
Expand Tree Data 	Expands the entire Tree List.
Collapse Tree Data 	Collapses the entire Tree List.
Filter Objects 	Opens the Object Filter dialog box.
Update Tree Data 	Updates the Collaboration Data list and shows only the object types you selected in the Collaboration Objects Filter list.
Properties section — Displays the properties for the change you are processing. The information displayed in the **Properties** pane changes according to what type of item you have selected (change file, design element). • Green text indicates added elements. • Red text indicates deleted elements. • Blue text indicates modified elements.	

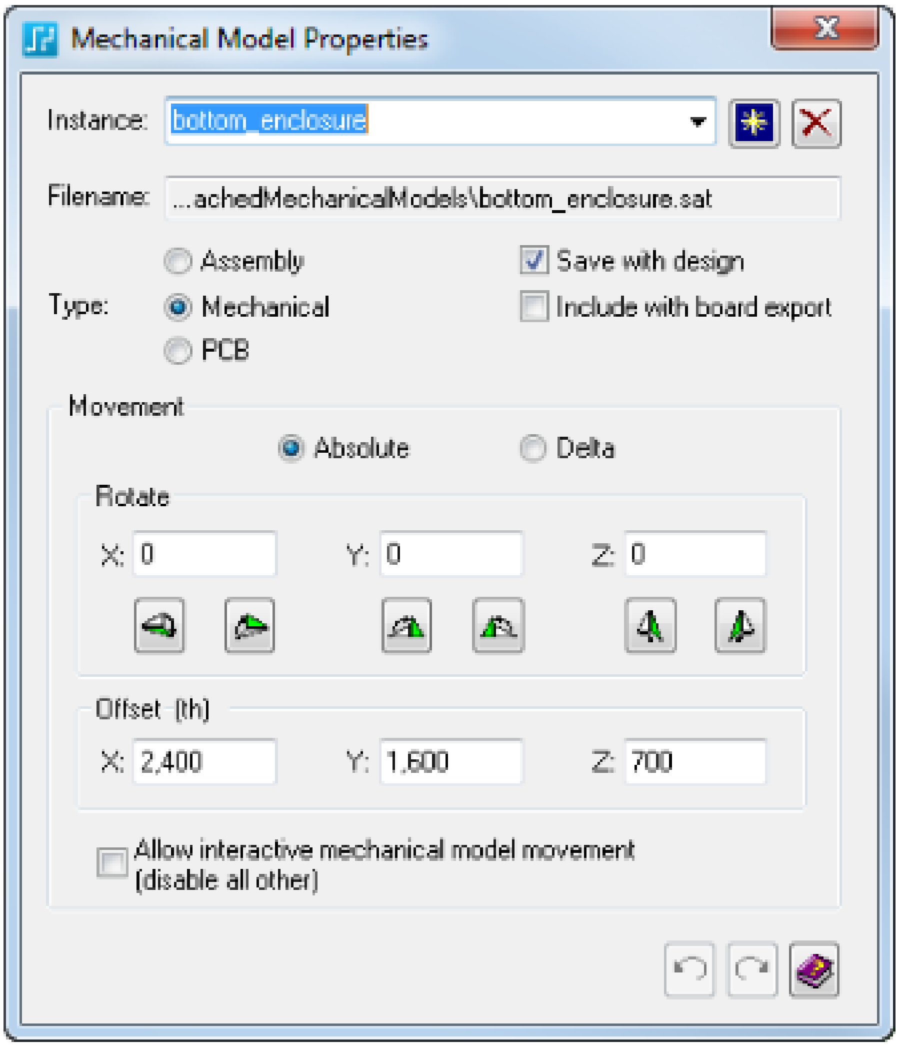
Related Topics

Exchanging Data with Mechanical Designers

Mechanical Model Properties Dialog Box

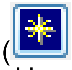
To access: Select **View > SailWind 3D** menu item, then click the **Mechanical Model Properties** button in the toolbar of the 3D dialog box .

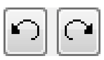
The Mechanical Model Properties dialog box allows you to associate mechanical models such as mounting brackets or covers with a PCB design. Use this dialog box to also modify certain properties (such as position) of mechanical model instances in your design.



Objects

Field	Description
Instance	Displays the name of a mechanical model instance used in the design.

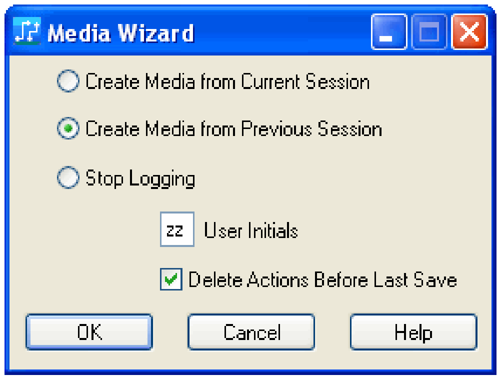
Field	Description
	• Click **Add** () to add a new mechanical model to the design. Clicking the add button opens the Import Mechanical Model dialog box to allow you to select a mechanical model. The imported model appears in the dropdown list with a default instance name of “Temp name *x*” where *x* is an iterative mechanical model instance. • Click **Delete** () to remove an existing model from the dropdown list. The model is also removed from the design. When the Mechanical Model Properties dialog box is open, select any mechanical model in the design (such as a bracket) to view its corresponding instance name in the box. You can also select a mechanical model from the dropdown list to select it. If desired, you can edit a mechanical model’s instance name by typing directly in the box.
Filename	Displays the filename and directory path for the model definition file.
Type	Defines the model type. Defining a model type for the mechanical model categorizes the model to allow you to apply specific 3D clearances (in the 3D Clearances Dialog Box) to the model. • **Assembly** — The model is used for assembly purposes (alignment hole, card rail, and so forth). • **Mechanical** — The model is a mechanical part (mounting hole, card ejector, and so forth). • **PCB** — The model is a daughter PCB that is connected to the PCB.
Save with design	Selecting the check box saves the modified model properties (such as its Rotation, Offset, and Type settings) with the current PCB layout design. This check box is selected by default.
Include with board export	Selecting this check box saves the mechanical model with the PCB design when you select the **Board only** option during design export. **Tip** Alternatively, you can select the All mechanical models option in the Export window to save all mechanical models with the design.
Movement	Defines the type of values for the Rotate and Offset options: • **Absolute** — The values you enter are absolute coordinate values based on a fixed XYZ origin. • **Delta** — The values you enter are difference values from the current XYZ location coordinates for the model.

Field	Description
	• **Rotate** — Defines the X, Y, and Z axis rotations for the model. Click the appropriate icon to set the orientation of the model. • **Offset** — Defines the X, Y, and Z axis offsets for the model location.
Allow interactive mechanical model movement (disable all other)	Selecting this check box allows you to reposition the mechanical model by dragging it with the mouse. When cleared, you can reposition the mechanical model using only the **Movement** fields.
Undo/Redo buttons 	Allows you to undo or redo alignment and rotation actions.

Media Wizard Dialog Box

To access: type BMW > press the Enter key

BMW (Basic Media Wizard) is a tool for recording and playing back SailWind Logic, SailWind Layout, and SailWind 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.



Objects

Field	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 SailWind tool crash. • **Create Media from Previous Session** — Use this procedure when the session you are recreating caused the SailWind 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.

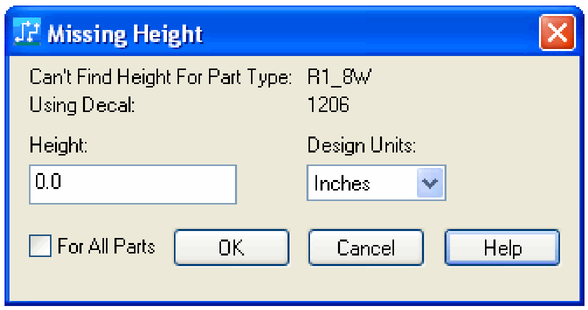
Related Topics

BMW and BLT

Missing Height Dialog Box

To access: 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.

Use the Missing Height dialog box to specify missing heights when exporting the design to IDF.



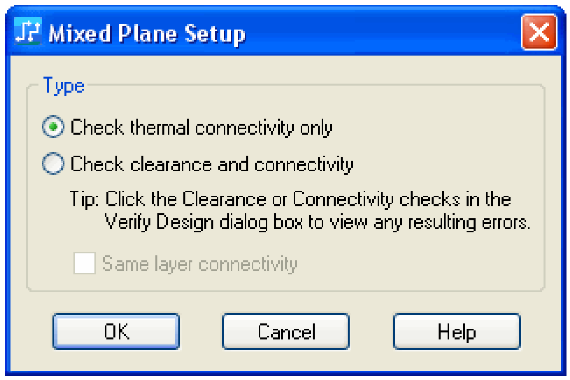
Objects

Field	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.

Mixed Plane Setup Dialog Box

To access: **Tools > Verify Design** menu item> Plane check > **Setup** button

Use the Mixed Plane Setup dialog box to set the type of plane checking.



Objects

Field	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, the plane is flooded 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.

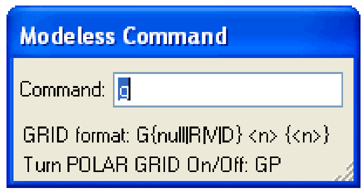
Related Topics

Plane Checking Setup

Modeless Command Dialog Box

To access: Type the modeless command (shortcut key) for the command you want.

You can set or change some settings and functions at any time using a shortcut key for the command called a *modeless command*.



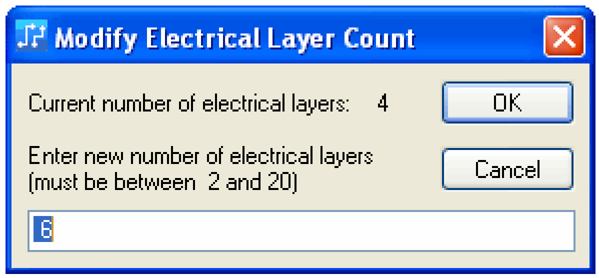
Objects

Field	Description
Command	Type a modeless command on page 83 and then press the Enter key to execute.

Modify Electrical Layer Count Dialog Box

To access: **Setup > Layer Definition** menu item > **Modify** button

Use the Modify Electrical Layer Count dialog box to increase or decrease the number of electrical layers in the design.



Objects

Field	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 cannot 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 cannot delete any layers because of data.

Related Topics

Modifying the Number of Electrical PCB Layers

Reassigning Electrical Layers

Reassign Electrical Layers Dialog Box

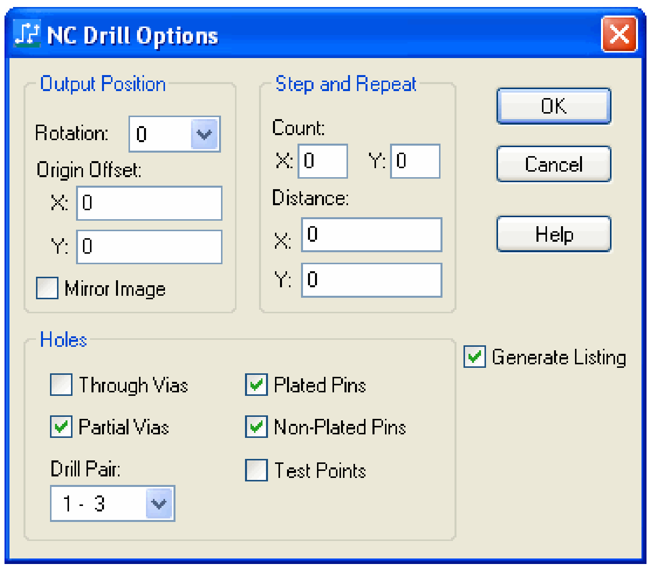
Layers Setup Dialog Box

NC Drill Options Dialog Box

To access:

- **File > CAM** menu item > **Add** button > select NC Drill from the Document Type list > **Drill** button > **Options** button
- **File > CAM** > select a document name > **Edit** button > select NC Drill from the Document Type list > **Drill** button > **Options** button

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.



Objects

Field	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.
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.

Field	Description
Partial Vias	Specifies to drill partial vias. **Restriction:** Available only if you have set up Drill Pairs on page 1337.
Drill Pair	Specifies which layer pair to drill. **Restriction:** Available only if you have 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

To access:

- **File > CAM** menu item > **Add** button > select NC Drill from the Document Type list > **Drill** button > **Device Setup** button
- **File > CAM** menu item > select a document name > **Edit** button > > select NC Drill from the Document Type list > **Drill** button > **Device Setup** button

Use the NC Drill Setup dialog box to set Excellon output or Drill Listing parameters.

Description

Figure 186. Excellon Tab

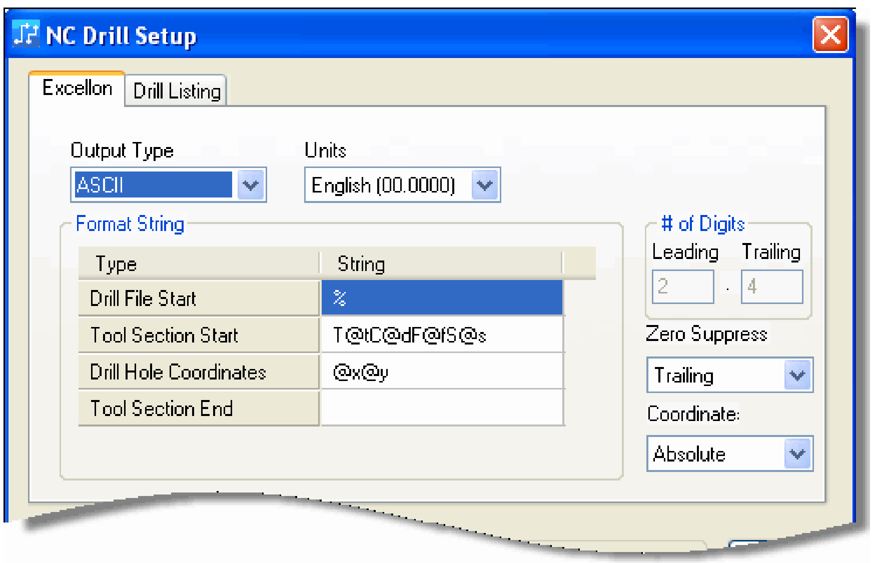


Figure 187. Drill Listing Tab

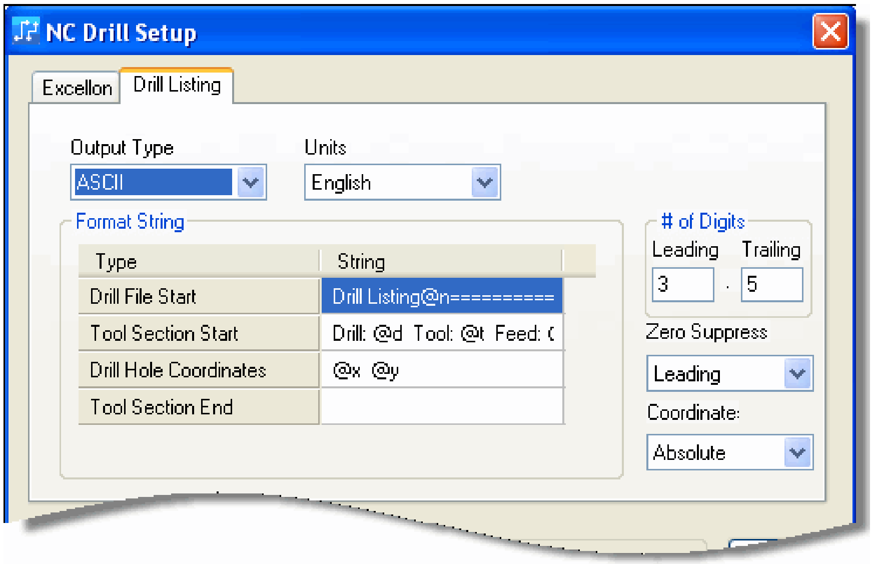
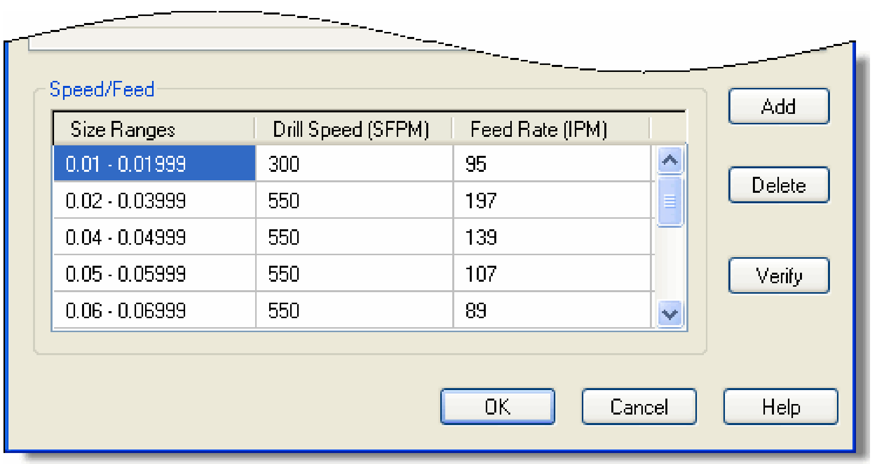


Figure 188. NC Drill Setup Dialog Box



Objects

Table 190. NC Drill Setup Dialog Box Fields

Field	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.

Table 190. NC Drill Setup Dialog Box Fields (continued)

Field	Description
String column	Lists the format string. Double click to edit. **Tip** • 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.
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

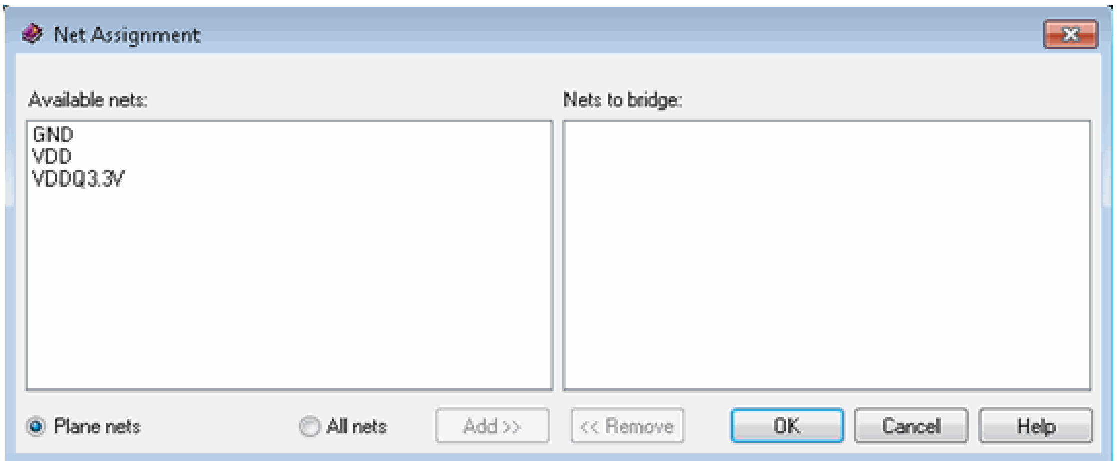
Creating an NC Drill File

Creating CAM Outputs to Manufacture Your PCB

Net Assignment Dialog Box

To access: Add copper to the design or open the properties of copper and in the Add Drafting or Drafting Properties dialog box on page 1054, select the “Copper bridge. Select the nets to bridge...” check box and then click the **Nets to bridge** button.

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 you enabled online Design Rule Checking (DRC), SailWind Layout does not flag the bridge copper as a violation, but catches all other accidental physical connections between the nets.



Objects

Field	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 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.
Plane nets	Filters the Available nets list to display only those assigned to planes (CAM or split/mixed layers), as set up in the Layers Setup Dialog Box.
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 of a Netlist Project

To access: Select a net > right-click > **Properties** popup menu item

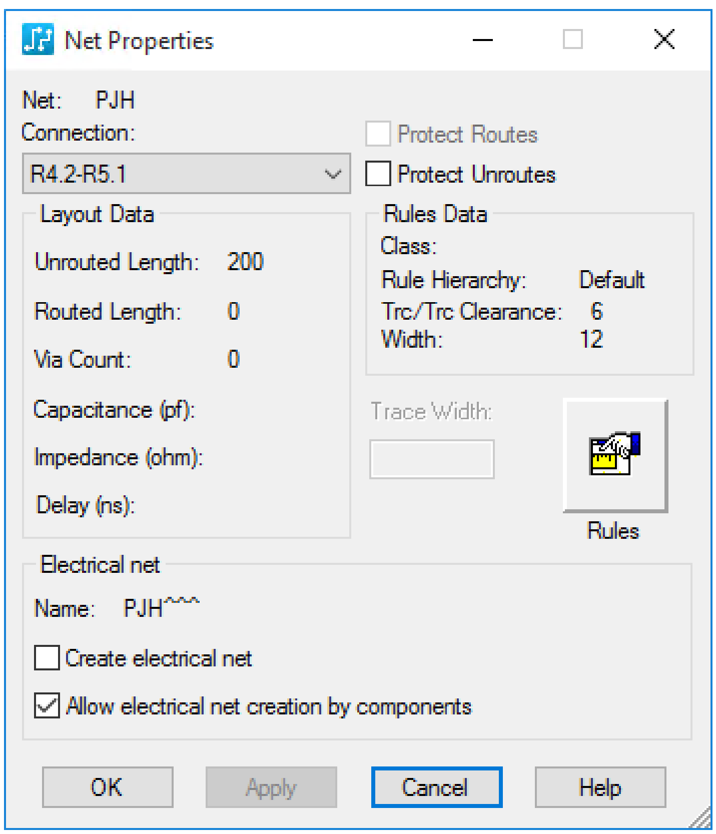
Displays the netname, the list of pin-to-pin connections, routing information, and rules data of one or more selected nets in a netlist project.

Description

You can also:

- set route protection for nets.
- set the trace width
- make settings that control the creation of electrical nets.

Restriction:
Some options in this dialog box are unavailable if the net is part of a physical design reuse or contains protected routes.



Objects

Field	Description
Net	Displays the name of the net.
Connection	Lists the pin pair connections available in the net.
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. Includes half the Discrete length value of each connected pin of components that have a Discrete length assigned. When routed to the pad, measurements are made to the origin of the pad. • **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 Parameter Setup 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 on page 1775 after you set up a High Speed check on page 1363 to check Capacitance. It is not verified by online DRC on page 1843. • **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 Parameter Setup 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 on page 1775 after you set up a High Speed check on page 1363 to check Impedance. It is not verified by online DRC on page 1843. • **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 Parameter Setup 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 on page 1775 after you set up a High Speed check on page 1363 to check Delay. It is not verified by online DRC on page 1843.
Protect Routes	Protects selected routes on page 1855 or traces on page 1867 from being moved. Similar to gluing components but you can allow protected traces to be edited by SailWind Router in the Routing Rules on page 1666.

Field	Description
	See also: "Route and Unroute Protection".
Protect Unroutes	Protects unrouted connections and the unrouted portions of partial routes on page 1846. 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 on page 1854 instead of the full route-completion target on page 1854. See also: "Route and Unroute Protection".

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. on page 1167 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. on page 1167 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 electrical net are selected, displays the name of the electrical net to which the selected net(s) belong. If nets belonging to more than one electrical net are selected, no electrical net names are listed. An electrical net name has a suffix of three caret symbols.
Create electrical net	Select the check box to create an electrical net with one or more attached components. **Tip** You can also set this check box (and create electrical nets) using the Create Electrical Net popup command for selected nets. See "Electrical Net Creation" on page 338. Clear the check box to remove any selected net that is part of an electrical net. Clear both check boxes to exclude the selected nets from inclusion in any electrical net.

Field	Description
Allow electrical net creation by components	Select the check box to allow the selected nets to be included in electrical nets by: • Specifying, in the Electrical Nets dialog box, the refdes prefix of the nets' joining component, or • Selecting the Create electrical net check box in the Component Properties dialog box. Clear the check box to disable electrical net creation by component (including by refdes prefix). **Tip** This check box is also cleared by the Disable Electrical Net Creation popup command for selected nets. **Restriction:** Nets that have the Create electrical net check box selected are not disabled from electrical net creation by clearing this check box. Clear both check boxes to exclude the selected nets from inclusion in any electrical net.

Related Topics

Modifying Net Properties

Net Properties Dialog Box - Design Reuse

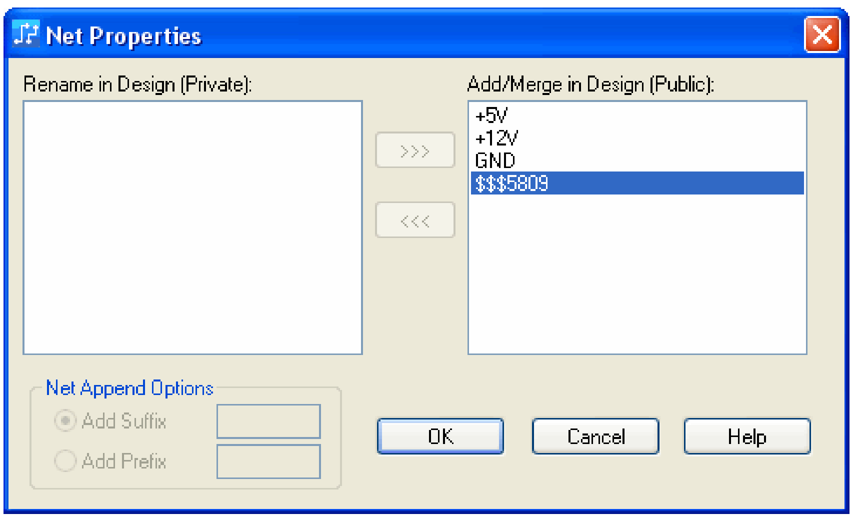
To access: Select a reuse with nets > right-click > **Properties** popup menu item > **Net Properties** button

Use the Net Properties dialog box to resolve netname conflicts between netnames in a physical design reuse and netnames in the design.

Restriction:
You cannot 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.

Description

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.



Objects

Field	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.

Field	Description
Add/Merge in Design (Public)	Lists nets in the physical design reuse to either add to the design (because the net does not 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.
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.

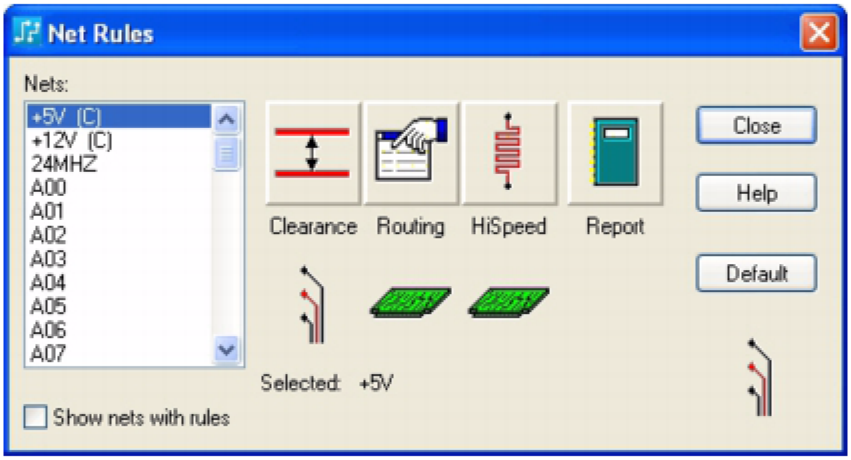
Related Topics

Reusing Designs or Parts of Designs

Net Rules Dialog Box

To access: **Setup > Design Rules** menu item > **Net** button

Use the Net Rules dialog box to define design rules that apply to nets.



Objects

Field	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

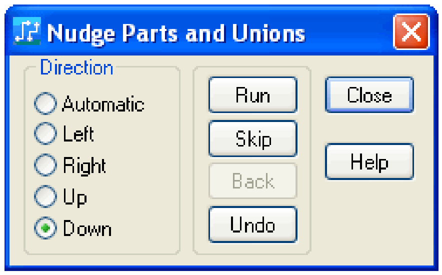
Resetting Net Rules to Default Rules

Design Rule Hierarchy

Nudge Parts and Unions Dialog Box

To access: Automatic or Prompt mode must be set in the Nudge area of **Tools > Options** menu item > **Design** category. Move a part or a union over another part or union. The dialog box opens.

Use the Nudge Parts and Unions dialog box to control the nudge process.



Objects

Field	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.

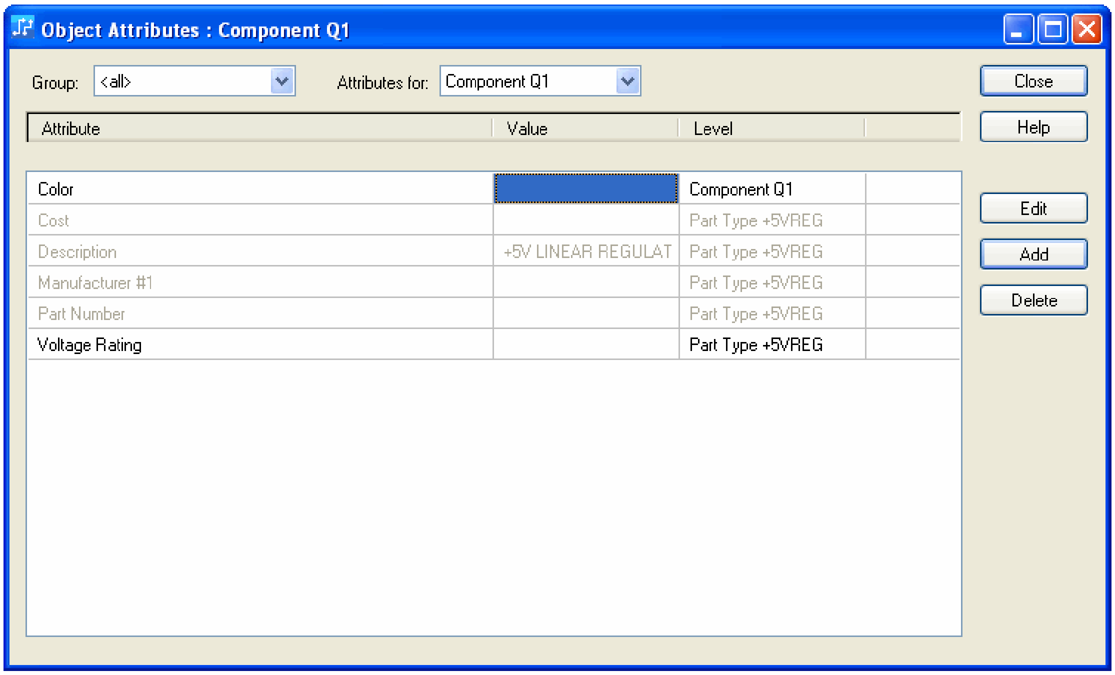
Related Topics

Nudge Overlapping Parts

Object Attributes Dialog Box

To access: Select object(s) > right-click > **Attribute** popup menu item

Use the resizable Object Attributes dialog box to add, modify, or remove attributes of single objects or multiple objects of the same type.



Objects

Field	Description
Group	Filters the Attributes list. You can choose an attribute group on page 1811 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. 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.
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

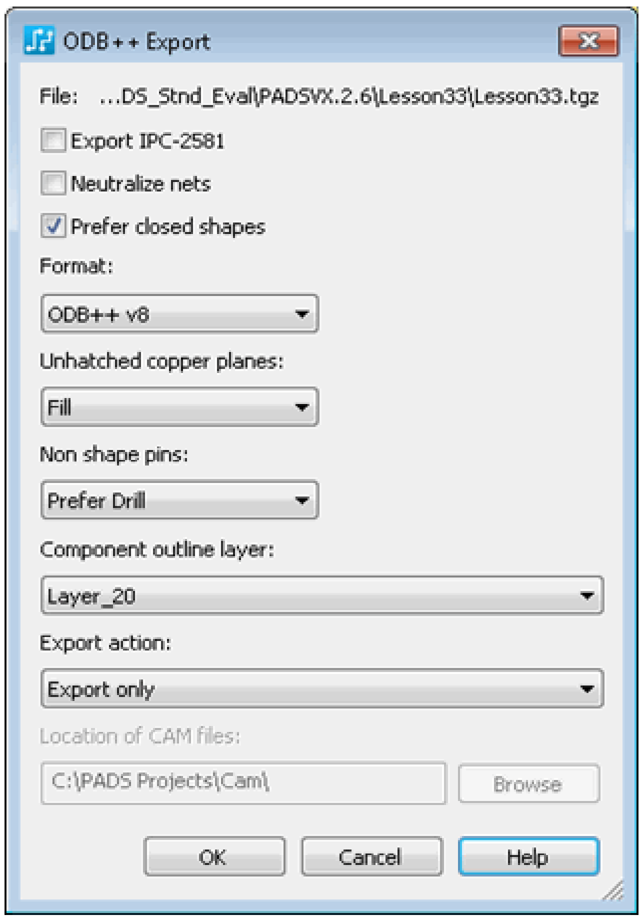
Design Object Attributes

ODB++ Export Dialog Box

To access: Choose the **File > Export** menu item and choose “ODB++ (*.tgz)” from the “Save as type” dropdown list; then click **Save**.

Use this dialog to export a design to an ODB++ file.

Tip
Virtual pins in the design are exported as vias.



Objects

Field	Description
Export IPC-2581	Exports IPC-2581 standard data for assembly and fabrication. Creates the file <design name>.xml in the same location as the ODB++ .tgz file.
Neutralize Nets	Renames nets numerically.
Prefer closed shapes	Specify how to determine component outlines:

Field	Description
	• 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.
Format	Choose the output format (v7 or v8) required by your manufacturer. The v8 format contains the addition of impedance constraints from nets (layer, minimum and maximum impedance, and the impedance value).
Unhatched copper planes	Specify what to do with unhatched copper planes: • **Fill** — Flood all copper planes before translating. • **Discard** — Discard unhatched copper planes. This list is unavailable if there are no unhatched copper planes in the design. If you want unhatched copper planes, close and reopen the design.
Non Shape Pins	Specify what to do with pins for which no pad shape is defined or when you do not want to use the defined pin shape. Non-shape pins are pins for which the design contains no pin definition, or for which the pin definition has zero shape or a very small shape of one or two basic units (one basic unit = 1/400,000 MM). By default, when such a situation is encountered, the translator attempts to extract a pin shape from the padstack that is assigned to the pin. If there is no padstack, or if the shape based on the padstack is very small, the translator tries to use the shape of the padstack drill. You can use parameter None Shape Pins to control how to generate shapes for non-shape pins, or to specify from which section of the file to take pin shapes. • **Discard** — Do not translate these pins. (Do not use this option for pins assigned to a net.) • **Add empty** — Translate these pins as-is (with no shape). • **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 a pad shape is defined: • **Prefer Drill** — Use the drill shape instead of the pin pad shape. • **Prefer Stack** — Use the padstack pad shape instead of the pin shape. • **Prefer Piece** — If the padstack pin contains associated coppers, use these instead of the pin shape.
Component Outline Layer	Specify the preferred component outline layer.
Export action	Choose an export action:

Field	Description
	• **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 the F1 key to open the CAM Compare help documentation.
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

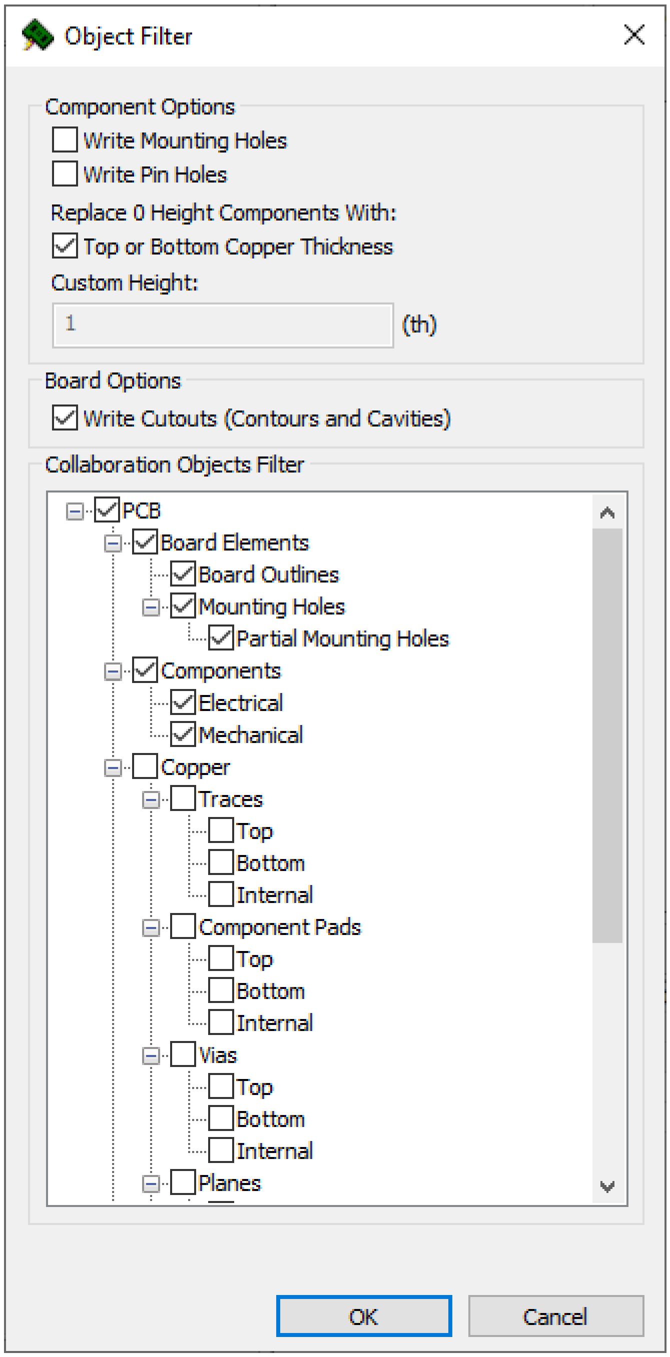
Exporting ODB++ Files

Object Filter Dialog Box

To access: Open the SailWind 3D window (**View > SailWind 3D** menu item), click the **PADS MCAD Collaborator** button  , then click the **Object Filter** button 

Use this dialog to specify the objects to include in the change data files generated by MCAD Collaborator.

Figure 189.



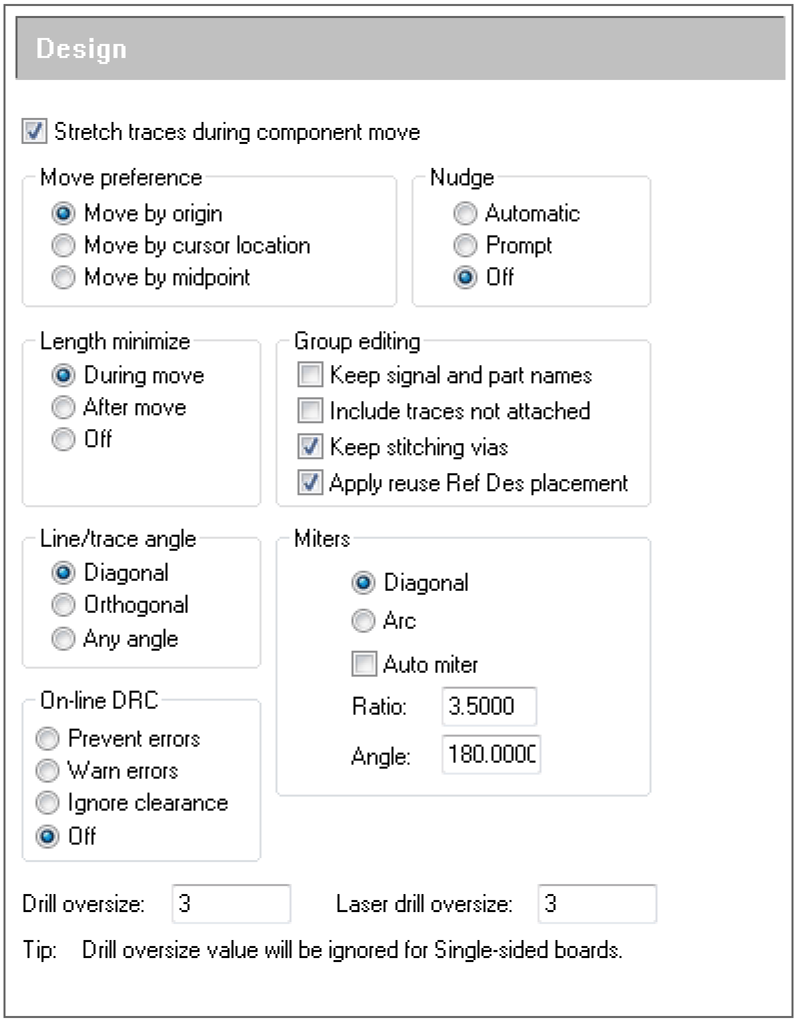
Objects

Field	Description
Component Options section — Defines the component elements to include in the change data files.	
Write Mounting Holes	Checked, include all component mounting holes.
Write Pin Holes	Checked, include all component pin holes.
Replace 0 Height Components With:	Defines a replacement value for zero height components. • **Top or Bottom Copper Thickness** — Checked, replaces the “0” height value for a zero height component with the thickness of the copper cladding on the top or bottom layer, depending on where the component is placed. • **Custom Height** — Defines a custom height value as a replacement for the “0” height value of a zero height component.
Board Options section — Defines the board elements to include in the change data files.	
Write Cutouts (Contours and Cavities)	Checked, includes all cutouts (contours and cavities).
Collaboration Objects Filter section — Defines the Collaboration Data objects to include in the change data files. Checked items are included in the generated data files. Unchecked items are not included in the generated data files. **Note:** The following additional objects (not shown in the list) are extracted if you check specific objects. If you check “Pads, Traces and Vias”, then fiducials are also extracted.	

Options Dialog Box, Design Category

To access: **Tools > Options** > **Design** category

Use the Design category to specify place and route options.



Objects

Field	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.

Field	Description
	See also: "Moving Items With Stretch Traces During Component Move"
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 on page 500 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.
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 menu > Paste. When items are copied to a new .pcb database you can set Keep Signal and Part Names to retain the netnames of traces and the reference designators. However, if you paste into an existing design and these netnames or reference designators already exist, incoming netnames or reference designators are sequentially updated. The default is

Field	Description
	to automatically generate new default reference designators and netnames when you paste to a new file. • **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 on page 829, delete component on page 829, swapping a pin on page 839, swapping all pins on page 838, change component on page 826 • 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. • **Apply reuse Ref Des placement** — When using Make Like Reuse, the properties of the reference designators match the reuse definition instead of the properties of those existing in the design. All properties of a reference designator are matched to the reuse - the location, font style, height, width, rotation, number of labels, layers on which they are placed, and so on.
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.
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 (for example, 2D lines and copper). To auto miter traces with 90 degree corners, you must select pin pairs or nets, right-click and click the **Add Miters** popup menu item. • **Ratio** — Affects the size of a diagonal miter or the radius of an arc miter. **Tip** • For diagonal miters, in the case of drafting items, the distance from the virtual corner to either of the breakpoints is created by the Ratio multiplied by the drafting line width. For example, a 10-mil line 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. In the case of traces the distance from the virtual corner to either of the breakpoints is created by the Ratio multiplied by half the trace width. For

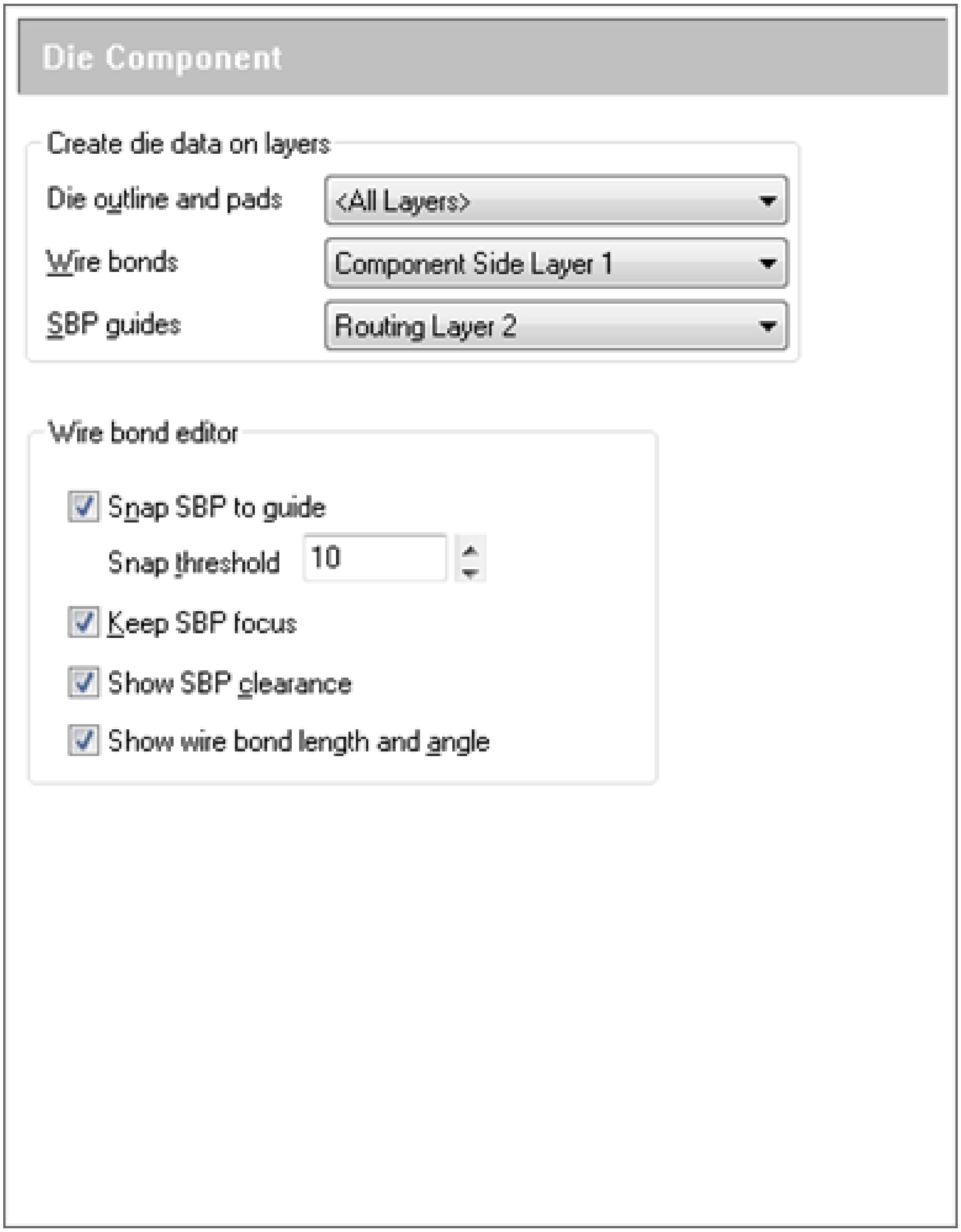
Field	Description
	example, a 10-mil trace and a ratio of 5 produces a 45-degree angle 25 mils apart from the virtual corner. • For arc miters, the radius is set to the ratio multiplied by half the trace width. For example, a ratio of one sets the arc radius equal to the half the trace width. Also, a 10-mil trace and a ratio of 5 produces a radius of 25 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 - allows parts to touch but not overlap each other. • **Off** — Disables design rule checking. You can 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.”
Drill oversize	Type a value to globally apply an oversize to plated holes. Since hole sizes are finished size with plating, the drill oversize value is required to verify against design rules to ensure that the larger holes drilled during manufacture are not going to cause problems. Clearance, and Latium Design Verification checking use the value. This value is ignored if Single-sided board is selected on the Layers Setup Dialog Box. **CAUTION:** The Drill oversize value does not apply to the Drill Drawing table, or NC Drill outputs - those values are finished hole sizes. The value affects copper plane flooding clearances, based on the drill-to-copper clearance and its CAM output (if the “Use design rules for thermals and antipads” setting is enabled in the **Copper Planes** category > **Thermals** subcategory). Your PCB fabricator normally oversizes plated holes to match the drill size that you specify in the pad stacks (finished hole size). Your PCB fabricator may also increase the apertures of antipads in pour areas using pre-production software and you may need to disable the drill oversize value after you have verified your design is production ready and you are ready to create your gerber files. **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.
Laser drill oversize	Type a value to globally apply an oversize to partial vias that will be laser drilled. Partial vias are set as laser drilled in the “Pad Stacks Properties Dialog Box” on page 1566. See the “Drill oversize” description for more information.

Options Dialog Box, Die Component Category

To access: **Tools > Options** > **Die Component** category

Use the Die Component category to specify die part creation and modifying options.

Note:
This information applies only to the BGA toolkit.



Objects

Field	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	

Field	Description
Snap SBP to Guide	Enables snap-to-guide mode when moving substrate bond pads. If 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. If you are moving multiple SBPs, each SBP is snapped to the nearest guide independently.
Snap threshold	Specifies the distance in current design units at which a moved substrate bond pad snaps to the nearest substrate bond pad guide. Snap SBP to guide does not work above this value. The threshold value applies to all SBP guides for all die components in the design.
Keep SBP focus	Automatically rotates the substrate bond pad to match the direction of the wire bond.
Show SBP clearance	If 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 on page , spinning on page 1861 SBPs, or adding fanouts on page . 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. For more information, see “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. For more information, see Design Rule Checking”.
Show wire bond length angle	If 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 are moving, adding on page , spinning on page 1861 SBPs, or adding fanouts on page or wire bonds on page . 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.

Related Topics

Moving a Substrate Bond Pad in the Layout Editor

Adding Substrate Bond Pads

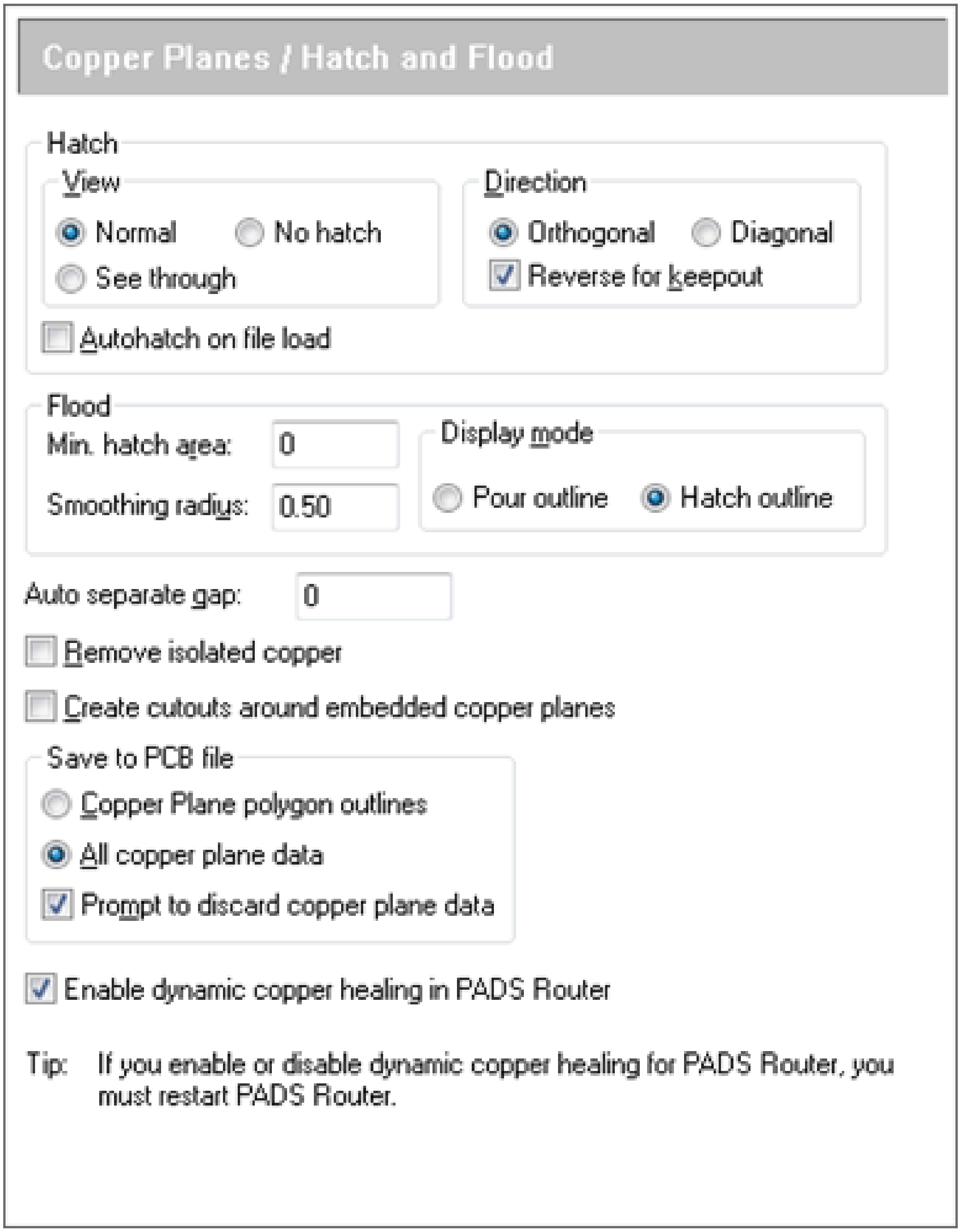
Spinning a Substrate Bond Pad

Adding Wire Bonds

Options Dialog Box, Copper Planes Category, Hatch and Flood Subcategory

To access: Choose the **Tools > Options** menu item > **Copper Planes** category >

Use the **Hatch and Flood** subcategory to view and edit options for copper planes.



Objects

Field	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 vertical and horizontal lines. • **Diagonal** — Hatching consists of diagonal 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	Hatches the copper planes automatically when you load the file. If you do not enable this feature, you must apply hatching through the Copper Plane Manager each time you open the file.
Flood area	
Min. hatch area	Specifies the square root, in current design units, of the minimum hatch outline area - the smallest island area created by flooding. Any areas that are smaller do not flood. Example: If you do not want to display islands smaller than 9 square design units, type 3 into the box.
Smoothing radius	Controls the radius of copper plane corners. <corner radius> = <shape outline width> * <smoothing radius> The smoothing radius parameter can be set between 0 and 5.
Display mode area	• **Pour Outline** — Display the copper plane outline with no hatching • **Hatch Outline** — Display the copper plane outline and hatching **Note:** If you attempt to flood a copper plane 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.
Auto separate gap	Specifies the plane-to-plane gap used by the “Auto Separate” on page 683 command and when “generating a copper plane from the board outline” on page 668. **Tip** The default value is the Board-to-Trace clearance.
Remove isolated copper	Specifies to automatically remove isolated copper areas, which are areas not connected to nets, during the flood operation of copper planes.

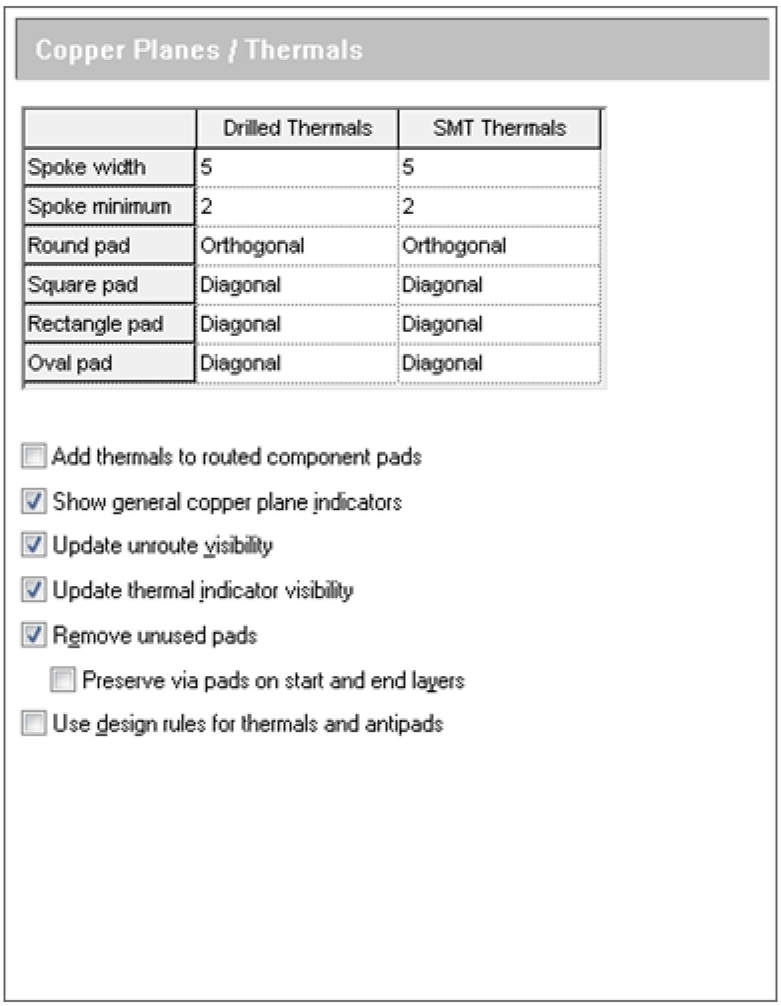
Field	Description
Create cutouts around embedded copper planes	Automatically creates cutouts around copper planes if the copper plane is placed inside another. **Tip** The cutout is combined with the external area.
Save to PCB file	Use these options to specify whether to save various forms of copper plane data in the design file. • **Copper Plane polygon outlines** — Saves only the mixed plane area polygons in the SailWind Layout file. Discards generated thermals and antipads. Saving only the plane polygon outlines results in a smaller SailWind Layout file and a faster design load time. • **All copper plane data** — Saves all data associated with mixed planes in the SailWind Layout file. **Note:** The fill or hatching of copper planes 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 copper plane data” option is selected.
Enable dynamic copper healing in SailWind Router	Opens the design in SailWind Router with the dynamic copper healing feature enabled automatically. Dynamic copper healing displays real-time edits you make to copper flooding in SailWind Router, such as changes to clearances around pads, vias, and other objects. Refer to the SailWind Router Help for more information.

Options Dialog Box, Copper Planes Category, Thermals Subcategory

To access: Choose the **Tools > Options** menu item >

Use the **Copper Planes** category > **Thermals** subcategory to specify options for copper planes that SailWind Layout automatically generates for existing plane outlines.

Note:
Some of these options only apply to copper planes when there are no custom thermals in the pad stack or when the “Use design rules for thermals and antipads” check box is selected.



Objects

Field	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.
Min. Spoke	Specifies the minimum number (1-4) of spokes. **Tip** If the pad intersects the boundary of a copper plane, 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 plane to flood over vias. See also: "Flood and Hatch Options Dialog Box". • **No Connect** — Creates no thermal reliefs.
Add thermals to routed component pads	Enables you to place thermals on routed pads. Thermals are normally generated only on unrouted pads, and vias. **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 copper plane indicators	Enables the display of a general copper 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. **Tip** • General copper 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 copper plane indicators can sometimes be covered during editing. Redraw the screen to view all of the indicators.
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.

Field	Description
Remove unused pads	Removes unused pads and replaces them with antipads. All pads are removed on all internal “No Plane” and “Split/Mixed” layers unless the pin is connected to a trace or a copper plane on that layer. Prior to PADS VX.2.4, unused pads were removed only from “Split/Mixed” layers. **Tip** CAM planes using custom antipads defined in the Pad Stacks require this setting, otherwise the default antipad setting is 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 a plane, specifies to use the hierarchical Pad to Copper clearance rule for thermals and the Drill to Copper clearance rule for antipads. **Tip** • 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.

Related Topics

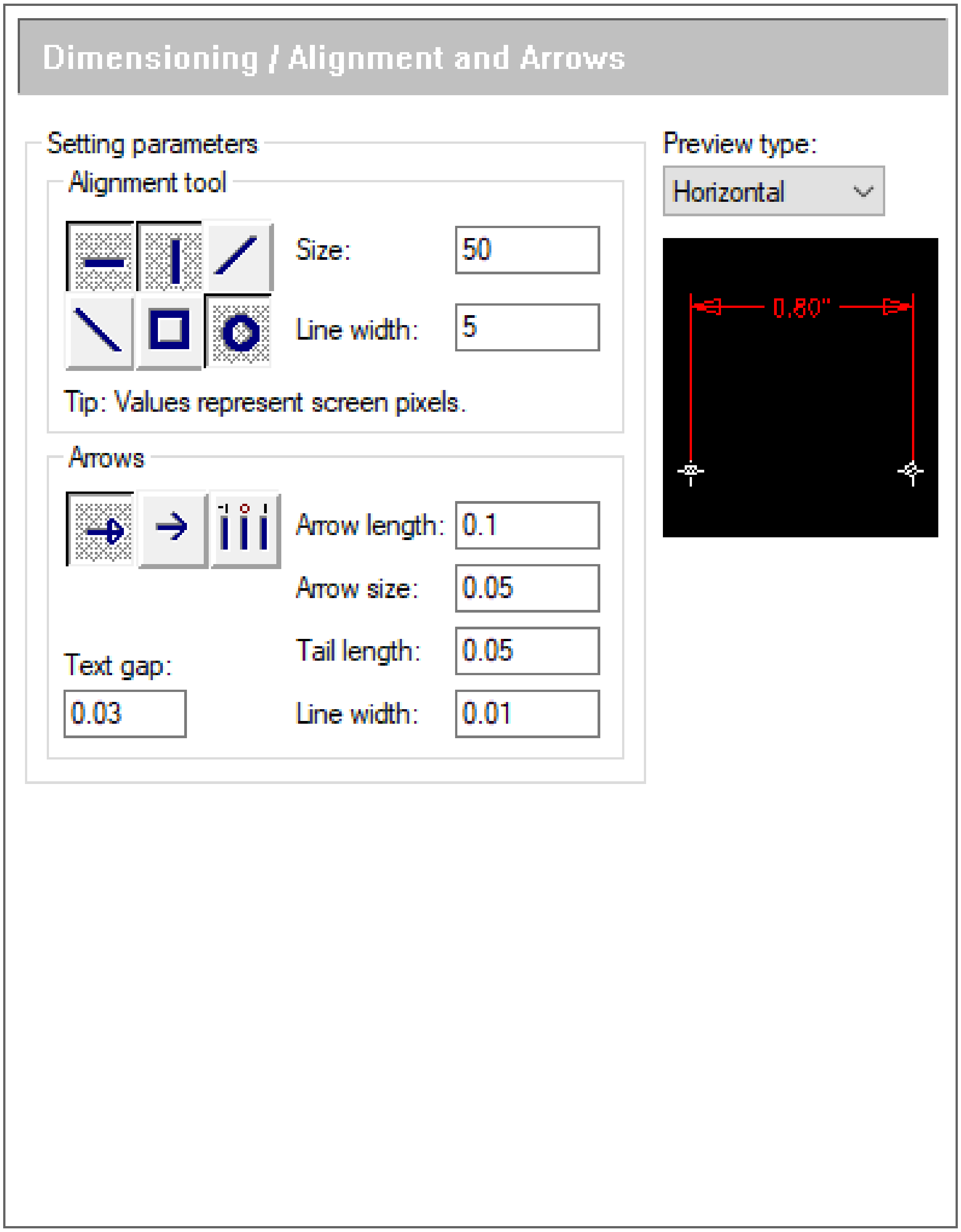
Customizing Design Rule Thermals

Flood Over Pads in a Copper Plane

Options Dialog Box, Dimensioning Category, Alignment and Arrows Subcategory

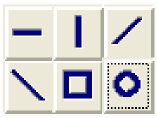
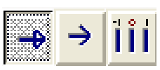
To access: **Tools > Options** menu item >**Dimensioning** category > **Alignment and Arrows** subcategory

Use the **Alignment and Arrows** subcategory to specify the appearance of dimensioning alignment tools and arrows. Options in this subcategory affect the appearance of text, lines, and so on, used to show clearances.



Objects

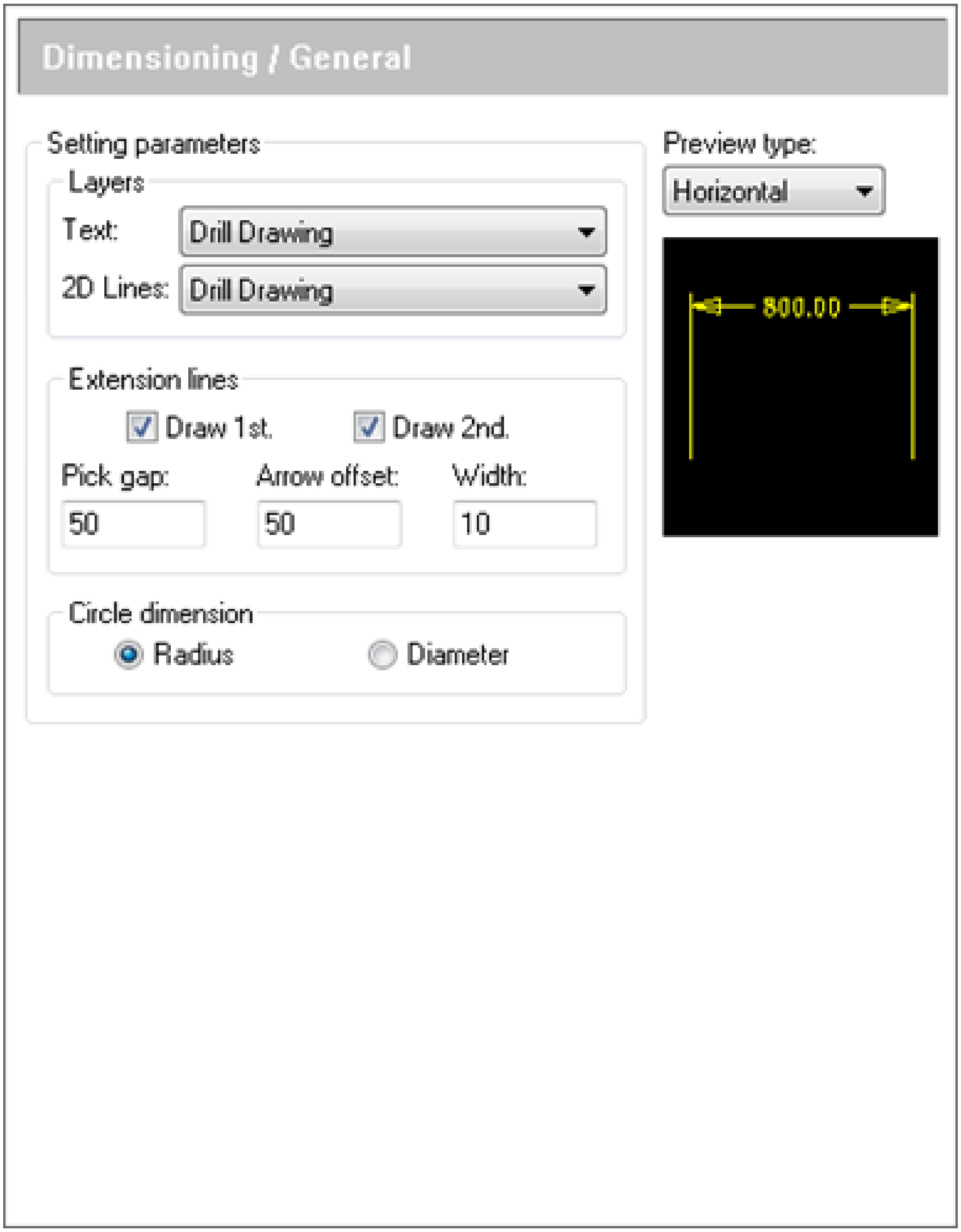
Field	Description
Preview type	Specifies the way the image is shown in the Preview area. You have the choice of:

Field	Description
	• Horizontal • Vertical • Aligned • Angular • Circular
Alignment tool area	
	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 in screen pixels.
Line Width	Specifies the alignment tool line width in screen pixels.
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 Category, General Subcategory

To access: **Tools > Options** menu item >**Dimensioning** category **General** subcategory

Use the **General** subcategory to specify the layers used to display dimensioning text and lines, the appearance of extension lines, and how to measure circles. Options in this subcategory affect the appearance of text, lines, and so on, used to show clearances.



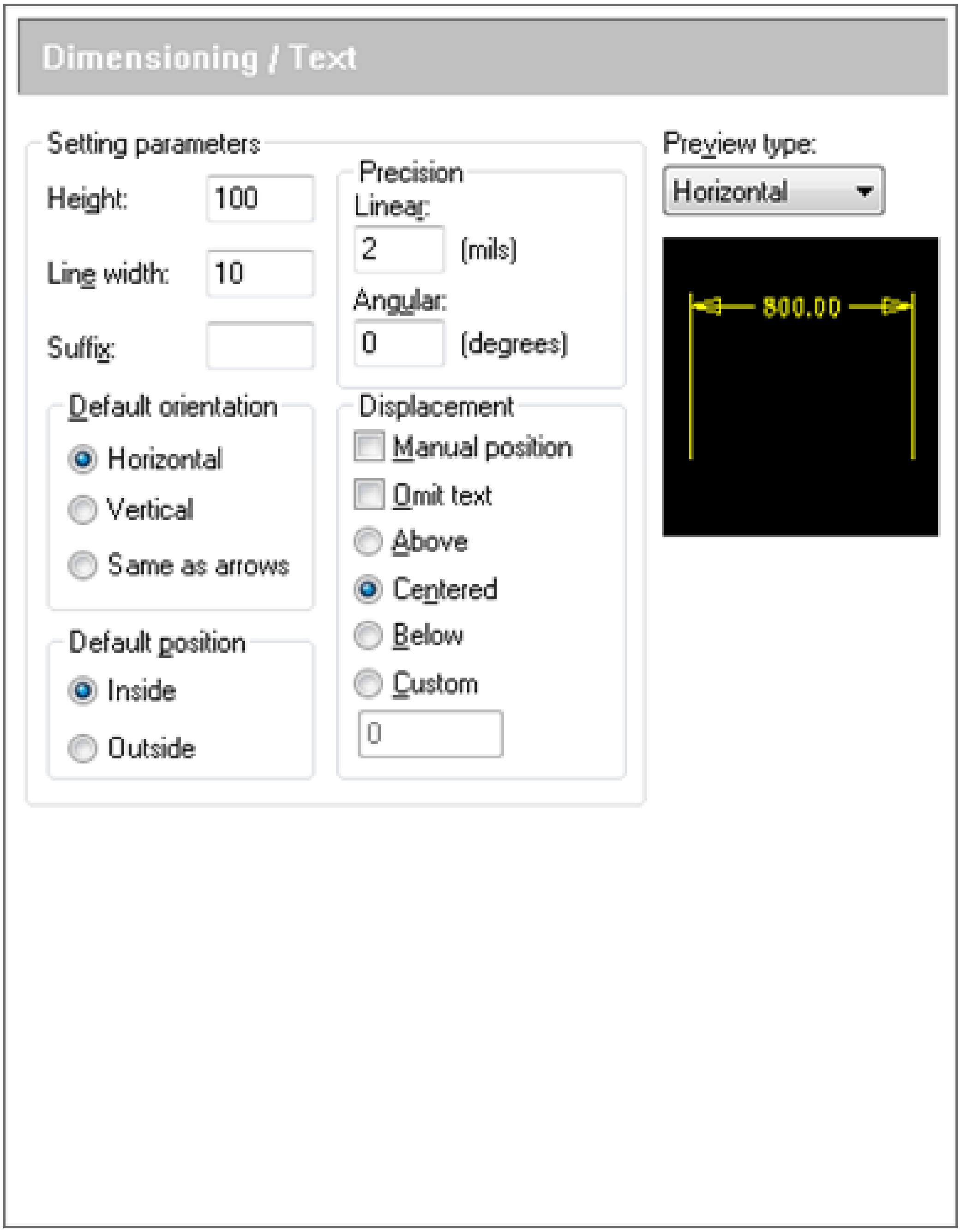
Objects

Field	Description
Preview type	Specifies the way the image is shown in the Preview area. You have the choice of: • Horizontal • Vertical • Aligned • Angular • Circular
Layers area	Specifies the layers to use for dimensioning text and lines. **Tip** • Dimensioning items ignore the current layer set in the Layer list on the Standard 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. All numerical values are in current design units.
Arrow offset	Specifies the overhang of the extension line beyond the arrowhead. All numerical values are in current design units.
Width	Specifies the width of the extension line. All numerical values are in current design units.
Circle dimension area	Specifies the circle measurement method you want.

Options Dialog Box, Dimensioning Category, Text Subcategory

To access: **Tools > Options** menu item >**Dimensioning** category **Text** subcategory

Use this subcategory to specify the appearance of dimensioning text that denotes clearances in your design.



Objects

Field	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.
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:

Field	Description
	• **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. 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 Category

To access: **Tools > Options** menu item > **Display** category

Use this category to set the text size of displayed net names and pin numbers, and the maximum allowable gap between net names on traces.

Objects

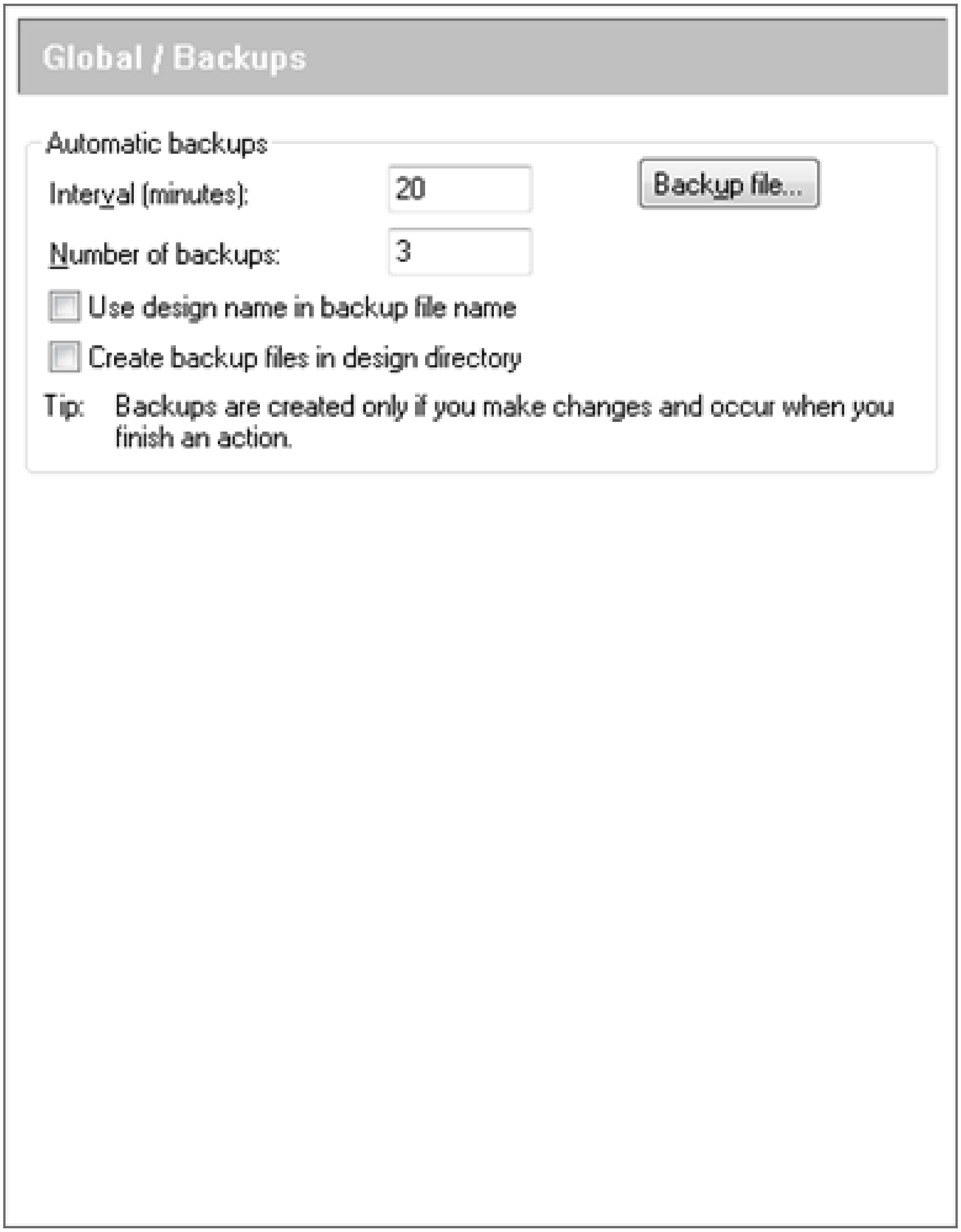
Field	Description
Minimum	Specifies the minimum size of displayed net names and pin numbers, in pixels.
Maximum	Specifies the maximum size of displayed net names and pin numbers, in pixels.
Maximum gap between net names on traces, pixels	Specifies the maximum allowable gap between displayed net names 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, Global Category, Backups Subcategory

To access: **Tools > Options** menu item > **Global** category > **Backups** subcategory

Use this subcategory to specify design data backup options.



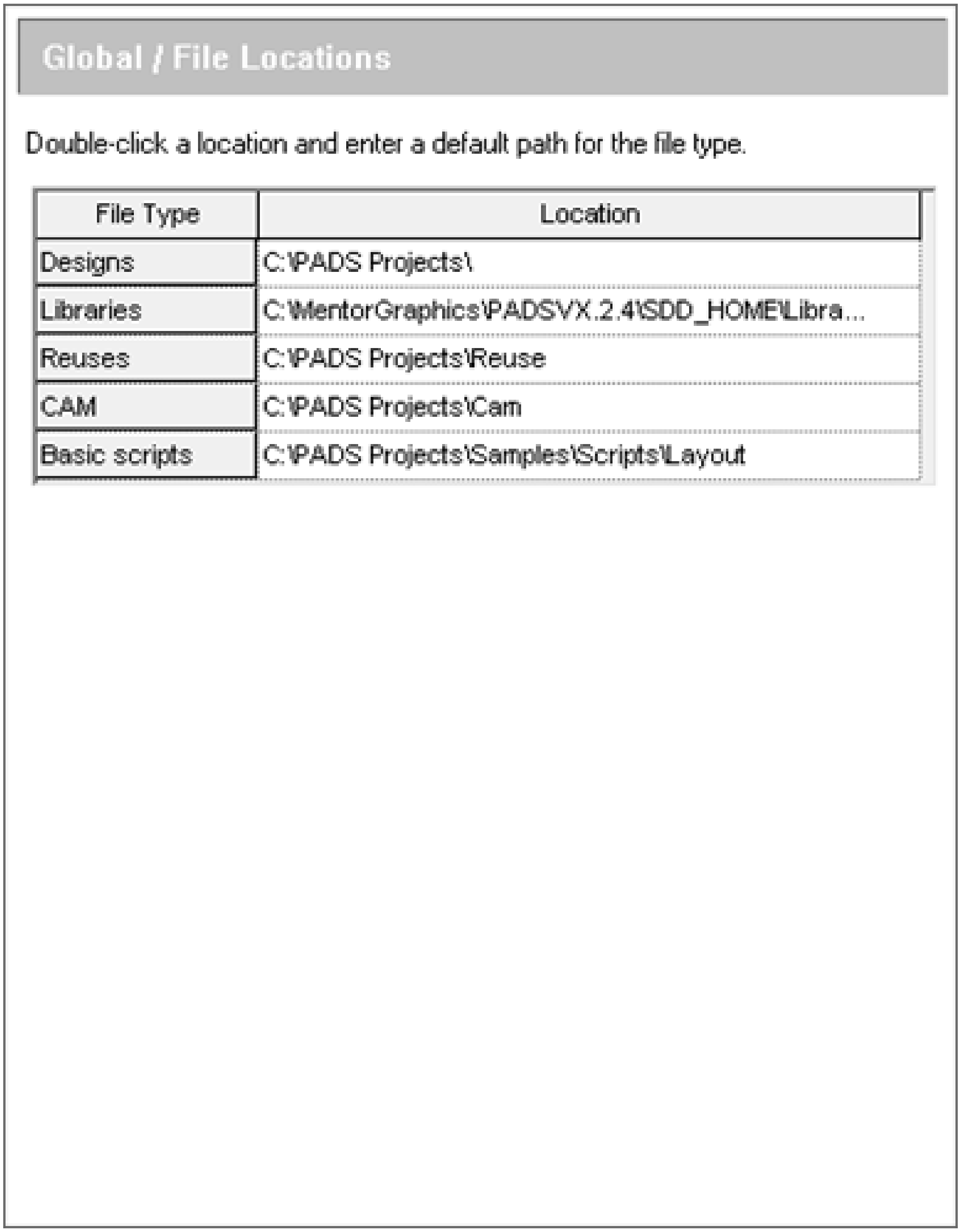
Objects

Field	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.
Backup file	Opens the Backup File dialog box where you can change to folder or name of the backup file. By default, backup files are named <design_name>.#, where # is a sequential number. For example, Layout1.pcb, Layout2.pcb, and so on.
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 Category, File Locations Subcategory

To access: **Tools > Options** menu item > **Global** category > **File Locations** subcategory

Use this subcategory to specify default design file location options.



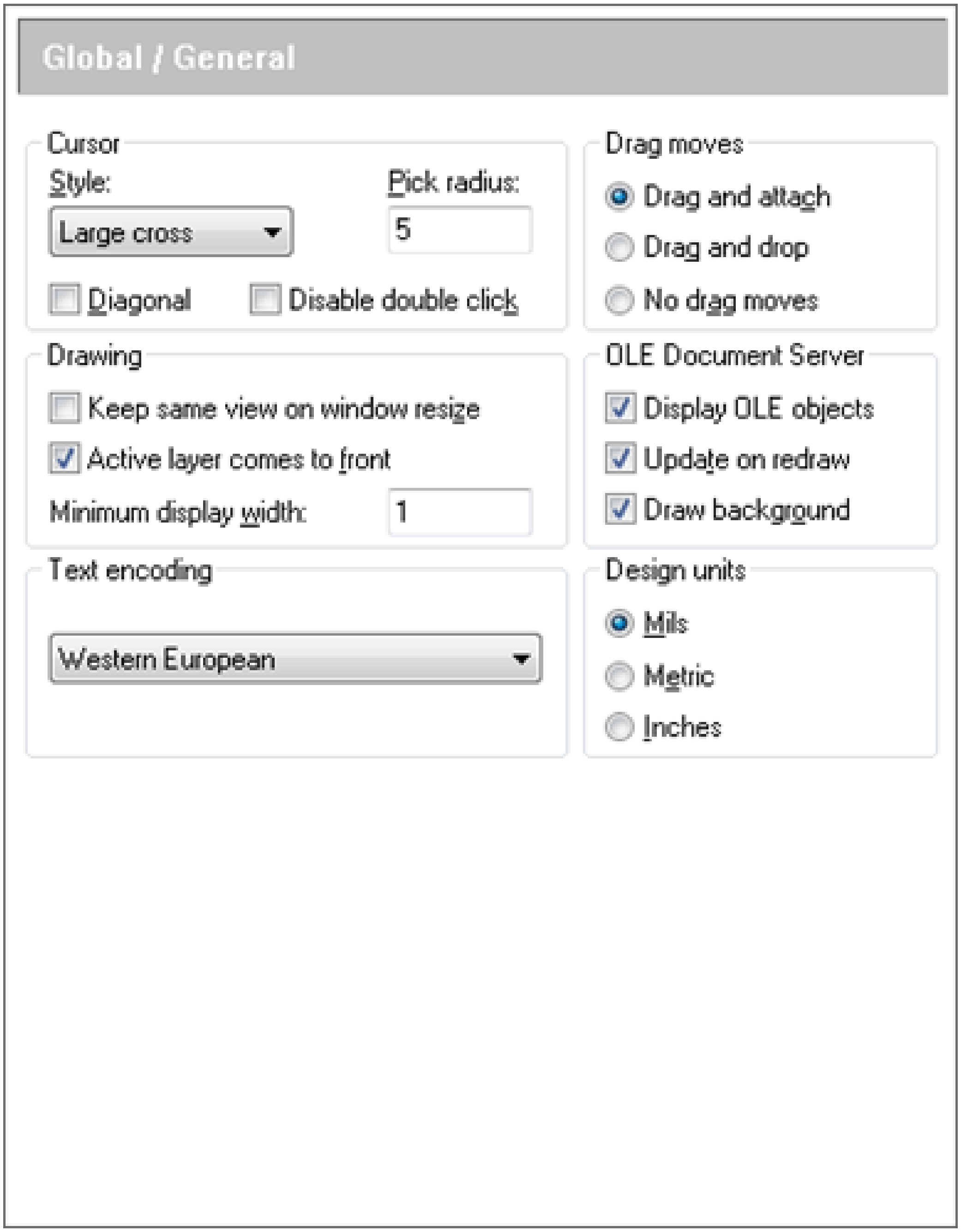
Objects

Field	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 Category, General Subcategory

To access: **Tools > Options** menu item > **Global** category > **General** subcategory

Use this subcategory to specify various workspace and design unit options.



Objects

Field	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. 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 +. **Restriction:** 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. **Tip** • 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: Right-click, click the **Move** popup menu item, move the object to the new location, and click to finish the move.
Drawing area	
Keep same view on window resize	Specifies to maintain the area view of the design when you resize the SailWind Layout window, by automatically zooming in or out.

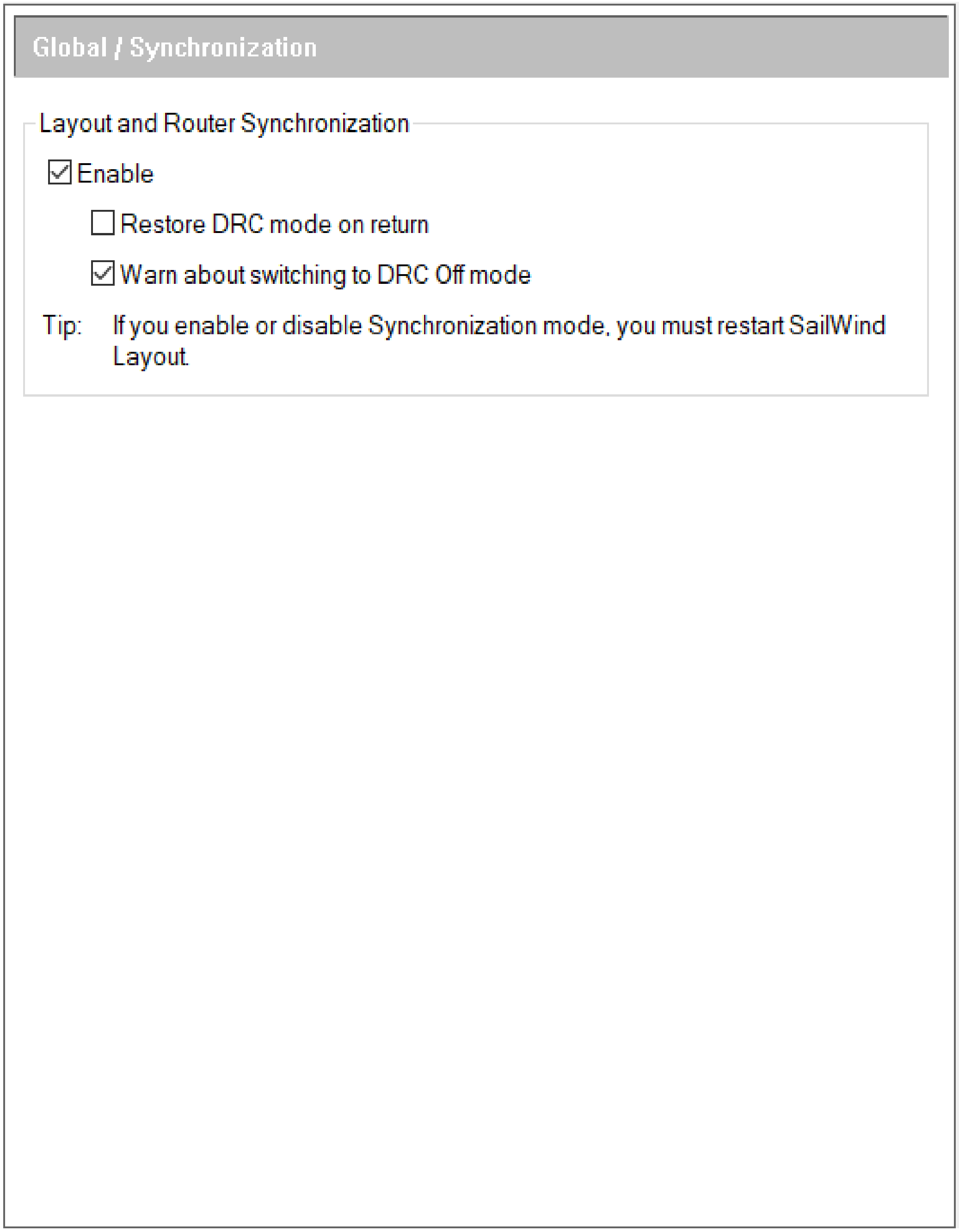
Field	Description
Active layer comes to front	Specifies to display the active layer on top of all other layers. **Tip** Specify the active layer in the Layers list on the Standard Toolbar.
Minimum display width	Specifies the minimum width in current design units of lines you want to draw at actual width. Lines smaller than this width are drawn only as center lines to save memory and redraw time. **Tip** • 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 SailWind Layout. You may want to disable this option to decrease redraw times when SailWind 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 SailWind 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 SailWind Layout background color in the linked or embedded object. When this option is disabled, the background of the SailWind Layout object is transparent and you can see through the object at the container application's background.
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

Field	Description
	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 Category Synchronization Subcategory

To access: **Tools > Options** menu item > **Global** category > **Synchronization** subcategory

Use this subcategory to specify the SailWind Layout and SailWind Router synchronization options



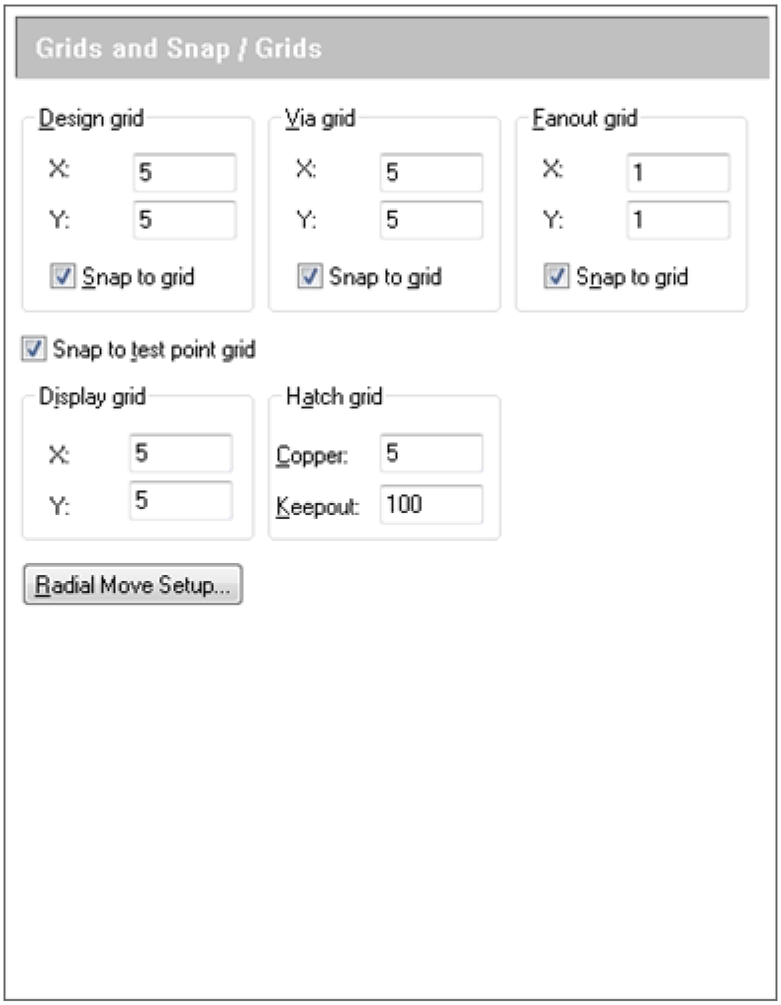
Objects

Field	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. **Note:** **Warning:** You must restart Layout for the change in Synchronization mode to take effect.
Restore DRC mode on return	Specifies, if you return to SailWind Layout from SailWind Router after switching, to change the DRC mode back to what you had before you switched: DRC Prevent, Warn, or Ignore. **Tip** • Switching to SailWind Router automatically places your design in SailWind Layout in DRC Off mode; the DRC mode in SailWind 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 SailWind Layout into DRC Off mode.

Options Dialog Box, Grids and Snap Category, Grids Subcategory

To access: **Tools > Options** menu item > **Grids and Snap** category > Grids subcategory

Use this subcategory to specify grid options.



Objects

Field	Description
Design grid area	Establishes the minimum snap distance for the pointer when editing (placing objects, drawing/editing drafting objects). The current Design grid settings appear to the right of the Status bar but when you move an object or use a drafting command, the grid readout is replaced by a Delta X and Y reading, calculated from the cursor selection point when the command starts. Negative numbers indicate left and down.

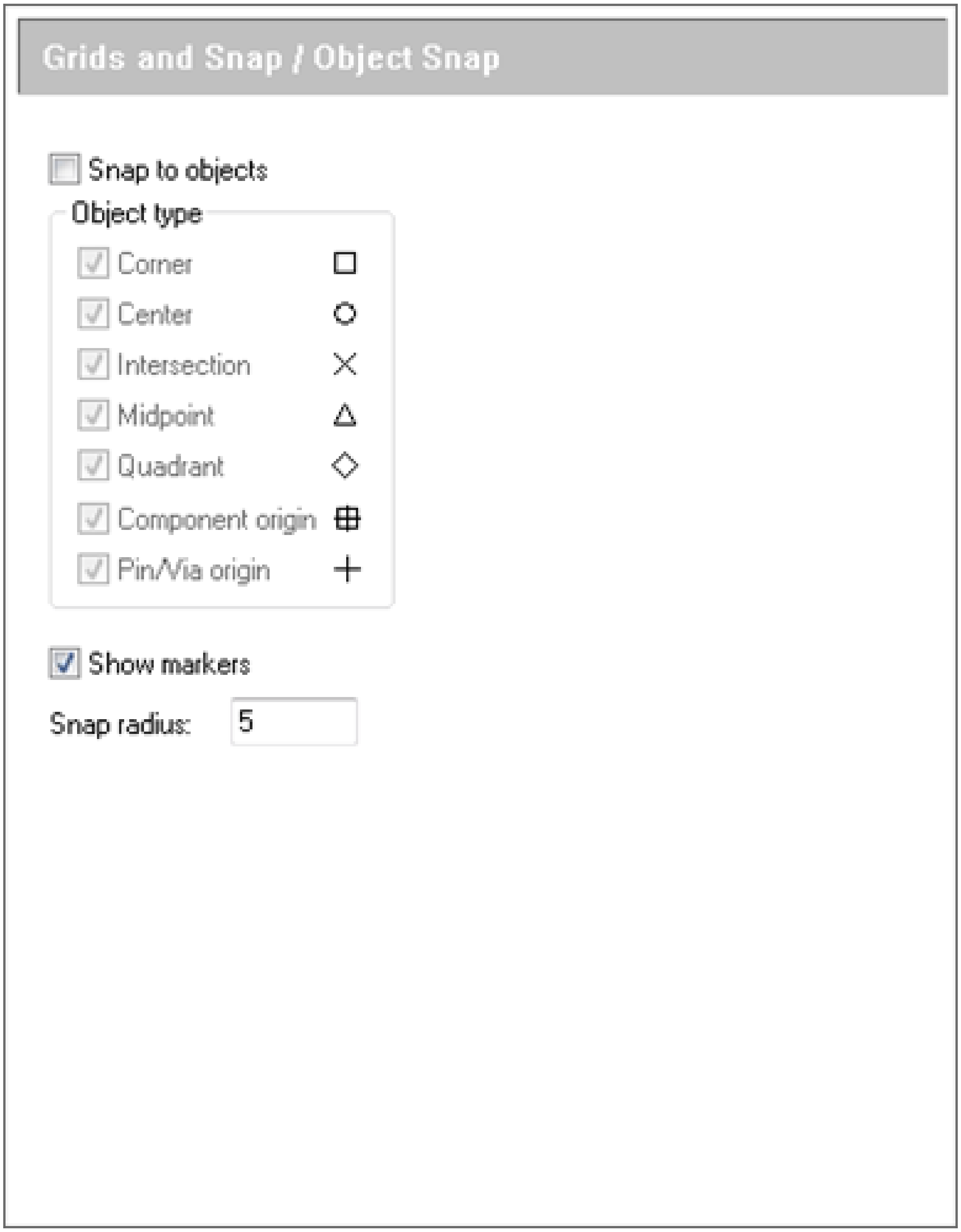
Field	Description
	The design grid is based on the origin. If you move a part across the board during design, the cursor may move smoothly but with Snap to Grid enabled, the part snaps from grid point to grid point and you cannot place a part off the design grid. • **X** — Specifies the distance between 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. • **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. If the Snap to Grid check box is selected, you cannot place a part off the grid. See also: “Setting the Design Grid”.
Via grid area	Controls the initial placement of vias. If you move a via after initial placement, the Design grid is used. • **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. 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. **Restriction:** This data is not used within SailWind Layout but is only passed to SailWind 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 SailWind Router.
Display grid area	Specifies the visible dot grid - a field of white dots that is a valuable drafting aid. Set the display grid to either match the Design grid or at larger multiples of the Design 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.

Field	Description
	Tip If you want to make the dot grid invisible, set the X and Y values to 0. See also: "Setting the Display Grid".
Hatch grid area	Specifies the distance between hatch lines of copper and keepout areas. The hatch grid, when combined with the line width, is used to create varying degrees of hatch through to a solid pattern. For more information, see "The Fill of Copper, and Copper Planes" on page 617. • **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.
Radial Move Setup	Opens the Radial Move Setup Dialog Box to specify the polar grid for radial moves. See also: "Setting Up a Polar Grid".

Options Dialog Box, Grids and Snap Category, Object Snap Subcategory

To access: **Tools > Options** menu item > **Grids and Snap** category > **Object Snap** subcategory

Use this subcategory to specify options for “snapping to” various objects when creating or modifying drafting shapes, placing components, or using the Q modeless command (Quick Measure).



Objects

Field	Description
Snap to objects	Snap to the specified object type(s) instead of to the grid. **Tip** You can use the OS modeless command on page 83 to toggle this setting.
Object type	Indicates the object type(s) to snap to: • **Corner** — corner of a drafting shape or line end. • **Center** — center of a circle or an arc • **Intersection** — cross point of two lines or arcs • **Midpoint** — midpoint of a line or arc • **Quadrant** — four orthogonal points on a circle or arc • **Component origin** • **Pin/via origin** You can use the OS <n> Modeless Commands to set the object types.
Show markers	Specifies to show the object type markers in your design as your pointer enters the Snap radius: • **Corner** — □ • **Center** — ○ • **Intersection** — × • **Midpoint** — △ • **Quadrant** — ◇ • **Component origin** — ⊞ • **Pin/via origin** — +
Snap radius	Specifies the radius of the “searching area” around the cursor in which snapping points are activated. Uses current Design units on page 1531. You can use the OSR modeless command on page 83 to specify this value.

Related Topics

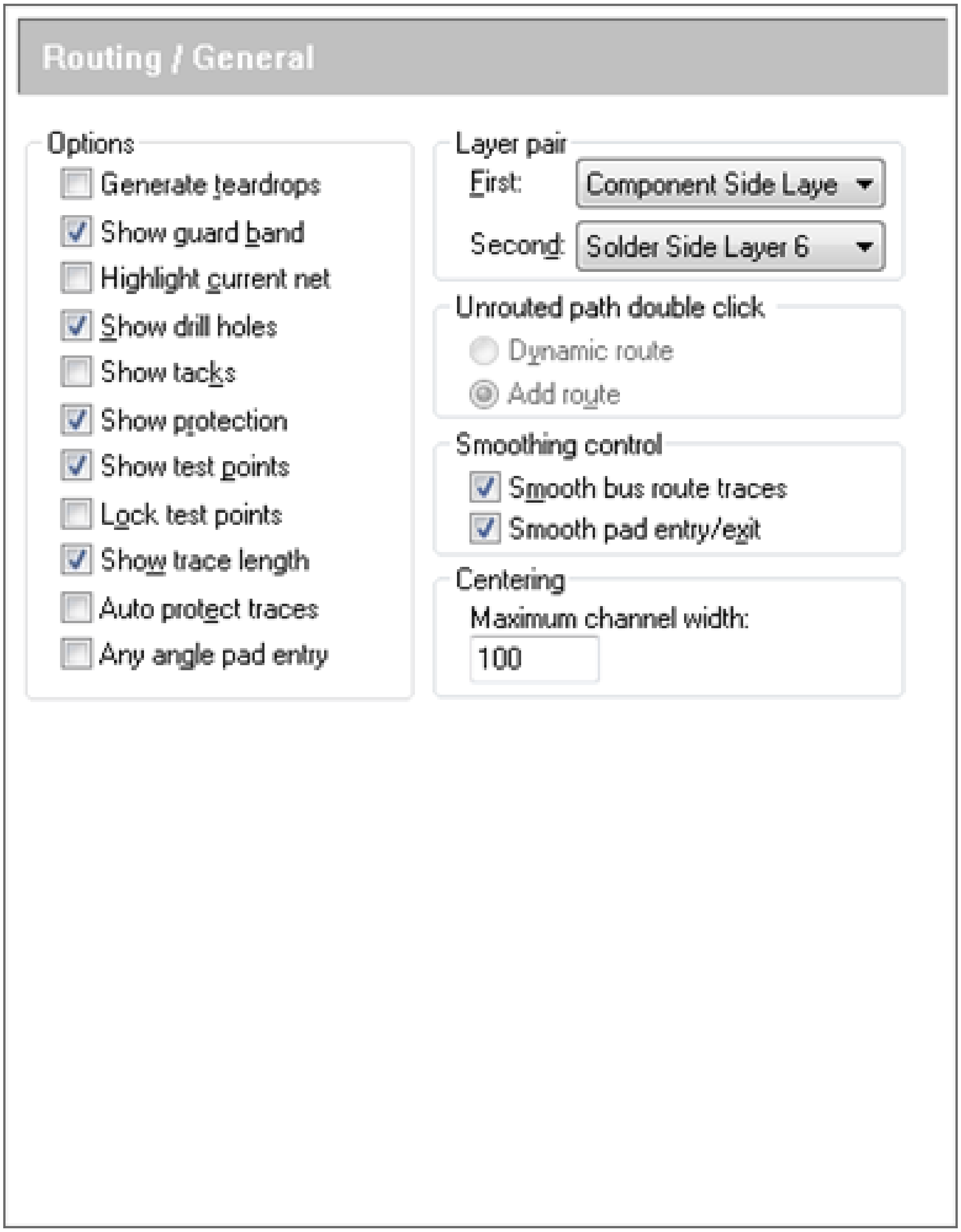
Object Snap

Set Values Before Creating a Drafting Object

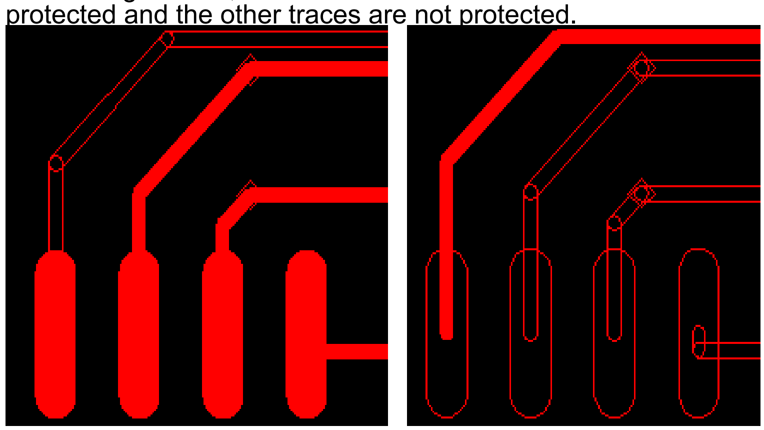
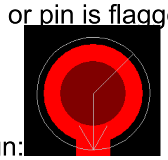
Options Dialog Box, Routing Category, General Subcategory

To access: **Tools > Options** menu item > **Routing** category > **General** subcategory

Use this subcategory to specify routing options.



Objects

Field	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.
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. When the via or pin is flagged as a test point, an arrow is drawn on it in the design:

Field	Description
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
Show trace length	Specifies to display the trace length monitor at the pointer. The status bar always reports trace length. **Tip** • You can also use the shortcut key combination 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. 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: "The Layer Pair", "Layer Changes While Routing". **Tip** • 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.

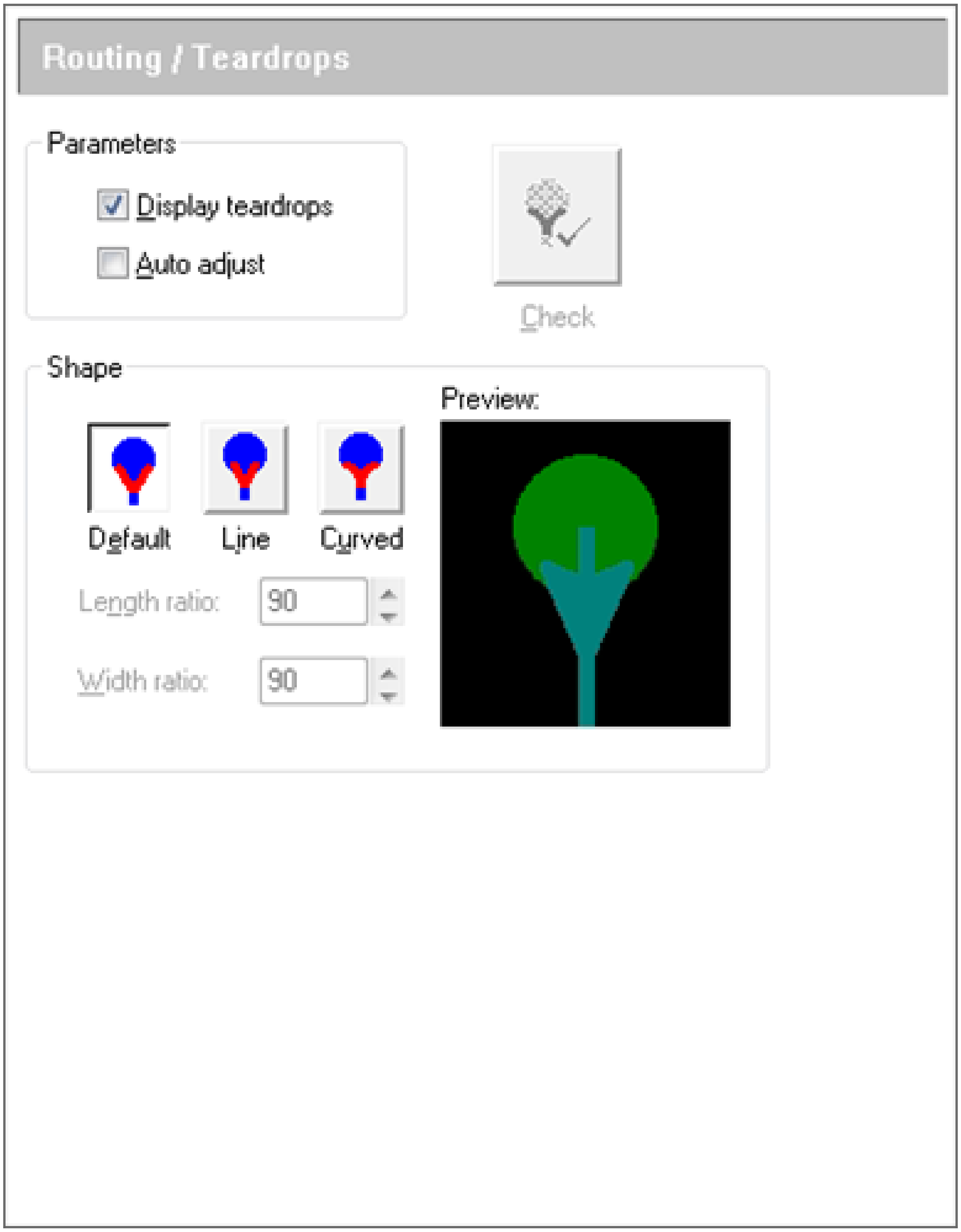
Field	Description
Smoothing control area	
Smooth bus route traces	Specifies to run smoothing after bus routing. This option affects only the Bus Route command, and it inhibits the global smoothing pass for all traces of the current bus.
Smooth pad entry/exit	Specifies to have traces that enter a pad 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. **Restriction:** This option is used only by SailWind Router.

Options Dialog Box, Routing Category, Teardrops Subcategory

To access: **Tools > Options** menu item > **Routing** category > **Teardrops** subcategory

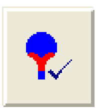
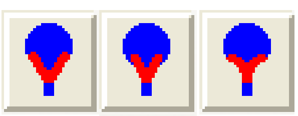
Use this subcategory to specify the display and physical appearance of teardrops on newly routed traces.

Tip
Use the **Routing** category > General subcategory on page 1542 to enable teardrops.



Objects

Field	Description
Parameters area	
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.

Field	Description
	Tip You create the teardrops using the Generate teardrops check box on the **Routing** category > General subcategory on page 1542. **Note:** **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 do not 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 SailWind 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 — these settings only apply to traces routed after the Generate Teardrops check box is selected in the **Routing** category > **General** subcategory. They do not apply to teardrops that are generated on existing trace segments connected to pads and vias.	
	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.
Length ratio	Specifies the length ratio in percent of pad diameter. **Restriction:** • You cannot set a ratio over 1000. • Not available for the Default shape.

<table>
<tr><th>Field</th><th>Description</th></tr>
<tr><td></td><td>Tip
This option sets the length of the teardrop relative to the attached pad. The formula used to calculate the teardrop length follows:
<code>teardrop length = (pad diameter) * (length ratio in %)</code>
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.</td></tr>
<tr><td>Width ratio</td><td>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.</td></tr>
</table>

Options Dialog Box, Routing Category, Tune and Diff Pairs Subcategory

To access: **Tools > Options** menu item > **Routing** category > **Tune/Diff Pairs** subcategory

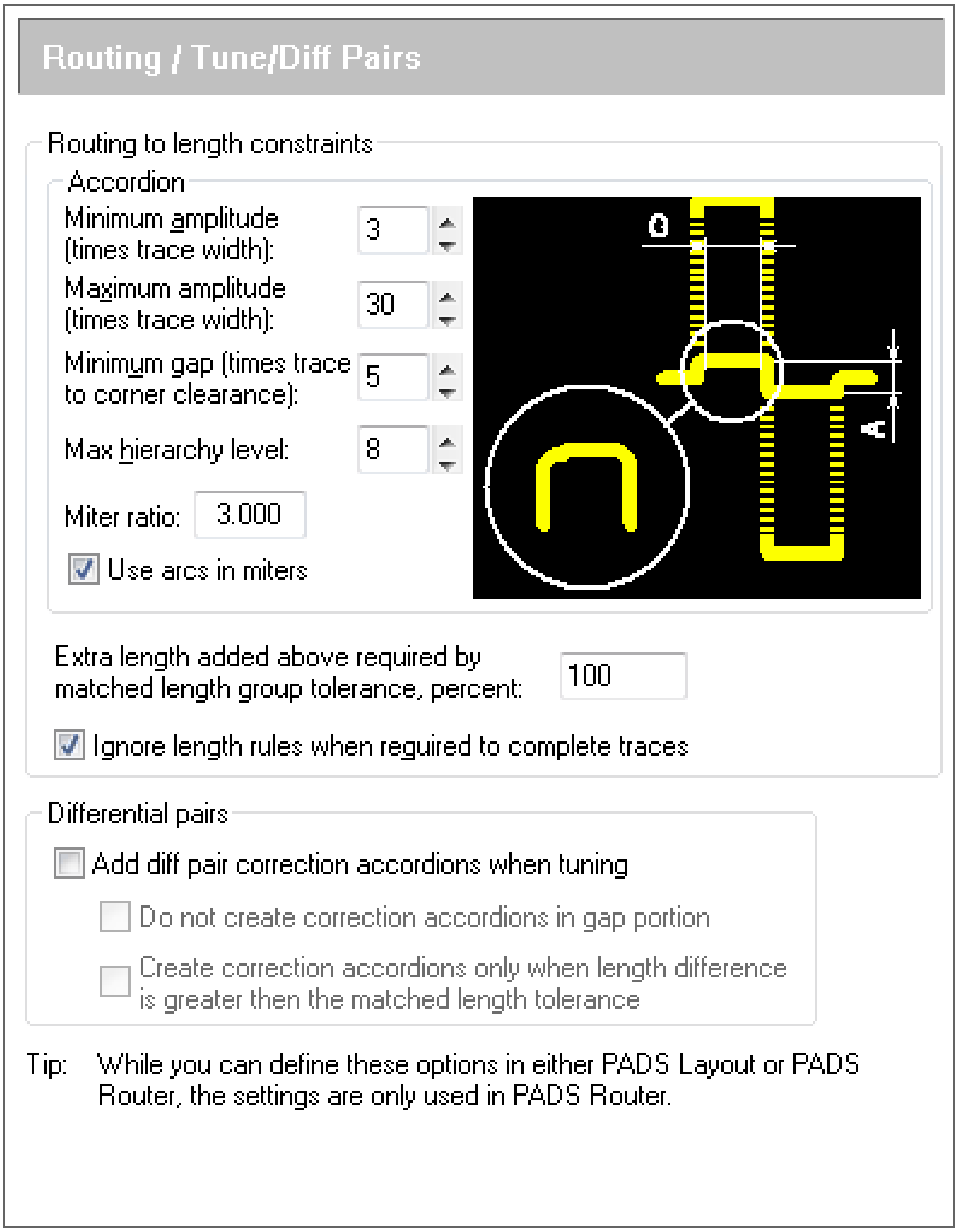
Use this subcategory to specify options for using accordions in routing to length constraints and tuning differential pairs.

Restriction:
While you can define these options in either SailWind Layout or SailWind Router, the settings are only used in SailWind Router.

Description

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 (SailWind Router only) Matched Length Properties dialog boxes.



Objects

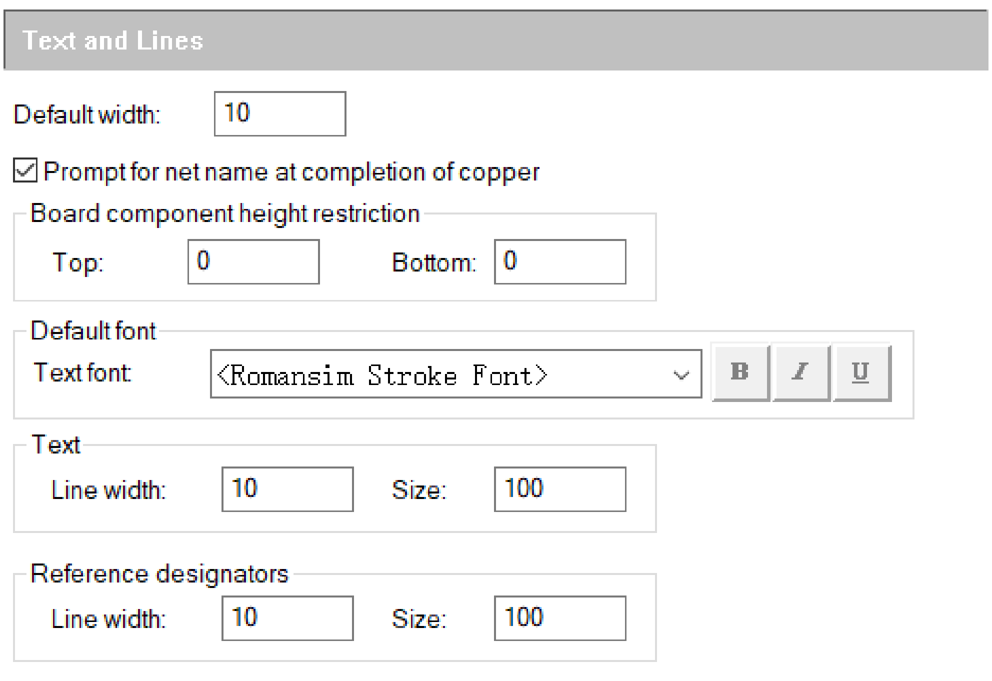
Field	Description
Accordion area	
Minimum amplitude	Specifies the minimum height (for horizontal accordions) or width (for vertical accordions).

Field	Description
Maximum amplitude	Specifies the maximum height (for horizontal accordions) or width (for vertical accordions).
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” in theTune Pass
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 greater 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, Text and Lines Category

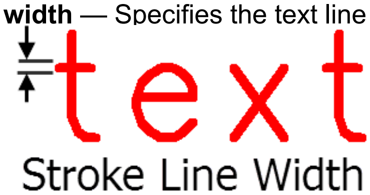
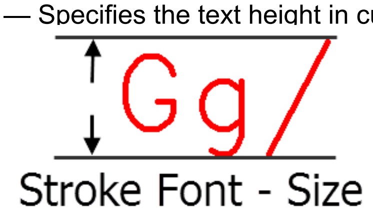
To access: Choose the **Tools > Options** menu item > **Text and Lines** category

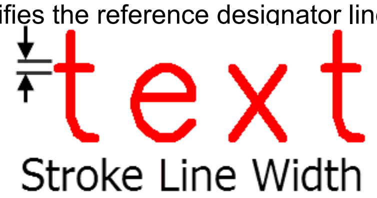
Use this category to view and edit options for drafting objects.



Objects

Field	Description
Default width	Specifies the default width to use when adding drafting objects. **Tip** • To change the width of an existing shape, select the shape, right-click, and click the **Properties** popup menu item. • To search for and change all shapes of a similar width, on the Edit menu, click **Find**.
Prompt for net name at completion of copper	Specifies whether you are prompted to assign a net to the new copper or copper plane. 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.

Field	Description
	Sets the maximum height for all design components that use the Geometry.Height attribute. See also: "Restricting Heights on Component Layers." **Tip** • 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"
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. **Tip** • 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.
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.
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.

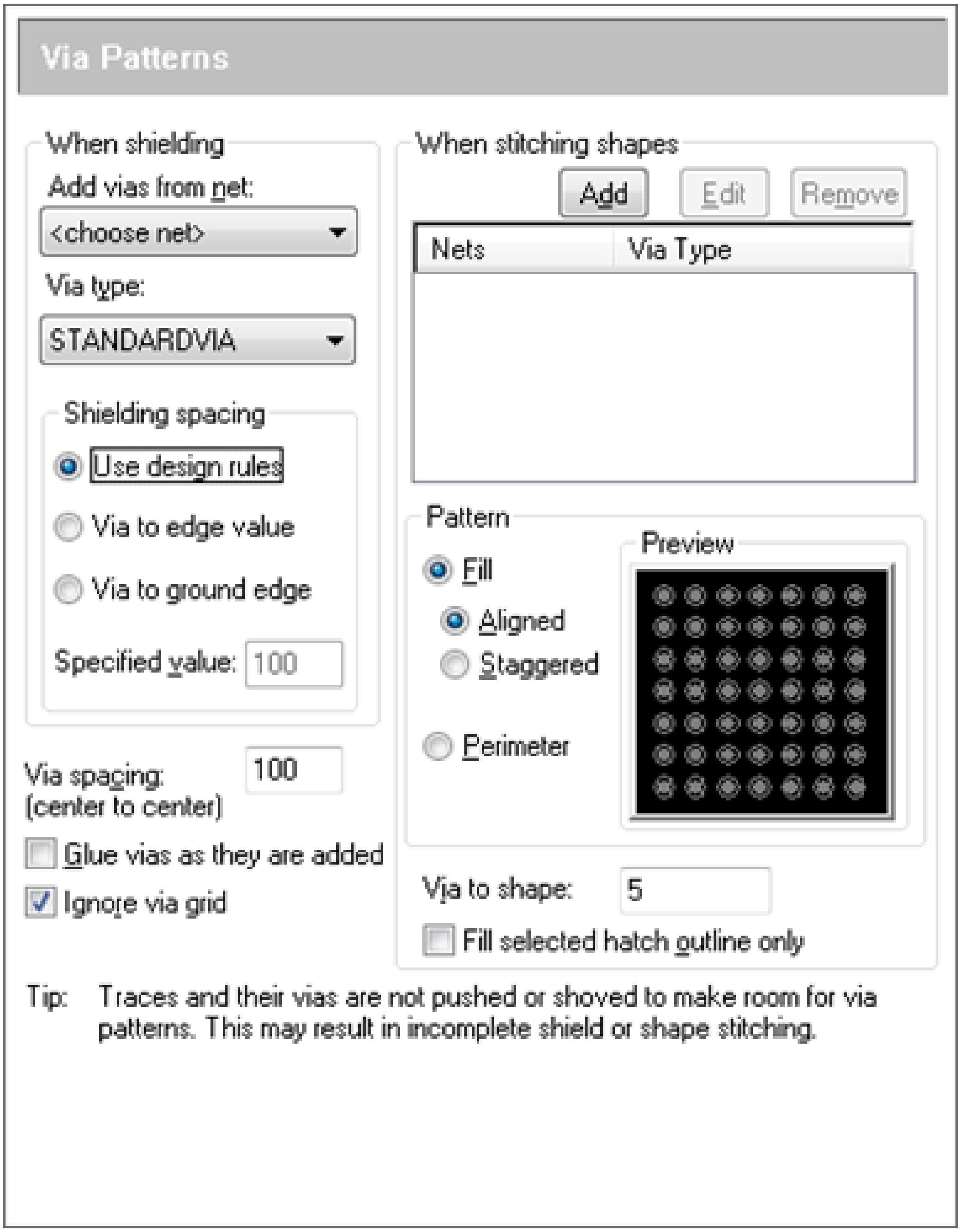
Field	Description
	• **Line width** — Specifies the reference designator line width in current design units.  **Restriction:** Reference designator width is stored in the library, but the pin number is not. • **Size** — Specifies the reference designator height in current design units.  **Tip** If you 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 the **Properties** popup menu item.

Options Dialog Box, Via Patterns Category

To access: **Tools > Options** menu item > **Via Patterns** category

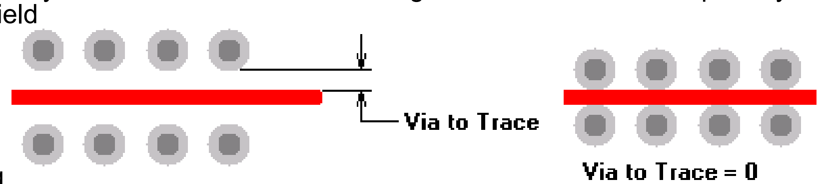
Use this category of the Options dialog box to set options for the via shielding and via stitching operations. The Add Via Shield and Via Stitch operations do not move traces in order to make room for the vias. For best results, you should add via shielding or via stitching early in design layout when your design has fewer traces.

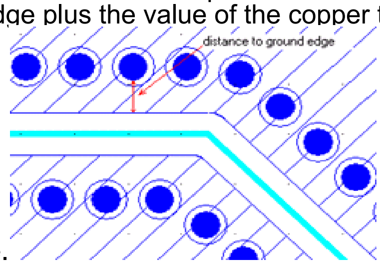
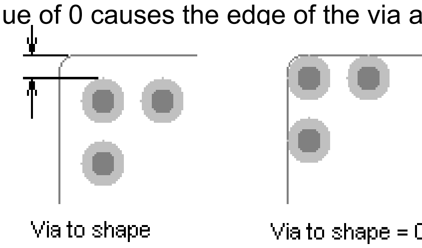
Tip
You must have a design open in order to set Via Patterns options and set the Via Patterns options before using the Add Via Shield or Via Stitch commands.

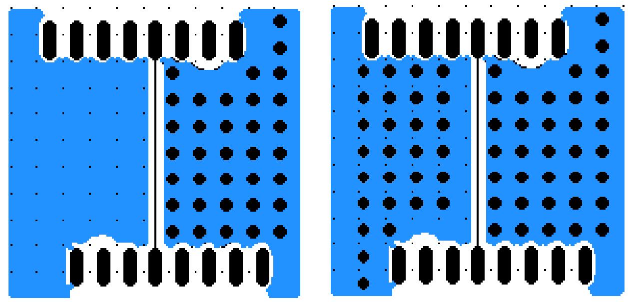
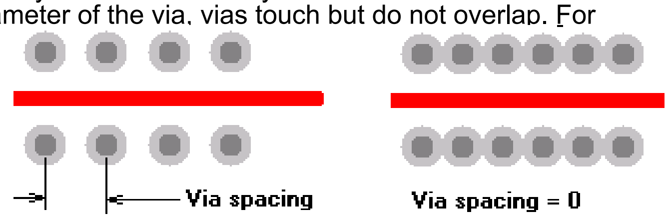


Objects

Name	Description
When shielding area — Use these options to specify the via type and spacing of vias used for shielding.	
Add vias from net	Specify the net assigned to the vias used for shielding. This also filters the Via type list to only display the vias selected in the Routing rules on page 1666 for use with the net you have chosen.

Name	Description
Via type	Specify the via type of vias used for shielding. If you need to create a new via for shielding, see “Via Setup”. **Note:** **Recommendation:** First, select the net to assign to the shielding vias in the Add vias from net list in order to choose a via that is available for use with the net. The Routing rules on page 1666 determine which via types are available for the selected net.
Shielding spacing area	Specify the distance from the shielding vias to the trace or shape they are shield  ing **Note:** **Recommendation:** Select the Ignore via grid check box. Otherwise, vias snap to the via grid. Select one of the following: • **Use design rules** — 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. **Exception:** Although design rule values are used to place vias in relation to the selected trace or copper, that placement can violate clearances with other objects—another trace, for example. To prevent addition of any via that causes clearance errors, set On-line DRC (Design Rule Checking) 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.

Name	Description
	• **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. 
When Stitching Shapes area — Specifies the via type used in stitching a copper shape for a given net. To prevent addition of any via that causes clearance errors, set On-line DRC (Design Rule Checking) to Prevent Errors. For more information, see "DRC and the Via Stitching and Shielding Operations".	
Nets/Via Type list	Specify the via type used per net for stitching shapes. By default, this list is empty. The Routing rules on page 1666 determine which via types are available for the selected net. If you need to create a new via for shielding, see "Via Setup". 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. 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, 0 to 1 inch, or 0 to 25.4 mm. 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.

Name	Description
Fill selected hatch outline only	Use this setting when you have a single copper 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 plane outline of which the selected hatch outline is a part.  Option selected Option cleared For example:
Use the settings below for both the Add Via Shield and Via Stitch commands.	
Via spacing	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, 0 to 1 inch, or 0 to 25.4 mm. 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:.  **Tip** • 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.)
Glue vias as they are added	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.
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 Shielding spacing area and the Via spacing field.

Related Topics

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.

To access: 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 open or close the Output window.

The Output window has two tabs:

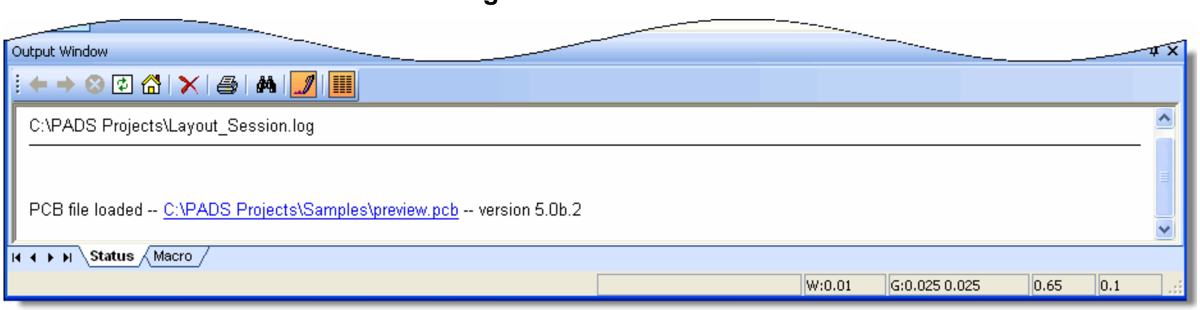
- Status Tab - Displays information on the current session.
- Macro Tab - Allows you to run, edit and debug macro scripts.

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).

To access: Click the **Output Window** button and then click the **Status** tab.

Figure 190. Status Tab



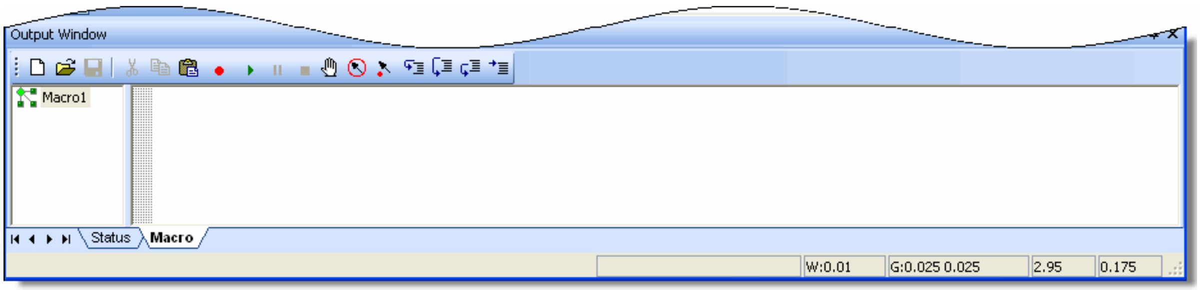
Macro Tab

You can edit, run, and debug macro scripts in the **Macro** tab. You can open multiple macros and nest macros using the macro editor.

To access: Click the Output Window button and then click the **Macro** tab.

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.

Figure 191. Macro Tab



Related Topics

User Interface Elements

The Status Tab

Macros and the Macro Language [SailWind Layout Command Reference Manual]

Chapter 52
GUI Reference Elements P

Read the sections that follow to learn more about dialog box elements in SailWind Layout.

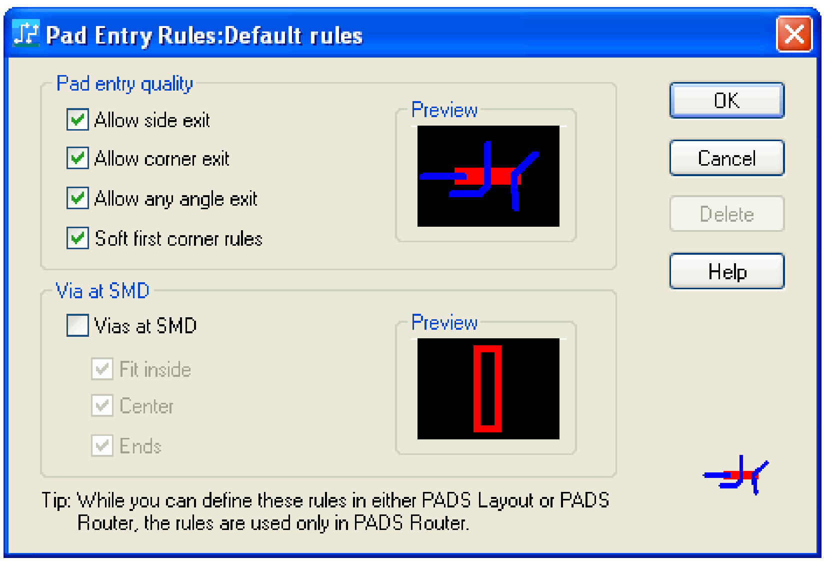
Pad Entry Rules Dialog Box

To access:

- **Setup > Design Rules** menu item > **Default** button > **Pad Entry** button

Use the Pad Entry Rules dialog box to specify how traces enter and exit pads.

Restriction:
While you can define these rules in either SailWind Layout or SailWind Router, the rules are only used in SailWind Router.



Objects

Area	Description
Pad entry quality	Specify the options that SailWind 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.

Area	Description
	• **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 SailWind Layout to locate possible acid traps. You can ignore first corner clearance rules, because it is a soft rule on page 1860.
Via at SMD	Specify to enable SailWind Router to place vias on SMD pads while routing. **Restriction:** 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. **Restriction:** 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 [SailWind Router User's Guide]

Pad Stacks Properties Dialog Box

To access: **Setup > Pad Stacks** menu item

Establish the size and shape of each pad and drilled hole in a given pad stack, including component, via, and virtual pin pad stacks.

Description

Figure 192. Pad Stacks Properties Dialog Box

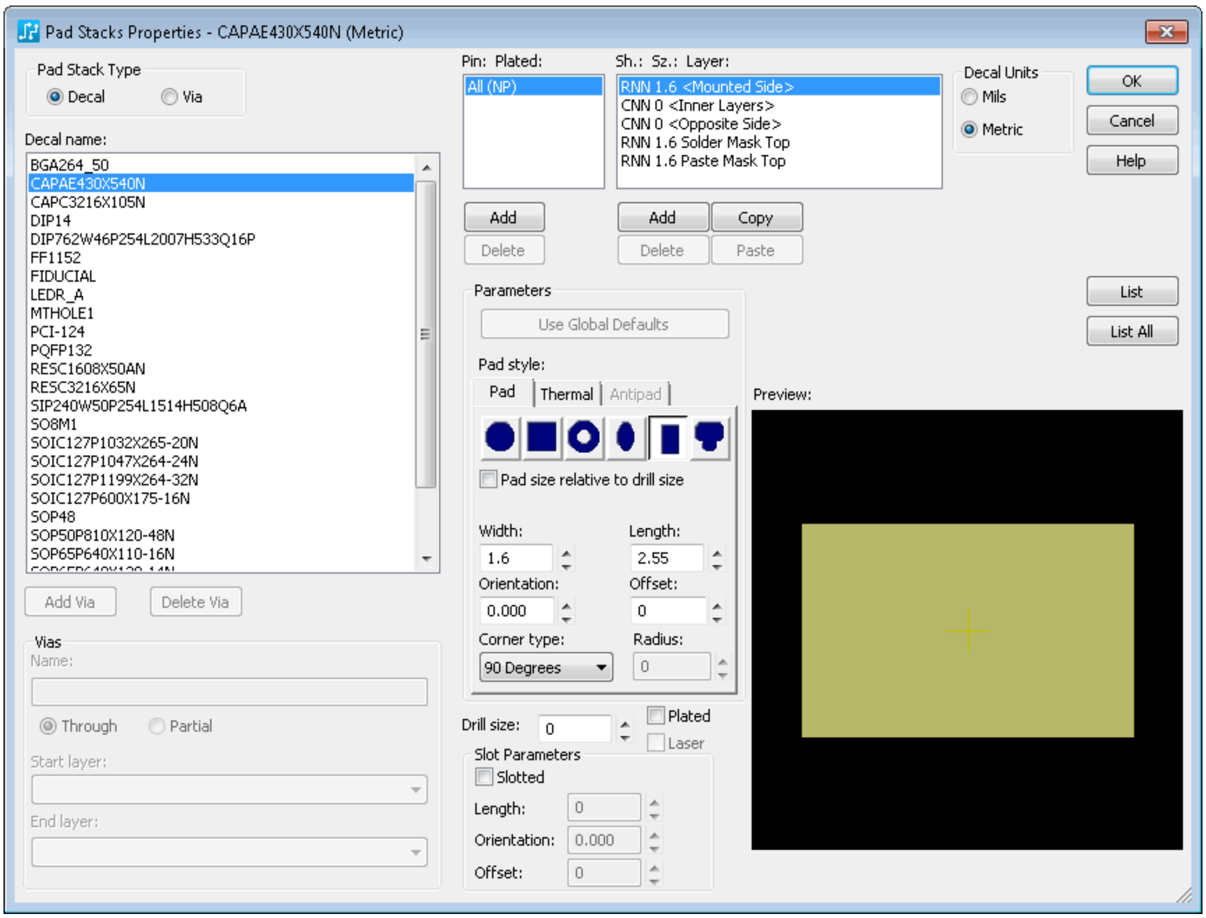


Figure 193. Pad Style Pad Tab

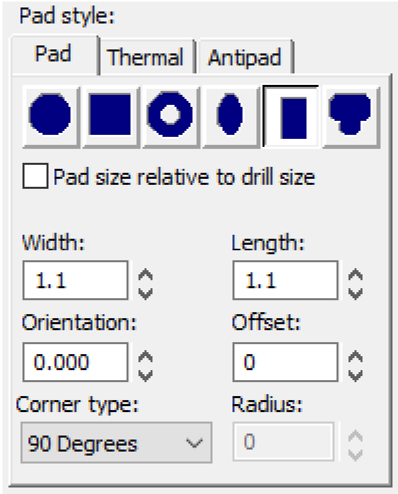


Figure 194. Pad Style Thermal Tab

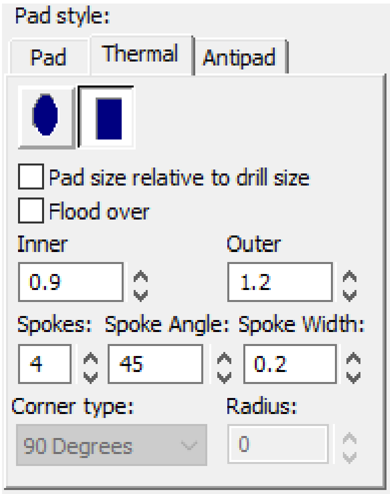
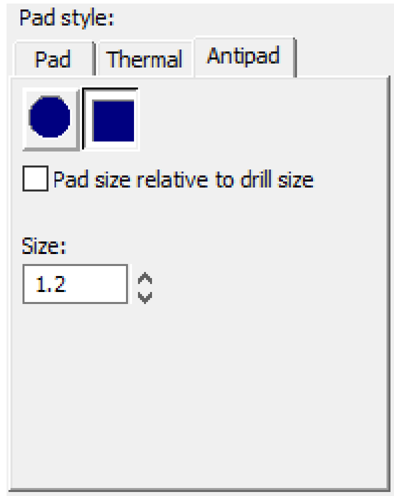


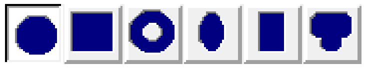
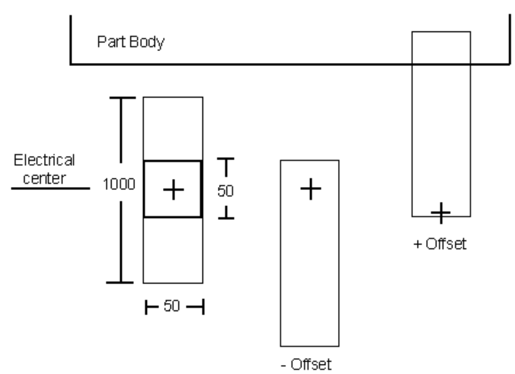
Figure 195. Pad Style Antipad Tab

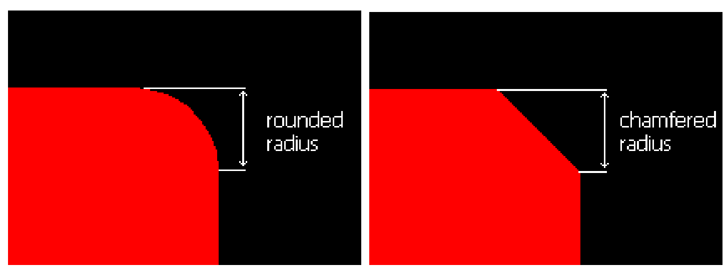


Objects

Area	Field	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 type. 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.
	Add Via	Adds a via type to the Decal Name list. Available only when Via is selected in the Pad Stack Type area. See also: "Creating a Through-hole Via Type", Creating a Partial Via Type".
	Delete Via	Deletes the selected via type from the Decal Name list. 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	This area is available only when Via is selected in the Pad Stack Type area.	
	Name	Assigns a name for a new via type.
	Through	Sets the current via type as a through via, through all layers of the board.
	Partial	Sets the current via type as a partial via, through certain layers of the board. Click Partial and click a Start Layer and an End Layer.

Area	Field	Description
	Start Layer/End Layer	Sets the start and end layer for a partial via type. Available only when Via is selected in the Pad Stack Type area, and Partial is the type of via.
Pin: Plated:	Pin number and plated status. You can select one or more pins and customize the pad stacks for your selection. The Pin, Plated setup does not apply to via pad stacks; all vias are considered plated. Available only when Decal is selected in the Pad Stack Type area.	
	Add	Opens the Add Pin Dialog Box.
	Delete	Removes the selected Pin.
Sh.: Sz.: Layer:	Shape, Size, Layer. Select one or more layers to specify the shape and size of the available pad, thermal, and/or antipad. In the shape column, the characters describe the shapes of the pad, thermal and antipad. C=circular, S=square, A=annular, O=oval, R=rectangular, D=odd. “N” indicates that none has been defined. 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 custom layers.
	Copy/Paste	Copy the shape and size information from one layer to one or multiple layers. Paste into other decals also. Via layer shape and size settings can be copied to a decal, but decal layer shape and size settings cannot be copied to a via. If antipad information is copied to a layer that does not support antipads (Mounted Side, Opposite Side) the antipad settings are ignored. Drill size value, Plated setting and Slot Parameters are not copied. See also: “Control of Solder Mask and Paste Mask”.
Parameters Specify the shape and size of the pad, thermal, and antipad for each layer in the list. Thermals and Antipads apply to plane areas on split/mixed layers and CAM negative planes (for RS-274X output). If thermals and antipads are not specified in the pad stack, they will be generated based on the design rules. For more information, see “Design Rule Versus Pad Stack - Thermals and Antipads” on page 797.		
	Use Global Defaults	When the Thermal or Antipad tab is active, sets thermal or antipad shapes to those specified in the Options dialog box > Copper Planes category > Thermals subcategory on page 1514. See also: “Design Rule Versus Pad Stack - Thermals and Antipads” on page 797.
	Pad tab Assign a pad shape to the pad style for the selected layers in the Sh: Sz: Layers: list. If a layer in the selection already has an assigned shape it will be reassigned if you click one of the shapes. You can assign pad shapes as round, square, annular, oval, rectangular or odd.	

Area	Field	Description
		
	Annular pad shape	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 pad shape	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 “Creation of Custom Pad Shapes” for more information.
	Orientation/Offset	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 196. Offset Example**  Maximum offset = ½ length or (1000/2) = 500
	Thermal tab	The size, shape and orientation of thermals are inherited from the normal pad. Prior to version 9.2, the size, shape and

Area	Field	Description
		orientation of thermals for slotted pads were derived from the length, drill size and orientation of the slotted hole. You can also 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 Options dialog box > Copper Planes category > Thermals subcategory on page 1514, 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. • **Flood over check box** — 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. Available only when a pad shape button is selected.
	Antipad	The length of an antipad with a non-zero drill size equals the slot length minus the drill size, plus the width. 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.
	Pad size relative to drill size	Displays inner and outer pad sizes relative to the drill size.
	Corner type	Available for only Square and Rectangular shapes. The figure below illustrates the rounded and chamfered corner types. 
	Radius	Available only when Chamfered or Rounded corners are selected as a corner type.
	Drill Size	Specifies the finished hole size. If the hole is plated, this value remains the same. 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 Category.
	Plated	Sets whether the drill hole is plated with copper. Normally, pad stacks with a hole are plated. To create a non-plated hole with no copper, such as a mounting hole, clear this Plated check box. Non-plated holes are drilled to true drill diameter, without oversize, and any drill oversize does not apply to them. Tips:

Area	Field	Description
		• The batch clearance checking functions consider the added (plated hole) drill and laser drill oversize value when flagging errors. The oversize settings are found in the Options Dialog Box, Design Category. • If the board is marked as Single-sided in the Layers Setup Dialog Box, this check box is ignored.
	Laser	Sets the partial via as a laser drilled via to use the separate “Laser drill oversize” setting in the Options Dialog Box, Design Category. Available only when creating the pad stack of a partial via. For more information, see “Creating a Partial Via Type” on page 330.
Slot Parameters	When slotting Round pads, the Diameter value becomes the Width. When slotting Square pads, the Size value becomes the width.	
	Slotted check box	Enables a slotted hole for the selected pad stack or pin. You must select this check box to activate the Length, Orientation, and Offset boxes.
	Length	Sets the length of the slotted hole.
	Orientation	Sets the orientation of the slotted hole. A custom thermal or antipad for a slotted hole has the same orientation as the 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. A custom thermal or antipad for a slotted hole has the same offset as the slot. 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. See also: “Slotted Holes”.
	Decal Units	Shows and sets the current units of the selected decal. This area is unavailable if you select Via in the Pad Stack Type area or when you use this dialog box in the PCB Decal Editor. Switching the decal units can cause errors if rounding off occurs.
	List	Produces a report describing the selected pad stack showing pad stack, slotted hole, and unit information. Vias are output in design units. For more information, see “Pad Stack Report” on page 189.
	List All	Produces a report describing the selected pad stack showing pad stack, slotted hole, and unit information. Vias are output in design units. For more information, see “Pad Stack Report” on page 189.
	Preview	Shows pad style, shape and size for the current options. When multiple layers are selected and there are different shapes assigned, the preview displays the last selected layer shape. When you see the message “Multiple Layer Values” it indicates

Area	Field	Description
		that multiple layers are selected in the Sh.: Sz.: Layer: list and the values of one or more fields are different. The field that is different between selected layers is blank and missing any specific value.
	Negative check box	Changes the preview area to a negative view for only the Thermal pad style.

Related Topics

Customizing Pad Stacks of Decal Pins

Flood Over Pads in a Copper Plane

Flood Over Vias in a Copper Plane

Pad Stack Report

Pad Sizes and Pad Stacks

Slotted Hole Offset Versus Pad Offset

Pad Stack Properties for Pin Dialog Box

To access: PCB Decal Editor > select a terminal > right-click > **Pad Stacks** popup menu item

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.

Description

Figure 197. Pad Stack Properties for Pin

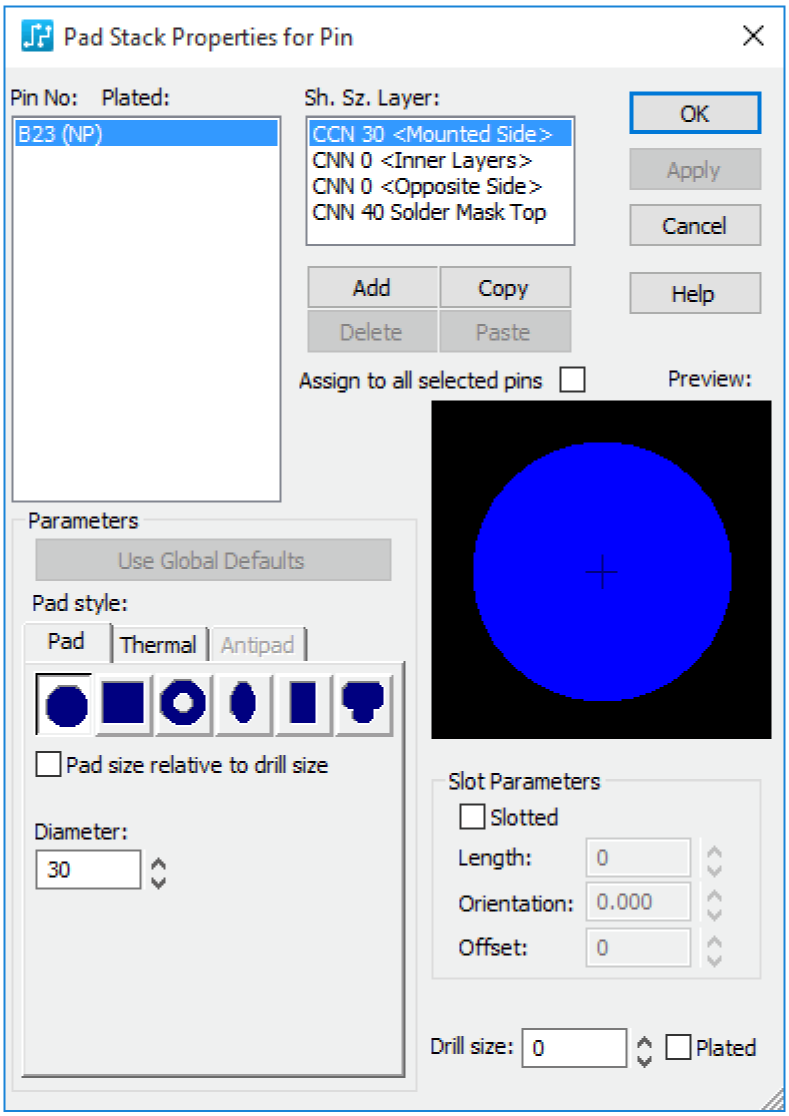


Figure 198. Pad Tab

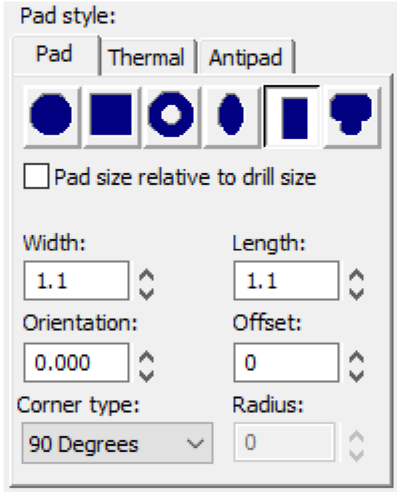


Figure 199. Thermal Tab

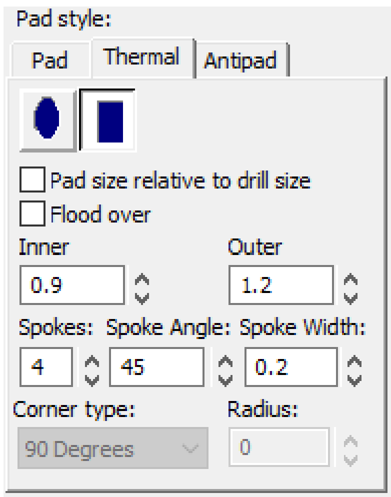
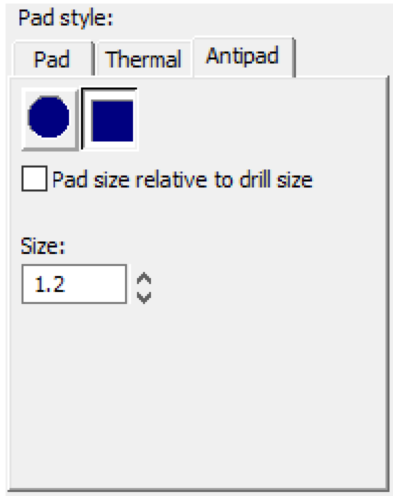


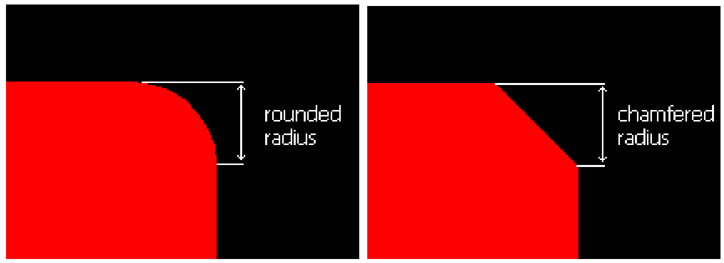
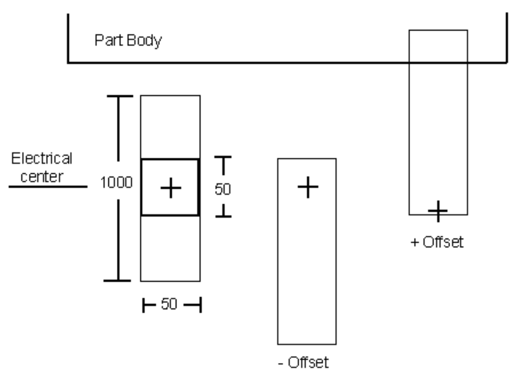
Figure 200. Antipad Tab



Objects

Field	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. • **Add** — Opens the Add Layer dialog box. • **Delete** — Removes custom layers. • **Copy/Paste** — Copy the shape and size information from one layer to one or multiple layers. Paste into other decals also. If antipad information is copied to a layer that does not support antipads (Mounted Side, Opposite Side) the antipad settings are ignored. Drill size value, Plated setting and Slot Parameters are not copied. See also: “Pad Sizes and Pad Stacks”.
Assign to all selected pins	When multiple terminals are selected simultaneously, modifies all the terminals listed under the Pin No: Plated list to match the terminal currently selected within the Pin No: Plated list.
Use Global Defaults	When the Thermal or Antipad tab is active, sets thermal or antipad shapes to the defaults specified in the Options dialog box > **Copper Planes** category > Thermals subcategory on page 1514. See also: “Design Rule Versus Pad Stack - Thermals and Antipads” on page 797.

Field	Description
Pad Style	Specify the shape and size of the pad, thermal, and antipad for each layer in the list. Thermals and Antipads apply to plane areas on split/mixed layers and CAM negative planes (for RS-274X output). If thermals and antipads are not specified in the pad stack, they will be generated based on the design rules. For more information, see “Design Rule Versus Pad Stack - Thermals and Antipads” on page 797. **Pad size relative to drill size** — Displays inner and outer pad sizes relative to the drill size.
Thermal notes	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. **Flood over check box** — 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. Available only when a pad shape button is selected. You can also 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 Options dialog box > **Copper Planes** category > Thermals subcategory on page 1514, 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.
Antipad notes	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.
Shape notes	Assigns a pad shape to the pad style for the selected layers in the Sh: Sz: Layers: list. If a layer in the selection already has an assigned shape it will be reassigned if you click one of the shapes. You can assign pad shapes as round, square, annular, oval, rectangular or 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.

Field	Description
	See "Creation of Custom Pad Shapes" for more information. • **Square and Rectangular** — The figure below illustrates the rounded and chamfered corner types. 
Orientation/Offset	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 201. Offset Example**  Maximum offset = ½ length or (1000/2) = 500
Drill size	Sets the drill size of the pad.
Plated	Sets whether the pad is plated with copper.

Field	Description
	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.
Slot Parameters	When slotting Round pads, the Diameter value becomes the Width. When slotting Square pads, the Size value becomes the width. • **Slotted check box** — Enables a slotted hole for the selected pad stack or pin. You must select this check box to activate the Length, Orientation, and Offset boxes. • **Length** — Sets the length of the slotted hole. • **Orientation** — Sets the orientation of the slotted hole. A custom thermal or antipad for a slotted hole has the same orientation as the 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. A custom thermal or antipad for a slotted hole has the same offset as the slot. 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. See also: “Slotted Holes”.
Preview	Shows pad style, shape and size for the current options. When multiple layers are selected and there are different shapes assigned, the preview displays the last selected layer shape. When you see the message “Multiple Layer Values” it indicates that multiple layers are selected in the Sh.: Sz.: Layer: list and the values of one or more fields are different. The field that is different between selected layers is blank and missing any specific value.

Related Topics

Customizing Pad Stacks of Decal Pins

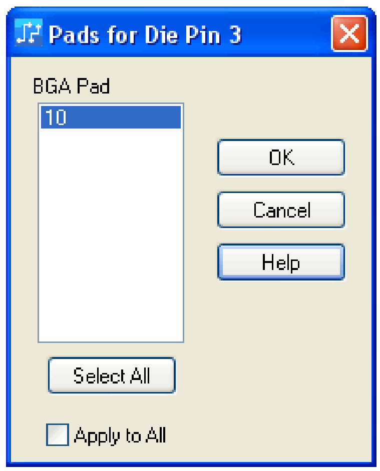
Pad Sizes and Pad Stacks

Pads for Die Pin Dialog Box

To access: **BGA Toolbar** button > select a BGA Pin > double-click in a BGA Pad cell > **Ellipsis** button

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 only to the BGA toolkit.



Objects

Field	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.

SailWind Router Link Dialog Box

To access: **Tools > SailWind Router**

The contents of the SailWind Router Link dialog box depend on your Synchronization mode settings.

Description

When you are not in Synchronization mode, the SailWind Router Link sets up autorouting options for SailWind Router and passes information from SailWind Layout to SailWind Router. Data is stored in the design file. Using the link you can either run SailWind Router and automatically open the current design file in the foreground, so you can view the autorouter's progress, or you can run SailWind Router in the background.

When you are in Synchronization mode, the SailWind Router link sets up autorouting options for SailWind Router and switch to SailWind Router. Using the link you can either run SailWind Router and automatically open the current design file in the foreground, so you can view the autorouter's progress, or you can run SailWind Router in the background.

Figure 202. SailWind Router Link Dialog Box

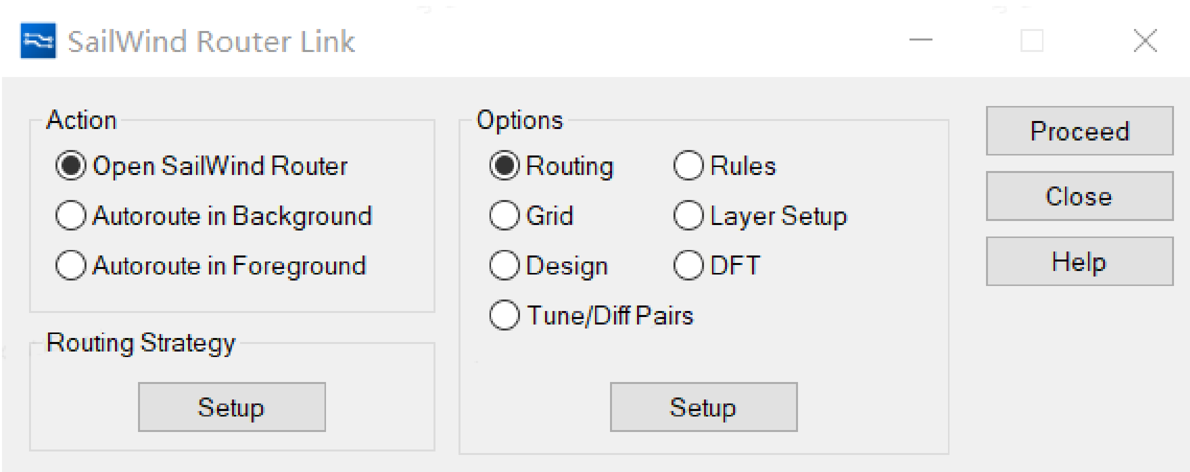
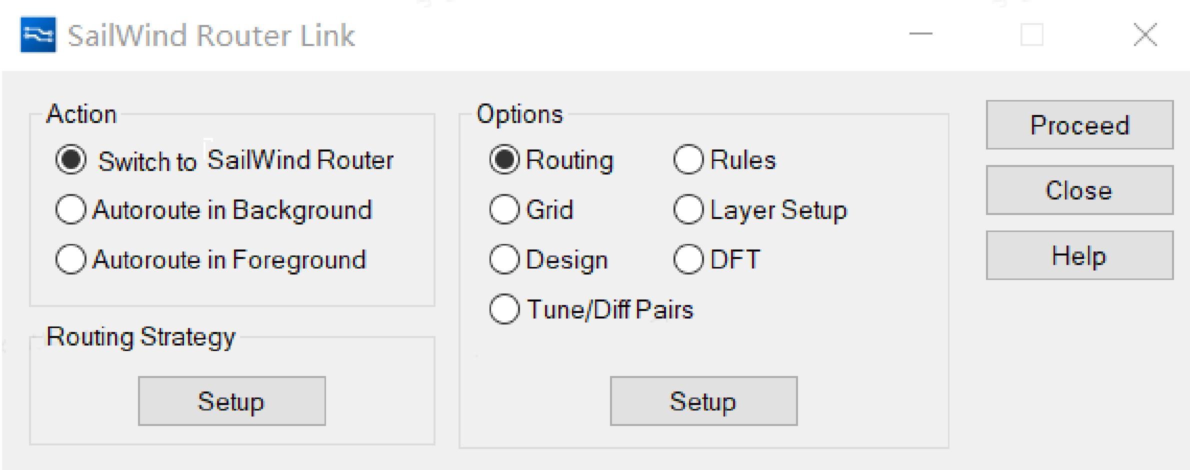


Figure 203. SailWind Router Link Dialog Box - Synchronization Mode



Objects

Table 191. SailWind Router Link Dialog Box Fields

Field	Description
Action	Specifies the action you want to perform. The options are: • **Open SailWind Router** — Opens SailWind Router and loads the design file into it. You can make changes to the design file in SailWind Router (rather than SailWind Layout) before routing. Non-Synchronization mode only. • **Switch to SailWind Router** — Links SailWind Router and SailWind Layout. Any changes you make to the design file in SailWind Router are reflected in SailWind Layout. Synchronization mode only. • **Autoroute in Background** — Opens SailWind Router and SailWind Router Monitor, but runs SailWind 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 SailWind Router and SailWind Router Monitor, and runs SailWind Router in the foreground making it the active program. If you are in Synchronization mode, opens SailWind 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

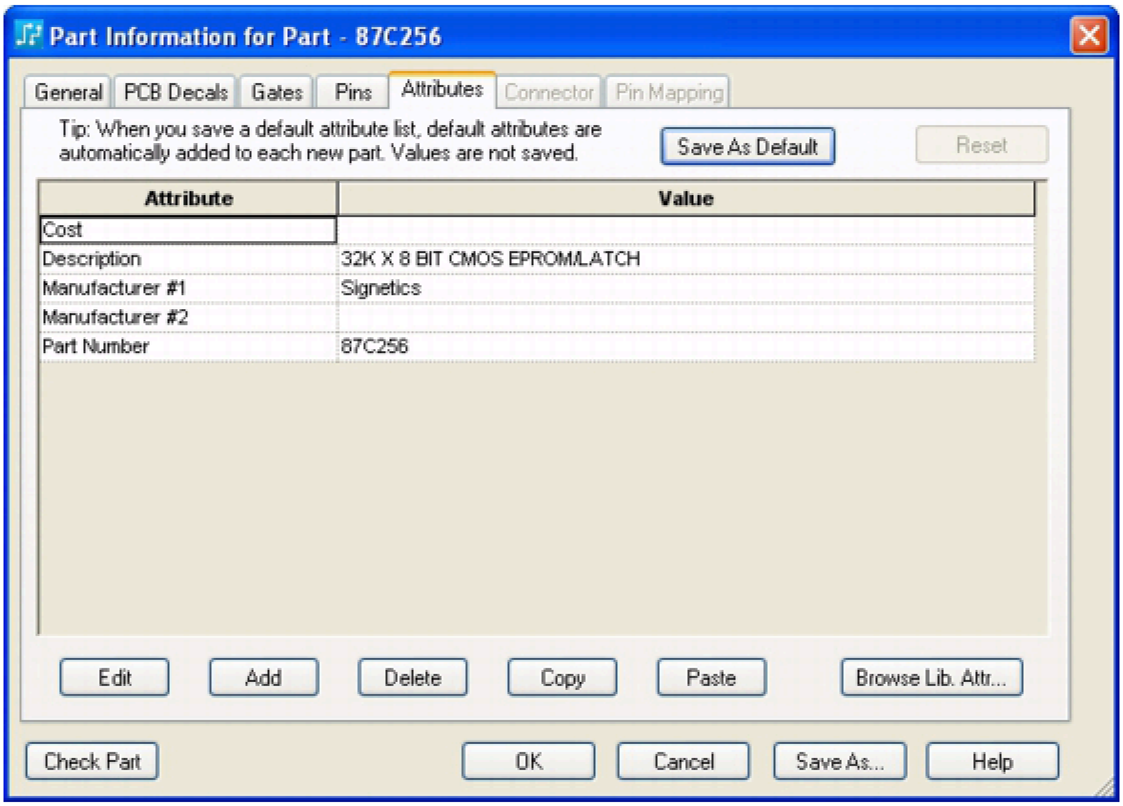
Autorouting Your Design Using SailWind Router

Synchronization Mode

Part Information Dialog Box, Attributes Tab

To access: **File > Library** menu item > select a library > **Parts** button > select part > **New** or **Edit** button > **Attributes** tab

Manage attributes for the selected part, and save as default attributes for new parts.



Objects

Object	Description
Save As Default	Save a default set of attributes to automatically use for each new part. Attribute values are not saved - only the names. Once the list of attributes is correct, click the button. The default attribute list is saved to C:\<install_folder>\PADS<latest_release>\Settings\defaultattribute.txt, which is shared between SailWind Logic and SailWind Layout. For more information, see "Defining Default Attributes for New Parts" on page 244.
Reset	Undo all changes for this tab only.

Object	Description
Attribute/Value list	The attributes are added automatically if they do not already exist. They are added during the creation of new parts, during synchronization, and during migration of libraries. The attributes are placed in the SailWind Designer symbol instances and are used during design packaging.
Edit	Click a cell and then click this button to edit the contents. As a faster alternative, double-click a cell to edit the contents. The attribute is edited for only the selected part. To manage attributes design-wide or in all libraries, use the “Manage Library Attributes Dialog Box” on page 1461.
Add	Start a new row with the cursor active in the Attribute cell ready to type the attribute name. Tab over to the Value cell to type the value. Net attributes are added to the Part level of the attribute hierarchy. For more information see “Attribute Hierarchy” on page 356.
Delete	First click a cell in the row to be deleted and then click the delete button to remove the row.
Copy	Click in a cell of the row to copy and then click the button.
Paste	Add a new empty row and then paste the contents.
Browse Lib. Attr	Opens the “Browse Library Attributes Dialog Box” on page 1139 where you can search for an existing attribute in all libraries.
Check Part	Checks for missing or inconsistent information. **Tip** Even if you do not 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.

Related Topics

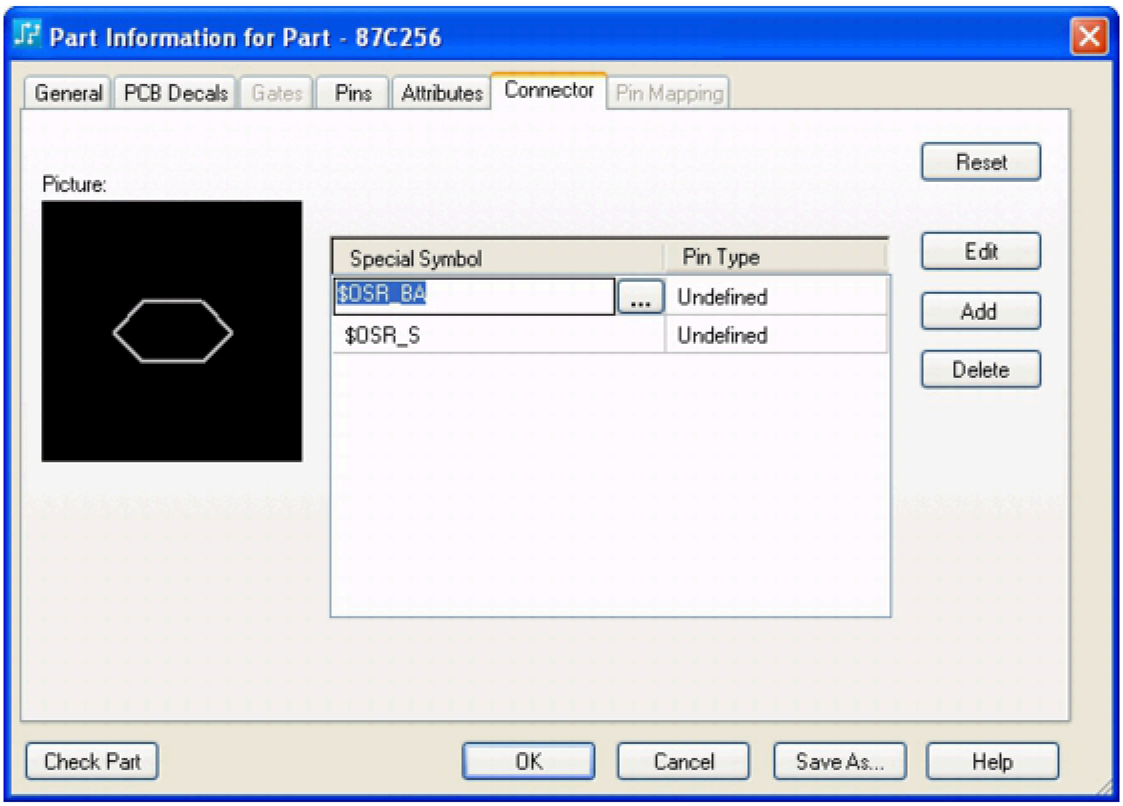
Adding and Modifying Part Type Attributes

Part Information Dialog Box, Connector Tab

To access: **File > Library** menu item > select a library > **Parts** button > select part > **New** or **Edit** button > **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.

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.



Objects

Field	Description
Picture	Displays a picture of the selected Special Symbol.

Field	Description
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. 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.
Check Part	Checks for missing or inconsistent information. **Tip** Even if you do not 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

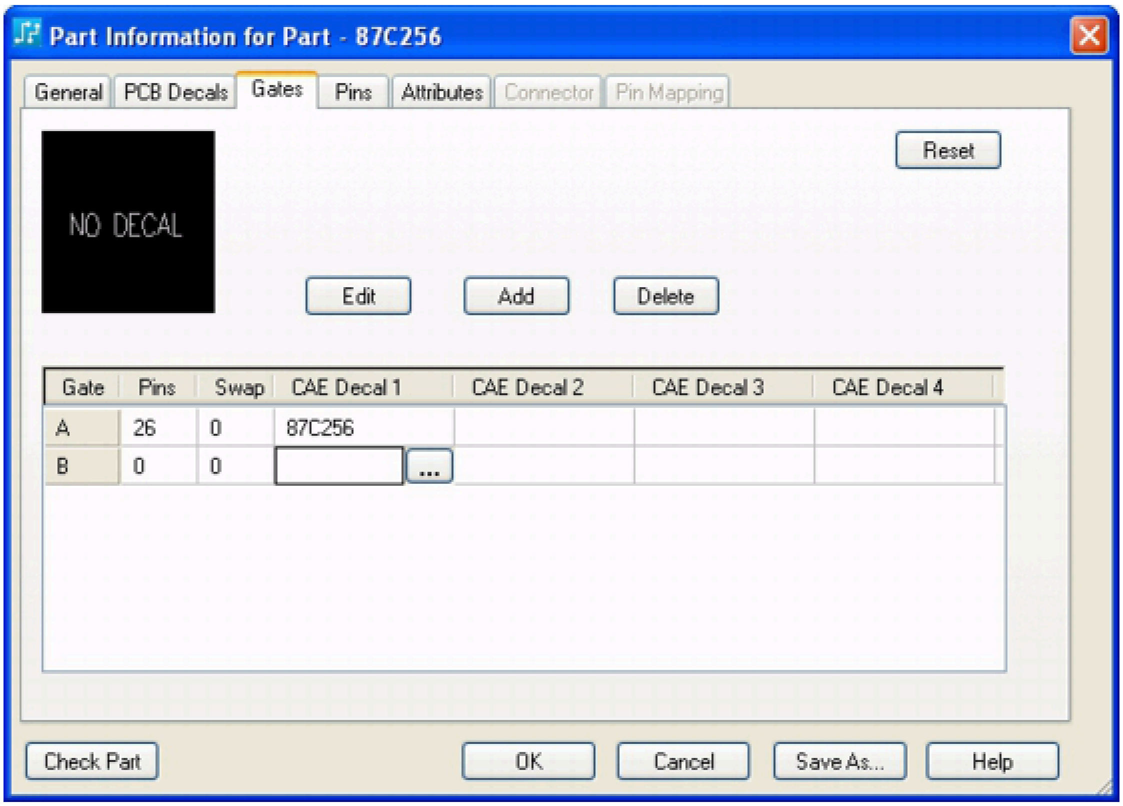
Creating a Connector Part Type

Creation of a New Part Type

Part Information Dialog Box, Gates Tab

To access: **File > Library** menu item > select a library > **Parts** button > select part > **New** or **Edit** button > **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.



Objects

Field	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.

Field	Description
	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.
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. 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. 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 do not 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.

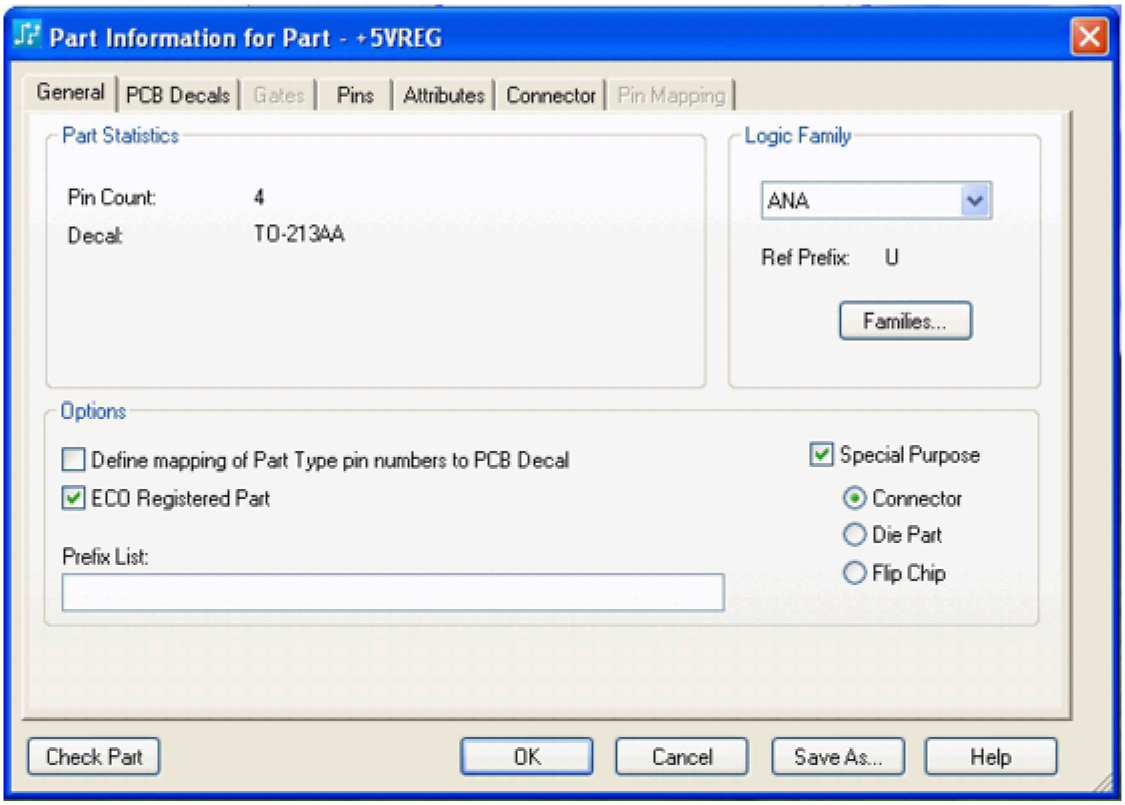
Related Topics

Creation of a New Part Type

Part Information Dialog Box, General Tab

To access: **File > Library** menu item > select a library > **Parts** button > select part > **New** or **Edit** button > **General** tab

Use the **General** tab in the Part Information dialog box to specify pin count, logic family, and various options for a part.



Objects

Field	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.

Field	Description
	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).
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 on page 1600 tab, where you map the numerical physical pins in the decal to the alphanumeric logical pins in the part type. **Restriction:** • 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 on page 1586 tab available. Connectors do not require a prefix list or gate definitions. **Restriction:** • 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.
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

Field	Description
	file. You can specify it when generating an *.eco* file using either the ECO Toolbar on page 1352, or the Compare/ECO dialog box on page 1181. 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. **Note:** **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. **Note:** **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. Even if you do not 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.

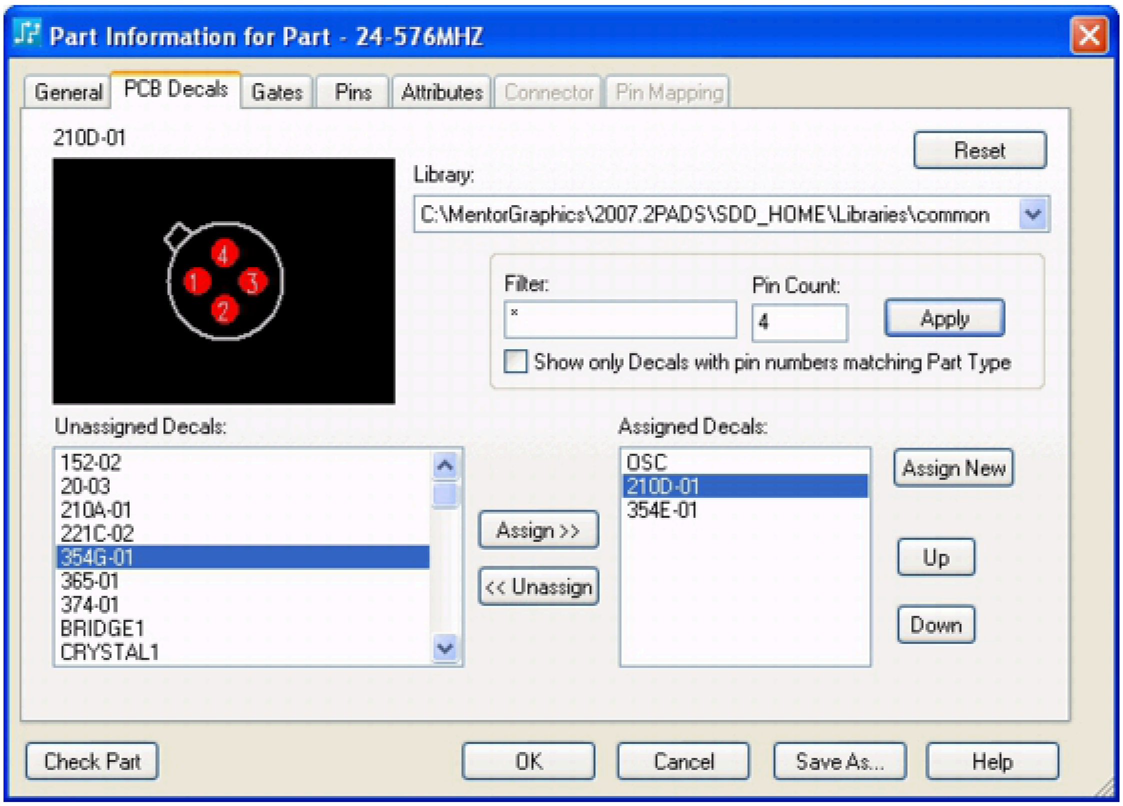
Related Topics

Creation of a New Part Type

Part Information Dialog Box, PCB Decals Tab

To access: **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **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.



Objects

Field	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. **Tip**

Field	Description
	• You can use wildcards on page 155 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. 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.
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. 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.
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 on page 1352**.** If you have not 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 on page 1763 tool, but you do not select PCB decals to be updated, the updated alternates will not be listed in the Decal list for selection in the Component Properties Dialog Box.

Field	Description
	Tip • 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. When your logic symbol uses alphanumeric pin numbering and the PCBDecal uses numeric pin numbering, you use the **Pin Mapping** tab to map the alphanumeric symbol pins to the numeric decal pins.
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 on page 1596 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. The decal at the top of the list is the default decal and is used when you add the part to the design.
Check Part	Checks for missing or inconsistent information. Even if you do not 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.

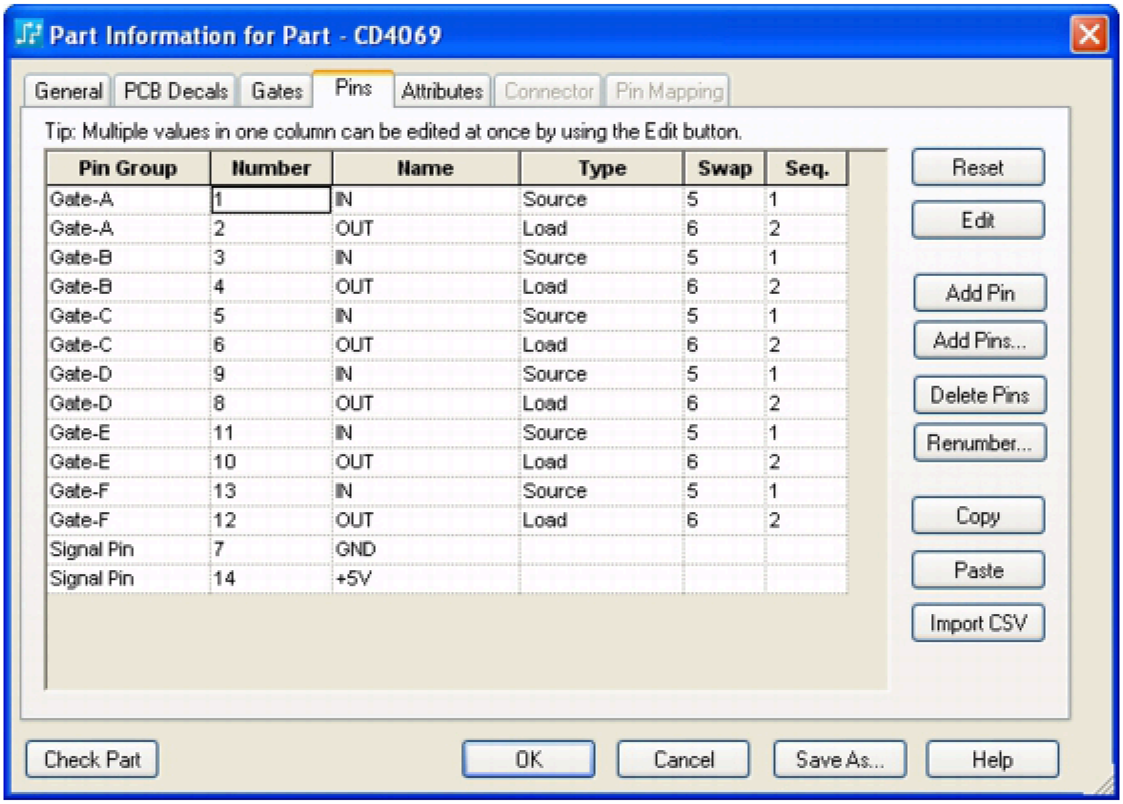
Related Topics

Creation of a New Part Type

Part Information Dialog Box, Pins Tab

To access: **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **Pins** tab

Use the **Pins** tab in the Part Information dialog box to assign gate pins, signal pins, and unused pins to the part. Typically, pin numbers on the **Pins** tab need to match those of the PCB decal. If your logical schematic symbol uses alphanumeric pin numbering and the PCB decal uses numeric pin numbering, use the **Pin Mapping** tab to overlay logical (schematic) alphanumeric pin numbers onto the physical numeric PCB decal.



Objects

Field	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.
Pin Group column	Select from Gate, Signal Pin, Connector Pin or Unused Pin.

Field	Description
	• **Gate Pins** — Assign to gates you have added to the part on the Gates on page 1588 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 SailWind 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 on page 1590 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. The pin number must match the PCB decal. For example, alphanumeric to alphanumeric. **Tip** If your logical schematic symbol uses alphanumeric pin numbering and the PCB decal uses numeric pin numbering, use the **Pin Mapping** tab to overlay logical (schematic) alphanumeric pin numbers onto the physical numeric PCB decal.
Name column	Specifies the pin signal or function name of the pin. The pin must have a name to be valid.
Type column	Specifies the type of pin. This column is used with gate pins only.
Swap column	Specifies an identical number between pins that can be swapped. Type 0 to disable swapping.
Seq. column	Specifies the 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.
Reset button	Undoes all changes you made in the **Pins** tab.
Edit button	Makes the selected cell available for editing. Press the Ctrl key or the Shift key 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:

Field	Description
	• **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. You must add a pin number to make the pin valid. **Tip** • 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.
Add Pins button	Opens the Add Terminals Dialog Box. **Restriction:** Total pins for the part can not exceed 32,767 pins. **Tip** • 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. 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.

Field	Description
	Tip • 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 *\SailWind Projects\Samples* folder.
Check Part button	Checks for missing or inconsistent information. Even if you do not 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

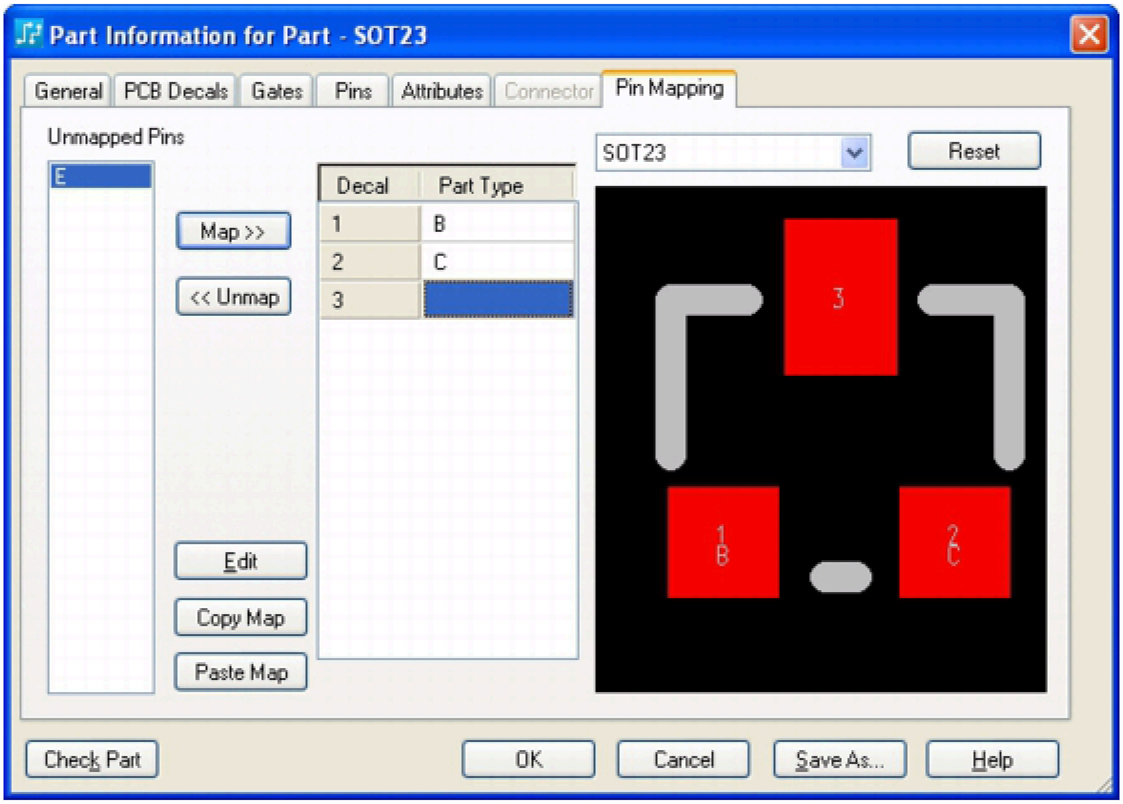
Renumbering Pins in the Pins Table

Creation of a New Part Type

Part Information Dialog Box, Pin Mapping Tab

To access: **File** menu > **Library** > select a Library > **Parts** button > select part > **New** or **Edit** button > **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, you could not save alphanumeric pin numbers in PCB decals.



Description

Note:
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.

Objects

Field	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 pin numbers assigned to the pins in the Decal. • **Part Type column** — The pin numbers that are mapped to the Part Type. These pin numbers represents the values assigned to the pins on the schematic symbol. These can be derived from the Unassigned Pins list (if a decal has been assigned on the **Pins** tab) or they can be manually entered in this table. **CAUTION:** If there are any errors in the pin numbering that has been automatically populated from the **Pins** tab, correct them by editing the table on the **Pins** tab. Do not attempt to renumber pins in the table on the **Pin Mapping** tab.
Edit	Makes the selected cell available for editing. You can also double-click the cell to edit the contents.
Copy Map	Places the map information into the paste buffer to paste into Microsoft Excel where you can make mass edits. Copy Map only works with the whole pin mapping table and not selective rows.
Paste Map	Pastes the map information from the paste buffer. 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.

Field	Description
	Even if you do not 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

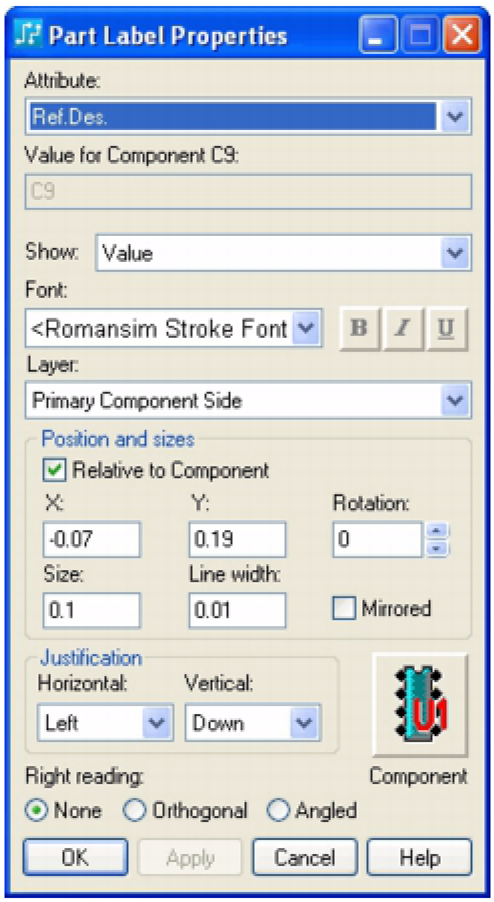
Creation of a New Part Type

Part Label Properties Dialog Box

To access: Select a part label > right-click > **Properties** popup menu item

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.



Objects

Field	Description
Attribute	The attributes available to you. If you are creating labels for jumpers, Reference Designator is the only available attribute.

Field	Description
	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.
Value for	The value of the selected attribute. **Tip** • 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 SailWind 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 on page 1862, 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. **Tip** • 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.
Size	Specifies the size of the font. Size (pts): This is font size in points and appears for system fonts

Field	Description
	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. **Tip** • 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 the **Justify Horizontally** popup menu item, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking the **Justify Vertically** popup menu item, 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

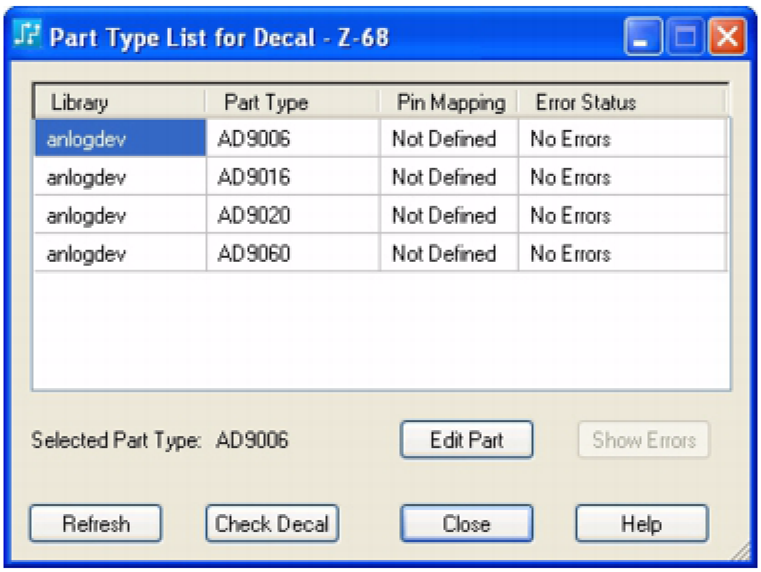
Modifying Part Label Properties

Part Type List for Decal Dialog Box

To access:

- Opens automatically when you open a decal in the Decal Editor
- **Tools > PCB Decal Editor** menu item > **Tools > Part Types** menu item

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.



Objects

Field	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.
Edit Part	Opens the Part Information dialog box on page 1590.

Field	Description
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

To access: **Tools > PCB Decal Editor**

SailWind 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 SailWind 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 click **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.

Tip

- SailWind 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.

Note:
Recommendation: Place free text and attribute values on the Silkscreen Top layer to avoid DRC violations or shorts.

Related Topics

Creation of a New Part Type

PDF Configuration Dialog Box

To access: **File > Create PDF** menu item

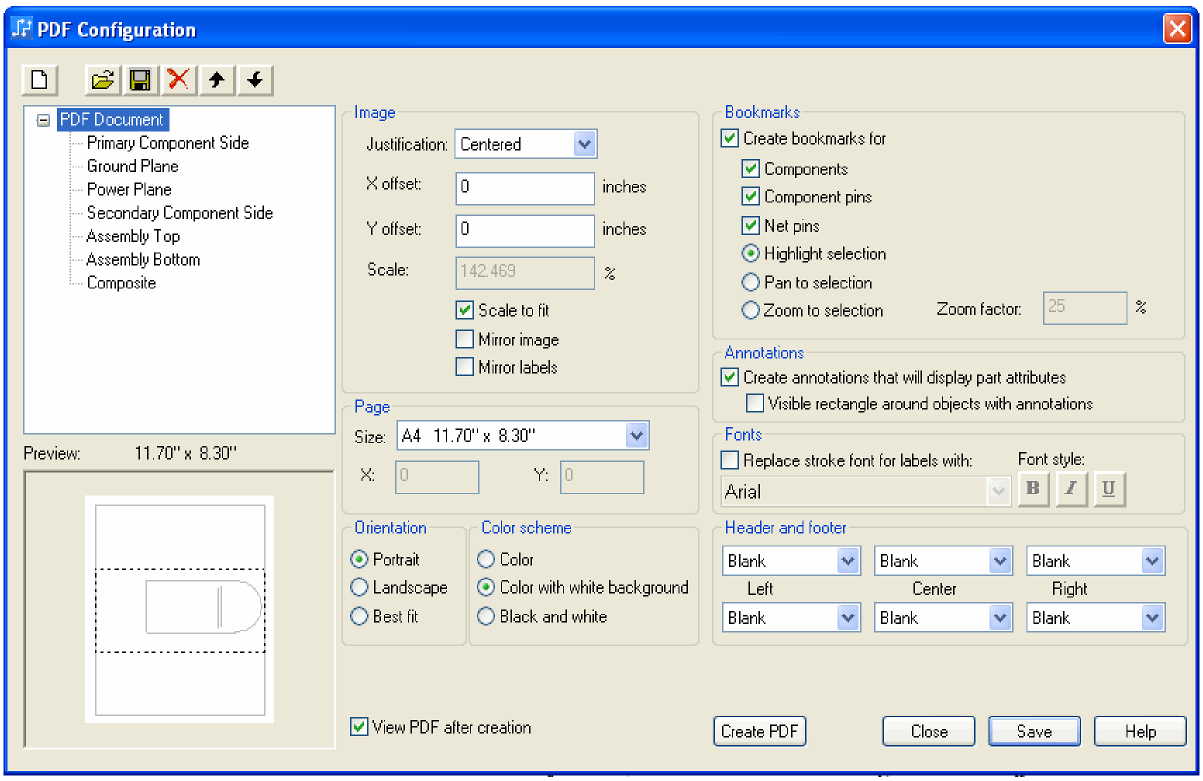
Use this dialog box to save one or more layers of your design as images in a PDF document. You can specify the contents of the document, its format, and its behavior, including the kinds of design items readers can search for, and how they are displayed.

Description

This dialog box displays its controls in two views--a “document” view and a “page” view.

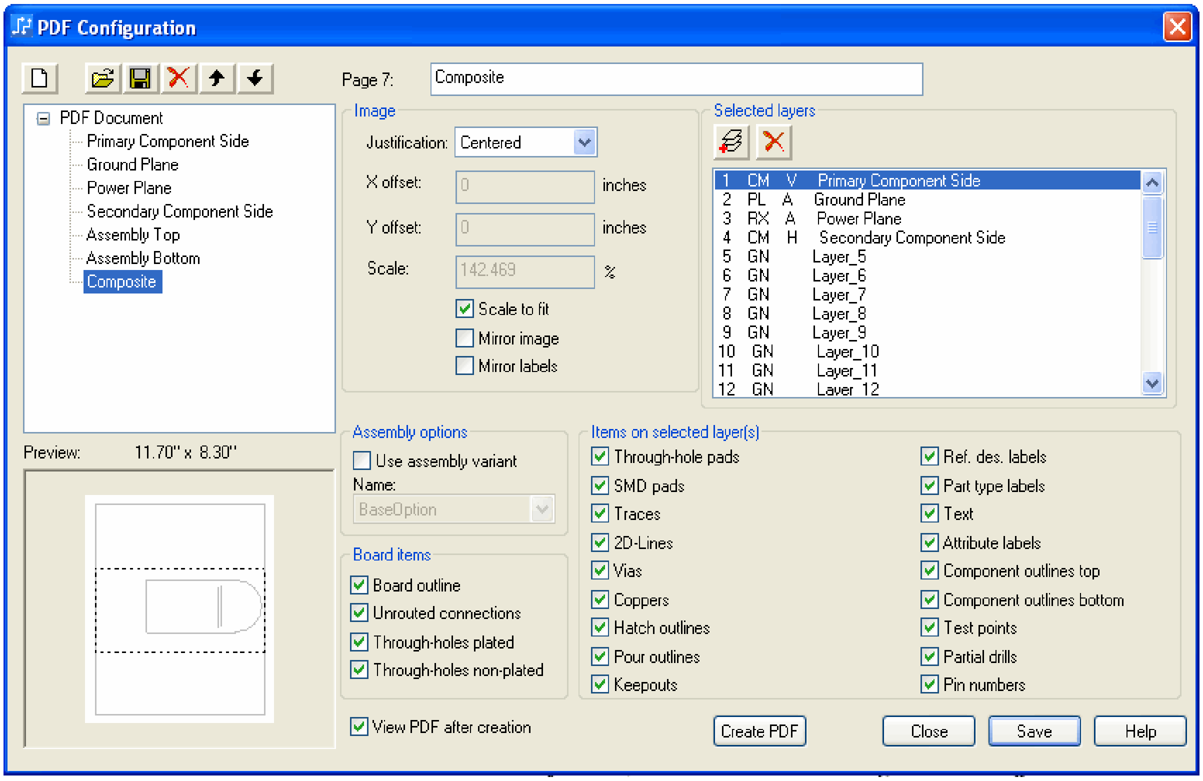
The document view appears when the root item (a PDF configuration) is selected in the page list at the top left of the dialog box. Settings made in the Document view affect all pages in the document.

Figure 204. 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 205. PDF Configuration Dialog Box, Page View



Objects

Table 192. 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. 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)	

Table 192. PDF Configuration Dialog Box Controls (continued)

Control/Area	Description
	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.
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:

Table 192. PDF Configuration Dialog Box Controls (continued)

Control/Area	Description
	• **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 check box 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 check box 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 check box 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)	
	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)	

Table 192. PDF Configuration Dialog Box Controls (continued)

Control/Area	Description
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 a part's attributes when you left-click on the item. The list of attributes and their order is hard coded. Attributes cannot be added to the list. Here is the list and order: • Ref Des • Description • Manufacturer #1 • PCB Decal • Part Number • Part Type
Visible rectangle around object with annotations	Draws a dotted line rectangle around each annotated item. **Note:** Clickable boxes around components are created as follows: • Clickable boxes for top-mounted components will only be created on pages that include the top layer, solder mask top layer, paste mask top layer, silkscreen top layer or assembly top layer. • Clickable boxes for bottom-mounted components will only be created on pages that include the bottom layer, solder

Table 192. PDF Configuration Dialog Box Controls (continued)

Control/Area	Description
	mask bottom layer, paste mask bottom layer, silkscreen bottom layer or assembly bottom layer. • For pages that do not contain specific references to the layers associated with top or bottom component layers, no clickable boxes are created.
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.
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.

Table 192. PDF Configuration Dialog Box Controls (continued)

Control/Area	Description
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.

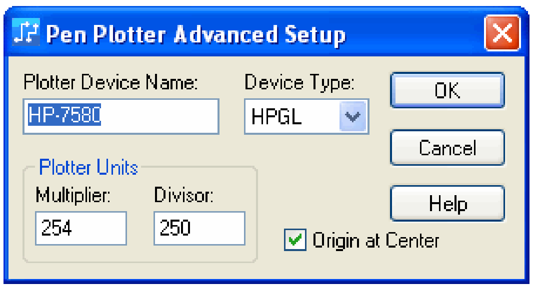
Pen Plotter Advanced Setup Dialog Box

To access:

- **File > CAM** menu item > **Add** button > **Pen** button > **Device Setup** button > **Advanced** button
- **File > CAM** menu item > select a document name > **Edit** button > Pen button > Device Setup button > Advanced button

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.



Objects

Field	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

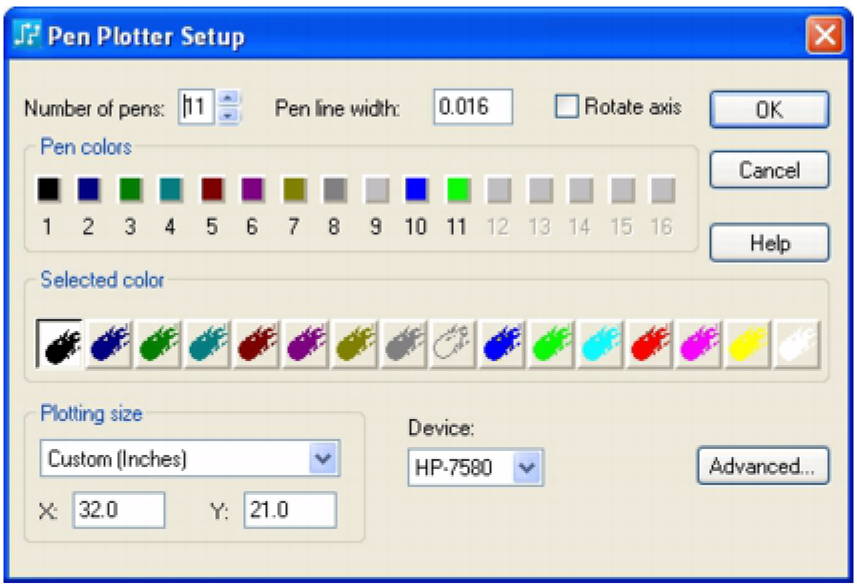
Creating CAM Outputs to Manufacture Your PCB

Pen Plotter Setup Dialog Box

To access:

- **File > CAM** menu item > **Add** button > **Pen** button > **Device Setup** button
- **File > CAM** menu item > select a document name > **Edit** button > **Pen** button > **Device Setup** button

Use the Pen Plotter Setup dialog box to set various pen plot options.



Objects

Field	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
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.

Field	Description
	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. 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.

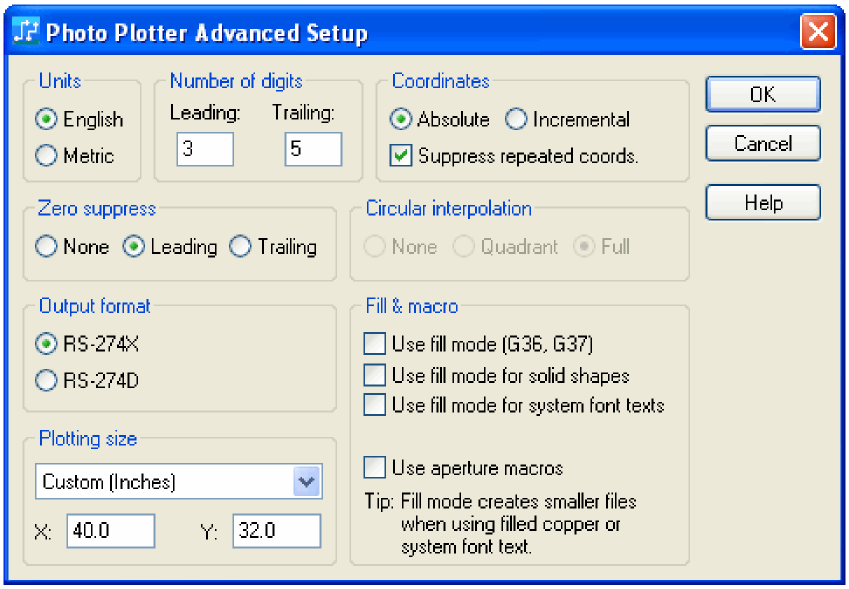
Related Topics

Creating CAM Outputs to Manufacture Your PCB

Photo Plotter Advanced Setup Dialog Box

To access: **File > CAM** menu item > **Add** or **Edit** button > **Photo** button > **Device Setup** button > **Advanced** button

Use the Photo Plotter Advanced Setup dialog box to customize the output of photo plot files.



Objects

Field	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.

Field	Description
	• **None** — retains leading and training zeros. • **Leadin** — suppresses zeros before the decimal point. • **Trailing** — suppresses zeros after the decimal point.
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. **Restriction:** 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.
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. **Restriction:** Cleared and unavailable when RS-274D is selected as the output format.

Field	Description
	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.
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. **Restriction:** • Cleared and unavailable when RS-274D is selected as the output format. • Selected and unavailable when “Use fill mode(G36, G37)” 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.
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. **Restriction:** • Cleared and unavailable when RS-274D is selected as the output format. • Selected and unavailable when “Use fill mode(G36, G37)” 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.
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

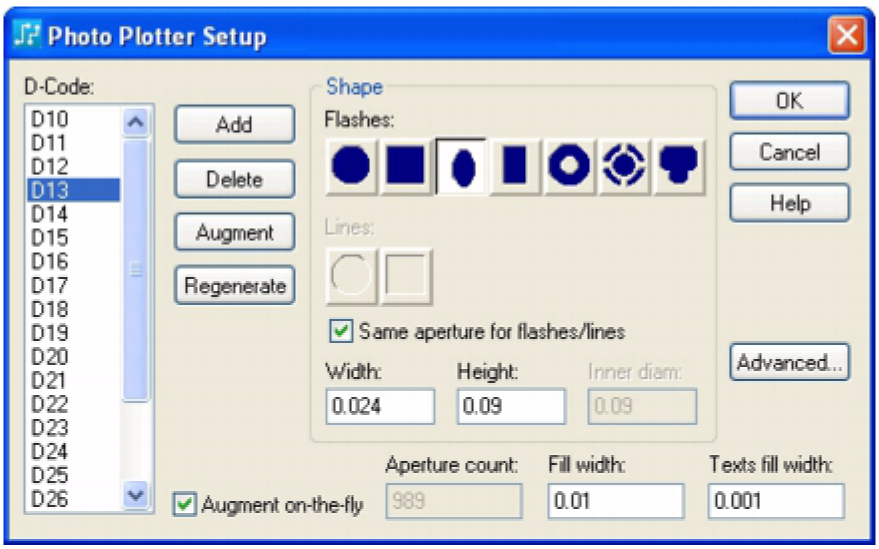
RS-274-X Format

Photo Plotter Setup Dialog Box

To access:

- **File > CAM** menu item > **Add** button > **Photo** button > **Device Setup** button
- **File > CAM** menu item > select a document name > **Edit** button > Photo button > Device Setup button

Use the Photo Plotter Setup dialog box to set up photo plotting and send your output to a plotter



Objects

Field	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. 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.

Field	Description
	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.
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.

Field	Description
Advanced button	Opens the Photo Plotter Advanced Setup Dialog Box

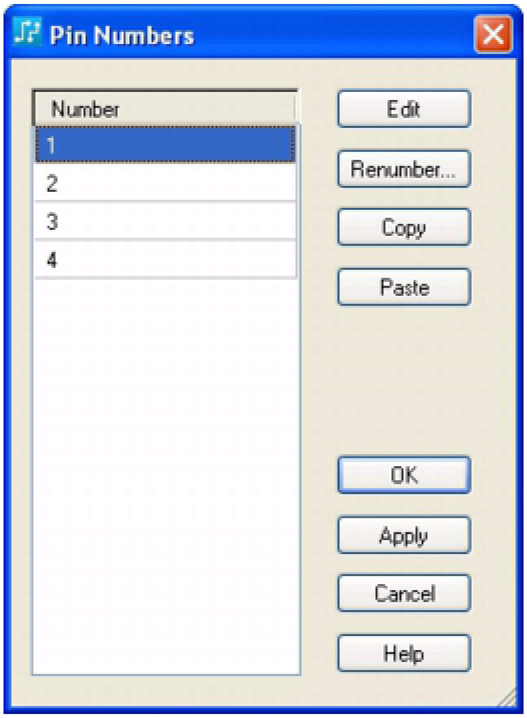
Related Topics

Creating CAM Outputs to Manufacture Your PCB

Pin Numbers Dialog Box

To access: (PCB Decal Editor) **Setup > Pin Numbers** menu item

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.



Objects

Field	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. Data only pastes into selected cells.

Related Topics

Renumber Using the Pin Numbers Dialog Box

Pin Pair Properties Dialog Box

To access: Select a pin pair > right-click > **Properties** popup menu item

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.

Restriction:
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.

Description

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.

Objects

Field	Description
Net	Displays the net name of the pin pair.
Connection	Displays the component pin and/or virtual pin connections that define the pin pair. For each component pin connection point, the reference designator of the component is listed first, followed by a period, then the pin number. For each virtual pin, the instance of the virtual pin in the format VP<number> is listed
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. Includes half the Discrete length value of each connected pin of components

Field	Description
	that have a Discrete length assigned. When routed to the pad, measurements are made to the origin of the pad. • **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 Parameter Setup 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 on page 1775 after you set up a High Speed check on page 1363 to check Impedance. It is not verified by online DRC on page 1843. • **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 Parameter Setup 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 on page 1775 after you set up a High Speed check on page 1363 to check Delay. It is not verified by online DRC on page 1843.
Protect Routes	Protects selected routes on page 1855 or traces on page 1867 from being moved. Similar to gluing components but you can allow protected traces to be edited by SailWind Router in the Routing Rules on page 1666. See also: "Route and Unroute Protection".
Protect Unroutes	Protects unrouted connections and the unrouted portions of partial routes on page 1846. Protected unroutes cannot be modified or routed. See also: "Route and Unroute Protection".

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.

Field	Description
	• **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. on page 1167See 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. on page 1167This 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 on page 1487 of the net to which the pin pair belongs.

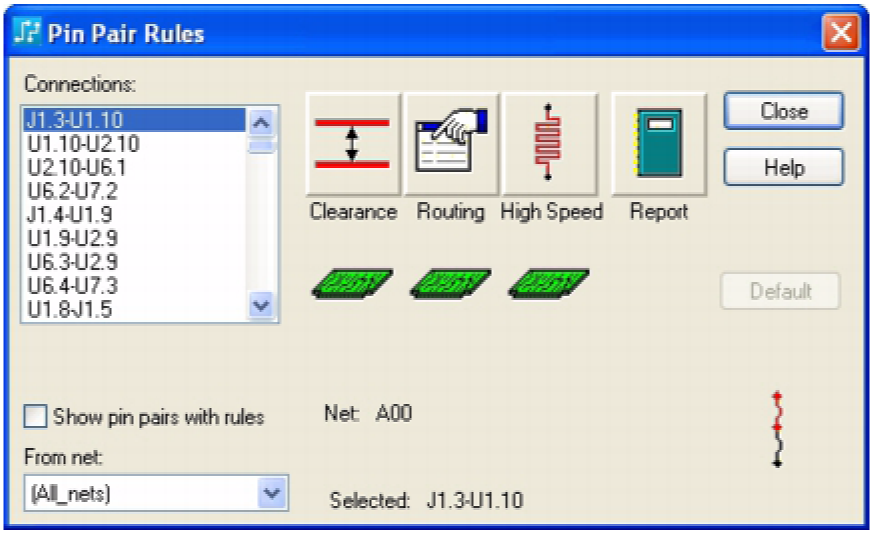
Related Topics

Modifying Pin Pair Properties

Pin Pair Rules Dialog Box

To access: **Setup > Design Rules** menu item > **Pin Pairs** button

Use the Pin Pair Rules dialog box to define design rules that apply to pin pairs.



Objects

Field	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. 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.

Field	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

Resetting Pin Pair Rules to Default Rules

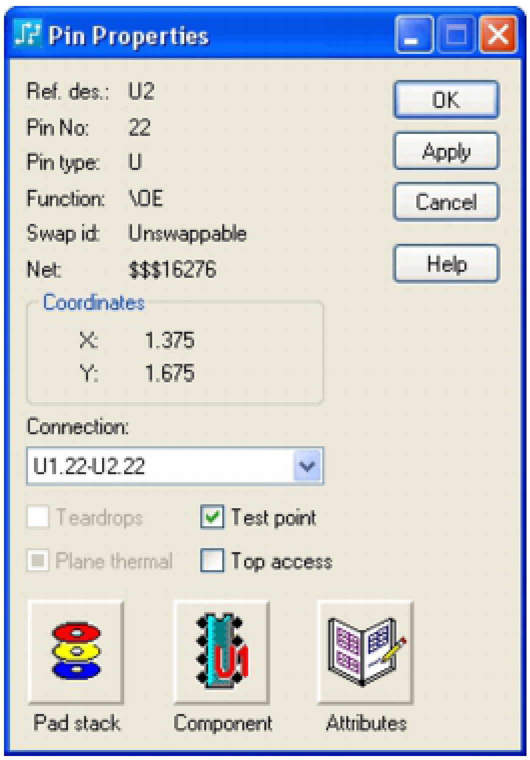
Design Rule Hierarchy

Pin Properties Dialog Box

To access: Select a pin > right-click > **Properties** popup menu item

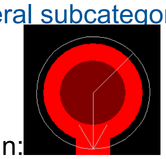
The Pin Properties dialog box displays the component's reference designator, the pin number, pin type, function (pin name), swap identification, netname, and coordinates of the selected pin.

Note:
If you select another pin while the dialog box is open, the dialog box updates with information about the newly selected pin.



Objects

Field	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.

Field	Description
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 Options dialog box > **Routing** category > General subcategory on page 1542, this option is unavailable.
Plane Thermal	Determines whether the pin or via is eligible to receive a thermal on page 1866. **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: "Setting Pins and Vias as Thermals on page 797".
Test Point	Makes the via or pin a test point. This is a three-state check box on page 1866. See also: "Performing a Test Point Audit". 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 in the Options dialog box > **Routing** category > General subcategory on page 1542, an arrow is drawn on it in the design: 
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".

Field	Description
	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 on page 995 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.

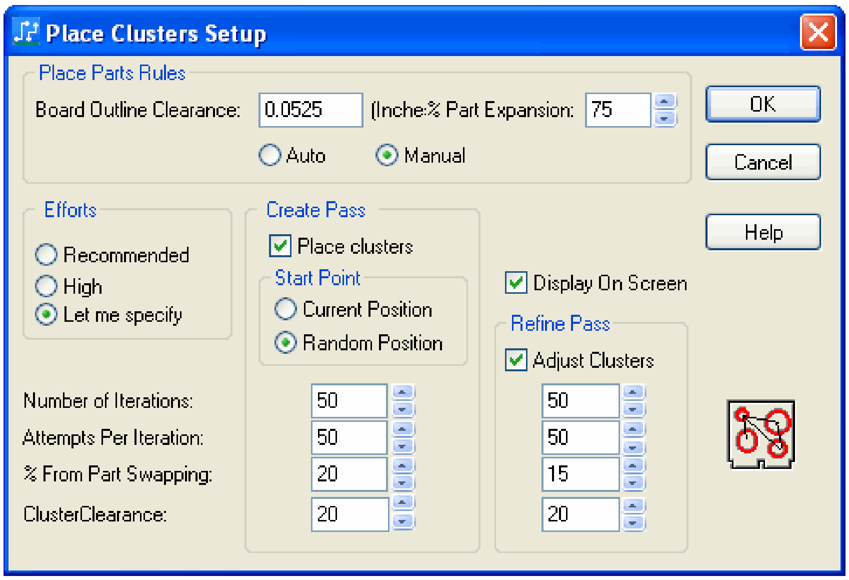
Related Topics

Modifying Pin Properties

Place Clusters Setup Dialog Box

To access: **Tools > Cluster Placement** menu item > **Place Clusters** button > **Setup** button

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.



Objects

Field	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.

Field	Description
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.
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

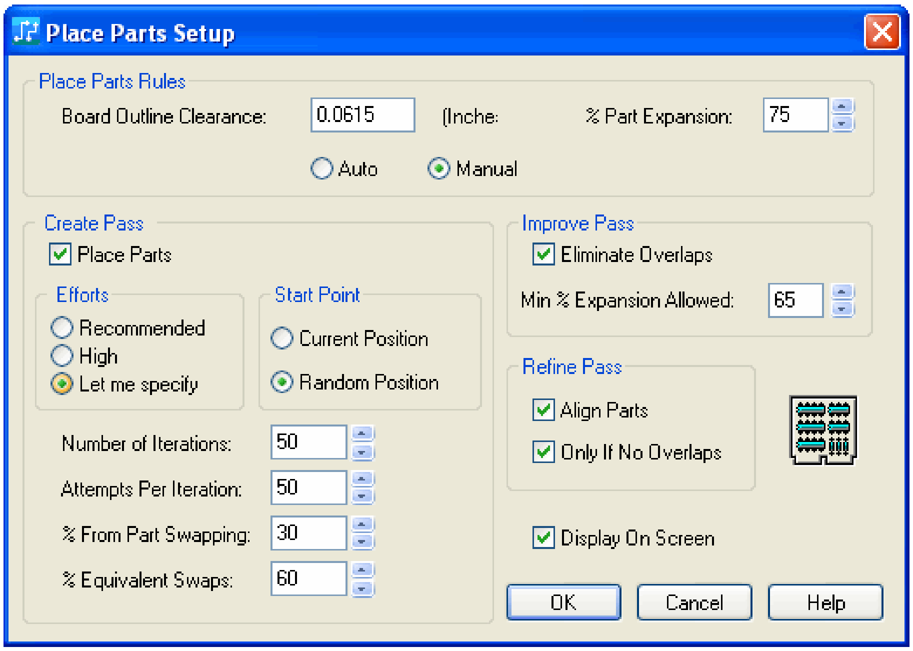
Automatic Placement

Cluster Placement

Place Parts Setup Dialog Box

To access: **Tools > Cluster Placement** menu item > **Place Clusters** button > **Setup** button

Use the Place Parts Setup dialog box to control how SailWind Layout handles parts during Automatic Placement.



Objects

Field	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.

Field	Description
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.
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.

Field	Description
Display On Screen	Displays an outline of each part and its movement throughout the automatic placement process.

Related Topics

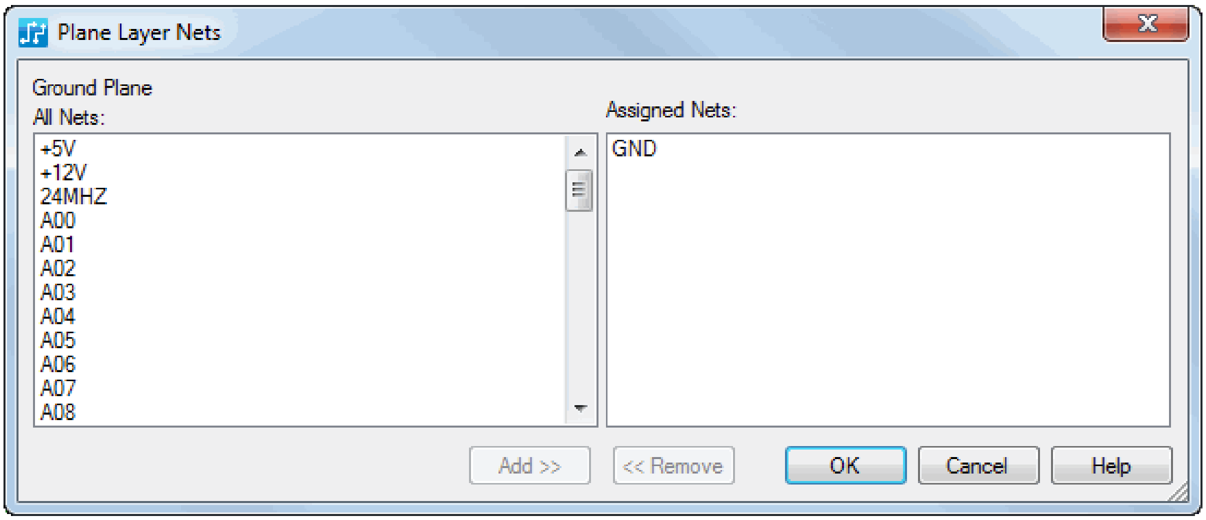
Automatic Placement

Cluster Placement

Plane Layer Nets Dialog Box

To access: **Setup > Layer Definition** menu item > select a plane layer > **Assign Nets** button

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 that components can access for ground and power. CAM output produces thermal relief connection pads for nets associated with the plane layer.



Objects

Field	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

To access:

- **File > CAM** menu item > **Add** button > **Options** button
- **File > CAM** menu item > select a document name > **Edit** button > **Options** button. If you selected NC Drill as the Document Type, the NC Drill Options Dialog Box opens instead.

Use this dialog box to set plotting options.

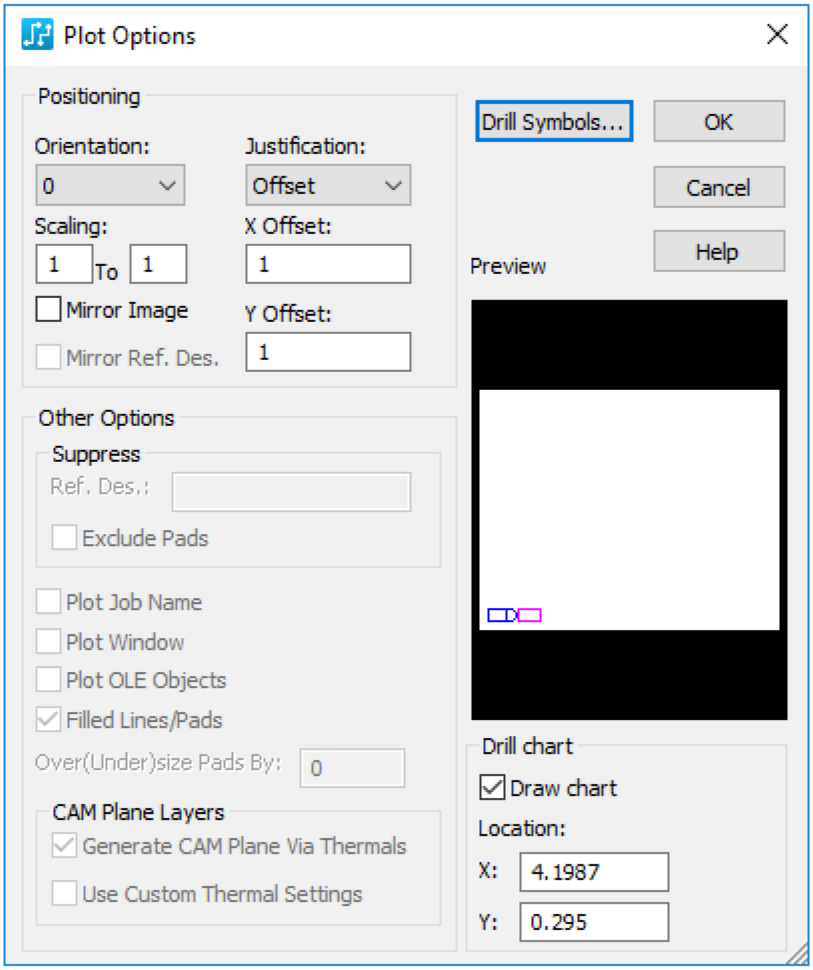
Note:
You can also use this dialog box to gain access to the “Drill Symbols dialog box” on page 1334.

Description

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.



Objects

Field	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.

Field	Description
	• **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.
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.

Field	Description
	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" on page 209. **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" on page 960. **Tip** • 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.
Use Custom Thermal Settings	Specifies to use thermal and aperture settings defined in the Pad Stacks Properties dialog box. on page 1566 If you disable this setting, CAM plane thermals are generated according to a set of parameters. See the Results section of "Creating a CAM Plane Gerber-format File" on page 951. **Note:** **Requirement:** CAM planes using custom antipads defined in the Pad Stacks require that you select the Remove Unused Pads check box in the Options dialog box > **Copper Planes** category > Thermals subcategory on page 1514. Default antipad settings are used if this setting is not enabled. 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.

Field	Description
Drill Symbols button	Opens the "Drill Symbols dialog box" on page 1334 where you can select the drill symbols you want to use with the drill drawing. **Restriction:** The Drill Symbols button is only available when you select the Drill Drawing document type.
Preview area	Displays the positioning of the design with regard to the plot area or printed page as you make changes to the options. document. The blue outline shows the board outline of your design with respect to the plot area. When creating a drill drawing, the preview area also displays the position of the Drill Chart (if enabled) with a magenta-colored rectangular outline. You may need to reposition the drill chart to keep from overlapping your design. See "Creating a Drill Drawing with Drill Table" on page 954 for more information.
Draw Chart	Specifies to include the drill chart in the plot.
X/Y Location	Specifies the location of the drill chart in design units.

Related Topics

Creating CAM Outputs to Manufacture Your PCB

Process Status Dialog Box

To access: Compare/ECO dialog box on page 1185 > select Generate ECO File check box > click **Run**.

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 an *.eco* file.

Description

This dialog box has two different views depending on whether Forward Annotation or Backward Annotation are occurring.

- **Forward Annotation dialog** — described in Table 193.
- **Backward Annotation dialog** — described in Table 194.

Figure 206. Process Status Dialog Box - Forward Annotation

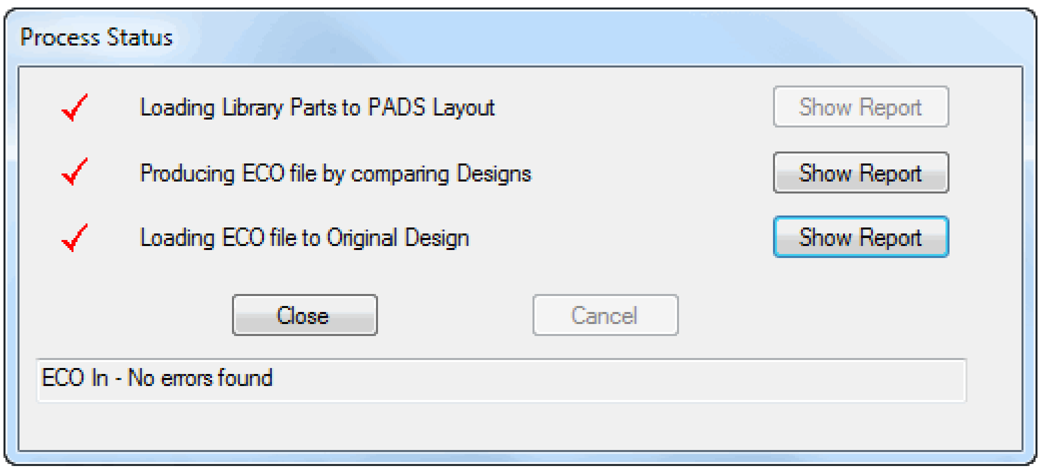
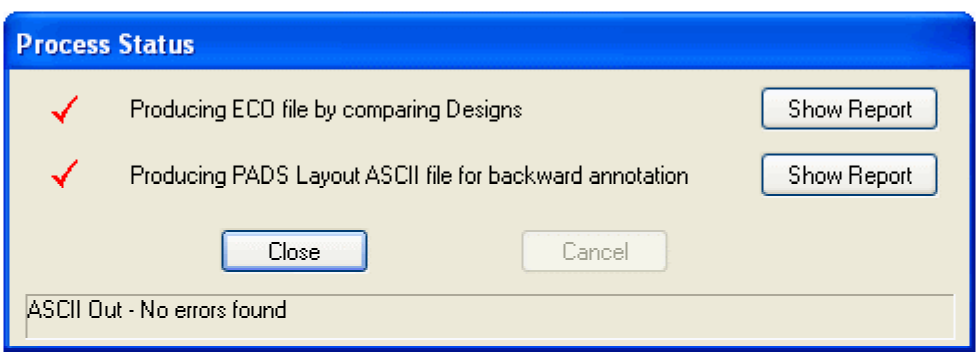


Figure 207. Process Status Dialog Box - Backward Annotation



Objects

Table 193. Process Status Dialog Box - Forward Annotation Contents

Field	Description
Loading Library Parts to SailWind Layout	Displays the status of importing the library part changes into SailWind Layout. This option only appears when you choose to update part types in the library by importing the *.p* file generated by PADS Designer.
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.
Loading ECO file to Original Design	Displays the status of importing the *.eco* file into SailWind Layout to update the design with changes forward annotated from the schematic software. This option only appears when you choose to update the original design on forward annotation Click Show Report to display the *eco.err* file.

Table 194. Process Status Dialog Box - Backward Annotation Contents

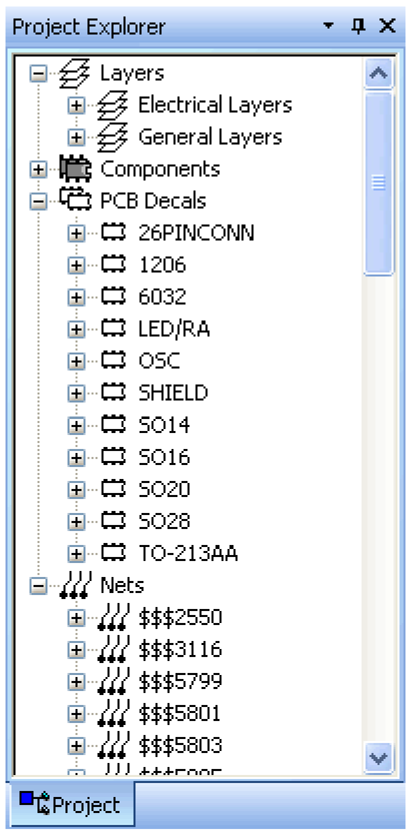
Field	Description
Producing ECO file by comparing Designs	Displays the status of generating the *.eco* by comparing designs. Click Show Report to display the *.eco* file and the *.log* file.
Producing SailWind Layout ASCII file for backward annotation	Displays the status of exporting the *.asc* file used in backward annotation to PADS Designer. **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

To access: Click the **Project Explorer** button. The Project Explorer is not available in the PCB Decal Editor.

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 updates automatically to reflect the changes you make.

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



Objects

In the right-click menu, if you enable the "Expand all on selection" option, the corresponding branch is expanded when an object is selected in the design. For example you select a net in the design and it expands the Nets branch of the Project Explorer.

Objects in the Project Explorer are placed in object groups. Object groups are of two types: primary and secondary. You cannot remove or rename primary object groups. Modification of secondary group items is only available in SailWind Router.

The following table lists and describes the primary and secondary object groups.

Table 195. Object Groups and Subgroups

Primary Group	Secondary Group	Description
Layers	Electrical layers	Lists all electrical layers, including plane layers and routing layers
	General layers	Lists all other layers except electrical
Components		Lists all components and pin pairs
Part decals/PCB decals		Lists all part decals in the design or all components that use the selected part decal
Nets		Lists all nets in the design
PCB decals		Lists the PCB decals used in the design

Related Topics

Selecting Design Objects Using the Project Explorer

Zooming to a Design Object Using the Project Explorer

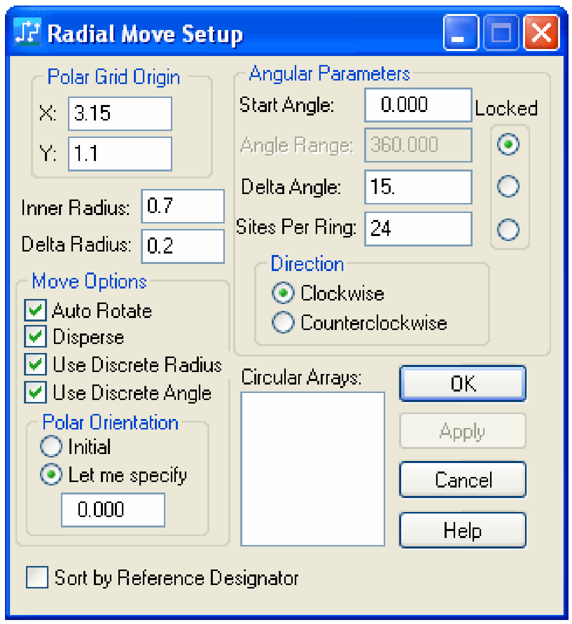
Chapter 53
GUI Reference Elements Q Through R

Read the sections that follow to learn more about dialog box elements in SailWind Layout.

Radial Move Setup Dialog Box

To access: **Tools > Options** > **Grids** tab > **Radial Move Setup** button

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.



Objects

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.
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.

Name	Description
	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.
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

Setting Up a Polar Grid

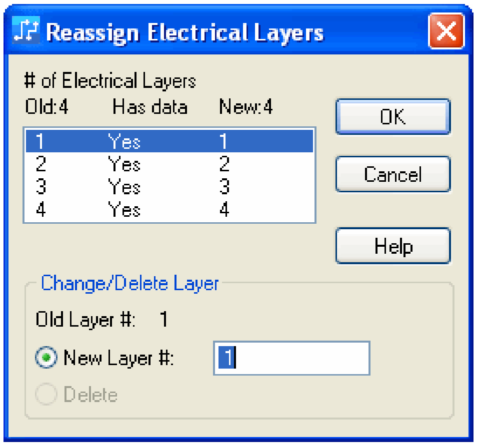
Moving Design Objects Radially

Component Arrays

Reassign Electrical Layers Dialog Box

To access: **Setup > Layer Definition** menu item > **Reassign** button

Use the Reassign Electrical Layers Dialog box to move data from one electrical layer to another.



Objects

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

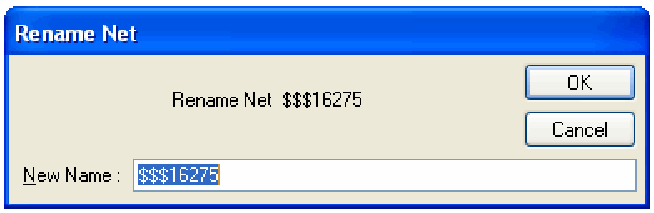
Reassigning Electrical Layers

Rename Net Dialog Box

To access: Select a net > **ECO Toolbar > Rename Net** button

Use the Rename Net dialog box to change the name of a net. Changing the name of a net also changes the netlist. It is therefore 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.



Objects

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

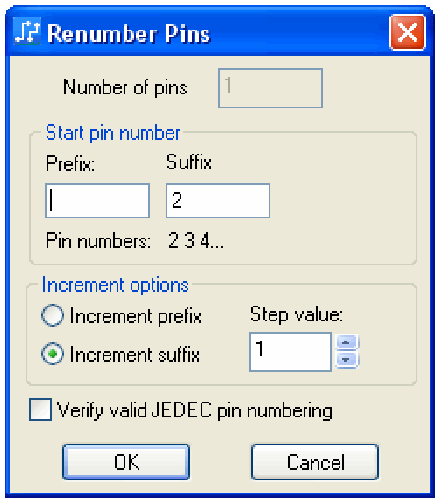
Renumber Pins Dialog Box

To access:

- PCB Decal Editor > select a starting terminal > right-click > **Renumber Terminals** popup menu item

- Part Information dialog box > Pins tab on page 1596 > select pin(s) > **Renumber** button

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.



Objects

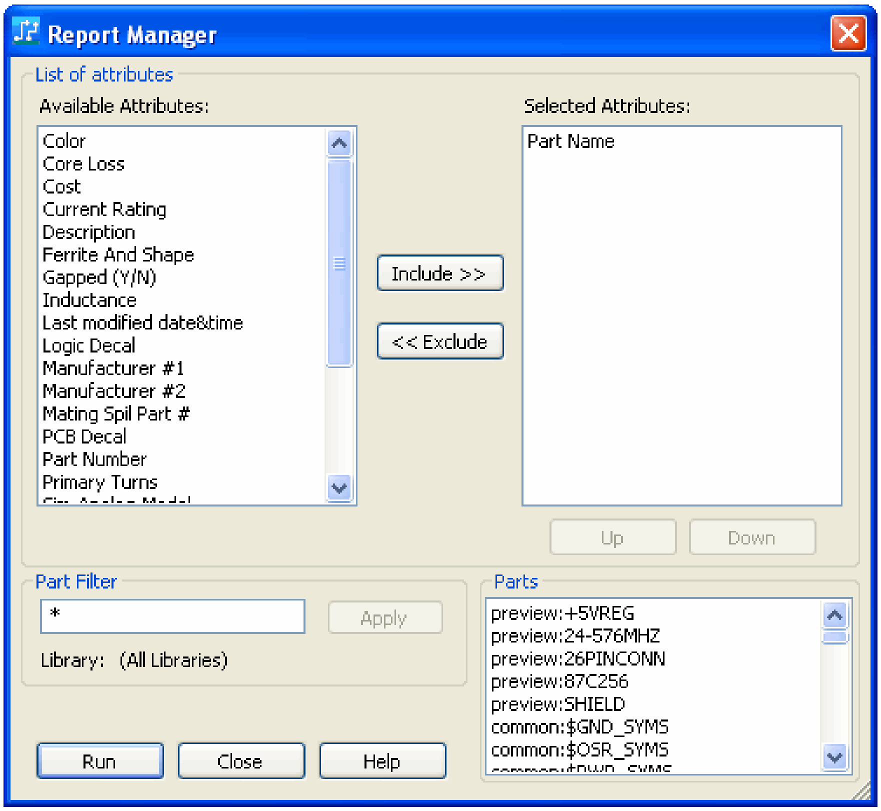
Name	Description
Number of pins	The number of pins available for renumbering.
Prefix/Suffix	For a single pin number, use either Prefix or Suffix box, and void the other box. Use both boxes if you want to increment one of the values. Alphabetic and numeric values can be used in either box.
Pin numbers	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.

Name	Description
	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.

Report Manager Dialog Box

To access: **File > Library** menu item > filter by Parts > **List to File** button

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.



Objects

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 the Ctrl key and click each attribute.) Click **Include >>** to include selected attributes in the report.
Selected attributes	Attributes in the report. Click an attribute in the list of select it. (To select additional attributes, press the Ctrl key 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.

Field or Button	Description
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

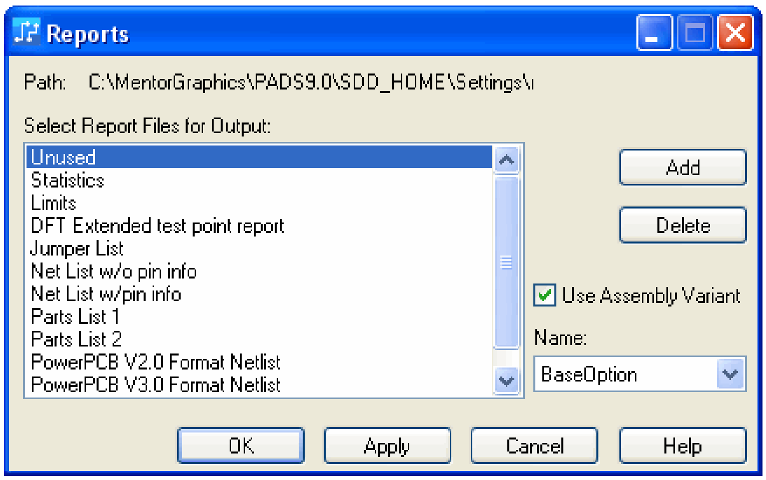
Creating a Report of the Parts in a Library

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

Reports Dialog Box

To access: **File > Reports** menu item

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.



Objects

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

Running a Report

Reuse Properties Dialog Box

To access: Select a reuse > right-click > **Properties** popup menu item

Use this dialog box to modify the selected physical design reuse.

Description

Figure 208. Reuse Properties Dialog Box When Adding a Reuse

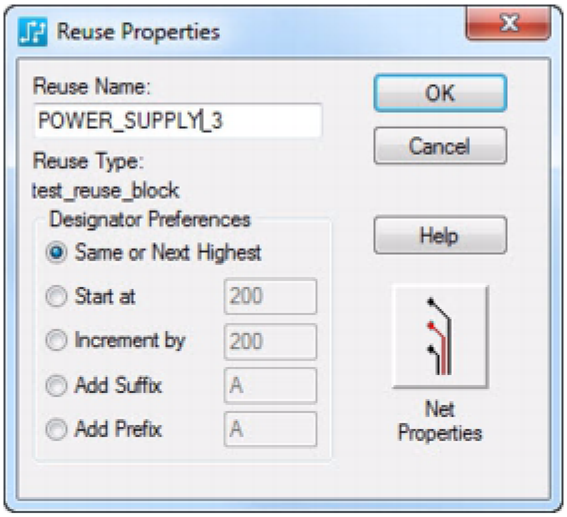
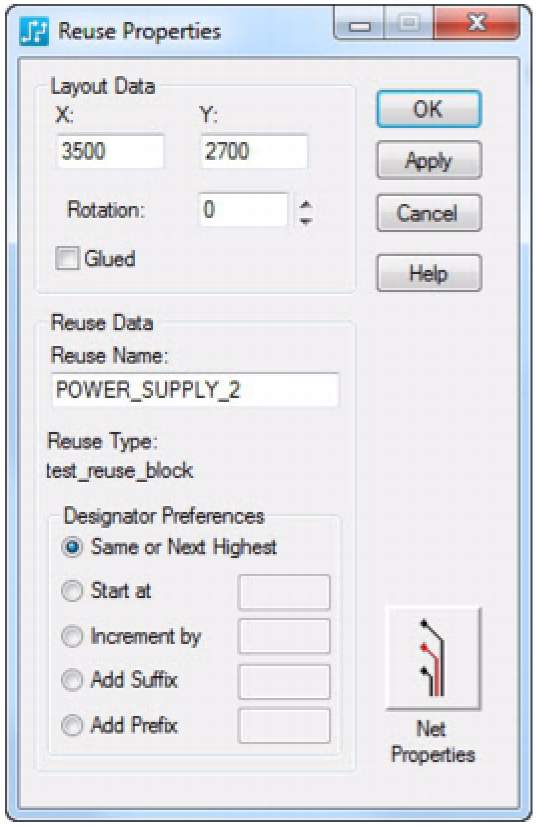


Figure 209. Reuse Properties Dialog Box of Reuse in the Design



Objects

Table 196. Reuse Properties Dialog Box Fields

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 196. Reuse Properties Dialog Box Fields (continued)

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 on page 1854. 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.
Reuse Type	Displays the Reuse Type name. **Tip** There is no way to display the reuse name and reuse type in the design.
Designator Preferences area — 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 section is not available unless you are in ECO mode with the ECO Toolbar open.	
Same or Next Highest	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.
Start at	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	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	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

Table 196. Reuse Properties Dialog Box Fields (continued)

Name	Description
	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 Prefix	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 on page 1491 where you can resolve netname conflicts between netnames in a physical design reuse and netnames in the design.

Related Topics

Modifying Reuse Properties in the Design

Routing Rules Dialog Box

To access:

- **Setup > Design Rules** menu item > choose a hierarchy level > **Routing** button
- PCB Decal Editor > **Setup > Design Rules** menu item > **Routing** button

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 SailWind Layout or from within the Decal Editor.

Description

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.

Figure 210. Routing Rules Dialog Box

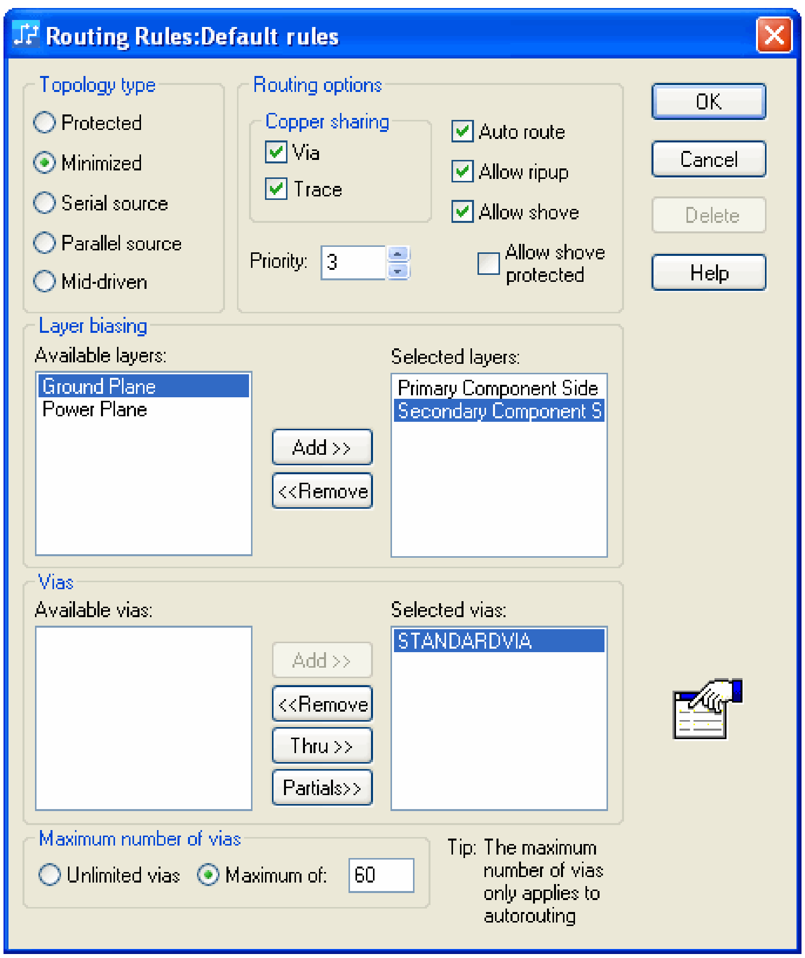
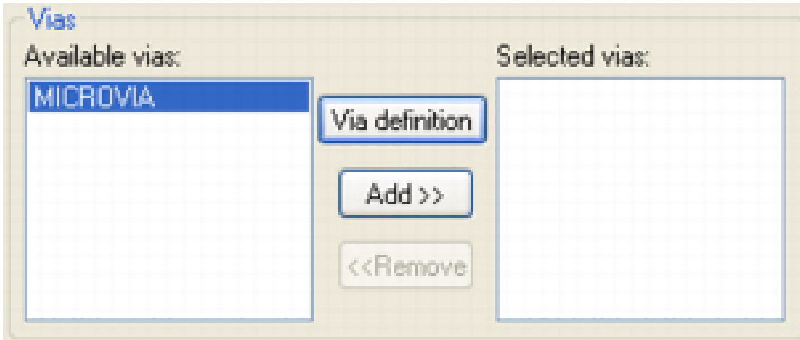


Figure 211. Vias Area of PCB Decal Editor Routing Rules Dialog Box



Objects

Table 197. Routing Rules Dialog Box

Area	Description
Topology type	Specifies the topology on page 1867 of the connections (ratsnest on page 1852) 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 on page 1837 but still allows routing of the net. For information about protecting the unroutes to prevent routing, see Protecting Unroutes in a Net and Protecting Unroutes in a Pin Pair. • **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 on page 474".

Table 197. Routing Rules Dialog Box (continued)

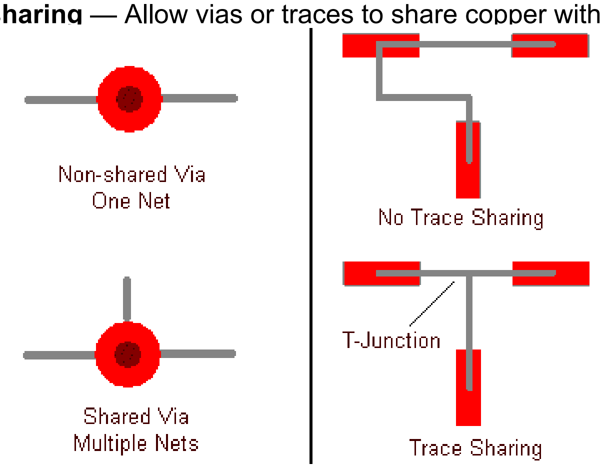
Area	Description
Routing options	• **Copper sharing** — Allow vias or traces to share copper with another object.  **Restriction:** This rule is used only in SailWind Router, although you can define this rule in SailWind Logic, SailWind Layout, or SailWind Router. • **Priority** — Assign a priority from 0 to 100. Nets with higher priority are routed first. **Restriction:** SailWind 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.
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. **Tip**

Table 197. Routing Rules Dialog Box (continued)

Area	Description
	• 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 on page 1833. 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 per net. 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

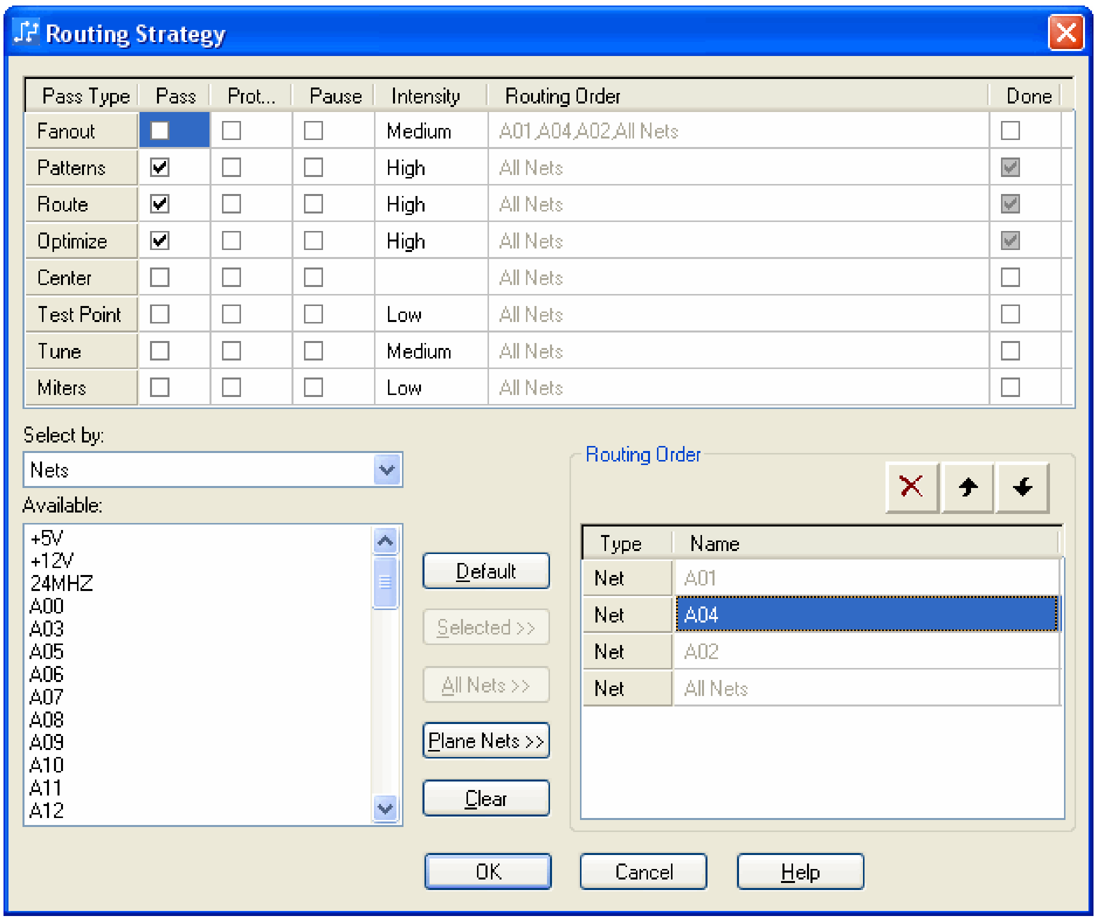
Protecting Unroutes in a Net

Protecting Unroutes in a Pin Pair

Routing Strategy Dialog Box

To access: **Tools > SailWind Router** menu item > **Setup** button

Use the Routing Strategy dialog box to define a strategy for autorouting your design in SailWind Router. You indicate what passes SailWind Router should perform, whether to protect the resulting traces, and what intensity to assign to objects.



Objects

Name	Description
Pass Type	Lists the types of passes that SailWind Router can make. SailWind 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.

Name	Description

	• **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, SailWind 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 While Interactively Routing”, “Assigning Test Points During Autorouting” and “Assigning Test Points After Autorouting” topics in the SailWind 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.
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 SailWind Router, the pass and the point within the pass are stored. When you resume autorouting, SailWind 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:

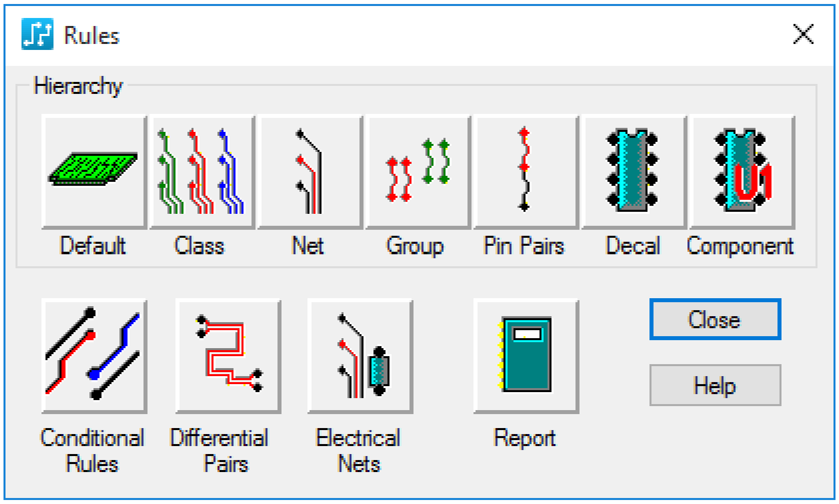
Name	Description
	• **Low** — Sets low effort, routing as quickly as possible with a low completion rate, using maximum vias and trace length. Use this setting when you want to complete autorouting quickly. The trade-off is your completion percentage. • **Medium** — Sets medium effort, routing more slowly 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.
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.

Name	Description
	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

To access: **Setup > Design Rules**

Use the Rules dialog box to assign place and route constraints to the design.



Objects

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 SailWind Layout; however, these rules are used in SailWind Router only.
Component	Opens the Component Rules Dialog Box. **Restriction:** You can define Component Rules in SailWind Layout; however, these rules are used in SailWind Router only.
Conditional Rules	Opens the Conditional Rule Setup Dialog Box.
Differential Pairs	Opens the Differential Pairs Dialog Box.

Name	Description
	Restriction: You can define Differential Pairs Rules in SailWind Layout; however, these rules are used in SailWind Router only.
Electrical Nets	Opens the “Electrical Net Rules Dialog Box” on page 1360.
Report	Opens the Rules Report Dialog Box.

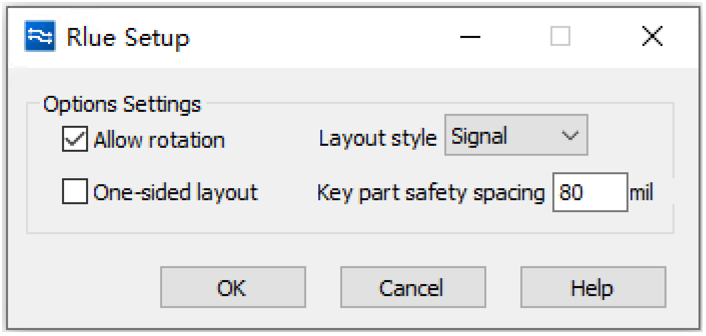
Related Topics

Design Rule Hierarchy

Rule Setup Dialog Box

To access: **AI > Intelligent Layout** menu item > **Rule Setup** button

Use this dialog box to define a layout strategy for componnets by intelligently identified group.



Objects

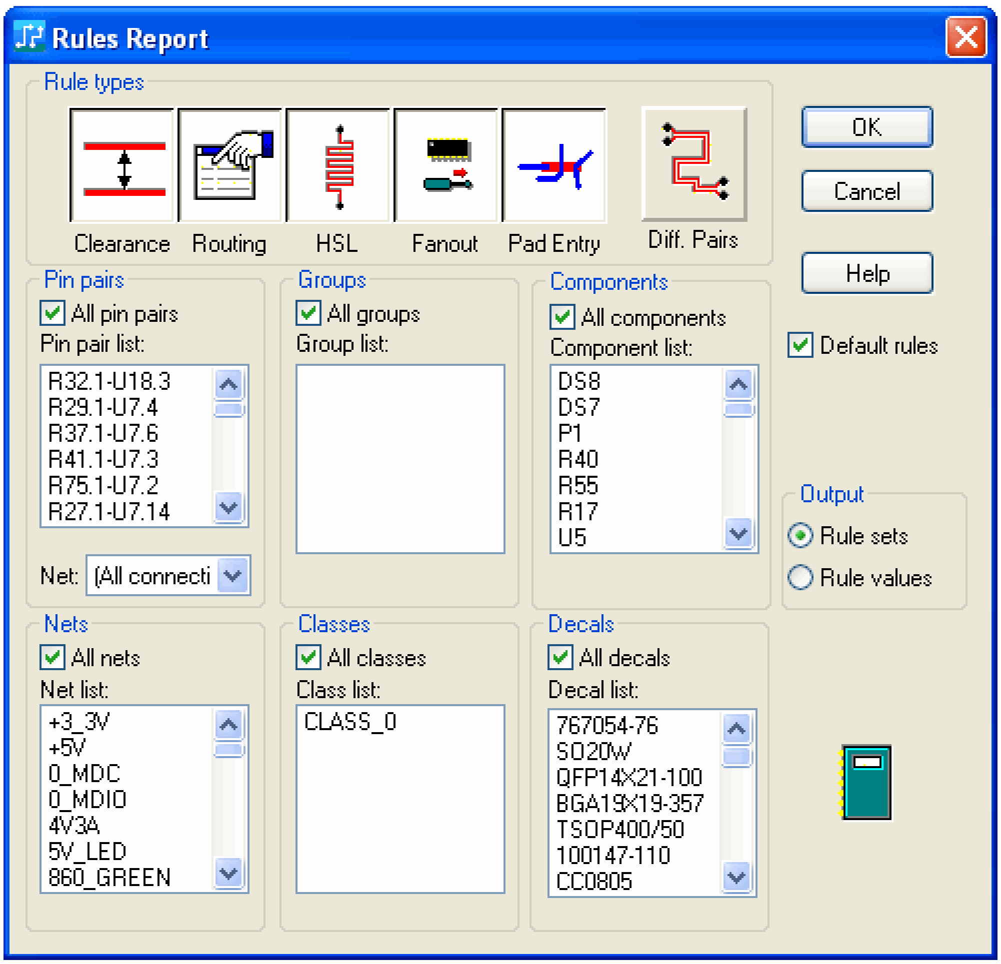
Name	Description
Allow rotation	Selects to allow rotating components by 90 degrees.
Layout style	Selects the layout style from the drop-down list.
One-sided Layout	Selects this option to enable the single-sided layout. By default, this option is unchecked for double-sided layout.
Key part safety spacing	Specifies the minimum clearance from the intelligently identified key component within a group.

Rules Report Dialog Box

To access:

- **Setup > Design Rules** menu item > **Report** button
- **Setup > Design Rules** menu item > choose a hierarchy level > **Report** button

Use the Rules Report dialog box to create reports containing information about design rules for the design.



Objects

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.

Name	Description
	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

Chapter 54
GUI Reference Elements S

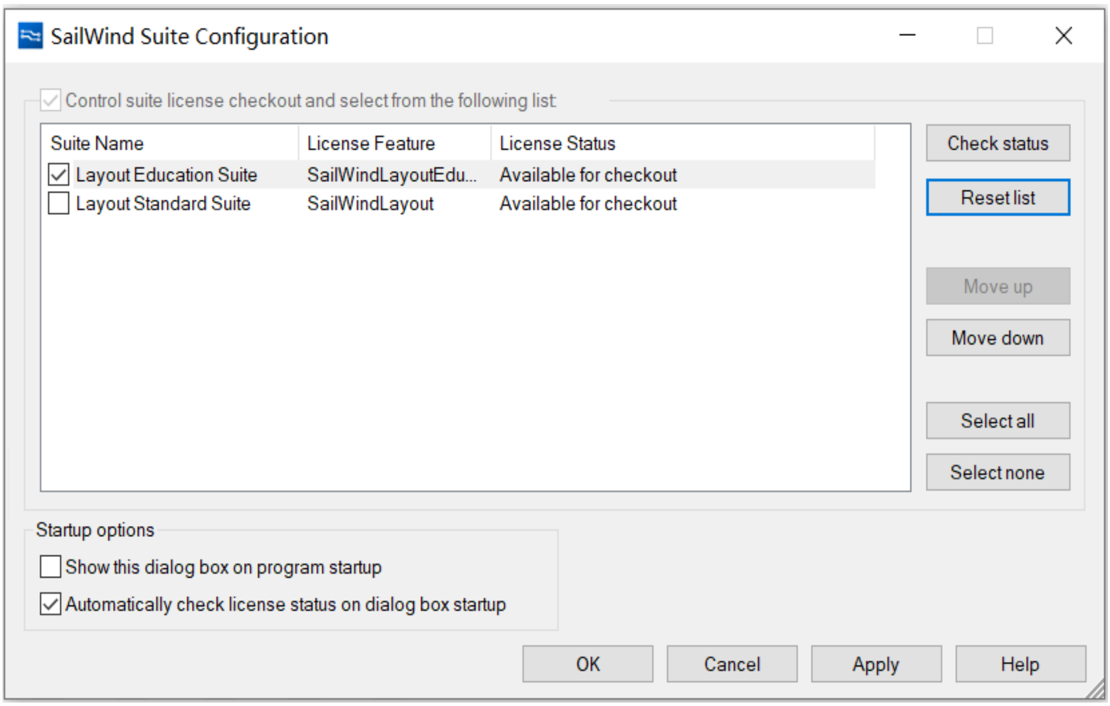
Read the sections that follow to learn more about dialog box elements in SailWind Layout.

SailWind Suite Configuration Dialog Box

To access:

- **Help > Installed Options** menu item > **Suite Configuration** button
- Optional: Opens on program startup

Use the SailWind Suite Configuration dialog box to manage SailWind Suite (composite) licenses.



Objects

Field	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.

Field	Description
Reset List button	Specifies to reset the list of suite licenses to only those detected in your licensing environment.
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 SailWind Suite Configuration dialog box when SailWind Layout starts.
Automatically check license status on dialog box startup	Specifies to check the status of the licenses when you open the SailWind Suite Configuration dialog box.

Related Topics

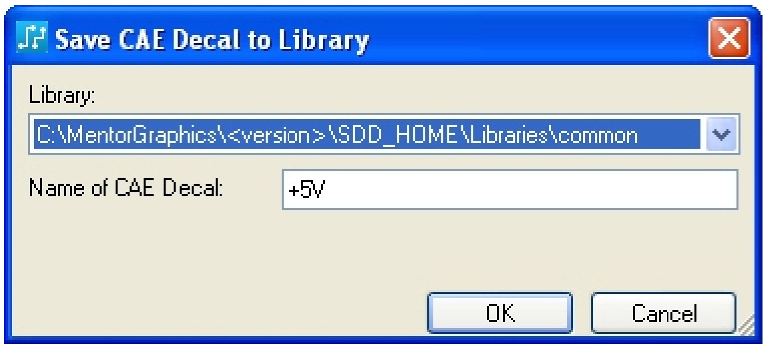
Installed Options Dialog Box, Options Tab

Checking Out Suite Licenses

Save CAE Decal to Library Dialog Box

To access: **File > Library** menu item > select a library > **Logic** filter button > select a CAE decal > **Copy**

Use the Save CAE Decal to Library dialog box to copy a CAE decal to another name or another library.



Objects

Field	Description
Library	Select the library for the copied CAE decal.
Name of CAE Decal	Type the name for the copied CAE decal.

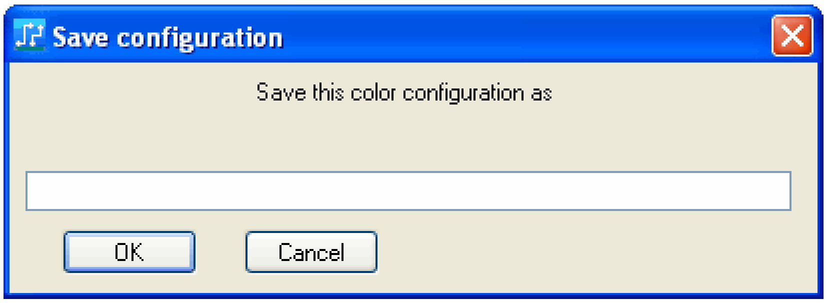
Related Topics

Copying a Library Item

Save Configuration Dialog Box

To access: **Setup > Display Colors** > **Save** button

Use the Save Configuration dialog box to save the color assignments and settings you have made in the Display Colors Setup dialog box.



Objects

Field	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

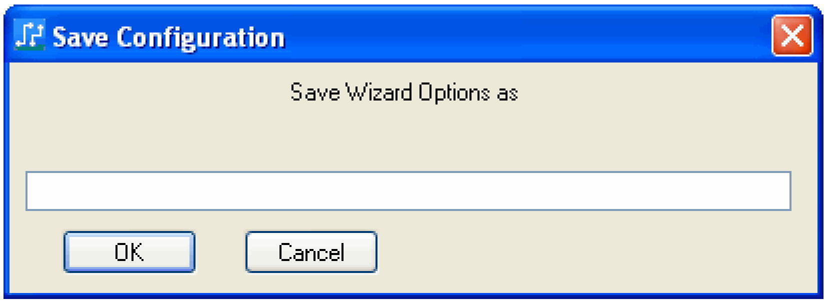
Display Colors Setup Dialog Box

Display Colors Setup Dialog Box in the Decal Editor

Save Configuration Dialog Box in the Decal Wizard Options

To access: Decal Wizard Options on page 1252 dialog box > **Save As** button

Use the Save Configuration dialog box to save the settings you have made in the Decal Wizard Option dialog box.



Objects

Table 198. Save (Decal Wizard Options) Configuration Dialog Box Content

Field	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 on page 1252.

Save Drafting Item to Library Dialog Box

To access:

- **File > Library** menu item > select a library > **2D Lines** filter button > select a line item > **Copy**

- Select a drafting shape in the design > right-click > **Save to Library** popup menu item

Use this dialog box to add a drafting shape from the design into the library or to save a library item under a different name or into a different library.

Objects

Table 199. Save Drafting Item to Library Dialog Box Content

Field	Description
Library	Select the library for the copied line item.
Name of Item	Type the name for the copied line item.

Related Topics

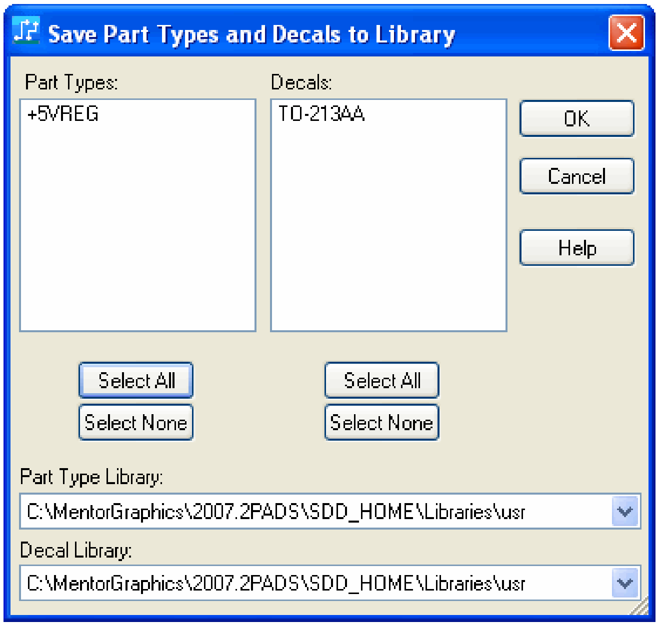
Copying a Library Item

Saving a Drafting Item to a Library

Save Part Types and Decals to Library Dialog Box

To access: Select one or more components in the design> right-click > **Save to Library** popup menu item

Use this dialog box to save modified decals or parts to libraries.



Objects

Field	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.

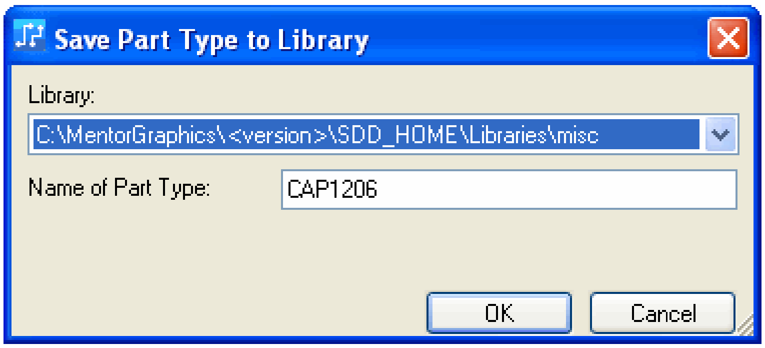
Related Topics

Saving Modified Decals and Parts to Libraries

Save Part Type to Library Dialog Box

To access: **File > Library** menu item > select a library > **Parts** filter button > select a part type > **Copy**

Use this dialog box to copy a Part Type to another name or another library.



Objects

Field	Description
Library	Select the library for the copied part type.
Name of Part Type	Type the name for the copied Part Type.

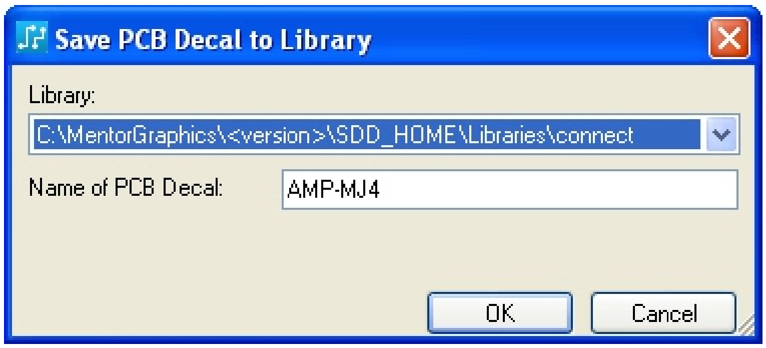
Related Topics

Copying a Library Item

Save PCB Decal to Library Dialog Box

To access: **File > Library** menu item > select a library > **Decal** filter button > select a PCB decal > **Copy**

Use this dialog box to copy a PCB decal to another name or another library.



Objects

Field	Description
Library	Select the library for the copied PCB decal.
Name of PCB Decal	Type the name for the copied PCB decal.

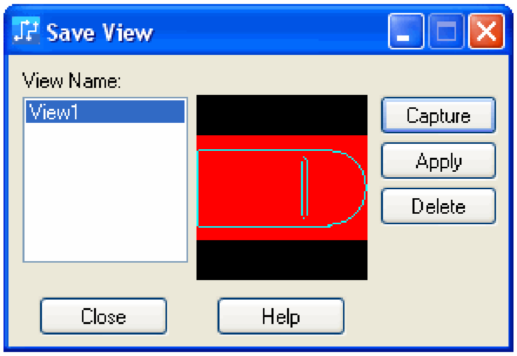
Related Topics

Copying a Library Item

Save View Dialog Box

To access: **View > Save View** menu item

Use the Save View dialog box to save a work area view. You can save up to nine different views.



Objects

Field	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

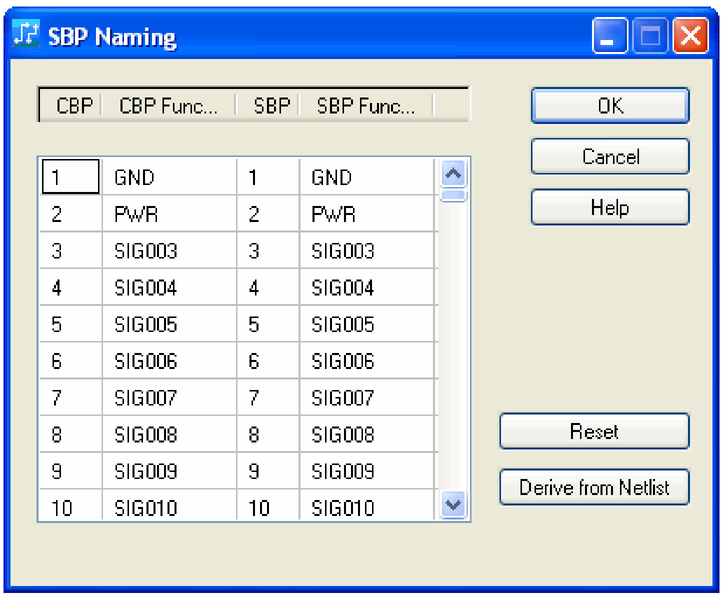
Saving and Restoring a View

SBP Naming Dialog Box

To access: **BGA Toolbar > Wire Bond Wizard** button > **SBP Naming** button

Use the SBP Naming dialog box to specify the numbers and functions of all newly created substrate bond pads, if different from the defaults.

Note:
This information applies only to the BGA toolkit.



Objects

Field	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

Setting SBP Names

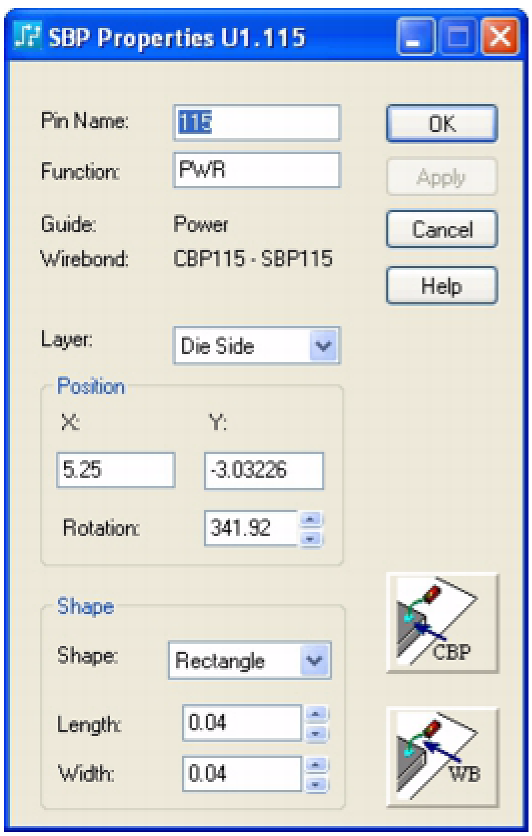
SBP Properties Dialog Box

To access: **BGA Toolbar > Wire Bond Editor** button > select an SBP > right-click > **Properties** popup menu item

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 the **Properties** popup menu item to edit the SBP Properties.

Restriction:
This information applies only to the BGA toolkit.

Note:
If you have multiple objects selected, only those properties that are common to all selected objects will appear.



Objects

Field	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.
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

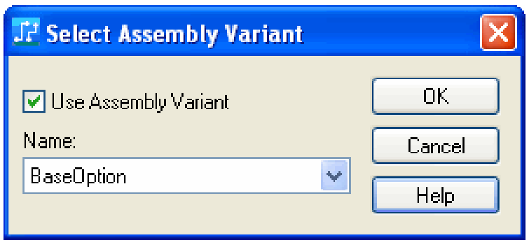
Editing Substrate Bond Pad Properties

Select Assembly Variant Dialog Box

To access:

- **File > CAM** menu item > **Add** button > select Assembly from the Document Type list > **Assembly** button
- **File > CAM** menu item > select a document name > **Edit** button > select Assembly from the Document Type list > **Assembly** button

You can create assembly drawings of your assembly variants by selecting a variant in the Select Assembly Variant dialog box.



Objects

Field	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 on page 1853.
Name	Specifies the variant you want to use.

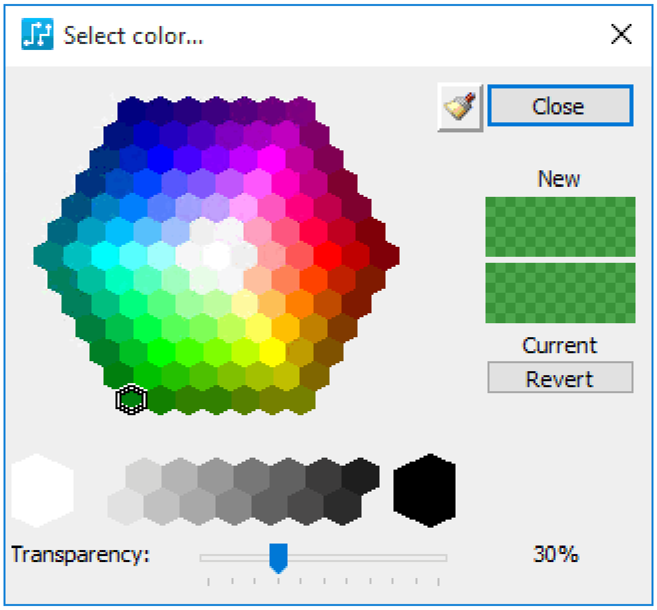
Related Topics

Creating CAM Outputs to Manufacture Your PCB

Select Color Dialog Box for 3D Options and Objects

To access: **SailWind 3D window > 3D General Toolbar > 3D Display Control button > click a color tile for a model listed under Options or Objects**

Set the color and transparency for Options and Objects in the 3D Display Control dialog box. The color tile might show the letter "P" if no custom color is assigned, indicating that the model is displaying the default photorealistic color.



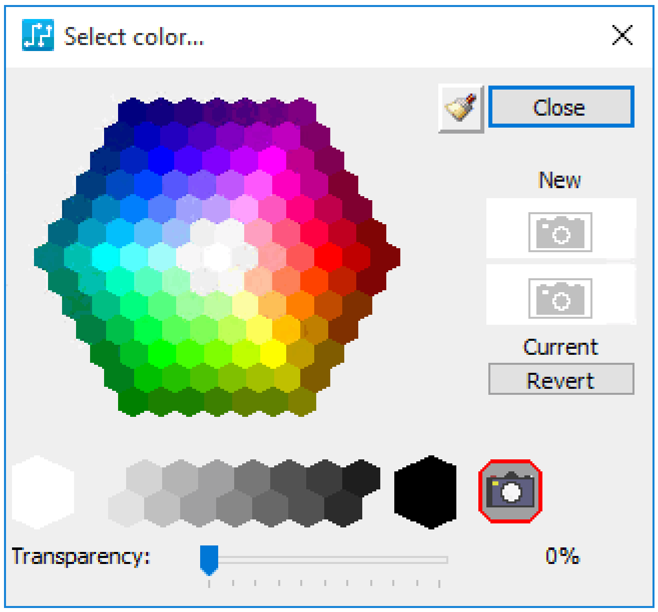
Objects

Object	Description
Color and Grayscale palettes	Choose a color or a scale of gray from the palettes. The color appears in the New color preview.
Transparency slider	Move the slider to set the transparency of the assigned color. The transparency appears in the New color preview.
 button	Click and apply the selected color to other color tiles in the 3D Display Control. The Select color dialog remains open.
Current and Previous color preview	Previews the new color selected and displays the current color to be replaced. Shows the camera icon if the color of the object is set to photo-realistic. Previews the transparency of the color also.
Revert button	Cancels a new color selection and replaces it with the Current color.

Select Color Dialog Box for 3D Models

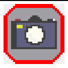
To access: **SailWind 3D window > 3D General Toolbar > 3D Display Control button > click a color tile for a model listed under Components, Assemblies, Mechanicals, or PCBs**

Set the color and transparency for 3D Models listed in the 3D Display Control dialog box. The color tile might show the letter “P” if no custom color is assigned, indicating that the model is displaying the default photorealistic color.



Objects

Object	Description
Color and Grayscale palettes	Choose a color or a scale of gray from the palettes. The color appears in the New color preview.
Transparency slider	Move the slider to set the transparency of the assigned color. The transparency appears in the New color preview.
 button	Click and apply the selected color to other color tiles in the 3D Display Control. The Select color dialog remains open.
Current and Previous color preview	• Previews the new color selected and displays the current color to be replaced. • Shows the camera icon if the color of the object is set to photo-realistic. • Previews the transparency of the color also.
Revert button	Cancels a new color selection and replaces it with the Current color.

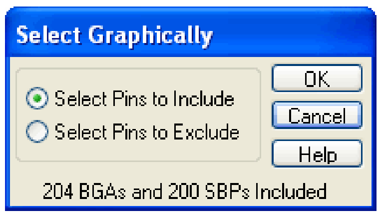
Object	Description
 button	Applies a photo-realistic color to the model.

Select Graphically Dialog Box

To access: **BGA Toolbar** button > **Route Wizard** button > **Select Pads** tab > **Select Graphically** button

Graphical Selection mode has two modes, namely “Include” and “Exclude.” The default mode is Include mode. The work area highlights pins that are currently included on the Substrate Bond Pads and BGA Pins lists.

Restriction:
This information applies only to the BGA toolkit.



Objects

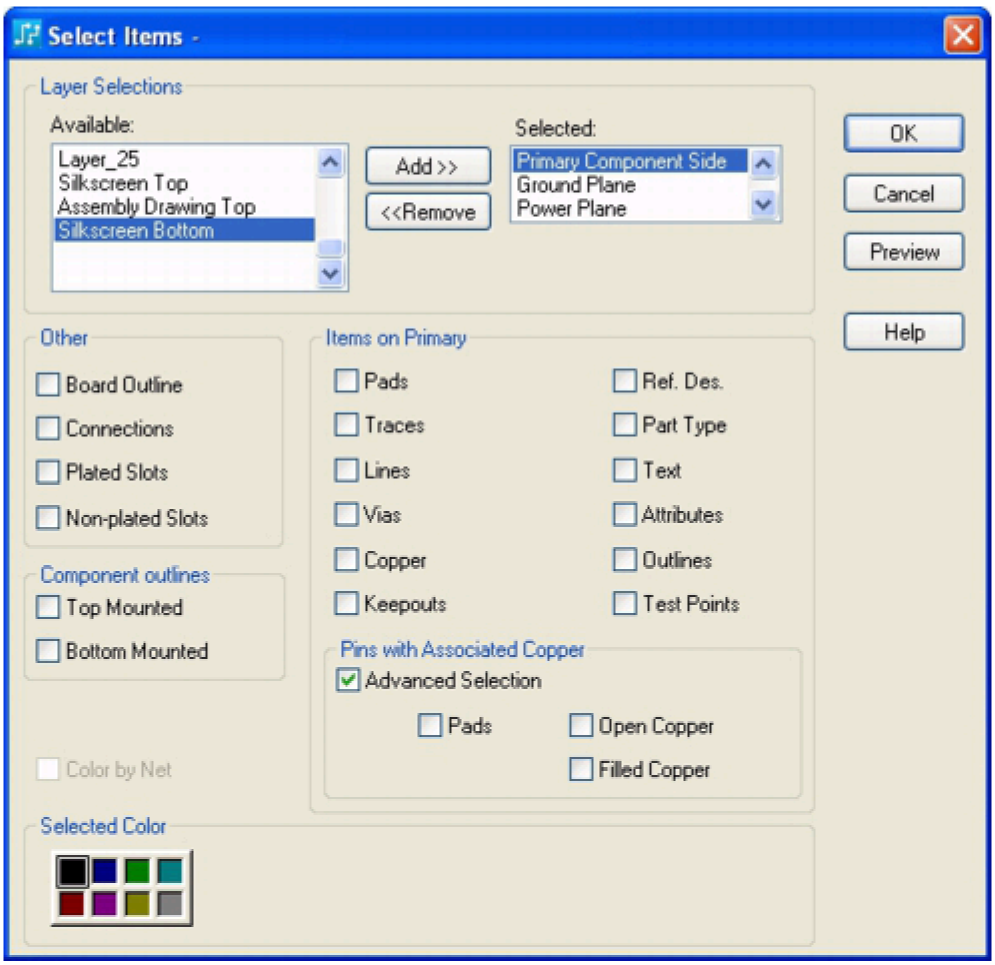
Field	Description
Select Pins to Include/Exclude	Places Graphical Selection in Include or Exclude mode.
Included/Excluded BGAs and SBPs	Lists the number of BGA pads and SBP pins currently selected for inclusion in or exclusion from processing.

Select Items Dialog Box

To access:

- **File > CAM** menu item > **Add** button > **Layers** button
- **File > CAM** menu item > select a document name > **Edit** button > **Layers** button

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.



Objects

Field	Description
Available list	Lists the layers available for display in the output.

Field	Description
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.
Items on Primary area	Select a check box to enable the display of the object from the Selected layer. • **Pads—** Displays pads on the selected layer. • **Traces—** Displays traces on the selected layer. • **Lines—** Displays 2D lines on the selected layer. • **Vias—** Displays vias on the selected layer. • **Copper—** Displays copper on the selected layer. • **Keepouts—** Displays keepouts on the selected layer. • **Ref. Des.—** Displays Reference Designator labels on the selected layer. • **Part Type—** Displays Part Type labels on the selected layer. • **Text** — Displays text on the selected layer. **Note:** **Exception:** 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. • **Attributes—** Displays Attribute labels on the selected layer. **Note:** 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 on page 590. • **Outlines—** Displays component outlines on the selected layer. • **Test Points** — 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.

Field	Description
Pins with Associated Copper area	CAM interprets pads and other objects differently when they are associated with copper in the PCB Decal Editor. Interpretation is as follows: • Terminals with associated copper 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. 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. Open copper is drawn using the Path setting. • **Filled Copper** — displays the closed copper shapes that are associated to pins. Filled copper is drawn using the Polygon, Circle, and Rectangle settings. **Tip** Once a pin has copper associated with it, it receives special handling, even if the layer you are outputting is not specifically the one that the associated copper is placed on. A through hole pin with associated copper is treated as a via in CAM so you can choose to enable vias on the solder mask layer for "Items on primary" or you can enable the "Advanced Selection" check mark for the solder mask layer and add a check mark for pads. For more information, see "Associating Copper with Terminals".
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. **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.
Selected Color	Provides a palette of grayscale or colors to use for objects in the output document. This helps distinguish items in the printout or pen plot. **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. 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.

Field	Description
	Tip Click a color in the palette and then click the box beside a design object. You must select the check box of a design object before the object color swatch appears. 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 document with selections assigned in this dialog.

Related Topics

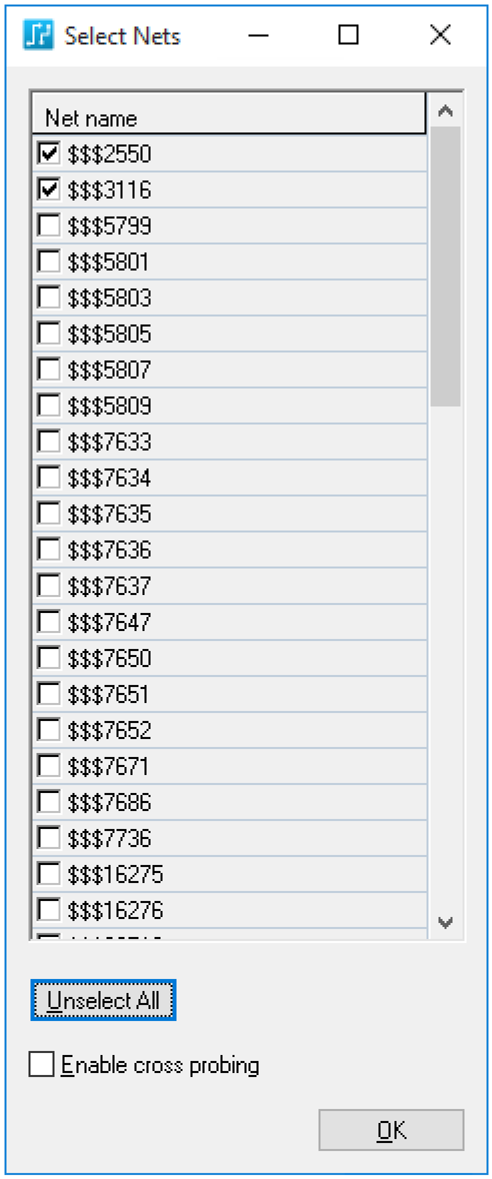
Colors in CAM Documents

Creating CAM Outputs to Manufacture Your PCB

Select Nets Dialog Box

To access: **SailWind 3D window > 3D General Toolbar > Export button > select one of the Copper Element Options > Selected Nets > Choose Nets**

Limit the export of pads, planes or traces to specific nets in a SAT or STEP file from the SailWind 3D window by selecting their nets.



Objects

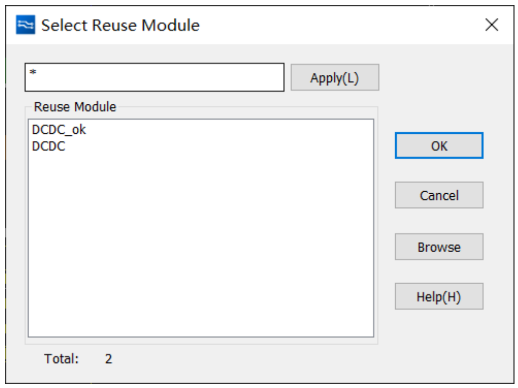
Object	Description
Net name list	Lists the nets in the design. Unconnected pins display as (Net0).

Object	Description
Unselect All	Clears the check boxes of all nets in the Net name list.
Enable cross probing	Enables you to select nets in the 2D design space instead of the dialog.

Select Reuse Module Dialog Box

To access: **Design Toolbar > Make Like Reuse**

Use this dialog box to select the reuse type, based on which to make a new physical reuse design.



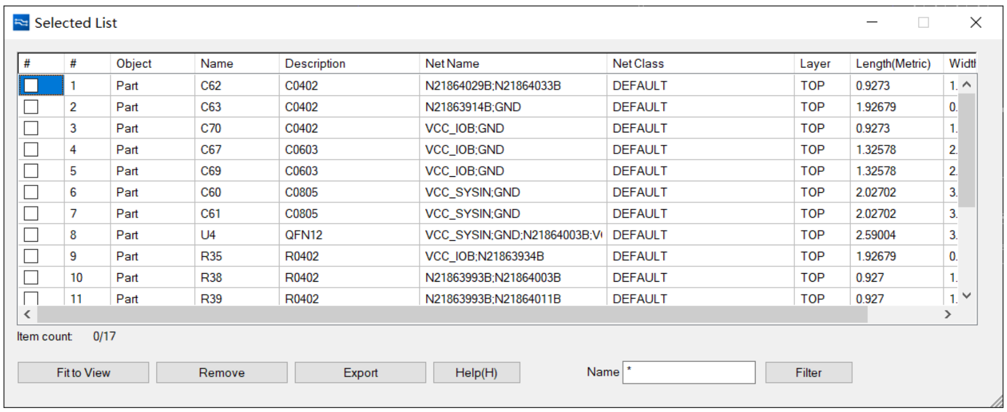
Objects

Object	Description	Remark
Filter box	Used to filter the reuse type within the design. To filter the reuse, type a wildcard or expression in the box and click **Apply** or press the **Enter** key.	
Reuse Module list	Lists the reuse type within the design or matched by the filter.	
Total	Displays the total number of what in the Reuse Module list.	
OK	Opens the Matching Result Dialog Box. Select a reuse type in the list prior to clicking the **OK** button.	You can select a reuse in either of the two ways.
Browse	Opens the "Open Reuse File" dialog box, in which to select a reuse file (.reu) locally and click **Open** to open the Matching Result Dialog Box.	

Selected List Dialog Box

To access: With object(s) selected in the workspace, right-click to choose **Selected List** in the pop-up menu

Use this dialog box to filter what you want from the selected more efficiently and precisely.



Objects

Table 200. Parameter Description

Field	Description	Remark
☐	Selects the check boxes for the object(s) you want, which will be highlighted in the highlight color in the workspace.	
Object	Displays the object type.	
Name	Displays the object name: reference designators for parts and blank cells for other objects requiring no name.	
Description	Displays the object information: PCB decal for parts and coordinate location for other objects.	
Net Name	Displays all nets with which the object is associated.	Blank cells indicate that no such information is found.
Net Class	Displays all net classes to which the nets belong.	
Layer	Displays the layer(s) on which the object is placed.	
Length	Displays the length and width of the objects, in current design units.	
Width		
Fit to View	Fits selected objects into the workspace.	

Table 200. Parameter Description (continued)

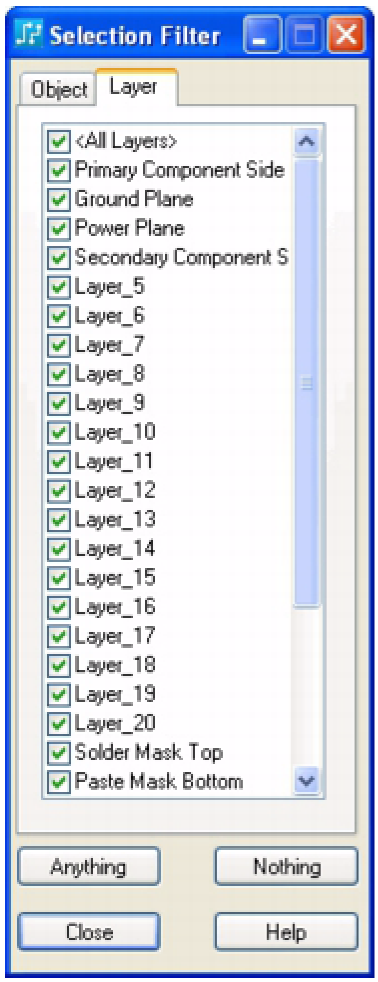
Field	Description	Remark
Remove	Removes the selected item(s) from the list, which will then be deselected in the workspace.	To clear the list, you can also click in an empty area of the workspace.
Export	Exports what currently in the list to a local CSV file.	
Name	Filters objects by name, with operations as follows: 1. Type a wildcard or expression in the Name box. If you do not want to restrict the results with a filter, you can display all items by typing * (asterisk). 2. Click **Filter** or press the Enter key.	

Selection Filter Dialog Box, Layer Tab

To access: **Edit > Selection Filter** menu item > **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.



Objects

Field	Description
Layers list	Specifies the layers on which you want to enable selection.

Field	Description
Anything	Specifies that you want to select anything in the design. **Note:** **Exception:** Clusters, unions, stitching vias, pin pairs, nets, and board outline shapes are not selected.
Nothing	Specifies that you do not want to select anything in the design.

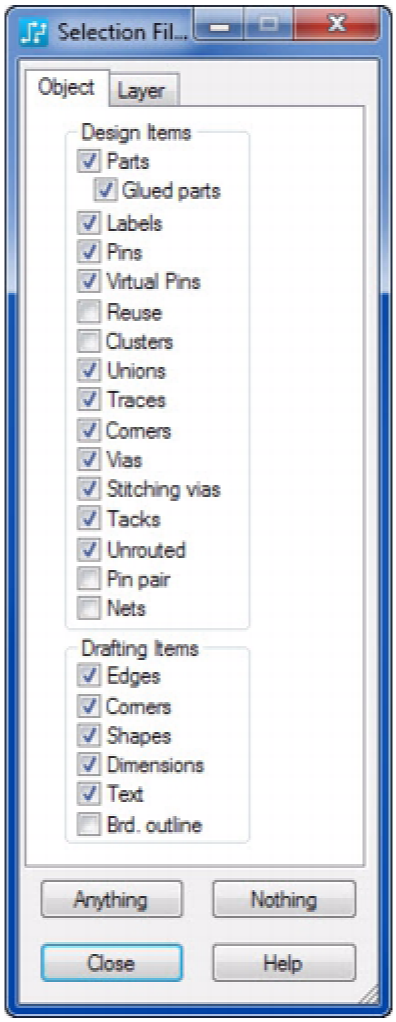
Related Topics

The Selection Filter

Selection Filter Dialog Box, Object Tab

To access: **Edit > Selection Filter** > **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.



Objects

Field	Description
Design Items	Specifies the design items you want to be able to select in the design. **Tip**

Field	Description
	• 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. **Note:** **Exception:** Clusters, unions, stitching vias, pin pairs, nets, and board outline shapes are not selected.
Nothing	Specifies that you do not want to select anything in the design.

Related Topics

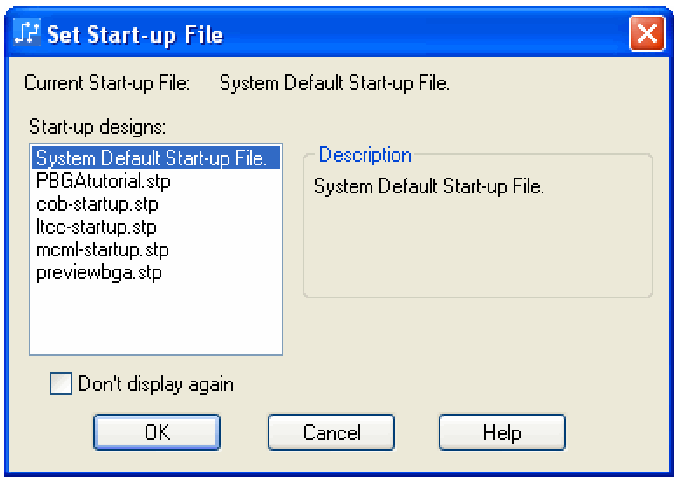
The Selection Filter

Set Start-up File Dialog Box

To access: **File > Set Start-up File** menu item

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.

Note:
The startup file affects only new designs and specifying a new startup file does not affect existing designs.



Objects

Field	Description
Current Start-up File	The name of the start-up file you are 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

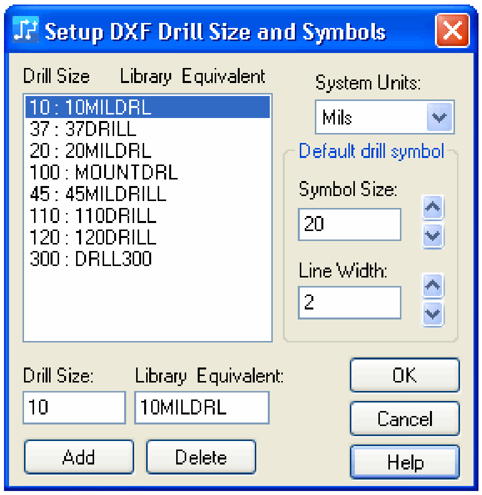
Specifying the Start-up File

Start-up Files

Setup DXF Drill Size and Symbols Dialog Box

To access: **File > Export** > select DXF File > **Save** button > **Setup** button

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.



Objects

Field	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 (+).

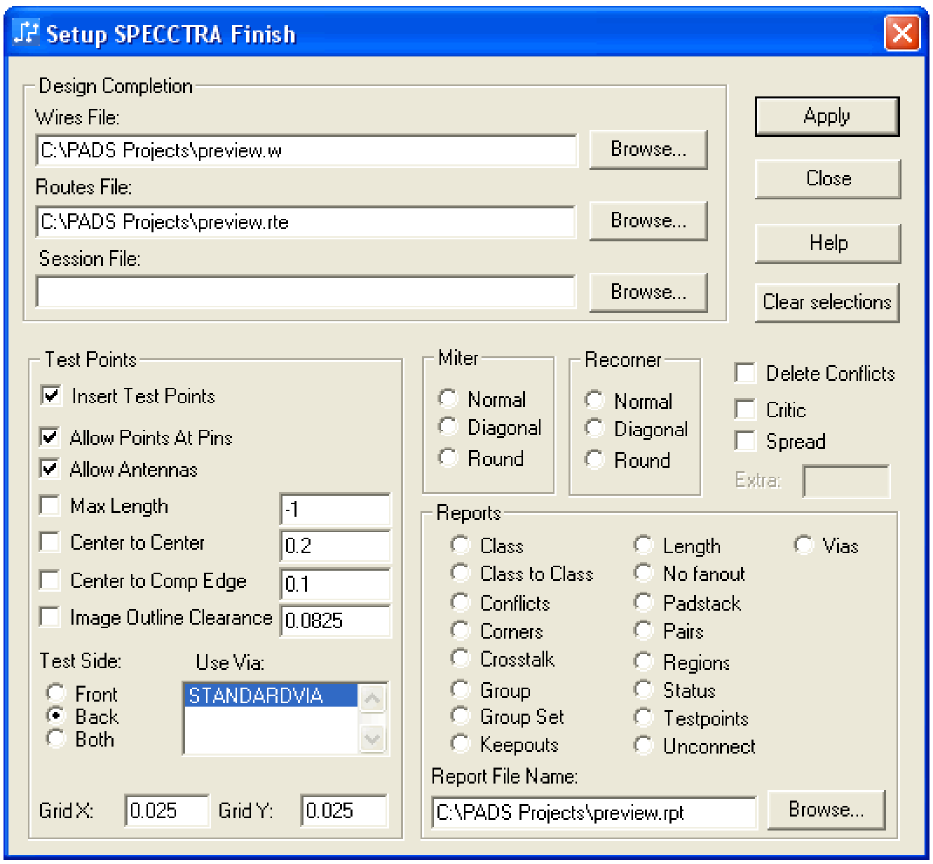
Related Topics

Specifying DXF Drill Sizes and Symbols

Setup SPECCTRA Finish Dialog Box

To access: **File > Export** > select SPECCTRA Files > **Save** button > in the SPECCTRA Link > **DO File** button > **Finish** button

Use the Setup SPECCTRA Finish dialog box to specify output file locations. Use this dialog to also set instructions for the SPECCTRA router regarding actions to perform when routing is completed, such as running the mitering pass, running re-cornering, and inserting test points.



Objects

Field	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.
Session File	Specifies the location of the session file.

Field	Description
	Tip Click Browse to locate the file.
Test Points area	Specifies the options you want for test points installed by SPECCTRA. See also: "Data Passed 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

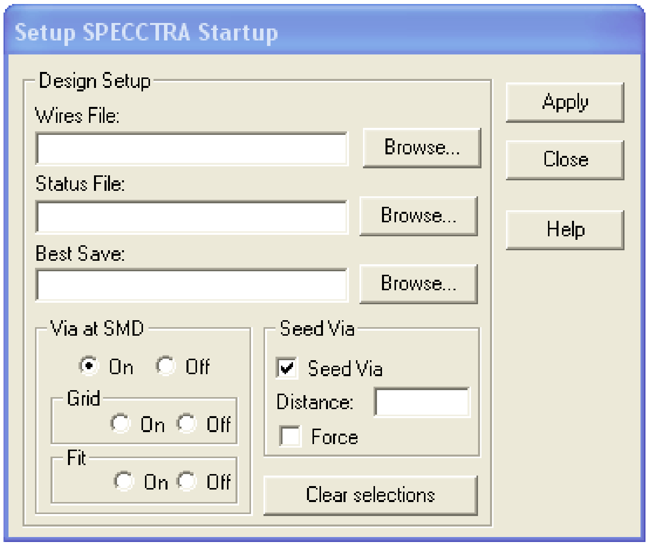
To access: **File > Export** menu item > select SPECCTRA Files > **Save** > in the SPECCTRA Link, type a name or browse to the file you want in the DO File Name box > **Do File** button > **Startup** button

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.

Description

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.



Objects

Field	Description
Wires File	Specifies the wires file. Click Browse to locate the file.
Status File	Specifies the status file. Click Browse to locate the file.
Best Save	Specifies the best save file. 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.

Field	Description
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.

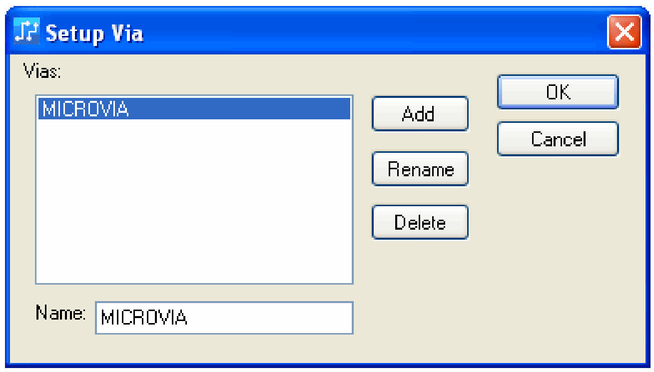
Related Topics

Setting up SPECCTRA .do File Startup Options

Setup Via Dialog Box

To access: PCB Decal Editor > **Setup > Design Rules** menu item > **Routing** button > **Via definition** button

Use the Setup Via dialog box to add new via names to the Available Vias list. The Setup Via dialog box does not reference via padstacks in the design. These via names should reference the real via names in your design, where the internal padstack structure is defined.



Objects

Field	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.
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

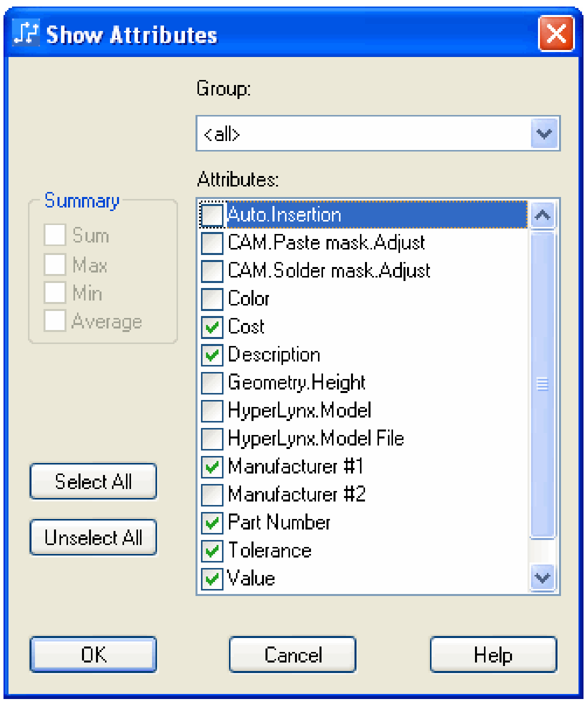
Creating Via Routing Rules in the Decal Editor

Show Attributes Dialog Box

To access: **Edit > Attribute Manager** menu item > **Show** button

Use the Show Attributes dialog box to select attributes to list in the multi-column list of the Attribute Manager.

CAUTION:
Hidden attributes are not available to the Attribute Manager and are not listed in the Show Attributes dialog box.



Objects

Field	Description
Group	Filters the Attributes list. You can choose an attribute group on page 1811 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.

Field	Description
	• **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 available only 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.

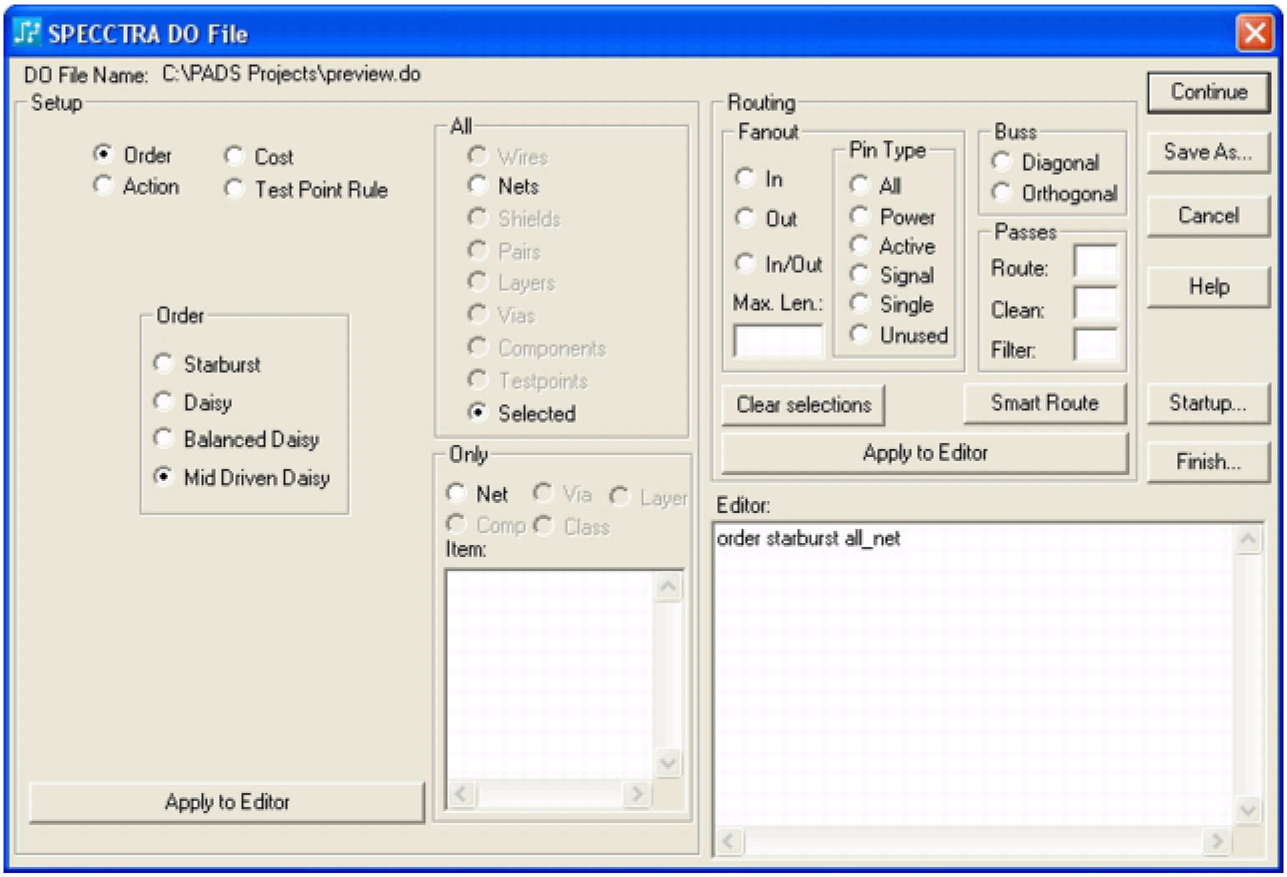
Related Topics

Attribute Manager

SPECCTRA DO File Dialog Box

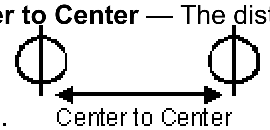
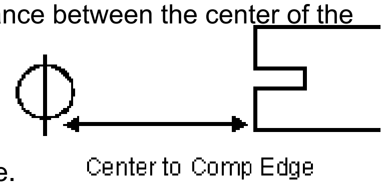
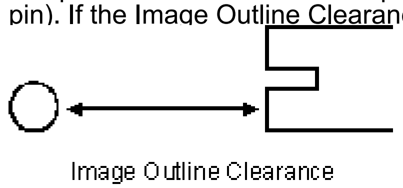
To access: **File > Export** > select SPECCTRA Files > **Save** > in the SPECCTRA Link, type a name or browse to the file you want in the DO File Name box > **Do File** button

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, it reads the *.do* file you specify in the SPECCTRA Link dialog box for editing.



Objects

Field	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*

Field	Description
	• **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: “Data Passed 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.
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.

Field	Description
	• **Use Via** — Uses vias as test points. • **Grid X, Y** — The test point grid. See also: "Options Dialog Box, Grids and Snap Category, Grids Subcategory on page 1537".
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.
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

To access:

- **File > Export** menu item > select SPECCTRA Files > **Save**
- Use Windows Explorer to navigate to your *C:\<install_folder>\SailWind<version>\Programs* directory, and double-click *pads2sp.exe*

When you start SPECCTRA from within SailWind Layout, the SPECCTRA Link dialog box enables you to load in and out of SPECCTRA automatically.

Note:
If you start SPECCTRA independently of SailWind Layout, use the Stand-alone SPECCTRA Link dialog box to load in and out of SPECCTRA manually on page 1012.

Figure 212. SPECCTRA Link Dialog Box - from Layout

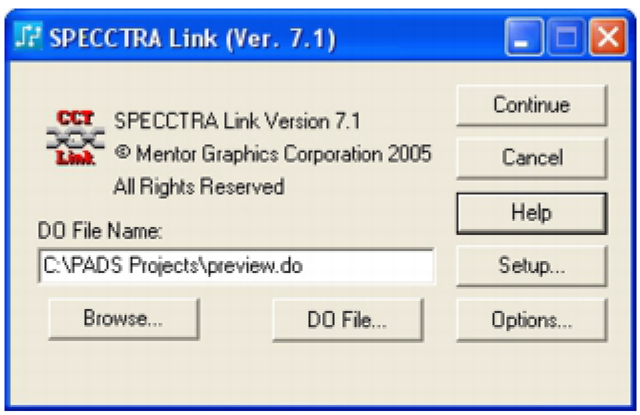
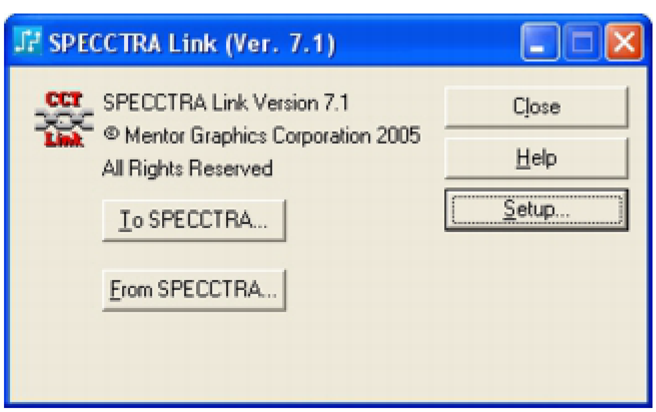


Figure 213. SPECCTRA Link Dialog Box - Stand-alone



Objects

Table 201. SPECCTRA Link Dialog Box Fields

Field	Description
DO File Name	Specifies the DO file you want to use. 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 on page 1727. **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

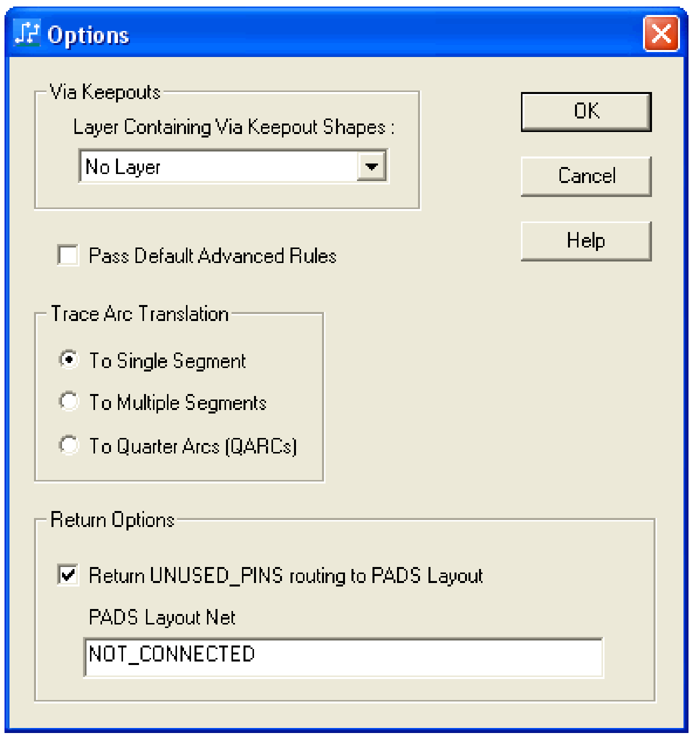
SPECCTRA Options Dialog Box

To access: **File > Export** menu item > select SPECCTRA Files > **Save** > in the SPECCTRA Link, click the **Options** button

The Options dialog box also appears when you click the Options button on the TO SPECCTRA dialog box (stand-alone), or the FROM SPECCTRA dialog box (stand-alone).

Description

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. For additional related information, see "Unused Pins Net".



Objects

Field	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: "SailWind Layout to SPECCTRA Rules Conversion".

Field	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 SailWind Layout. Type the name of the net in the SailWind 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 SailWind 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 SailWind Layout is 47 characters. You can use any alphanumeric characters except for brackets{ }, asterisks *, or spaces.

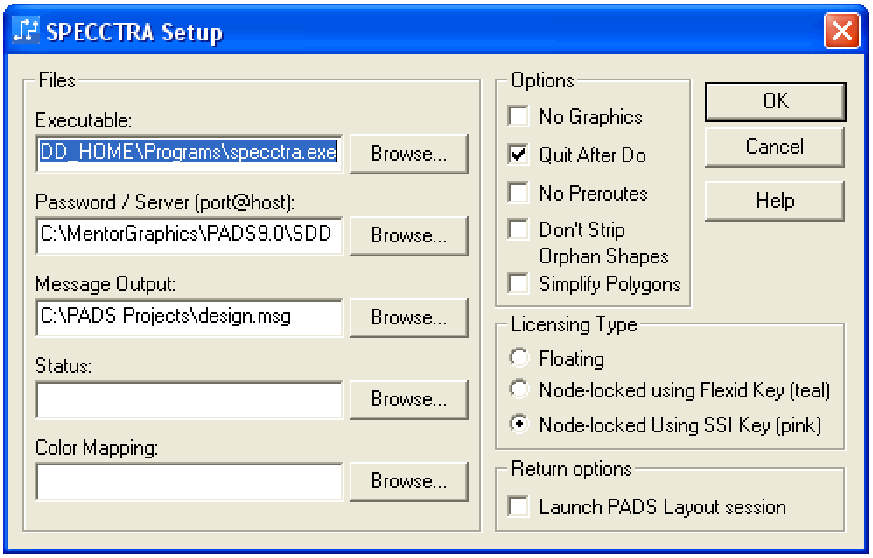
Related Topics

Setting SPECCTRA Options

SPECCTRA Setup Dialog Box

To access: **File > Export** menu item > select SPECCTRA Files > **Save** > in the SPECCTRA Link, click the **Setup** button

Use the SPECCTRA Setup dialog box to set SPECCTRA automatic startup information.



Objects

Field	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.

Field	Description
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.
Licensing Type area	Select the licensing type you want: Floating, Node-locked with Flexid Key, or Node-locked with SSI key.
Launch SailWind Layout session	Specifies to reload the routed design back into SailWind Layout after it processes the *.do* file commands.

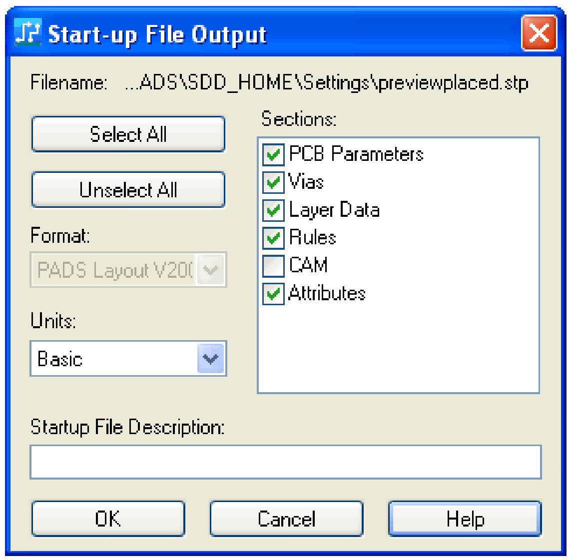
Related Topics

Setting the SPECCTRA Automatic Startup Information

Start-up File Output Dialog Box

To access: **File > Save as Start-up File**

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 save in the startup file.



Objects

Field	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

Field	Description
	• **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

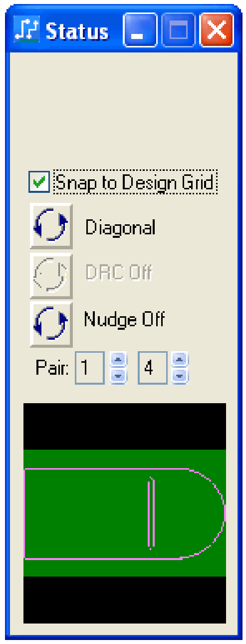
Specifying the Start-up File

Start-up Files

Status Dialog Box

To access: Ctrl+Alt+S

Use the Status window to view selection information and gain access to frequently used Options settings.



Objects

Field	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 on page 1537.
Line/Trace Angle	Toggles the Line/trace angle of the Design Options on page 1503.
On-line DRC	Toggles the On-line DRC setting of the Design Options on page 1503. **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 on page 1503.
Layer Pair	Toggles the Layer pair setting of the Routing Options on page 1542.
Postage Stamp	Displays a miniature of the current view.

Step and Repeat Dialog Box

To access: PCB Decal Editor > select one or more terminals, 2D line items, text items, copper items, copper cut outs, or keepouts > right-click > **Step and Repeat** popup menu item

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.

Description

Three types of Step and Repeat array replications are available and are controlled by the three tabs of the dialog box. The lower section of the dialog box is used to control the incrementing of pin numbers or text.

- **Linear Tab** — For planar replication, in the horizontal and vertical directions. Described in Table 202.
- **Polar Tab** — For angular replication, rotation around the decal origin. This is useful for creating polar arrays of terminals. Described in Table 203.
- **Radial Tab** — For radial replication, along the radial direction starting from the decal's origin. This is useful for creating polar arrays of terminals. Described in Table 204.
- **Pin numbering section** — Used to specify the incrementing format of pin numbers. Described in Table 205.
- **Text section** — Used to specify the incrementing of text. Described in Table 206.

Figure 214. Step and Repeat Dialog Box Linear Tab

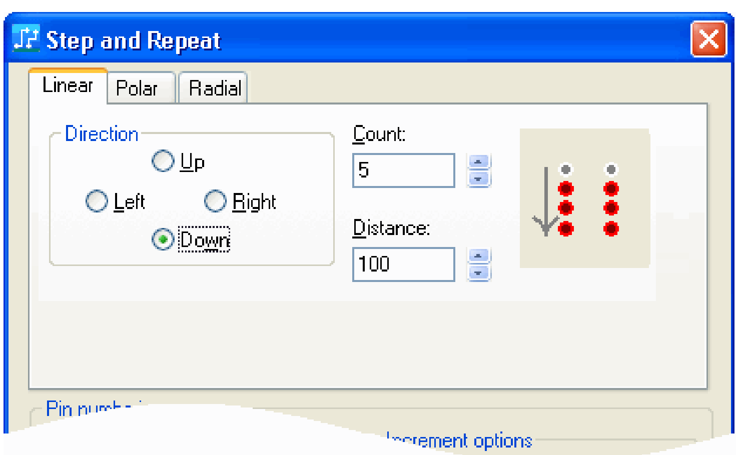


Figure 215. Step and Repeat Dialog Box Polar Tab

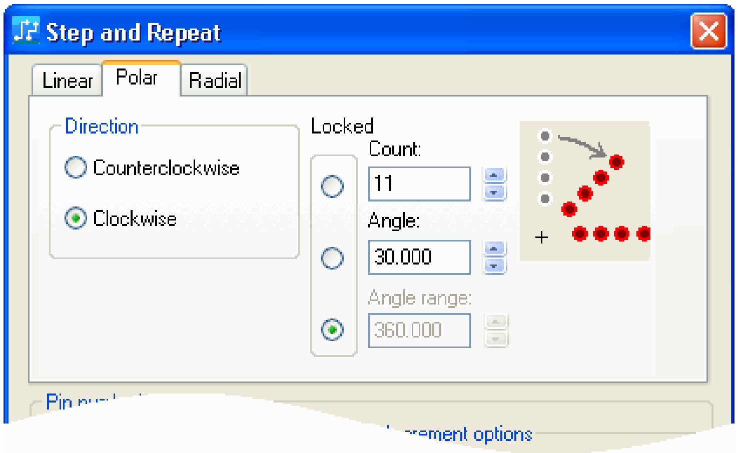


Figure 216. Step and Repeat Dialog Box Radial Tab

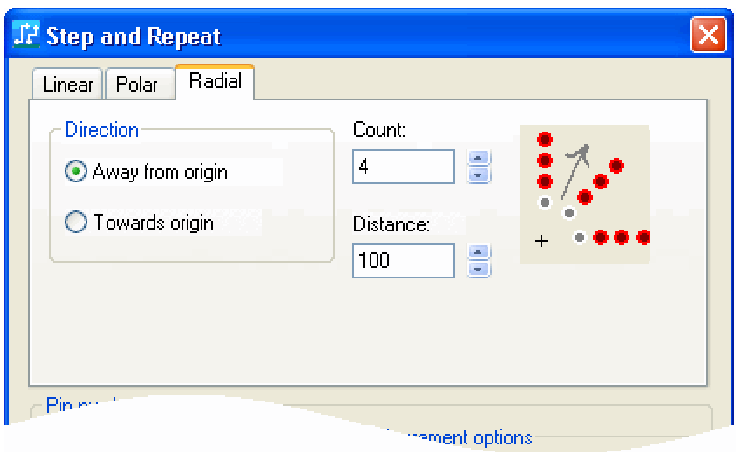


Figure 217. Step and Repeat Dialog Box - Pin Numbering

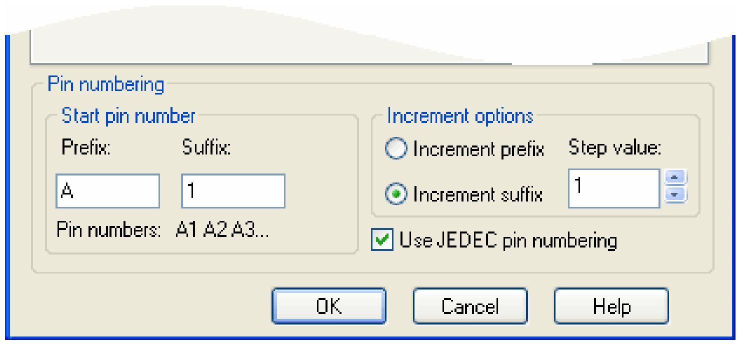
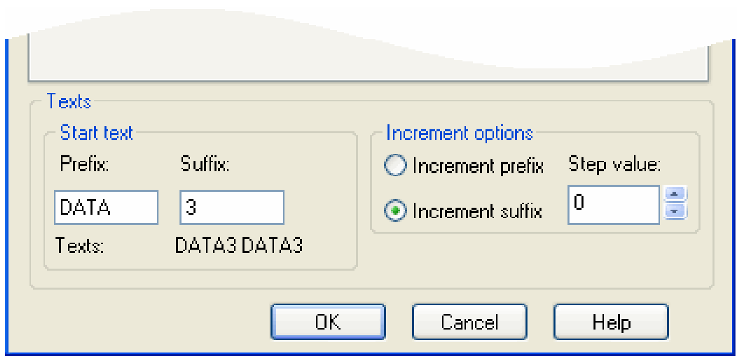


Figure 218. Step and Repeat Dialog Box - Texts



Objects

Table 202. 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.

Table 203. 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

Table 203. Step and Repeat Polar Tab Contents (continued)

Command	Description
	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. **Restriction:** • 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, SailWind 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.

Table 204. 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.

Table 205. 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.

Table 205. Pin Numbering Area Contents (continued)

Command	Description
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.

Table 206. 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.

Related Topics

Using Step and Repeat to Add an Array of Terminals

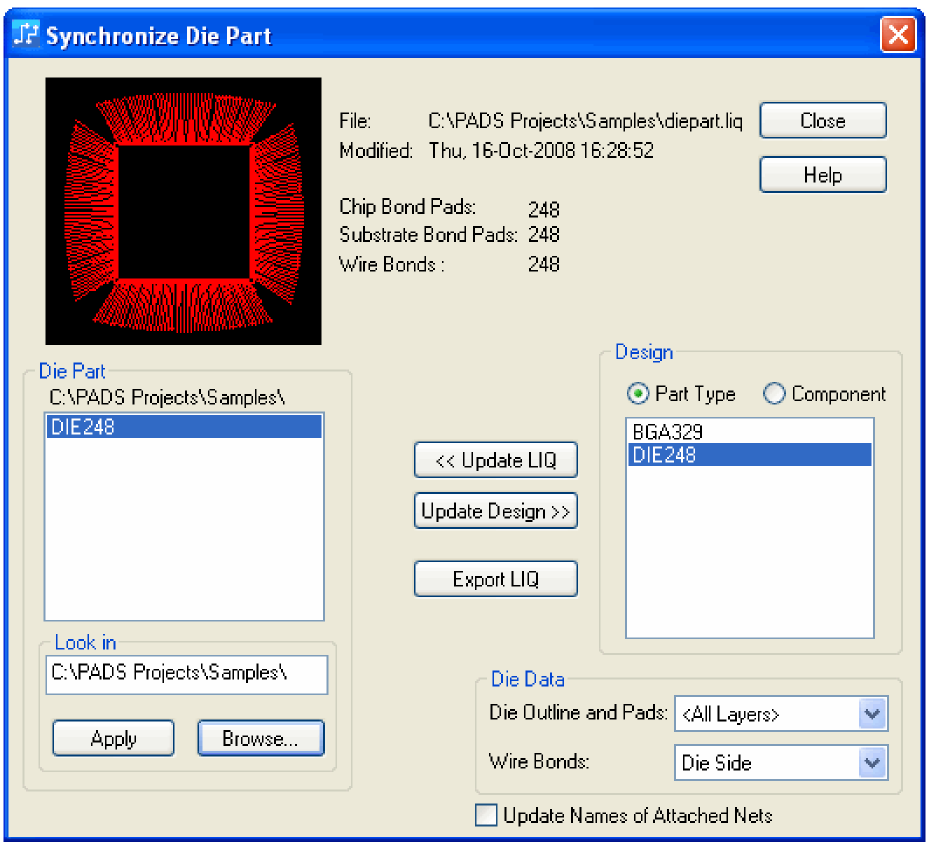
Synchronize Die Part Dialog Box

To access: **BGA Toolbar > Synchronize Die Parts** button

Library IQ is outmoded by the Wire Bond Wizard. This dialog box was used to synchronize die part data between SailWind Layout and Library IQ — a primitive Visual Basic tool to help reuse text-based die part data.

Restriction:
This information applies only to 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 SailWind Layout; so the original family designation (and reference designator) of these parts is lost in the export/import process.



Objects

Field	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.
Look in	Displays the die part search folder. 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\SailWind 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:

Field	Description
	• 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 does not 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.

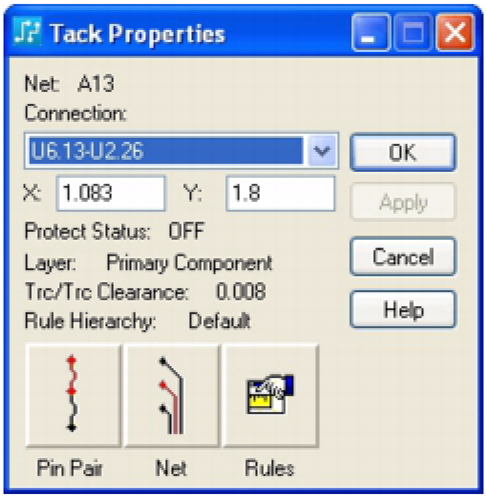
Chapter 55
GUI Reference Elements T Through Z

Read the sections that follow to learn more about dialog box elements in SailWind Layout.

Tack or Trace Corner Properties Dialog Box

To access: Select a trace corner > right-click > **Properties** popup menu item

Use the Tack or Trace Corner Properties dialog box to obtain information on and to modify a selected trace corner or tack.



Objects

Field	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 on page 1487.
Rules button	Opens the Pin Pair Rules Dialog Box.

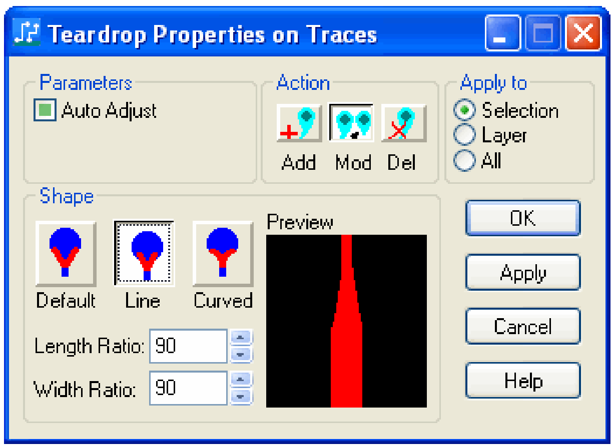
Related Topics

Modifying Trace Corner or Tack Properties

Teardrop Properties on Traces Dialog Box

To access: Select one or more traces to which the teardrop is attached > right-click > **Teardrop Properties** popup menu item

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.



Objects

Field	Description
Auto Adjust	Use Auto Adjust to set a custom length and width ratio. With Auto Adjust selected, SailWind 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.
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.
Shape area	
Default button	Creates a standard teardrop shape that is compatible with early SailWind Layout (that is, PowerPCB) versions. **Restriction:**

Field	Description
	• You cannot set a length or width ratio with a Default-shaped teardrop. • You cannot place default-shaped teardrops on square pads.
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. **Restriction:** This button is unavailable if you do not have the Analog license feature.
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. **Restriction:** This button is unavailable if you do not have the Analog license feature.
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. **Restriction:** This setting is not available for the Default teardrop shape.
Width ratio	Sets the width of the teardrop relative to the pad to which it is attached. You cannot set a ratio over 100. **Restriction:** This setting is not available for the Default teardrop shape.
Preview	Shows the currently configured teardrop.

Related Topics

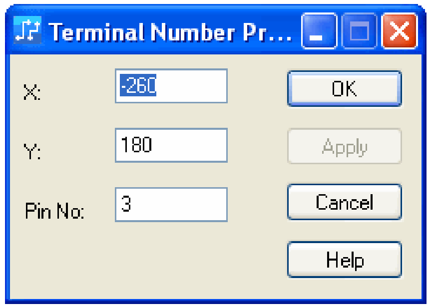
Modifying Teardrop Properties

Teardrops

Terminal Number Properties Dialog Box

To access: PCB Decal Editor > select a terminal number > right-click > **Properties** popup menu item

Use the Terminal Number Properties dialog box to modify the properties of the terminal number.



Objects

Field	Description
X, Y	The X,Y coordinates of the selected terminal number.
Pin No	The pin number of the selected terminal number

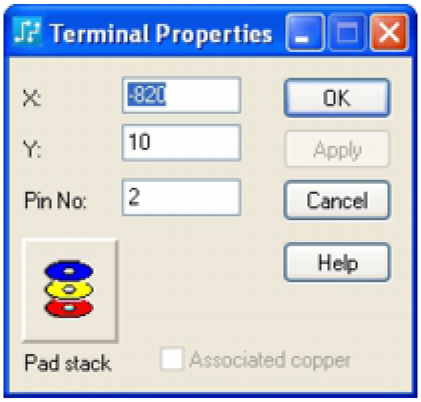
Related Topics

Modifying Terminal Number Properties

Terminal Properties Dialog Box

To access: PCB Decal Editor > select a terminal > right-click > **Properties** popup menu item

Use this dialog box to modify terminal properties.



Objects

Field	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

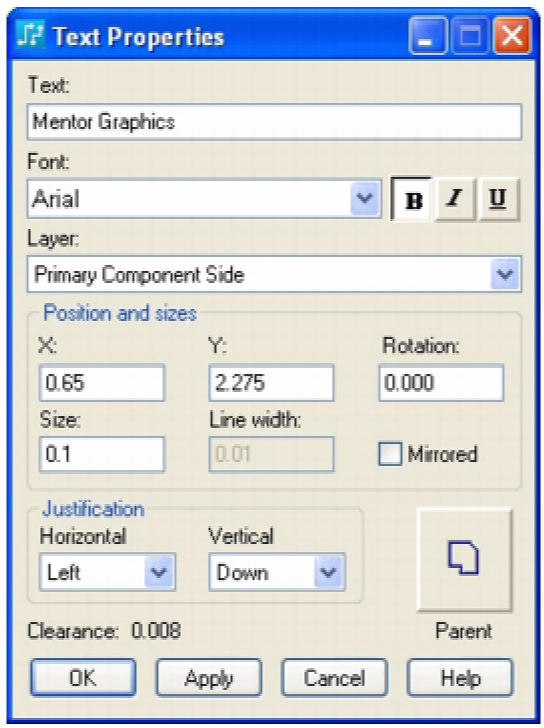
Modifying Terminal Properties

Text Properties Dialog Box

To access: Select text > right-click > **Properties** popup menu item

Use the Text Properties dialog box to modify free text. You can change the font, font style, layer assignment, orientation, rotation, size, line width, justification, and mirror settings. 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 "Creating Reusable Fabrication Notes" for a tip on saving multiple lines of text to the library for reuse.



Objects

Field	Description
Text	The text string you want to use. **Restriction:** There is a maximum of 128 characters per text string.
Font	The fonts available to you. **Tip**

Field	Description
	• 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.
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. **Tip** • 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 the **Justify Horizontally** popup menu item, and then clicking **Left**, **Center**, or **Right**; and by right-clicking and clicking the **Justify Vertically** popup menu item, 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.

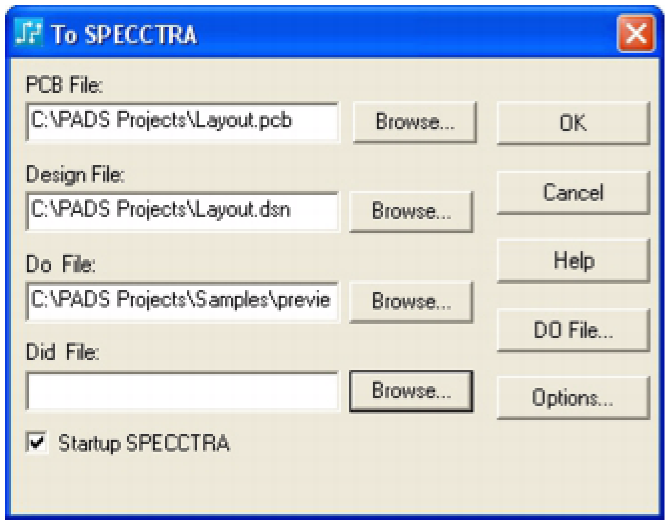
Related Topics

Modifying Text Properties

To SPECCTRA Dialog Box

To access: In Windows Explorer, navigate to your *C:\<install_folder>\SailWind<version>\Programs* directory, and double-click *pads2sp.exe* and then click the **To SPECCTRA** button

Use the To SPECCTRA dialog box to translate a *.pcb* design file into a SPECCTRA design file.



Objects

Field	Description
PCB File	Specifies the file to send to SPECCTRA. Click Browse to locate the file.
Design File	Specifies the design file (*.dsn*) that SPECCTRA inputs. 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. 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. 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.
Options button	Opens the Options dialog box on page 1727.

Related Topics

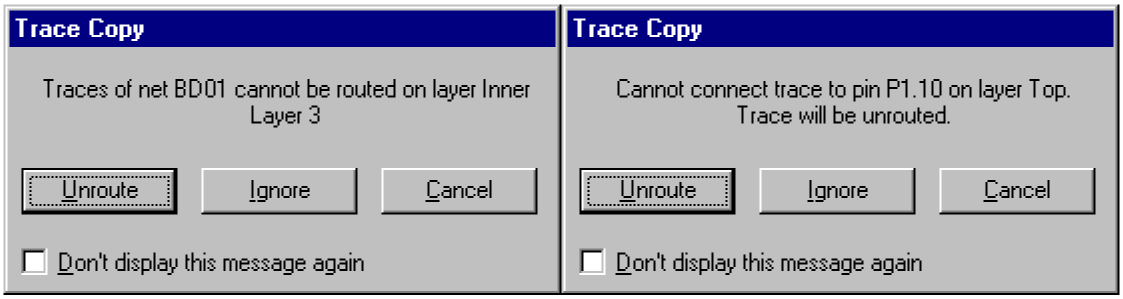
Translating Design Data from SailWind Layout to SPECCTRA

Trace Copy Dialog Box

To access: Appears 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 219. Trace Copy Dialog Box



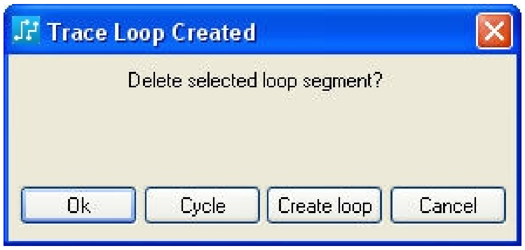
Objects

Field	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

To access: Automatically opens when you create a trace loop.

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.



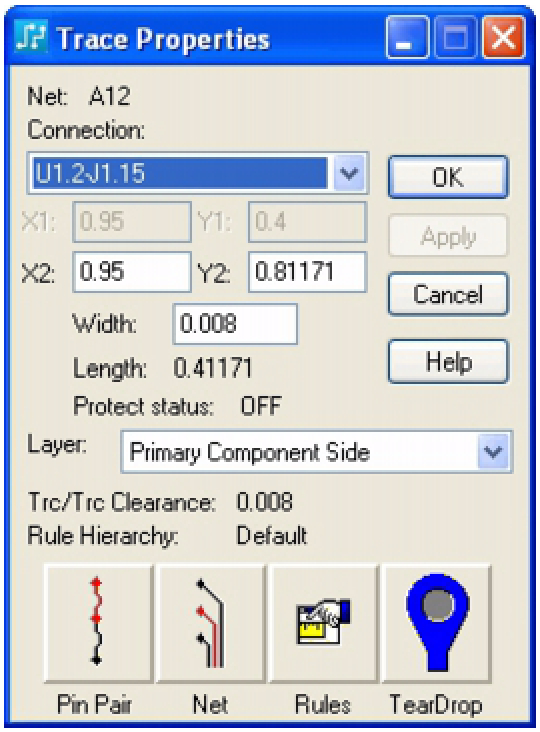
Objects

Field	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

To access: Select a trace > right-click > **Properties** popup menu item

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.



Objects

Field	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.

Field	Description
	Restriction: You cannot enter a value that is not within the minimum and maximum trace width settings in the Clearance Rules on page 1167. If you enter an illegal value, you get the error message, “Wrong width value.”
Length	Displays the routed length of the trace. Does not include any Discrete lengths.
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”.
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 on page 1487.
Rules button	Opens the Pin Pair Rules Dialog Box.
TearDrop button	Opens the Teardrop Properties on Traces Dialog Box.

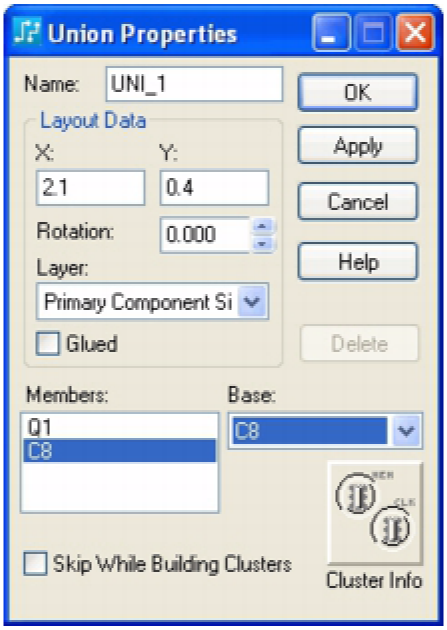
Related Topics

Modifying Trace Segment Properties

Union Properties Dialog Box

To access: Select a union > right-click > **Properties** popup menu item

Use the Union Properties dialog box to obtain information on, and to modify, a selected union.



Objects

Field	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.

Field	Description
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.

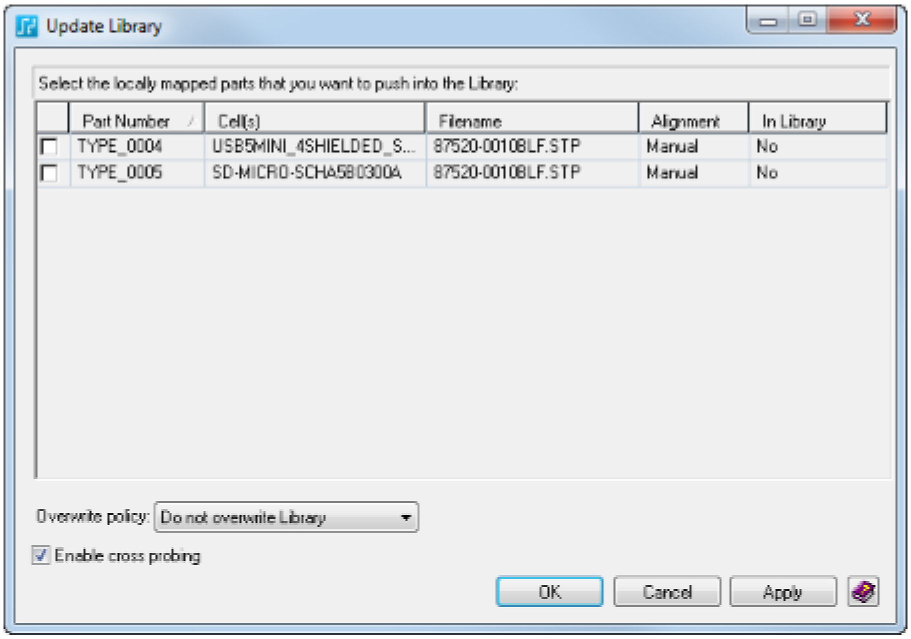
Related Topics

Unions

Update Library Dialog Box

To access: **View > SailWind 3D** menu item > click the **Update Library** button  .

Use this dialog box to export 3D model mappings from a PCB design to the Reuse folder location. Exporting 3D model mappings is optional; however, exporting them allows you to reuse the 3D model with the associated decal in future designs.



Objects

Field	Description
Part Number	Displays the part type of all components in the PCB design.
Cell(s)	Displays the decal name associated with each part in the PCB design.
Filename	Provides the file name of the 3D model mapped to each decal in the PCB design.
Alignment	Indicates whether the 3D model has been aligned manually or automatically. SailWind Layout imports 3D models into the PCB design in the default layout with which they were originally created (automatic alignment). You can manually adjust the alignment of the 3D model using the Align 3D Models Dialog Box. Updating a decal in the Reuse folder location with a 3D model that has been manually aligned results in a new default alignment.

Field	Description
In Central Library	Indicates whether or not the 3D model mapping is already saved in the Reuse folder location.
Overwrite policy dropdown list	Select one of three options when encountering any change conflicts in the central library: • **Do not overwrite Library —** Prevents overwriting of any existing 3D model mappings. Decals mapped in the PCB to 3D models that differ from those mappings in the Reuse folder location remain in the PCB design only. • **Overwrite all in Library —** Overwrites any existing 3D model mappings already in the Reuse folder location with those in the PCB design. • **Prompt for each conflict —** Provides a warning whenever encountering a conflict with Reuse folder location, allowing you to overwrite the existing 3D model mapping or retain it.
Enable cross probing check box	Select this check box to enable you to interactively select a part in the design workspace and make it selected also in the **Update Library** list.

Related Topics

Assigning a 3D Model to a Component

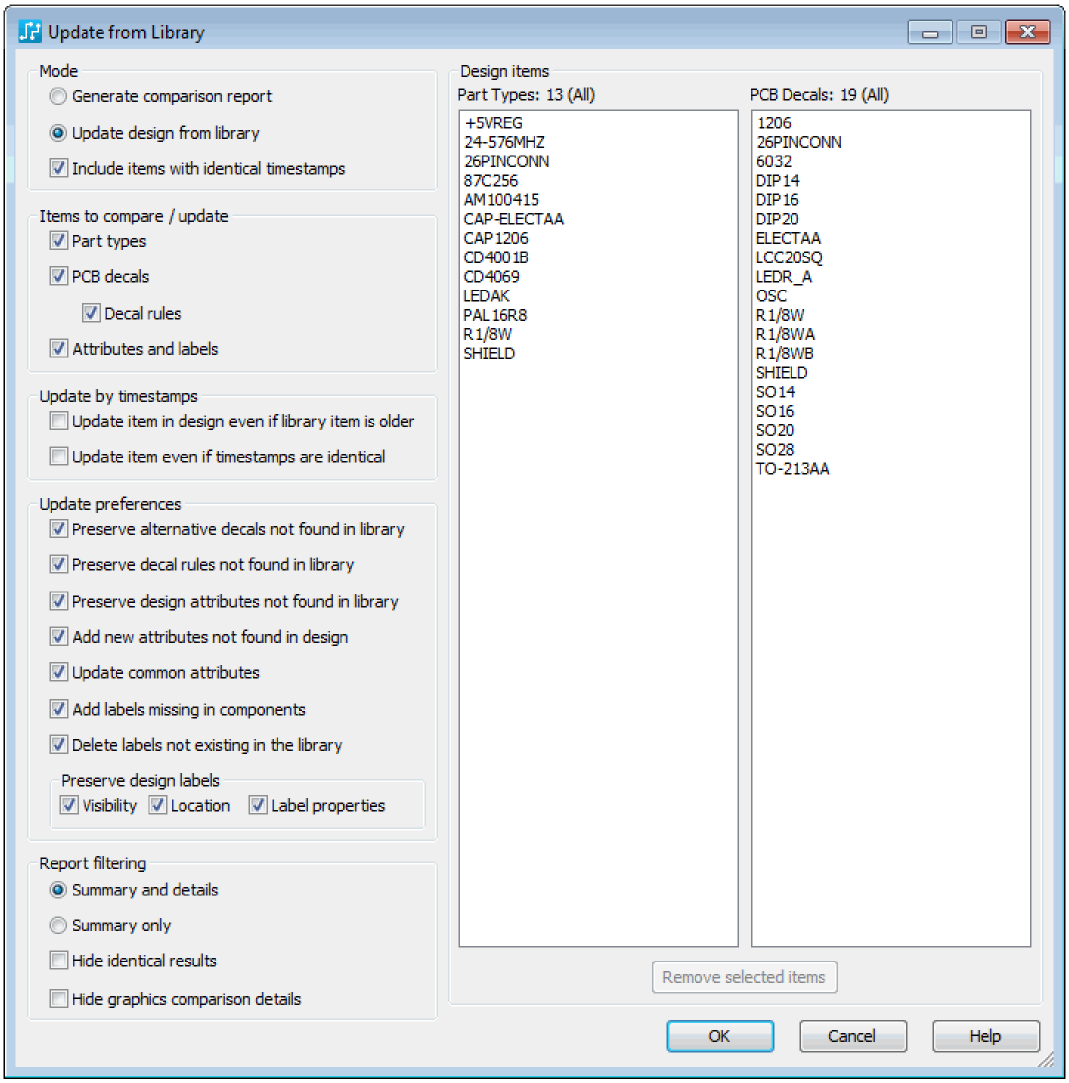
3D Model Mapping

Update from Library Dialog Box

To access: **Tools > Update from Library**

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.



Objects

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 SailWind 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 on page 1593.
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.
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:

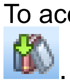
Control	Description
	• Select the check box to preserve the alternative decals in the design part type. • Clear the check box to remove them from the design part type (that is, leave only the alternative decals that exist in the library version).
Preserve decal rules not found in library	Use this check 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. See the “Label” check boxes below for additional attribute-related controls. • 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).
Add labels missing in components	Compares the labels on components in the design against those of the library decal and adds labels that might have been deleted from the component in the design or added to the decal in the library after the part was placed in the design. For example, a silkscreen Reference Designator deleted from the component in the design, or a second Reference Designator added to the assembly layer in the library. Labels that exist in the library decals are propagated to all instances of decals in the design. For an exact match of library labels to component labels, select the “Delete labels not existing in the library” check box also and clear the three “Preserve design labels” check boxes.
Delete labels not existing in the library	Compares the labels on components in the design against those of the library decal and deletes labels that might have been added to the component in the design or deleted from the decal in the library after the part was placed in the design.

Control	Description
	Note: **Requirement:** You must also clear the three check boxes in the "Preserve design labels" area. Labels on components in the design are removed if matching labels are not found in the library decals. For an exact match of library labels to component labels, select the "Add labels missing in components" check box also and clear the three "Preserve design labels" check boxes.
Preserve design labels	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. **Tip** These check boxes need to be cleared if you have selected the "Delete labels not existing in the library.
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.

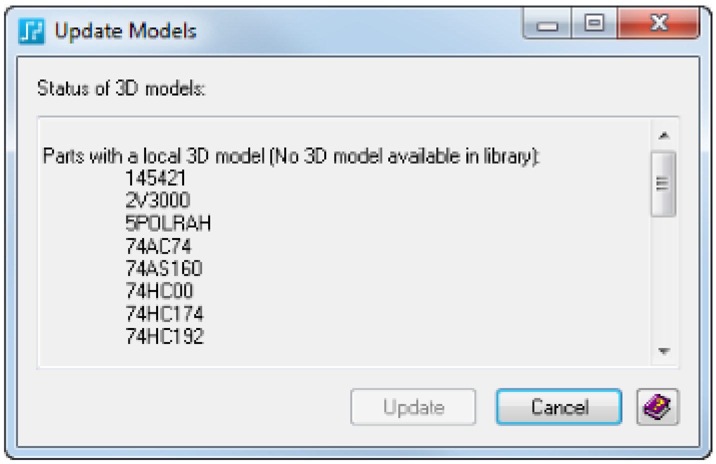
Related Topics

Updating a Design from the Library

Update Models Dialog Box

To access: Open the 3D view (**View > SailWind 3D** menu item), then click the Update Models button .

Use this dialog box to update the 3D models in the PCB design with 3D model information from the Reuse folder location.



Objects

Field	Description
Status of 3D models	Displays a short report on the current state of all parts in the 3D design and their current 3D model mappings: • **Parts with a local 3D model (no 3D model available in library)** — Displays a list of all parts that have 3D models mapped to them in the PCB design but do not have 3D models mapped in the Reuse folder location. • **Parts with a local 3D model (A 3D model is available in library)** — Displays a list of parts in the PCB design that have a 3D model with differences from the 3D model mapped to the part in the Reuse folder location. • **Parts without 3D model in library** — Displays a list of parts in the PCB design that have not been mapped to a 3D model. Parts not mapped to 3D models are represented by 2D extruded models instead. • **Parts with up-to-date 3D model** — Displays a list of parts in the PCB design that have 3D models mapped in the Reuse folder location that have information unchanged from the corresponding 3D model information in the PCB design. • **Parts with modified 3D model mapping** — Displays a list of parts with 3D model information in the PCB layout that require updating with 3D model changes saved in the central library or Reuse folder location.

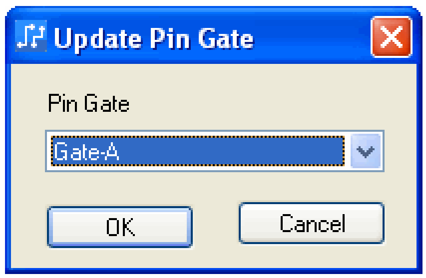
Field	Description
Update button	If any parts in the PCB design are mapped to 3D models in both the local PCB design and the Reuse folder location, and the 3D models in the Reuse folder location have differences, you can update the corresponding 3D models in the local PCB design by clicking **Update**

Update Pin Gate Dialog Box

To access:

- **File > Library** menu item > select a Library > **Parts** button > **New** > Pins tab on page 1596 > select cells in the Pin Group column > **Edit** button
- **File > Library** menu item > select a Library > **Parts** button > select a part > **Edit** button > **Pins** tab > select cells in the Pin Group column > **Edit** button

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.



Objects

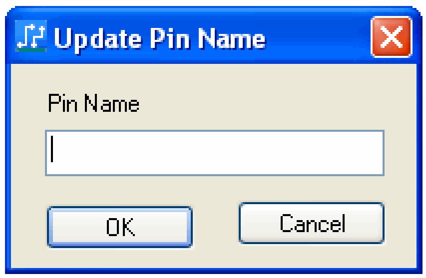
Field	Description
Pin Gate list	Specifies the setting to assign selected pins in the pins table. Possible settings include: **Gate-n** — assign to gate pins. This selection is only available if you have added gates on the Gates tab on page 1588. **Signal pin** — assign to implicit pins (not shown on 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 on page 1590.

Update Pin Name Dialog Box

To access:

- **File > Library** menu item > select a library > **Parts** button > **New** > Pins tab on page 1596 > select cells in the Name column > **Edit** button
- **File > Library** menu item > select a library > **Parts** button > select a part > **Edit** button > Pins tab on page 1596 > select cells in the Name column > **Edit** button

Use this dialog box to update the Name column of selected pins in the Pins table (on the Pins tab) of the Part Information dialog box.



Objects

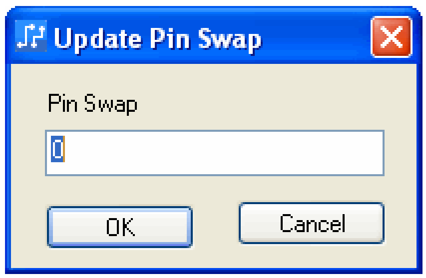
Field	Description
Pin Name box	Specifies the name to assign selected pins in the pins table.

Update Pin Swap Dialog Box

To access:

- **File > Library** menu item > select a library > **Parts** button > **New** > Pins tab on page 1596 > select cells in the Swap column > **Edit** button
- **File > Library** menu item > select a library > **Parts** button > select a part > **Edit** button > Pins tab on page 1596 > select cells in the Swap column > **Edit** button

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.



Objects

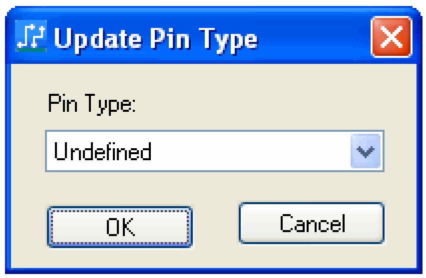
Field	Description
Pin Swap box	Specifies the swap number to assign selected pins in the pins table. Pins with the same swap number can have their connections swapped in the design using the Swap Pin or Auto Swap Pin features on the ECO Toolbar.

Update Pin Type Dialog Box

To access:

- **File > Library** menu item > select a library > **Parts** button > **New** > Pins tab on page 1596 > select multiple cells in the Type column > **Edit** button
- **File > Library** menu item > select a library > **Parts** button > select a part > **Edit** button > Pins tab on page 1596 > select multiple cells in the Type column > **Edit** button

Use the Update Pin Type dialog box to update the Type column of multiple selected pins in the Pins table (on the Pins tab) of the Part Information dialog box.



Objects

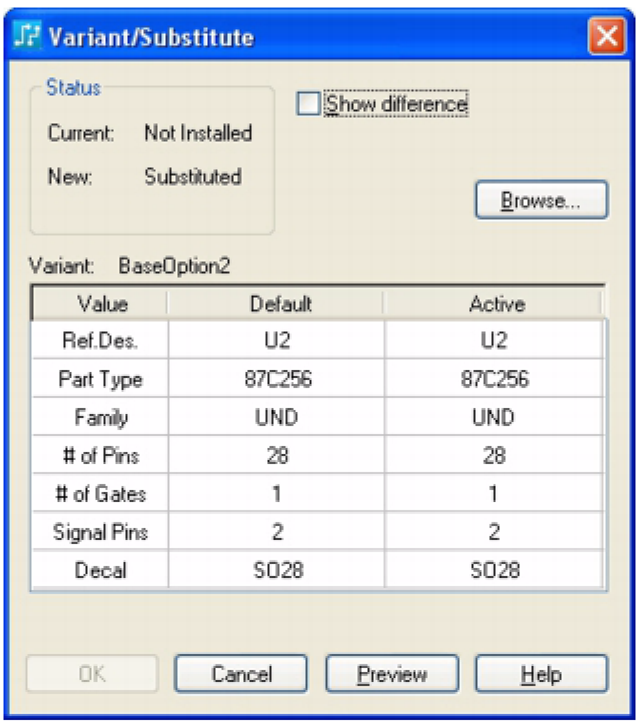
Field	Description
Pin Type list	Specifies the type to assign selected pins in the pins table.

Variant/Substitute Dialog Box

To access: **Tools > Assembly Variants** and then one of the following:

- 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.
- Select a component in the multicolumn list, and click Substitute in the Status area.
- Click Substitute from the Verb Mode list, and select a component in the Layout Editor.

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.



Objects

Field	Description
Current	Displays the state before you click Substituted.
New	Displays the state after you click Substituted.

Field	Description
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 on page 1820 value, and its Active on page 1808 value (the substitution).
Preview	Opens the Preview for dialog box where you can see the substitutions you have made.

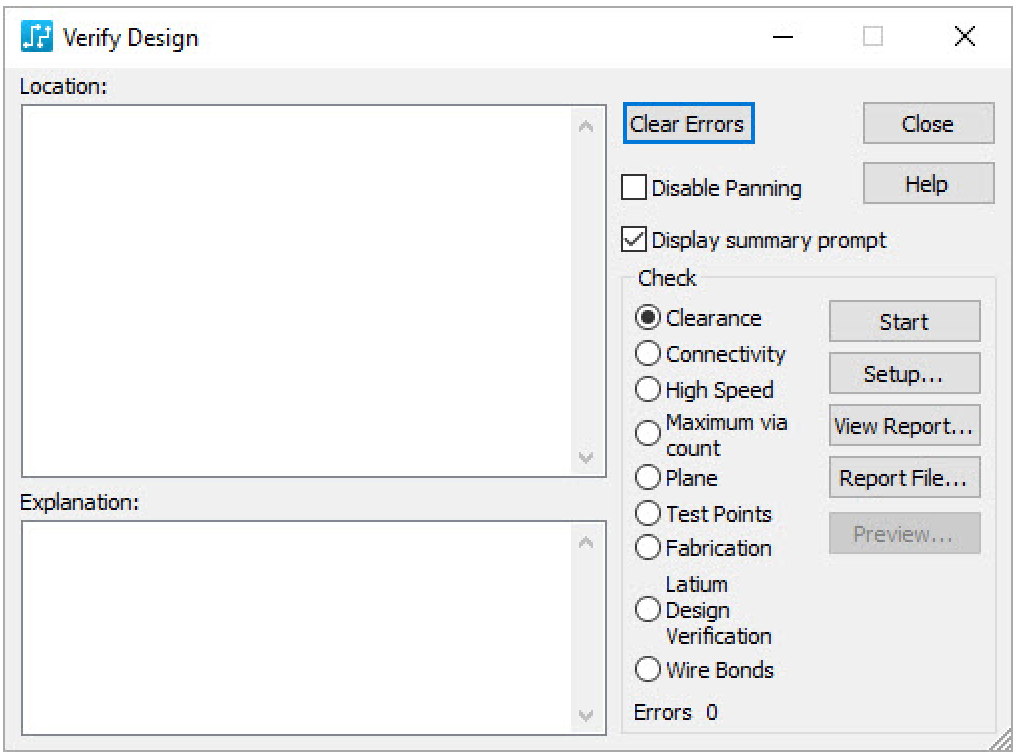
Related Topics

Substitute a Component for Assembly Variants

Verify Design Dialog Box

To access: **Tools > Verify 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. You cannot use this dialog box to check reference designator, part type, or attribute labels for clearance violations.



Objects

Field	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 on page 885. **CAUTION:** 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	Enables the error-summary prompt window. The prompt window appears after the check has run and lists the total number of errors found.

Field	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 and visible objects of the design. For example, if you have a trace running across the via pad of a different net on inner layer 4 and you have the color for vias assigned the same color as the background color, the clearance check will not identify any clearance error on that via and trace short. 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. **Tip** • 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 Subnet 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**.
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. **Tip** DFT treats virtual pins as pins. See also: "Performing a Test Point Audit".

<table>
<tr><th>Field</th><th>Description</th></tr>
<tr><td></td><td>
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 SailWind Layout and SailWind 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 SailWind Layout or by using errors that are backward annotated from CAM350.
Note:
Requirement: You need the CAM350 Link license option to use this. See also: Fabrication Checking Setup Dialog Box and “Back-annotating CAM350 Files”.
Latium Design Verification — Performs clearance checking on only the visible area of the design. Any Latium rule on page 1835 errors found appear in the Location list. The design is passed to SailWind Router to perform the verify design check, and is passed back to SailWind Layout when the process is complete. If you used SailWind Router to work on your design, it might contain advanced rules (differential pairs, vias at SMD, matched length traces) which only SailWind Router can check.
The Latium Design Verification:
a. Saves the current SailWind Layout database to a temporary file
b. Starts SailWind Router (if not already running)
c. Starts the SailWind Router Monitor on page in Verify Design Mode
d. Loads the saved SailWind Layout design into SailWind Router
e. Executes the selected checking operations
f. Saves the file in SailWind Router
g. Re-loads the SailWind Router file into SailWind 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 on page .

</td></tr>
<tr><td>Start button</td><td>Runs the check with your specified options.</td></tr>
<tr><td>Setup button</td><td>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
</td></tr>
</table>

Field	Description
	• **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 *SailWind 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.

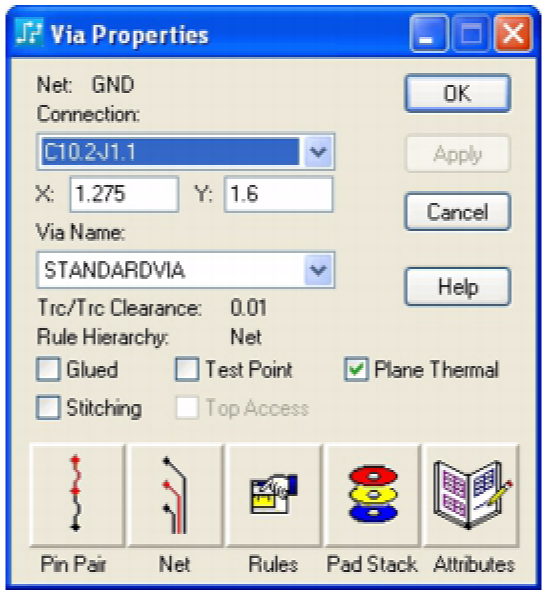
Related Topics

Verify the Design

Via Properties Dialog Box

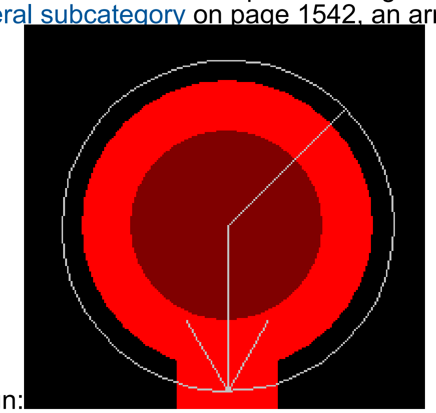
To access: Select a via > right-click > **Properties** popup menu item

The Via Properties dialog box displays the netname to which the via belongs, the via name, coordinates, and the place where the via is connected.



Objects

Field	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 Options dialog box > **Routing** category > General subcategory on page 1542 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 on page 995 appears.
Trc/Trc Clearance	Displays the trace to trace clearance for this via.

Field	Description
Rule Hierarchy	Displays which rules are apply to the via. See also: “Design Rule Hierarchy”.
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 in the Options dialog box > **Routing** category > General subcategory on page 1542, an arrow is drawn on it in the design: 
Plane Thermal	Determines whether the pin or via is eligible to receive a thermal on page 1866. 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: “Setting Pins and Vias as Thermals on page 797.”
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

Field	Description
	are treated differently from regular vias (which can be ripped up or shoved by the dynamic router).
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 on page 1487.
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

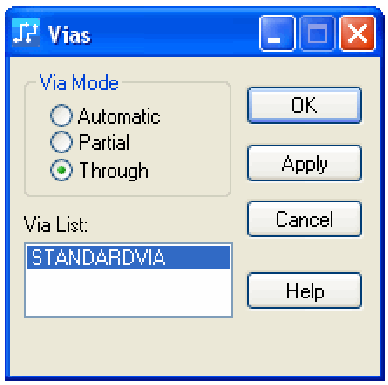
Modifying Via Properties

Vias Dialog Box

To access: Type v > press the Enter key

Use the Vias dialog box to specify how SailWind Layout should determine the via type to use when you add a via or a virtual pin. When you choose a via type, SailWind Layout uses it with the Add Via, Add Via to Guide, and Add Virtual Pin commands.

Tip
You can also change the via type during routing.



Objects

Field	Description
Via Mode area	• **Automatic** — When you add a via or virtual pin, SailWind Layout chooses from all via types, through or partial, that can handle the particular layer change, if any. (Virtual pins, unlike vias, do not necessarily change layers.) If SailWind Layout finds partial via types dedicated to the layer change, it chooses from them. If SailWind Layout cannot find a dedicated partial via type, it selects any through via types for a through or partial layer change even if the via type is not one that is specifically selected in the Via list. It then checks the Routing Rules Dialog Box for via types that are allowed for the net to which you are adding a via or virtual pin. If more than one via type still passes, SailWind Layout installs the one with the smallest drill diameter or smaller pad size. Automatic allows only via types that begin and end on the Layer Pair shown in the Options dialog box > **Routing** category > General subcategory on page 1542. To use automatic via mode, the layer pair for routing and the layer pair for the partial via type need to match. For example, if you have a partial via type 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 via types only.

Field	Description
	• **Through** — The list of through via types becomes active. Click the via type you want to use as the default and click Apply. This is the via type which will be installed every time you add a via or virtual pin.
Via List	Lists through hole via types available in the design; partial via types are not listed.

Related Topics

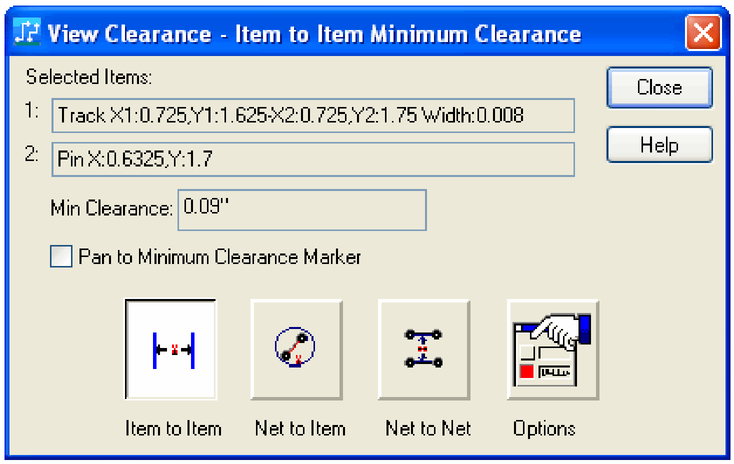
Via Types

Changing the Via Type While Routing

View Clearance Dialog Box

To access: **View > Clearance** menu item > click the button for the type of clearance > select two items in the design

Use the View Clearance dialog box to find the minimum clearance between two items.



Objects

Field	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	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.
Item to Item button	Finds the minimum clearance between two selected items and dimensions appear on the design showing the location and size of the minimum clearance. Supported item types are board outlines, pads, vias, jumpers, traces, 2D lines, copper, 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.

Field	Description
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, and component outlines.
Net to Net button	Finds the minimum clearance between two selected nets.
Options button	Opens the Options dialog box > **Dimensioning** category > General subcategory on page 1520, 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 Options dialog box > **Dimensioning** category > General subcategory on page 1520.

Related Topics

Viewing the Clearance Between Items and Annotating Dimensions

Viewing the Clearance Between a Net and an Item

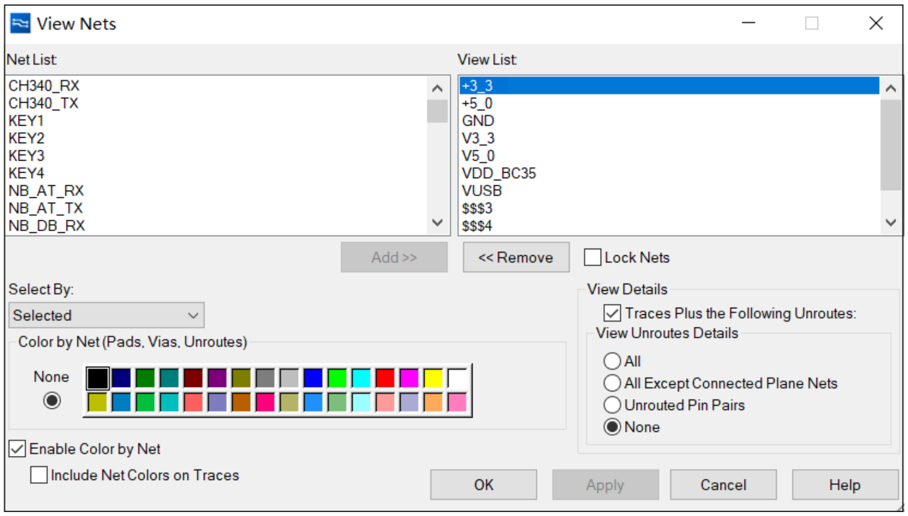
Viewing the Clearance Between Nets

View Nets Dialog Box

To access: **View > Nets**

Use the View Nets dialog box to selectively hide or display connections or to hide or show routed or unrouted paths by netname.

CAUTION:
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.



Objects

Field	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.
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 colors to the critical signals and then create correct copper planes with greater ease. All net color assignments are saved with the design.

Field	Description
	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 do not need to view the power and ground unroutes. • Assign color for critical nets so their static unroutes are clearly visible during placement.
Enable Color by Net	Allows assignment of colors to pads, vias, and unroutes. Selecting this check box also allows you to optionally select the “Color Traces by Net” check box to apply color assignments to traces and pins. You can also use the modeless command Y to toggle this setting. Clearing this check box returns colors to their default assignments (as assigned in the Display Colors Setup Dialog Box). **Note:** Re-selecting this check box restores any previously defined net color assignments.
Include Net Colors on Traces	Shows traces, for nets in the View List box, using the assigned color. Traces, pins, copper, and copper planes appear in the assigned color. Selecting the “Enable Color by Net” check box enables this check box.
Lock Nets	Specifies whether to lock the net(s) currently selected in the View List. Note the following: • To lock or unlock multiple nets, use Ctrl/Shit for multiple selections. Nets are locked/unlocked immediately after this checkbox is selected/cleared. • Options made in the View Unroutes Details area also affect the state of the Lock Nets checkbox: option "None" for selecting the checkbox automatically; the others for clearing this checkbox. • For locked nets, you can neither change their display color by "Assign Color to Net" feature nor toggle the display of unrouted connections by commands Show All Connection or Hide All Connection.
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 planes, or CAM planes. • **Unrouted Pin Pairs** — Displays only unroutes on fully unrouted pin pairs. • **None** — Displays no unroutes.

Related Topics

Modifying Net Properties

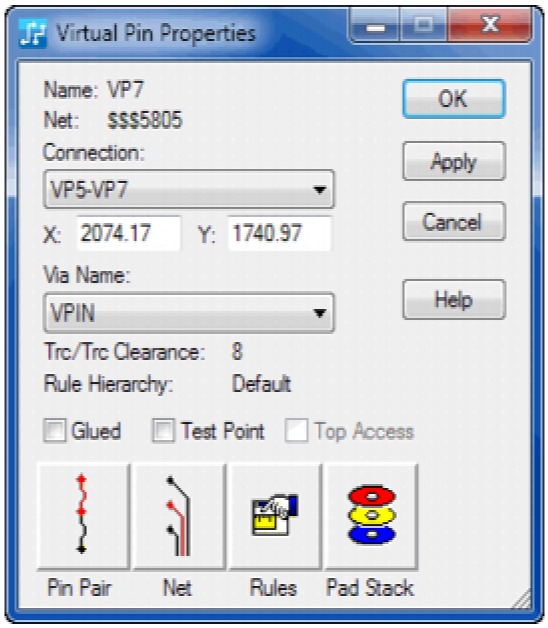
Assigning Colors to Nets

Net Properties Dialog Box of a Netlist Project

Virtual Pin Properties Dialog Box

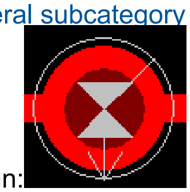
To access: Select a virtual pin > right-click > **Properties** popup menu item

The Virtual Pin Properties dialog box displays the netname to which the virtual pin belongs, the virtual pin padstack name, coordinates, and the place where the virtual pin is connected.



Objects

Field	Description
Name	Displays the reference designator of the virtual pin.
Net	Displays the name of the net to which this virtual pin belongs.
Connection	Lists pin pairs to which the virtual pin belongs.
X/Y	The X and Y location of the virtual pin. Type in these fields to change the location.
Via Name	Lists the name of the via type used by the virtual pin. You can click a different via type from the list and apply it.
Trc/Trc Clearance	Displays the trace to trace clearance for this virtual pin.
Rule Hierarchy	Displays which rules are apply to the virtual pin. See also: "Design Rule Hierarchy".
Glued	Glues the virtual pin so you cannot move it. If you try to move the virtual pin, the following message appears in the Output window on

Field	Description
	the **Status** tab “At least one object is glued, cannot proceed with command.”
Test Point	Makes the virtual pin a test point. This is a three-state check box on page 1866. When you click OK or Apply, the pad stack is automatically checked to see if the virtual pin can be a test point; for example, you cannot make a buried virtual pin a test points because a probe cannot access a buried virtual pin. If it cannot be made a test point, a warning message pops up and the Test Point check box is automatically cleared. **Tip** When the virtual pin is flagged as a test point, and Show Test Points is checked in the Options dialog box > **Routing** category > General subcategory on page 1542, an arrow is drawn on it in the design:  **Tip** The points of a bed of nails test apparatus are matched to the test point locations, which can be virtual pins. So if you redesign and remanufacture the board, glued test points prevent the virtual pins from moving in the new design, thus preventing costly retooling of the test equipment. Consider locking test points with the Lock test points check box in the Routing options on page 1542.
Top Access	Attempts to probe the test point from the top and bottom in DFT Audit. **Restriction:** This check box is unavailable unless you select the Test point check box first. 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 OK or 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.
Pin Pair button	Opens the Pin Pair Properties Dialog Box.
Net button	Opens the Net Properties dialog box on page 1487.
Rules button	Opens the Pin Pair Rules Dialog Box.
Pad Stack button	Opens the Pad Stacks Properties Dialog Box.

Related Topics

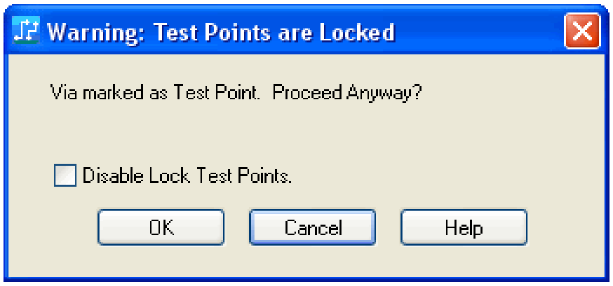
Virtual Pins

Warning: Test Point Locked Dialog Box

To access: Modify any of the following:

- A cluster that contains test points
- Vias, pins, virtual pins or jumper pins that are locked test points
- Routes that are connected to locked test points

This warning dialog box performs different functions depending on whether you are modifying vias, pins, virtual pins, or routes.



Objects

Field	Description
Disable Lock Test Points	Turns the Lock Test Points check box off in the Options dialog box > **Routing** category > General subcategory on page 1542
OK button	Applies the change and maintains the test point status.

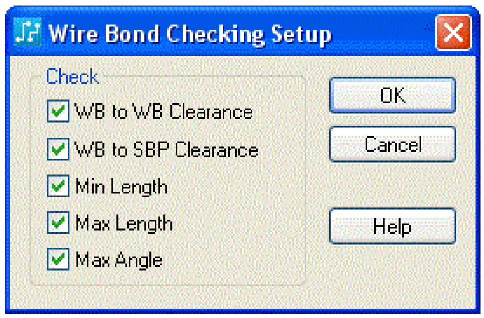
Related Topics

Warning: Test Point Locked Dialog Box

Wire Bond Checking Setup Dialog Box

To access: **Tools > Verify Design** > Wire Bonds check > **Setup** button

Use the Wire Bond Checking Setup dialog box to specify which wire bond rules to check during a Wire Bonds verification.



Objects

Field	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

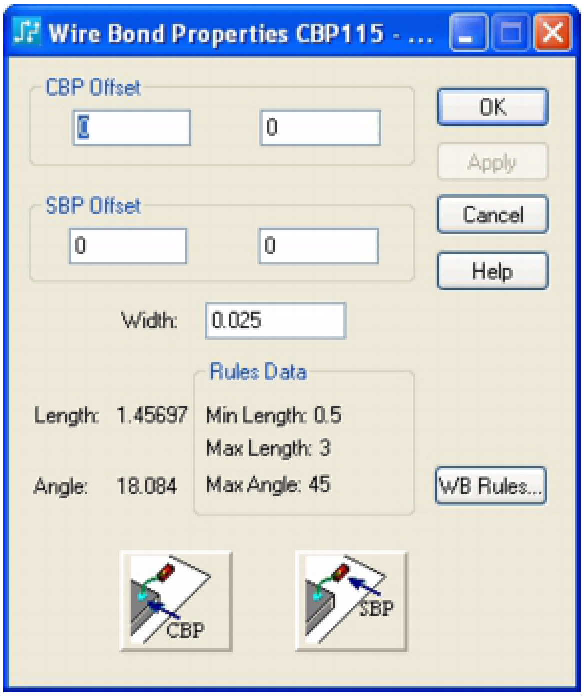
Wire Bond Properties Dialog Box

To access: **BGA Toolbar > Wire Bond Editor** button > select a wire bond > right-click > **Properties** popup menu item

Use the Wire Bond Properties dialog box to edit the wire bond.

Restriction:
This information applies only to the BGA toolkit.

Note:
If you have multiple objects selected, properties that are not common to all selected objects will not appear.



Objects

Field	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.

Field	Description
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.
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.

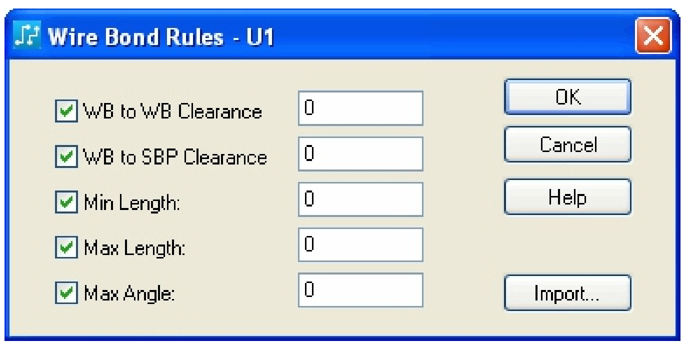
Wire Bond Rules Dialog Box

To access:

- **BGA Toolbar** button > **Wire Bond Wizard** button > **Wire Bond Rules** button
- **BGA Toolbar** button > **Wire Bond Editor** button > select a die component > right-click and click the **Wire Bond Rules** popup menu item
- In the layout editor select a substrate bond pad, right-click and click the **Wire Bond Rules** popup menu item

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.



Objects

Field	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.”

Field	Description
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

Wire Bond Rules Checks

Wire Bond Wizard Dialog Box

To access: **BGA Toolbar > Wire Bond Wizard** button > select die part

Use the Wire Bond Wizard dialog box to define the SBP rings, set rules, assign the component bond pads to rings, define the wire bond fanout geometry, and create a wire bond fanout.

Description

The Die Wizard also has 4 tabs as shown below the main graphic.

- "Figure 221" on page 1799 with details
- "Figure 222" on page 1799
- "Figure 223" on page 1800
- "Figure 224" on page 1800

The features of those tabs are described in tables at the end of this topic.

Figure 220. Wire Bond Wizard Dialog Box

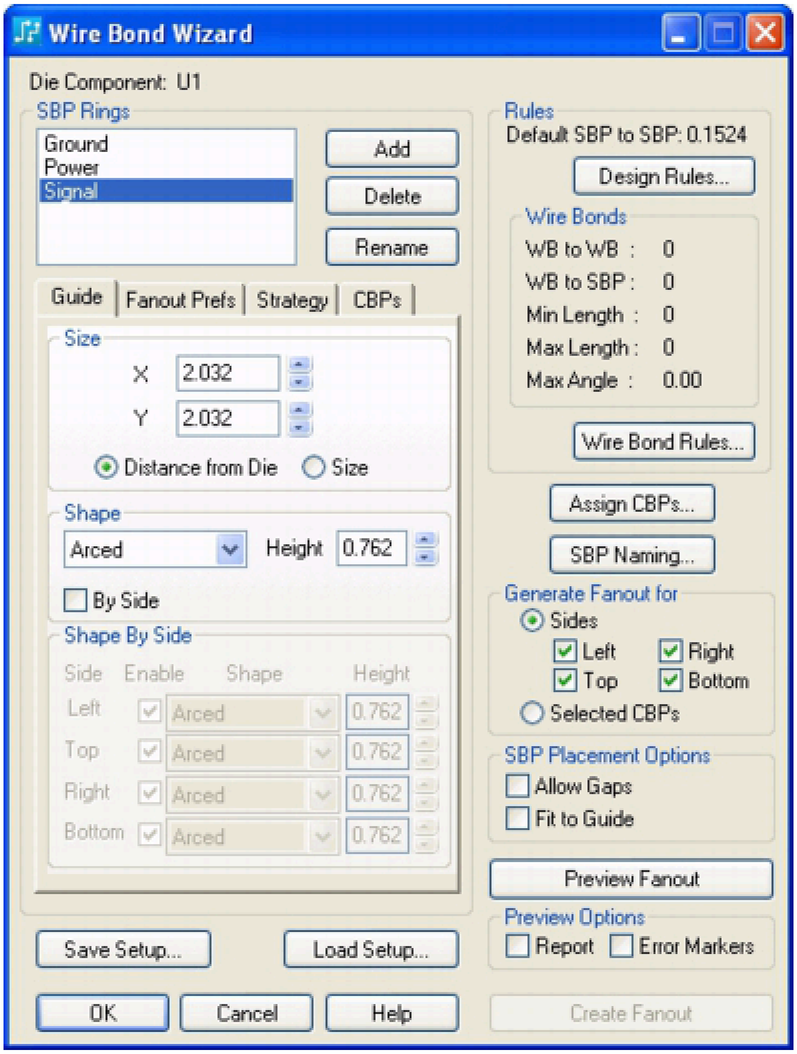


Figure 221. Wire Bond Wizard, Guide tab

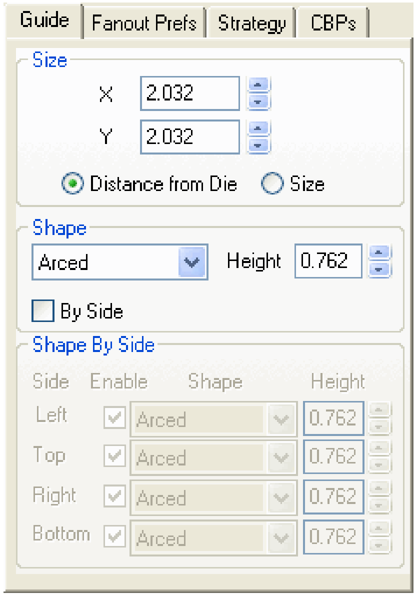


Figure 222. Wire Bond Wizard, Fanout Prefs tab

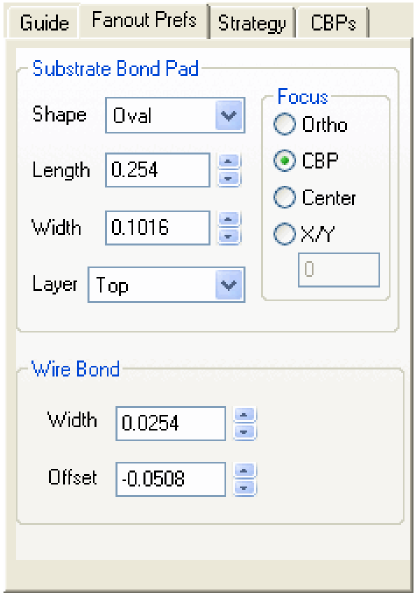


Figure 223. Wire Bond Wizard, Strategy tab

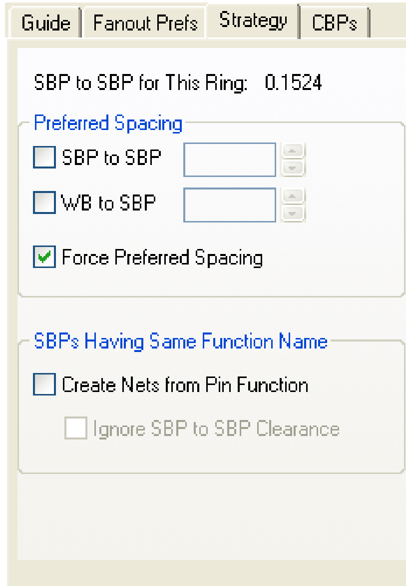
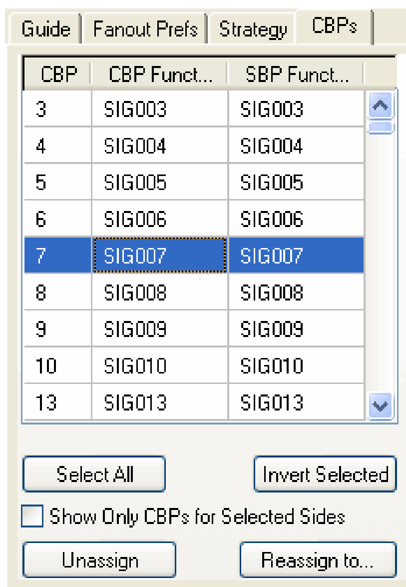


Figure 224. Wire Bond Wizard, CBPs tab



Objects

Table 207. Wire Bond Wizard Dialog Box Fields

Field	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.

Table 207. Wire Bond Wizard Dialog Box Fields (continued)

Field	Description
	You can also add new rings, delete rings, and rename rings.
Add button	Opens the Add SBP Ring dialog box on page 1072, 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 on page 1072, where you can rename the currently selected SBP ring.
tabs	• Table 208 on page 1802 • Table 209 on page 1803 • Table 210 on page 1803 • Table 211 on page 1804
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.
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.

Table 207. Wire Bond Wizard Dialog Box Fields (continued)

Field	Description
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.

Table 208. Guide tab contents

Field	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.

Table 208. Guide tab contents (continued)

Field	Description
	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.

Table 209. Fanout Prefs contents

Field	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.

Table 210. Strategy tab contents

Field	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 210. Strategy tab contents (continued)

Field	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.

Table 211. CBPs tab contents

Field	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.
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

Creating a Wire Bond Fanout

Glossary

absolute coordinates

Coordinates of a location based upon their distance from the origin (coordinates 0,0) of the design area.

accelerator keys

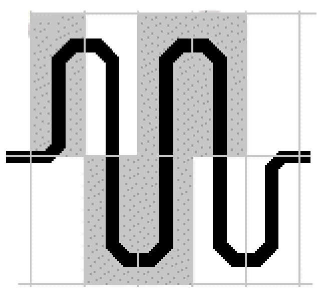
Key sequences used to invoke commands and change system settings without using the mouse. Accelerator keys are called shortcut keys in the SailWind product documentation.

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

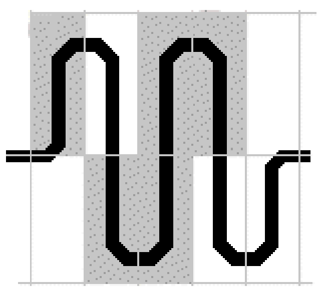
A trace pattern resembling a signal wave that adds length to traces. The trace patterns are contiguous and do not include layer changes.



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.

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 on page 83.

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.

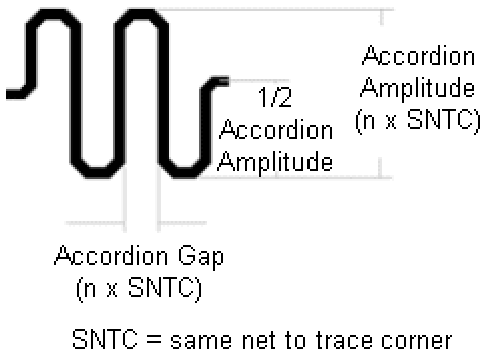
alphanumeric pins

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.

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 board

A board with mostly discrete components and minimal integrated circuits.

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.

annular ring

The conductive pad material surrounding the hole. The annular ring radius = pad diameter-(finished hole size) / 2.

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. SailWind 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

See *electrical net*.

associating component

A component through which an electrical 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:

- 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. SailWind 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.

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.

Pass	Description
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.

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 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 re-import the information to *.pcb* format, export in basic units.

Conversions are:

- 1 mil = 38100 basic units
- 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. SailWind 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.

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 on page 629

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 connects only inner layers.

bus

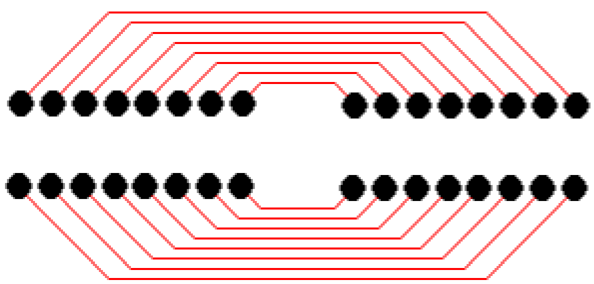
A series of connections that share a common use, such as memory array or data array, and are usually routed parallel to each other.

bus routing

Routing two or more pin pairs simultaneously and in close proximity to each other in neat, flowing patterns.

C 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 SailWind 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 *SailWindpcb.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 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

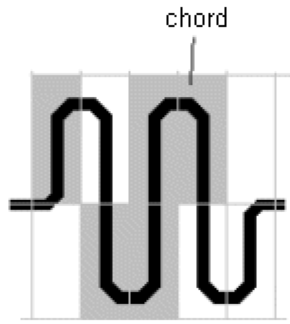
The packaging configuration in which a chip is bonded directly to a circuit board or substrate.

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.

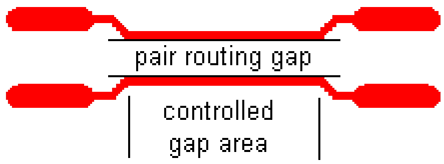
container application

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 SailWind product, the SailWind product is the container application. When you insert a SailWind 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:

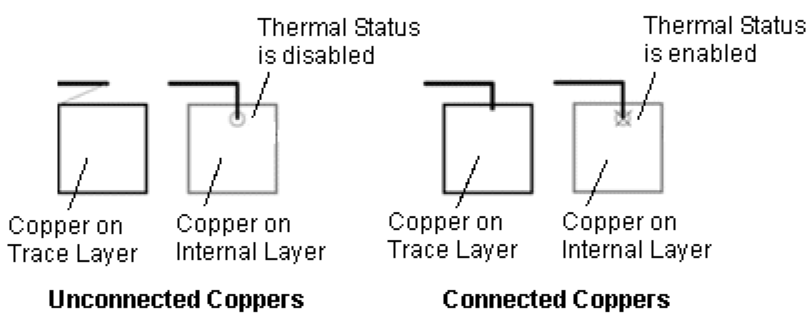
- Minimum/maximum length
- Matched length
- Differential pairs

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 plane

A copper shape with insulation areas around traces and pins that pass through the copper, but are not attached or connected to the copper. Can be placed on any layer, except CAM plane layers.

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 SailWind Layout and PADS Designer.

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 plane, or board outline. In copper or copper planes, a cutout results in an area absent of copper.

See also *overlapping cutouts*

cycle picking

To sequentially select objects in the vicinity of the selection point using the Tab key.

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 on page 922

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

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:\<install_folder>\<version>\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 *SailWindpcb.ini* file.

See also *increased layer mode*

defaults

Conditions or options that are set when the SailWind 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 category

The Options category 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:\<install_folder>\<version>\Programs* folder that contains CAM printer and plotter driver data. This file must exist in the same folder specified by the UserDir *SailWindpcb.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*

digital board

A board with mostly integrated circuits in proportion to the analog components.

DIP

An acronym for Dual In-line Package.

DisableCaching

A *SailWindpcb.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 SailWind 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.

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 *SailWindpcb.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 to complete the connection.

ecad hint.map

A user-defined text file that you create, edit, and maintain. This file enables the replacement of approximated parts from SailWind 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 SailWind 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 on page 1352.

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.

electrical net

A series of nets connected by one or more components. Length, differential pair and matched length rules can be applied to an electrical net as though it were a single net.

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 *SailWindpcb.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 electrical net.

end no via

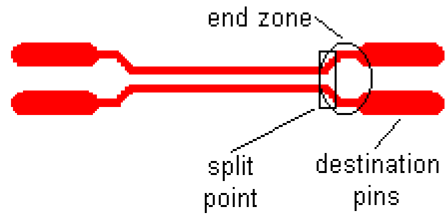
The mode initiated in the routing shortcut menus that, while routing, ends a partial route without a via.

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.



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.

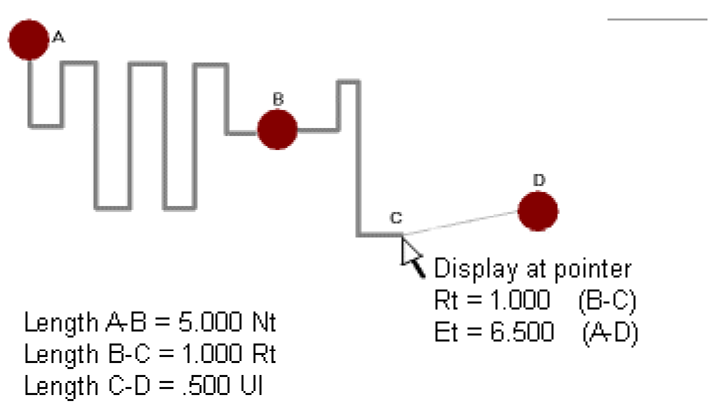
Esc

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), the unrouted length (U1) of the trace being routed, and includes half the Discrete length value of each connected pin of components that have a Discrete length assigned (not shown).

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.

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, enabling 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 enable easy routing.

Use fanouts to:

- enable 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 for alignment and calibration of images on a multi-board panel.
- **Board fiducials** — used 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 (local) 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 enables 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 *SailWindpcb.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.

finished hole size

The size of a drilled or routed hole after plating and/or solder reflow has been applied.

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 plane.

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 *SailWindpcb.ini* file entry that determines the maximum size of the undo buffer before SailWind 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, SailWind 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.

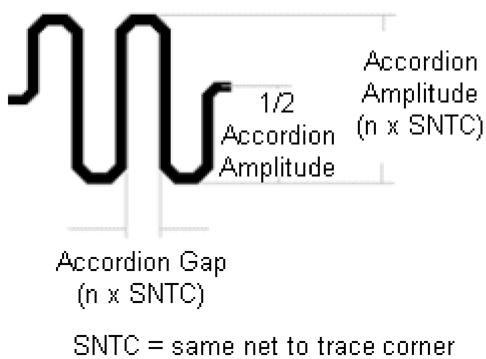
free disk space

The physical amount of space available on your hard drive that is available for use by programs.

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.

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

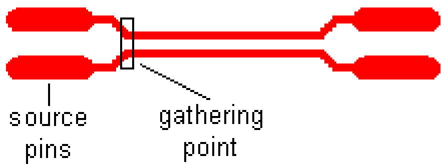
Gate swapping involves exchanging the entire element for a like element.

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.



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.

Geometry.Height

This attribute is used to indicate the height of the part. The attribute enables SailWind Layout to prevent the component from being placed in an area of the PCB which is height restricted.

In SailWind Layout, you can set board height restrictions for the top and bottom layers in the Drafting Options on page 1553 or area height restrictions using a Keepout area with a “Component height” restriction. This attribute is also passed from SailWind Layout to mechanical tools where it can be used in 3D simulations to determine whether the part will meet spatial requirements.

In addition to the value, use one of the following units:

- Use the quotation symbol " for inches. The SailWind Layout Attribute Dictionary specifies the following limits of acceptable values. Min=0.00000", Max=25.00000".

  Example: GEOMETRY.HEIGHT=3.26548"

- Use the abbreviation mil in upper or lower case. The SailWind Layout Attribute Dictionary specifies the following limits of acceptable values. Min=0.00mil, Max=25000.00mil

  Example: GEOMETRY.HEIGHT=12654.83mil

- Use the abbreviation mm in upper or lower case. The SailWind Layout Attribute Dictionary specifies the following limits of acceptable values. Min=0.00000mm, Max=635.00000mm.

  Example: GEOMETRY.HEIGHT=123.21348mm

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

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 *SailWindpcb.ini* and *SailWindlogic.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

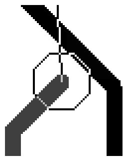
A collection of pin pairs that share common design rules.

grow

An cluster placement feature that adds additional parts to an existing cluster.

guard band

An octagonal shape that appears at the end of a trace during routing operations whenever the head of the trace meets a clearance obstacle that it cannot shove. The guard band only appears when online design rules are enabled.



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.

hard rule

A rule that is always followed. See also *soft rule*, and Hard and Soft Rules.

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.

HDI

An acronym for High Density Interconnect.

heat dissipating component

While all components generate heat, these components generally have a published wattage rating. Care must be taken to ensure components or materials adjacent to these heat generating components do not exceed their max temperature ratings. If more than one of these components are used in a design, they should be spread out and should also be positioned to not impede airflow.

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.

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* filename 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.

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. Type the I modeless command, then press the Enter key.

intensity

A value assigned to objects such as vias to weigh decisions made during the autorouting process in SailWind 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 SailWind 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 plane that are not attached to anything.

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.

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 planes, 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.

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 SailWind Router. Some constraints that you can set in Layout are only used by SailWind Router. For examples, see the Tune/Diff Pairs options on page 1550, Fanout Rules on page 1375, Pad Entry Rules. on page 1564The 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 enables 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 *\SailWind 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 *SailWindpcb.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 SailWind product or created by the user.

Library Manager

The SailWind 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 *SailWindpcb.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 *SailWindpcb.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.

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

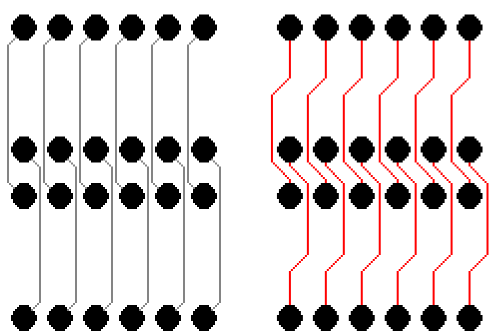
There are three material conditions when creating component decals. They are Maximum (providing the most robust solder joint), Nominal (providing a general purpose solder joint) and Minimum (providing the least possible solder joint for very dense designs).

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 *SailWindPCB.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.

microvia

Vias that have a narrow drill hole. Because of their specific diameter to depth ratio, they are typically blind or buried vias and do not pass through many layers of the design.

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.

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.

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 and Adding Board Mounting Holes on page 268 in the *SailWind Layout User's Guide and Reference Manual*

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.

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 SailWind Logic schematic into your SailWind 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.

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:\<install_folder>\<version>\Programs* folder when the *SailWindpcb.ini* entry LOGMode is equal to one. The *next.ini* file is a copy of your *SailWindpcb.ini* file at the time *next.log* is written.

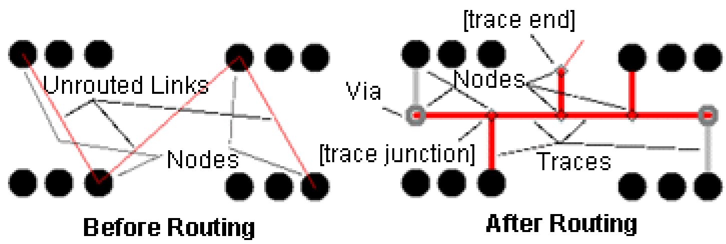
next.log

A file produced in the *C:\<install_folder>\<version>\Programs* folder when the *powerpcb.in*i entry LOGMode is equal to one. The *next.log* file records all activities within a SailWind Layout session so that they can be replayed to reproduce a series of steps or used to illustrate a problem.

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, virtual pins, vias, or floating endpoints. Specifically, a node can be any pin, virtual 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. Non-plated holes are not drilled with an oversize to accommodate the solder flow.

To determine plating status, in SailWind Layout, use the Pad Stacks Properties dialog box. In SailWind Router, use the Pad Stack tab in the Pin Properties dialog box.

nudge

A placement feature that relocates parts to make room for new parts being placed. Movement is based on previously defined clearance rules.

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, for example protected pins, vias, traces, keepouts, and board outlines.

odd pad shape

A pad that requires a special aperture, or plot sequence, to create. In SailWind 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 SailWind products, a net must have a minimum of two pins.

online DRC

A SailWind 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

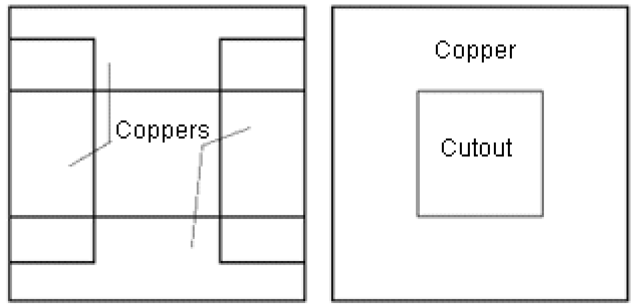
Rearranging placed parts and/or swapping pins and gates on parts to minimize trace lengths and reduce the number of vias required for routing.

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.



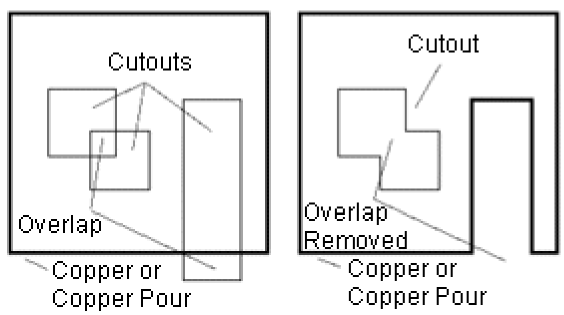
Overlapping Coppers

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.

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 SailWind 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.

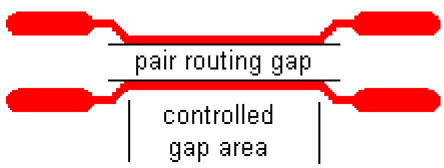
Pad oversize is measured from the center of the pad, not the perimeter. For example, if you have a 3 mil oversize, the measurement is actually1.5 mils in each direction from the center of the pad.

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.

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 Decal Wizard Dialog Box, BGAPGA Tab on page 1232.

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 to optimize an existing placement.

plane hatch outline

The outline of a copper plane shape after it has been flooded to differentiate it from the plane pour outline as originally drawn. When you draw the copper 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 command PO.

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. You can switch plane display modes using the shortcut modeless command PO.

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.

power board

A board that is designed to control power to other circuit boards.

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.

SailWindpcb.ini

The SailWind Layout initialization file for default settings.

powerpcb.mdb

The SailWind Layout message file that contains error messages, prompts, and other miscellaneous text strings. This file must be located in the same folder as *SailWindPCB.exe*.

powerpcb.reg

A file that defines all Registry keys required for the proper registration of SailWind Layout OLE components. In addition, other programs acting as clients access the SailWind Layout Automation Server through this Registry file.

The installation program automatically creates this file and saves it in the same folder as *SailWindPCB.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 enable 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 component side

The mounted side of the board when using through-hole components. See also *secondary component side*

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:

- layers
- components
- part decals
- net objects (including nets and pin pairs)
- 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 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.

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, enable 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.

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 SailWind 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 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. SailWind Layout permits you to renumber the reference designators in one of several specific patterns enabling you to quickly find a part among thousands on the manufactured board. The reshuffled numbers that are rearranged in SailWind 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 on page 1666 of SailWind Layout or Via Biasing properties in SailWind 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 SailWind stroke font, used to display text in SailWind 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.

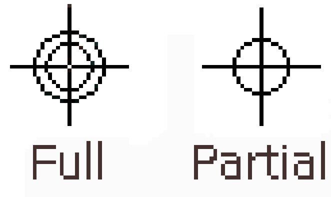
route

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 on page 1627).



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, SailWind 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. Includes half the Discrete length value of each connected pin of components that have a Discrete length assigned. If you start routing from the endpoint of a partially routed trace, 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.

• *center pass* • *fanout* • *miters pass* • *optimize pass*	• *patterns pass* • *route pass* • *test point pass* • *tune pass*

routing strategy

The collective information SailWind Router uses to autoroute a design. This information includes which pass types SailWind 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.

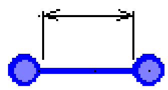
rules set

A specific set of user-assigned nondefault rules such as pin pair, groups, or classes.

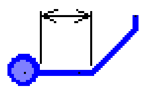
same net checking

Checks clearances between objects along the same net, as specified in the Clearance Rules dialog box. Object to object checking includes:

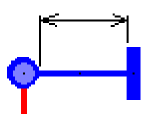
- Pad edge to pad edge.



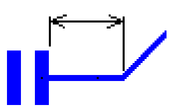
- Pad edge to inside corner of trace.



- 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 component side

The side opposite the mounted side for through-hole components. This side is typically wave soldered.

See also *primary component side*

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:

- net class
- pin pair group
- conditional rule
- matched length net group
- matched length pin pair group
- 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 SailWind 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

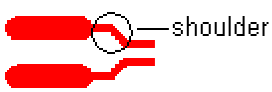
A key sequence that starts a command directly from the keyboard and without navigating through menus.

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 SailWind Logic during part type creation.

signal via

Via used to continue a signal from one layer to another.

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 SailWind Layout command that reroutes existing traces by enabling 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 *SailWind Router 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. 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 or a gang opening.

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 command that rotates a component or object around its axis or selection point.

split

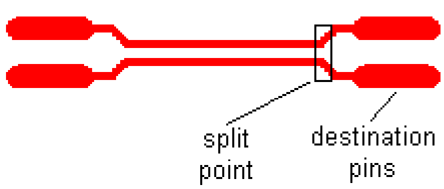
The command that creates a new corner at the pick point of the selected trace, enabling it to be rerouted.

split plane

A solid copper plane layer divided into two or more sections 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

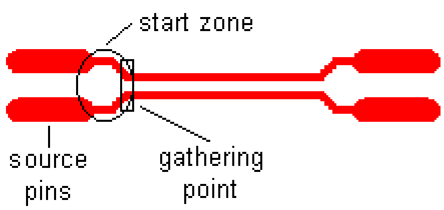
A program used to recommend set statement settings to properly adjust port access times for a security key.

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.

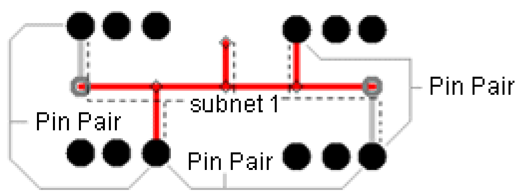
subnet

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*.

Figure 225. One subnet in a net

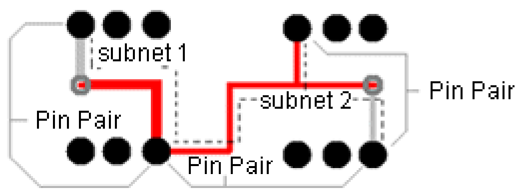


subnets

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 *differential pairs*, *differential pairs*.

Figure 226. Multiple subnets in a net



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 SailWind product configuration files.

system attribute

An attribute that is set by, used by, and critical to a SailWind 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 SailWind product configuration files.

T junction

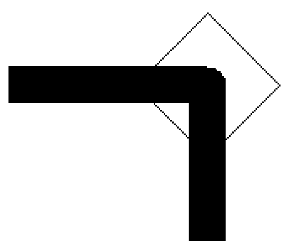
A trace that branches into another.

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.

tented vias

Vias that are covered by solder mask on both sides of the board to seal the hole and protect it from wave solder.

terminal

The electrical center of a pin, as defined in the part decal.

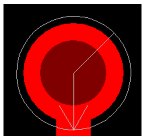
terminator

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.

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 copper plane.

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 plane, to the surrounding copper. Thermal reliefs provide good pin soldering by preventing heat from dissipating throughout the plane layer.

thermal via

Via used to dissipate heat from an area or component.

thick-film process

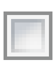
A hybrid microelectronic process where conductors, insulators, and passive components are screened from special pastes onto the substrate.

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.

three-state check box

A three-state check box helps to identify the state of the check box in a collection of objects.

Check box	Description
	Cleared check box. The item or collection of items all have the check box in the cleared or unchecked state.
	Selected check box. The item or collection of items all have the check box in the selected or checked state.
	Indeterminate or mixed state check box. The collection of items have different states of the check box.

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 do not 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 will not 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.

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 Editing Pad Stacks on page 188

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 electrical net or pin pair. Includes half the Discrete length value of each connected pin of components that have a Discrete length assigned.

Total length is reported for pin pairs when all the following are true: length rules are defined for the pin pair, the electrical net is a high-speed net, and copper sharing is disabled.

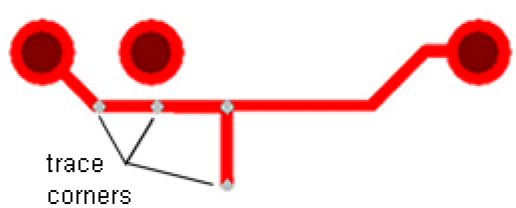
If pin pair rules are reported, the estimated total length of the pin pair is shown; otherwise, total length for nets is reported.

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 endpoint of a partially routed trace. The trace segments may be in line.

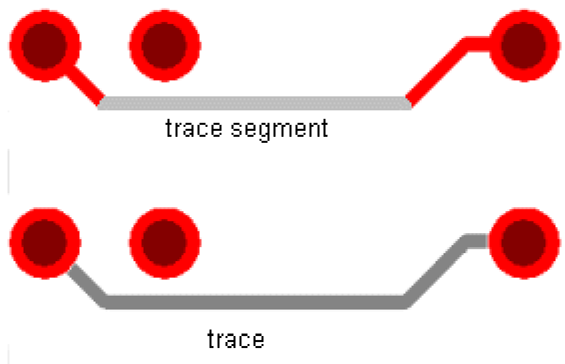


trace paths

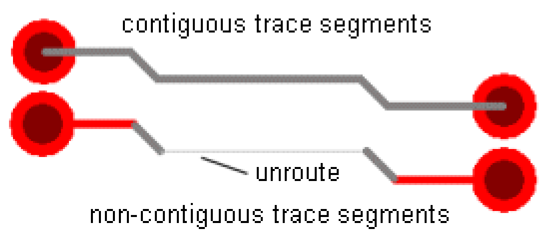
A continuous sequence of trace segments in the same trace on the same layer. Paths start and end at nodes, and cannot pass through a node.

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 SailWind 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 on page 1189.

By default, TrueLayer mode is enabled. To disable it, use the /NTL command-line switch. See Software Launch Options on page 50.

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.

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 enables 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 *SailWindpcb.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 endpoint 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 SailWind product configuration files.

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*.

via type

A via or virtual pin padstack definition that is defined and named in SailWind Layout in the Pad Stacks Properties dialog box.

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 pin

A net object that, like a component pin, serves as a pin pair end, but uses the pad stack of a via. The pad stack can be through-hole or partial, or it can be a single-layer pad.

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 scripting language developed by the Microsoft Corporation to enable users to customize 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*.

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

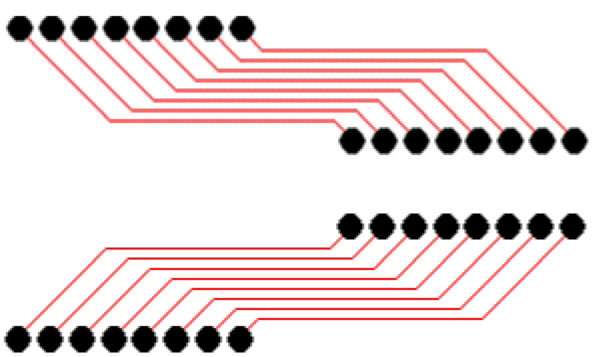
The actual work area where a design is created.

yield

The ratio of the number of acceptable units to the maximum number possible.

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*.

Third-Party Information

Details on open source and third-party software that may be included with this product are available in the *<your_software_installation_location>/ThirdParty* directory.