PAIZI

SailWind Router Guide

Release SailWind 3.0
Document Revision 1.1

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	2024-03-15
1.1	Updated descriptions on suite configuration in "Help > Installed Options" menu.	2024-09-27

Table of Contents

Chapter 1
Autorouting QuickStart

Welcome to SailWind Router, a product containing an autorouter implemented on Latium technology.

Perform the following steps to get started quickly.

Alternatively, watch the SailWind Layout and SailWind Router Getting Started Videos for Netlist Projects or for all the advanced features in SailWind Router for placing and routing your design, see SailWind Router Introductory Videos for Netlist Projects.

- Step 1 - Load a SailWind Layout File
- Step 2 - Assign Default Design Properties
- Step 3 - Assign Routing Options
- Step 4 - Define a Strategy
- Step 5 - Define a Routing Order
- Step 6 - Autoroute the Design
- Step 7 - Generate Reports

Step 1 - Load a SailWind Layout File

SailWind Router reads and writes native SailWind Layout design files.

Procedure

1. On the standard toolbar, click the **Open** button. 
2. In the Files of type area, click **PCB Design Files (*.pcb,*.bre)**.
3. Click the SailWind Layout design file you want and click **Open**. For example, open the file named *previewplaced.pcb* in the *\My Documents\SailWind Projects\Samples* folder. The file opens in SailWind Router.
4. If the file you load contains traces, delete all the traces using Unroute.
    a. Right-click in the design area and click the **Select Nets** popup menu item.
    b. Right-click and click **Select All**. All nets are selected.
    c. Right-click and click the **Unroute** popup menu item.

    Alternative: Press the Delete key.

Step 2 - Assign Default Design Properties

All design properties and design rules set in SailWind Layout transfer automatically to SailWind Router when you load a SailWind Layout file. You can, however, change them in SailWind Router.

Procedure

1. On the standard toolbar, click the **Properties** button.

> **Note:**
> Ensure nothing is selected in the design when you click the **Properties** button or the resulting dialog box displays properties of the selected object.

2. Assign properties using the tabs in the dialog box. See Table 1.

3. Click **OK** to close the Properties dialog box. Access to some tabs may require optional features not available in your SailWind Router configuration.

Table 1. Properties Dialog Box—Tabs

Tab	Use to
Clearance	Assign default object to object clearances.
Same net	Assign default same net object to object clearances.
Routing	Assign default trace width, default ripup, and shove preferences.
Via Biasing	Assign default via type bias.
Layer Biasing	Assign default layer bias.
Test Points	Assign default test point clearances and stub length.
Layers	Enable layers and assign routing directions.
Grid	Assign default grid values.
Fanout	Assign parameters for fanning out from SMD pads.
Pad Entry	Assign rules for how traces enter and exit pads.
Topology	Assign trace topology preferences.

For more information on each tab, click **Help** in the tab.

Step 3 - Assign Routing Options

All design and display options you set in SailWind Layout transfer automatically to SailWind Router. You can, however, change them in SailWind Router. Additionally, SailWind Router contains a few unique autoroute options.

Procedure

1. On the standard toolbar, click the **Options** button. 
2. Assign autorouting options using the tabs in the dialog box. Access to some tabs may require optional features not available in your SailWind Router configuration.

   Click the **Help** button for more information about the features on each tab.

Step 4 - Define a Strategy

The autoroute function performs several autorouting operations serially. You can specify the operations and the order in which they are performed in SailWind Router by defining the *autorouting strategy*.

Procedure

1. Click the **Tools > Options** menu item > **Routing** category > **Strategy** subcategory.
2. Select the check box in the Pass Type column for each pass that you want to include in the autorouting operation.
   a. Select the corresponding Protect check box to preserve all routed traces after the pass is completed.
   b. Select the corresponding Pause check box to pause autorouting immediately after the pass completes.

Step 5 - Define a Routing Order

You can define the order in which the autorouter routes all components and net objects for each pass type. This is called the routing order.

Procedure

1. Click the **Tools > Options** menu item > **Routing** category > **Strategy** subcategory.
2. In the “Pass definition” area, click the row of a pass type name to select it for routing order setup.
3. In the “Routing order definition” area, click components or net objects from the left-hand list and click **Selected** to move them to the Routing Order list (right-hand side).
4. Change the order of items in the Routing Order list by selecting items and clicking the **Up** or **Down** button.
5. Click **Delete** to remove an item from the list.
6. Click **OK**.

Step 6 - Autoroute the Design

Use the Routing toolbar to control the autorouter.

Procedure

1. On the standard toolbar, click the **Routing toolbar** button. 

   Table 2 describes the buttons that appear in the Routing toolbar.

Table 2. Autorouter Controls

Button	Control
Start Autorouting 	Begins the autorouting process.
Resume Autorouting 	Continues the autorouting process after you pause it.
Pause Autorouting 	Pauses or interrupts the autoroute process. The software resumes from the point where it paused when you click the Resume Autorouting button.
Stop Autorouting 	Halts the autorouting process. The software leaves any portions that have already been routed in place.

2. Click the **Start Autorouting** button to begin autorouting the design.

Step 7 - Generate Reports

After the autorouting routine completes, it generates a routing report detailing the results. A link to the report appears in the **Status** tab of the Output window.

Procedure

1. If the Output window is not already open, click the **Output Window** button on the Standard toolbar.
   

2. Click the link in the **Status** tab to open the routing report. Additional reports can be generated at your request by clicking the **File > Reports** menu item and then clicking either the **Design**, **Testability**, or **Test Points** report category.

Chapter 2
Interactive Routing QuickStart

If desired, you can route all or part of your design manually rather than using the autorouter process.

Perform the following steps to get started quickly.

Alternatively, watch the SailWind Layout and SailWind Router Getting Started Videos for Netlist Projects or for all the advanced features in SailWind Router for placing and routing your design, see SailWind Router Introductory Videos for Netlist Projects.

Restriction:
Access to some tabs or features may require options not available in your SailWind Router configuration

---

Step 1 - Open a SailWind Layout File

SailWind Router reads and writes native SailWind Layout design files. Perform the following steps to load a SailWind Layout file.

Procedure

1. On the standard toolbar, click the **Open** button.
2. In the Files of type list, select **PCB Design Files (*.pcb,*.bre)**.
3. Click the SailWind Layout design file you want and then click **Open**. For example, open the file named *previewplaced.pcb* in the *C:\SailWind Projects\Samples* folder. The file opens in SailWind Router.
4. If the file you load contains traces, delete all the traces using Unroute.
   a. Right-click in the design area and click the **Select Nets** popup menu item.
   b. Right-click and click the **Select All** popup menu item. All nets are selected.
   c. Right-click and click the **Unroute** popup menu item.

      Alternative: Press the Delete key.

Step 2 - Set Default Design Properties

All design properties and design rules set in SailWind Layout transfer automatically to SailWind Router, although you can change them in SailWind Router. SailWind Router also has some additional unique properties.

Procedure

1. On the standard toolbar, click the **Properties** button. 
2. Assign properties using the tabs in the dialog box. See Table 3.
3. Click **OK** to close the Properties dialog box. Access to some tabs or features may require options not available in your SailWind Router configuration.

Table 3. Design Properties Tabs

Tab	Use to
Clearance	Assign default object to object clearances.
Same net	Assign default same net object to object clearances.
Routing	Assign default trace width, default ripup, and shove preferences.
Via Biasing	Assign default via type bias.
Layer Biasing	Assign default layer bias.
Test Points	Assign default test point clearances and stub length.
Layers	Enable layers, set their costs, and assign their routing directions.
Grid	Assign default grid values.
Fanout	Assign parameters for fanning out from SMD pads.
Pad Entry	Assign rules for how traces enter and exit pads.
Topology	Assign trace topology preferences.

4. For more information on each tab, click **Help** in the tab.

Step 3 - Route Interactively

Interactive routing allows you to choose the desired route manually. You can set the level of automatic software input during interactive routing by setting certain options and enabling certain features beforehand.

Setting interactive routing options

Set your design options before routing interactively.

You can change the options as desired; however, the following procedure provides recommended starting settings.

Procedure

1. Click the **Tools > Options** menu item > **Routing** category > **General** subcategory.
2. In the "Routing angle" area, select the angle you prefer (Orthogonal, Diagonal, or Any angle).
3. In the Interactive routing area, clear the "Dynamically route" check box.

   Tip
   When routing interactively, one segment is added at a time. When you click the Dynamically route check box, multiple segments and corners are added as you move the pointer.

4. In the Plower area, select the "Turn on plower" check box. Specific software configurations have the ability to push and shove traces while interactively adding traces. This is done by turning on the plower. The plower will be used in a later step. The plower pushes and shoves traces aside to help you route.
5. Click the "Plow with pointer" option. This allows you to plow without having to click to engage the plower.
6. Click Real-time plowing. This plows immediately without having to guide the trace to an open area.
7. Review the other settings in the Interactive routing area and on the other tabs of the Options dialog box.

   Note:
   On the **Colors** tab, ensure the "Connection" color is set to a visible color.

8. Click **OK**.

Interactive routing

SailWind Router can route your connections intelligently, responding to obstacles in the path and adjusting the route accordingly to ensure design rules are met.

Procedure

1. Right-click and click the **Select Unroutes/Pins** popup menu item.
2. Select an unroute or pin in the workspace.
3. Right-click and click the **Interactive route** popup menu item.

   Alternative: Press the F3 key.
4. Move the pointer and click to create each corner in the trace.

   Tip
   When you select the Dynamically route check box (in the Options dialog box > **Routing** category > **General** subcategory), you do not need to click to add corners. Corners and segments are automatically added.
5. Click on the object where you want to complete the trace when a single or double circle is visible over the completion point.

   Alternative: When the pointer is near the object at which you want to complete the trace, right-click and click the **Complete** popup menu item.

Step 4 - Plow traces while routing interactively

With the plower enabled, the interactive router clears a path for the trace you are routing while maintaining existing connections.

Procedure

1. Select an unroute or pin, right-click and click **Interactive route**. While routing, note that if you attempt to cross another trace, or get closer than the minimum trace-to-trace clearance, the other trace shoves aside to maintain the minimum clearance.
2. While routing a trace, right-click and notice your plower options. Experiment with the plower options. See Table 4.
3. Complete the connection. While routing, you can temporarily change plower settings to complete a trace.

Table 4. Plower Options

Command	Use to
Push Trace Behind	Allow traces to be pushed behind the pointer as you interactively route
No Plowing	Turn off plowing for the trace in progress

Command	Use to
Plow After Click	Reroute obstacles after you click a corner. If you select Plow After Click, it only plows after you click to enter a corner or a via.
Plow with Pointer	Reroute obstacles as you move the pointer
Rip Up Obstructing Traces	Unroute obstacles

Chapter 3
Preparing a Design

Your design must meet certain requirements before you import it into SailWind Router. After carefully preparing your design for import and ensuring it meets all the prerequisites, you can import it using a synchronization process for switching back and forth between SailWind Layout and SailWind Router.

Advanced Features in SailWind Router

SailWind Router provides advanced features that enable you to perform routing, placement, and design rule checking at a level of detail beyond that of SailWind Layout.

Tip
Alternatively, you can watch videos of the advanced features in SailWind Router for placing and routing your design. See SailWind Router Introductory Videos for Netlist Projects.

SailWind Router includes the following advanced features.

- **Unique component placement grid** — Having components on a separate grid allows you to use another grid with a different resolution simultaneously for routing the design.
- **Visual design rule checking while placing components** — Visual indicators, such as markers for clearance violations, appear while placing components. A description of each error appears simultaneously in the Spreadsheet window, providing quick viewing during placement.
- **Advanced design rule checking** — While performing design rule checks, you can specify which design items to check and what action you want SailWind Router to take for any violations or errors it encounters (such as ignore them). With design rule checking enabled, you can also pause when errors appear during actions such as routing or component placement. You can examine such errors in detail in the Spreadsheet window, allowing you to modify your design when necessary.
- **Advanced routing capabilities** — SailWind Router provides additional routing tools not available in SailWind Layout. For example, you can implement design rules for specialty design features such as differential pairs, accordions, and fanouts. Other capabilities include the following:
    - Tuning
    - Pad centering
    - Visual design rule checking indicators while routing

    - Automatic test point additions
    - Net scheduling
    - Trace plowing

- **Customization of Keepout fill** — Typically, keepouts created in SailWind Layout appear hatched in SailWind Router. If desired, you can change the fill pattern of your keepouts using the extended keepout fill options in SailWind Router to distinguish them from copper, fills, and other plane areas. You can access fill pattern settings for both keepouts and copper, fills, and planes in the Options dialog box > **Global** category > **General** subcategory.

- **Dynamic copper healing** — Flooded copper in copper planes adjusts automatically to changes made within the shape. You do not need to flood the shape to see updates to clearances in the fill. For example, if you move a component, the copper floods to the required clearances around its new position immediately. You can enable the dynamic copper healing feature in the Options dialog box > **Flooding** category.

- **Navigation window** — This window provides an alternate view of the workspace while navigating your design:

    - Displays a magnified view of the design when nothing is selected
    - Displays cross sectional details when a pin or via is selected
    - Magnifies the view of any component selected in the design
    - Magnifies the view of any net objected selected in the design
    - Monitors and displays the routed length of any length-controlled net during interactive routing

  The Navigation window features command buttons that let you show, hide, or zoom in and out from various design features (such as traces or components) from the view in the Navigation window.

- **Spreadsheet window** — This window lets you view and modify design object properties or cross probe design rule checking violations flagged in the design. For example, design rule violation details appear in the **Error** tab of the window. You can thus view design error details as they occur while routing or placing components, allowing you to make design modifications as necessary.

- **Autorouting capability** — You can choose to perform batch autorouting and you can also select individual nets, components, or pin pairs for selective autorouting.

- For batch autorouting, determine your strategy first. For example, route or plan to route denser components (such as BGAs) before routing other components.

> **Tip**
> Even if you are planning to manually route your design, you can use the autorouter to make a “first pass” at the design to determine the tightest channels and most problematic areas.

- For selective autorouting, select a component or net, then right-click and choose one of the manual routing options (Route, Fanout, Optimize, Tune, or Center). For best results, protect the routes when finished to ensure they remain unchanged during any subsequent batch autorouting.

Related Topics

Spreadsheet Window

Options Dialog Box, Global Category, General Subcategory

Options Dialog Box, Flooding Category

Prerequisites for Opening a Design in SailWind Router

You must complete a number of prerequisite tasks prior to bringing a design into the SailWind Router workspace. Your particular design may or may not require some tasks.

Requirements

You must complete these tasks in SailWind Layout prior to opening the design in SailWind Router:

- Create the board outline
- Define the layer structure of the board including the number of layers and their functional assignments (as a minimum you need to define component mounting layer(s), routing layer(s), and if they are required - power/ground planes)
- Define any keepouts:
    - Placement (including component height restrictions)
    - Component drill
    - Trace and copper
    - Copper planes
    - Via and jumper

    - Test points
    - Accordions

- Import the design netlist from your schematic tool
- Define the padstacks for all of the vias that you will be using to route the board (standard, micro/laser, blind, and/or buried)
- Define any coppers/copper cutouts, copper planes/copper plane cutouts, hatch areas (Note: Router sees hatch outline differently than pour outlines)
- Disperse the components
- Define any Reuse elements required by the design (functionality not available in SailWind Router)
- Create any component unions that may be required
- Check to be sure you have assigned the geometry.height attribute to components if you will be using any component height keepout restrictions

Recommendations

Though it is not a prerequisite to perform the following tasks before opening a design in SailWind Router, if you eventually need any of these design features, keep in mind that you can only complete them in SailWind Layout:

- Create any cutouts or slots in the board outline
- Define additional items in the layer structure of the board such as silkscreen, paste mask, solder mask, drill, assembly, or any other specific-use layers
- Add any non-ECO registered components such as tooling holes, mounting holes, fiducials, heat sinks, and hardware (optional, if used)
- Create any 2D line items required such as title block, board markings (text on board), graphic elements, solder mask reliefs (copper on SM layer), company logos, and other items that your company standards require

Synchronization Mode

Synchronization mode links SailWind Layout and SailWind Router. As you work in SailWind Router, changes you make also appear in SailWind Layout; as you work in SailWind Layout, the changes appear in SailWind Router. When you are in Synchronization mode, you can instantly switch between the two programs, but you are restricted in what you can do in the inactive program.

If you perform an unsupported command or change an unsupported option, the inactive program becomes unsynchronized.

Note:
You must and then restart SailWind Layout before Synchronization mode takes effect.

Despite synchronizing with SailWind Router, you must actively switch back to SailWind Layout in order to work on your design in SailWind Layout.

You can still use automation; however, most Automation methods will not return any valuable results and will not make any changes in the database.

Enabling Synchronization Mode
Sending a Design to SailWind Router in Synchronization Mode
Switching to SailWind Layout
Unsupported Actions in Synchronization Mode
Resynchronizing When the Design Becomes Unsynchronized
Exiting Synchronization Mode

Enabling Synchronization Mode

Synchronization mode links SailWind Layout and SailWind Router so that the design appears in both programs simultaneously, enabling you to work in either tool without closing the design in one program first.

CAUTION:
Synchronization mode can only be set from SailWind Layout and must be initiated prior to trying to send a design to SailWind Router.

Procedure

1. In SailWind Layout, click the **Tools > Options** menu item.
2. In the Options dialog box, click the **Global** category, **Synchronization** subcategory.
3. In the Layout and Router Synchronization area, select the Enable check box.

   Switching to SailWind Router automatically places the design inside SailWind Layout in DRC Off mode (the DRC mode in SailWind Router is not affected); therefore, you may want to select the “Restore DRC Mode on return” check box. This setting restores the DRC mode that existed before switching to SailWind Router when you switch back to SailWind Layout. Depending on the size of your design, restoring the DRC mode may take a few minutes.

4. 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” (this check box is not available if you select the “Restore DRC mode on return” check box in step 3).

5. Click **OK**.

6. Restart SailWind Layout.

Sending a Design to SailWind Router in Synchronization Mode

Use Synchronization mode to transfer your design to SailWind Router while keeping the design open in SailWind Layout.

Procedure

1. Enable Synchronization mode as described in “Enabling Synchronization Mode” on page 33.

2. Switch to SailWind Router using one of the following:

   - On the Standard Toolbar, click the **Route** button. 

   - Click the **Tools > SailWind Router** menu item, and in the SailWind Router Link dialog box, select an Action and click **Proceed**.

Results

SailWind Router opens and loads your design.

Related Topics

Exiting Synchronization Mode

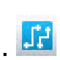
Resynchronizing When the Design Becomes Unsynchronized

Unsupported Actions in Synchronization Mode

Switching to SailWind Layout

You can quickly move your design from SailWind Router back into the SailWind Layout environment.

Procedure

1. On the main toolbar, click the **Layout** button. 

2. SailWind Layout opens and your design is loaded.

> **Tip**
> To switch back to SailWind Router, click the **Route** button on the main toolbar in SailWind Layout.

Unsupported Actions in Synchronization Mode

Synchronization mode does not support some commands and options. If you perform one of these commands, or change one of the options, SailWind Layout and SailWind Router become unsynchronized.

Unsupported commands

- Changing a drafting object: copper, copper planes, text, 2D lines, keepouts, labels, board outlines, and dimensioning. However, you can move a component that has drafting objects attached to it without causing the programs to become unsynchronized.
- 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

- Design For Fabrication (DFF)
- Design for Test (DFT)
- Verify Design
- Drafting
- Split/Mixed Plane
- Thermals

- ECO Options
- Via Patterns

Related Topics

Resynchronizing When the Design Becomes Unsynchronized

Resynchronizing When the Design Becomes Unsynchronized

If you perform an unsupported command or change an unsupported option, the inactive program becomes unsynchronized. The status bar displays the synchronization status - Active, Inactive, or Out of sync. You must resynchronize the design in SailWind Layout or SailWind Router.

Procedure

To re-synchronize the two programs:

- If SailWind Router is “Active” and SailWind Layout is “Out of sync”, on the main toolbar in SailWind Router, click the **Layout** button.
- If SailWind Layout is “Active” and SailWind Router is “Out of sync”, on the main toolbar in SailWind Layout, click the **Route** button.

Note:
To turn off the display of licensing warnings when switching between SailWind Layout and SailWind Router, type the following text into the *SailWindRouter.ini* file (you do not need to restart SailWind Router afterward):

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

Results

The previously out of sync program is updated with the latest design changes and becomes the active program.

Related Topics

Unsupported Actions in Synchronization Mode

Exiting Synchronization Mode

You can exit Synchronization mode at any time after making design changes.

Procedure

1. Determine which application you would like to continue working in.
2. To exit Synchronization mode, close the other application.

   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; therefore, 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 Router, but no longer want to be in Synchronization mode, close SailWind Layout. When prompted, save your design.

Chapter 4
File Operations

SailWind Router users can perform basic operations from within the software, including opening design files, scheduling automatic file backups, and archiving designs.

About Opening Files

SailWind Router can open certain file types. Opening each file type, however, has associated limitations.

File Types

SailWind Router allows you to open SailWind Layout and PowerPCB or PowerBGA files. You can also open SailWind Router backup files.

SailWind Router supports the following file types:

- SailWind Layout 2005 (or later) *.pcb* files
- PowerPCB and PowerBGA versions 3.5 (or later) *.pcb* files
- SailWind Router backup (*.bre*) files

You cannot open a file that another user has open.

When you open a file, the software runs an integrity test to ensure the database is stable. The session log records the names and paths of the tested file and the test report as links.

Tip
Click the corresponding link in the sessions log if you want to reopen a file you worked on earlier in the session.

Note:
When you exit SailWind Router, all current settings (including the position and size of toolbars, dialog boxes, and windows) save to the Windows Registry. SailWind Router uses these saved settings the next time you run the program.

Related Topics

Testing Database Integrity

Text Fonts When Opening a File

When you open a *.pcb* file in SailWind Router, any text that uses system fonts displays it with the same font, font size, and font style as in SailWind Layout.

If any text in the file uses fonts not installed on your computer, a message in the Status Window indicates that text using those fonts will be replaced by empty boxes. These boxes appear in the same size as the text they replace, so you can examine the empty boxes for clearances, even though you cannot view the actual text.

Differential Pair Trace Width Settings When Opening a File

You can set the width and gap for differential pair traces for all layers, or per layer. (For early versions, SailWind Router set the width and gap.)

For information on setting the width and gap per layer, see "“Setting Differential Pair Properties” on page 184”.

For *.pcb* files created prior to PADS2005 SPac2, SailWind Router sets a width and gap for each differential pair, as follows:

- Sets the width for all layers as the larger of the recommended widths for each member of the differential pair.
- Sets the gap assigned to the differential pair for all layers.

Layer Count for Security When Opening a File

When you open a file, SailWind Router checks the number of layers available for routing in your design against your security limits.

The following layers are counted as layers available for routing:

- CAM plane layers with routes
- Top and bottom layers with components
- Electrical layers
- Split/mixed plane layers

If your design has more layers available for routing than those for which you are licensed, all split/mixed plane layers without routing or components are disabled for autorouting. (You can view the disabled layers in the **Layers** tab of the Design Properties dialog box.)

For example, assume your design is a six-layer board (four routing layers and two split/mixed layers without routing or components), and you have a four-layer license. When SailWind Router reads this design, it disables the split/mixed plane layers that have no routing or components. (Split/mixed plane layers with routing or components are not disabled.) The resulting layers available for routing count of four is within your security limits (a four-layer license), and the design can be routed.

To enable a disabled layer, you must obtain a higher layer count option in your license file. For floating security, you may be able to check out a higher limit. For node-locked licensing, you may need to purchase a new configuration. In either case, you must restart the program for the new settings to take effect.

T-Junctions When Opening a File

When you open a SailWind Layout design in SailWind Router and the design violates the no trace sharing rule, SailWind Router disconnects the affecting trace, depending on the structure of the route pattern.

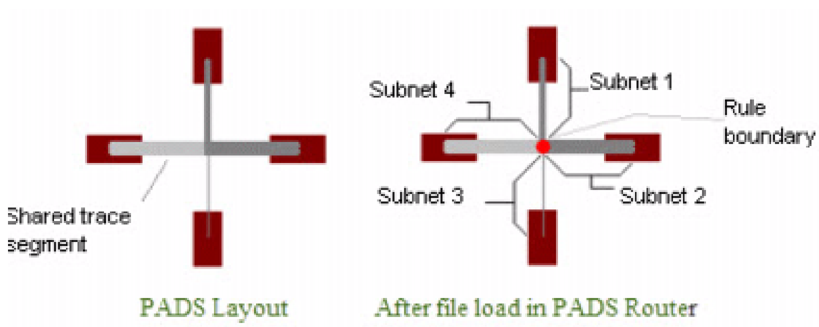
If no more than two pin pairs share the segments, SailWind Router establishes a rule boundary automatically to make the design comply with the program's rules. "Rule boundaries" on page 42 improve the management of these conflicts and eliminate the requirement to disconnect traces when opening a file.

Subnet Conflicts When Opening a File

When you open a design and this design has subnet overlaps, the nets with subnet conflicts convert to structures the program can manage. Subnet conflicts occur when a trace or via belongs to several subnets.

SailWind Router adds rule boundaries to avoid unrouting portions of a trace when pin pairs of a net share a single trace segment.

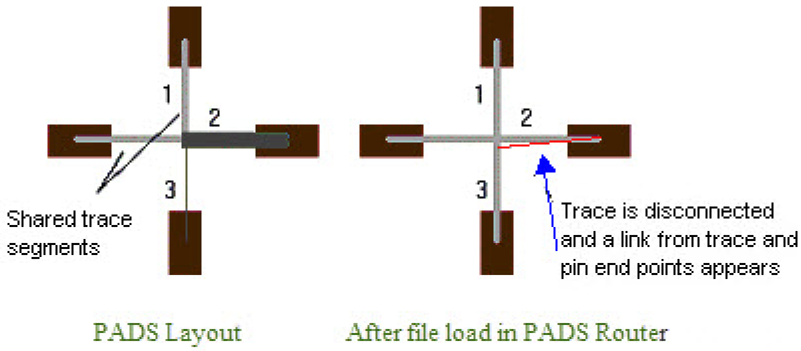
Figure 1. Rule Boundaries When Pin Pairs Share a Trace Segment



When pin pairs of the same net share two trace segments that converge at one point, the software creates two overlapping rule boundaries. Because the software cannot resolve this type of conflict, it disconnects some traces.

Figure 2 shows how the software disconnects such traces.

Figure 2. Disconnected Traces



Opening a File

You can open SailWind Layout *.pcb* design files and SailWind Router *.bre* backup files.

When you open a file:

- Design Rule Checking (DRC) is enabled, regardless of the setting in SailWind Layout.
- In-line trace corners are removed.

For more detailed information about opening files, and about processes that occur during opening, see ““About Opening Files” on page 40.”

Restrictions and Limitations

SailWind Router does not open *.bre* files saved using a different SailWind Router version.

Procedure

On the standard toolbar, click the **File > Open** menu item or click the **Open** button on the main toolbar.

Results

If SailWind Router discovers incompatibilities when importing an ASCII file created in a version prior to PADS 9.2, it displays a warning prompt and writes the incompatibilities to the ascii.err file.

Scheduling File Backups

You can schedule backups to occur automatically while a design is open for editing.

Procedure

1. Click the **Tools > Options** menu item > **Global** category > **Backups** subcategory.
2. In the Options Dialog Box, Global Category, Backups Subcategory, in the Interval box, type the time in minutes between backups.
3. In the Number of Backups box, type the maximum number of backups you want to keep.

   When the number of backups reaches this limit, the oldest existing backup is deleted whenever a new backup is created.

   Tip
   This limit can specify the maximum number of backups for all designs, or the number of backups for each design. For more information, see the table "Table 88" on page 438.

   ---

4. Select the "Use design name in backup file name" check box to use "<*design name*>_Router" instead of "Router" in the backup file name.

   For example: <*design_name*>_Router_<*date_time*>.bre, instead of Router_<*date_time*>.bre
5. Select the "Create backup files in design directory" check box to place backup files in the same directory as the design.
6. Click **OK**.

Results

Backups will be saved according to settings made in this procedure, either in the design directory, or in the default Backup location specified in the "Options Dialog Box, Global Category, File Locations Subcategory" on page 439. Also, a link to the saved backup is recorded in the session log that appears in the **Status** tab of the Output window.

Related Topics

About Opening Files

Opening a File

Archiving Your Design

You can create a folder or a *.zip* file that contains all of your design files and supporting files for archive purposes. The folder or zip file may include the design itself, a schematic file, libraries, and any additional files or folders you want to save.

Procedure

1. Open the design you want to archive.
2. Click the **File > Archive** menu item.
3. Select the files and folders you want to archive (see "Archiver Dialog Box" on page 389).
4. Click **OK**.

Results

- If you chose to not compress the files, SailWind Router archives the selected items in the target folder you specified.

- If you chose to compress the files, SailWind Router creates a *.zip* file containing the selected items in the target folder you specified. The filename has the following format:

  ```
  <project_name>YYYYMMDDHHMMSS.zip
  ```

  Where YYYY is the year, MM is the month, DD is the day, HH is the hour (using the 24-hour clock system), MM is the minute, and SS is the second of the time you created the file.

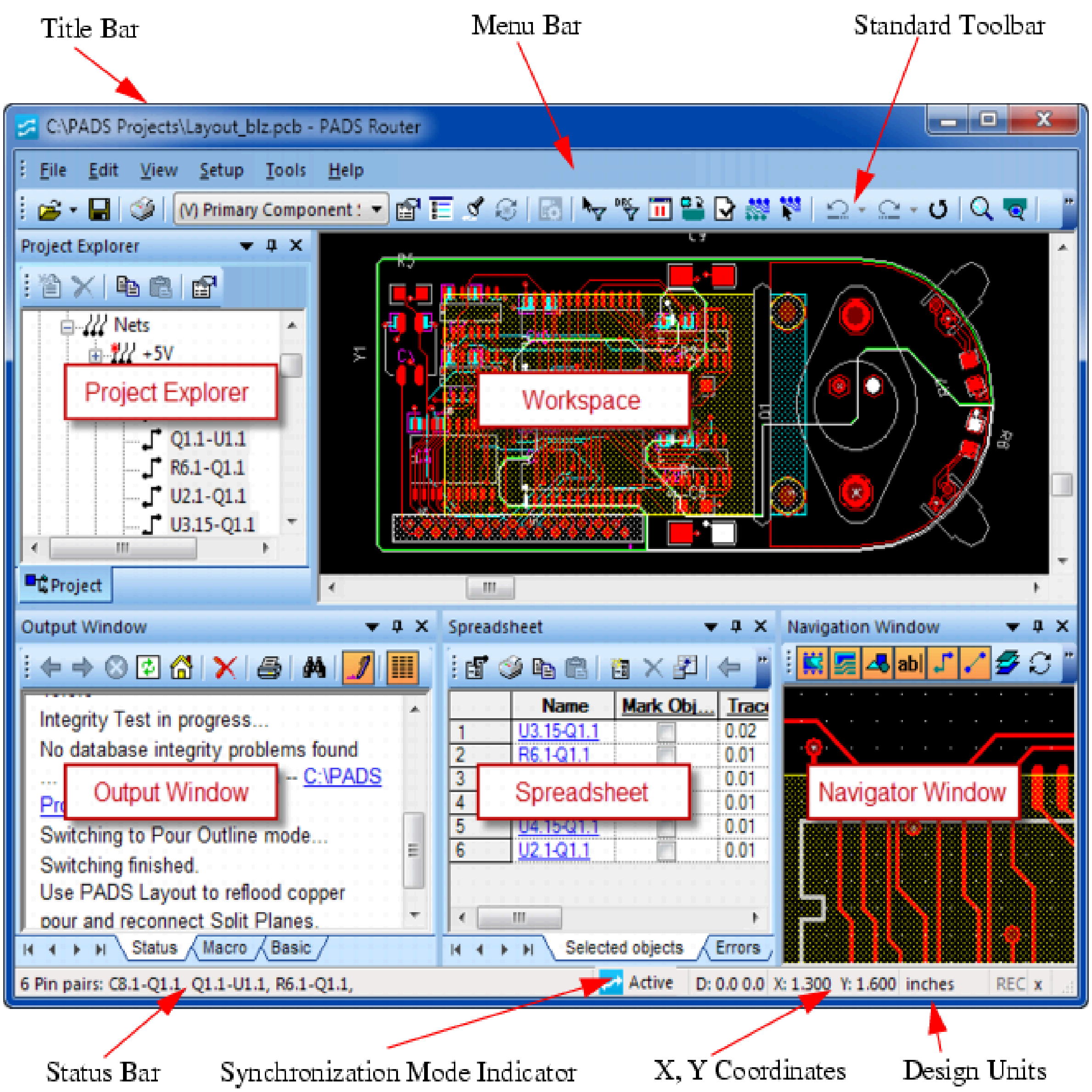
Related Topics

Archiver Dialog Box

Chapter 5
Interface Elements

The SailWind Router interface provides standard Windows menus and buttons for accessing commands and settings.

Figure 3. SailWind Router User Interface



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 Router in practice shall prevail.

- **Title Bar** — The application icon, document name, and application name appear in 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 Router" appears as the document name. When a design file is loaded, the path, the filename, and the file extension (.pcb or .bre) appear in the title bar.

- **Menu Bar** — The menu bar lists SailWind Router commands. The menus also show the appropriate command icons, access keys, and 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 the design, and access the toolboxes.

- **Status Bar** — The status bar contains status messages, cursor coordinates, and a system status indicator. When the system status indicator is green, the system is idle or ready for operation. When the system status indicator is red, the system cannot receive user input. The status bar also has a ToolTip that is a routing progress indicator. For more information see ““Routing Progress Indicator” on page 277.”

- **Synchronization Mode Indicator** — Indicates that 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. Enable synchronization mode from SailWind Layout to make this indicator available. See “Enabling Synchronization Mode” on page 33 for more information.

- **Line Width**— Displays the current line width setting.

- **X,Y Coordinate**— Displays the horizontal distance of the cursor from 0,0 as the x-coordinate and the vertical distance of the cursor from 0,0 as the y-coordinate. Also indicates the polar radius and the polar angle if you are using a polar grid.

- **Units** — Displays the current units in the design: inches, mils, millimeters, or microns. All values in dialog boxes display in these units and values you enter are interpreted in these units. For information about setting the design units, see “Options Dialog Box, Global Category, General Subcategory” on page 441.

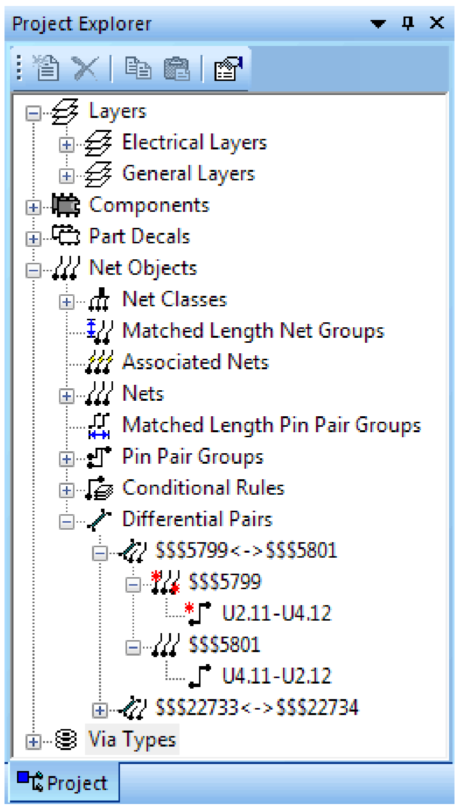
Project Explorer
Output Window
Navigation Window
Spreadsheet Window

Project Explorer

Project Explorer gives you a view of the objects in your design that is different from the view in the workspace. Instead of showing design objects in their places on the board, it lists them in a hierarchical alphabetized list. The Project Explorer view can make it easier to find and select objects.

Tip
To allow objects selected in the Project Explorer to also highlight in the workspace, right-click in the window and click **Allow Selection**. To zoom to the item you select in Project Explorer, right-click and click **Zoom to Selection**.

Figure 4. Project Explorer



In the Project Explorer list, an asterisk in the top left corner of an object indicates that it has rules that are not the default rules. An asterisk in the bottom right corner of an object indicates that its secondary objects contain rules that are not the default rules.

When you change your design, the hierarchical list automatically updates to reflect the changes.

- Project Explorer Objects
- Project Explorer Operations
- Splitting the Project Explorer View
- Sorting Objects in Project Explorer
- Creating a New Object in a Secondary Group

Project Explorer Objects

Design objects in Project Explorer are organized into *primary groups* and *secondary groups*.

Table 5 shows the hierarchy of groups and objects.

Table 5. Hierarchy of Objects in Project Explorer

Primary Groups	Secondary Groups	Description
Layers	Electrical Layers	Lists all layers enabled for routing, including plane layers and routing layers.
	General Layers	Lists all non-electrical layers, such as solder mask and silkscreen.
Components		Lists all components and their corresponding pin pairs.
Part decals/PCB decals		Lists all part decals in the design or all components that use the selected part decal.
Net objects	Net Classes	Lists all nets belonging to net classes.
	Matched Length Net Groups	Lists all net groups set up to have equal total net lengths. You can create matched length net groups directly from within the Project Explorer. For more information, see Creating Matched Length Net Groups.
	Nets	Lists all nets in the design.
	Electrical Nets	Lists all electrical nets in the design. An *electrical net* is an array of nets jointed by discrete components. For more information, see Electrical Nets.
	Matched length pin pair groups	Lists all pin pair groups set up to have equal total net lengths. You can create matched length pin pair groups directly from within the Project Explorer. For more information, see Creating Matched Length Pin Pair Groups.
	Pin pair groups	Lists all nets belonging to pin pair groups (containing pin pair rules).
	Conditional rules	Lists all nets with conditional rules.
	Differential pairs	Lists all nets assigned as differential pairs. You can create differential pairs directly from within the Project Explorer. For more information, see Creating Differential Pairs.

Table 5. Hierarchy of Objects in Project Explorer (continued)

Primary Groups	Secondary Groups	Description
Via types		Lists the via types used in the design.

Project Explorer Operations

Use the Project Explorer to create delet rename and sort secondary group objects (rules objects).

Operations on Primary Groups in Netlist Projects

All objects listed under the Layers, Components, Part Decals, Via Types, and Net Objects/Nets groups are netlist objects. Because SailWind Router is a non-ECO product and does not support netlist changes, you cannot delete, rename, move, or cut any of these objects. You can, however, copy objects from these groups to create new members of secondary groups. As an example, you cannot delete a net from the Nets group; but you can copy a net into the secondary group Net Classes to create a new net class containing this net.

Operations on Secondary Groups in Netlist Projects

With the exception of the Nets group, you can create new members in all the secondary groups under Net Objects, and you can edit all members of those groups to some degree.

To create a new member of a group, copy/paste or drag and drop appropriate objects into the group. For instance, to create a new net class, drag and drop one or more nets into the Net Classes group.

Tip
You can also create and name a new *empty* member of the Net Classes, Matched Length Net Groups, Pin Pair Groups, and Matched Length Pin Pair Groups using the right-click > **New** command.

Table 6 lists the objects you can add to each of these secondary groups to create a new member of the group. Table 7 lists the operations you can perform on members of the groups.

Table 6. Objects You Can Add to Secondary Groups

Secondary Group	Objects You Can Add
Net Classes	Net **Tip** You can drag and drop an electrical net into a net class, but this adds the individual nets of the electrical net to the net class, not the electrical net itself.
Electrical Nets	Electrical net
Matched Length Net Groups	Net class, net, electrical net
Pin Pair Groups	Pin pair

Table 6. Objects You Can Add to Secondary Groups (continued)

Secondary Group	Objects You Can Add
Conditional rule	Net class, net, pin pair group, pin pair, and layer. For more information, see "“Conditional Rules” on page 175.”
Matched Length Pin Pair Groups	Pin pair group, pin pair
Differential pair	Net, pin pair, electrical net **Tip** A differential pair contains exactly two pin pairs, two nets, or two electrical nets.

Table 7. Operations on Members of Secondary Groups

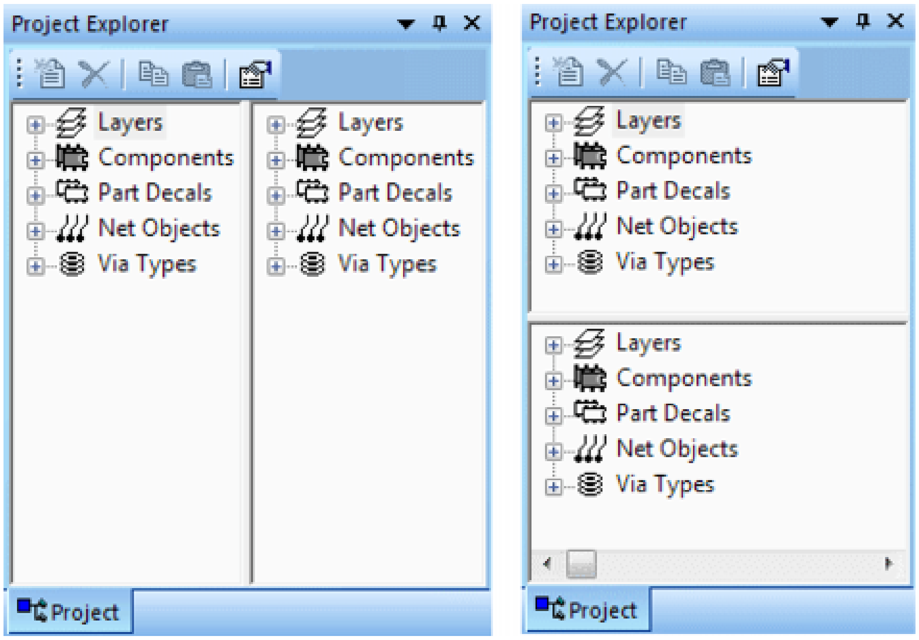
Secondary Group	Available Operations on Members
Net classes	Copy/paste or drag and drop into group to create, new, rename, delete
Electrical Nets	New, delete
Matched Length Net Groups	Copy/paste or drag and drop into group to create, new, rename, delete
Pin Pair Groups	Copy/paste or drag and drop into group to create, new, rename, delete
Conditional Rules	Delete **Tip** You cannot remove individual elements of a conditional rule.
Matched Length Pin Pair Groups	Copy/paste or drag and drop into group to create, new, rename, delete
Differential Pairs	Copy/paste or drag and drop into group to create, delete **Tip** You cannot remove individual elements of a differential pair.

Splitting the Project Explorer View

You can split the view of Project Explorer to create a second instance of the tree. Splitting the view is useful when you repeatedly copy and paste or drag and drop objects into Net Objects subtrees; for example, when you copy nets to create a Net Class.

You can split the view horizontally or vertically. The vertical split displays more of the tree and requires less scrolling, while the horizontal split allows wider views of the tree and viewing full names of design objects without scrolling left and right, and without sacrificing design space.

Figure 5. Project Explorer Split Views



Vertical Split Horizontal Split

Procedure

1. Position the pointer in the tree area of Project Explorer.
2. Right-click and click the **Vertical Split** or the **Horizontal Split** popup menu item.

Sorting Objects in Project Explorer

You can sort objects either alphabetically or by rules. Sorting by rules lists objects with rules at the top of the list.

Procedure

To sort the view by rules, right-click and click the **Sort by Rules** popup menu item. To sort alphabetically again, click the **Sort by Rules** popup menu item again.

Creating a New Object in a Secondary Group

With the exception of the Nets group, you can create new members in all the secondary groups under Net Objects. You can also edit all members of those groups to some degree.

Secondary groups include net classes, pin pair groups, conditional rules, matched length net groups, matched length pin pair groups, electrical nets, and differential pairs.

The following table provides a summary of methods for creating new members of secondary object groups.

To create this...	Perform these steps:
Net class	Either: • Drag and drop nets from primary object groups into the Net Class primary group. This action creates a new net class automatically that you can then name. • Select the Net Class primary group. Right-click and click **New**. Name the net class, and then drag and drop nets into it. **Tip** You can drag and drop an electrical net into a net class, but this adds the individual nets of the electrical net to the net class, not the electrical net itself. For more information, see "Creating Net Classes" on page 172.
Pin pair group	Either: • Drag and drop pin pairs from primary object groups into the Pin Pair Groups primary group. This action creates a new pin pair group automatically that you can then name. • Select the Pin Pair Groups primary group. Right-click and click the **New** popup menu item. Name the new group, and then drag and drop pin pairs into it. For more information, see "Creating Pin Pair Groups" on page 174.
Conditional rule	Drag and drop net objects from primary object groups into the Conditional Rules primary group. This action creates a new conditional rule and name automatically. For more information, see "Creating Conditional Rules" on page 178.
Matched length net group	Either: • Drag and drop nets and/or electrical nets from primary object groups into the Matched Length Net Groups primary group. This action creates a new group automatically that you can then name. • Select the Matched Length Net Groups primary group. Right-click and click the **New** popup menu item. Name the group, and then drag and drop nets into it. For more information, see "Creating Matched Length Net Groups" on page 179.
Matched length pin pair group	Either: • Drag and drop pin pairs from primary object groups into the Matched Length Pin Pair Groups primary group. This action creates a new group automatically that you can then name. • Select the Matched Length Pin Pair Groups primary group. Right-click and click the **New** popup menu item. Name the group, and then drag and drop nets and/or electrical nets into it.

To create this...	Perform these steps:
	For more information, see “Creating Matched Length Pin Pair Groups” on page 180.
Differential pair	Select two nets in the Nets primary group, or two electrical nets in the Electrical Nets primary group, right-click, click the **Copy** popup menu item, select the Differential Pairs primary group, right-click, and then click the **Paste** popup menu item. This action creates a new differential pair automatically. Alternative: Select two nets or two electrical nets in the design workspace, right-click, and then click the **Make Differential Net or Make Differential Electrical Net** popup menu item. For more information, see “Creating Differential Pairs” on page 181.

Removing an Object From a Secondary Group

You can remove an object from Net Classes, Electrical Nets, Pin Pair Groups, Matched Length Net Groups, or Matched Length Pin Pair Groups.

You cannot remove individual objects from Differential Pairs and Conditional Rules.

Procedure

Right-click the object to remove and click the **Delete** popup menu item.

Results

The objects are removed from the secondary group only; they are *not* deleted from the design.

Renaming Secondary Group Elements

You can rename secondary group elements of Net Classes, Matched Length Net Groups, Matched Length Pin Pair Groups, and Pin Pair Groups.

You cannot rename an Electrical Net or Differential Pair Group.

Procedure

1. Right-click the secondary group name, and click the **Rename** popup menu item.
2. Type a new name over the previous name, and press Enter.

Output Window

Use the three tabs of the Output window to display reports and session logs, edit and debug macros, and do custom programming and debugging.

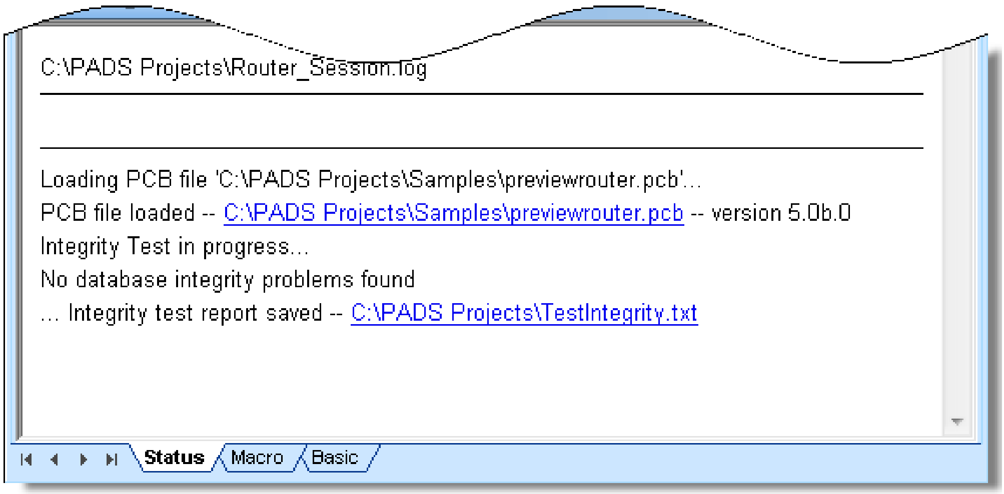
The Output window is located in the lower left section of the interface. You can dock or float the Output window. You can also open or close the Output window.

- Status Tab
- Macro Tab
- Basic Tab

Status Tab

To access: From the Standard toolbar, click the **Output Window** button, and then choose the **Status** tab.

Use this tab to record, display, and print session logs, report files, web pages, and to open program files.



Description

The **Status** tab displays information about the current session. It specifies the filename 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 auto-routing (or performing other tasks), the Output window opens with the **Status** tab active and the error appears in red. The Output window appears in its most recent state (floating or docked).

Objects

Object	Description
Session log	The *session.log* file contains all program output for the current session, including names of open and saved files, integrity test results, routing statistics, and messages. SailWind Router constantly updates the session log during autorouting. The session log presents types of information in different colors. Underlined items are links. • Red — Errors Green — Warnings Black — Messages Blue — Links to files, web pages, and database objects.
Navigation buttons	Navigate through the pages of reports in the session log.
Filter popup menu item	Filter session log messages according to type:

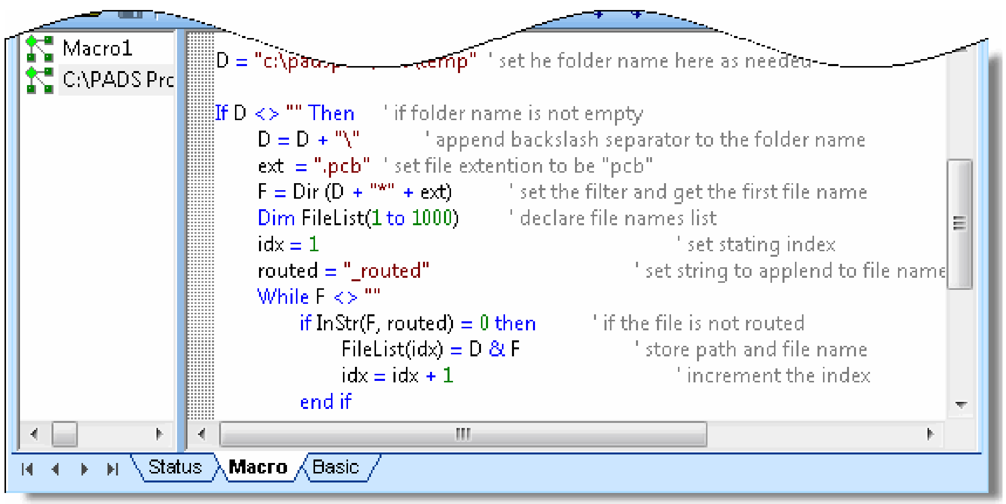
Object	Description
	• Error • Warning • Message • Show all
Find popup menu item	Right-click and click the **Find** popup menu item to search the session log contents.
Print popup menu item	Print the entire session log using the **Print** popup menu item or print an individual report by right-clicking it and clicking the **Print** popup menu item.
View popup menu item	Click any report to view it in detail.
Log to File popup menu item	Saves the session log for future reference. If a session log file already exists, new information is appended. If a session log file does not exist, a new file is created. The default path (*\SailWind Projects*) for the session log file is set when you install the program. In SailWind Router, you can change the default filename and path in the "Options Dialog Box, Global Category, File Locations Subcategory" on page 439. **Note:** You can clear the session log display each time you open a file. Clearing the session log prevents you from accidentally viewing information from a previously opened file; however, it does not delete the log file.

Macro Tab

You can edit, run, and debug macro scripts in the **Macro** tab. You can open multiple macros and nest macros using the macro editor.

A macro is any combination of commands, keystrokes, and mouse clicks that you record to replay as a single action. You can record virtually any set of procedural steps for replay, thereby simplifying redundant activities, such as setting preferences and layer/display settings.

Figure 6. Macro Tab



Creating Macros
Managing Macros
Playing Back a Macro
Stopping a Playing Macro
Debugging Macro Scripts
Accessing Help on the Macro Language

Creating Macros

You can create macros to execute often-performed actions. You can record a series of procedural steps for replay as a single action; you can also nest macros.

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

Creating a New Macro

Set up a new macro for tasks that SailWind Router may perform frequently.

Procedure

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

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

Mouse Movement Recording

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

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

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

Saving the Macro

Save your macro after completing it.

Procedure

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

Managing Macros

Read the following topics to learn more about managing macro files.

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

Opening an Existing Macro File

SailWind Router stores the macros you create in (*.mcr*) files. You can use either the menus or the toolbar to open an existing macro file.

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

Procedure

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

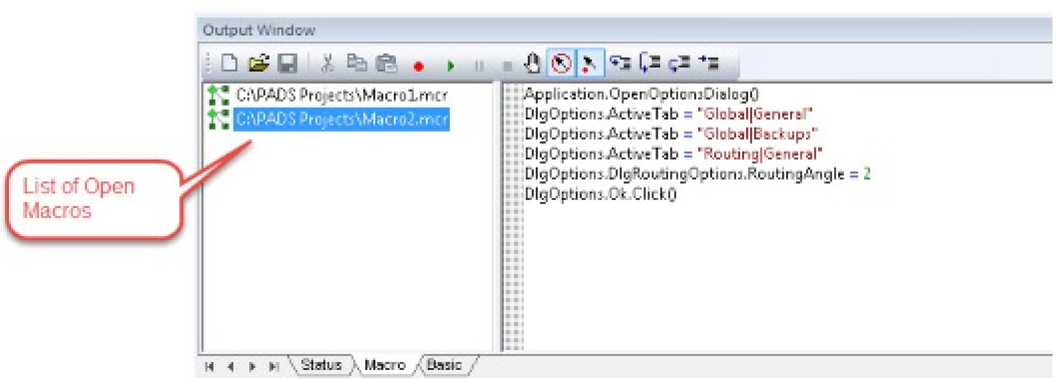
Viewing Multiple Open Macros

You can open multiple macros in the macro editor and switch between them.

Procedure

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

Figure 7. The Macro Tab



Editing a Macro

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

Procedure

1. Select the text you want to copy or cut.
2. Right-click in the Output window and click the **Copy** or **Cut** popup menu items.
3. Right-click in the Output window and click the **Paste** button. You will see that your selection has been pasted in the Output window at the insertion point.

Saving the Macro Edits

When you have completed editing the macro, you must save it as described in the following procedure.

Procedure

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

Playing Back a Macro

You can play back a macro after creating it. (When you play a macro, you cannot use the mouse in the workspace.)

Procedure

1. On the **Macro** tab, click the **Open** button and open a macro (.mcr) file. Recent macros can be found by clicking the **Tools > Macros** menu item. Alternatively, you can right-click in the **Macro** tab and click **Run**.
2. On the **Macro** tab toolbar, click the **Run** button.

Tip
Use the **Pause** and **Play** buttons in the **Macro** tab toolbar to pause and play the running macro.

Stopping a Playing Macro

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

Procedure

Right-click and click the **Stop** popup menu item.

Debugging Macro Scripts

You can play back a macro by either running it step-by-step, or by allowing it to run to a certain point in the script. To perform these debugging tasks, insert breakpoints in the macro script at places where you want the macro to stop.

Setting and Removing Breakpoints

Set and remove breakpoints before debugging a macro. If the macro engine encounters a breakpoint during macro playback, it pauses the macro.

Procedure

1. Place the cursor on the line where you want to add a breakpoint.
2. Right-click in the **Macro** tab and click the **Toggle Break** popup menu item. This inserts a breakpoint at the current cursor location. A breakpoint marker appears in the gutter area. Alternatively, on the **Macro** tab toolbar, click the **Toggle Breakpoint** button.

Results

When the macro engine encounters a breakpoint while playing back a macro, it pauses the macro. It also marks the next line in the macro with the instruction pointer.

Debugging the Macro Scripts

You can debug your macros after inserting breakpoints.

Use the commands in the following table for debugging:

Table 8. Macro Script Debugging Commands

To Do This	Perform These Steps
Play a single line of the macro.	• Right-click in the **Macro** tab and click the **Step Over** popup menu item or • On the **Macro** tab toolbar, click the **Step over** button
Perform a subroutine call on the current line.	• Right-click in the **Macro** tab and click the **Step Into** popup menu item or • On the **Macro** tab toolbar, click the **Step into** button

Table 8. Macro Script Debugging Commands (continued)

To Do This	Perform These Steps
Return from a subroutine to the point from which it was called.	• Right-click in the **Macro** tab and click the **Step Out** popup menu item or • On the **Macro** tab toolbar, click the **Step out** button
Play back a macro to a given point.	• Right-click in the **Macro** tab and click the **Step to Cursor** popup menu item or • On the **Macro** tab toolbar, click the **Step to cursor** button
Continue the execution from the current point.	• Right click in the **Macro** tab and click the **Run** popup menu item or • On the **Macro** tab toolbar, click the **Run** button

Correcting Run-Time Errors

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

Accessing Help on the Macro Language

You can access Help on the macro language calls while scripting or running macros. Help topics provide term information and sample scripts.

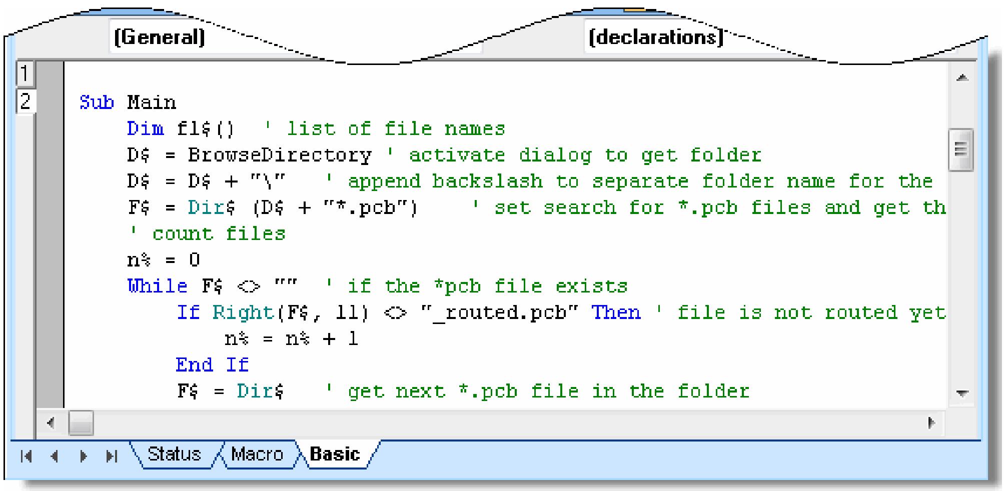
Procedure

Click in the edit area of the **Macro** tab and press the F1 key.

Basic Tab

Use the Basic Script Editor in the **Basic** tab to create, edit, run, and troubleshoot Basic scripts from SailWind applications.

Figure 8. Basic Tab



Managing Scripts
Creating Scripts
Running Scripts
Debugging Scripts
Accessing Help on the Basic Language

Managing Scripts

You can open, edit, print, and save scripts directly from within SailWind Router.

Opening an Existing Script

Use the procedure that follows to open a script in SailWind Router. Scripts that SailWind Router can open have a *.bas* extension. The default location for *.bas* files is *C:\SailWind Projects*.

Tip
You can have up to nine scripts open at the same time.

Procedure

1. In the Basic Script Editor, click the **Open** button.
2. In the Open Script window, select the script and then click **Open**.

Managing Open Scripts

The commands on the Sheet submenu provide script management commands. As you can have up to nine scripts open at the same time, you can open #uses, close sheets, close multiple sheets, and choose scripts to view and edit.

After you have inserted breakpoints in the script, you can debug it using the commands in the following table.

Table 9. Sheet Submenu Commands

To Do This	Perform These Steps in the Basic Script Editor
Open a #uses module. (#Uses modules are Basic scripts that are called from within other scripts.)	Right-click and click **Sheet > Open Uses**. The #uses modules called in the script appear as script sheets in the Basic Script Editor. They are assigned a numbered tab and you can edit or run them.
Close an open script.	• Right-click and click the **Sheet > Close** menu item or • Double-click the script's numbered tab in the gutter

Table 9. Sheet Submenu Commands (continued)

To Do This	Perform These Steps in the Basic Script Editor
Close all open scripts.	Right-click and click the **Sheet > Close All** popup menu item.
Select an open script to view. **Tip** You can have up to nine scripts open at the same time	• Right-click and click the **Sheet** popup menu item; then click the script you want to view from the list of open scripts on the submenu or • Click the script's numbered tab in the gutter.

Editing a Script

You can copy or cut selected text from the Basic Script Editor to the Clipboard and paste a selection from the Clipboard into the text window. You can also paste text from the Clipboard into other applications.

Procedure

1. In the Basic Script Editor, select the text you want to copy or cut.
2. Right-click and click the **Copy** or the **Cut** popup menu item.
3. Right-click and click the **Paste** popup menu item to paste the script text. Your selection is pasted in the Output window at the insertion point.

   Alternatively, you can click the Copy, Cut, and Paste buttons on the Basic Script Editor toolbar.

Editing a User Dialog Box

You can edit a user dialog graphically. A User Dialog is defined by a “Begin Dialog...End Dialog” block in a Basic script.

Procedure

1. In the Basic Script Editor, put your cursor in a User Dialog block of the script.
2. Click the **Edit User Dialog** button. (This button is at the far right of the toolbar; you may need to expand the Output Window to see it.)

   For more information, see the *Sax Basic Editor Online Help* (*C:\<install_folder>\<version>\Programs\sbe5_000.hlp*)

Finding an Automation Statement

If you are working with a long script, you can search for particular statements.

Procedure

1. In the Basic Script Editor, click the Object list and select an object type. The Object list shows all the objects for the current module. The (General) object groups all of the procedures that are not part of any specific object.
2. Click the Procedure list and select a bold procedure. The Procedure list shows all the procedures for the current object. Selecting a procedure that is bold locates the procedure in the script.

Results

The statement appears in the Basic Script Editor.

Printing a Script

You can print a Basic script from within the Basic Script Editor.

Procedure

1. Open the script in the Basic Script Editor.
2. On the toolbar, click the **Print** button.

Saving a Script

Keep your Basic scripts for future use by saving them.

Procedure

1. In the Basic Script Editor, click the **Save** button.
2. Type a filename and then click **Save**.

Watching a Variable

Use Quick Watch to show the value of the expression under the cursor in the immediate window.

Procedure

1. Right-click and click the **Quick Watch** popup menu item, or click the **Quick Watch** button.
2. See also: *Sax Basic Editor On Line Help* (*C:\<install_folder>\<version>\Programs\sbe5_000.hlp*).

Creating Scripts

You can create scripts to execute often-performed actions in SailWind Router.

Creating a Script

You can write scripts for SailWind Router using the Basic Script Editor.

Procedure

1. Click the **Tools > Basic Scripts > Basic Script Editor** menu item.

   The **Basic** tab opens in the Output window.

   Note:
   In SailWind Layout and SailWind Logic, the SAX Basic Engine dialog box appears.

2. Click the **New** button.

Inserting an Automation Statement Using the Object and Procedure Lists

Use the Object and Procedure lists to select and insert a statement. These lists contain the most commonly used statements.

Procedure

1. Click the Object list and click an object type. The Object list shows all the objects for the current module. The (General) object groups all of the procedures that are not part of any specific object.

2. Click the Procedure list and click a non-bold procedure to insert. The Procedure list shows all the procedures for the current object. Selecting a procedure that is not bold inserts the proper procedure definition for that procedure.

Results

The statement appears at the bottom of the script.

Inserting an Automation Statement Using the ActiveX Automation Members Dialog

Use the Object and Procedure lists to select and insert a statement.

CAUTION:
If the pointer is on any line in the script other than the bottom line, the line is overwritten.

Procedure

1. In the Basic Script Editor, right-click and click the **Browse** popup menu item.
2. Use the ActiveX Automation Members dialog box to select and insert a statement. This dialog box contains an extensive list of statements.

Setting the Next Statement

You can force a particular line in a script to run next. (You can only select statements in the current subroutine or function.)

Procedure

1. In the Basic Script Editor, put your cursor on the line you want to run next.
2. Right-click and click the **Set Next Statement** popup menu item.

Results

An instruction pointer appears next to the selected line. This line, and only this line, will run next. If you go to other parts of the script, you can return to this line by clicking **Show Next Statement**.

Showing the Next Statement

You can use the “Show Next Statement” feature in the Basic Script Editor to locate the next point in the script to run. This feature is especially useful during debugging operations.

Procedure

In the Basic Script Editor, right-click and click the **Show Next Statement** popup menu item.

Results

An instruction pointer indicates the next statement to run. Pausing a running script or setting a statement to run next sets the next statement. You can locate the set statement from anywhere in the script.

Saving the Script

Store your scripts for future use by saving them from within the Basic Script Editor.

Procedure

1. In the Basic Script Editor, click the **Save** button.
2. Type a file name and then click **Save**.

Running Scripts

You can run an existing script using Run or resume the playback of a paused script. (When you run a script, you cannot use the mouse in the workspace.)

Running a Script

Use the Basic Script Editor to open and run scripts.

Procedure

1. In the Basic Script Editor, open a script file.
2. Right-click and click the **Run** popup menu item. Alternatively, on the Basic Script Editor toolbar, click the **Run** button.

Pausing and Resuming a Running Script

When running a long script, you can pause it to perform some other design activity, and then resume it.

Procedure

1. In the Basic Script Editor, right-click and click the **Pause** popup menu item. (Or click the **Pause** button in the toolbar.)
2. When you are ready to resume running the script, right-click in the Basic Script Editor and click the **Run**, **Step Over**, or **Step to Cursor** button.

Stopping a Running Script

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

Procedure

In the Basic Script Editor, right-click and click **Stop.** (Or click the **Stop** button in the toolbar.)

Debugging Scripts

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

Setting or Removing the Breakpoints

The ability to set or remove breakpoints is useful when you debug a script. If the Basic engine encounters a breakpoint when running a script, it pauses the execution of the script. An instruction pointer marks the next line in the script.

Procedure

1. Place the cursor on the line where you want to add a breakpoint.
2. On the Basic Script Editor toolbar, click the **Toggle Breakpoint** button. Alternatively, in the Basic Editor, right-click and click the **Toggle Break** popup menu item.

Results

A breakpoint is inserted at the current cursor location. A breakpoint marker appears in the gutter area.

Debugging the Scripts—Commands

You can begin debugging a script after you have inserted breakpoints.

Debug your script using the following commands:

Table 10. Basic Script Debugging Commands

To Do This	Perform These Steps in the Basic Script Editor
Run a single line of the script	Click the **Step over** button on the toolbar.
Perform a subroutine call on the current line.	Click the **Step into** button on the toolbar. (Or right-click in the edit area and click the **Step into** popup menu item.)
Return from the subroutine to the point from which it was called.	Click the **Step out** button on the toolbar.
Run a script to a given point.	In the edit area, set the cursor on the line at which you want to pause, then right-click and click the **Step to cursor** popup menu item
To continue the execution from the current point.	Click the **Run** button on the toolbar. (Or right-click and click the **Run** popup menu item.)
Remove all breakpoints.	In the edit area, right-click and click the **Clear All Breaks** popup menu item.

Correcting Run-time Errors

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

Accessing Help on the Basic Language

While writing or running scripts, you can access Help that provides information and a sample script using the Basic language statements.

Procedure

Select or click in an item in color in the edit area of the Basic Script Editor and then press the F1 key.

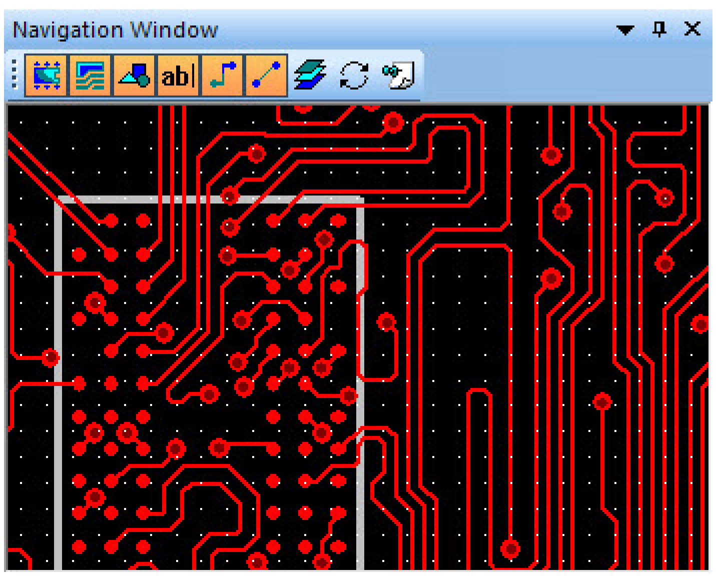
Results

Help appears for the current statement.

Navigation Window

Use the Navigation window to display a different view of the workspace, in which you can filter the items displayed and see information about objects selected in the workspace.

Figure 9. Navigation Window



The content of the Navigation window depends on what is selected in the workspace, as described in the following topics.

Nothing Selected

The view in the Navigation window is magnified, and centers on the location of the pointer in the workspace; when the pointer moves, the view in the Navigator window follows it.

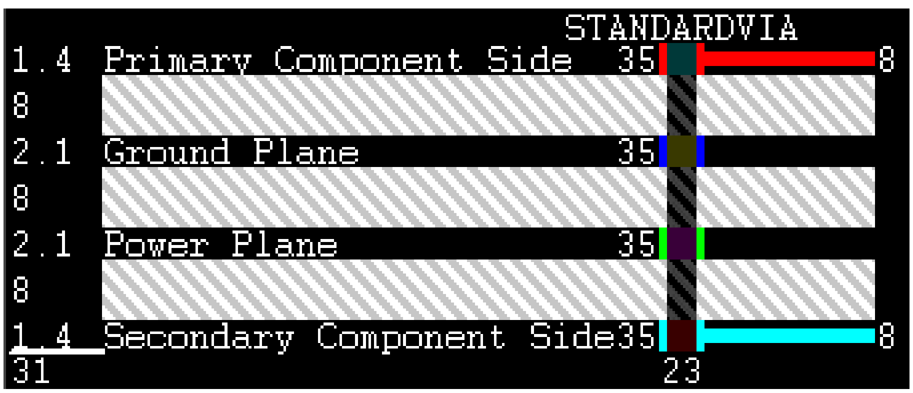
Pin or Via Selected

When you select a pin or via, the Navigation window displays a cross section of the object.(Figure 10). Information that appears includes:

- Pad sizes
- Thermal attachments
- Test point status
- Width
- Layers of attached traces
- Drill size
- Component or via name

The size of the substrates is proportional to their thickness, which is set in the SailWind Layout Setup Layers dialog box. The pad size reflects non-plane nets on any layers. For plane nets on CAM or split/ mixed layers, the antipad size or thermal pad (showing the inner diameter for a thermal pad) appears.

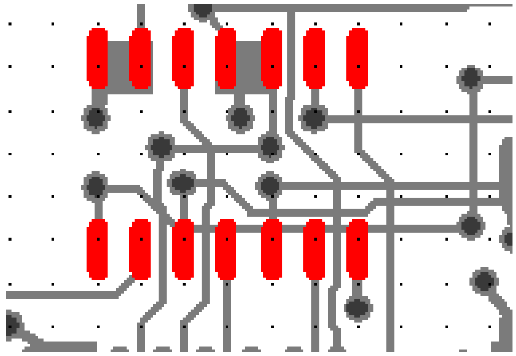
Figure 10. Pin/Via Cross-Section



Component Selected

When a component is selected, the navigation window zooms in on the selected component, (Figure 11). The navigation window shows the attached traces and fanouts as well as the component.

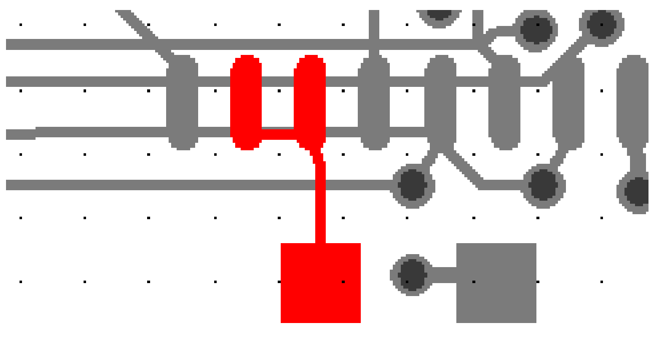
Figure 11. A Selected Component in the Navigation Window



Net Selected

If a net object is selected, the navigation window zooms in on the selected net class, net, pin pair group, pin pair, electrical net, differential pair, or matched length group (Figure 12).

Figure 12. A Selected Net in the Navigation Window

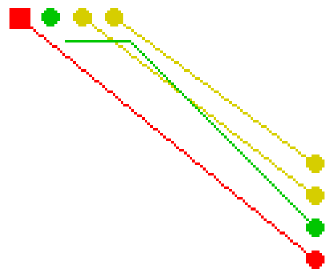


When Routing Length-Controlled Objects, or Selecting and Placing Components

When interactively routing length-controlled objects (for example, matched length groups and differential pairs, nets, electrical nets, or pin pairs with length rules), the navigation window monitors the route length. It shows only those nets, electrical nets, or matched length groups that have length rules and are affected by the current operation.

The navigation window shows the same information if you select or place a component that has length-controlled objects attached to it. Length-controlled objects appear in different colors (Figure 13).

Figure 13. Length-Controlled Objects in the Navigation Window



For more information, refer to Monitoring Trace Length.

Navigation Window Commands

Use the Navigation window commands shown in the following table to control the display in the Navigation window.

Tip
You can also select these commands by right-clicking in the Navigation Window.

Table 11. Navigation Window Commands

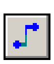
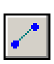
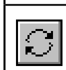
Button	Command
	Display components.
	Display coppers and pours.
	Display 2D lines.
	Display text.
	Display traces and vias.
	Display unroutes.
	Display all layers. When checked, all electrical layers appear. When not checked, the layers set in the shortcut menu appear.
 or F7	Swap views between the Navigation window and the workspace. **Tip** You can also press the F7 key to do this.
 or F8	Lock the view displayed in the Navigation window. The view will not follow the pointer in the workspace.

Table 11. Navigation Window Commands (continued)

Button	Command
	Tip You can also press the F8 key to do this.
F5	Zoom in.
F6	Zoom out.

Related Topics

Pan, Zoom, and Scroll the Display

Spreadsheet Window

Use the Spreadsheet Window to view and modify properties for all design objects. You can customize how information appears. You can also export the information to an HTML file, letting you share it over the Web with other members of the design team.

By default, the Spreadsheet window has the following tabs:

- **Selected Object** — Displays and changes information about design objects
- **Errors** — View and organize errors and find the errors in the design. Use the **Error** tab to view the type of error, the layer the error is on, the coordinates of the error, and the error description. These fields are for information only and cannot be modified. Also use the **Error** tab to ignore an error.
- **Net Length Monitor** — Dynamically displays trace length values for net objects during interactive routing and route editing. Includes half the Discrete length value of each connected pin of components that have a Discrete length assigned.
- **Electrical Net Length Monitor** — Dynamically displays trace length values for electrical net objects during interactive routing and route editing. Includes half the Discrete length value of each connected pin of components that have a Discrete length assigned.

Displaying the Spreadsheet Window

You can show or hide the Spreadsheet window.

Procedure

Click the **Spreadsheet Window** button on the Standard toolbar to turn display of the Spreadsheet window on or off (or click the **View > Spreadsheet Window** menu item).

Spreadsheet Window Tab Content

You can use the “Display data for object type” drop-down list to specify what content the selected **Spreadsheet** tab displays.

You can specify to display any *one* of the following:

- Objects currently selected in Project Explorer
- Objects currently selected in the Workspace
- All objects of *one* of the following object types:
    - Components
    - Decals
    - Layers
    - Net classes
    - Nets
    - Electrical nets
    - Matched length pin pair groups
    - Matched length net groups
    - Pin pair groups
    - Pin pairs
    - Differential pairs
    - Keepouts
    - Via types
    - Coppers
    - Errors
    - Jumpers

Changing Design Data

You can change the values of some design properties, such as component location, trace width, and layer, from within the Spreadsheet window.

Note:
You can choose to allow SailWind Router to save your changes to the database automatically by clicking the **Auto update data** button. Otherwise, you must click the **Update data and apply changes** button each time you want to save your changes.

If you copy one cell, you can then paste the contents of that cell into a group of multiple cells. The group does not have to be contiguous.

Restrictions and Limitations

When pasting a group of cells, you must paste into an area that is smaller than the selection.

Procedure

1. In the spreadsheet, double-click on the cell whose value you want to change.
2. Type the new value in the cell then press the Enter key.
3. The change is saved to the database automatically if you have clicked the **Auto update data** button; otherwise, click the **Update data and apply changes** button to save the change.

Related Topics

Copying and Pasting To and From the Spreadsheet

Copying and Pasting To and From the Spreadsheet

You can copy selected cells in a **Spreadsheet** tab to the Windows Clipboard in HTML table format. Then you can paste the information into Windows applications, such as Microsoft Word and Excel, following standard Windows procedures. You can also paste into another part of the tab or into a different tab. For example, you can copy a value (from an individual cell) to the clipboard, and then select a group of cells to which you want to paste this value.

Copying Spreadsheet Cells

You can copy cells and paste them elsewhere in the spreadsheet.

Procedure

1. Select the cells that you want to copy.

Tip
When pasting a group of cells, you must paste into an area that is smaller than the selection.

2. Click the **Copy** button.

Examples

You may want to copy a single value, for example a component's Y location, into multiple cells. This allows you to easily change the Y location of several components at one time.

Pasting Cells in the Spreadsheet

You can paste copied spreadsheet cells into other cells in the spreadsheet.

Restrictions and Limitations

You cannot delete a predefined scheme.

Procedure

1. Copy the cells you want to paste.
2. Click the tab into which you want to paste the cells.
3. Select the cells into which to paste the data.

Tip
You can paste the contents of a single cell into multiple cells. The cells do not have to be contiguous.

Note:
When pasting a group of cells, you must paste into an area that is smaller than the selection.

4. Click the **Paste** button.

Related Topics

Changing Design Data

Pasting Cells into Another Windows Application

You can copy information from cells in a spreadsheet and paste it into other applications such as a text editor.

Procedure

Open the application into which you want to paste the data, and paste as appropriate for that application.

Synchronizing a Tab with Project Explorer

Tabs synchronized with Project Explorer display the objects currently selected in Project Explorer.

Synchronization operates as follows:

- If you select a specific object type in the Project Explorer (such as Pin Pair), then all objects of that type are displayed in the Spreadsheet window.
- If you select certain objects such as specific nets, then only those objects are displayed.
- If you select objects of multiple types at the same time, no objects are displayed.

Procedure

In the Display data for object type dropdown list, select “Sync with Project Explorer”. (You may need to close the Output window and/or the Navigation window to see the dropdown list.)

Tip
To pause synchronization, select “Keep current objects” from the Display data for object type list. To resume synchronization, click “Sync with Project Explorer” again.

Synchronizing a Tab with Workspace Selection

When you synchronize the selection tab of the Spreadsheet window with the workspace, the selection tab displays the objects selected in the workspace.

Restrictions and Limitations

If you select objects of multiple types at the same time, no objects are displayed.

Procedure

In the Display data for object type dropdown list, select “Sync with selection”. (You may need to close the Output window and/or the Navigation window to see the dropdown list.)

Tip
To pause synchronization, select “Keep current objects” from the Display data for object type dropdown list. To resume synchronization, click “Sync with Project Explorer” again.

Related Topics

Organizing Spreadsheet Tabs

Organizing Spreadsheet Tabs

Use the Column Organizer dialog box to select which columns appear in the selected tab and to change the display order of columns. You can also define a non-scrolling area.

Procedure

1. In the Spreadsheet window, click the **Organize columns** button to open the Column Organizer dialog box. You may need to close the Output window and/or the Navigation window to see the button.

2. To add columns to the tab, in the Available columns box, select the columns you want to include, and click the right arrow button (**>>**) to move them to the Displayed columns box.

3. To remove columns from the tab, in the Displayed columns box, select the columns you want to remove, and click the left arrow button (**<<**) to move them to the Available Columns box.

4. To change the order of columns in the spreadsheet, in the Displayed columns box, select the column you want to move and click the down arrow button to move the column to the right in the spreadsheet or the up arrow button to move the column to the left.

5. To define a non-scrolling area, select “Lock position of columns when scrolling” check box. Enter the number of columns you want to define in the “Number of columns locked (from left)” box.

Tip
To remove the non scroll property, clear the “Lock position of columns when scrolling” check box.

Spreadsheet Window Customization

You can change the appearance of the Spreadsheet window by adding, renaming, and removing tabs. You can also change the width of columns and align text.

Adding a Tab to the Spreadsheet Window

You can display additional content in a spreadsheet by adding tabs.

Procedure

1. Right-click any tab on the Spreadsheet and click the **Add Tab** popup menu item. A new tab appears to the right of existing tabs.
2. Double-click the title of the added tab and type the new name.

Renaming a Tab in the Spreadsheet Window

You can change the name of any tab in the spreadsheet.

Procedure

1. Right-click the tab you want to rename and click the **Rename Tab** popup menu item.
2. Type the new name.

Removing a Tab from the Spreadsheet Window

You can delete any tab from the spreadsheet.

Procedure

Right-click the tab you want to remove and click the **Delete Tab** popup menu item.

Changing Column Width

You can make the column width fit the text.

Procedure

1. Select the columns you want to size.
2. Click the **AutoFit to Contents** button. You may need to close the Output window and/or the Navigation window to see the button.

> **Tip**
> You can also double-click the right-hand separator of a column to auto-size it.

Aligning Text

You can align text to the left, center, or right-hand side of columns.

Procedure

1. Select the columns you want to align.
2. Click the proper alignment button for the type of alignment you want (“Align left,” “Center,” or “Align right”). You may need to close the Output window and/or the Navigation window to see the alignment buttons.

Sorting the Spreadsheet

Sort the spreadsheet to find the most important information quickly.

Procedure

1. Click the **Define sort order** button. You may need to close the Output window and/or the Navigation window to see the button.
2. Double-click each column header in the sort order you want to create, starting with the most important column.

   A number is placed in the column header to show the priority of that column in the sort order, and the spreadsheet is re-sorted to include that column.

> **Tip**
> Double-clicking the column again reverses the sort order for the column.

Printing a Spreadsheet Tab

If desired, you can print the contents of any tab in a spreadsheet.

Procedure

1. Click the tab you want to print.
2. Click the **Print** button to print the tab using the current printer setup.

Finding Items in the Spreadsheet Window

You can find items in Selection tabs by specifying search criteria in the Find dialog box. You can search on one condition, or create a more complex search containing multiple conditions.

Creating a Simple Search

You can create a simple search expression by defining a single condition.

Procedure

1. In the Spreadsheet window, click the **Find object** button. This opens the Find dialog box. (You may need to close the Output window and/or the Navigation window to see the **Find object** button.)
2. If the **Less** button appears in the Find dialog box, click it to collapse the lower portion of the dialog box. Only the simple search items remain.
3. Using the Property, Condition, and Value controls, specify the property you want to search on and the condition and value to test the property against.

   Example: If you want to find nets whose Routed Length property is less than 1.1, select Routed Length from the Property list, “Is less than” from the Condition list, and enter “1.1” in the Value box.
4. If you want to start the search from the bottom of the Spreadsheet, select the Backward Search check box.
5. Click **Find** to find the first object and close the dialog box, or click **Find Next** to search successively for multiple objects.

Related Topics

Find Dialog Box

Creating a Multiple-Condition Search

You can create a Boolean search expression that has multiple conditions.

Procedure

1. In the Spreadsheet window, click the **Find object** button. This opens the Find dialog box. (You may need to close the Output window and/or the Navigation window to see the button.)
2. In the Find dialog box, if the **More** button is available, click it to expand the dialog box.
3. Click the **New** button to add the first filter condition to the filter expression.
4. Using the Property, Condition, and Value controls, specify the property you want to search on and the condition and value to test the property against.

   Example: If you want to find nets whose Routed Length property is less than 1.1, select Routed Length from the Property list, “Is less than” from the Condition list, and enter “1.1” in the Value box.

5. Add another condition to the search expression, as follows
    a. Click the **New** button.
    b. Select a Boolean operator.
    c. Using the Property, Condition, and Value controls, specify the property you want to search on and the condition and value to test the property against.
6. If you have another condition to add, go to Step 5; otherwise continue with Step7.
7. If you want to start the search from the bottom of the Spreadsheet, select the Backward Search check box.
8. Click **Find** to find the first object and close the dialog box, or **Find Next** to search successively for multiple objects.

Related Topics

Find Dialog Box

Filtering the Spreadsheet Window

You can filter information that appears in Selection tabs by creating a filtering expression in the Edit Filter dialog box then applying the filter by clicking the **View filtered data** button. You can create a simple filter containing one condition, or create a more complex filter containing multiple conditions.

Tip
When you create a filter, it becomes part of the currently active scheme.

For more information, see "Edit Filter Dialog Box."

Creating a Simple Filter

Create a simple filtering expression with a single condition.

Procedure

1. In the Spreadsheet window, click the **Define filter settings** button. This opens the "Edit filter" dialog box. (You may need to close the Output window and/or the Navigation window to see the button.)
2. In the Edit filter dialog box, if the **Less** button is available, click it to collapse the lower portion of the dialog box.
3. Using the Property, Condition, and Value controls, specify the property you want to filter by and the condition and value to test the property against.

   Example: If you want to display nets whose Routed Length property is less than 1.1, select Routed Length from the Property list, "Is less than" from the Condition list, and enter "1.1" in the Value box.
4. Click **OK** to save the filter.

Results

The filter is saved as part of the currently active scheme. To apply the new filter, click the View filtered data button.

Creating a Multiple-Condition Filter

Create a Boolean filter expression with multiple conditions.

Procedure

1. In the Spreadsheet window, click the **Define filter settings** button. This opens the "Edit filter" dialog box. (You may need to close the Output window and/or the Navigation window to see the button.)
2. In the Edit filter dialog box, if the **More** button is available, click it to expand the dialog box.

3. Click the **New** button to add the first search condition to the search expression.
4. Using the Property, Condition, and Value controls, specify the property you want to filter by and the condition and value to test the property against.

   Example: If you want to display nets whose Routed Length property is less than 1.1, select Routed Length from the Property list, “Is less than” from the Condition list, and enter “1.1” in the Value box.
5. Add another condition to the filter expression, as follows
   a. Click the **New** button.
   b. Select a Boolean operator.
   c. Using the Property, Condition, and Value controls, specify the property you want to filter by and the condition and value to test the property against.
6. If you have another condition to add, return to Step 5; otherwise continue with Step7.
7. Click **OK** to save the filter.

Results

The filter is saved as part of the currently active scheme. To apply the new filter, click the **View filtered data** button.

Related Topics

Using Spreadsheet Schemes (Formats)

Find Dialog Box

Exporting a Spreadsheet

You can save data in HTML format to view with a Web browser or import into another Microsoft application such as Excel.

Procedure

1. Select the rows or columns you want to save.

   Tip
   If you do not select part of the spreadsheet, the entire spreadsheet is saved.
2. Click the **Export to HTML file** button.
3. Specify the filename and location to which you want to save the file.
4. Click **Save**.

Using Spreadsheet Schemes (Formats)

A *scheme* saves the way you customize a tab in the Spreadsheet window. You can associate any number of schemes with each object type.

The program contains predefined schemes for each object type. You cannot delete a predefined scheme.

Creating a New Scheme

Create a new spreadsheet scheme to customize the appearance of your spreadsheet workspace.

Procedure

1. From the "Display data for object type" dropdown list, select the object type for which you want to create the scheme. You may need to expand the Spreadsheet window to see the list.
2. Use the Customize dialog box to customize your workspace; for example, your workspace, toolbars, menus, and shortcut keys.

   For more information, see: "Customizing the Appearance of the Screen" on page 565
3. Click the **Create new scheme** button. SailWind Router creates the new scheme and saves it with the default name Scheme *N* (where N is a number). If you do not want to use the default name, type a name for the new scheme in the Scheme dropdown list.

Deleting a Scheme

You can remove a scheme provided it is not one of the predefined schemes.

Restrictions and Limitations

You cannot delete a predefined scheme.

Procedure

1. From the Scheme dropdown list, select the scheme you want to delete.
2. Click the **Delete the current scheme** button of the toolbar.
3. When prompted, click **Yes**.

Results

The selected scheme is deleted, and the current scheme becomes the predefined scheme for the object type.

Changing to a Different Scheme

You can select another spreadsheet scheme from within the Spreadsheet window.

Procedure

1. In the Open Scheme list, select the scheme that you want to change.
2. To return to a previous scheme, click the **Show Previous Scheme** button.
3. To advance to the next scheme on the list, click the **Show Next Scheme** button.

Chapter 6
Modeless Commands and Shortcut Keys

As an aid to increase your design session productivity, SailWind Router offers a broad array of modeless commands and shortcut keys to allow for quicker execution of common design tasks.

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 clicking Enter. 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 by pressing the g key, typing the new setting, and pressing the Enter key.

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

The following table lists the conventions used in modeless commands and shortcut keys.

Table 12. Modeless Command and Shortcut Key Table Conventions

Convention	Indicates
< >	A variable, or something that you can type
{ }	An optional command argument
click	Click the left mouse button
middle-click	Click the middle mouse button or wheel
right-click	Click the right mouse button
wheel back	Rotate the wheel backward, where the top of the wheel rotates away from your palm
wheel forward	Rotate the wheel forward, where the top of the wheel rotates toward 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 and select W1, while S SW1 means to search for SW1.

The following tables list all of the modeless commands:

Table 13. Modeless Commands for Setting the Design Units

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

Table 14. Modeless Commands for Grid Settings

Name	Command	Description
Global Grid Setting (Except Display)	G<*x*> {<*y*>}	Sets all grids on page 491 (Routing, Test Point, Via, Fanout, Component), except the Displayed grid. 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.
Component Grid Setting	GC <*x*> {<*y*>}	Sets the Component grid on page 491. The second parameter is optional. If you type one parameter, it applies the same value to both <x> and <y>. For example, GC 8-1/3, GC 25 25, or GC 100.
Displayed (Dot) Grid Setting	GD <*x*> {<*y*>}	Sets the Displayed (Dot) grid on page 491. 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.
Fanout Grid Setting	GF <*x*> {<*y*>}	Sets the Fanout grid on page 491. The second parameter is optional. If you type one parameter, it applies the same value to both <x> and <y>. For example, GF 8-1/3, GF 25 25, or GF 100.

Table 14. Modeless Commands for Grid Settings (continued)

Name	Command	Description
Routing Grid Setting	GR <*x*> {<*y*>}	Sets the Routing grid on page 491. 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, or GR 100.
Grid Snap On/Off (Routing Grid Only)	GS	Toggles snap to the routing grid on page 491 on or off.
Test Point Grid Setting	GT <*x*> {<*y*>}	Sets the Test point grid on page 491. The second parameter is optional. If you type one parameter, it uses the same value for both <x> and <y>. For example, GT 8-1/3, GT 25 25, or GT 100.
Via Grid Setting	GV <*x*> {<*y*>}	Sets the Via grid on page 491. 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.
Grid Statistics (Hard)	GGH	Create the Hard Grid Statistics report. The report is displayed in the Output window. **Requirement:** You must select the "Snap Objects to Grid" check box for the Routing grid in order to see Off grid statistics in the report. Use the GS modeless command to quickly toggle this setting. For more information, see Grid Statistics Report in ""Grid Rules" on page 165."
Grid Statistics (Medium)	GGM	Create the Medium Grid Statistics report. The report is displayed in the Output window. **Requirement:** You must select the "Snap Objects to Grid" check box for the Routing grid in order to see Off grid statistics in the report. Use the GS modeless command to quickly toggle this setting. For more information, see Grid Statistics Report in ""Grid Rules" on page 165"
Grid Statistics (Soft)	GGS	Create the Soft Grid Statistics report. The report is displayed in the Output window. **Requirement:** You must select the "Snap Objects to Grid" check box for the Routing grid in order to see Off grid statistics in the report. Use the GS modeless command to quickly toggle this setting. For more information, see Grid Statistics Report in ""Grid Rules" on page 165"

Table 15. Modeless Commands for Line/Trace Angle Settings

Name	Command	Description
Any Angle	AA	Any angle mode. Sets the routing angle in the Options dialog box > **Routing** category > **General** subcategory to “Any angle” (no angle restrictions).
Diagonal Angle	AD	Diagonal angle mode. Sets the routing angle in the Options dialog box > **Routing** category > **General** subcategory to “Diagonal” (45 degree angles only).
Orthogonal Angle	AO	Orthogonal angle mode. Sets the routing angle in the Options dialog box > **Routing** category > **General** subcategory to “Orthogonal” (90 degree angles only).

Table 16. Modeless Commands for Line/Trace Width Settings

Name	Command	Description
Reduce Display Width	R <n>	Sets the minimum display line width. Traces that have a width less than this value are not displayed at their true width, but are displayed at a single pixel width. This feature simplifies the selection of tiny trace 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 17. Modeless Commands for DRC Settings

Name	Command	Description
Toggle DRC Mode On/Off	DRC	Toggles on-line DRC mode. Sets the **On-line DRC** button on the DRC Toolbar on/off.
Suspend	SUS	Suspend. You can suspend (pause) the move or spin of a component to free the cursor to examine DRC issues on the **Errors** tab of the Spreadsheet window. Press Esc to return to the move or spin mode.

Table 18. 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 (supports regular expressions ?*[])	S <*s*>	Search string. Moves pointer to the named object, for example S U3.10. You can substitute alphanumeric characters in the object name with the ? * and [] regular expression pattern

Table 18. Modeless Commands for Searching (continued)

Name	Command	Description
		operators. For examples, see "Expressions in Shortcut Keys" on page 560.
Search Relative	SR <*x*> <*y*>	Search relative. Moves the pointer by the specified X and Y offset, for example SR -100 -50.
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. Moves the pointer to the named object, then selects it, for example SS U10. You can also search and select more than one component by typing multiple entries, for example SS U10 U5 R6. You can substitute alphanumeric characters in the object name with the ? * and [] regular expression pattern operators. For examples, see "Expressions in Shortcut Keys" on page 560. **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.
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.

Table 19. Modeless Commands for Object Visibility

Name	Command	Description
Distinguish Protected Objects	DP	Distinguishes protected objects by displaying them in outline mode. Command toggles this mode on and off.
Distinguish Protected Objects On/Off	I	Distinguishes protected objects by displaying them in outline mode. Command toggles this mode on and off.

Table 19. Modeless Commands for Object Visibility (continued)

Name	Command	Description
		Tip The shortcut dialog does not appear for this command, but the status of the change appears in the Status bar.
Outline Mode	O	Toggles between showing only object outlines or showing filled objects. **Tip** The shortcut dialog does not appear for this command, but the status of the change appears in the Status bar.
Pin Number Display On/Off	PN	Toggles pin number display on/off. **Tip** This command toggles the “Pin Num.” column check box in the Options dialog box > **Colors** category.
Transparent Mode On/Off	T	Toggles Transparent mode on or off.
Unroute Display On/Off	U	Toggles Unroute display on or off.
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” on page 538.

Table 20. Modeless Commands for Net Name Visibility

Name	Command	Description
Toggle Visibility of Net Names	NN	Toggles the visibility of net names on pins, traces and/or vias based upon the individual settings of the NNP, NNT and NNV commands. **Tips:** • • This command toggles the “Net Nm.” check box in the Options dialog box > **Colors** category. • • When net name display is turned off with the NN command, no net names are displayed, irrespective of the NNP, NNT, and NNV settings. • • When net name display is turned on with the NN command, net names are displayed or not displayed, according to the NNP, NNT, and NNV settings. • • When the “Net Nm.” column check box is enabled, net name visibility is still restricted

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

Name	Command	Description
		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.
Toggle Display of Net Names on Pins	NNP	Toggles the display of net names on pins on/off. **Tip** This command toggles the Net names on...Pins check box in the Options dialog box > **Colors** category. **Restriction:** The display of net names also depends on the “Net Nm. column (visibility) check box and the presence of a non-background color tile on the layer in the Options dialog box > **Colors** category.
Toggle Display of Net Names on Traces	NNT	Toggles the display of net names on traces on/off. **Tip** This command toggles the Net names on...Traces check box in the Options dialog box > **Colors** category. **Restriction:** The display of net names also depends on the “Net Nm. column (visibility) check box and the presence of a non-background color tile on the layer in the Options dialog box > **Colors** category.
Toggle Display of Net Names on Vias	NNV	Toggles the display of net names on vias on/off. **Tip** This command toggles the Net names on...Vias check box in the Options dialog box > **Colors** category. **Restriction:** The display of net names also depends on the “Net Nm. column (visibility) check box and the presence of a non-background color tile on the layer in the Options dialog box > **Colors** category.

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

Name	Command	Description
Quick Layer View	Z	With no command arguments, Z displays the initial layer view.

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

Name	Command	Description
Add or Remove Layer from Current Set of Displayed Layers	Z {+<layer>} {-<layer>}	Adds or removes a layer from the current set of displayed layers. For example: • 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>	Allows you to restrict the display to a specified range of layers. For example, typing “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> …}	Allows you to restrict the display to only the layers you type. For example, typing “Z 2 4 d” displays layers 2, 4, and the documentation layer.
View All layers	Z *	Displays all layers in the design. **Restriction:** Z supports only the asterisk * regular expression.
View Items on Layer 0	Z 0	View only items that are visible on all layers. Items placed on Layer 0 are visible on all layers.
View the Active Layer	Z A	Displays the currently active layer. If the active layer is changed, it has no effect on the display.
View Only the Bottom Layer	Z B	Restricts the display to show the bottom layer only.
View Only the Current Layer	Z C <-C>	Changes the display to show the current layer only. 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	Displays all documentation layers in the design.
View All Electrical Layers	Z E	Displays all electrical layers in the design.

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

Name	Command	Description
		Tip You can also display an individual electrical layer (using Z C to make the current layer visible) and then switch the display to an adjacent electrical layer (using Alt+wheel backward or Alt+wheel forward), skipping any non-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 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.
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 Enter 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>	Sets the current layer to the number or name you enter, for example (L 2) or (L top).
Paired Layer Command	PL <n1> <n2>	Paired layers command, where<n1> and <n2> can be the layer number or name, for example, PL 1 2 or PL top bottom.Sets the Layer pair setting in the Options dialog box > Routing category General subcategory.

Table 23. Modeless Commands for Vias

Name	Command	Description
Toggle Between End Via Modes	E	Cycles the End Via mode between the following three settings: • 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** The shortcut dialog does not appear for this command, but the status of the change appears in the Status bar.
Select Via Type	V <name>	Via type is set to the name you enter.
Automatic Via Selection	VA	Via type set to automatic via selection.
Via Guide Toggle	VG	Toggles Via guide on or off.
Use Partial Via	VP	Via type set to partial vias only.
Use Through Hole Via	VT <name>	Via type set to through-hole vias only.

Table 24. Modeless Commands for Highlighting

Name	Command	Description
Highlight Specific Net On/Off (supports regular expressions ?*[])	H {<net>}	Net highlighting on or off. If you specify a net, you can substitute alphanumeric characters in the net name with the ? * and [] regular expression pattern operators. For examples, see "Expressions in Shortcut Keys" on page 560.
Highlight Component Pads with Rules	HCR	Highlights component pads with the current Highlighted object display on page 441 setting to indicate that the component has component rules. Other indicators are displayed when using component rules. For more information, see "Creating Traces Using Component Rules" on page 308.
Highlight in Dim Color Mode On/Off	HD	Highlight Dimming. Sets the Highlighted object display on page 441 setting to Distinguish highlighted objects by Dimming other object colors.
Highlight in Gray Scale Mode On/Off	HG	Highlight Graying. Sets the Highlighted object display on page 441 setting to Distinguish highlighted objects by Graying other objects.
Highlight in Highlight Mode On/Off	HH	Highlight Hatching. Sets the Highlighted object display on page 441 setting to Distinguish

Table 24. Modeless Commands for Highlighting (continued)

Name	Command	Description
		highlighted objects by crosshatching the highlighted object.
Highlight On/Off (Current Object)	HO	Toggles highlighting on or off for highlighted objects. Turns off the Highlighted object display on page 441 setting or turns it on using the last used setting. For example, objects receive highlighting when you use the Edit > Highlight command or when you route a net.

Table 25. Modeless Commands for Plower Settings

Name	Command	Description
Push Trace Behind Mode	PB	Toggles Push trace behind on or off (temporarily) for the current trace during interactive routing.
Plow After Click Mode	PC	Enables Plow after click mode (temporarily) for the current trace during interactive routing.
Plow With Pointer Mode	PP	Enables Plow with pointer mode (temporarily) for the current trace during interactive routing.
Rip Up Mode	PR	Enables Plower rip up mode (temporarily) for the current trace during interactive routing.
No Plowing	PX	Disables Plowing (temporarily) for the current trace during interactive routing.

Table 26. Modeless Command for Pour Outlines

Name	Command	Description
Pour Outline On/Off	PO	Pour outline on/off. This toggles the display mode of copper planes between pour outline and hatch outline.

Table 27. Modeless Commands for General Mode Settings

Name	Command	Description
Smooth Traces on Complete On/Off	SC	Toggles Smooth Traces on Complete on or off.
Direction Guides	DG	Toggles the direction guides on or off when you move a trace segment. The direction guides are lines that radiate out from the pointer to indicate the direction in which you can proceed.

Table 27. Modeless Commands for General Mode Settings (continued)

Name	Command	Description
		The guides represent the routing angle in the Routing options on page 450.

Table 28. Modeless Command for Measurement

Name	Command	Description
Quick Measure	Q	Quick measure with dynamic ruler. Attaches a measurement line to the pointer and displays dx, dy, and hypotenuse information, depending on pointer movement. Place the pointer at the starting point, then type the "q" modeless command. Drag the pointer to create a line between the start and end point of your measurement. Snaps to the design grid when the Snap to grid is on. Measurements are gridless when Grid Snap is off. Dynamically reports delta x, delta y, and delta x,y in current design units. You can measure precise Euclidean distances between polar grids using this command. For more information about using the Q shortcut key, see "Measuring Distance Between Objects" on page 211. **Tip** The shortcut dialog does not appear for this command, but the Status gives tips on usage.

Table 29. Miscellaneous Modeless Commands

Name	Command	Description
Move Sequential	MS	Move sequential. After selecting a predetermined set of components, use Move Sequential to sequentially place components on your pointer for placement in the design. For more information, see "Placing Components Sequentially" on page 207.
Set Arc Radius	RAD	Arc radius set to the <radius> value, for example, RAD 45.

Table 30. 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 "Crash Detection" on page 576

Table 30. Modeless Commands for the Basic Media Wizard/Log Test (continued)

	Command	Description
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 replaced using the BLT modeless command. • BMW is also a command line option. See also “Crash Detection” on page 576
Start BMW Session Logging	BMW ON	Starts BMW session logging.
Stop BMW Session Logging	BMW OFF	Stops BMW session logging.

Shortcut Keys

Using shortcut keys, you can start commands directly from the keyboard or mouse without navigating through menus. You can use shortcut keys (shortcuts) to start commands, change system settings, or start macros. You can also create new shortcuts.

Shortcuts are context-sensitive and mode sensitive. For example, in normal mode, pressing Ctrl+A selects all selectable design objects. In interactive routing mode, however, pressing Ctrl+A adds an arc.

A shortcut is unavailable when its associated command cannot be started. For example, the shortcut for the SaveAs command is unavailable when no design is loaded.

Displaying Available Shortcuts

SailWind Router provides a number of different methods that allow you to view your available shortcut choices.

The following table shows how to display the available shortcuts:

Table 31. How to Display Shortcuts

Source	Contains	How to display
Online help	Standard shortcuts only	See also "“Shortcut Keys” on page 106”.
HTML report	All shortcuts, which are sorted into general and mode-specific groups	See also "“Listing Available Shortcut Keys” on page 560”.
Shortcut dialog box	Standard and custom shortcuts available for the current mode	Open the Shortcut dialog box from the standard toolbar or from the View menu.

Starting Commands with Shortcuts

You can start commands by clicking on the workspace and typing the key sequence for the shortcut and any arguments it may have. As you type, the command may start immediately or shortcut dialog boxes may open to help you check the accuracy of the command argument or choose from similar shortcuts.

You can also start commands using mouse pointer events such as click and wheel rotation. However, you cannot create a shortcut that combines shortcut keys and mouse pointer events.

The program distinguishes between long and short shortcuts. A long shortcut consists of two or more keystrokes and possibly a combination of Ctrl, Alt, or Shift modifiers for the first keystroke. A short shortcut consists of one keystroke, or a mouse pointer event, and possibly a combination of Ctrl, Alt, or Shift modifiers:

Table 32. Responses to Shortcuts

Response	Why
Starts the command immediately. No shortcut-related dialog boxes open.	The shortcut has all of the following characteristics: • No command arguments • Short shortcut • Not similar to other shortcuts
Opens the Shortcut dialog box where you can view the key sequence you are typing, shortcut information, and syntax, if available.	The shortcut has any of the following characteristics: • One or more command arguments • Long shortcut • Similar to other shortcuts
Opens the Command Arguments dialog box, prompting you for required command arguments.	The shortcut cannot start the command until you provide the required command arguments.

Using a Shortcut to Start a Command with Arguments

You can use a shortcut to start the Change Layer command; then interactively supply the required layer number argument.

Procedure

1. Open a design.
2. Type L (lowercase), which is the standard shortcut for the Change Layer command.

   The Command Arguments dialog box opens with “Set Layer” displayed in the text box.
3. Type the layer number in the text box and press Enter to change the layer.

Using a Shortcut to Start a Customized System Command

You can use a customized shortcut to start the Change Layer command and supply the layer number.

Procedure

1. “Create a custom shortcut” on page 558 for the Change Layer command that includes a command argument value. For this example, assume that you have created a shortcut named L3 that includes 3 as the command argument value.
2. Open a design.

3. Type the key sequence L3, which is the custom shortcut you created in step 1.

   The Shortcut dialog box opens with L3 displayed in the text box.

4. Press Enter to change the layer to 3.

> **Note:**
> For more information, refer to “Creating a New Shortcut Key” on page 558 and “Listing Available Shortcut Keys” on page 560.

Starting a Shortcut

The Shortcut dialog box displays the shortcut key sequence as you type, allowing you to check its accuracy before starting the shortcut. Shortcut information and syntax, if available, appear below the text box. Parentheses enclose optional arguments. Similar shortcuts are displayed in the Available commands and macros box; click **More** to display the box.

Procedure

1. Start typing the shortcut; the Shortcuts dialog box opens automatically.

2. Do one of the following:

   - Type the shortcut into the text box.

   - Select the shortcut from the Available commands and macros box. To re-sort the contents of the Available commands and macro box, click a column header.

3. If needed, type space-delimited command arguments into the text box.

> **Tip**
> Use the wildcard character * to substitute for any number of characters, and use the wildcard character ? to substitute for one character. For example, to highlight all nets whose names start with the letter A, type H A*.

4. Press Enter.

> **Tip**
> To immediately start multiple-character shortcuts that do not have command arguments, clear the “Wait until <Enter> before executing long shortcuts” check box in the Customize dialog box (**Tools > Customize** menu item, **Options** tab).

Note:
To define new shortcuts using the Customize dialog box, right-click over most areas on the Shortcut dialog box, click **Customize**, and then click the **Keyboard and Mouse** tab.

Restriction:
If you opened the Shortcut dialog box by typing a shortcut, you cannot delete the initial character in the text box. To display all available shortcuts, open the Shortcut dialog box by clicking the **Shortcut Dialog** button on the main toolbar.

Related Topics

Modeless Commands and Shortcut Keys

Shortcut Keys

Starting a Shortcut with Command Arguments

The Command Arguments dialog box displays the command arguments key sequence as you type, allowing you to check its accuracy before starting the shortcut. The Command Arguments dialog box opens automatically when you type a shortcut if you have not yet provided a required command argument.

Procedure

Type the space-delimited arguments into the text box and press Enter.

Note:
Shortcut information and syntax, when available, appear below the text box. Parentheses enclose optional arguments.

Tip
Use the wildcard character * to substitute for any number of characters, and use the wildcard character ? to substitute for one character.

Examples

To highlight all nets whose names start with the letter A, type the key sequence H A*.

Related Topics

Shortcut Keys

Modeless Commands and Shortcut Keys

Function Keys

Function keys allow you to use the standard keyboard function keys to initiate specific operations.

The following table defines the function key command assignments:

Table 33. Function Key Command Assignments

Function Key	Description
F1	Open Help (context sensitive)
F2	Unassigned
F3	Interactive Route (Dynamics Enabled)
F4	Layer Pair Toggle during routing. **Note:** F4 no longer repeats the last command.
F5	Zoom In (Navigation Window)
F6	Zoom Out (Navigation Window)
F7	Swap Workspace and Navigation Window contents
F8	Lock View (Navigation Window)
F9	Autorouting, start
F10	Autorouting, resume
F11	Autorouting, pause
F12	Autorouting, stop

Keypad Keys

You can initiate specific commands using the numeric keypad keys on your keyboard.

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

Table 34. Keypad Key Command Assignments (continued)

Keypad Keys	Description
Keypad (2)	Pans the workspace down one increment
Keypad (3)	Zooms out at the pointer
Keypad (4)	Pans the workspace left one increment
Keypad (5)	Starts Zoom from center (Num Lock only)
Keypad (6)	Pans the workspace right one increment
Keypad (7)	Zoom to the Board
Keypad (8)	Pans the workspace up one increment
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

For menu-free operation, start commands using shortcut keys.

Tip
You can also create or change shortcut key assignments on page 558.

The following tables list all 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.
Zoom to Board	Ctrl + B	Zooms to the board. Fits the board outline into the workspace.
Zoom Extents	Ctrl-Alt + E	Zooms to extents. Fits all objects in the design into the workspace.
Zoom Area In/Out	MMB (drag)	Zooms area in or out. Drag pointer up to zoom in. Drag pointer down to zoom out.
Start Zoom from Corner	Shift + MMB (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)	MMB	Centers the view at the pointer.
Center View (Using Pointer Location)	<Insert>	Centers the view at the pointer.
Zoom In at Pointer (Zoom Mode)	LMB Click	Zooms in at the pointer (zoom mode).
Zoom Out at Pointer (Zoom Mode)	RMB Click	Zooms out at the pointer (zoom mode).
Zoom In at Pointer (Zoom Mode)	<spacebar>	Zooms in at the pointer (zoom mode).
Zoom In at Pointer	<PgUp>	Zooms in at the pointer.
Zoom Out at Pointer	<PgDn>	Zooms out at the pointer.
Zoom In at Pointer	Ctrl + Wheel Fwd	Zooms in at the pointer.
Zoom Out at Pointer	Ctrl + Wheel Back	Zooms out at the pointer.
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.
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.

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

Name	Shortcut Keys	Description
Dynamic Panning	Alt + MMB (Drag)	Pans the sheet area below the pointer to the center of the workspace.
Pan Workspace Down (One Line)	Wheel Back	Pans workspace down one line.
Pan Workspace Up (One Line)	Wheel Fwd	Pans workspace up one line.
Pan Workspace Right (One Line)	Shift + Wheel Back	Pans workspace right one line.
Pan Workspace Left (One Line)	Shift + Wheel Fwd	Pans workspace left one line.
Pan Workspace Down (One Pixel)	Ctrl-Alt + Wheel Back	Pans workspace down one pixel.
Pan Workspace Up (One Pixel)	Ctrl-Alt + Wheel Fwd	Pans workspace up one pixel.
Pan Workspace Right (One Pixel)	Alt-Shift + Wheel Back	Pans workspace right one pixel.
Pan Workspace Left (One Pixel)	Alt-Shift + Wheel Fwd	Pans workspace left one pixel.

Table 36. Keyboard Shortcuts for Selection

Name	Shortcut Keys	Description
Select	LMB 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	LMB (Start Drag)	Starts an area selection.
Area Complete	LMB (End Drag)	Completes an area selection.
Cancel Area Selection	LMB (Cancel Drag)	Cancels an area selection.
Cycle Selection	<Tab>	Cycles the selection of adjacent objects within the pick radius.
Toggle Selection	Ctrl + LMB Click	Toggles object selection.
Toggle Area Selection	Ctrl + LMB (Drag)	Toggles an area selection

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.
Print/plot	Ctrl + P	Print/plot
Save (Quick Save)	Ctrl + S	Saves the design file

Table 38. Keyboard Shortcuts for Opening Menus and Dialog Boxes

Name	Shortcut Keys	Description
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	RMB	Opens the Shortcut menu.
Properties (Current Object)	LMB DblClick	Displays Properties for the currently selected object.
Properties (for Selected)	Alt+<Enter>	Opens Properties dialog box for selected object or, if no object is selected, for the design.
Properties (for Selected)	Ctrl + Q	Opens Properties dialog box for selected object or, if no object is selected, for the design.
Options	Ctrl + <Enter>	Opens the Options dialog box.
Open Colors Category of the Options Dialog	Ctrl-Alt + C	Opens the Colors category of the Options dialog box.
Open the Options Dialog (Last Viewed)	Ctrl-Alt + D	Opens the last viewed category of the Options dialog box.
Open the Options Dialog (Last Viewed)	Ctrl-Alt + G	Opens the last viewed category of the Options dialog box.
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	Ctrl-Shift + R	Rotates a component (90 degrees)
Rotate (During Move)	<Tab>	Rotates a component (during component move mode).
Rotate Component (90)	Ctrl + R	Rotates the selected components(90 degrees).
Spins Component	Ctrl + I	Spins a component.
Flip Side (Selected)	Ctrl + F	Flips an object to the opposite side of the board.
Flip Side (Component)	Shift + F	Flips a component to the opposite side of the board.

Table 40. Keyboard Shortcuts for Routing Operations

Name	Shortcut Keys	Description
Add Corner	LMB Click	Adds a corner (interactive routing).
Add Corner	<Spacebar>	Adds a corner (interactive routing).
Add Arc of Radius	Alt + R	Adds an arc of radius (interactive routing). Set the radius with the RAD shortcut.
Invert Arc	Alt + I	Inverts an arc (interactive routing).
Add Via (Interactive Routing)	Shift + LMB Click	Adds a via (interactive routing).
Add Arc (Interactive Route)	Alt + A	Adds an arc (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	LMB Click	Complete (interactive routing) when pointer is near a valid completion point.
Complete	LMB DblClick	Complete (interactive routing) when pointer is near a valid completion point.
End (routing)	Ctrl + LMB Click	End (interactive routing).
Length Minimization	Ctrl + M	Runs length minimization.
Reset Length Monitor to 0	Crtl + <PgDn>	Resets length monitor to zero (interactive routing).

Table 40. Keyboard Shortcuts for Routing Operations (continued)

Name	Shortcut Keys	Description
Length Monitor On/Off	Ctrl + L	Toggles the Length Monitor On/Off
Estimated Length Toggle (Pin Pair vs. Net)	Shift + E	Toggles the display of the estimated length (Et) in the length monitor between pin pair length and net length.
Move Trace to Current Layer	Ctrl + MMB Click	Move trace to current layer.
Add Via While Routing	Alt-Shift + LMB Click	Add via while routing.
Cycle Via Pattern (Diff Pairs)	Ctrl + <Tab>	Cycles through the via pair patterns, differential pair routing.
End Routing	Ctrl-Alt + LMB Click	End Routing.
Previous Base Point	Shift + <Tab>	Previous Base Point.
Next Base Point	Ctrl + Alt + <Tab>	Next Base Point.
Accordion Beginning (Interactive Routing Mode)	Shift + A	Begin an accordion, interactive routing mode.
End Accordion	LMB DblClick	Ends an accordion, interactive routing.
Dynamic Reconnect On/Off	Shift + N	Toggles Dynamic Reconnect on or off.
Dynamics Mode On/Off	Shift + D	Toggles Dynamics mode on or off, interactive routing.
Grid Snap On/Off (Routing Grid Only)	Alt + G	Toggles snap to routing grid on or off.
Guard Bands On/Off	Shift + G	Toggles guard bands on or off.
Switch Rules	Shift + R	Switches rules.
Switch Trace (Route Separately Mode)	<Tab>	Switches trace, route separately mode.
Show Length of Alternate Trace (Trace Length Monitor in Diff Pairs Mode)	<Tab>	Shows the length of the alternate trace in the differential pair. Use this with the trace length monitor to check the length of each member of the differential pair.
Split Pair Around Obstacle (Diff Pairs Mode)	Shift + X	Split pair around obstacle, routing differential pairs.
Route Separately (Diff Pairs Mode)	Shift + Z	Routes separately, routing differential pairs.

Table 41. Keyboard Shortcuts for Layer List Navigation

Name	Shortcut Keys	Description
Change Current Layer to Previous in List	Alt + <Up Arrow>	Changes the current layer to previous in list.
Change Current Layer to Next in List	Alt + <Down Arrow>	Changes the current layer to next layer in list.
Change Current Layer to Previous in List	Alt + Wheel Back	Changes the current layer to previous in list.
Change Current Layer to Next in List	Alt + Wheel Fwd	Changes the current layer to previous in list.

Table 42. Keyboard Shortcuts for Protection of Objects

Name	Shortcut Keys	Description
Protect Objects	Shift + P	Protects objects.
Unprotect Objects	Shift + U	Unprotects objects

Table 43. Keyboard Shortcuts for DRC Operations

Name	Shortcut Keys	Description
DRC Setting Dialog	Alt + D	Opens the DRC Settings dialog box.
Explain Last Error	Alt + X	Explains the last DRC error (component move mode).

Table 44. Keyboard Shortcuts for Editing

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

Table 44. Keyboard Shortcuts for Editing (continued)

Name	Shortcut Keys	Description
Undo	Ctrl + Z	Undo.
Highlight Object	Ctrl + H	Highlights Object.
Unhighlight	Ctrl + U	Removes highlighting from current object.
Toggle Highlight	Alt + LMB Click	Highlighting on or off.
Area Highlight	Alt + LMB (Drag)	Highlights objects in the enclosed area.

Table 45. Keyboard Shortcuts for Viewing

Name	Shortcut Keys	Description
Next View	Alt + N	Displays the next view.
Previous View	Alt + P	Displays the previous view.
Previous View	Ctrl + Alt + P	Displays the previous view.

Table 46. 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 47. Miscellaneous Keyboard Shortcut Commands

Name	Shortcut Keys	Description
Check Clearance Toggle for Current Operation	Shift + C	Checks clearance.
Database Integrity Test	Shift + I	Runs database integrity test.

Chapter 7
View

SailWind Router provides controls for changing the display of a design, from changing display colors to scrolling and zooming across the details.

Setting Colors for the Design

As you work on a design, you can customize display colors to make it easier to see objects as you place them.

Use the Colors category of the Options dialog box to perform the following tasks:

- 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 custom color scheme. You can then switch among color schemes you commonly use to work with a design.

Procedure

1. Click the **Tools > Options** menu item.
2. In the Options dialog box, click the Colors on page 422 category.
3. Set the color(s) for design objects:

a. To set the color for one object, in the Color selection 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. (Visibility of the object must be enabled first by clicking the Visible check box for the corresponding row.)

Figure 14. Enabling Object Viewing

> **Tip**
> To change the palette of colors from which you can select, see "“Changing the Color Palette” on page 121”.

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 Color selection 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 Color selection 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).

e. To copy a color definition from one row or column and paste it to another row or column:

  i. Select the row or column with the color definition that you want to copy and press Ctrl+C.

  ii. Select the row or column to which you want to paste the color definition.

  iii. Press Ctrl+V.

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

Results

When you save the design, SailWind Router stores the setting with design data.

You can save display color settings you commonly use as you work on a design. See ""Saving Color Schemes" on page 121".

You can use modeless commands to toggle the display of pin numbers and net names. See ""Table 19" on page 97".

Changing the Color Palette

You can change the colors that appear in the color palette.

Procedure

1. Click the **Tools > Options** menu item.
2. In the Options dialog box, click the Colors on page 422 category.
3. In the "Color selection" area, click the color tile that you want to change.
4. Click **Custom**.
5. 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.
6. 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.
7. Click **OK** to close the Options dialog box.

Results

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

Related Topics

Setting Colors for the Design

Saving Color Schemes

Saving Color Schemes

You can save to file the color assignments you have specified in the Colors category of the Options dialog box.

Procedure

1. Click **Tools > Options**.
2. In the Options dialog box, click the Colors on page 422 category.
3. Set the color assignments on page 119 for the scheme.
4. In the Color scheme area, click **Save As**.
5. In the Save Scheme dialog box that appears, type the scheme name.

Results

To use a previously saved configuration, select the configuration name from the “Color scheme name” list.

Related Topics

Setting Colors for the Design

Highlighting and Unhighlighting Design Objects

You can highlight or clear the highlighting from objects in your design to distinguish them from other design objects. Highlights help you visually locate objects when working on a dense design.

You can set how highlighting is displayed using the "Distinguish highlighted objects by" list in the "Options Dialog Box, Global Category, General Subcategory" on page 441.

Highlighting Objects

You can highlight one or more objects in the design.

Procedure

Press and hold the Alt key then select one or more objects. Alternatively, select an object then click the **Edit > Highlight** menu item.

Tip
You can also use the HG, HD, and HH shortcut keys to highlight objects. See the "Table 24" on page 102.

Unhighlighting Objects

Remove highlighting from one or more objects in the design.

Procedure

Press and hold Alt, and click the highlighted object or objects. Alternatively, select one or more highlighted objects then click the **Edit > Unhighlight** menu item.

Unhighlighting All Objects

You can remove highlighting from all objects in the design with one command.

Procedure

Press and hold Alt, and click in an empty area of the design, or click the **Edit > Unhighlight All**.

Tip
You can also use the HG, HD, and HH shortcut keys to do this. See the "Table 24" on page 102.

View Controls

You can control which areas of a design are visible.

Use the commands in the following table to control the display in the workspace.

Table 48. Display Commands

Command	Description
Zoom	Moves closer to or farther from the design. Access this control through the main toolbar or the View menu.
View Board	Fits the board outline into the workspace. Access this control through either the main toolbar or the View menu.
View Extents	Fits the entire design, including objects outside the board outline, into the workspace. Access this control through either the main toolbar or the View menu.
View Selection	Fits a selected object or area in the workspace. Access this control through the View menu.
Previous View	Displays the previous view in the workspace. Access through the View menu.
Next View	Displays the next view in the workspace. Access through the View menu.
Layer List	Displays a specific layer on top of other layers. Access as a dropdown list on the main toolbar.

Controlling the View

Using the keys on the keypad located at the far right of most keyboards, you can control which part of the design is visible on the screen. NumLock can be on or off except where noted.

Table 49. View Commands

Keys	Description
Home	Fits the board into the workspace.
End	Redraws the current view.
Arrows	Pan the viewing window when NumLock is on. Move one-half the screen width in the direction of the arrow. With NumLock off, move by grid unit.
Page Up	Zooms in centered at the pointer location.
Page Down	Zooms out centered at the pointer location.
Ins	Centers the view at the current pointer location, without zooming.

Use the vertical and horizontal scroll bars to pan the view vertically or horizontally.

Displaying Selected Nets

Use the View Nets dialog box (accessible from the View menu) to select and view nets. The View Nets dialog box shows View details for selected nets. For each net, you can hide or show connections, routed paths, or unrouted paths. When you move nets into the View list box, you can specify color and show or hide various unroutes.

Viewing Nets by Color

Use the color palette to show a selected net in a specific color. When you assign a color to a net, the pads, vias, and connections on all layers appear in the assigned color. This makes it easier to identify nets when you split planes. Any net color assignments you make are saved when you save the design.

Pan, Zoom, and Scroll the Display

Use the mouse wheel to scroll, pan, and zoom the workspace display.

The following mouse/key combinations affect the display:

- Roll the mouse wheel to scroll the display up and down.
- Press and hold the Shift key while rolling the mouse wheel to pan left and right.
- Press and hold the CTRL key while rolling the mouse wheel to zoom in and out.

Scrolling Through Layers Using the Wheel

If desired, you can press and hold Alt and roll the mouse wheel to scroll through layers of the design. Use this feature for previewing layers while routing interactively and for displaying individual electrical layers.

Previewing Layers While Interactively Routing

The ability to preview layers during routing and trace editing helps you determine which layer has the most space for additional traces.

Procedure

1. While interactively routing, press Alt and roll the wheel button backward or forward.
2. Do one of the following:
   - To transition layers at the current pointer location, press Shift and click. A via appears at the pointer location, allowing you to continue routing from that point on the layer you previewed.
   - To move the trace in progress (from the last corner location), press Ctrl and click the middle mouse button. A via appears at the last corner location and moves the trace to the layer you previewed.

Displaying Individual Electrical Layers

You can cycle the display through the set of individual electrical layers. Switching the display to adjacent electrical layers skips over non-electrical layers and also renders all other layers invisible.

Procedure

1. Type the shortcut key Z C to make the current layer visible and make all other layers invisible. The current layer does not have to be an electrical layer.
2. To switch the display to an adjacent electrical layer, press Alt and roll the wheel button backward or forward. As an alternative, you can use Alt+Arrow up instead of Alt+wheel backward. You can use Alt+Arrow down instead of Alt+wheel forward. When you try to switch beyond the outermost electrical layer, the display loops back to the outermost electrical layer on the other side of the board.

Viewing by Layer Using the Layers List

SailWind Router provides a Layers list view to allow you to display a specific layer on top of other layers.

Note:
The board outline, unroutes, thermal indicators, and test point indicators always appear on top of other objects, regardless of layer selection.

Procedure

1. On the standard toolbar, open the Layers list.
2. Click the layer you want to display on top.

   Tip
   If you select All Layers, layers appear from bottom to top.

Showing Previous or Next Views

Use Previous View and Next View to display previous and next views recorded by the program. Show Previous and Show Next work like multilevel undo and redo commands. You can save up to 100 views. When the view buffer is full, newer views replace the oldest views.

Procedure

To show the previous view, click the **View > Show Previous** menu item. To show the next view, click the **View > Show Next** menu item.

Chapter 8
Undo, Redo, Repeat, Delete

Use the Undo, Redo, Repeat, and Delete commands to cancel or repeat your last operation on a design object. Undo and Redo clear all objects in the design before running. The Undo command updates graphics in the workspace and updates the Project Explorer.

An undo buffer contains recorded database changes. When the buffer is full, the oldest change is deleted.

New, Open, Save, Save As, Restore, and Autoroute clear the undo and redo buffer, and make Undo and Redo unavailable.

Use the Undo and Redo list boxes on the main toolbar to open the list of recent operations. Select the operations in the list you want to undo or redo. Use these commands to undo and redo your last command operation. Undo and Redo deselect all objects in the design before running.

Undoing a Command

Undo the last command on a selected object.

Procedure

Click the **Edit > Undo** menu item, or click the **Undo** button on the main toolbar.

Redoing a Command

Redo the last undo on a selected object

Procedure

Click the **Edit > Redo** menu item, or click the **Redo** button on the main toolbar.

Undoing Multiple Commands

You can undo multiple actions at one time.

Procedure

1. Click the arrow next to the **Undo** button on the standard toolbar to see a list of the most recent actions you can undo.
2. Click the action you want to undo. When you undo an action, you also undo all actions above it in the list.

Redoing Multiple Undos

You can redo multiple undos.

Procedure

1. Click the arrow next to the **Redo** button on the standard toolbar to see a list of the most recent actions you can redo.
2. Click the action you want to redo. When you redo an action, you also redo all actions above it in the list.

Repeating Operations

You can repeat a series of operations, thus simplifying a repetitive sequence of steps.

Procedure

1. Select an object in the design.
2. Perform a command or series of commands.
3. Select another object.
4. On the standard toolbar, click the **Repeat** button.

   Alternatively, click the **Edit > Repeat** menu item.

Examples

You want to unroute and re-fan out two or more components in your design. You also want to highlight the components. To do this:

1. Select one of the components.
2. Right-click and click the **Unroute** popup menu item.
3. Right-click and click the **Fanout** popup menu item.
4. Click the **Edit > Highlight** menu item.
5. Select the next component.

6. Click the **Edit > Repeat** menu item.

7. Repeat steps 5 and 6 for any additional components.

Chapter 9
Search and Selection

You can select design objects directly in the workspace, in Project Explorer, and in the Spreadsheet window.

Selection Modes

You can execute a command on a design object in two ways: Select the object and then select the command (known as *object mode*), or select the command and then select the object (known as *verb mode*).

Object Mode

In object mode, you click an object to select it before choosing a command. After selecting the object, you choose a command using one of the following methods:

- Right-click and select a command from the shortcut menu.
- Select a command in a toolbar or from a menu.

Verb Mode

In verb mode, you select the command you want to execute from a toolbar or menu, then you click one or more objects on which you want to perform the command. Verb mode automatically sets the selection filter to items that respond to the active command. For example, if you are in Move Component verb mode, the selection filter changes to allow selection of components only. After moving one component, you can move another simply by selecting it.

Verb mode commands are also available on toolbars.

Selecting Objects

SailWind Router provides you with multiple ways of selecting a design object.

You can select a design object using one of the following methods:

- Click directly on the object in the workspace.
- Click near it and use the **Cycle** button to cycle through nearby objects.
- Use the Find Dialog Box to locate it and select it.
- Click on the object in Project Explorer. (If necessary, right-click and click the **Allow Selection** popup menu item to enable selection of objects in the Project Explorer.)
- Use the Find Dialog Box, Spreadsheet Window to locate it and select it.

When you select an object, all its elements are selected and highlighted. For example, when you select a component, all of its pins, copper, and text are also selected. Any previously selected objects are unselected.

If you double-click an object, it is selected and its properties dialog box appears. If you press Esc or right-click and click Cancel, all objects are unselected.

You can set how selected objects appear on screen using the “Distinguish highlighted objects by” list in the Options dialog box > **Global** category > General on page 441 subcategory.

The table below summarizes the different ways to select objects.

Table 50. Object Selection Methods

To Select	Action
Single object	Click an object.
Objects in a rectangular area	Starting at one corner of the area, click and hold the left mouse button, and drag to the diagonally opposite corner. Release the button.
All objects	On the Selection Filter toolbar, click the **Select All** button. You can also press Alt+A. Select All selects all objects according to the type enabled on the Selection Filter toolbar.
Series of objects	Press Ctrl and click items to add or remove, in sequence.
Extended objects	Press Shift and click to select objects related to selections. For example, if a pin is selected, and you Shift and click it, all other pins in the net are selected.
Objects in a dense or crowded area	Use the Selection Filter toolbar to prevent certain objects from being selected.
Part of objects	Use the Selection Filter toolbar to select only part of objects, such as trace corners.
Trace paths	Using the **Path** button on the Selection Filter toolbar, click at the start of the path and then click at the end of the path.

Selection Status on the Status Bar

When you select objects, the status bar shows which objects are selected.

For single selection, the following information appears:

[object type] [object name (if any)] [additional information]

For example:

Component: U1, Top

For multiple selections of the same things (for example, all nets), up to ten object names appear. For multiple selections of different things (for example, a mixture of nets and components), up to five object names appear for each type of object.

For errors, the selected error count is listed along with the error type for the last error in the selection.

Extending Selections

Pressing and holding the Shift key while clicking an object selects both the object and its associated items. Selected objects change to the color set for Selections in the **Display** tab of the Options dialog box.

Table 51 summarizes the effect of clicking an object while holding the Shift key.

Table 51. Objects and their Associated Items

Object	Shift and click selects
Component	Component and all attached traces
Pin	All pins in the net connected to the selected pin
Net	All nets in the net class if the net is a part of a net class
Unroute, trace, pin pair, via	Connected nets
Trace segment or corner	Connected nets
Keepout, text, error	No associated items
Copper, copper plane	Connected nets

Restriction:
The following restrictions apply:

- You cannot select pin pairs and nets at the same time.
- You cannot Undo or Redo a selection.

Select Part of an Object

You can select part of an object using special filters on the Selection Filter toolbar.

Note:
You can set how selected objects appear on screen using the “Distinguish highlighted objects by” list in the Global / General Options on page 441.

Use the following Selection Filter commands to filter selections:

- **Segments and corners**— Selects trace segments and corners.
- **Path**— Selects a continuous portion of a trace. The trace path can start and end at any point along a continuous sequence of trace segments.

Selecting a Trace Corner

You can select a trace by a corner. Selecting a corner allows you to move or relocate it.

Procedure

1. Click the **Tools > Options** menu item > **Global** category > **General** subcategory.
2. In the “Pointer settings” area, set the pick box size in the “Pick radius” box. Click **OK**.
3. On the Selection Filter toolbar, click the **Segments and Corners** button.
4. Click the trace corner to select.

Tip
If you have short trace segments that are larger than the pick radius, either zoom in or cycle selections.

Selecting a Trace Path

If desired, you can select only a specific trace portion of a trace to move or edit it.

Procedure

1. On the Selection Filter toolbar **View > Toolbars > Selection Filter Toolbar** menu item), click the **Path** button.
2. Click the starting point of the path to select. The pointer changes to a path selection symbol that marks the start point of the path.

3. Click the ending point of the path.

Tip
To deselect a path, press Esc. In SailWind Layout, you can right-click and click the **Cancel** popup menu item, which you cannot do in SailWind Router.

Filtering Selections

Use the Selection Filter toolbar buttons to set the objects you can select. You can also set the selection filter using the shortcut menu that appears when you right-click in the workspace with nothing selected.

Procedure

1. On the Selection Filter toolbar (**View > Toolbars > Selection Filter Toolbar** menu item), click the **Layers** button.
2. In the Selection Layers list (of the Selection Layers dialog box), select the check boxes for each layer that you want to be able to select in the design; clear a check box if you want to disable selection of a layer.

   Alternatively, you can select the All electrical layers box at the top of the dialog box to enable all electrical layers for selection.
3. Click **Close**.
4. On the Selection Filter toolbar, click the buttons for the objects for which you want to enable or disable selection.

Restriction:
You cannot select pin pairs and nets at the same time.

Cycling Through Selections

If you are working in a densely populated area of a board, it may be difficult to select the correct object, even with the Selection Filter. You can cycle through all objects near a selected object, making it easier to select the correct object.

Procedure

1. Select an object near the object you want to select.
2. Press the Tab key.

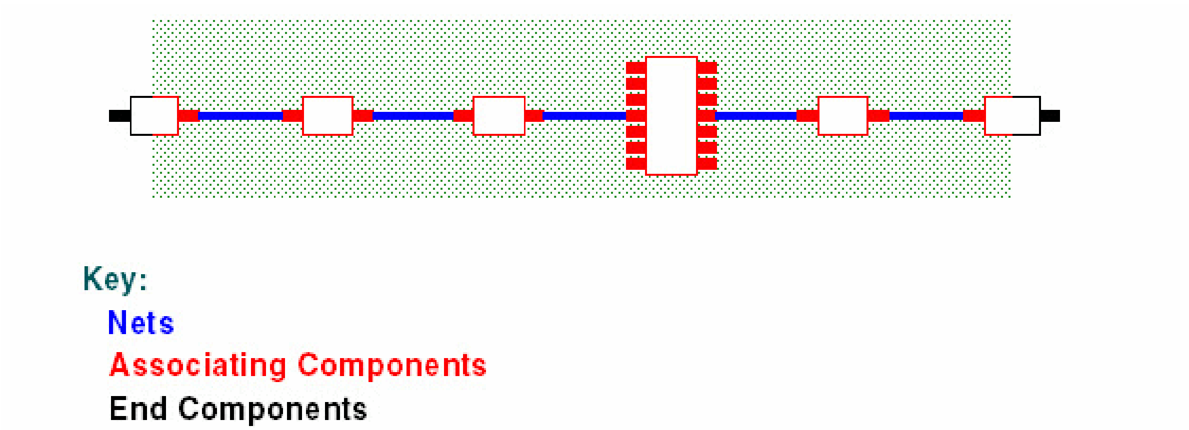
   Each time you press the Tab key, another object in the area is selected. If only one object is in the area, the Tab key has no effect

Chapter 10
Electrical Nets

You can associate an array of nets joined by discrete components, creating an electrical net to which you can apply 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 following figure shows an example of an electrical net.

Figure 15. Electrical Net Components and Nets



Length, differential pair and matched length rules can apply to an electrical net as though it were a single net.

- Electrical Net Creation and Modification
- Deletion of Electrical Nets
- Excluding Nets from Electrical Net Creation
- Excluding Components from Electrical Net Creation
- Canceling Electrical Net Creation by Nets
- Canceling Electrical Net Creation by Components
- Selecting Electrical Nets
- Setting Electrical Net Properties
- Creating a Matched Length Group of Electrical Nets
- Creating a Differential Pair of Electrical Nets
- Conditions Governing Electrical Net Creation

Electrical Net Creation and Modification

You can create or modify electrical nets to meet your design requirements.

There are three ways to create and modify your 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 want to mix methods in your design, you must first understand how the methods interact, as described in "Conditions Governing Electrical Net Creation" on page 151.

General Restrictions and Prerequisites

Consider certain restrictions and prerequisites before creating electrical nets.

The following restrictions apply to all 3 electrical net creation methods:

- A plane net cannot be part of an electrical net.
- A net having more pins than the limit specified in the "Maximum non-plane-net pin count" field cannot be part of an electrical net.

> **Tip**
> Virtual pins are not included in a net's pin count.

- An electrical net that would have more nets than the limit specified in the "Maximum net count per electrical net" field of the Electrical Nets dialog box is not created.
- Components through which an electrical net passes must be either discrete two-pin components, or multiple pin components that conform to the following conditions:
    - All the pins must connect to a gate.
    - Each gate must have exactly two pins.

  No component that does not conform to these conditions can create an electrical net, that is, no electrical net can go through it.

The following prerequisite is recommended for all 3 electrical net creation methods.

Before you begin creating electrical nets, you should make sure that you set the "Maximum net count per electrical net" and "Maximum non-plane-net pin count" fields in the Electrical Nets on page 409 dialog box appropriately for your design. These fields 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. If these values are set inappropriately for your design, SailWind Router does not create electrical nets that it would otherwise create, or the software creates them differently than you expect, as follows:

- The software does not create an electrical net if the potential electrical net exceeds the “Maximum net count per electrical net” limit.
- If any net in the potential electrical net exceeds the “Maximum non-plane-net pin count” limit, the software may split the electrical net, truncate it, or not create it at all, depending on the location of the faulty net.

Tip
Virtual pins are not included in a net’s pin count.

For more information, see “Conditions Governing Electrical Net Creation” on page 151.

Creating Electrical Nets by Selecting the Nets

Use this method if you want to create electrical nets by selecting nets.

Tip
Whenever you create, delete, or change electrical nets, the software regenerates all electrical nets in the design. Therefore, always verify that the results of an electrical nets operation are what you expect by reading the messages in the Output Window.

Procedure

1. In the workspace, select the nets you want to make into electrical nets.
2. Right-click and click the **Create Electrical Net** popup menu item.

   As an alternative, in the Net Properties dialog box, on the **Electrical Net** tab, select the Create electrical net check box.

Results

The selected nets are made into an electrical net, and the Net Properties Create electrical net check box is selected for all the selected nets.

Are the results different from what you expected?

- SailWind Router may have split or truncated an expected electrical net, or not created it at all, because one or more of its nets or components did not conform to the General Restrictions and Prerequisites for creating electrical nets.
- 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, it cannot create an electrical net, that is, no electrical

net can go through it. (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 151.

Related Topics

Canceling Electrical Net Creation by Nets

Excluding Nets from Electrical Net Creation

Creating Electrical Nets by Selecting the Components

Use this method if you want to create electrical nets by selecting components to associate its connected nets.

Tip
Whenever you create, delete, or change electrical nets, SailWind Router regenerates all electrical nets in the design. Therefore, always verify that the results of an electrical nets operation are what you expect by reading the messages in the Output Window.

Procedure

1. In the workspace, select the components whose nets you want to make into electrical nets.
2. Right-click and click the **Create Electrical Net** popup menu item.

   As an alternative, in the Component Properties dialog box, on the **Electrical Nets** tab, select the Create electrical net check box.

Results

The nets attached to the selected components are made into electrical nets, and the Component Properties "Create electrical net" property is set for all the selected components.

Are the results different from what you expected?

- SailWind Router may have split or truncated an expected electrical net, or not created it at all, because one or more of its nets or components did not conform to the General Restrictions and Prerequisites for creating electrical nets.
- **Mixed-method restriction:** If you have excluded a net that is attached to a selected component 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 nets and Allow electrical net creation... check boxes are both cleared.)

  For more information, see "Conditions Governing Electrical Net Creation" on page 151.

Related Topics

Excluding Components from Electrical Net Creation

Canceling Electrical Net Creation by Components

Creating Electrical Nets Automatically by Refdes Prefix

If you want to create electrical nets automatically, use this method to specify the refdes prefixes of any components through which the nets pass.

Tip
Whenever you create, delete, or change electrical nets, SailWind Router regenerates all electrical nets in the design. Therefore, always verify that the results of an electrical nets operation are expected by reading the messages in the Output Window.

Prerequisites

This procedure creates electrical nets for all components that have the specified refdes prefixes and conform to the General Restrictions and Prerequisites for creating electrical nets. Ensure the refdes prefixes of any set of components you plan to group into electrical nets are unique.

Procedure

1. Click the **Setup > Electrical Nets** menu item.
2. In the Maximum net count per electrical net box, type the maximum number of nets to be allowed in electrical nets.
3. In the Maximum non-plane-net pin count box, type the maximum number of pins allowed in nets belonging to electrical nets. Nets that have pin counts higher than this maximum are considered plane nets, and are not included in electrical nets.

   Tip
   Virtual pins are not included in a net’s pin count.

4. In the Discrete component prefixes area, enter the refdes prefixes of the components you want to be creating electrical nets.

   Tip
   The text entry field names are only for convenience; prefixes for any type of component can be entered in any field.

Results

When you click **OK** or **Apply**, electrical nets regenerate based on the new or changed set of specified refdes prefixes.

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.

Are the results different from what you expected?

- SailWind Router might have split or truncated an expected electrical net, or not created it at all, because one or more of its nets or components did not conform to the General Restrictions and Prerequisites for creating electrical nets.
- **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. (SailWind Router excludes a component or net 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 151.

Related Topics

Deleting Electrical Nets Manually

Excluding Nets from Electrical Net Creation

Excluding Components from Electrical Net Creation

Deletion of Electrical Nets

You can delete electrical nets manually or—if you created them automatically through the Electrical Nets dialog box—you can delete them automatically.

Deleting Electrical Nets Manually

You can remove electrical nets manually from the workspace or from Project Explorer.

Tip
Deleting an electrical net does not delete physical nets.

Procedure

1. In Project Explorer or the workspace, select the electrical nets you want to delete.
2. Right-click and click either the **Delete** or the **Delete Electrical Net** popup menu item.

Results

The Create electrical net check box is cleared for all the physical nets of the selected electrical nets. If any physical nets have also been made into electrical nets 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 which you have created using the refdes prefix method.

Tip
Whenever you create, delete, or change electrical nets, SailWind Router regenerates all electrical nets in the design. 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.

Procedure

1. Choose the **Setup > Electrical Nets** menu item.
2. Delete the refdes prefixes of the components that should not be creating electrical nets.
3. Click **OK**.

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 also has the Component Properties “Create electrical net” check box selected, the nets of that component remain in an electrical net.
    - If a component having the deleted refdes prefix has the “Allow electrical net creation by refdes prefix and nets” 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 checked, those nets will remain in an electrical net.

  For more information, see “Conditions Governing Electrical Net Creation” on page 151.

Related Topics

General Restrictions and Prerequisites

Excluding Components from Electrical Net Creation

Excluding Nets from Electrical Net Creation

You can select physical nets for exclusion from any electrical net.

Tip
This procedure clears 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 151.

CAUTION:
Whenever you create, delete, or change electrical nets, SailWind Router regenerates all electrical nets in the design. Therefore, always verify that the results of an electrical nets operation are expected by reading the messages in the Output Window.

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, on the **Electrical Net** tab, clear both the “Create electrical net” and “Allow electrical net creation by components” check boxes.

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

Conditions Governing Electrical Net Creation

Canceling Electrical Net Creation by Nets

Excluding Components from Electrical Net Creation

Excluding Components from Electrical Net Creation

You can prevent an individual component from creating electrical nets—a component through which an electrical net passes. 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, excluding it removes it from the electrical net.

Note:
This procedure clears 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 151.

Tip
Whenever you create, delete, or change electrical nets, SailWind Router regenerates all electrical nets in the design. Therefore, always verify that the results of an electrical nets operation are expected by reading the messages in the Output Window.

Procedure

1. In the workspace, select the components you want to exclude from creating 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 nets” check boxes.

Results

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.

Are the results different from what you expected?

- Nets attached to the excluded component may still be made into electrical nets through other components.

Related Topics

Conditions Governing Electrical Net Creation

Excluding Nets from Electrical Net Creation

Canceling Electrical Net Creation by Nets

You can cancel or end the “electrical net creation by the net” method.

Tip
Whenever you create, delete, or change electrical nets, SailWind Router regenerates all electrical nets in the design. Therefore, always verify that the results of an electrical nets operation are expected by reading the messages in the Output Window.

Procedure

1. In the workspace or Project Explorer, select the nets you want to remove from electrical nets.
2. Right-click and click the **Properties** popup menu item.
3. In the Electrical Nets tab, clear the Create electrical net check box.

   Tip
   If you want to exclude the nets from electrical net creation altogether, also clear the “Allow electrical net creation by components” check box.

Results

The selected nets are removed from existing electrical nets.

Are the results different from what you expected?

- **Mixed-method restriction:** Nets that are also made into electrical nets by the component and/or the refdes prefix method remain in electrical nets.

Related Topics

Conditions Governing Electrical Net Creation

Canceling Electrical Net Creation by Components

Cancel the creation of an electrical net that is created by the components it passes through.

Tip
Whenever you create, delete, or change electrical nets, SailWind Router regenerates all electrical nets in the design. Therefore, always verify that the results of an electrical nets operation are what you expected by reading the messages in the Output Window.

Procedure

1. In the workspace or Project Explorer, select the components whose electrical net creation you want to cancel.
2. Right-click and choose the **Properties** popup menu item.
3. In the Electrical Nets area, clear the “Create electrical net” check box.

   Tip
   If you want to exclude the components from electrical net creation by any method, also clear the “Allow electrical net creation by refdes prefix and by nets” check box.

Results

Electrical net creation by the selected component is canceled.

Are the results different from what you expected?

- **Mixed-method restriction:** Nets that conform to the following criteria remain in electrical nets throughout the selected components:
    - The nets are attached to selected components, and
    - The nets are also made into electrical nets by the net and/or the refdes prefix method

Selecting Electrical Nets

Use a popup menu item to select an electrical net. Selecting the electrical net also selects all physical nets in the electrical net.

Procedure

1. With nothing selected, right-click and click the **Select Nets** popup menu item.
2. In the workspace, select one of the nets in the electrical net you want to select.

3. Right-click and click the **Select Electrical Net** popup menu item.

> **Tip**
> You can also select electrical nets in the Project Explorer.

Setting Electrical Net Properties

Use the Electrical Net Properties dialog box to set or modify electrical nets properties.

Procedure

1. Select the electrical nets whose properties you want to set.
2. Right-click and click the **Properties** popup menu item.
3. In the Electrical Net Properties dialog box, make the appropriate settings in the Electrical Net on page 411 and Length on page 413 tabs, and click **OK**.

Creating a Matched Length Group of Electrical Nets

You can group electrical nets together to create a matched length group.

Procedure

1. Select the electrical nets on page 149.

> **Tip**
> You can also select nets to add to the group.

2. Right-click and click the **Make Matched Length Net Group** popup menu item.
3. As an alternative, in the Project Explorer, select the electrical nets and copy/paste or drag and drop them into a matched length net group, or into the Matched Length Net Groups group.

Creating a Differential Pair of Electrical Nets

You can select electrical nets and create a differential pair.

Procedure

1. Select the electrical nets on page 149.
2. Right-click and click the **Make Differential Electrical Net** popup menu item.
3. As an alternative, in the Project Explorer, select the electrical nets and copy/paste or drag and drop them into the Differential Pairs group.

Conditions Governing Electrical Net Creation

A number of conditions regulate the creation of electrical nets.

Electrical net creation is controlled by the following dialog box settings:

- The “Create electrical net” and “Allow electrical net creation by components” check boxes in the Net Properties dialog box.
- The “Create electrical net” and “Allow electrical net creation by refdes prefix or by nets” check boxes in the Component Properties dialog box.
- Refdes prefixes you specify in the Electrical Nets dialog box.
- The “Maximum net count per electrical net” and “Maximum non-plane-net pin count” fields in the Electrical Nets dialog box.

Tip
Virtual pins are not included in the net pin count.

The default settings for these check boxes are:

- The “Create electrical net” nets check boxes, which create electrical nets, are cleared by default for all nets and components. So by default, no electrical nets created.
- The “Allow electrical net creation...” check boxes, which allow the selected net or component to be added to an electrical net by other methods, are selected by default for all nets and components. So by default, you create electrical nets by any of the three methods.
- The “Maximum net count per electrical net” default value is 5.
- The “Maximum non-plane-net pin count” default value is 25.

A discrete component and its attached nets become part of an electrical net only if the following conditions 1 and 2 are true:

1. You set the component and net properties to meet one of the following conditions:
    - Component Properties “Create electrical net” is set and neither attached net is excluded from electrical net creation. (A component or net is excluded from electrical net creation if you clear its “Create electrical net” and “Allow electrical net creation...” check boxes.)
    - 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 you clear its “Create electrical net” and “Allow electrical net creation...” check boxes.)

   - Component Properties “Allow electrical net creation...” check box is selected and no attached net is excluded from electrical net creation. Also the component’s refdes prefix is specified in the Electrical Nets dialog box. (A component or net is excluded from electrical net creation if you clear its “Create electrical net” and “Allow electrical net creation...” check boxes.)

2. The component, its nets, and the potential electrical net to which it will belong, all conform to the General Restrictions and Prerequisites for creating electrical nets.

Chapter 11
Set Up Rules

Design rules allow you to exercise precise control over your routing results. A wide range of settings let you specify simple clearances between design objects or expand the rules to include complex relationships between nets, classes groups, pin pairs, and layers.

Setting Design (Global and Default) Properties

Use the Design Properties dialog box to view and modify the properties of your design when no object is selected. The Design Properties dialog box sets global rules properties.

Procedure

1. With nothing selected, click the **Properties** button or right-click and click the **Properties** popup menu item.
2. Click the design properties tab for the properties that you want to modify:
    - Clearance on page 468
    - Fanout on page 163
    - Grid on page 167
    - Layer Biasing on page 168
    - Layers on page 168
    - Pad Entry on page 504
    - Routing on page 189
    - Same net on page 519
    - Test Points on page 521
    - Topology on page 291
    - Via Biasing on page 169
3. Modify the properties on the tab as required.
4. Click another tab to continue setting properties.
5. When you are finished, click **OK**.

Related Topics

Setting Object Properties

Design Rules

Design rules allow you to assign constraints to your design objects. Assigning constraints allows you to control clearances between objects, assign vias to specific layers, control which layers will be used for routing, set up complex relationships between objects (or groups of objects), and define numerous other design constraints.

Hard and Soft Rules

There are two kinds of design rules: *hard* and *soft*. A hard rule is always followed, even if the result is an uncompleted route. A soft rule is ignored when it would result in an uncompleted route.

Most rules are permanently either hard or soft; a few rules are switchable—that is, they can be changed from hard to soft and back again.

Hard Rule Examples

- All clearance rules
- All via biasing rules

Soft Rule Examples

- Fanout rules
- Routing restrictions on component layers

Switchable Hard/Soft Rule Examples

- Length rules— Hardness/softness is controlled by the “Ignore length rules when required to complete traces” check box in the **Tune** tab of the Options dialog box.
- Same net first corner rules— Hardness/softness is controlled by the “Ignore first corner rules to complete traces when required” check box in the **Pad Entry** tab of the Design Properties dialog box.

Rules Hierarchy

Certain rules have priority over other rules. For example, a pin pair rule overrides a group rule, and a group rule overrides a net rule. This structure is known as the *rules hierarchy*.

Default rules, at level 1, have the lowest priority and represent the lowest level of the rules hierarchy. Electrical Net rules, at level 34, have the highest priority and represent the highest level of the rules hierarchy.

1. Default
2. Default with conditional layer rule
3. Class
4. Class with conditional layer rule
5. Net
6. Net with conditional layer rule

7. Group
8. Group with conditional layer rule
9. Pin Pair
10. Pin Pair with conditional layer rule
11. Class against Class
12. Class against Class with conditional layer rule
13. Net against Class
14. Net against Class with conditional layer rule
15. Net against Net
16. Net against Net with conditional layer rule
17. Group against Class
18. Group against Class with conditional layer rule
19. Group against Net
20. Group against Net with conditional layer rule
21. Group against Group
22. Group against Group with conditional layer rule
23. Pin Pair against Class
24. Pin Pair against Class with conditional layer rule
25. Pin Pair against Net
26. Pin Pair against Net with conditional layer rule
27. Pin Pair against Group
28. Pin Pair against Group with conditional layer rule
29. Pin Pair against Pin Pair

30. Pin Pair against Pin Pair with conditional layer rule

31. Decal

32. Component

33. Differential Pair

34. Electrical Net

Design Rules Passed from SailWind Layout

SailWind Layout passes most of the design rules in a design to SailWind Router. Upon completion of your routing session, SailWind Router passes the rules (and any modifications made to them) seamlessly back with the design to SailWind Layout.

With the exception of high-speed rules and rules on pin pairs of plane nets on page 159, SailWind Router uses the same design rules as SailWind Layout (Table 52).

Table 52. SailWind Layout Rules Verified and Maintained in SailWind Router

Rule		Maintained by: SailWind Layout	Maintained by: SailWind Router
Clearance	Clearance	Yes	Yes
	Body to body spacing	Yes	Yes **Note:** Converted to "Minimum spacing between components" value
	Same Net	Yes	Yes
	Trace Width	Yes	Yes
Routing	Length Minimization	Yes	Yes
	Routing Options	Yes	Yes
	Layer Biasing	Yes	Yes
	Via Biasing	Yes	Yes
High-Speed	Parallelism	Yes (EDC)	No
	Shielding	No	No
	High-speed	Yes (EDC)	No
	Length Matching	Yes (EDC)	Yes

Table 52. SailWind Layout Rules Verified and Maintained in SailWind Router (continued)

Rule		Maintained by: SailWind Layout	Maintained by: SailWind Router
Conditional	All combinations	Yes	Yes
Differential Pairs	All definitions	No	Yes
Test Point	Probe to Trace	Yes	Yes
	Probe to Pad	Yes	Yes
	Test Point Grid	Yes	Yes
	Assignment	Yes	Yes

SailWind Layout Design Rules vs. SailWind Router Properties

In SailWind Layout, you use Design Rules to assign constraints to your design objects. Such constraints include clearances, via assignments, layer assignments, and other complex design object relationships. SailWind Router supports most of these same rules; however, SailWind Router refers to them as *Properties*.

This table shows the location of specific Design Rules in SailWind Layout and the corresponding Properties location in SailWind Router.

Table 53. Design Rule Mapping between SailWind Layout and SailWind Router

SailWind Layout setting	SailWind Layout dialog box	SailWind Router setting	SailWind Router dialog box
Clearances	Default Rules, Clearance Rules	Clearances	Design Properties, **Clearance** tab
	Default Rules, Clearance Rules: Body to body	Minimum spacing between components	Options, Fabrication category
Same Net Clearance	Default Rules, Clearance Rules	Same Net Clearance	Design Properties, **Same Net** tab
Trace Width	Default Rules, Clearance Rules	Trace Width	Design Properties, **Routing** tab
Topology	Default Rules, Routing Rules	Topology	Design Properties, **Topology** tab
Routing Options	Default Rules, Routing Rules	Route Options	Design Properties, **Routing** tab
Layer Biasing	Default Rules, Routing Rules	Layer Biasing	Design Properties, **Layer Biasing** tab

Table 53. Design Rule Mapping between SailWind Layout and SailWind Router (continued)

SailWind Layout setting	SailWind Layout dialog box	SailWind Router setting	SailWind Router dialog box
Vias	Default Rules, Routing Rules	Via Biasing	Design Properties, **Via Biasing** tab
Maximum Number of Vias	Default Rules, Routing Rules	Maximum Number of Vias	Design Properties, **Routing** tab
Test Point Audit	DFT Audit, Properties	Test Points	Design Properties, **Test Points** tab
Test Point Assignment	DFT Audit, **Assignment** tab	Test Point Assignment	Component Properties, **Test Points** tab Project Explorer, Via Type Properties, **Test Points** tab
No Equivalent		Layers	Design Properties, **Layers** tab
Grid	Grids and Snap Options, **Grids** tab	Grid	Design Properties, **Grid** tab

Pin Pairs and Plane Nets

SailWind Router does not allow Pin Pair rules or Pin Pair Group rules defined for Plane nets. These rules affect SailWind Router performance as they present a technically complex scenario.

If plane net pin pairs are assigned in SailWind Layout they are removed after the design is opened in SailWind Router. You cannot assign these rules in SailWind Router. In either case the following messages appear in the Output window:

- **For plane net pin pair rules** — “Pin Pair rules cannot be assigned in plane nets.”
- **For plane net pin pairs in a group** — “Pin Pairs cannot be included to Group in plane nets.”

For non-plane nets, the limit can be controlled in the *SailWindRouter.ini* file by the following:

```
[General]
NonPowerNetMaxPinCount=50
```

With this setting, if a net has more than 50 pins, SailWind Router considers it to be a large net and does not support pin pair rules for it.

Same Net Rules

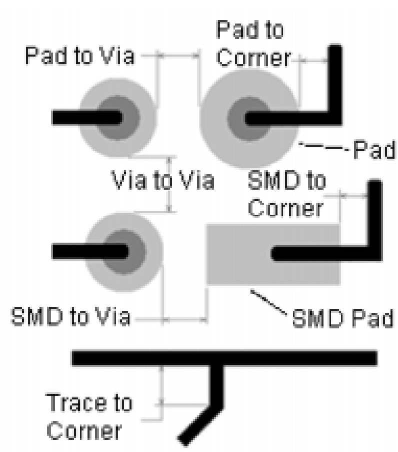
Same net rules specify the minimum spacing between items in a net.

Some same net rules are permanently hard; you can make others hard or soft, using the "Ignore first corner rules to complete traces when required" check box in the **Pad Entry** tab of the Design Properties dialog box. Table 54 and Figure 16 show the different rule types for items in the same net.

Table 54. Edge-to-edge Clearance Values that Apply to Items in the Same Net

Clearance rule	Rule type	Description
Pad-to-Corner	Switchable hard/soft	Minimum spacing between a non-SMD pad and the first trace bend point.
SMD-to-Corner	Switchable hard/soft	Minimum spacing between a surface mount pad and the first trace bend point.
SMD-to-Via	Hard	Minimum spacing between a surface mount pad and a via.
Trace-to-Corner	Switchable hard/soft	Minimum spacing between a segment and the bend point of an adjacent trace segment; for example, when a trace splits at a T-junction and one of the two traces has a bend point. Trace-to-corner also serves as a trace-to-trace rule. The trace-to-corner rule is used for routing only and is not checked by design verification.
Via-to-Via	Hard	Minimum spacing between vias.

Figure 16. Same Net Rules



First Corner Rules

First corner rules determine how clearances are enforced on traces as they exit a pad or via.

There are three first corner rules:

- Pad-to-corner(for through pins and vias),
- Trace-to-corner,
- SMD-to-corner

Pad-to-corner and SMD-to-corner rules prevent corners from being added to traces that enter or exit a pad at less than the minimum corner rules. The pad-to-corner or SMD-to-corner rules also prevent the trace from getting too close to the pad after the first corner is added. Figures Figure 17 and Figure 18 show the first corner rules in SailWind Layout and SailWind Router.

Figure 17. First Corner Rule in SailWind Layout

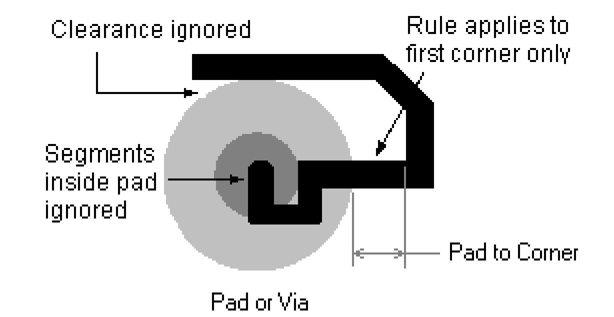
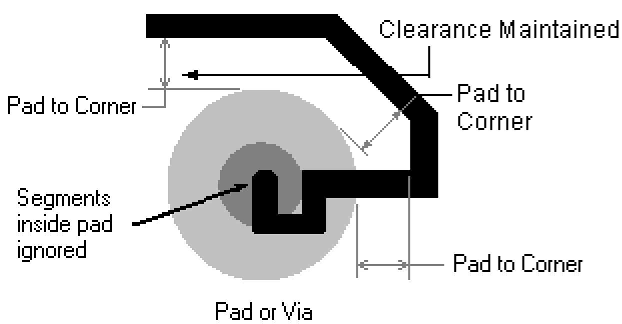


Figure 18. First Corner Rule in SailWind Router



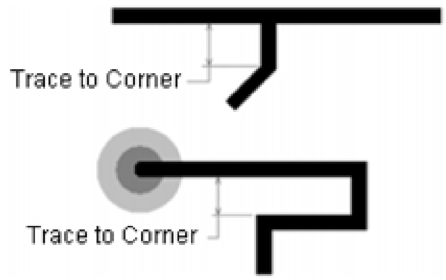
Trace-to-Corner Rule

The trace-to-corner rule is similar to the pad-to-corner rule and also serves as a trace-to-trace rule. During autorouting, SailWind Router violates this rule only to avoid an incomplete trace. You can repair such violations during Smooth or Optimize passes.

The software uses the trace-to-corner rule (Figure 19) for routing only; it does not check this rule during design verification.

Tip
For best results, include an Optimize pass each time you autoroute.

Figure 19. Trace-to-Corner Rule

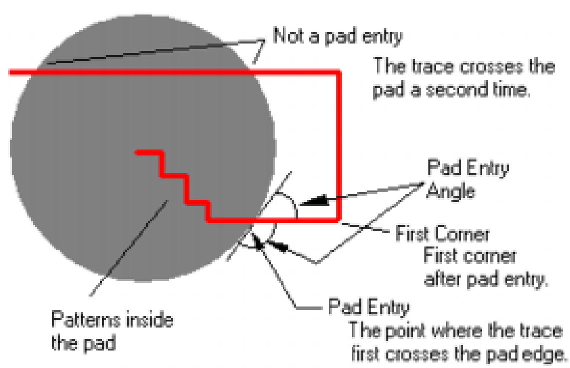


Pad Entry Rules

Pad entry is the point at which a trace entering or exiting a pin first crosses the edge of a pad. You can define the rules for pad entry in the design properties setup.

The quality of a pad entry is measured by the angle of the pad entry. A perfect pad entry has pad angles equal to, or greater than, 90 degrees. Figure 20 shows an unsatisfactory pad entry; the lower pad entry angle is greater than 90 degrees, but the upper pad entry angle is less than 90 degrees.

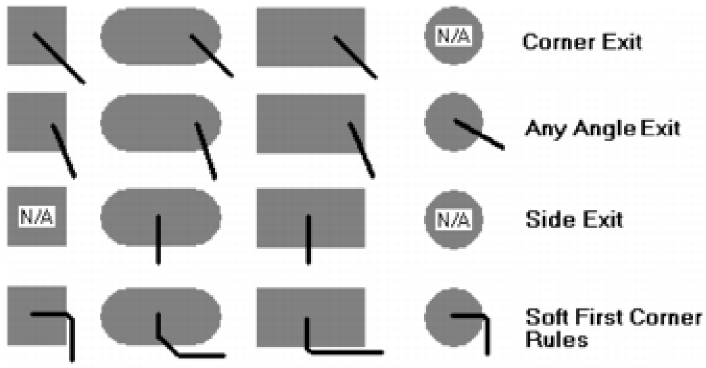
Figure 20. Unsatisfactory Pad Entry



Perfect pad entry is attempted on all traces entering pins and vias regardless of the pad shape. The pad entry controls let you choose pad exit types (any angle, side, or corner) and soft corner rules. For more information, see "Hard and Soft Rules" on page 155.

SailWind Router uses different pad entries as required to complete traces (Figure 21). The software achieves best results when all pad entry types, including soft first corner rules, are permitted. The software makes poor or unsatisfactory pad entries only for the sake of trace completion.

Figure 21. Pad Entry Types



To maintain pin pair rules, SailWind Router does not share a pad entry or exit on nets with pin pair rules.

Topology Restrictions

Topology rules allow you to specify the net topology type and whether you want to allow multiple connections to vias and traces in your design.

Set topology rules from the **Topology** tab of the properties setup dialog box. The following restrictions apply:

- The “Allow junctions on” settings are unavailable for all topology types except Minimized.
- The “Maximum stub length” setting is unavailable for the Minimized topology type.

Fanout Rules

Fanout rules specify the fanout type, via placement, and the multiple connection option for pins, SMDs, vias, and traces. You can give fanouts unlimited length, or you can specify a maximum length. Fanout rules are soft rules, meaning SailWind Router ignores them in circumstances where following them would result in one or more uncompleted routes.

Decal Rules

Decal rules specify routing, pad entry, same net, fanout, via biasing, and clearance rules for all components that use a particular decal.

Error Rules

Error rules specify whether to ignore an error during verification operations.

Clearance Rules

Clearance rules specify the minimum allowable air gap between object types in the design.

Object types supported include:

- Traces
- Vias
- Through-hole pads
- Surface mount (SMD) pads
- Copper
- Drill holes
- Text
- Board

Other Clearance Rules

Additional clearances rules maintain clearances between component bodies and drill holes.

Table 55 summarizes other clearance rules.

Table 55. Other Clearance Rules Included

Rule	Description
Minimum spacing between components	The distance between the centerlines of the lines used to define the placement outline of each component. This includes auto-generated placement outlines and outlines derived from SailWind Layout Layer 20 placement outlines. **Note:** When a design is opened, this value populates with the “Body to body” Clearance Design Rule value from SailWind Layout.
Minimum spacing between holes	Minimum edge-to-edge spacing between two drill holes

The rules in Table 55 appear on the **Fabrication** tab of the Options dialog box.

Setting Clearances

View and modify minimum spacing clearances using the **Clearance** tab of the properties setup dialog box.

If you selected an object, you are setting the clearance properties for that object. If you selected nothing, you are setting clearance properties for the entire design.

Procedure

1. Right-click and click the **Properties** menu item; then click the **Clearance** tab.
2. Type the clearance value in the box for the clearance to allow between the object listed in the row heading and the object listed in the column heading.

   Tip
   To set the same value for an entire row or column, click on a column heading, row heading, or **All**. Type a value and click OK to apply the value.

3. Click **OK**.

Related Topics

Setting Design (Global and Default) Properties

Properties Dialog Box, Same Net Tab

Creating Conditional Rules

Grid Rules

SailWind Router provides you with a number of different grid types to assist you during the placement, routing, and test point stages of your design.

SailWind Router uses the following five routing grids:

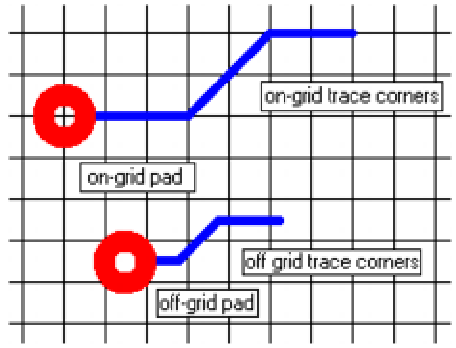
- Test point
- Via
- Fanout
- Component
- Routing

SailWind Router has two additional grids, the Displayed and Copper Hatch grids, which it does not use during routing.

Set the routing grid using Snap to Grid on the **Grid** tab of the Design Properties dialog box. If you clear the Snap to Grid check box, grid settings are ignored.

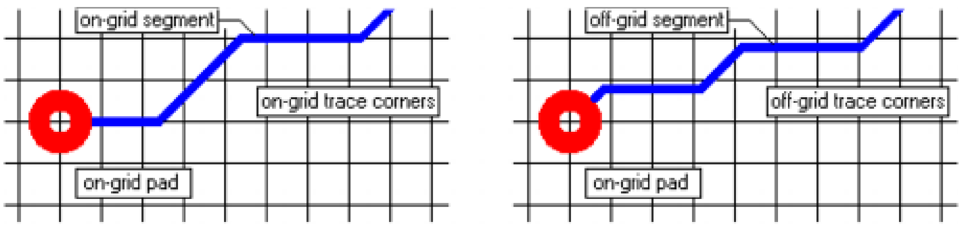
The routing grid establishes the default routing grid for routing tasks other than components, test points, vias, and vias added during fanout operations (Figure 22).

Figure 22. Default Routing Grid



A trace corner is on-grid if at least one of its coordinates is on the grid. A trace segment is on-grid if both of its end points are on the grid in the same plane. Figure 23 shows the routing grid for trace corners and trace segments.

Figure 23. Routing Grid for Traces



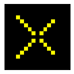
The rule to maintain on-grid trace corners is soft. If a trace corner cannot be placed on-grid, the corner is placed off-grid to complete the trace.

Grid Statistics Report

You can create a grid statistics report in the Output window to report on pins, vias, and trace segments that are on or off the grids.

You must select the Snap Objects to Grid check box for the Routing on page 491grid before you create the grid statistics report. This is required to generate the Off-grid statistics in the report and to show the markers in the design on the locations of all the pins, vias, or trace segment endpoints that are off-grid according to the Hard/Soft rules. Markers disappear when the screen is redrawn.

Figure 24. Grid Statistics Off-Grid Marker



The Grid Statistics reports are run using the following Shortcut (Modeless) commands.

- **GGS**— Soft grid statistics: Trace segments are considered on-grid if one of the endpoints has one of either the X and Y coordinates on the Routing grid.

- **GGM**— Medium grid statistics.

- **GGH**— Hard grid statistics: Trace segments are considered on-grid if one of the endpoints has both X and Y coordinates on the Routing grid.

Tip
The Snap value in the report always displays “Off” when the grid is set to 00000_00000 regardless of the Snap Objects to Grid setting.

Sample Report

This is a sample of a Soft grid statistics report.

```
______Total_____On grid___Off grid___Snap_____Soft_Grid
Pin___00206__|___00030__|__00176___|__On___|_00050_00050
Via___00072__|___00000__|__00072___|__Off__|_00000_00000
Trace_00587__|___00151__|__00436___|__On___|_00050_00050
```

Setting Grids

Use the **Grid** tab to view and modify all grid settings in your design.

Set a grid to restrict placement of the routing objects to specific increments. The following table lists objects and their grid placement.

Table 56. Objects and Grid Placement

Object	Is placed on this grid
Trace corners	Routing
Vias, via pairs	Via
Test points	Test Points

A trace segment is considered “on grid” if at least one of its coordinates (X or Y) is on the grid. You do not have to place both coordinates on the grid.

If you select the Snap Objects to Grid box for a grid, then pointer movements snap to the grid. This prevents you from placing objects between grid points. For more information, see “Grid Rules” on page 165.

Procedure

1. Open the Design Properties grid by right-clicking in the workspace (with nothing selected) and choosing the **Properties** popup menu item.
2. Click the **Grid** tab. In the grid settings table, type the value in each box to which you want to set the grid line spacing.
3. Select the “Snap Objects to Grid” check box to enable snap. Clear the check box to disable snap.
4. Click **OK**.

Related Topics

Setting Design (Global and Default) Properties

Design Rules

Setting Layer Biasing Rules

Layer biasing rules specify whether SailWind Router can use a particular layer while routing.

Use the **Layer Biasing** tab to restrict layers available for nets and pin pairs to certain layers for routing. Layer biasing provides an additional level of control over routing on layers.

For example: You want to route power and ground nets only on outer layers, and force the routing of other nets on internal layers. To accomplish this, with nothing selected, enable outer layers for routing in the general design properties, but disable outer layers in the **Layer Biasing** tab. Then, in the Net Properties dialog box for the power and ground nets, enable the outer layers for routing in the **Layer Biasing** tab.

If you selected an object, you are setting layer biasing properties for that object. If you selected nothing, you are setting layer biasing properties for the entire design.

Procedure

1. Right-click and click the **Properties** popup menu item; then, in the properties dialog box, click the **Layer Biasing** tab.
2. Select the Allow Routing check box to enable routing on a layer. Clear the check box to disable it.
3. Click **OK**.

Related Topics

Setting Design (Global and Default) Properties

Setting Layer Rules

Layer rules specify routing parameters for the specified layer. Only electrical layers display.

Use the **Layers** tab to set routing parameters for a specific layer.

Procedure

1. With nothing selected in the workspace, right-click and click the **Properties** popup menu item; then, in the properties dialog box, click the **Layers** tab.
2. In the “Routing parameters for layers” area, select the Route check box to enable a layer for routing. Clear the check box to disable it.
3. Click in the Direction column for the layer and select the routing direction you want from the list.
4. Click in the Cost column for the layer and type a number 0 and 100 for the cost value.

   The cost value specifies the level of usage for the layer during routing. A higher number specifies *less* use of the layer.

5. Click **OK**.

Restriction:
The Type column, which displays the layer type, can only be changed in SailWind Layout.

Related Topics

Setting Design (Global and Default) Properties

Setting the Via Types to Use on Specific Layers

Via biasing rules specify whether SailWind Router can use a particular via type while routing.

Use the **Via Biasing** tab of the Properties dialog box to restrict the via types that SailWind Router uses while routing.

Restrictions and Limitations

- You cannot change the size of vias or add additional vias in SailWind Router. To change the size of a via or create new vias, open the design in SailWind Layout and use the Pad Stacks Properties dialog box in SailWind Layout.

If you selected an object, you are setting via biasing properties for that object. If you selected nothing, you are setting via biasing properties for the entire design.

Procedure

1. Right-click and click the **Properties** popup menu item; then, in Properties dialog box, click the **Via Biasing** tab.
2. Select the Allow check box to enable SailWind Router to use the via for routing. Clear the check box to disable use of the via.
3. Click **OK**.

Related Topics

Setting Design (Global and Default) Properties

Setting Object Properties

Object properties allow you to specify design rules for all of the objects in your design. You can also construct complex relationships between groups of objects to accommodate your most demanding design constraints.

Tip
The Changed Properties icon appears on the tab if the design properties differ from the properties of the selected object. The differing fields appear with a yellow background.

Procedure

1. Select an object then right-click and click the **Properties** popup menu item.

   The properties that appear in the resulting dialog box depend on the selected object. Properties set in this way affect the selected object only and not global settings. Table 57 lists the available tabs for each object type.

Table 57. Property Dialog Box — Available Tabs for Object Types

Object	Available tabs
Component	• Clearance on page 468 • Component on page 469 • Fanout on page 486 • Pad Entry on page 504 • Routing on page 517 • Same Net on page 519 • Test Points on page 523 • Via Biasing on page 532
Conditional rule	• Clearance on page 468
Copper Plane	• Copper Plane on page 475 • Flooding on page 489
Part decal	• Clearance on page 468 • Component on page 469 • Fanout on page 486 • Pad Entry on page 504 • Routing on page 517 • Same Net on page 519 • Via Biasing on page 532
Differential pair	• "Pair" on page 184
Error	• Error on page 366
Keepout	• Keepout on page 495
Matched length group	• Group on page 493

Object	Available tabs
Net	• Clearance on page 468 • Layer Biasing on page 497 • Length on page 501 • Net on page 502 • Routing on page 517 • Same Net on page 519 • Test Points on page 521 • Topology on page 524 • Via Biasing on page 532
Net class	• Clearance on page 468 • Layer Biasing on page 497 • Length on page 501 • Routing on page 517 • Same Net on page 519 • Topology on page 524 • Via Biasing on page 532
Pin	• Component on page 469 • Drill on page 480 • Pad Stack on page 506 • Pin on page 513
Pin pair	• Clearance on page 468 • Layer Biasing on page 497 • Length on page 501 • Routing on page 517 • Via Biasing on page 532
Pin pair group	• Clearance on page 468 • Layer Biasing on page 497 • Length on page 501 • Routing on page 517 • Via Biasing on page 532
Text	• No properties on page 169
Trace	• Trace on page 526
Trace corner	• Trace Corner on page 479

Object	Available tabs
Trace segment (path, or partial selection)	• Trace Segment on page 528
Unroute (connection)	• No properties on page 169
Via	• Via on page 530 • Pad Stack on page 508 • Drill on page 482
Via type	• Pad Stack on page 508 • Drill on page 482 • Test Points on page 523

Some selected objects may not have properties. For example, an unroute. In this case the message "No properties available" appears. If multiple object types are selected, the message "Multiple selection" appears.

2. Click a tab and make modifications as appropriate.

3. To clear the current rules for the selected object, click the **Delete Level** button. The next rules level in the hierarchy is applied to the object. The object's hierarchy level appears in the Rule level box.

Tip
The **Delete Level** button is unavailable if design rules are used.

4. Click **OK**.

Related Topics

Setting Design (Global and Default) Properties

Working with Multiple Object Properties

Set Up Rules

Creating Net Classes

Create net classes and assign Properties (Rules) by selecting nets in the Project Explorer and/or in the design workspace.

Video

Procedure

1. Use one of the following methods, or a combination, to select the nets for the net class:

    - **Using the Project Explorer**— Use the list of net names in the Project Explorer to select the nets.

      Tip
      You might find it useful to split the Project Explorer on page 52 for tasks like this.

        i. In the Project Explorer, expand the Net Objects list and then expand the Nets list.

        ii. In the list of nets, select one or more nets to add to the net class. You can add more nets to the class later or delete nets from the class if needed.

        iii. Right-click and click Copy, or drag your selection from the list of nets and then right-click and click Paste, or drop them on the Net Classes list.

    - **Using the Workspace**— Use the pointer to select nets in your design.

        i. In the design, select one or more nets to add to the net class. You can add more nets to the class later or delete nets from the class if needed.

        ii. Right-click and click the **Make Class** popup menu item.

2. In the Project Explorer, expand the Net Objects list and then expand the Net Classes list. The Net Classes list expands to reveal a new Class<n>, for example, Class1.

3. Rename the class if desired to make the name more significant.

    a. Click to select the new class.

    b. Click the text name of the selected class. The text is selected for renaming.

    c. Rename the class and press the Enter key.

4. Right-click in the Project Explorer and make sure the Allow Selection setting is checked. Selecting the class in the Project Explorer also needs to simultaneously select the nets in the design area in order to open the Properties of the new class.

5. Place the pointer over the new class name, right-click and click the **Properties** popup menu item.
6. In the Net Class Properties dialog box, set unique properties for the class. For each tab in the dialog box, you can click the **Help** button to get details for each setting.

Creating Pin Pair Groups

Create groups of pin pairs and assign Properties (Rules) by selecting pin pairs in the Project Explorer and/or in the design workspace.

Restrictions and Limitations

Plane nets cannot be added to Pin Pair Groups. For more information, see "Pin Pairs and Plane Nets."

Procedure

1. Use one of the following methods, or a combination, to select the pin pairs for the pin pair group:

   - **Using the Project Explorer**— Use the lists of pin pairs of nets in the Project Explorer to select the pin pairs.

     Tip
     You might find it useful to split the Project Explorer on page 52 for tasks like this.

     i. In the Project Explorer, expand the Net Objects list and then expand the Nets list. You must expand an individual net to view the pin pairs of the net.

     ii. Select one or more pin pairs to add to the pin pair group. You can add more pin pairs to the group later or delete pin pairs from the group if needed.

     iii. Right-click and click the **Copy** popup menu item, or drag your selection from the lists of pin pairs and then right-click and click the **Paste** popup menu item. You can also drop them on the Pin Pair Groups list.

   - **Using the Workspace**— Use the pointer to select pin pairs in your design.

     i. In the design, select one or more pin pairs to add to the pin pair group. You can add more pin pairs to the group later or delete pin pairs from the group if needed.

     ii. Right-click and click the **Make Pin Pair Group** popup menu item.

2. In the Project Explorer, expand the Net Objects list and then expand the Pin Pair Groups list. Pin Pair Groups expands to reveal a new Group<n>, for example, Group1.
3. Rename the group if desired to make the name more significant.

   a. Click to select the new group.

   b. Click the text name of the selected group. The text is selected for renaming.

   c. Rename the group and press Enter.

4. Right-click in the Project Explorer and make sure the Allow Selection setting is checked. Selecting the group in the Project Explorer also needs to select the pin pairs in the design area in order to open the Properties of the new group.
5. Place the pointer over the new group name, right-click and click the **Properties** popup menu item.
6. In the Pin Pair Group Properties dialog box, set unique properties for the group. For each tab in the dialog box, you can click the **Help** button to get details for each setting.

Conditional Rules

Conditional rules extend basic rule assignments and provide more control over complex designs. For example, you could set a conditional rule for a net, which comes into effect only if the net is adjacent to a specific net. Conditional rules have some additional restrictions, which make them different from other rules.

Conditional rule names are not user-definable. An automatic naming process is used for conditional rules. For more information, see Table 59 on page 178.

You must specify a set of objects before you can create a conditional rule.

The following lists the conditional rules that you can create and describes how to create them.

- **Default clearance and trace width rules on a specific layer** — Copy or drag the specific electrical layer and paste or drop it on Conditional Rules in the Project Explorer.
- **Class clearance and trace width rules on a specific layer** — Copy or drag the net class and specific electrical layer and paste or drop them on Conditional Rules in the Project Explorer.
- **Net clearance and trace width rules on a specific layer** — Copy or drag the net and specific electrical layer and paste or drop them on Conditional Rules in the Project Explorer.
- **Group clearance and trace width rules on a specific layer** — Copy or drag the pin pair group and specific electrical layer and paste or drop them on Conditional Rules in the Project Explorer.
- **Pin Pair clearance and trace width rules on a specific layer** — Copy or drag the pin pair and specific electrical layer and paste or drop them on Conditional Rules in the Project Explorer.
- **Class against Class rules** — Copy or drag the two net classes and paste or drop them on Conditional Rules in the Project Explorer.
- **Class against Class rules on a specific layer** — Copy or drag the two net classes and specific electrical layer and paste or drop them on Conditional Rules in the Project Explorer.
- **Net against Class rules** — Copy or drag the net and net class and paste or drop them on Conditional Rules in the Project Explorer.
- **Net against Class rules on a specific layer** — Copy or drag the net, net class, and specific electrical layer and paste or drop them on Conditional Rules in the Project Explorer.
- **Net against Net rules** — Copy or drag the two nets and paste or drop them on Conditional Rules in the Project Explorer.

- **Net against Net rules on a specific layer** — Copy or drag the two nets, and specific electrical layer and paste or drop them on Conditional Rules in the Project Explorer.
- **Group against Class rules** — Copy or drag the pin pair group and net class and paste or drop them on Conditional Rules in the Project Explorer.
- **Group against Class rules on a specific layer** — Copy or drag the pin pair group, net class, and specific electrical layer, and paste or drop them on Conditional Rules in the Project Explorer.
- **Group against Net rules** — Copy or drag the pin pair group and net and paste or drop them on Conditional Rules in the Project Explorer.
- **Group against Net rules on a specific layer** — Copy or drag the pin pair group, net, and specific electrical layer, and paste or drop them on Conditional Rules in the Project Explorer.
- **Group against Group rules** — Copy or drag the two pin pair groups and paste or drop them on Conditional Rules in the Project Explorer.
- **Group against Group rules on a specific layer** — Copy or drag the two pin pair groups, and specific electrical layer, and paste or drop them on Conditional Rules in the Project Explorer.
- **Pin Pair against Class rules** — Copy or drag the pin pair and net class and paste or drop them on Conditional Rules in the Project Explorer.
- **Pin Pair against Class rules on a specific layer** — Copy or drag the pin pair, net class, and specific electrical layer, and paste or drop them on Conditional Rules in the Project Explorer.
- **Pin Pair against Net rules** — Copy or drag the pin pair and net and paste or drop them on Conditional Rules in the Project Explorer.
- **Pin Pair against Net rules on a specific layer** — Copy or drag the pin pair, net, and specific electrical layer, and paste or drop them on Conditional Rules in the Project Explorer.
- **Pin Pair against Group rules** — Copy or drag the pin pair and pin pair group and paste or drop them on Conditional Rules in the Project Explorer.
- **Pin Pair against Group rules on a specific layer** — Copy or drag the pin pair, pin pair group, and specific electrical layer and paste or drop them on Conditional Rules in the Project Explorer.
- **Pin Pair against Pin Pair rules** — Copy or drag the two pin pairs and paste or drop them on Conditional Rules in the Project Explorer.
- **Pin Pair against Pin Pair rules on a specific layer** — Copy or drag the two pin pairs, and specific electrical layer and paste or drop them on Conditional Rules in the Project Explorer.

Conditional Rules Objects

You can create conditional rules using drag and drop or copy and paste operations in the project explorer. You must first select the objects that you want included in the conditional rule.

Table 58. Objects You Can Add to the Conditional Rule Group to Create a New Rule

First Object	Second Object	Third Object	Resulting Rule	Automatic Naming Example
Layer			All: All (Layer)	All: All (Top)
Layer	Class		Class: All (Layer)	CLS1: All (Bottom)
Layer	Net		Net: All (Layer)	+5V: All (Bottom)
Layer	Group		Group: All (Layer)	GRP1: All (Bottom)
Layer	Pin Pair		Pin Pair: All (Layer)	R1.1-U1.2: All (Bottom)
Class	Class		Class: Class (All Layers)	CLS1: CLS2 (All Layers)
Class	Class	Layer	Class: Class (Layer)	CLS1: CLS2 (Top)
Class	Net		Class: Net (All Layers)	CLS1: +5V (All Layers)
Class	Net	Layer	Class: Net (Layer)	CLS1: +5V (Top)
Class	Group		Class: Group (All Layers)	CLS1: GRP1 (All Layers)
Class	Group	Layer	Class: Group (Layer)	CLS1: GRP1 (Top)
Class	Pin Pair		Class: Pin Pair (All Layers)	CLS1: R1.1-U1.1 (All Layers)
Class	Pin Pair	Layer	Class: Pin Pair (All Layers)	CLS1: R1.1-U1.1 (Top)
Net	Net		Net: (All Layers)	GND: +5V (All Layers)
Net	Net	Layer	Net: (Layer)	GND: +5V (Top)
Net	Group		Net: Group (All Layers)	GND: GRP1 (All Layers)
Net	Group	Layer	Net: Group (Layer)	GND: GRP1 (Top)
Net	Pin Pair		Net: Pin Pair (All Layers)	GND: R1.1-U1.1 (All Layers)
Net	Pin Pair	Layer	Net: Pin Pair (All Layers)	GND: R1.1-U1.1 (Top)
Group	Group		Group: Group (All Layers)	GND: GRP1 (All Layers)
Group	Group	Layer	Group: Group (Layer)	GND: GRP1 (Top)
Group	Pin Pair		Group: Pin Pair (All Layers)	GRP1: R1.1-U1.1 (All Layers)
Group	Pin Pair	Layer	Group: Pin Pair (All Layers)	GRP1: R1.1-U1.1 (Top)
Pin Pair	Pin Pair		Pin Pair: Pin Pair (All Layers)	R2.1-R3.1: R1.1-U1.1 (All Layers)
Pin Pair	Pin Pair	Layer	Pin Pair: Pin Pair (All Layers)	R2.1-R3.1: R1.1-U1.1 (Top)

Conditional Rules Automatic Naming

Because you cannot define names for conditional rules, automatic naming differentiates one rule from another.

As shown in Table 59, automatic names consist of the names of objects in the rule and the name of the layer for which the rule applies. Automatic names appear both in the **Object View** tab and in the captions of the Properties dialog box for the conditional rule.

Table 59. Automatic Conditional Rule Naming

Rule	Automatic Name
Net +5V against Net GND on Layer Top	+5V : GND (Top)
All Objects against All Objects on Layer Bottom	All : All (Bottom)
Net Class GROUND against Pin Pair R1.1 - U10.2 on all layers	GROUND : R1.1-U10.2 (All Layers)

Conditional Rule Objects Modification

You cannot modify the objects assigned to a conditional rule. You must delete conditional rules completely, and then recreate the conditional rule for the new set of objects.

Creating Conditional Rules

Create conditional groups Properties (Rules) by selecting items in the Project Explorer and copying them to Conditional Rules.

Procedure

1. In the Project Explorer, select the items you need to make up the conditional rule. For specific information, see “Conditional Rules” on page 175.

   Tip
   You might find it useful to split the Project Explorer on page 52 for tasks like this.

2. Right-click and click copy, or drag your selection and then right-click and click Paste, or drop it on **Conditional Rules**. You cannot change the objects assigned to a conditional rule; you must delete the conditional rule, and then recreate the conditional rule for the new set of objects.

3. In the Project Explorer, expand **Net Objects** and then expand **Conditional Rules**.

4. Conditional Rules expands to reveal a new condition in the format of *Source:Against (Layer)*, for example, +12V:+5V (Top Layer).

5. Place the pointer over the new conditional rule, right-click and click the **Properties** popup menu item.

6. In the Conditional rule dialog box, set unique properties for the condition.

Related Topics

Setting Clearances

Creating Matched Length Net Groups

Create matched length net groups and assign Properties (Rules) by selecting nets in the Project Explorer and/or in the design workspace.

Procedure

1. Use one of the following methods, or a combination, to select the nets for the matched length net group:

   - **Using the Project Explorer**— Select from the list of nets in the Project Explorer.

     > **Tip**
     > You might find it useful to split the Project Explorer on page 52 for this task.

     i. In the Project Explorer, expand the Net Objects list and then expand the Nets list.

     ii. In the list of nets, select one or more nets to add to the matched length net group. You can add more nets to the group later or delete nets from it if needed.

     iii. Right-click and click the **Copy** popup menu item, or drag your selection from the list of nets and then right-click and click the **Paste** popup menu item, or drop them onto the Matched Length Net Groups list.

   - **Using the Workspace**— Use the pointer to select nets in your design.

     i. In the design, select one or more nets to add to the matched length net group. You can add more nets to the group later or delete nets from the group if needed.

     ii. Right-click and click the **Make Matched Length Net Groups** popup menu item.

2. In the Project Explorer, expand the Net Objects list and then expand the Matched Length Net Groups list. Matched Length Net Groups expands to reveal a new MLNetGroup<n>, for example, MLNetGroup1.

3. Rename the group if desired to make the name more significant.

   a. Click to select the new group.

   b. Click the text name of the selected group. The text is selected for renaming.

   c. Rename the group and press Enter.

4. Right-click in the Project Explorer and make sure the Allow Selection setting is checked. Selecting the group in the Project Explorer also needs to select the nets in the design area to open the properties of the new group.

5. Place the pointer over the new group name, right-click and click the **Properties** popup menu item.
6. In the Matched Length Group Properties dialog box, set unique properties for the group:
    a. Type a value in the Tolerance box to set the difference between the shortest member and longest member of the group.
    b. Select the "Restrict length" check box to restrict the length of the group. Unlike SailWind Layout, this program counts overlapping trace segments in the trace length calculation.
        i. Type the minimum value in the Minimum length box.
        ii. Type the maximum value in the Maximum length box.
7. Click **OK**.

Related Topics

Monitoring Trace Length

Creating Matched Length Pin Pair Groups

Create matched length pin pair groups and assign Properties (Rules) by selecting pin pairs in the Project Explorer and/or in the design workspace.

Procedure

1. Use one of the following methods, or a combination, to select the pin pairs for the matched length pin pair group:
    - **Using the Project Explorer**—Use the lists of pin pairs of nets in the Project Explorer to select the pin pairs.

    > **Tip**
    > You might find it useful to split the Project Explorer on page 52 for tasks like this.

        i. In the Project Explorer, expand the Net Objects list and then expand the Nets list. You must expand an individual net to view the pin pairs of the net.
        ii. Select one or more pin pairs to add to the matched length pin pair group. You can add more pin pairs to the group later or delete pin pairs from the group if needed.
        iii. Right-click and click **Copy**, or drag your selection from the lists of pin pairs and then right-click and click **Paste**, or drop them onto the Matched Length Pin Pair Groups list.
    - **Using the Workspace**—Use the pointer to select pin pairs in your design.

i. In the design, select one or more pin pairs to add to the matched length pin pair group. You can add more pin pairs to the group later or delete pin pairs from the group if needed.

ii. Right-click and click **Make Matched Length Pin Pair Group**.

2. In the Project Explorer, expand the Net Objects list and then expand the Matched Length Pin Pair Groups list. Matched Length Pin Pair Groups expands to reveal a new MLPinPairGroup<n>, for example, MLPinPairGroup1.
3. Rename the group if desired to make the name more significant.
   a. Click to select the new group.
   b. Click the text name of the selected group. The text is selected for renaming.
   c. Rename the group and press Enter.
4. Right-click in the Project Explorer and make sure the Allow Selection setting is checked. Selecting a pin pair group in the Project Explorer must also select the pin pairs in the design area to open the properties of the new group.
5. Place the pointer over the new group name, right-click and click **Properties**.
6. In the Matched Length Group Properties dialog box, set unique properties for the group.
   a. To set the difference between the shortest member and longest member of the group, type a value in the Tolerance box.
   b. Select the "Restrict length" check box to restrict the length of the group. Unlike SailWind Layout, this program counts overlapping trace segments in the trace length calculation.

      i. Type the minimum value in the Minimum length box.

      ii. Type the maximum value in the Maximum length box.
7. Click **OK**.

Related Topics

Monitoring Trace Length

Differential Pairs Rules

Differential pair rules assign length, gap parameters, and obstacle handling to selected pairs of nets, electrical nets, or pin pairs. When setting up your design rules, you can pair two nets or two pin pairs and define rules related to the pair. If desired, you can use differential pair rules to also restrict a layer change.

Creating Differential Pairs

Create Differential Pairs of nets or pin pairs and assign Properties (Rules) by selecting nets or pin pairs in the Project Explorer and/or in the design workspace.

Procedure

1. Use one of the following methods, or a combination, to select the nets or pin pairs for the matched length pin pair group:

   - **Using the Project Explorer** — Use the nets or lists of pin pairs of nets in the Project Explorer to select the pin pairs.

     > **Tip**
     > You might find it useful to split the Project Explorer on page 52 for tasks like this.

     i. In the Project Explorer, expand the Net Objects list and then expand the Nets list. Expand an individual net to view its pin pairs.

     ii. Select either two nets or two pin pairs to add to the differential pair.

     iii. Right-click and click the **Copy** popup menu item, or drag your selection from the list of nets or lists of pin pairs and then right-click and click the **Paste** popup menu item, or drop them onto the Differential Pairs list.

   - **Using the Workspace** — Use the pointer to select nets or pin pairs in your design.

     i. In the design, select two nets or two pin pairs to make up the differential pair.

     ii. Right-click and click the **Make Differential Net** or the **Make Differential Pin Pair** popup menu item.

2. In the Project Explorer, expand the Net Objects list and then expand the Differential Pairs list. The Differential Pairs list expands to reveal a new Net<->Net or Pin Pair<->Pin Pair. For example, CLK1<->CLK2.

3. Right-click in the Project Explorer and make sure the Allow Selection setting is checked. Selecting a differential pair in the Project Explorer must also select the nets or pin pairs in the design area to open the properties of the differential pair.

4. Place the pointer over the new differential pair, right-click and click the **Properties** popup menu item.

5. In the "Differential Pair Properties Dialog Box, Pair Tab" on page 511, set unique properties for the pair. For specific instructions, see "Setting Differential Pair Properties" on page 184.

6. Click **OK**.

Related Topics

Differential Pair Layer Hierarchy

Assigning the Same Rules to Multiple Differential Pairs

Set the Properties (rules) of multiple pin pairs at the same time.

Procedure

1. In the Project Explorer, expand the Net Objects list and then expand the Differential Pairs list.
2. Right-click in the Project Explorer and make sure the Allow Selection setting is checked. Selecting differential pairs in the Project Explorer must also select the nets or pin pairs in the design area to open the properties of the differential pair.
3. Use Shift or Ctrl to select multiple differential pairs.
4. Place the pointer over one of the differential pairs, right-click and click **Properties**.
5. In the "Differential Pair Properties Dialog Box, Pair Tab" on page 511, set unique properties for the pair. For specific instructions, see "Setting Differential Pair Properties" on page 184.

   Some existing settings may be unique to individual differential pairs. If some values appear blank, it is because the values are different between the various differential pairs. If you leave the value blank, the original unique value will be retained for each differential pair. If you overwrite the value, then all differential pairs inherit that value.

   The <All Layers> is always displayed as a grayed text. However, if a specific layer appears in the list, but it displays with grayed text, it indicates that some differential pairs call out that specific layer while others do not.

   A check box that is neither selected nor cleared but shows a filled blue square indicates that some differential pairs have the box checked and others do not. Leaving the check box in this filled blue state retains the unique settings on the Differential Pairs. Clearing or selecting the check box disables or assigns that state to all the selected differential pairs.
6. Click **OK**.

Related Topics

Creating Differential Pairs

Differential Pair Layer Hierarchy

You can assign differential pair width and gap values to layers and categories of layers; however, a layer may also fall into one or more categories.

For example, Layer 2 may also be a plane layer, and an outer layer. Therefore, the following hierarchy is followed to define which layer settings take priority:

1. All Layers
2. Plane Layers
3. Outer Layers

4. Inner Layers

5. Individual Layers

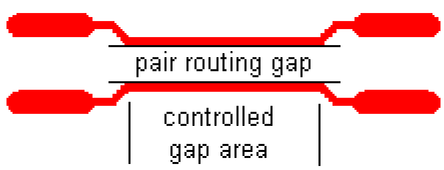
Tip
Individual Layers has highest priority.

Setting Differential Pair Properties

Use the **Pair** tab of the Differential Pair Properties dialog box to set the gap, trace width, length, and obstacle handling for differential pairs.

You can set different properties for differential pairs, which affects their routing, whether you are routing interactively, autorouting, or editing traces. Differential pair properties determine the gap between the traces in the controlled gap area, the minimum and maximum trace lengths and widths, and obstacle behavior in the controlled gap area.

Figure 25. Controlled Gap Area



Tip
When creating differential pairs, the following should be noted:

- SailWind Router routes differential pairs using the default trace width value from the Differential Pair Properties dialog box.
- Setting the differential pair width and gap per layer allows you to better control impedance.

Procedure

1. Select a differential pair then right-click and click the **Properties** popup menu item; then click the **Pair** tab.

2. In the “Set trace width and gap for the pair by layer” area, type Width and Gap values in the <All layers> row.

   Restriction:
   You cannot delete the <All layers> row.

3. To set the width and gap per layer, click **Add**, click in the Layer cell in the newly added row, and select the layer for which to set width and gap values. Then type Width and Gap values in the appropriate cells. The gap rule overrides any other rule defining a clearance between the differential pair members. Therefore, the gap is the minimal clearance and must be provided when possible.

Note:
If you select multiple differential pairs, and a layer setting does not belong to all of the selected pairs, the Layer box is unavailable for that layer. If you type a new value, it applies to the differential pairs that have this layer setting only.

Tip
Keep the following tips in mind:

- If you select multiple differential pairs that have the same layer setting, but the Width and Gap values do not match, the Width and Gap cells will appear empty. You can, however, type a new value, and the new value will be applied to all selected differential pairs when you click **OK** or **Apply**.
- You can also set the Width and Gap values using the Spreadsheet window, sorted for differential pairs. The spreadsheet will show the width and gap value set for <All layers>. If a specific layer has a unique setting, an asterisk appears next to the width or gap value.

For more information, see Differential Pair Layer Hierarchy.

4. Type the minimum length value in the Minimum box. Type the maximum length value in the Maximum box.

Tip
You can lengthen the differential pair by adding an accordion.

5. Select the “Restrict layer changes during autorouting” check box to force routing of the pair on a single layer. (This setting does not restrict layer changes when routing interactively.)

6. Select the “Allow pair to split around obstacles” check box to temporarily exceed the pair routing gap. This setting applies to autorouting and does not restrict splitting around obstacles when routing interactively.

7. Type the maximum number of obstacles to route around in the Maximum number of obstacles box.

Note:
SailWind Router does not count obstacles in the start zone or end zone. This is the zone between the connection point and the shoulder of the differential pair.

8. Type 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.

Note:
SailWind Router does not check the obstacle size in the start zone or end zone. This is the zone between the connection point and the shoulder of the differential pair.

9. Click **OK**.

Related Topics

Setting Object Properties

Creating a New Object in a Secondary Group

Component Rules

To provide improved control over the trace width for fine pitch parts or densely populated BGAs, you can specify trace width rules at the component level resulting in different trace width rules within the component boundary.

You can also set pad entry, fanout, clearance, via biasing, and same net rules at the component level. The pad entry and fanout rules are applied to all pins in the component.

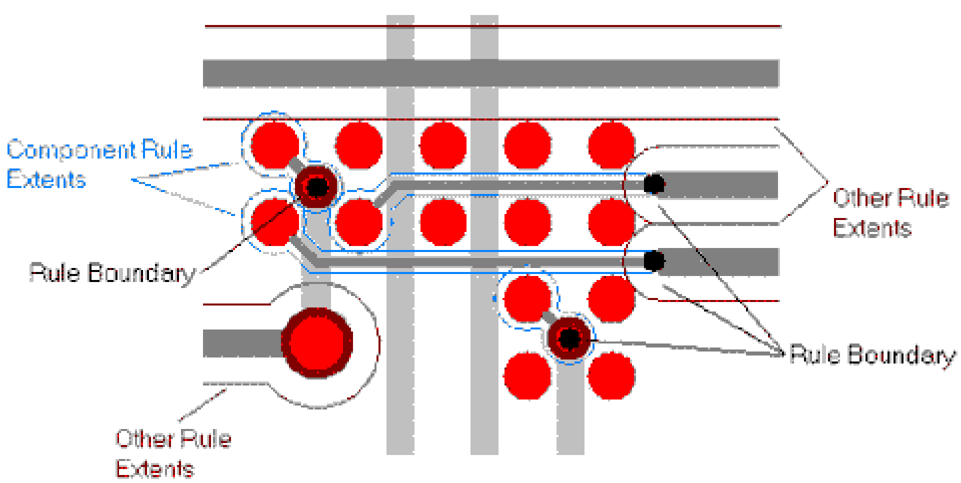
Component clearance rules are based on pin clearances and other component-specific design rules. SailWind Router creates a boundary automatically to indicate where component rules start and stop.

SailWind Router often assigns boundaries at vias and trace vertices. The software automatically determines boundary locations when it autoroutes traces; however, when interactively routing, you can define when to start and stop using component rules.

When a component rule is in use, it applies to trace objects between the rule boundary and a component pin. If a trace passes under a component but does not connect to the component, the software does not check it against the component's clearance rules.

Rule boundaries (Figure 26) also have an effect on trace editing (for more information, refer to "“Creating Traces Using Component Rules” on page 308").

Figure 26. Rule Boundaries



Via biasing rules apply to the first and last vias connecting to the component. When routing, SailWind Router starts at one pin and follows a path to the next pin in the chain. Therefore, the software assigns a non-component rule via type to the first and last vias along the trace path. If this creates a spacing violation, then the software uses a component rule via type instead.

Tip
Exception: A unique situation exists where the component boundary is not calculated solely upon the design rules. For example, in SailWind Router, you exit a fine pitch BGA using a narrow trace width, but the trace does not automatically exit component rules at the edge of the component boundary. You decide when to use the Switch Rules command to exit component rules to change to a larger trace width according to the most relevant rule from the design hierarchy. In this case, you can extend the component rules far from the component and outside of the calculated area of the component boundary.

SailWind Layout does not understand these extended Component Rules and it may lose extended boundary locations. However, if you do not edit the design in SailWind Layout and you reopen it in SailWind Router, the extended boundary locations likely recalculate correctly.

Also, in SailWind Router, if you change the component rules after extending the component rules with a trace, the recalculated component boundary will correspond to the new component rules and may differ from the old boundary - creating an invisible split in the trace. When the design is opened in SailWind Layout, this location becomes a tack.

Length Rules

Length rules specify the minimum and maximum trace length. You can specify length rules for a net, electrical net, or pin pair, and matched trace lengths for two or more nets, electrical nets, or pin pairs.

You can make length rules hard or soft by using the “Ignore length rules when required to complete traces” check box in the **Tune** tab of the Options dialog box. See Hard and Soft Rules.

You can assign length rules to nets and pin pairs at many levels of the hierarchy. This includes assigning the following:

- Net rules at the net and class rule levels
- Pin pair rules at the group and pin pair rule levels
- Net or pin pair length rules to members of a differential pair
- Length rules to electrical nets
- Matched length rules at the net, class, electrical net, group, and pin pair rule levels by grouping these objects into matched length groups
- Matched length net groups can contain mixtures of nets, electrical nets, and net classes

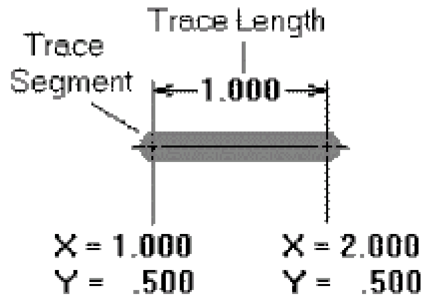
For more information, see “Rules Hierarchy” on page 155.

Calculating Length

The delta between end points of a segment is the length of a trace segment.

For example, a horizontal segment having a common Y coordinate for both end points but having a difference of one inch in the X coordinate for the end points has a trace length of one inch (Figure 27).

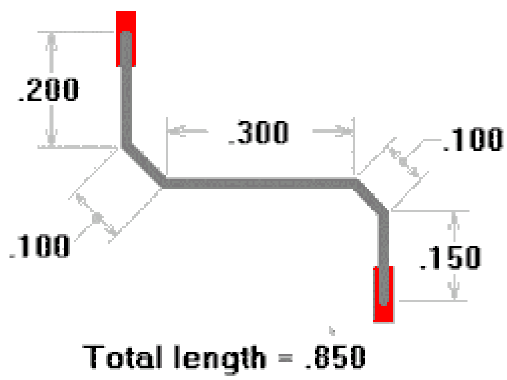
Figure 27. Calculating Trace Segment Length



SailWind Router calculates the length of a collection of trace segments, a pin pair, or a net by totaling the length of all trace segments, including the portions embedded within pads or vias (Figure 28). The total trace length calculation includes trace segments that overlap pads or vias.

Tip
Unlike SailWind Layout, SailWind Router counts overlapping trace segments in the trace length calculation.

Figure 28. Calculating Total Length

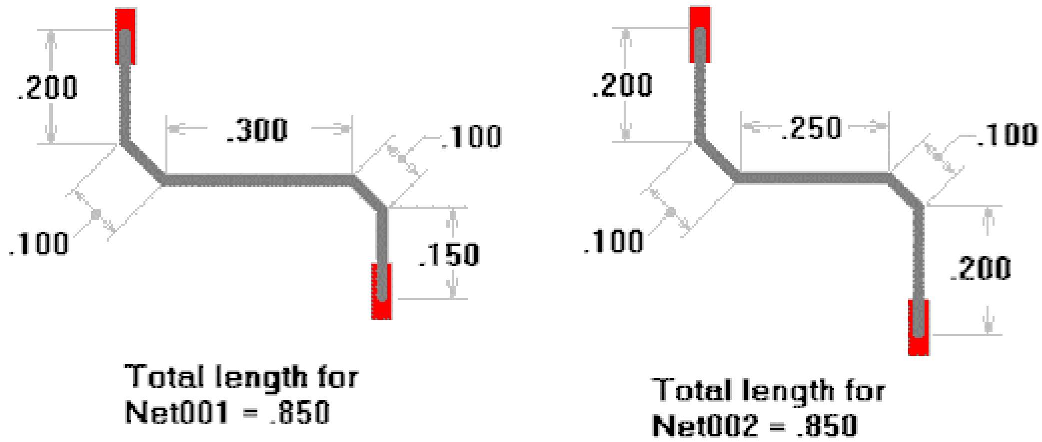


Matched Length

Matched length is defined as two or more nets, electrical nets, or pin pairs having equal total net lengths. Matched length is typically specified with a tolerance.

Figure 29 shows two different nets with a matched length, even though the trace pattern is slightly different.

Figure 29. Matched Lengths

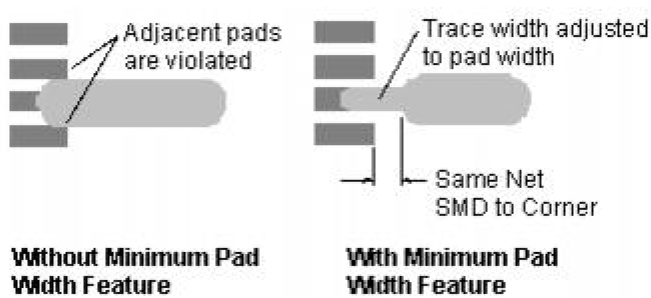


Routing Rules

Routing rules specify trace width allowances and what should happen during autorouting operations.

A SailWind Router feature automatically reduces trace width to the pad, provided the trace width is within the minimum and maximum width rules (Figure 30). The routing rules for the trace must have a minimum width equal to or less than the width of the pad; otherwise SailWind Router does not use the feature and does not complete the trace.

Figure 30. Routing With and Without Minimum Width Specification



Working with Multiple Object Properties

You can assign or edit the design properties of multiple design objects simultaneously. Simultaneous editing reduces assignment tasks and aids in providing the same settings to multiple objects.

Procedure

1. Select two or more objects of the same type (for example, two nets or two components).
2. Right-click and click the **Properties** popup menu item.

Results

If multiple object types are selected, the No Properties dialog box appears. Multiple object type selection is not supported.

If multiple objects are selected, the dialog box controls behave differently, depending on the status of the objects selected.

For example, if you select a pin that serves as a test point, and another pin that does not (mixed selection), the state of the “Serve as test point” check box is dimmed with a check mark.

Table 60. Multiple Objects— Dialog Box Behavior

Control Type and Status	State	Allowed actions
Check box — Cannot set (database restrictions)	Dimmed	Non-modifiable
Check box —Mixed	Dimmed with check mark	All on or All off
Check box — All on	Check mark	All off
Check box — All off	Clear	All on
Box — Cannot set (database restrictions)	Dimmed	Non-modifiable
Box — Cannot set (all read-only)	Dimmed	Non-modifiable
Box — All the same	Same value	Editable value
Box — Mixed	No value	Clear - editable
Static text — All the same	Same value	Non-modifiable
Static text — Not all the same	Object count provided	Non-modifiable
Preview — All the same	Same view	Non-modifiable
Preview — Not all the same	Dimmed	Non-modifiable

Related Topics

Properties Dialog Box, No Properties Tab

Setting Object Properties

Chapter 12
Turn on Design Rule Checking

Design Rule Checking (DRC) checks for design rule violations during interactive routing, route editing, and placement editing. When you enable DRC, SailWind Router checks your design for errors continuously, based on the defined design rules.

When you open a file in SailWind Router, it uses the existing status of DRC and not the setting from SailWind Layout. You can use the DRC Filter toolbar to enable or disable DRC for individual design rules (clearance, trace width, same net, placement, and length). The DRC Filter Toolbar reflects the settings in the Design Rule Checking dialog box.

When DRC is off, the software does not perform checks for violations, but it does not permit subnet violations. Traces between subnets may cross but they cannot join at the intersection. Layer Biasing and Via Biasing violations are not permitted when DRC is off. Autorouting always operates as though DRC is enabled and allows no errors.

Setting Design Rule Checking

You can set DRC options from the Design Rule Checking dialog box.

Tip
If the items on the DRC toolbar are not available (and are grayed out), click the **DRC On/Off (DRC)** button on the DRC toolbar to enable the ability to select and edit the specific DRC checking options. Once you have enabled the DRC editing capability, you can use the toolbar buttons to enable and disable specific DRC checks.

Procedure

1. Click the **Tools > DRC Settings** menu item, or click the **DRC Filter** toolbar button then click the **DRC Settings** button (on the DRC Filter toolbar).

2. In the Design Rule Checking dialog box, in the Enable column, select the check box for each design rule you want to check.

3. In the Error Response column, click the cell of the design rule for which you want to set an error response. Select the appropriate response for how you want SailWind Router to handle design rule violations when it encounters them during interactive routing, trace editing, and placement editing operations.

   For more information on the DRC error response behaviors, see “Design Rule Checking Dialog Box” on page 405.

4. Click **Close**.

Results

The desired design rule options are set, and the DRC Filter toolbar is updated with the same information.

Component Spacing

SailWind Router provides automated features to ensure adequate spacing between components.

Spacing Basics

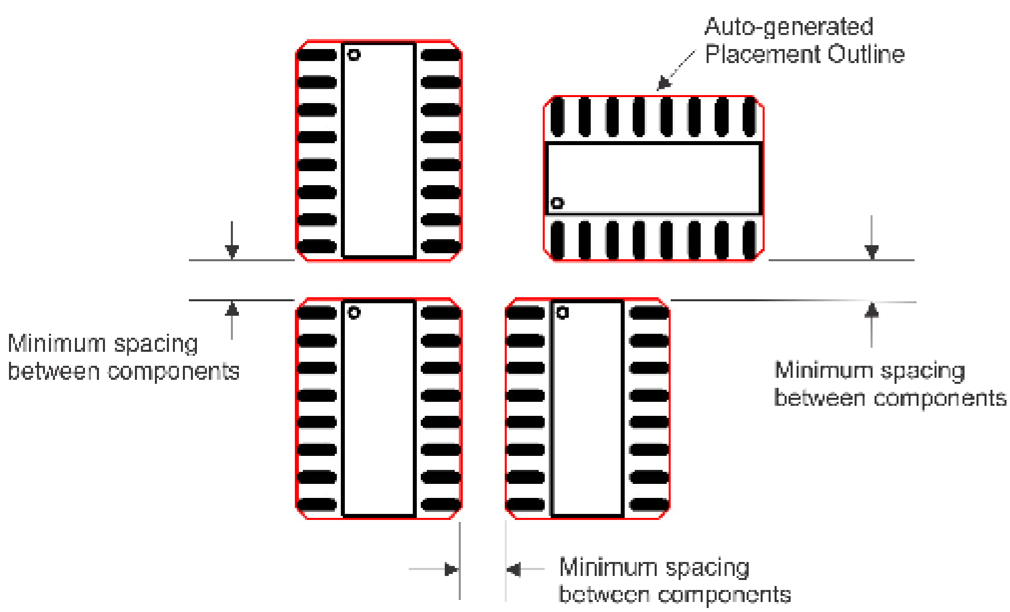
The software places components using the units of the Design grid when you enable the “snap to grid” setting.

The software prevents components from overlapping if both of the following conditions are met:

- On the **Tools > Options** menu item, **Fabrication** category, you have set up a value for “Minimum spacing between components” in the Component assembly area
- On the DRC Toolbar, you have enabled Placement DRC (design rule checking), and in the DRC Setup dialog box, you have set Placement to Prevent Errors mode

Auto-generated Component Outlines

The body of a component is the furthest extent of any decal object. Upon loading the design, if the system does not detect a component placement outline on Layer 20, it auto-generates an outline that encompasses all objects contained within the decal. The DRC function uses this visible outline to establish a reference component boundary.



In the preceding illustration, the “Minimum spacing between components” value applies in all cases when Placement DRC is enabled.

Note:
SailWind Router supports separate Clearance DRC checking. If Clearance DRC checking is enabled during a Placement DRC operation, the system performs both checks simultaneously. If the TH or SMD pads of a component encounter other TH or SMD pads of another component, and the placement outlines of the components do not violate the "Minimum spacing between components" value, then the system uses the Pad to Pad, SMD to SMD, or Pad to SMD rule.

The "Minimum spacing between components" value applies to the auto-generated outline representing the furthest extent of any decal object on the following layers:

- Component layer (top or bottom)
- Associated silkscreen layer for component layer (silkscreen top or bottom)
- Layer 0 (All layers).

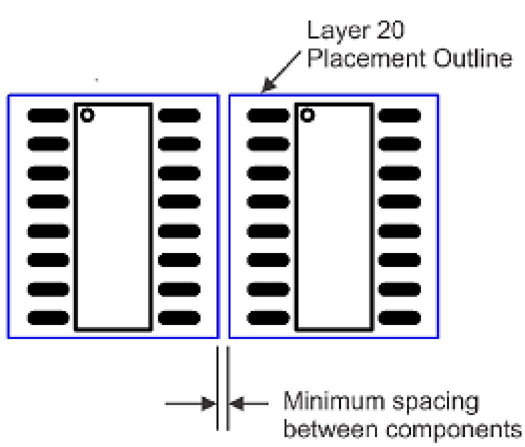
There is only one "Minimum spacing between components" value that must apply to all components.

Advanced Spacing

While a single "Minimum spacing between components" value 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 outline during the placement process to create more spacing around components. The extra, larger outline is often called a component courtyard or nudge outline. The "Minimum spacing between components" value applies between Layer 20 outlines.

Tip
In extended layers mode, Layer 20 becomes Layer 120.



Spacing Verification

If you are unsure if components have been placed outside of the design rules that you have set up, run a Clearance design verification check. Be sure that all aspects of your components, including the color for Top Plc. and Bot. Plc. objects are visible since the Clearance Placement Outline check only verifies objects that are visible and within view during its checking routine.

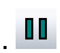
When setting up the Clearance check in the **Tools > Options** menu item, in the **Design Verification** category, ensure that you have selected the check box for “Placement outline” to check for the “Minimum spacing between components” value between the placement outlines.

Checking DRC Errors During Operations

Certain operations may be unsuccessful—such as manual routing and component placement—if they violate design rules. If you encounter an error, you can choose to suspend the operation and view the DRC errors.

Procedure

1. Right-click and click the **Explain Last Error** popup menu item.

   Alternatively, on the Suspend toolbar, click the **Explain** button. 

2. Click an error in the Error Type column in the Spreadsheet Window to display the DRC error message in the status bar and highlight the error location in the workspace.

3. Right-click and click the **Cancel** popup menu item to resume the command or click the **Continue with Errors** popup menu item to continue the operation despite the DRC violation.

Related Topics

Design Rule Checking and Interactive Routing

Setting Design Rule Checking

Chapter 13
Place Components

SailWind Router provides a comprehensive selection of placement commands that allow very precise positioning of components. SailWind Router also has commands that allow you to move, flip, rotate, and spin components during or after initial placement.

Placing Components

Use placement commands to move components on a board.

SailWind Router's component placement commands differ slightly from those in SailWind Layout. Familiarize yourself with the placement rules and options to fully understand placement commands and results.

You can modify component placement manually or you can edit component properties. To manually place a component or component group, you use a combination of pointer operations and Move, Spin, Rotate, and Flip commands. To edit a component or component group's placement, you modify the X/Y location, rotation, or layer properties in the **Component** tab of the Component Properties dialog box.

The component's origin snaps to the Placement grid during placement operations except in the following cases:

- You place the component using the Spin command.
- You place the component by editing its X/Y or rotation properties.
- You clear the Snap check box for the Placement grid on the **Grid** tab of the Design Properties dialog box.

You can move components after you set options and preferences defining how you want to move them and how you want to perform DRC. You can also spin components, rotate them by 90 degrees, or flip them to the other side of the board.

Use the Placement toolbar to activate placement commands.

Tip
Placement commands are also available in Object mode from the shortcut menu when you right-click over a selected component.

In general, the placement commands behave like SailWind Layout placement commands. The following list summarizes the main placement behaviors that are different in this program:

- Set an origin for a component or component group.
- Spin, rotate, and flip a component group as a whole, using one origin for the entire group.
- Rotate components counter-clockwise, regardless of the side of the board on which the component resides.
- Set the angular increment value for Spin.
- Reroute during placement.
- Prevented from moving unions, reuses, arrays, an arbitrary set of components and traces, coppers, texts, 2D-lines, or keepouts.

  Tip
  You can place a component by editing its properties. Modify the X/Y location, rotation, or layer properties in the **Component** tab of the Component Properties dialog box.

Component Placement Rules

During placement operations, components adhere to specific rules that define clearances between design objects.

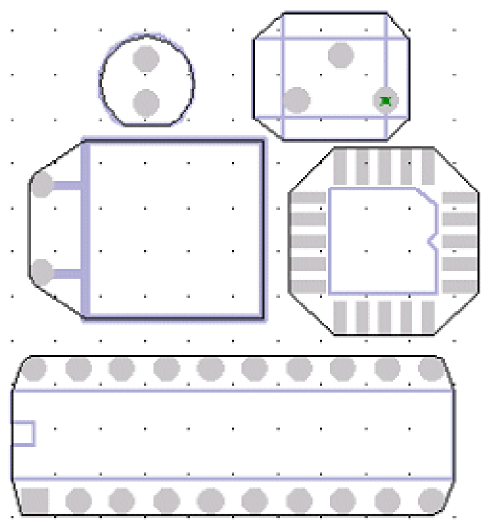
Clearance between two components is defined as the minimum distance between the component placement outlines. During placement operations, SailWind Router checks the clearance between a selected component and other components for the following object relationships:

- Component-to-component clearance
- Component-to-board outline clearance
- Component-to-board cutout outline clearance

The software takes measurements from the centerline of the line used to create the component placement outline to the following:

- The centerline of another component placement outline
- The centerline of the board outline
- The centerline of a line defining a board cutout

Figure 31. Example Placement Outlines



The SailWind Layout Decal Editor enables you to use 2D lines to create a representation of the component placement outline on Layer 20. The software recognizes these placement outlines as follows:

- When a part is brought into a design in SailWind Layout, this outline converts from a group of 2D lines into a Placement Outline design object on Layer 20. Visibility is controlled by color tiles on Layer 20 in the Top/Bottom Object columns of the Display Colors dialog box.
- When the design opens in SailWind Router, the software recognizes this Placement Outline object on Layer 20 and converts it to a SailWind Router Placement Outline object whose visibility is controlled separately by color tiles in the Top Plc. or Bot. Plc. columns in the **Options** menu item, Colors category. The Placement Outline design object visibility is no longer associated with Layer 20.
- If no Layer 20 Placement Outline is detected for a component, the system auto-generates an outline that encompasses all of the objects in the pcb decal. The software recognizes it as a SailWind Router Placement Outline object whose visibility is also controlled separately by color tiles in the Top Plc. or Bot. Plc. columns in the **Options** menu item, Colors category.
- Placement Outline objects derived from Layer 20 Placement Outlines and auto-generated Placement Outline objects are treated equally in SailWind Router, and no distinctions are drawn between them by the DRC or Design Verification processes.
- The auto-generated SailWind Router Placement Outline design objects are temporary for the current design session. When the design returns to SailWind Layout, the software removes these design objects from the database. The visibility of any Layer 20 Placement Outline design objects revert back to control by Layer 20 in SailWind Layout.

During placement operations, SailWind Router checks both the height of the selected component and the keepout height settings. If you place a component into a keepout, and the height of the component exceeds the height setting of the keepout, a placement DRC violation is reported.

Tip
Using the Layout Editor in SailWind Layout, you can set the geometry of a component height keepout on the primary component side, the secondary component side, or both sides of the board. You can edit the height of the keepout by editing its properties.

Placing Components and DRC

SailWind Router checks design rules between placed components and other objects but does not check design rules between objects on the placed component or component group. When you use Move or Spin, placement-only DRC violations appear highlighted before you complete the placement.

Tip
No design rule checking occurs for a component or component group that you place completely outside of the board outline. However, design rule checking does occur if you place the component or component group partly or entirely inside the board outline.

When a DRC violation occurs, the Error Response setting in the Design Rule Checking dialog box determines how the placement command completes and it reports the violation.

Table 61. Effects of Error Response Setting

Error response	Placement response to DRC violation	DRC violation reporting
Prevent	The software rejects the new placement location and returns the component to its original location. For Move and Spin the component remains attached to the pointer until you select an error-free location, suspend placement (to view DRC violations), or cancel placement. For Rotate and Flip, the component briefly appears at the new location and then returns to the original position.	To view DRC violation reporting when the component remains attached to the pointer during Move or Spin, right-click and then click **Explain Last Error**. The spreadsheet window displays the DRC violations. To view DRC violation reporting when the component returns to the original position during Rotate or Spin, click the **Explain Last Error** button on the Suspend toolbar. The spreadsheet window displays the DRC violations.
Explain	Placement suspends to enter Display Last Error mode.	The spreadsheet window displays the DRC violations.
Warn	Placement completes.	The spreadsheet window displays the DRC violations.

Tip
When more than one Error Response list setting is selected in the Design Rule Checking dialog box, the setting with the highest priority applies. “Prevent” has the highest priority and “Warn” has the lowest priority.

Setting Fabrication Rules to Control Placement

You can set the minimum spacing between components clearance value and the default component height limits.

Tip
You cannot place a component if its distance to another component is less than the “Minimum spacing between components” clearance value that you set in the **Options** menu item, Fabrication category.

Tip
If you set a Geometry.Height attribute for a component in SailWind Layout, you cannot place the component in SailWind Router if the attribute value exceeds the height you set in the **Options** menu item, Fabrication category.

Procedure

1. Click the **Tools > Options** menu item; then, in the Options dialog box, click the Fabrication category.
2. In the Component assembly area, specify placement rule settings.
3. Click **OK** to accept the settings and close the Options dialog box.

   Tip
   To enable placement rule checking, on the DRC Filter Toolbar, select Placement, and then, in the Design Rule Checking dialog box, set the desired response action.

Related Topics

Placing Components

Adding Keepouts

You can add keepout areas to your design 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.

Keepout areas you create in SailWind Router transfer with the design when you open it in SailWind Layout.

Procedure

1. Click the **Drafting Operations** toolbar button. 
2. On the Drafting Operations toolbar, click the Keepout button. 
3. Create one of the following shapes for the keepout area:
    - Creating a Polygon Drafting Object
    - Creating a Circle Drafting Object
    - Creating a Rectangle Drafting Object
4. In the “Keepout Properties dialog box” on page 495 that appears, specify the layer where you want to place the keepout area and the restrictions that you want to apply.

    Restriction:
    When you choose layer assignments, if a restriction is not available for that layer, its corresponding check box is unavailable. For example, if you choose a non-placement layer, the Placement check box is not available.

    ---

5. Click **OK**.

    The software creates the keepout. If you create other keepouts, they use the restrictions you set here as the default.

Results

The keepout displays with the hatch filling selected in the Options dialog box > Global category > “General subcategory” on page 441.

Related Topics

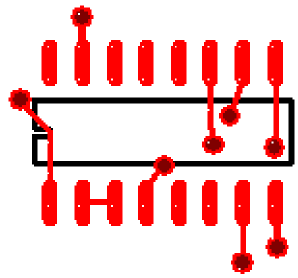
Properties Dialog Box, Keepout Tab

Moving Routed Components

SailWind Router manipulates the traces attached to the pins of a placed component according to the Move Fanouts and the Reroute Traces settings in the **Placement** tab of the Options dialog box. When you enable the Move Fanout setting, SailWind Router places the traces and vias recognized as fanout with the component.

The following figure shows a selected component and its fanout.

Figure 32. Fanout of a Selected Component



The “Move components with Fanout” setting has priority over the Reroute Mode setting. If you select the “Move components with Fanout” check box, the fanout traces and vias move with the component. In contrast, if you clear the check box, the fanout traces and vias are subject to the Reroute Mode setting.

If SailWind Router cannot route the nets free of DRC violations, and you set the Error Response setting in the Design Rule Checking dialog box to “Prevent” or “Explain,” it leaves the nets unrouted. However, if you set the Error Response setting to “Warn,” the software routes in a straight rat’s nest pattern any nets that it cannot otherwise reroute free of DRC violations.

Related Topics

Options Dialog Box, Placement Category

Setting the Placement Origin

You can change the placement origin for a component or component group.

Procedure

1. Click the **Tools > Options** menu item; then, in the Options dialog box, click the Global category > General subcategory.
2. In the “Object movement” area, click the “Move object by” list and select “Origin.”
3. Click **OK**.
4. On the Placement toolbar, click the **Select Mode** button.
5. Select the component or component group.
6. Right-click and click the **Set Move Origin** popup menu item.
7. Move the pointer and click to set the new origin.

   The origin you specify applies only for the current selection. When you make a new selection, the move origin reverts to its original location on the component.

   If you do not specify a new origin, SailWind Router uses the original origin of the selected component. If you select a group of components, the software calculates the origin as the lower left of the selected group.

8. Click **Yes** to confirm the new origin or click **No** to select another origin.

> **Tip**
> Set Move Origin is not available in Verb mode.

Related Topics

Placing Components

Placing Members of a Component Group

If you use the Move, Spin, Rotate, or Flip commands with a component group, the component group origin becomes the focal point for the placement operation while relative positions among members remain unchanged. In contrast, if you edit the component properties, the individual component origins become focus points for placement operations and the relative positions among members of the component group do change.

Figure 33 and Figure 34 show how the Rotate command places the members of a component group.

Figure 33. Default Component Group Origin and Initial Component Positions

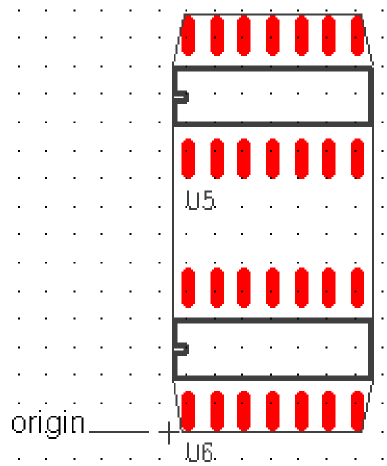


Figure 34. Component Positions After the Rotate Command Has Completed

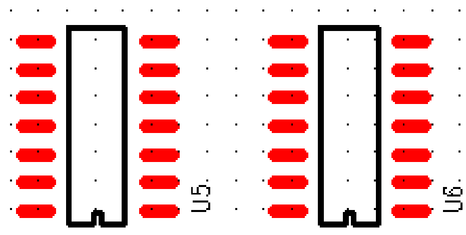


Figure 35 and Figure 36 show how setting the Orientation property places the members of a component group.

Figure 35. Default Component Origins and Initial Component Positions

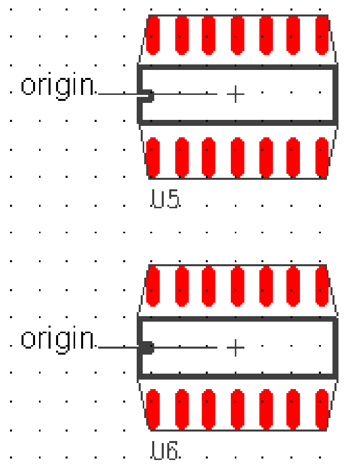
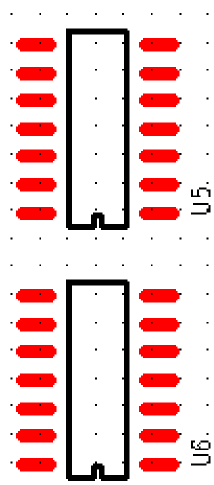


Figure 36. Component Positions after Setting Orientation Property for the Component Group to 90



Moving Components

You can move components individually or as groups. SailWind Router provides several methods for moving components.

Tip
When the component is attached to the pointer, for all modes except drag and drop, you can right-click to start another command, change component properties, or use shortcut keys to change the workspace view.

Moving Mode	Description
Moving Components Using Object Mode	Move a component by selecting it first and then choosing to move it.
Moving Components Using Verb Mode	Move a component by choosing the Move mode and then selecting the component.
Moving Components Using Drag and Attach Mode	Move a component by dragging it with the mouse and clicking to place it.
Moving Components Using Drag and Drop Mode	Move a component by dragging it with the mouse and "dropping" it in place by releasing the mouse button.

Moving Components Using Object Mode

Move a component by selecting it first and then choosing to move it.

Prerequisites

Set the object move properties. For more information, see The Object Movement area of the Global / General Options on page 441

Procedure

1. Select the components you want to move.
2. Right-click and click the **Move** popup menu item. Alternatively, on the Placement Toolbar, click the **Move Component** button.
3. Move the pointer to the new location and click to complete the move.

   If DRC errors result from the new component location, and you set the DRC setting to Prevent in the Design Rule Checking dialog box, the Move command remains attached to the pointer. You can suspend the Move command to examine DRC errors. For more information, see "Checking DRC Errors During Operations" on page 194

Moving Components Using Verb Mode

Move a component by choosing the Move mode and then selecting the component.

Prerequisites

Set the object move properties. For more information, see The Object Movement area of the Global / General Options on page 441

Procedure

1. On the Placement toolbar, click the **Move Component** button.
2. Select the components you want to move.
3. Move the pointer to the new location and click to complete the move.

   If DRC errors result from the new component location, and you set the DRC setting to Prevent in the Design Rule Checking dialog box, the Move command remains attached to the pointer. You can suspend the Move command to examine DRC errors. For more information, see "Checking DRC Errors During Operations" on page 194
4. Repeat steps 4 and 5 as needed to move additional components.
5. Press the Esc key to exit Verb mode.

Moving Components Using Drag and Attach Mode

Move a component by dragging it with the mouse and clicking to place it.

Prerequisites

Set the object move properties. For more information, see The Object Movement area of the Global / General Options on page 441

Procedure

1. Click the **Tools > Options** menu item; then, in the Options dialog box, click the Global/General page.
2. In the "Object movement" area, click the "Drag object using" list and select "Drag and attach."
3. Click **OK**.
4. Select the components you want to move.
5. Click and hold the left mouse button over the components, start dragging the components, and then release the left mouse button to attach the components to the pointer.
6. Move the pointer to the new location and click to complete the move.

   If DRC errors result from the new component location, and you set the DRC setting to Prevent in the Design Rule Checking dialog box, the Move command remains attached to the pointer. You can suspend the Move command to examine DRC errors. For more information, see "Checking DRC Errors During Operations" on page 194.

Moving Components Using Drag and Drop Mode

Move a component by dragging it with the mouse and "dropping" it in place by releasing the mouse button.

Prerequisites

Set the object move properties. For more information, see The Object Movement area of the Global / General Options on page 441.

Procedure

1. Click the **Tools > Options** menu item; then, in the Options dialog box, click the Global / General page.
2. In the "Object movement" area, click the "Drag object using" list and select "Drag and drop."
3. Click **OK**.
4. Select the components you want to move and release the left mouse button.
5. Click and hold the left mouse button over the components, then move the pointer to initiate a drag move.
6. Move the pointer to the new location and release the left mouse button to complete the move.

   If DRC errors result from the new component location, and you set the DRC setting to Prevent in the Design Rule Checking dialog box, the Move command remains attached to the pointer. You can suspend the Move command to examine DRC errors. For more information, see "Checking DRC Errors During Operations" on page 194.

Related Topics

Placing Components

Placing Components Sequentially

Use the Move Sequential command to attach a set of selected components to your cursor one at a time for placement in the design.

There are two ways to move components sequentially:

Method	Description
Placing Components Using the Find Dialog Box	Use the Find dialog box to select the components and also to apply the Move Sequential command.
Placing Components by Selecting Components in the Workspace	You select components in the working area and use a modeless command to apply the Move Sequential command.

Placing Components Using the Find Dialog Box

Use the Find dialog box to select the components and also to apply the Move Sequential command.

Procedure

1. Click the **Edit > Find** menu item.
2. In the Find dialog box, set the “Find by” option to “Ref. Designator, Part Type,” or “Decal.”

   Tip
   Select multiple Ref. Des. prefixes, part types, or decals to expand the list of available components.

3. From the resulting list, select all the parts you want to move.

   Tip
   Use Shift+click for a range of components, or Ctrl+click for multiple components.

4. In the Action list, select “Move Sequential.”
5. Click **OK**.
6. In the prompt, “Proceed with next object? (*value*)” click one of the following options:

   - **Yes** to attach parts to the cursor successively, but prompt every time after a 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.
- **Cancel** to abort the process. You return to the Find dialog box.

7. The first part attaches to the pointer for placement (parts attach to the pointer in ascending reference designator order). Click to place the component.

Tip
With a part attached to the pointer, you can right-click to access the Rotate 90, Spin mode, or Flip commands and also use modeless commands.

Note:
You can move a component by either its origin or midpoint, as set in the **Global** category > **General** subcategory of the Options dialog box.

Related Topics

Find Dialog Box

Options Dialog Box, Global Category, General Subcategory

Placing Components by Selecting Components in the Workspace

You select components in the working area and use a modeless command to apply the Move Sequential command.

Procedure

1. In the working area, with nothing selected, right-click and click the **Select Components** popup menu item.
2. Using Ctrl+click, select all the parts you want to move.

Tip
The parts are attached to the pointer in the reverse order in which they are selected. The last component selected is the first one attached to the pointer.

3. Type the modeless command MS and then press Enter.
4. In the prompt, “Proceed with next object? (*value*)” click one of the following options:

- **Yes** to attach parts to the cursor successively, but prompt every time after a 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 a part and go to the next.
- **Cancel** to abort the process. You return to the Find dialog box.

5. The first part is attached to the pointer for placement. Click to place the component.

Tip
With a part attached to the pointer, you can right-click to access the Rotate 90, Spin mode, or Flip commands, or use modeless commands.

Note:
You can move a component by either its origin or midpoint, as set in the General options on page 441.

Manipulating Components

You can rotate, spin, or flip a component from one side of the PCB to the other.

Procedure

Manipulate components using either the Object mode or the Verb mode as follows:

Mode	Steps
Object mode	1. Select the component or component group to manipulate. 2. Right-click and click the **Flip Side**, **Rotate 90**, or **Spin** popup menu item. 3. If you choose the **Spin** popup menu item, move the pointer to the new location and release the left mouse button to complete the spin. If a DRC error results from manipulating the component, and the DRC setting is set to “Prevent,” the component returns to its original position (or in the case of the **Spin** popup menu item, remains attached to the cursor).
Verb mode	1. On the Placement toolbar, click the **Flip Side**, **Rotate 90**, or **Spin** button. 2. Click a component you want to manipulate. 3. (Spin button only) Click to indicate the new rotation angle to complete the component spin. 4. Press the Esc key to exit the Verb mode

Related Topics

Placing Components

Checking DRC Errors During Operations

Measuring Distance Between Objects

You can measure the distance between components to aid in accurate placement.

Use either of the following methods to measure the distance between objects:

Measuring Distance Between Objects Using In-Place Query
Measuring Distance Between Objects Using the Q Shortcut Key

Measuring Distance Between Objects Using In-Place Query

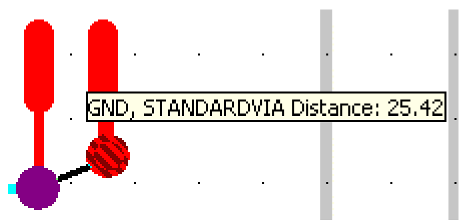
In-place Query allows you to measure the distance between components by selecting the objects.

Procedure

1. On the main toolbar, click the **In-Place Query** button. 
2. Select an object.
3. Point to a second object.

   The shortest distance between the objects appears, as shown in Figure 37

Figure 37. Distance Measured from the Via on the Left to the Via on the Right



Measuring Distance Between Objects Using the Q Shortcut Key

The Q shortcut key attaches a measurement line to the pointer and displays dx, dy, and hypotenuse information, depending on pointer movement. It snaps to the routing grid when Grid Snap is on. When Grid Snap is off, the Q shortcut key makes the measurements without the grid.

Procedure

1. Locate the pointer over the first object.
2. Press the q key and click **Execute**.
3. Locate the pointer over the second object to display the dx, dy, and hypotenuse (“d”) information.

Tip
Click to restart the measurement.

Tip
Ctrl+click to attach a new measurement line to the current measurement line and to display the total length.

4. Press the Esc key to exit the measurement function.

Related Topics

Measuring Distance Between Objects Using In-Place Query

Chapter 14
Virtual Pins

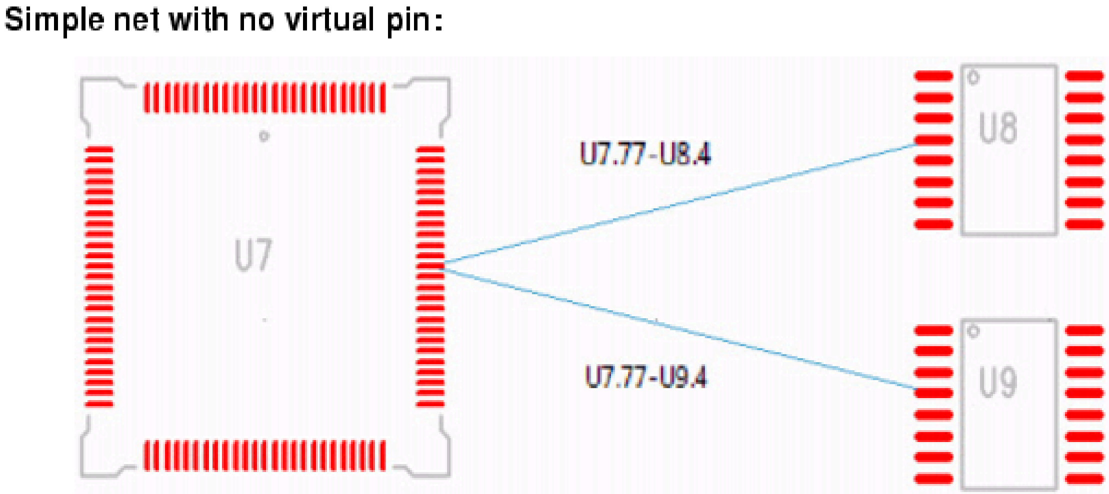
SailWind Router supports the use of *virtual pins*, which can act as nodes for adding pin pair start or end points.

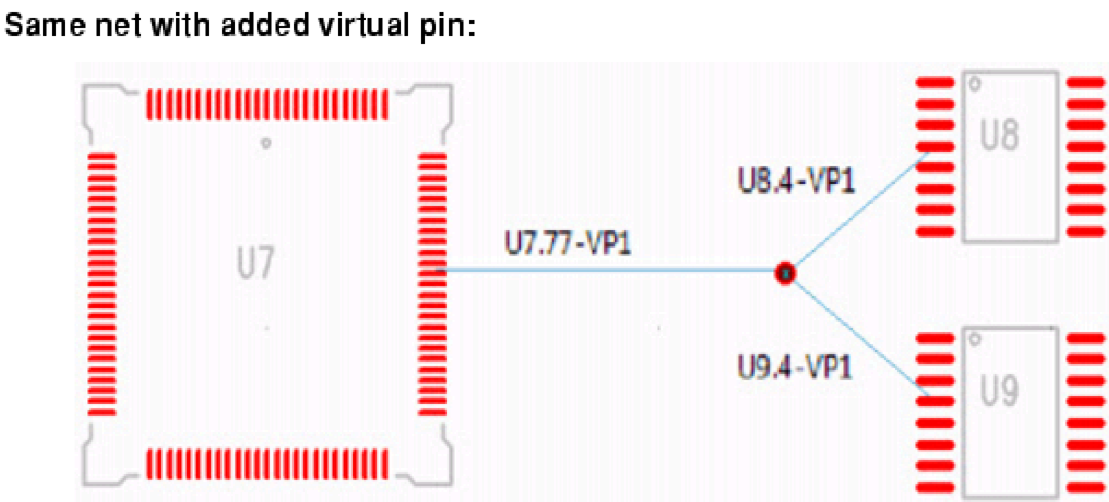
The Virtual Pin

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 Layout.

A virtual pin uses the pad stack of a via. The pad stack can be a through-hole type or partial, or it can be a single-layer pad. You add virtual pins *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:

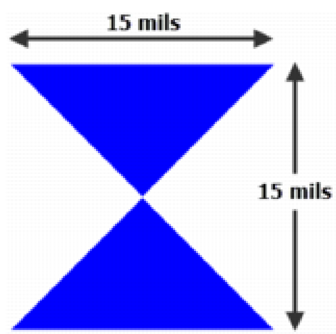




You can assign rules to pin pairs ending on virtual pins in the same way as to pin pairs ending on component pins.

In the display, virtual pins are identified with a marker that uses the color of the board outline.

Figure 38. Virtual Pin Marker



To work with virtual pins, you need the Advanced Rules license.

Virtual Pin Setup

Use SailWind Layout to set up the virtual pins in your design.

See Via Setup in the *SailWind Layout Guide* for information on how to create, edit, and delete pad stacks for vias and virtual pins.

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 conforms to the following:

- 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—the software creates the pin pairs 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 the software creates a starburst pattern.*

  Note that if you previously added a virtual pin to the net and deleted it, SailWind Router sets the net topology to Protected; so the virtual pin you are adding, even though it appears to be the first virtual pin in the net, connects 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 connects to the nearest component pin only. The software does not create a starburst pattern.

- If the virtual pin you are adding is not the first virtual pin in the net, it connects to the nearest virtual pin only. The software does not create a starburst pattern.

Tip
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 virtual pin appears on the pointer.
3. Click to place the new virtual pin. To stop adding virtual pins, right-click and click the **Cancel** popup menu item.

Results

- If you create a starburst pattern, the software generates the new pin pairs and sets the net topology 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 software uses the same format (VP<*number*>) in pin pair names, for example VP7—U2.1.

  For information on setting design rules for the new pin pairs, see "Properties Dialog Box, Pin Pair Tab."

- If you create a non-starburst configuration, or if you create a starburst configuration and want to reconfigure your pin pairs, you need to reschedule the net. See ""Rescheduling Nets" on page 252" for more information.

Deleting a Virtual Pin

You can remove 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 software deletes both the virtual pin and its connection.
- If the deleted virtual pin has two or more connections, all unrouted, the software deletes the virtual pin and makes new connections between the component pins and virtual pins to which it was connected.
- If the deleted virtual pin has two or more connections, some or all of which are routed:
    - The software deletes the virtual pin and all its attached pin pairs.
    - Routing (traces and vias) of the deleted pin pairs remains, and the software assigns it new pin pairs.
    - If traces from different layers were attached to the deleted virtual pin, SailWind Router creates one or more zero-length unroutes between junctions on different layers.
    - Unroutes might connect to junctions that replace the virtual pin.

Protecting a Virtual Pin

You can control the protection of virtual pins through the "Protect; disallow editing" check box in the **Virtual Pin** tab of the Virtual Pin Properties dialog box.

Tip
This check box is also toggled by the **Protect** and **Unprotect** commands in the virtual pin's popup menu.

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 then right-click and click the **Properties** popup menu item.
3. On the **Virtual Pin** tab of the Virtual Pin Properties dialog box, select the "Protect; disallow editing" check box.
4. Click **OK**.

Changing a Virtual Pin's Via Type

Change the type of via that a virtual pin uses through its Properties.

Restrictions and Limitations

- You cannot change a protected via unless you first unprotect it.

Procedure

1. Select the virtual pin, right-click and click the **Properties** popup menu item.
2. On the **Virtual Pin** tab of the Virtual Pin Properties dialog box, select the new via type from the Type list.
3. Click **OK**.

Chapter 15
Copper Planes and Copper Shapes

SailWind Router provides tools for creating and customizing the shape of copper planes and copper shapes, including merging and subtracting shapes, creating arcs, and adding or deleting voids.

Any copper planes and shapes that you create or modify in SailWind Router transfer with the design when you open it in SailWind Layout.

Migration to Copper Planes

Prior to the VX.2.4 release, PADS 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, the software 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 “Replace unused pads with antipads” setting, found in the “Flooding Options” on page 447, 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 “Connect to copper plane with thermal” check box (in the Pin, or Via Properties dialog boxes). 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 “Connect to copper plane with 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 “Flooding Options” on page 447. 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.

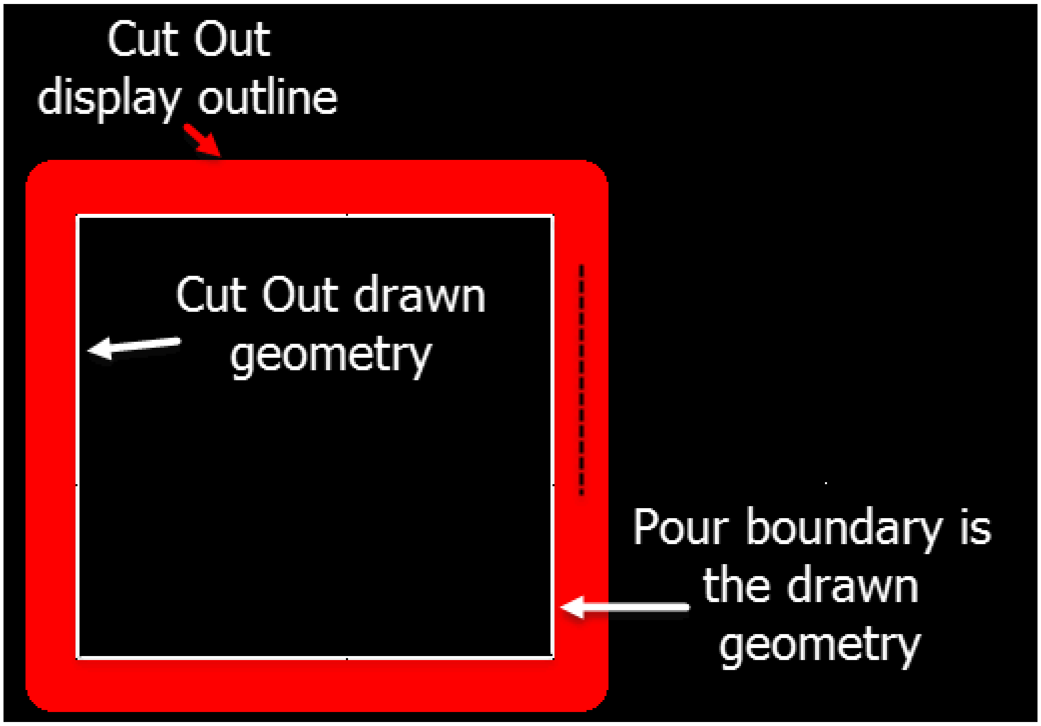
- One smoothing radius setting in the “Flooding Options” on page 447 applies to all copper planes, regardless of the layer on which they reside. Previous releases had separate smoothing radius settings for copper pours and plane areas. The copper pours value from the flooding of an existing design is retained in VX.2.4 and later releases. It may be necessary to adjust the smoothing radius value if you experience connectivity errors in VX.2.4 that were not present in the prior version.

- The setting “Remove violating thermal spokes” is removed in SailWind Layout. 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.

- When flooding drafting shapes, the copper floods to the inner edge of completed cut out outlines (not the centerline).

Figure 39. Cut Out Pour Boundary



- 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 40. Real Lower-Left Hatch Origin

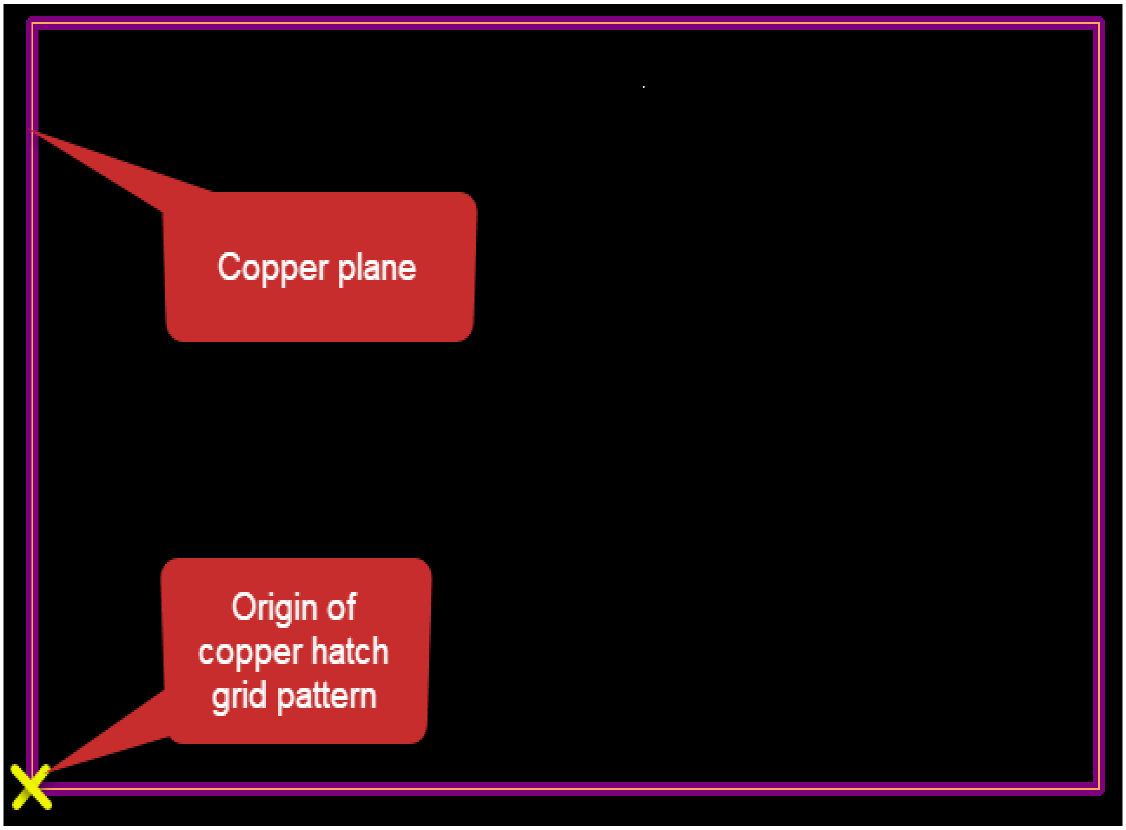
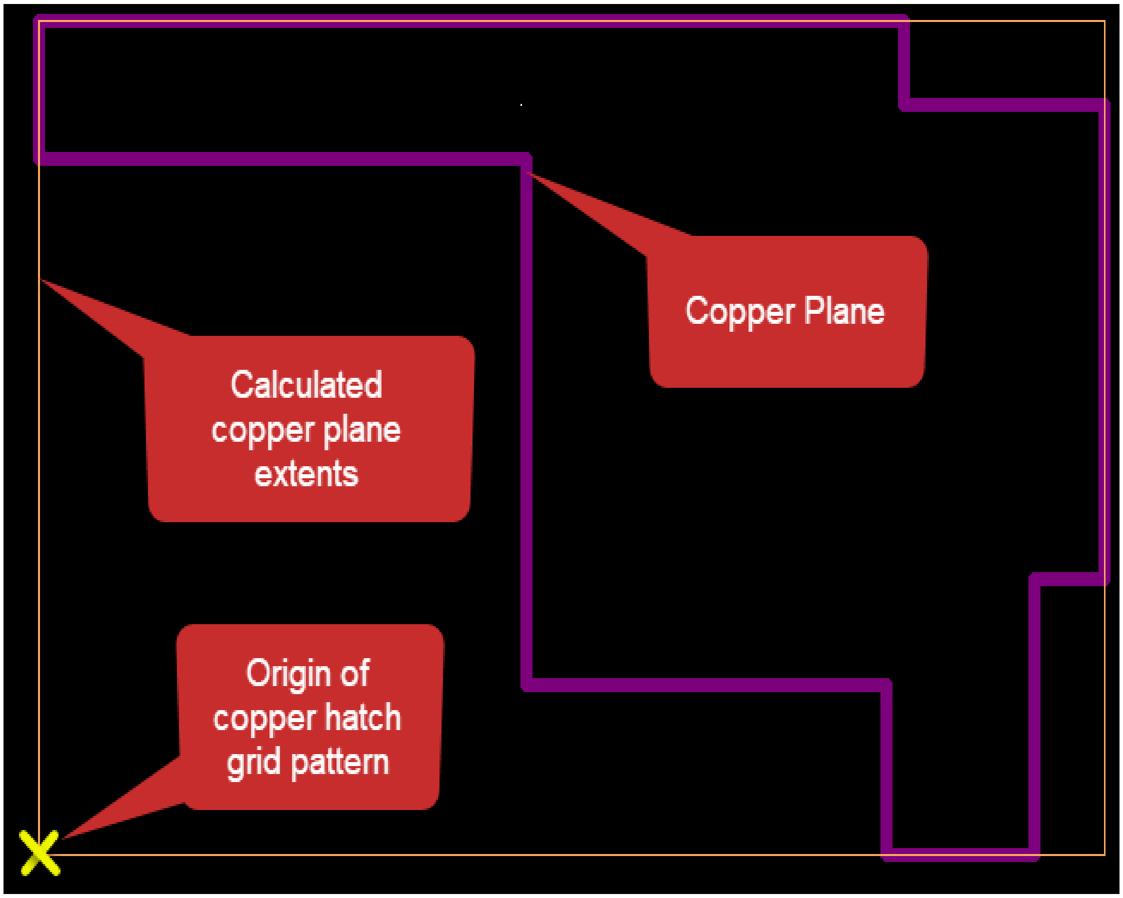


Figure 41. Calculated Lower-Left Hatch Origin



Set Drafting Options Before Creating a Drafting Object

You can set various drafting options before you begin using the Drafting Operations Toolbar to add copper shapes, copper planes, cut outs, and voids.

Set Line Width

You can use 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.
- Click either the **Copper**, **Copper Plane**, or the **Copper Plane Cutout** button: 

Right-click and choose the **Width > Set** popup menu item to type a new value.

- Set the default width value in the "Text and Lines options" on page 462. (When you set the default line width in the Options dialog box, you can also enable the software to prompt you for a net to associate with any copper shape or copper plane you create.)

Set Layer Placement

Choose one of the following methods to select the layer for the placement of your drafting object:

- Before clicking any of the drafting buttons, type L<n>, where <n> is a layer number.

- Click either the **Copper**, **Copper Plane**, or the **Copper Plane Cutout** button:

  Specify the layer using the Layer dropdown list on the Standard Toolbar.

Note:
You can change the layer of a drafting object after creating it by selecting the object and choosing a new layer from its Properties dialog box or by choosing a layer from the Layer dropdown list. All active segments of the drawing change to the new layer selection.

Set Corner Angle

Set the corner angle after clicking either the **Copper**, **Keepout**, **Copper Plane**, or the **Copper Plane Cut Out** button.

Before you begin drafting, right-click and choose either the **Orthogonal**, **Diagonal**, or **Any Angle** popup menu item. If desired, you can right-click and switch angle settings at any time while drafting a shape. This capability allows you to change drafting angles multiple times for a single shape.

Edge Precision of Drafting Shapes

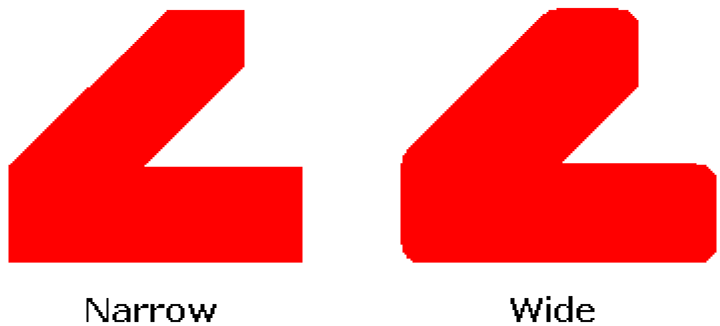
You often need to precisely control the edge of drafting shapes, as many factors contribute to that edge.

Line Width

The precision of edges is controlled by the line width used to create the shape. The default line width is set in 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.

Note:
In some cases, setting the line width too wide results in creation of a self-intersecting polygon. For more information, see "Self-Intersecting Polygons" on page 237.

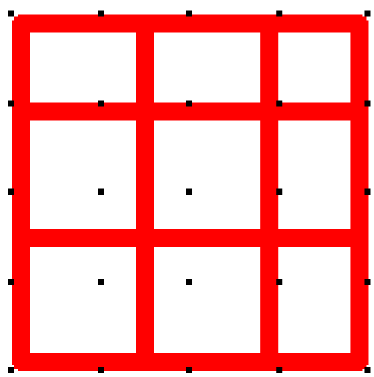
Figure 42. Narrow Versus Wide Outline Width



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.

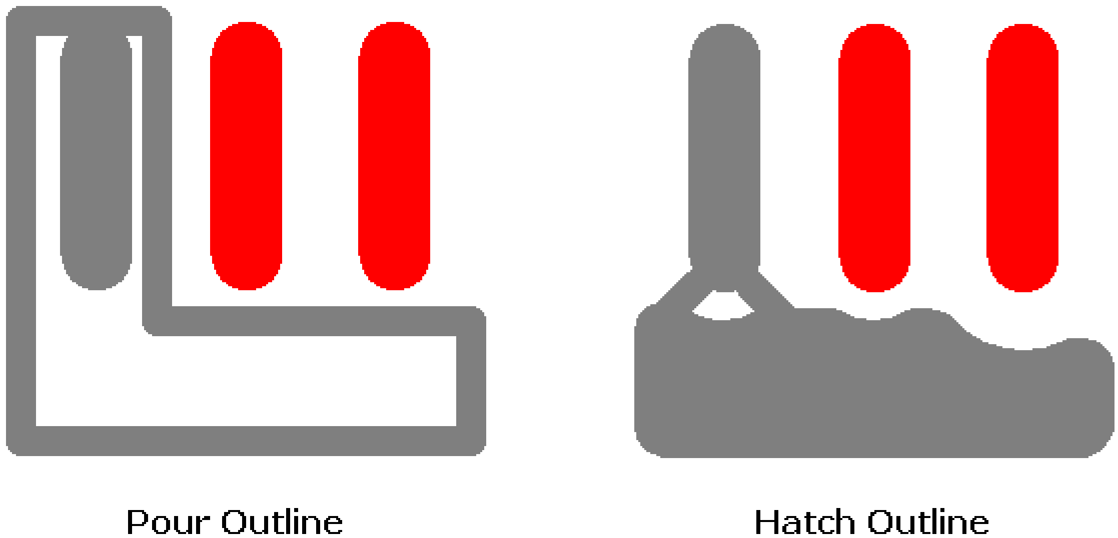
Figure 43. Hatch Grid



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 44. Clearance Rule Effect on the Hatch Outline



Creating a Copper Shape

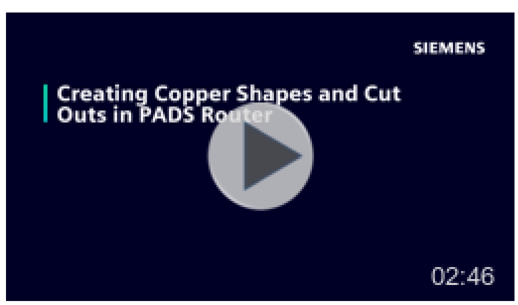
Copper is used to create heat sinks, shielding, and net bridges. You can assign nets to copper shapes.

Prerequisites

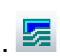
- Ensure you select a layer for placement of the copper. You cannot create a copper shape on all layers (layer number zero). If you need the same copper shape on many layers, copy the shape to other layers in SailWind Layout (SailWind Router does not provide copying capabilities).

Video

The following video demonstrates fundamental steps for creating and editing copper shapes.



Procedure

1. On the Drafting Operations toolbar, click the **Copper**button. 
2. Right-click and click a command to change the values of the drafting object as necessary.
3. Create the shape using one of the following:

- Creating a Circle Drafting Object
- Creating a Polygon Drafting Object
- Creating a Rectangle Drafting Object

4. After completing the shape, if you have selected the "Prompt for net name at completion of copper" check box in the Options dialog box (**Text and Lines** category), the "Copper Properties dialog box" on page 471 displays. Make changes to the copper properties.

   For example, you can assign a net by selecting the net name from the Net list. You can also specify the line width, the layer on which the copper is located, and whether or not the copper shape is solid (filled).

   Note:
   Select the "Solid" check box in the Copper Properties dialog box to enable or disable solid fill of the shape. If you clear the check box, whenever you enable copper or copper plane hatching to display, the copper drafting object fills according to the settings in the **Global** category > **General** subcategory of the "Options dialog box" on page 441.

5. Click **OK** to close the Copper Properties dialog box.
6. Verify the resulting shape displays as expected.
   - If the shape edges are not correct, see "Edge Precision of Drafting Shapes" on page 224.
   - If the shape needs to be modified, see "Drafting Object Properties" on page 244.
   - If you want to start over, see "Deleting a Drafting Segment or Object" on page 245.

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.

SailWind Router does not provide a dedicated copper cut out button. Create cut outs by subtracting one copper shape from another.

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.
- You must be in Pour Outline mode (modeless command PO) to select or modify copper shapes.

Prerequisites

- You must put the cut out on the same layer as the copper.

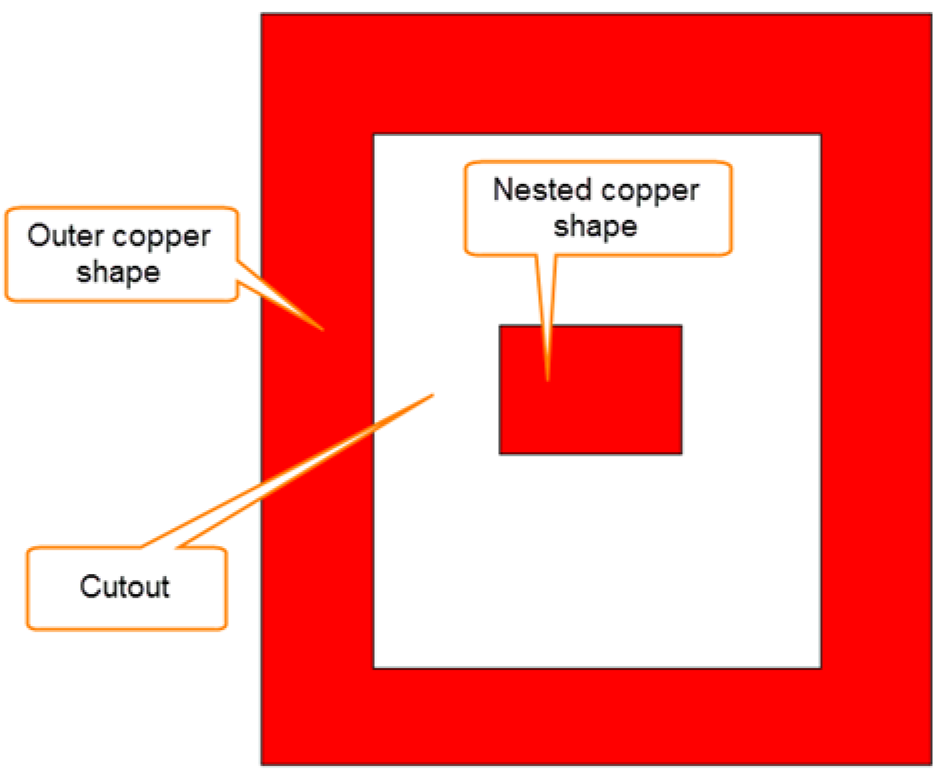
Procedure

1. On the **Drafting Toolbar**, click the **Copper** button. 
2. “Create one or more cut out shapes” on page 226 for the copper. 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.
4. Select the copper shape where you want to create the cut out.
5. Click the **Subtract Shapes** button. 
6. Select the shape that you want to use as the copper cut out. The software removes copper inside the selected cut out area.
7. Verify the resulting cut out displays as expected.

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, subtracting it from the outer shape, and then creating the new nested copper shape inside the copper cutout.

Figure 45. Example Nested Copper Shape



Prerequisites

- You must be in Pour Outline mode (modeless command PO) to modify or select copper shapes.

Procedure

1. In the outer copper shape, "create a copper cut out" on page 227 of appropriate size and shape; then subtract it from the outer shape by clicking the **Subtract Shapes** button.  See "Creating a Copper Cut Out" on page 227 for more information on subtracting shapes.
2. Create the new nested copper shape on page 226 within the copper cut out.

Results

The new nested copper shape displays inside the cut out of the outer shape.

Filling a Shape with Solid Copper

If desired, you can fill a shape with solid copper. The copper shape appears solid when you enable the Pour Outline mode (modeless command PO).

Procedure

1. Select a copper shape, right-click, and then click the **Properties** popup menu item.
2. In the "Copper Properties dialog box" on page 471, select the "Solid" check box.

   If the check box is selected, whenever you enable the display of copper hatching, the copper shape displays as solid. When the check box is cleared, and you enable hatching, the copper drafting object fills according to the settings in the **Global** category > **General** subcategory of the "Options dialog box" on page 441.

Tip
Solid copper uses trace clearance rules.

Creating a Copper Plane or Cut Out Manually

Use a copper plane to create a copper area that avoids objects not connected to the net assigned to the copper plane and makes thermal connections to objects that are connected to that net.

Copper planes are insulated from traces or pins that do not share the net name. Flooded copper planes create isolation areas around non-connecting pins and traces automatically.

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 plane. See “Creating Voids in Copper Planes” on page 246 for instructions on creating voids.
- If you create a copper plane cut out, you must put the cut out on the same layer as the copper plane.

Video

The following video demonstrates fundamental steps for creating and editing copper planes.

Procedure

1. On the Drafting Toolbar, click the **Copper Plane** or the **Copper Plane Cut Out** button. 

   The add command for that object type attaches to your pointer.

2. Right-click and click a command to change the values of the drafting object. For more information, see Set Drafting Options Before Creating a Drafting Object.

3. Create the shape using one of the following methods:

   - Creating a Polygon Drafting Object
   - Creating a Circle Drafting Object
   - Creating a Rectangle Drafting Object

4. In the “Copper Plane Properties dialog box” on page 475 that appears, assign a net by selecting the net name in the Net list and any other appropriate changes to the copper plane properties. 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.

5. Click **OK**.

6. Review the resulting shape.

- If the shape edges are not correct, see “Edge Precision of Drafting Shapes” on page 224.
- If the shape needs to be modified, see “Drafting Object Properties” on page 244.
- If you want to start over, see “Deleting a Drafting Segment or Object” on page 245.

Creating a Polygon Drafting Object

Use the polygon drafting tool to draw the shape of a copper plane, void, or cut out.

Procedure

1. On the Drafting Toolbar, click the **Copper**, **Keepout**, **Copper Plane**, or the **Copper Plane Cut Out** button. 

   The add command for that object type attaches to your pointer.

2. Right-click and choose the **Polygon** popup menu item.
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; then click to draw the segment.
   - Type SR<x> <y> if you want to specify relative coordinate values for each point and corner; then click to draw the segment.

   Tip
   Right-click and choose 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 Esc key to cancel the operation.

4. Double-click or right-click and choose the **Complete** popup menu item to end the shape.

Related Topics

Creating a Copper Shape

Creating a Copper Plane or Cut Out Manually

Creating a Circle Drafting Object

Use the Drafting Toolbar to create circular copper planes, voids, and cut outs.

Procedure

1. On the Drafting Toolbar, click the **Copper**, **Keepout**, **Copper Plane**, or the **Copper Plane Cut Out** button. 

   The add command for that object type attaches to your pointer.

2. Right-click and choose the **Circle** popup menu item.

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 at any time.

4. Click or press the spacebar to indicate the circle’s radius to complete the circle definition.

Related Topics

Creating a Copper Shape

Creating a Copper Plane or Cut Out Manually

Creating a Rectangle Drafting Object

Use the Drafting Toolbar to create rectangular copper planes, voids, and cut outs.

Procedure

1. On the Drafting Toolbar, click the **Copper**, **Keepout**, **Copper Plane**, or the **Copper Plane Cut Out** button. 

   The add command for that object type attaches to your pointer.

2. Right-click and choose the **Rectangle** popup menu item.

3. Click to indicate one corner of the rectangle.

4. Click to indicate the location of the diagonally opposite corner.

5. As an alternative to using the pointer, you can 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.

Related Topics

Creating a Copper Shape

Creating a Copper Plane or Cut Out Manually

Merge Shapes

Merge Shapes is used to combine two or more shapes (coppers, copper planes, or copper plane cutouts) into a single merged shape. Each location where a shape intersects another shape results in a single shape that expands to include the additional shapes.

The shapes merge into a single shape, which inherits the properties (such as width, net name, and fill type) of the last shape selected. Additionally, if you merge a copper plane with a copper plane cutout, then the merged shape is of the same type of the last shape selected. For example, if you select the copper plane cutout last, the resulting merged shape is a copper plane cutout.

Prerequisites

- You must select two (or more) shapes before merging shapes.
- You must be in Pour Outline mode (modeless command PO) to select and modify copper shapes. Use the modeless PO command to toggle the setting.
- You cannot merge copper shapes with copper planes.
- The shapes need to reside on the same layer.

Note:
If two or more shapes are selected and they are assigned to different net names then the merge is not completed.

- The shapes need to intersect at one (or more) locations to merge.
- You cannot merge shapes if they belong to a physical design reuse.

Procedure

1. Use Ctrl-click to select two or more existing shapes.
2. On the Drafting toolbar, click the **Merge Shapes** button, or right-click and click the **Merge Shapes** popup menu item.

   The selected shapes are merged into a single shape.

Related Topics

Creating a Copper Shape

Creating a Copper Plane or Cut Out Manually

Subtract Shapes

Subtract Shapes

Subtract Shapes is used to subtract one shape from another resulting in a single merged shape. This is not restricted to copper plane cutout shapes; a copper plane can be subtracted from a copper plane or a copper shape can be subtracted from another copper shape for ease of use.

Restrictions and Limitations

- A shape subtracted from another shape that completely severs the first shape results in two separate shapes.
- A shape completely within a plane which is subtracted is treated as a void and combined with the plane so that it moves with the plane (unlike a cutout which tracks separately). Subtracting the shape may not be the best choice in this case and it may be better to keep it as a cutout so the void shape can be more easily moved or copied. The copper plane will have an opening in the flooded shape either way.
- You must be in Pour Outline mode (modeless command PO) to select and modify copper shapes. Use the modeless PO command to toggle the setting.
- You cannot subtract a copper shape from a copper plane or a copper plane from a copper shape.
- You cannot subtract shapes if they belong to a physical design reuse.
- If subtracting a copper shape from another results in two or more separate copper shapes, each shape receives a new name and a new origin.

Prerequisites

- The shapes need to reside on the same layer.
- The shapes need to intersect.

Procedure

1. Select a single copper, copper plane, or copper plane cutout shape.
2. On the Drafting toolbar, click the **Subtract Shapes** button or right-click and click the **Subtract Shapes** popup menu item.

   The base shape (the shape from which you intend subtract another shape) displays with a colored dashed outline. Other shapes that are available for selection appear with a solid colored outline, while unavailable objects in the design change to gray scale, thus making it easy to discern between objects.
3. Select another single shape to subtract from the first shape.

   The function does not allow multiple shapes to be selected as subtraction operations can result in different effects depending on the order of subtraction.

   The selected shape is subtracted from the first shape. Repeat to subtract additional shapes as required.
4. When you have finished subtracting shapes press the Esc key or right-click and click the **Cancel** popup menu item to end the procedure.

Related Topics

Creating a Copper Shape

Creating a Copper Plane or Cut Out Manually

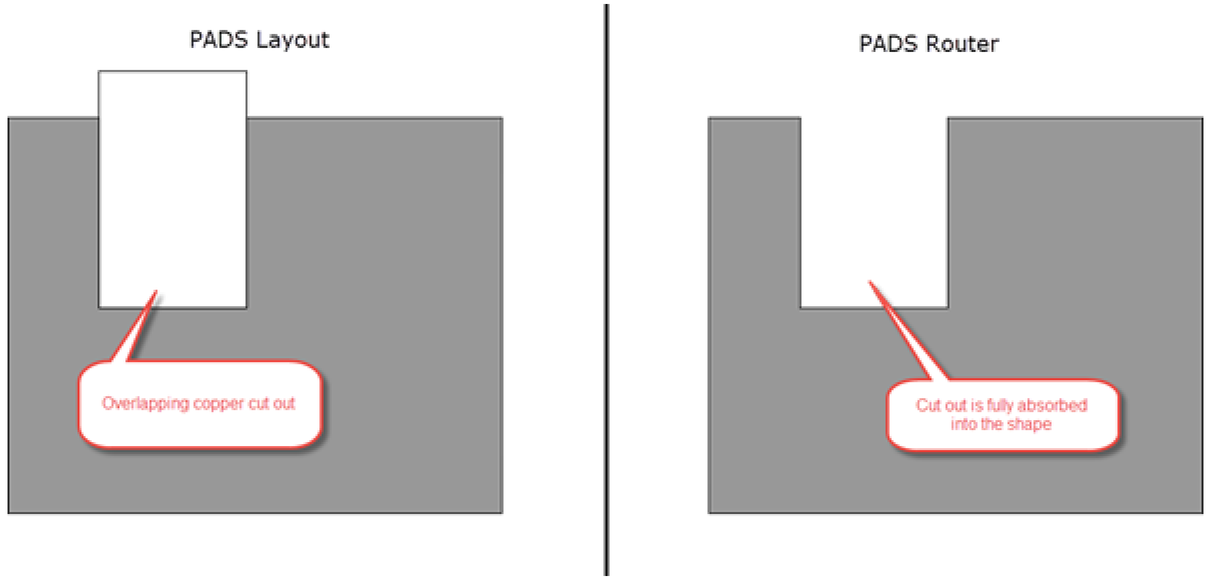
Merge Shapes

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 after you, they are no longer cut outs. Instead, the software absorbs them into the copper shape.

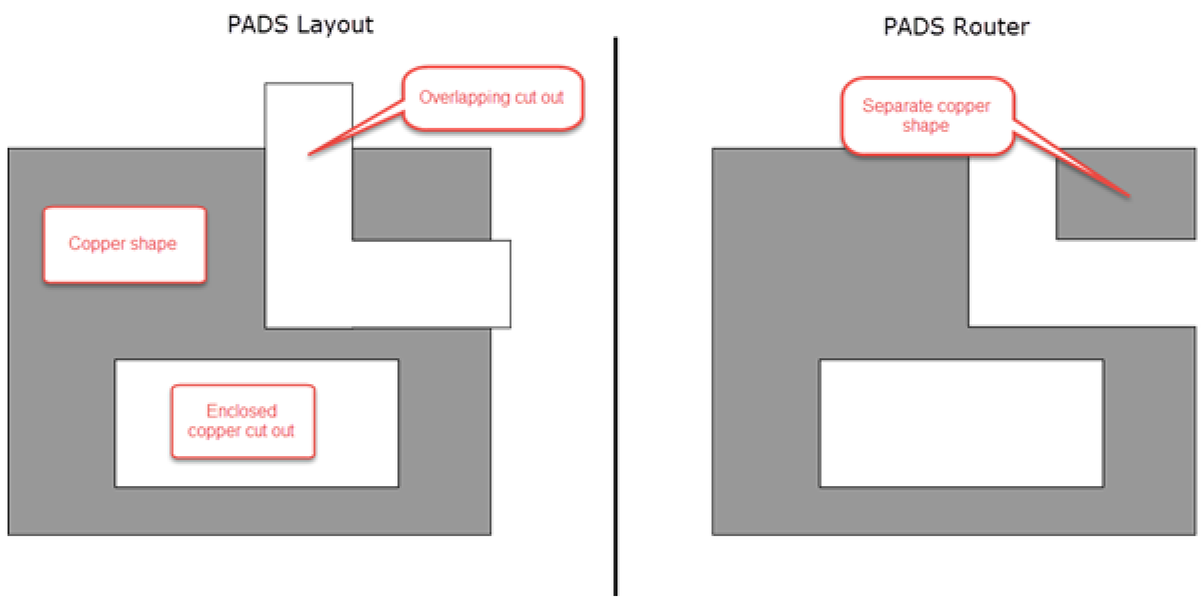
In some cases, copper cut outs are completely enclosed within the confines of a copper shape. If the copper cut out extends beyond the boundaries of the copper shape, however, and you merge the cut out with the copper shape in SailWind Layout (by right-clicking and clicking the **Combine** popup menu item), when you move the design over to SailWind Router, the software “absorbs” it into the outline of the copper shape.

Figure 46. 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 47. 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.

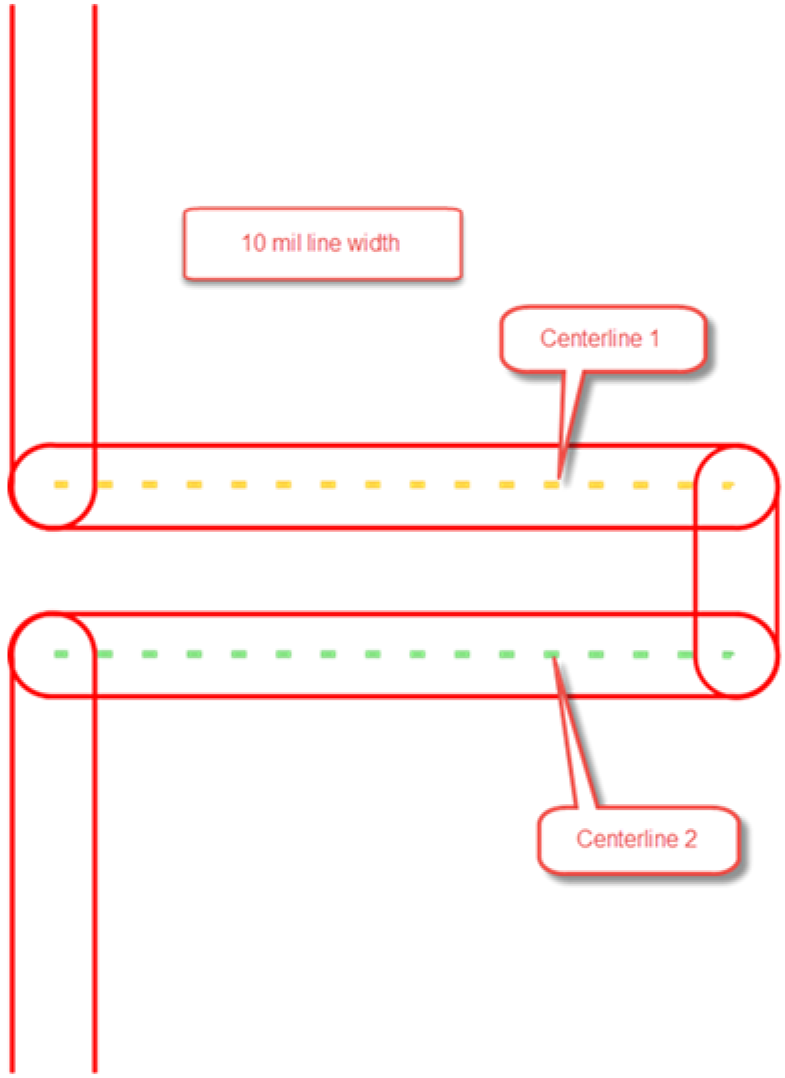
Note:
If SailWind Router is running in the foreground, the link to the report displays in the SailWind Router Output Window. If SailWind Router is running in the background (for example, during an autoroute routine or during Latium Design Verification), the link to the report displays in the SailWind Layout Output Window.

Self-Intersecting Polygons

A self-intersecting polygon occurs whenever a portion of a shape outline overlaps with another portion of the same shape. Giving a polygon a large outline width can cause the outline to overlap in some portions.

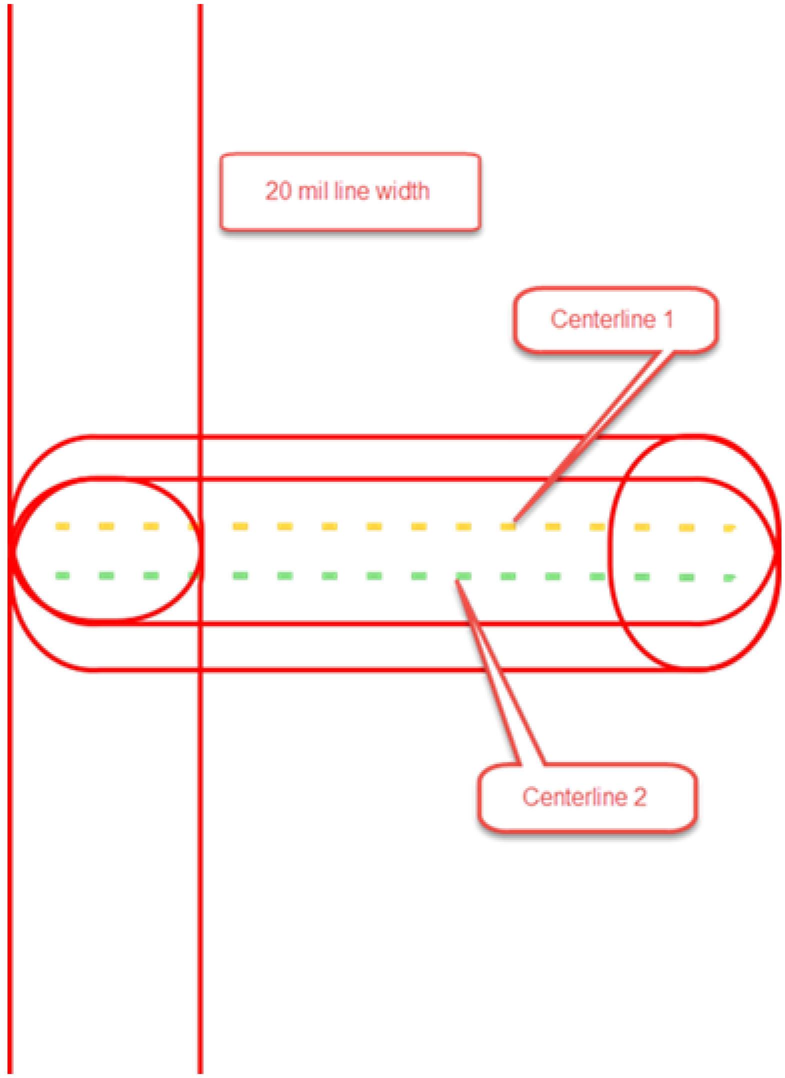
When drafting copper planes and shapes using polygons, you can set the outline line width in the Copper Plane Properties dialog box on page 475. Choosing a narrow line width enables you to create shapes with greater accuracy in detailed areas, as shown in Figure 48.

Figure 48. Close up of Shape Detail



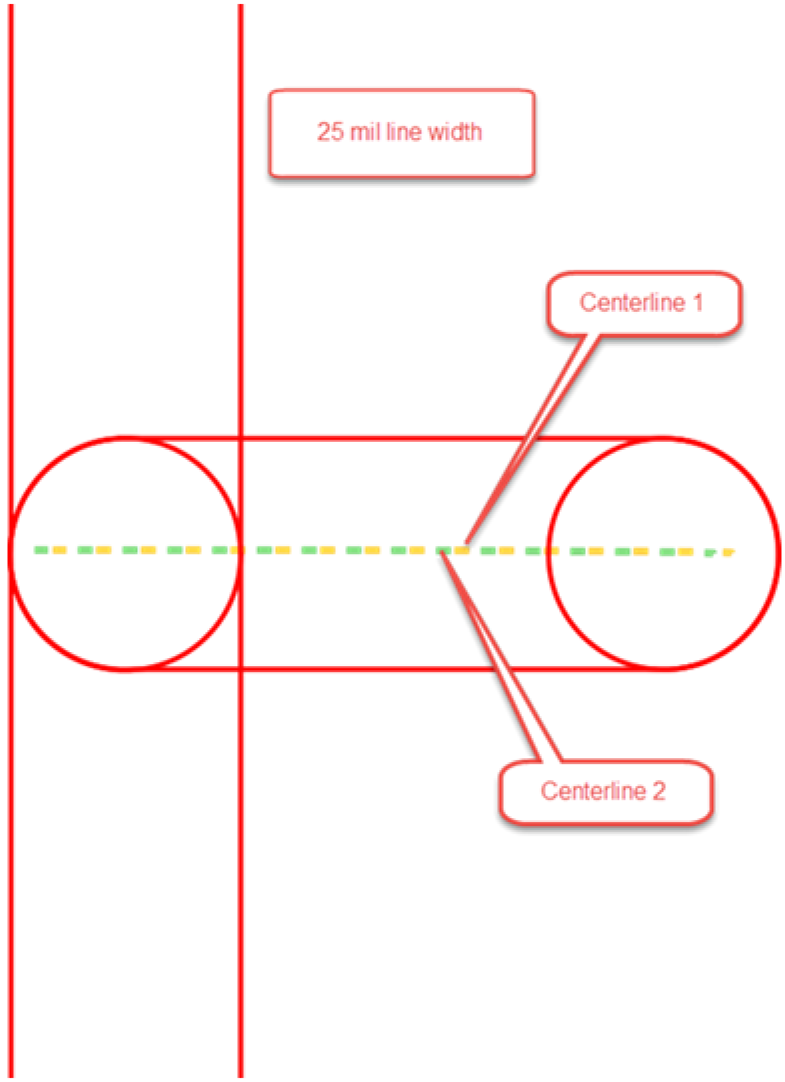
You can set the line width as required for your design; however, as the width increases, the shape resolution decreases. Referring to the example in the preceding figure, as the outline line width of a copper shape increases, the outer edge of the lines remain in place and the inner edges expand inward. The result appears in Figure 49.

Figure 49. 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-intersecting polygon.

Figure 50. 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.

Modification of Drafting Objects

After adding drafting objects to a design, you can move, edit, or delete them as your design needs change.

The following video demonstrates fundamental steps for modifying drafting objects.

Moving a Drafting Object

You can move a whole drafting object while maintaining its shape.

Restrictions and Limitations

- You cannot modify, move, or delete a polygon that is part of a physical design reuse.

Procedure

1. Select the drafting object to move using one of the following methods:

> **Note:**
> You must turn off flooding before you can select a shape.

- With nothing selected, right-click and choose the **Select Shapes** popup menu item; then click on a segment of the object. (The ***Select Shapes*** selection filter setting is the best filter for selecting a drafting object.)

- Select a segment of the object, right-click, and then choose the **Select Shape** popup menu item.

- Shift-click a segment of the object.

2. Right-click and choose the **Move** popup menu item. Alternatively, you can drag the selected object, if allowed by your Drag moves setting on page 441.

   The shape attaches to the cursor by the Move preference setting on page 441.

3. If you need to make additional changes to the drafting shape before you move it, right-click and choose the **Rotate 90**, **Rotate Group 90**, **Spin**, **Flip Horizontal**, or **Flip Vertical** popup menu items.

4. Move the drafting object to its new location and click to place it.

Modifying Drafting Object Segment Properties

You can edit or change the properties of copper shape, keepout, copper plane, and copper plane cut out segments.

Procedure

1. Right-click and click **Select Anything** as your selection filter.

2. Click a copper, keepout, copper plane, or copper cut out segment to select it.

3. Right-click and choose the **Properties** popup menu item. The “Copper Segment dialog box” on page 478, “Keepout Segment Properties dialog box” on page 496, Copper Plane Segment dialog box, or the Copper Cut Out Segment Properties dialog box on page 474 opens, displaying the drafting object type, coordinates, and dimensions.

   Tip
   Some of the options in this dialog box are unavailable if the segment is part of a physical design reuse.

4. Modify the segment coordinates and arc settings as desired; then click **OK**.

Modifying Drafting Corner Properties

You can edit or change the properties of drafting object corners, such as location.

Procedure

1. Right-click and click **Select Anything** as your selection filter.

2. Select a corner of a drafting object, right-click, and then choose the **Properties** popup menu item.

3. In the X and Y boxes, type the X and Y location of the drafting corner.

4. Click **OK**.

Setting the Origin of a Drafting Object

You can set the origin of a drafting object to move it or place it.

Procedure

1. Select the whole drafting object.

   > **Tip**
   > Right-click an object and click the **Select Shape** popup menu item or press the Shift key and click a segment to select the whole drafting object.

2. Right-click and click the **Set Origin** popup menu item.

3. Click to set the new origin point. A message displays, prompting you to confirm the origin point.

4. Click **Yes** to change the origin point or click **No** to use the previous origin point.

5. Press the Esc key to exit the Set Origin mode.

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 object.

Changing Line Widths
Changing Layers

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 choose the **Properties** popup menu item.
2. Type a new line width value in the Width box of the Properties dialog box.
3. Click **OK**.

Changing Layers

You can change the layer on which the drafting object exists.

Procedure

1. Select a drafting item, right-click, and then choose the **Properties** popup menu item.
2. Select a new layer from the “Located on layer” dropdown list of the Properties dialog box.
3. Click **OK** The drafting item moves to the new layer.

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 **Pull Arc**.
2. The segment or corner attaches to your pointer and changes to an arc.
3. Click to indicate the new position for the arc.

Tip
You cannot modify, move, or delete a polygon that is part of a physical design reuse.

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" on page 249.
2. Choose one of the following to delete the segment or object:

   - Press the Delete key
   - Choose the **Edit > Delete** menu item
   - Right-click and choose the **Delete** popup menu item.

Tip
You cannot modify, move, or delete a polygon that is part of a physical design reuse.

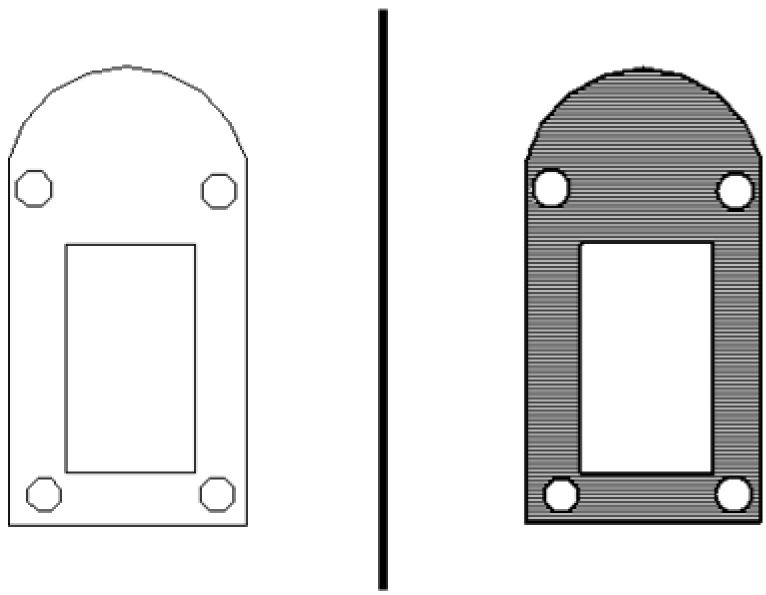
Copper Voids

Voids in copper planes represent gaps or missing areas in the copper fill. You can create and delete voids as necessary to accommodate any changes required to the copper plane.

A void is an integral part of the copper plane, and it behaves as part of the plane. If you move the plane, for example, the void moves with it. Conversely, a copper plane cut out is separate from the plane. Although you can select and move a cut out, you cannot move it by moving the copper plane.

Voids and cut outs appear the same when you flood the copper plane.

Figure 51. Voids in a Copper Plane Before and After Flooding



You create a void in a copper plane by subtracting one enclosed polygon from within another. You can remove a void later by deleting it. For more information, see “Creating Voids in Copper Planes” on page 246.

If you subtract a copper plane cutout from a copper plane, it becomes a void.

Creating Voids in Copper Planes

You can subtract a copper plane from another copper plane to create a void. You can also delete that void later if desired.

Prerequisites

- Place the software in copper plane outline mode by turning off copper plane hatching (using the Pour Outline (PO) command). If you do not turn off copper plane hatching, the shapes you create within a copper plane appear to be voids but the software treats them as cut outs instead.

Video

The following video demonstrates how to create copper planes and voids.

Procedure

1. Create the outline of the void by drawing another copper plane or cut out within the boundaries of the first copper plane. You can create a polygon, circle, or rectangle by right-clicking and choosing the desired shape from the popup menu.

   Note:
   You must draw the void within the boundaries of the first copper plane or the software cannot qualify it as a void.

   ---

   a. In the Copper Plane Properties dialog box, ensure the shape of the intended void is on the same layer as the copper plane.

   b. Click **OK**.

2. Press the Esc key to exit the copper plane drawing mode.

3. Click the outline of the copper plane to select it; then, on the Drafting Operations toolbar, click the **Subtract Shapes** button.

   Alternatively, right-click and choose **Subtract Shapes** from the popup menu item.

Either method places the software in shape subtraction mode, as indicated by a dashed line highlighting the copper plane.

4. Click the shape you want to use as the void. You can select more than one shape.
5. Press the Esc key to exit the shape subtraction mode when you are finished selecting shapes.
6. Enable copper plane hatching using the Pour Outline (PO) command and verify the void is formed properly.

Related Topics

Deleting Voids in Copper Planes

Deleting Voids in Copper Planes

You can remove voids from copper planes.

Prerequisites

- You have already created a void in a copper plane.

Procedure

1. Turn off the copper plane hatching using the Pour Outline (PO) command.

   Note:
   You must turn off copper plane hatching to select shape segments.

2. Select one or more segments of the void, and then right-click and choose the **Delete Void** popup menu item.

Related Topics

Creating Voids in Copper Planes

Creating Nested Copper Planes

You can create a copper plane within another “outer” copper plane. You create a copper void inside the outer shape and then create the new nested copper plane inside the copper cut out.

Note:
You can alternatively create a distinct, nested shape by giving it a different flooding priority than the underlying copper plane. See “Options Dialog Box, Flooding Category” on page 447.

Procedure

1. In the outer copper shape, “create a copper void” on page 246 of appropriate size and shape within the outer shape.
2. Create the new nested copper plane on page 229 within the copper void.

Results

The new nested copper plane appears inside the void of the outer copper plane.

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.

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 choose the **Select Anything** popup menu item.
2. Click the outline of the shape.
3. Right-click and choose a command from the popup menu to alter the segment as needed.

Selecting Whole Drafting Objects

Select the outlines of a drafting object to move the object or change its properties.

Note:
You must turn off copper flooding to select shapes and shape segments.

Use one of the following methods:

- With nothing selected, right-click, choose the **Select Shapes** popup menu item, and then click the object.
- While in Select Anything mode, shift-click along the outline of the object.
- While in Select Anything mode, drag a selection box over the entire outline of the object.

Chapter 16
Routing Setup

An important step before routing your design involves optimizing the topology and net distribution, and setting routing restrictions. Also important is defining how you want to add fanouts and test points to the design.

Routing Settings

SailWind Router offers a broad selection of setup options and constraints for configuring your design to achieve exceptional routing results.

Performing Length Minimization

Length minimization examines the design and assesses the current topology settings of the nets. Using these settings, SailWind Router reorders the nets to establish the shortest length between each of the connection points.

Procedure

1. Click the **Tools > Length Minimization** menu item.
2. Length minimization runs with the default topology type or the topology type you set on the **Topology** tab of the Properties dialog box.

Related Topics

Properties Dialog Box, Topology Tab

Rescheduling Nets

Net rescheduling lets you change distribution of unroutes without using SailWind Layout ECO operations. You can move one end of an unroute from the current source pin or virtual pin to another pin. You can reschedule nets even if a net topology has a type of “Protected” as assigned in the **Topology** tab of the Design Properties or Net Properties dialog boxes.

CAUTION:
The net must contain unroutes to reschedule it.

Restrictions and Limitations

You cannot reschedule the following unroutes:

- Unroutes connected to copper planes or T-junctions.
- Unroutes that span subnets.
- Unroutes belonging to differential pairs. You can reschedule differential pairs only if an unroute exists in each pair member.

Procedure

1. Select an unroute then right-click and click the **Reschedule** popup menu item.

   Everything in the workspace dims except the unroutes, pins, virtual pins, and protected vias that are valid end points for rescheduling.

2. Point to the pin where you want to reconnect the unroute. You can reconnect to pins or protected vias if they belong to the subnet of the selected unroute and if the connection maintains net integrity.

   When you find a valid connection point, the pointer graphic changes to a double-circle with a cross hair through it. In addition, a ToolTip appears informing you that the pin is available for connection —or if not, why.

3. Click to complete the rescheduling.

4. When you complete rescheduling, the net automatically switches to a Protected topology type (on the **Topology** tab of the Net Properties dialog box).

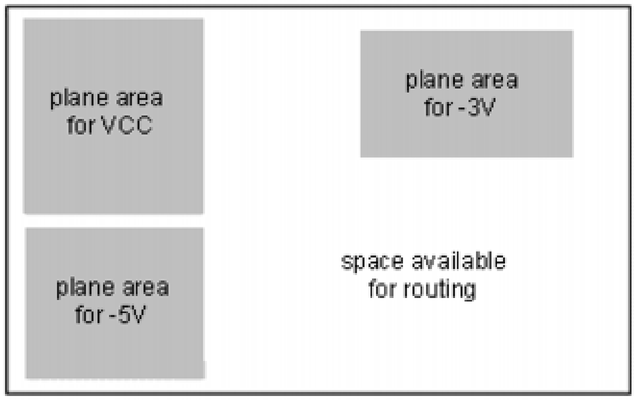
Tip
When you reschedule an unroute in a differential pair, a parallel rescheduling is performed in the other member of the pair. You can click **Undo** on the standard toolbar to undo this parallel rescheduling.

Split/Mixed Plane Layer Setup

SailWind Router routes on split/mixed plane layers only if the layer has an open area not covered by a plane area.

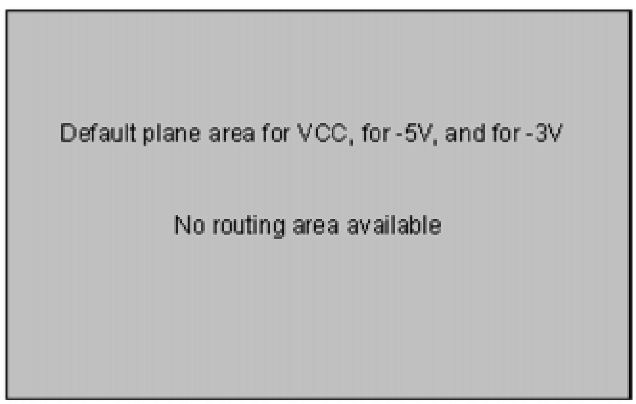
Scenario 1

All nets assigned to the layer have a plane area polygon. Open space is available for routing on the layer.



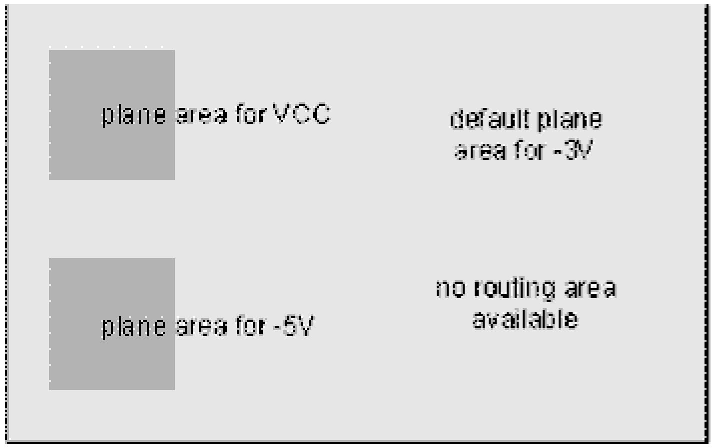
Scenario 2

No nets assigned to the layer have a plane area polygon. *No open space is available for routing on the layer*. The entire layer is reserved for plane areas. Pre-routing analysis posts the following warning to the Output window in this situation: “Split/mixed layers <named layers> have automatically created plane areas. SailWind Router will route only assigned nets on these layers. To correct the problem, use SailWind Layout to define plane area polygons.”



Scenario 3

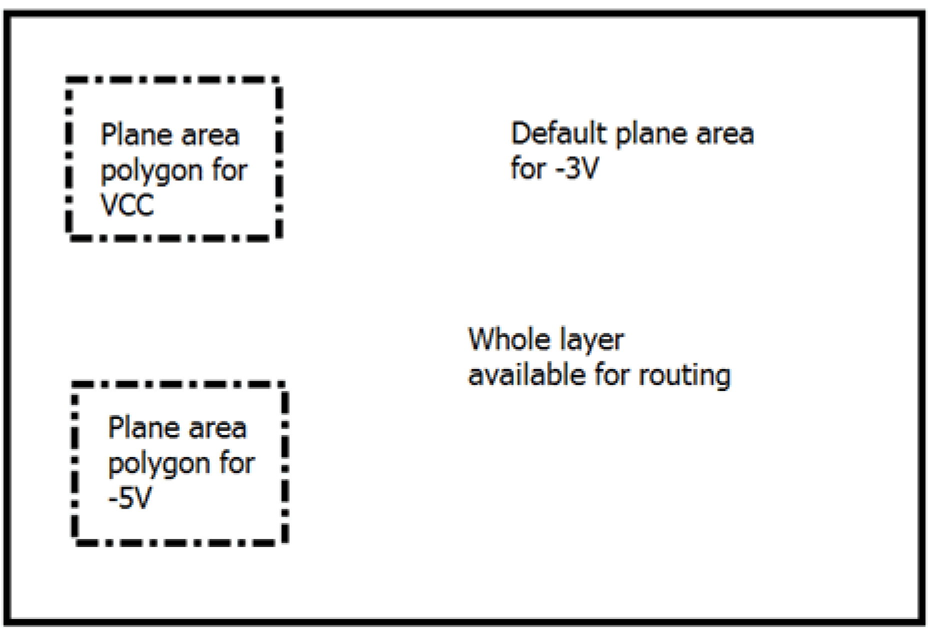
Some nets, but not all, that are assigned to the layer have plane area polygons. *No open space is available for routing on the layer*. The space without plane area polygons is reserved for plane area polygons of the nets assigned to the layer that do not yet have polygons. Pre-routing analysis posts the following warning to the Output window in this situation, “Split/mixed layers <named layers> have automatically created plane areas. SailWind Router will route only assigned nets on these layers. To correct the problem, use SailWind Layout to define plane area polygons.”



Note:
Nets with polygons route to the plane area. The nets with no polygons route to the plane *layer* as if the entire layer is one shared plane area. Therefore, a pin from a net with no polygon can be fanned out to a point inside a plane area polygon defined for another net. While nets will not short because the plane processing in SailWind Layout prevents it, this fanning out can lead to isolated thermals.

Scenario 4

Nets are assigned to the Split/Mixed plane layer and polygons exist on the layer, but the layer has been temporarily changed from a Split/Mixed layer to a Routing layer in SailWind Layout. The whole layer is available for routing. For instructions and tips, see "Routing on Split/Mixed Layers" on page 256.



Routing on Split/Mixed Layers

Plane areas on Split/Mixed Layers form obstacles to routing; therefore, you must take certain steps to permit routing on Split/Mixed layers.

Restrictions and Limitations

- Plane area polygons are obstacles to routing. Nets assigned to the layer without polygons flag the entire layer as a polygon. For more information see "Split/Mixed Plane Layer Setup" on page 253.
- When you define a component layer as a split/mixed plane layer, the layer will be used for connecting SMD pins, through hole pins or vias.

Procedure

1. In the Design Properties, on the **Layers** and **Layer Biasing** tabs, enable the layers for routing.
2. To permit routing on the Split/Mixed plane layers, you must do one of the following:
   a. Create plane area polygons for all nets assigned to the layer leaving open space for routing on the layer or plane area cutouts for the areas in which you want to permit routing.
   b. Temporarily set the layer as a No Plane layer in the SailWind Layout, Layer Setup dialog:
      i. On the Standard toolbar, click the **Layout** button to open the design in SailWind Layout.
      ii. Choose the **Setup > Layer Definition** menu item.
      iii. Select the Split/Mixed layer and click the **Assign Nets** button.
      iv. Because the assigned nets will be lost, record the net names assigned to the layer and then close the Plane Layer Nets dialog box. Polygon shapes do not lose their net assignments.
      v. In the Plane Type area, click No Plane and click **OK**.
      vi. On the Standard toolbar click the **Route** button to open the design in SailWind Router.
3. Route the design as required. If autorouting, it might be helpful to assign a high Cost to the Split/Mixed layer to prefer routing on other layers when possible. You set the Cost value in the Design Properties on the **Layers** tab.
4. If you followed instructions above to change the layer type to a No Plane layer, reverse that process:
   a. Open the design in SailWind Layout to set the layer as a Split/Mixed layer.
   b. Reassign the nets to the layer. The plane area polygon outlines automatically re-inherit the correct net name assignments.

Routing with Layer Restrictions

A routing-restricted layer is any electrical (but non CAM) plane layer that you disable for routing. This includes a component mounting layer specified as a split/mixed plane.

You can define routing with layer restrictions in two ways:

- Define a layer bias design rule. This rule limits the layers upon which you can route traces for nets, net classes, groups, pin pairs, or all net objects, when it is assigned as a default routing rule.
- Restrict an electrical layer in the **Layers** tab of the Design Properties dialog box.

Typically, you restrict a layer or bias net objects to avoid routing traces on specific layers. For example, you may want to disable the outer layers of the board and route traces mostly on internal layers to reduce the incidence of cross talk. You can assign layer bias rules to achieve the same result, but this limits the application to objects rather than restricting all routing from a specific layer in the design.

Related Topics

Setting Layer Biasing Rules

Setting Layer Rules

Routing with Restrictions on Component Layers

You can set up your design to impose layer restrictions that control which layers allow routing.

The following figure shows how this routing program would route a net containing only SMD pins with no layer restrictions, and how it would route the same net when you impose layer restrictions.

Figure 52. Possible Routing Patterns

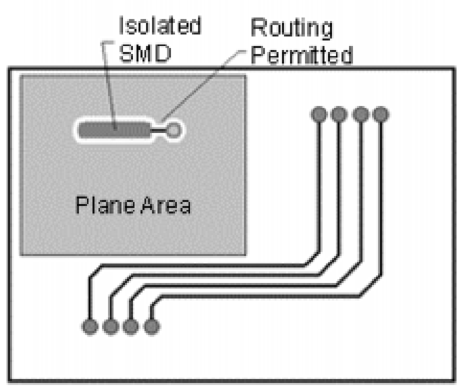
When you impose a layer restriction that excludes routing from component layers, routing is not eliminated on the component layers completely. If the restriction is respected literally, SMD pins on the restricted component layer may remain unconnected. Therefore, restrictions are respected for component layers as a soft rule to allow routing to SMD pins.

Routing on Restricted Component Layers Assigned as Split/Mixed Planes

If you choose to restrict routing on component layers designated as split/mixed planes, split/mixed plane area polygons are treated as obstacles. However, the program routes inside split/mixed plane area polygons on component layers to allow connections to SMD pins.

The following figure illustrates the principle:

Figure 53. Split/Mixed Plane with Space Available for Routing



Working with Advanced Vias

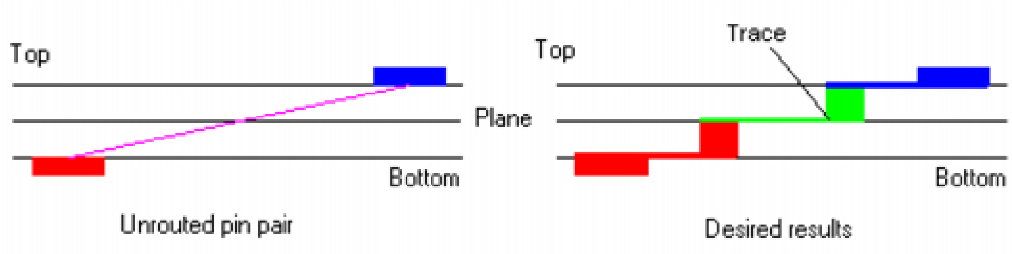
To support advanced via designs, SailWind Router routes within split plane areas on split/mixed plane layers. SailWind Router limits the trace length and routes only when blind or buried via configurations exist. Short traces route automatically between partial vias to complete routing of nets with partial vias on split/mixed planes.

When working with advanced vias:

- SailWind Router does not permit CAM planes routing layers for either autorouting or interactive routing operations.
- You can use assigned vias for routing only if these vias start or end on the split/mixed planes, but this does not guarantee that traces connecting the vias will be the shortest possible. Some other routing operations may adjust plane traces resulting in increased length.

Figure 54 shows an example design of three layers with an internal split/mixed plane layer. The design has only two blind vias that connect layer 1 to layer 2 and layer 2 to layer 3. The graphic on the left shows routing results without using partial vias and the graphic on the right shows routing results when partial vias are used. To achieve the best results, the router places a trace on the split/mixed plane layer.

Figure 54. Three-Layer Design with a Split/Mixed Plane Layer



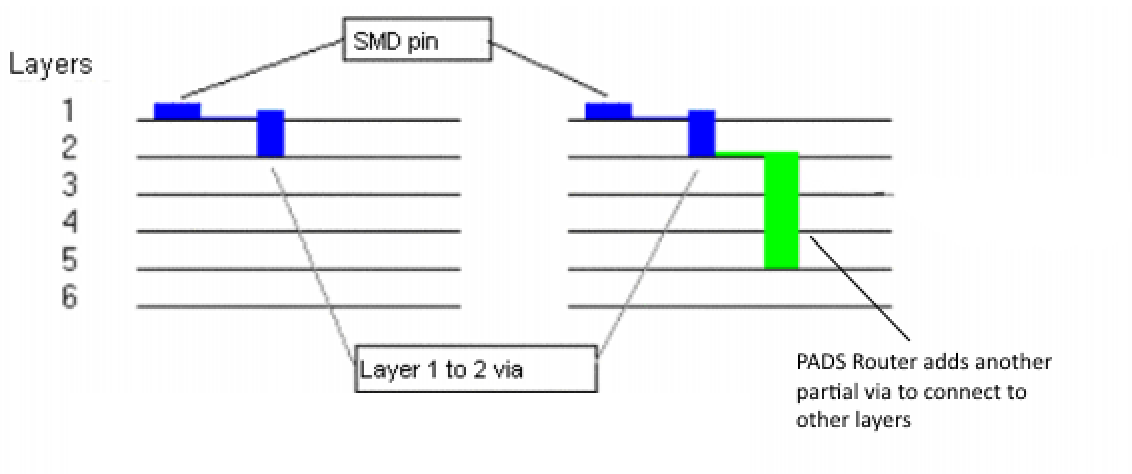
Working With Fanouts and Partial Vias

SailWind Router works well when you use through-vias during fanout operations. You can fan out SMD pins using a single through-via to provide access to the SMD on all layers.

With partial vias, however, SailWind Router does not produce proper fanouts. A single via is not sufficient for use with partial vias because only one via is added, spanning only a limited number of layers. To resolve the issue of limited access for fanout operations, the fanout autorouter pass adds more vias to enable access to all layers, except opposite layers.

For example, if the plane is on the second layer for a six-layer design, SailWind Router adds partial vias to connect the pin to the second layer and connect the second layer to all other layers except the last.

Figure 55. Six-Layer Design with Partial Vias



Tip
Any nets assigned to an internal plane layer are dispersed with partial vias, which does not add enough vias to connect to the plane layer.

Setting and Tuning Length Constraints Options

SailWind Router offers sophisticated length tuning options that allow you to create length constraints for individual nets as well as classes, groups and differential pairs.

Setting Up Routing to Length Constraints

Use routing-to-length constraints to adjust the length of length-controlled traces. SailWind Router automatically maintains length-based design rules for nets, electrical nets, classes, pin pairs, groups, and differential pairs during autorouting according to the constraint settings you set in the **Routing** category > **Tune** subcategory of the Options dialog box. The software adjusts or rips up traces during the tune pass based on their compliance with minimum and maximum trace length rules. If necessary, SailWind Router increases net and pin pair lengths to satisfy length rules by introducing accordion patterns to the trace (during the tune pass).

Tip
You can set length rules in Class, Net, Electrical Net, Pin Pair, Differential Pair, and Matched Length Properties dialog boxes.

Procedure

1. Click the **Tools > Options** menu item; then, in the Options dialog box, click the **Routing** category > **Tune** subcategory.

2. In the Minimum amplitude box, type a number to specify the minimum height or width for any accordion pattern that SailWind Router uses.

3. In the Maximum amplitude box, type a number to specify the maximum height or width of any accordion pattern that SailWind Router uses.

4. In the Minimum gap box, type a number to define the distance between accordions.

5. In the Max hierarchy level box, type a number to specify how many steps SailWind Router uses to create accordions. For additional information, see “Maximum Hierarchy Level” in the Tune Pass. on page 272

6. In the Miter ratio box, type a number to specify the miter ratio for accordion corners.

7. To use an arc instead of a diagonal segment in the accordion, click “Use arcs in miters.”

8. To determine how much extra length is added above the required matched length group tolerance (in percent of the tolerance), type a number in the Extra length added above required by matched length group tolerance box.

   Tip
   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.

9. If you need to complete traces that break length rules, click “Ignore length rules when required to complete traces.”

10. Click **OK**.

Related Topics

Add Length to Traces

Setting up Differential Pairs Tuning Options

Use the Differential Pairs area on the **Routing** category > **Tune** subcategory of the Options dialog box to control the use of differential pair accordions while tuning routing length.

Procedure

1. Click the **Tools > Options** menu item; then in the Options dialog box, click the **Routing** category > **Tune** subcategory.

2. To use an accordion to make differential pairs the same length, select the “Add diff pair correction accordions when tuning” check box.

3. If you do not want to allow correction accordions where two traces go together at the gap, select the “Do not create correction accordions in gap portion” check box.

4. Select the “Create correction accordions only when length difference is greater than the matched length tolerance” check box if you want to restrict the software to creating differential pair accordions only in instances where the length difference is greater than the matched length.

5. Click **OK**.

Chapter 17
Autorouting

Setting up a strategy of pass types and routing order ensures the best results when performing batch autorouting.

Creating a Prerouting Analysis Report

The Prerouting Analysis report provides information about design properties that may impede autorouting, such as grid settings, nets that have a disabled routing status, or pin pairs that have a shove-protected status.

Procedure

1. Click the **Tools > Pre-routing Analysis** menu item.
2. After the prerouting analysis finishes, review any errors or warnings that are sent to the **Status** tab of the Output window.

Related Topics

Reporting Design Information

Autorouting Strategy

The autorouting strategy is a set of options that define how a board should be autorouted. A strategy contains several passes. You can autoroute the entire design using a strategy, or you can select components and route them one by one using a pass type.

Note:
This section discusses autorouting the entire design and the pass types in the strategy. For more information on routing components one at a time, see Autorouting by Selection.

The autorouting strategy defines the sequential operations to perform during autorouting, including:

- Passes that the autorouter should run
- Pass types to run during autorouting
- Trace protection
- Pausing at the end of each pass
- Intensity of autorouting
- Order in which to autoroute components, nets, net classes, differential pairs, and matched length groups

You cannot save autorouting strategies. Set the strategy for each file you want to autoroute.

For more information, see "“Specifying the Passes to Route” on page 274" and "“Setting the Routing Order” on page 275."

Autorouting Pass Types

The strategy for autorouting contains several pass types. Each pass completes a specific task; it may also perform a number of subpasses.

- **Prerouting Passes** — Use these passes only when necessary because they can hinder routing completion. The default strategy starts with the Route pass and skips these prerouting passes.
    - **Fanout pass** — Only for complicated or dense components, like BGAs. If you use the Fanout pass for the entire design, the resulting fanouts occupy too much space on the board. As a result, routing becomes more difficult, more vias are placed, more time is needed to route, and completion rates drop.
    - **Patterns pass** — Only for boards with extremely regular connections. You can also use this pass to create several versions of the design, for comparison.
- **Postrouting Passes** — Center, Optimize, Miters, Test Point, and Tune are postrouting passes. The Intensity setting in the **Strategy** tab significantly affects these passes. For more information, refer to "“Autorouting Strategy” on page 263."

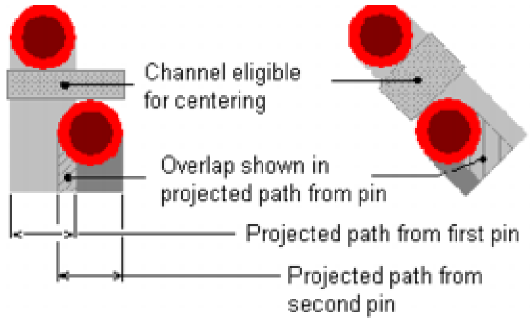
Center Pass

The center pass automatically places traces equidistant from the pads of adjacent component pins or vias to evenly distribute any available space in the channel. Centering does not violate design rules.

SailWind Router maintains differential pairs during centering. The shoulder portions of the differential pair, in the start and end zones, center as normal trace segments. The segments in the controlled gap portion center as one wide trace. Differential pairs that are split around obstacles do not center.

A channel is eligible for centering if a projected path from the pins or vias overlaps.

Figure 56. Channels Eligible for Centering



To limit the number of channels to center, set a maximum channel width. Any channel larger than this width is not eligible for centering. Set the Maximum channel width on the **Routing** tab of the Options dialog box.

The following are restrictions for the center pass:

- The Center pass only centers traces that are parallel to the channel orientation.
- The software centers traces by “stretching” the segments that are perpendicular to the channel. Trace centering does not take place if no perpendicular traces are available or if they do not have enough space to stretch.
- The Center pass does not add corners to achieve centering.
- The Center pass can only adjust traces that pass through a channel. It cannot adjust a nearby trace to accommodate centering.
- The Center pass does not adjust any angle trace segments.
- The Center pass does not maintain length rules or accordions. The Center pass adjusts trace segments that are part of accordions as regular trace segments; it does not preserve the accordion. However, it does not perform centering if it results in new errors.

For more information, refer to the Centering value in the ““Options Dialog Box, Routing Category, General Subcategory” on page 450” and ““Autorouting Strategy” on page 263.”

Fanout Pass

By performing the fanout pass, you shorten overall autorouting times and increase completion rates because this pass creates access to pins early in the process.

This pass places vias for inaccessible SMD component pins and routes from the vias to the pins. This pass locates all SMD pins in the design and routes a short trace, terminated with a via, from each pin. Set options for the fanout pass on the “Properties Dialog Box, Fanout Tab” on page 486.

Tip
Protect any fanouts you create to prevent post-fanout routing or editing operations from shoving or rerouting the fanout traces. These post-fanout operations ignore Fanout length settings in the **Fanout** tab of the Properties dialog box.

The fanout pass includes two processes: preplaced fanouts and free fanouts.

Preplaced Fanouts

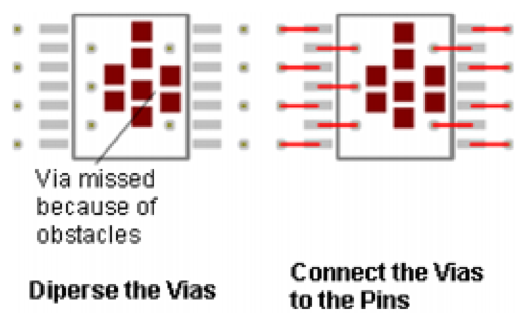
This process tries to place vias that obey the current minimum spacing rules and the via spacing you set. If the via placement violates a rule, SailWind Router does not place the via and does not create a fanout for the pin. After successfully placing vias, SailWind Router routes the fanout.

You can specify whether vias are placed on a fixed grid or spaced to accommodate one trace or two traces running between them. Set the spacing using the Via Spacing options in the **Fanout** tab of the Properties dialog box.

You can also specify whether pins and vias are shared to complete the fanout. You can fanout a number of pins by interconnecting them to a single, common via. Set this sharing feature through the Sharing options in the **Fanout** tab (of the Properties dialog box).

You can also customize the fanout pattern for vias to help avoid obstructions or unusable board areas. Set patterns in the Alignment list and Direction list in the **Fanout** tab.

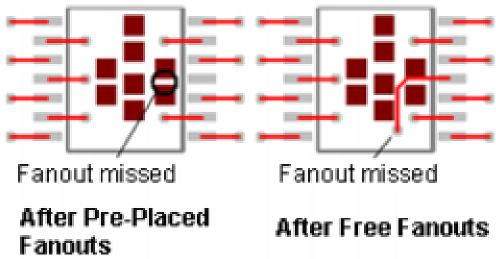
Figure 57. Customizing the Fanout Pattern



Free Fanouts

The free fanouts process completes fanouts that failed the prerouted fanouts process by searching for a path, using push and shove, and using “rip up” and “retry” features to make room for the fanout, as shown in the following figure. The free fanouts process ignores pattern settings to complete the fanout.

Figure 58. Free Fanouts Process

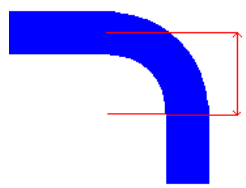


Miters Pass

The miters pass converts all route corners of a specified angle to diagonal corners. You can set Miter options on the **Routing** tab of the Options dialog box. Depending on the settings in the **Routing** tab, you can convert a corner of any degree corner to a miter.

SailWind Router adds miters to trace corners only when space permits. If a miter creates spacing or first corner rule violations, the software skips the corner and does not add a miter.

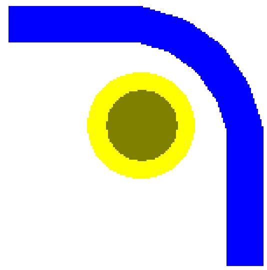
If you choose to use arcs for miters, the software creates a miter using the following calculation: <trace width> / 2 X ratio



Small segments and clearance violations provide two reasons to use arcs as miters.

If a trace has small segments, and it is impossible to create miters that are large enough, SailWind Router creates an arc for the three segments:

If it is impossible to create an arc of the required size because of a clearance violation, the software does not perform any pushing and plowing or trace adjustment. It creates a smaller arc instead:



Miters and Length Tuning

Tune operations assume traces are smoothed before you add accordions. If tuning is performed before the miter pass, the addition of miters changes the length of traces, potentially creating length violations. SailWind Router tries to add/remove length differences on the first parallel segments of modified traces or other traces belonging to the same net if it is required.

For differential pairs, miters can cause different violations on traces. In this case SailWind Router tries to adjust length of the controlled gap portion common for the both traces, as well as shoulders for each trace separately.

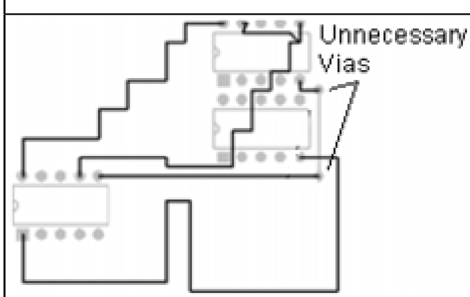
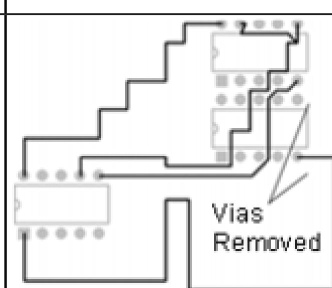
Optimize pass

The optimize pass analyzes each trace and tries to improve the quality of the route pattern by removing extra segments, reducing via usage, and shortening trace lengths. The optimize pass includes these processes: via minimize, glossing, and smoothing.

Via Minimize

The via minimize process reduces via usage in the design by ripping up existing traces and finding a new path that uses fewer vias, as shown in the following figure.

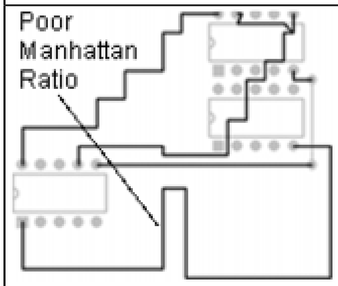
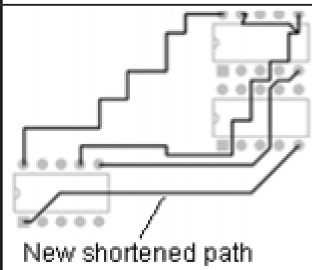
Table 62. Via Minimize Process

Before Minimize	After Minimize
	

Glossing

The glossing process reduces the Manhattan Ratio for traces, as shown in the following figure.

Table 63. Glossing Process

Before Glossing	After Glossing
	

Smoothing

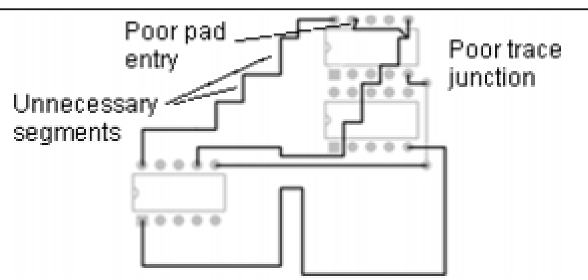
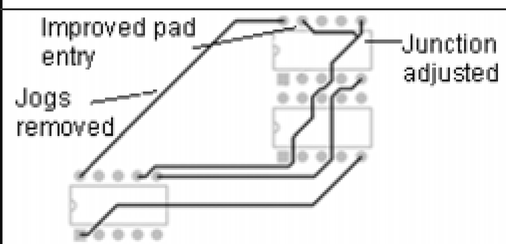
Smoothing removes unnecessary corners and segments in traces while maintaining the same path. Smoothing can also reduce length, which not only frees up room on the board for additional traces, but also helps meet and maintain pad entry and first corner rules.

The smoothing process does the following:

- Reduces the Manhattan ratio.
- Improves pad entryand trace junctions.
- Optimizestrace patterns without changing their topology.
- Eliminates same net trace crossings that push-and-shove processes can leave.
- Eliminates same net trace-to-corner violations by connecting the trace directly to the pin.
- Eliminates same net trace-to-trace violations.

Tip
Smoothing does not maintain length rules on length-controlled traces.

Table 64. Smoothing Process

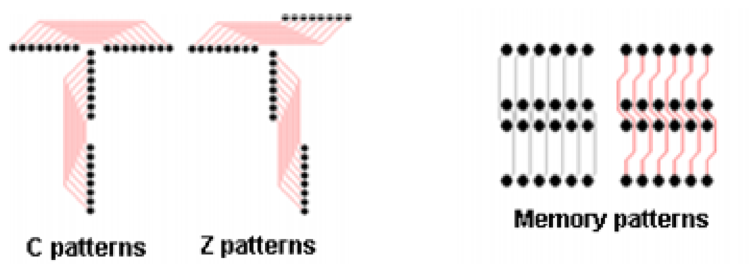
Before smoothing	After smoothing
	

For more information, refer to "“Interactive Routing Setup” on page 291” and "“Trace Smoothing” on page 351.”

Patterns Pass

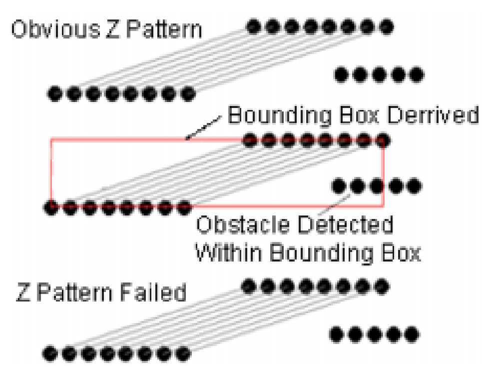
The patterns pass searches for groups of unrouted connections that it can complete using typical C routing patterns, Z routing patterns, and memory patterns. The patterns pass then routes them.

Figure 59. Patterns Pass



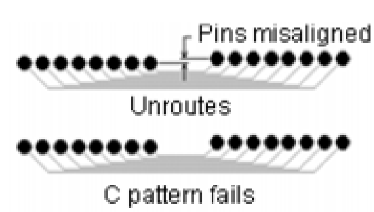
During the patterns pass, a bounding box surrounds the group of unroutes to pattern route. If enough space exists to create traces and obey clearance rules, SailWind Router creates the pattern. If obstacles appear within the bounding box or if the pattern violates rules, the pattern fails.

Figure 60. Bounding Box During a Patterns Pass



For C patterns, the pins that form the C pattern must align along the X-axis or Y-axis or the pattern fails.

Figure 61. C Patterns During a Patterns Pass



Route Pass

The route pass is the core pass, which performs the majority of autorouting. During this pass, SailWind Router tries to sequentially route each unroute until it has tried all connections. The route pass contains many processes -- serial, rip up and retry, push and shove, and touch and cross.

The following table displays the Route pass processes.

Table 65. Route Pass Processes

Process	Action
Serial	Converts unroutes to traces by finding the shortest path around the obstacles between the end points of the unroute. No obstacles can exist in the path. If a path is not found, SailWind Router skips the unroute.
Rip Up and Retry	Increases completion rates by unrouting existing traces and finding new paths to create space for other traces. If SailWind Router cannot find a a new path, it cancels the process and returns the trace to its original state.
Push and Shove	Pushes traces aside to create room for new traces. The software performs this process together with the rip up and retry process.
Touch and Cross	Eliminates touch and cross violations that are created by other autorouting operations that create or modify existing traces or patterns. For example, the push and shove process may force a trace to “hop” over a pin, creating a cross violation for the trace exiting the pin. This process successively attempts to reroute the trace to eliminate the violation. If the software cannot resolves a violation, it cancels the operation and returns the trace to its original state.

Tip
SailWind Router performs this process with all other passes and processes. It will not complete a trace with a violation. It either removes all violations or it cancels trace routing.

Tune Pass

The tune pass adjusts the length of length-controlled traces. The pass only examines trace lengths for completely routed nets, electrical nets, pin pairs, or differential pairs and analyzes the current length if you have enabled length rules and length control.

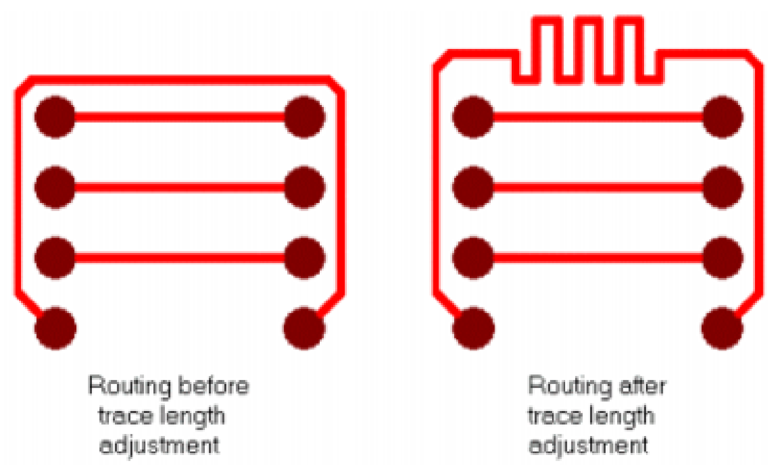
The tune pass makes adjustments based on the following conditions:

- 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.
- If only one member of a differential pair meets the matched length group rule, the pair is considered mismatched and the tune pass adjusts the noncompliant member of the pair.

- If the length delta between members of a differential pair is greater than 300 mils, the tune pass skips the pair, leaving it in its previous state (either unrouted or untuned).

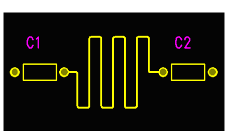
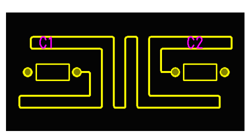
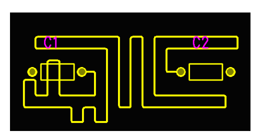
Tip
When tuning a differential pair, SailWind Router adds small accordions in the Shoulder area of the shorter trace, without plowing or shoving obstacles. If the Shoulder areas cannot accommodate the small accordion, SailWind Router adds the accordion in the Controlled gap area, as close as possible to the Shoulder area.

Figure 62. Before and After Length Adjustment



Maximum Hierarchy Level

Length adjustments are created in stages. The first stage is a regular horizontal or vertical accordion. If this is still not enough length, the accordion direction is turned 90 degrees and an additional accordion is created. SailWind Router will continue adding the accordions at 90 degrees until the length adjustment is met, or until the maximum hierarchy level is met.

		
Regular Accordion	Accordion with 90 degree accordion added	Accordion with a Maximum Hierarchy Level of 3

By default, Maximum Hierarchy Level values are set to 8 in designs created before PADS Router 9.0.

Setting Up the Autorouting Strategy

SailWind Router offers extreme flexibility in setting up an autorouting strategy. You can choose from a series of specific routing pass types to accomplish your desired autorouting tasks. You can also systematically set up your routing strategy to specify the order in which SailWind Router routes specific design objects.

For example, you could set up a strategy to perform the following:

- Fanout specific high pin count components and protect the routes
- Fanout the rest of the components
- Route critical signals such as clocks
- Route the power nets
- Route all of the nets associated with a specific component
- Route the data bus on specific layers for return path control
- Route the address bus on specific layers for net distribution control
- Route the differential pairs nets on specific layers for impedance control
- Route the balance of miscellaneous nets on specific layers

You have complete control over the routing order, the routing pass intensity, and the ability to start, stop, pause, and inspect your routing results at any point in the process.

Specifying the Passes to Route

Use the **Routing** category > **Strategy** subcategory to specify which passes to run, the routing intensity to use, whether or not to protect the generated traces, and whether to pause after a pass completes.

Procedure

1. Click the **Tools > Options** menu item; then, in the Options dialog box, click the **Routing** category > **Strategy** subcategory.
2. In the Pass column, select each pass type you want to run. You can run any combination of passes.
3. Select the “Protect” check box (In the Protect column of the table in the “Pass definition” area) if you want SailWind Router to protect any generated traces after performing the selected pass. For example, select the Protect check box for the Center pass type if you want SailWind Router to protect trace changes generated during the Center pass. (This protects traces and glues vias that are completed during the corresponding pass type.)
4. In the Pause column, select the pass check box if you want SailWind Router to pause routing after completing the corresponding pass.
5. 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.
6. Set the routing order on page 275.

7. Click **OK**.

8. After defining your strategy, click the **Start Autorouting** button on the Routing toolbar to autoroute the design.

Results

If the Options dialog box is open, a check mark appears in the Done column for each pass that completes. The check marks remain in the **Routing** category > **Strategy** subcategory until you modify the autorouting strategy, or restart autorouting. This is a read-only column; you cannot manually mark a pass as complete.

The strategy is saved by default in the *\SailWind Projects* folder.

Related Topics

Setting the Routing Order

Autorouting a Design

Autorouting Pass Types

Setting the Routing Order

Use the Routing order list to specify the order in which to autoroute components, nets, net classes, differential pairs, and matched length groups for the selected pass.

Restrictions and Limitations

Certain object types within the Routing order definition pane are not valid for routing order operations. The **Selected** button is available only when components, nets, net classes, differential pairs, and matched length groups are selected either in the object view pane, in the design itself, or in the Routing order definition pane.

Procedure

1. Click the **Tools > Options** menu item; then, in the Options dialog box, click the **Routing** category > **Strategy** subcategory.

2. In the Pass Type column (of the “Pass definition” table), click the pass type for which you want to set the routing order. The default routing order appears in the Routing Order box below the table.

3. To add items to the routing order, do any of the following:

   - To add all nets associated with plane layers, click **Plane Nets**.

   - To add all nets, click **All Nets**.

   - To add individual nets, select the component or nets to add in the left pane (Routing order definition), and then click **Selected**.

   - To remove all items from the Routing Order list, click **Clear**.

   You can also add nets attached to components to the routing order, using the following:

- **Object View tab of the Project Explorer**— You add nets from the **Object View** tab in exactly the same way as you add them using the Routing order definition list.

- **The work area**— Make sure you make nets available for selection on the Selection Filter toolbar.

4. Use the Routing Order pane buttons to delete and move items.

5. Click **OK**.

Results

The routing strategy is saved in the *\SailWind Projects* folder. When you autoroute the design, these settings are used. If the dialog box is open, a check mark appears in the Done column upon completion of each pass.

Related Topics

Autorouting Strategy

Autorouting a Design

Autorouting a Design

The Autoroute command starts the autorouter and begins performing the routing algorithms and passes you specify in the Options dialog box > **Routing** category > **Strategy** subcategory.

Restrictions and Limitations

- You cannot autoroute a design that has no board outline or has a board outline that is not a closed shape. Attempting to do so results in a “Cannot route design with open board outline” error message. Use SailWind Layout to create the board outline.

- If you choose to autoroute a design before you set a strategy on page 263, a message appears prompting you to set a strategy.

- When you Autoroute a selected object, the All Nets setting in the routing order (set in the Options dialog box > **Routing** category > **Strategy** subcategory) is ignored.

Procedure

On the Routing toolbar, click the **Start Autorouting** button or click the **Tools > Autoroute > Start** menu item. When you start autorouting or running pass types on selection, the program performs a prerouting analysis and reports any errors or warnings to the **Status** tab of the Output window. If you want to correct any of the reported issues, you must first stop autorouting.

Pausing Autorouting

Use Pause to pause autorouting between passes to check—for example—on statistics and completion rates before continuing or canceling the autoroute. The Pause feature stops routing at the end of the current subpass. SailWind Router stores the pass and point within the pass so you can resume autorouting from the point where you paused.

Procedure

1. On the standard toolbar, click the **Routing toolbar** button.
2. Click the **Pause** button.

Resuming Autorouting

Use Resume to start autorouting after pausing. Routing begins at the subpass where you paused. Use Stop to end autorouting.

Procedure

1. On the standard toolbar, click the **Routing** button.
2. Click the **Resume** button.

Stopping Autorouting

Use **Stop** to end autorouting at any time. The **Stop** button ends routing at the end of the current connection. SailWind Router then clears all of the Done check boxes in the Options dialog box > **Routing** category > **Strategy** subcategory.

You cannot resume autorouting when you choose **Stop**. You can only start autorouting from the beginning of the strategy.

Procedure

1. On the standard toolbar, click the **Routing** button.
2. Click the **Stop** button.

Routing Progress Indicator

The status bar contains a routing progress indicator with a ToolTip that displays the completion rate of each pass and subpass the router performs. This helps you assess the effectiveness of the strategy and the settings you selected for routing.

Reporting Routing Results

When you autoroute your design by running a pass type or by starting an autorouting command, SailWind Router reports two types of messages: prerouting analysis messages and routing status messages. These messages appear in a routing report that lists prerouting messages, the name of the design, the number of passes, the pass types performed, and other statistics for each pass.

The routing report, *RoutingReport.txt*, is located in the *\SailWind Projects* folder. A link to this file is automatically created in the **Status** tab of the Output window. You can open and view the file by clicking the link on the **Status** tab.

Related Topics

Autorouting Strategy

Autorouting Messages

Assigning Test Points During Autorouting

You can create an accessible net during autorouting. This process enables test point placement during the autorouting “Route” pass. As it adds traces, the autorouter attempts to also add test points for each net using vias or component pins along the net. The via or pin acts as the test point for the net whenever someone inserts the PCB into the Automatic Test Equipment (ATE). Adding test points during autorouting sacrifices completion rates for accessibility.

Procedure

1. Click the **Tools > Options** menu item; then in the Options dialog box, click the Test Points category.
2. Select the “Create test points during autorouting” check box. Set other testability options as needed. For more information, see “Options Dialog Box, Test Points Category” on page 459.
3. Click **OK**.
4. On the Routing toolbar, click the **Start** button. When you autoroute the design, SailWind Router uses the test point settings.

Tip
During test point placement, SailWind Router respects test point keepouts set in SailWind Layout.

Related Topics

Test Points Pass

Properties Dialog Box, Test Points Tab

Autorouting Messages

When you autoroute your design (by either using an autorouting command or running a pass type), SailWind Router reports two types of messages: prerouting analysis and routing status.

When autorouting completes, or when you stop autorouting, SailWind Router creates a routing report that details the results of the autoroute. For more information, see Routing Report.

Prerouting Analysis

When you start autorouting, SailWind Router analyzes the design for problematic conditions that may affect or cancel routing. Related messages appear in the **Status** tab of the Output window. These messages appear in the Routing Report only if the entire design is routed. To correct any of the reported issues, you must first stop routing.

Selection Dependency

As shown in the following table, the type of prerouting analysis depends on the current selection.

Table 66. Selection Dependency

Selection	Analysis Scope	Analysis Content
Nothing	Whole design	All
Net	Net	Thermal status Potential plane nets No routing settings Via settings Vias to planes
Pin Pair	Pin pair	Thermal status No routing settings
Unroute	Unroute with adjacent pins and vias	Thermal status No routing settings for owning net
Component	Pin pairs connected to component pins	Thermal status No routing settings
Pin	Pin pairs connected to the pin	Thermal status No routing settings

Routing Status

When you start autorouting, the current routing status appears. These messages appear on the status bar and in the **Status** tab of the Output window. Routing status messages appear in the session log under the heading “Batch Router Statistics.” Routing status messages update with each pass and subpass of the router.

For more information on the session log, see “Status Tab” on page 57.

Center Pass Messages

The following is a center pass message, whose parts are explained in the following table.

```
Centering gate 264/1738 (HorDir, layer6)
```

Table 67. Parts of a Center Pass Message

Part of Message	Description
gate	Current gate being processed
Dir	Direction of the current gate (Hor, Vert, or 135)
layer	Layer of the current gate

Fanout Pass Messages

The following is a fanout pass message, whose parts are explained in the following table.

```
Pins 162 of 198 (82%) Duration 00:00:31 (+00:00:20)
```

Table 68. Parts of a Fanout Pass Message

Part of Message	Description
Pins	Number of pins fanned out, followed by the total number of pins selected for fanout, and the fanout completion percentage in parentheses.
Duration	Amount of time the router spent on this pass, followed by the total time spent routing in parentheses.

Tip
SailWind Router reports individually the status of fanned out pins belonging to the same island. For example, if three pins belong to the same island (connected with each other by traces), SailWind Router fans out all three pins with one shared via during the fanout pass. In this case the message displays three pins fanned out (Pins 3 of 3 (100%) Vias 1 Duration 00:00:01 (+00:00:01)).

Patterns Pass Messages

The following is a patterns pass message, whose parts are explained in the following table.

```
Routed 15 Total 65 of 650 (10%) Vias 181 Duration 00:01:31 (+00:01:00)
```

Table 69. Parts of a Patterns Pass Message

Part of Message	Description
Routed	Number of connections routed in this pass, followed by the total number of routed and unrouted connections in the design, and the pattern completion percentage in parentheses.
Vias	Number of vias added during this pass.
Duration	Amount of time the router spent on this pass, followed by the total time spent routing in parentheses.

Route Pass Messages

The following is a route pass message, whose parts are explained in the following table.

```
Routed 635 Total 650 of 650 (100%) Vias 481 Duration 00:06:41 (+00:05:10)
```

Table 70. Parts of a Route Pass Message

Part of Message	Description
Routed	Number of connections routed in this pass, followed by the total number of routed connections in the design, and the routed connections completion percentage in parentheses.
Vias	Total number of vias added during this pass.
Duration	Amount of time the router spent on this pass, followed by the total time spent routing in parentheses.

Optimize Pass Messages

The following shows an optimize pass message, whose parts are explained in the following table.

```
Optimized 5% Vias 451 (-30) Trace length 66 (-5) inches Duration 00:07:41
 (+00:01:00)
```

Table 71. Parts of an Optimize Parts Message

Part of Message	Description
Optimized	Optimization completion percentage.

Table 71. Parts of an Optimize Parts Message (continued)

Part of Message	Description
Vias	Total number of vias in the design, followed by the number of vias reduced by optimization in parentheses.
Trace Length	Total length of traces routed in this pass, followed by the total length of traces reduced by optimization in parentheses.
Duration	Amount of time the router spent on this pass, followed by the total time spent routing in parentheses.

Tip
Inches are used for Mils and Inches settings. Meters are used for Microns and Metric settings.

Miters Pass Messages

The following shows a miters pass message, whose parts are explained in the following table.

```
Trace 15 of 172 Miters 20 Duration 00:08:52 (+00:01:11)
```

Table 72. Parts of a Miters Pass Message

Part of Message	Description
Trace	Total number of traces processed, followed by the total number of traces selected for mitering.
Miters	Total number of miters added in this pass.
Duration	Amount of time the router spent on this pass, followed by the total time spent routing in parentheses.

Test Point Pass Messages

The following shows a test point pass message, whose parts are explained in the following table.

```
Accessible nets 46 of 100 (46%) Duration 00:09:52 (+00:01:00)
```

Table 73. Parts of a Points Pass Message

Part of Message	Description
Accessible Nets	Number of nets made accessible in this pass, followed by the total number of nets with test point requirements in the design, and the test point completion in parentheses.
Duration	Amount of time the router spent on this pass, followed by the total time spent routing in parentheses.

Tune Pass Messages

The following shows a tune pass message, whose parts are explained in the following table.

```
Nets tuned 25 of 32 Pinpairs tuned 78 of 500 Duration 00:04:33 (+00:03:59)
```

Table 74. Parts of a Tune Pass Message

Part of Message	Description
Nets tuned	The number of nets tuned in this pass, followed by the total number of nets with length restrictions.
Pinpairs tuned	The number of pin pairs tuned in this pass, followed by the total number of pin pairs with length restrictions.
Duration	Total time the router spent routing, followed by the amount of time the router spent on this pass in parentheses.

Routing Done

SailWind Router displays a message in the status bar after finishing all passes.

The following message appears in the status bar after all passes finish:

```
Done> Unroutes 0 Routed 164 of 164 (100%) Vias 100 Duration 00:09:52
```

The following table explains the parts of the message:

Table 75. Parts of the Routing Done Message

Part of Message	Description
Unroutes	Total number of unroutes in the design.
Routed	Total number of routed links, followed by the total number of routed and unrouted links in the design, and the routing completion percentage in parentheses.
Vias	Total number of vias in the design.
Duration	Amount of time the router spent on the pass, followed by the total time spent routing in parentheses.

Routing Progress Indicator ToolTip

The status bar also provides a progress indicator ToolTip that displays the completion rate of each pass and subpass the autorouter performs. Place the pointer over the status bar during autorouting to view the progress indicator ToolTip. This ToolTip helps determine the effectiveness of the strategy and your routing settings.

The following is an example of the ToolTip message that appears when you place the pointer over the status bar. A ToolTip message can have the parts shown in Table 76.

```
Subpass 1 of 12 Attempting 15 of 100 Complete 10 (U1.1-Via)
```

Table 76. Parts of the Tooltip Message

Part of Message	Description
Subpass	Subpass, followed by the total number of subpasses in the current pass.
Attempting	Number of connections attempted in the current subpass, followed by the number of connections selected for the current subpass.
Complete	Number of successful attempts in the current subpass, followed by the current connection in parentheses.

Routing Report

A routing report is automatically created when autorouting completes or when you stop autorouting. It lists the name of the design being routed, the number of passes, the pass types performed, and other statistics for each pass. The routing report, *RoutingReport.txt*, is located in *\SailWind Projects*. A link to this file appears automatically in the **Status** tab of the Output window. You can open and view the file by clicking the link tab.

Tip
The prerouting analysis messages also appear in this report, but only if routing finishes. If you stop autorouting, the prerouting analysis messages do not appear in the report.

For more detailed information, see The Routing Report.

Chapter 18
Selective Autorouting

Instead of batch autorouting named design objects, you can autoroute components and nets by selecting them in the design. The autorouter then routes them immediately.

You can select objects whose nets you want to autoroute, and instead of using the Strategy dialog box, you can use a command to run a pass type on the selection. These autorouting commands are available on the Routing toolbar and on the shortcut menu that appears when you right-click the selected object.

The pass type SailWind Router uses when running pass types on your selections is similar to the pass type set in the Strategy dialog box for batch autorouting. However, you cannot change the settings for this pass type, and the pass types are limited to the following:

- **Route** — This lets you preroute objects in specific routing modes, such as routing a BGA in any-angle mode, and then protecting the routes before batch autorouting the design in diagonal mode. Follows options set in the **Routing** tab of the Options dialog box.
- **Fanout** — Fans out selected components before routing the rest of the board. Follows options set in the **Fanout** tab of the Options dialog box.
- **Optimize** — Uses the highest possible intensity, to let you interactively improve critical areas of the design after batch autorouting.
- **Tune** — Adjusts the length of selected traces (including differential pairs) or traces attached to selected components to meet length rules.
- **Center** — Places traces equidistant from the pads of adjacent component pins or vias to evenly distribute any available space in the channel.

Autorouting by Selection

You can start autorouting after you set options and preferences defining how you want to route your design.

Procedure

1. On the standard toolbar, click the **Routing** button or click the **View > Toolbars > Routing Toolbar** menu item.
2. Select the object to route.
3. Click the **Route** button.

Tip
SailWind Router routes the net or the net attached to the selected object by using the settings in the Options dialog box > **Routing** category > **Strategy** subcategory. You cannot route only a pin pair or link in this program; you must route the entire net.

Note:
When you start autorouting or running pass types on selection, the program performs a prerouting analysis and reports any errors or warnings to the **Status** tab of the Output window. If you want to correct any of the reported issues, you must first stop autorouting.

Running Pass Types in Object Mode

You can run a pass type on a selection, independently of any strategy settings in the Options dialog box, while operating in Object mode.

Procedure

1. Select the component, pin, net, unroute, or trace on which you want to perform the pass type. The pass type is run on the selected net or the net connected to the selected component or pin.

2. Right-click and click one of the available pass types.

Table 77. Available Pass Types for Object Types

Object	Pass Type
Component	Route, Fanout, Optimize, Tune
Pin	Route, Fanout, Optimize
Net	Route, Optimize, Tune, Center
Unroute	Route
Trace	Center

Tip
When you run pass types, the program performs a prerouting analysis and reports any errors or warnings to the **Status** tab of the Output window.

Related Topics

Running Pass Types in Verb Mode

Autorouting Messages

Running Pass Types in Verb Mode

You can run a pass type on a selection, independent of any strategy settings in the Options dialog box, while operating in Verb mode.

Procedure

1. On the Routing toolbar, click the pass type button you want to use: Route on page 271, Fanout on page 266, Optimize on page 269, Tune on page 272, or Center on page 265.
2. Select the component, pin, net, unroute, or trace on which you want to perform the pass type. The selection filter automatically updates, allowing you to select only an object to which the command applies.
3. Click the pass type button again or press the Esc key to exit Verb mode.

> **Tip**
> When you run pass types, a prerouting analysis runs and reports any errors or warnings to the **Status** tab of the Output window.

Related Topics

Running Pass Types in Object Mode

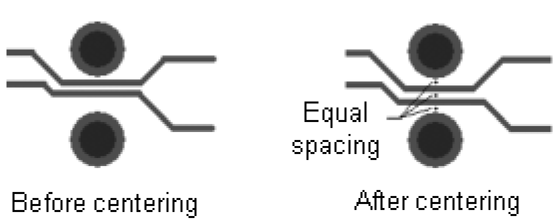
Setting Up the Autorouting Strategy

Centering Traces

Center is a pass of the autorouter that you can run on selected traces. The center pass places the traces equidistant from the pads of adjacent component pins or vias to evenly distribute any available space in the channel. Centering does not violate design rules.

The following figure illustrates trace centering.

Figure 63. Centering Traces— Example



To limit the number of channels to center, set a maximum channel width. Any channel larger than this width is not eligible for centering. Set the Maximum channel width in the Options dialog box > **Routing** category > **General** subcategory.

Procedure

1. Select a trace.
2. Right-click and click the **Center** popup menu item.

Assigning Test Points After Autorouting

This process adds test points (using the test point pass) to an already routed board. This process sacrifices accessibility for higher completion rates.

Procedure

1. Click the **Tools > Options** menu item; then in the Options dialog box, click the Test Points on page 459 category.
2. In the “Create test points” area, clear the “Create test points during autorouting” check box. Set other testability options as needed.
3. Click the **Routing** category > **Strategy** subcategory.
4. In the table of the “Pass definition” area, select the Pass check box for the Test Point pass type.
5. Click **OK**.
6. Click the **Tools > Autoroute > Start** menu item.

   You can use the Test Point pass with other pass types. Routing completes for the entire design and then SailWind Router adds test points.

   Note:
   Test point keepouts set in SailWind Layout are respected during test point placement.

Related Topics

Assigning Test Points During Autorouting

Properties Dialog Box, Test Points Tab (Component or Via Type)

Properties Dialog Box, Test Points Tab (Nets)

Selective Autorouting

Chapter 19
Manual Routing

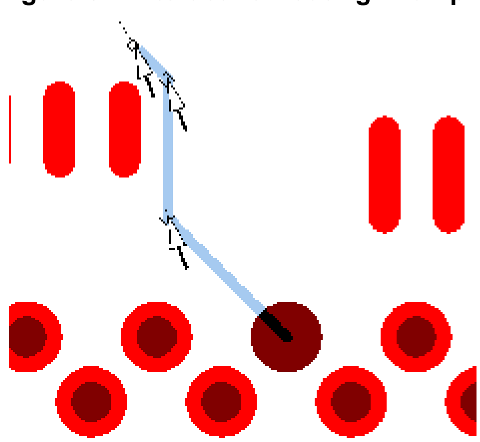
Interactively create traces to control all aspects of the routing.

Interactive Routing Workflow

Using interactive routing, you create new traces or reroute existing traces. Traces patterns follow the path of the pointer, connecting one point to another. You can also add corners, vias, and other routing objects.

The following figure illustrates making a connection from one point to another with the pointer.

Figure 64. Interactive Routing Example



Required Setup:

- Set design rules on page 154 such as trace sharing and component entry.
- Set net topologies on page 291.
- Set routing options on page 291, such as dynamics, plowing, the routing angle, and so on.
- Choose whether to protect traces on page 292.
- Set display options on page 294 for routing.
- Choose whether to monitor trace length on page 304.

Route Interactively:

1. Create a trace on page 304.
2. End the trace on page 306.

Add other routing elements:

- Add corners on page 318.
- Add arcs on page 321.
- Add vias on page 326.
- Add test points on page .
- Add Length to Traces.

Related Topics

Interactive Routing Setup

Interactive Routing Setup

Read the topics that follow to learn more about setting up interactive routing.

Setting Net Topology
Protected Objects
Display Options for Interactive Routing

Setting Net Topology

You can set different topology types for nets, which affects how they are routed whether you are routing interactively, autorouting, or editing traces. Topology types determine the order in which to connect pins in a net.

If you have nothing selected, the Design Properties dialog box appears, enabling you to set topology globally. If you selected a net or a net class, a specific properties dialog box appears, enabling you set topology for the selected net or nets.

Restrictions and Limitations

- The **Topology** tab does not appear in the Pin, Via, Electrical Net, Pin Pair Group, and Pin Pair Properties dialog boxes.

Procedure

1. Right-click and click the **Properties** popup menu item; then, in the Properties dialog box, click the Topology tab on page 524.
2. Click the net topology type you want.
3. Click the types of junctions you want to allow.
4. In the “Maximum stub length” box, type the maximum value. A value of 0 means no stubs are allowed.
5. Click **OK**.

Protected Objects

You can protect objects while interactively routing or trace editing. When you protect objects, you cannot plow, unroute, move, reroute, or otherwise edit them. Objects you can protect include trace segments, traces, vias, and test points.

SailWind Router automatically protects reuse objects and jumpers defined in SailWind Layout. It does not, however, protect unions.

Protecting Traces and Vias While Interactively Routing

You can protect traces and vias during interactive routing to prevent—for example—making inadvertent edits.

Procedure

1. Click the **Tools > Options** menu item; then, in the Options dialog box, click the **Routing** category > **General** subcategory.
2. Before interactively routing, select the “Protect trace and vias when creating segments” check box.

   As an alternative, while you are routing a trace, right-click and click the **Protect Traces and Vias** popup menu item. SailWind Router protects the current trace segment back to the last corner as well as any subsequent traces. The “Protect Traces and Vias” command only effects the current operation; it does not change the state of the check box in the **Routing** category > **General** subcategory.

   When you enable protection during interactive routing, SailWind Router protects objects as you create them. If you are not dynamically routing, SailWind Router protects trace segments when you click to add a corner. If you insert vias or test points while routing the trace, the software also protects them.

   Tip
   Select the “Distinguish protected traces and vias” box (Options dialog box > Global category > General subcategory) to show protected traces, nets, vias, and test points differently from unprotected objects. For example, this can show protected objects as outlines instead of filled traces. The display of traces changes after you finish routing, not as traces are added.

Results

To turn off protection: Right-click and click the **Protect Traces and Vias** popup menu item again.

Protecting Existing Objects

Protect traces and vias after placing them.

Procedure

1. Select an object.
2. Right-click and click the **Protect** popup menu item. As an alternative, on the Route Editing toolbar, click the **Protect** button.

Unprotecting Existing Objects

You can remove the protection from (“unprotect”) traces and vias that have been protected previously.

Restrictions and Limitations

You cannot unprotect reuse members or jumper vias.

Procedure

1. Select an object.
2. Right-click and click the **Unprotect** popup menu item. Alternatively, on the Route Editing toolbar, click the **Unprotect** button.

Display Options for Interactive Routing

Several features control how traces and obstacles appear in SailWind Router.

Dynamically Reconnecting

As you route, the unrouted portion of the trace dynamically reconnects to an end point in the net that is closest to the pointer. You can toggle the feature on and off.

Procedure

1. While interactively routing, right-click and click the **Dynamic Reconnect** popup menu item.
2. Repeating this procedure toggles the feature to the opposite state.

Results

- Differential pairs do not reconnect dynamically. However, if you route differential pairs separately, the individual trace reconnects dynamically according to net order.
- Nets with Protected and High-speed topology types reconnect dynamically to preserve the specified routing order.
- Unroutes do not reconnect dynamically to allow routing of a specific connection.

Related Topics

Setting Net Topology

Showing Via Guides

You can display a via guide at the pointer to help you place vias. When the Via grid differs from the Routing grid, a via guide appears, showing you the closest available locations for vias. In addition, if you are snapping to the via grid, a small matrix appears showing other valid grid points.

You can toggle the via guide on and off.

Restrictions and Limitations

If the via grid is set to 0, the via guide does not appear.

Procedure

Type "VG" and press the Enter key. This toggles the state of the via guides.

Completion Points

Graphics that appear at the pointer change automatically to indicate completion points. When the pointer nears a valid completion point, the pointer changes to a bullseye. In addition, when the pointer is close to the completion object, an outline of the object blinks.

You cannot turn this option on or off.

Showing Clearances Around Obstacles

You can graphically view the minimum clearance around obstacles. Transparent outlines, called *guard bands*, surround obstacles to the object being routed or edited. Guard bands show the minimum distance you must maintain between objects.

Procedure

1. Click the **Tools > Options** menu item; then, in the Options dialog box, click the **Global** category > **General** subcategory.

2. Select the "Show guard bands on object" check box.

Tip
You can set the guard band color in the Colors page on page 422 of the Options dialog box.

Showing Protected Objects

You can choose to show protected components, traces, vias, and test points differently from unprotected objects.

Procedure

1. Click the **Tools > Options** menu item; then, in the Options dialog box, click the **Global** category > **General** subcategory.

2. Select the "Distinguish protected traces and vias" check box. As an alternative, use the "I" shortcut key.

Results

If traces appear as filled objects, protected traces appear as outlines only.

Showing Unroutes

You can choose to show unroutes at the default trace width.

Procedure

1. Click the **Tools > Options** menu item; then, in the Options dialog box, click the **Global** category > **General** subcategory.
2. Select the “Show unroutes at recommended width” check box. Otherwise unroutes appear as thin connection lines.
3. Click **OK**.

Results

- You can use the **U** shortcut key to turn unroute visibility on and off. This shortcut key does not effect visibility of selected or highlighted unroutes.
- You can hide unroutes by matching the color of “Connection” to the “Background” color using the Colors page of the Options dialog box.
- You can use the View Nets dialog box to set unroute visibility either globally across the design or individually for selected unroutes.

Related Topics

View Controls

Highlighting Nets

You can highlight nets to make them easier to see in your design. You can use the highlighting options to change the way you view nets. If desired, you can also choose to view only the traces or nets you are routing.

Procedure

1. Click the **Tools > Options** menu item; then in the Options dialog box, click the Global category > General subcategory.
2. In the Highlighted object display area on page 441, in the “Distinguish highlighted objects by” list, choose “Dimming other object colors.” You can also use the H shortcut key to change this setting. For more information, see View Controls.
3. Click **OK**.
4. Select a net to highlight.
5. Click the **Edit > Highlight** menu item. You can also use the Highlight column in the View Nets dialog box. You can also use the **H** shortcut key with the net name to turn highlighting on and off for a net.

Related Topics

Highlighting and Unhighlighting Design Objects

Unhighlighting Objects

Setting Net Topology

[Modeless Commands](#)

[View Nets Dialog Box](#)

Design Rule Checking and Interactive Routing

Turn on design rule checking and set it to Prevent to avoid design rule violations during interactive routing, route editing, and placement editing operations.

When you enable clearance checking set it to “Prevent,” you generally cannot create a violation while moving the pointer because you cannot create or edit a routing object in a way that violates design rules. For example, if you attempt to route a trace between two pins, and the trace does not have enough clearance, the trace stops just before the clearance boundary of the pins. Clicking near the pins adds a corner at the last valid point before encountering an obstacle. Because the trace does not violate the clearance boundary, no violation occurs. However, placing vias while interactively routing can create violations. If placing a via creates a violation, the operation cancels without placing a via.

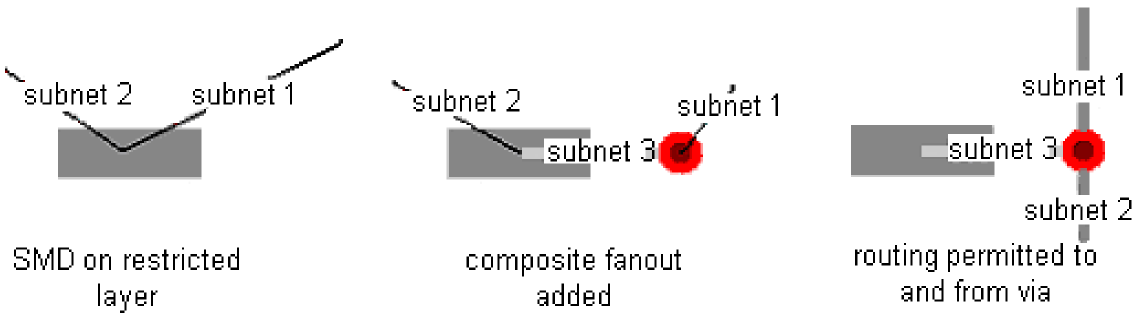
SailWind Router can also prevent violations that you might inadvertently create while editing object properties in one of the properties dialog boxes or in the spreadsheet window. For example, if you select a trace segment and change its width to a value that would not fit due to clearance violations, SailWind Router prevents the change.

Trace Sharing Rules

Trace sharing rules restrict the sharing of vias and traces. When sharing is restricted, you may not be able to route more than two traces to a single via. Set trace sharing rules in the “Allow junctions on” area on the **Topology** tab of the Design Properties dialog box.

You can share SMD pins mounted on outer layers (on which routing is restricted), sharing a pin that otherwise cannot be shared. To provide access to the pin on the restricted layer, create a fanout of a short trace ending with a via. The trace connecting the SMD pin to the via becomes a subnet. A subnet joins to other portions of the net at the via without sharing the segment with any other portion of the nets. This fanout is called a *composite fanout*.

Figure 65. Trace Sharing Rules



Trace Width and Component Rules

SailWind Router uses the recommended trace width when starting from a pin or via. When routing from a trace segment, trace corner, or end point, the software uses the current width of the selected trace. When this size cannot be used (due to clearances, for example), SailWind Router uses the recommended trace width instead. If the recommended trace width is still too large, SailWind Router tries to use the pad width, or finally, the minimum trace width. This occurs only when you select the “Allow trace necking at pads” check box on the **Routing** tab of the Options dialog box.

SailWind Router uses component rules during routing as well. One of the component rules specifies a trace width per component. When both component rules and trace width rules are defined, the software uses the width rule from the component rules set.

When you exit a pin of a component with unique trace width rules, the component width rule is applied.

Set the Maximum, Recommended, and Minimum trace widths on the **Routing** tab of the Design Properties, Net Properties, Pin Pair Properties, or Component Properties dialog boxes.

Same Net Rules and Corners

Same net corner rules determine how far from the edge of a pad you can place the first corner when routing a trace. These rules also represent the minimum distance between traces of the same net. You cannot add a corner too close to the edge of a component pin, via pad, or copper boundary unless you enable soft rules.

Set the same net rules on the **Same Net** tab of the Design Properties dialog box. Set whether first corner rules are hard or soft with the Ignore first corner rules to complete traces when required check box in the **Pad Entry** tab of the Design Properties or Component Properties dialog boxes.

Pad Entry Rules and Routing Interactively with Dynamics

When routing interactively with dynamics, any traces exiting a pad or copper respect the pad entry rules. If the “Dynamically route” check box is cleared, traces do not automatically maintain pad entry rules.

Pad entry rules define how traces enter and exit component pads and manage the quality of the trace pattern at pad and trace junctions. SailWind Router maintains pad entry rules for all routing angles and pad orientations.

Set pad entry rules on the **Pad Entry** tab of the Properties dialog boxes for selected components.

Topology Rules

When routing a net with a Protected topology type, SailWind Router disables the dynamic reconnect features to preserve net order. You can only connect the net in the order defined by the pin pairs; however, you can connect at any point in the branch. For example, you can connect to any trace or via that is attached to the targeted connection pin. If you are routing a high-speed net, you can click to reconnect to the final destination.

For more information, refer to Setting Net Topology.

Editing Traces and Same Net Clearance Rules

Same net rules are verified after you complete an editing operation. With Same net Pad to Corner rules, only the first corner after the trace exits the pad is maintained. Same net Trace to Corner rules are not maintained.

Interactive Routing

The interactive routing features let you route traces manually, by either clicking and entering a trace pattern with your pointer or automatically following the path drawn by the pointer.

For information on using design rules with interactive routing, see Design Rule Checking and Interactive Routing.

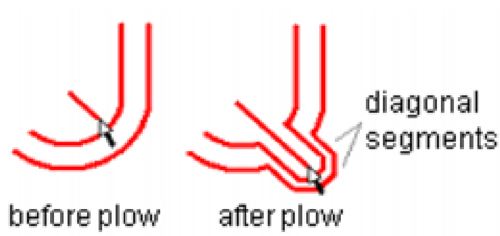
Trace Plowing

The plower moves existing traces and vias automatically as you route new traces.

Observe the following restrictions when plowing traces:

- You cannot plow vias that are set as protected test points.
- You cannot plow reuse objects.
- When you enable the plower the router attempts to plow objects. To allow the plowing of nets, select the “Shove traces to complete connections when required” check box (**Routing** tab of the Design Properties, Pin Pair Properties, or Net Properties dialog boxes).
- To allow the plowing of protected objects, select the “Shove protected traces when required” check box (**Routing** tab of the Design Properties, Pin Pair Properties, or Net Properties dialog boxes).
- SailWind Router dose not plow arcs as arcs. Instead, it interrupts the arc and adds trace segments to allow for the trace in progress.

Figure 66. Traces Before and After Plow



If the plowed traces encounter an immovable obstacle, like a protected via, the plowed traces “hop” over the obstacle. SailWind Router maintains clearance rules when traces hop over objects.

For more information, refer to “Interactive Routing Setup.”

Figure 67. Traces are Obstacles to New Traces

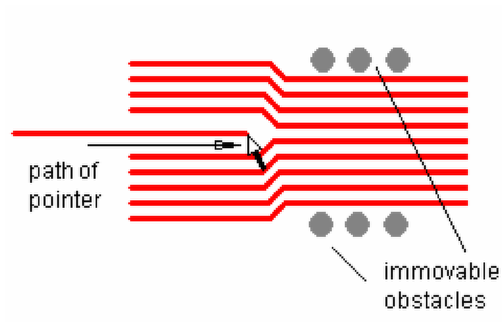
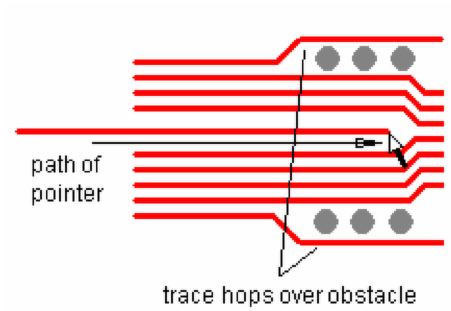


Figure 68. Traces are Plowed to Make Room for New Traces



Plower Types

The following table shows the available plower types.

Table 78. Plower Types

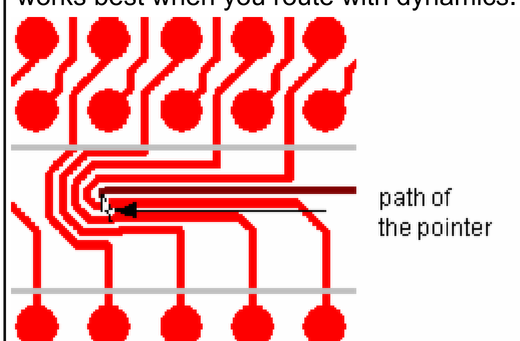
Plower type	Action
Plow with pointer	Plows obstructions as the pointer moves towards them. This feature works best when you route with dynamics.  Plow with pointer has two modes: • Real-time plowing— Plows obstacles immediately as the pointer passes them. • Guided plowing— Plows obstacles after you define a potential path with the pointer and then reach an open area.

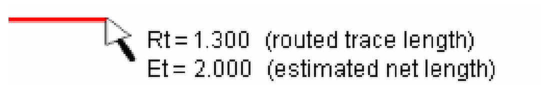
Table 78. Plower Types (continued)

Plower type	Action
Plow after corner click	Plows obstructions only after you click to enter a corner. If no room exists to move the obstruction, you cannot enter a corner. Visually, the ending trace pattern is the same as with Plow with pointer movement. **Tip** This feature works best when you route without dynamics.

Interactive Routing with the Trace Length Monitor

As you route traces interactively, the routed length and estimated total length appear near the pointer. The feature for displaying routed and estimated length is called the *trace length monitor*.

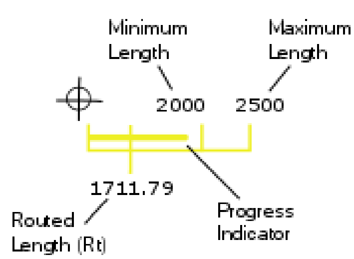
Figure 69. Routed Length and Estimated Total Length



Routing Traces with Length Rules

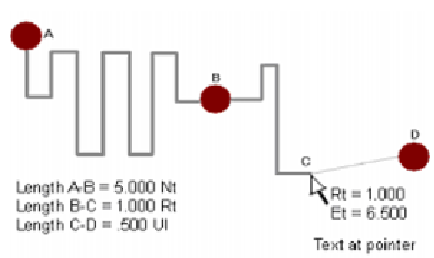
When routing traces with length rules, the trace length monitor shows the routed length, the minimum length rule, and the maximum length rule for the net, electrical net, or pin pair. A progress indicator bar also appears as shown in Figure 70.

Figure 70. Routing Traces with Length Rules



Calculating Length

The trace length monitor calculates estimated length as the combined total of routed and unrouted length of the trace being routed and includes half the Discrete length value of each connected pin of components that have a Discrete length assigned.



Routed Length

The trace length monitor calculates routed length as the cumulative length of the trace. It includes half the Discrete length value of each connected pin of components that have a Discrete length assigned. If you start routing from the end point of a partially routed trace, the routed length includes the partially routed trace length. If the trace has branches, then the trace length monitor calculates the length from the branch point.

Unrouted Length

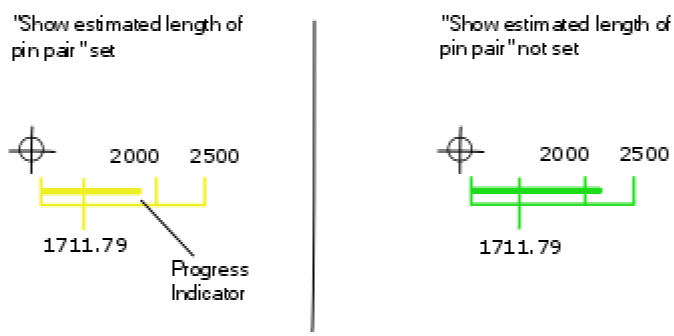
The trace length monitor calculates unrouted length as the distance from the end point of the current trace segment (attached to the pointer) to its destination. The unrouted length is recalculated as the unroute dynamically reconnects to connection points. The routing angle also effects this calculation.

Progress Indicator

For traces with length rules, the progress indicator displays the total *estimated* length for either the current pin pair or the entire net, depending on the “Show Estimated Length of Pin Pair” setting in the Routing/ General options on page 450. If you select this check box, the progress indicator shows the estimated length for the current pin pair. When you clear the check box, the indicator shows the estimated length for the entire net. Keep in mind this setting can affect whether or not the estimated length on the display appears to meet or violate length rules while routing from one pad to the next. Figure 72, for example, shows the progress indicator for the same net with the “Show Estimated Length of Pin Pair” check box both selected and cleared.

Figure 71. Calculated Length

Figure 72. The Progress Indicator with Different Settings



Monitor Display

As shown in the table below, the trace length monitor changes depending on what rules are set.

Table 79. Length Rules and Monitor Display

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, electrical 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.

Table 79. Length Rules and Monitor Display (continued)

Length rule	Information in report
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, electrical net, or pin pair rule.
No Length Rules	Shows the current routed length and the total estimated length

Colors Used in the Monitor

The Navigation Window shows only those nets, electrical nets, or matched length groups that have length rules and are affected by the current operation. The **Net Length Monitor** tab of the Spreadsheet window shows the nets, lengths, and rules in a table. The monitor, navigation window, and **Length Monitor** tab show length in different colors. A colored bar moves across the monitor. In the navigation window, the nets appear in colors. In the **Net Length Monitor** and **Electrical Net Length Monitor** tabs, lengths in the Length columns appear in different colors. Each color has a special meaning.

Table 80. Trace Length Monitor Colors

Color	Means the net length is
Yellow	Below the minimum limit
Green	Within the minimum and maximum limits
Dark Green	Within 10% of the minimum or maximum limits
Red	Over the maximum limit

Trace Length Display

You can monitor trace length in four places.

- **At the pointer** — Shows a graphic that monitors progress, routed length, estimated net or electrical net length, and any length rules. This information also appears on the status bar. The graphics that appear near the pointer change depending on whether the net, electrical net, or pin pair has length rules, rules other than the default, or if the rule is violated.
- **In the Navigation window** — Shows only those nets, electrical nets, or matched length groups that have length rules and are affected by the current operation.
- **In the Net Length Monitor tab of the Spreadsheet window** — Shows nets, lengths, rules routed length, unrouted length, estimated net length, and length rules. You can also change length rules here.
- **In the Electrical Net Length Monitor tab of the Spreadsheet window** — Shows electrical nets, lengths, rules routed length, unrouted length, estimated net length, and length rules. You can also change length rules here.

Effects of the Tune Pass on Trace Length

During interactive routing and rerouting operations, the trace length monitor may show a length greater than the actual final length. You may perform pad entry corrections on traces after you finish routing (depending on your routing preferences). These operations may change trace length slightly.

Monitoring Trace Length

You can monitor trace length when routing or editing traces. By monitoring trace length, you can ensure that traces meet length rules as you route. The trace length monitor at the pointer shows both the current routed trace length and the estimated total net or electrical net length.

Restrictions and Limitations

For differential pairs, the trace length monitor shows the length of one pair member at a time. Use the Tab key to switch between the pair members.

Procedure

1. Click the **Tools > Options** menu item; then in the Options dialog box, click the **Global** category > **General** subcategory.
2. In the "Pointer settings" area, select the "Length monitor" check box.
3. To estimate the total length of the current pin pair (total length + unrouted length), select the "Show estimated length of pin pair" check box on the **Routing** category > **General** subcategory. Otherwise, the length monitor estimates the total length of the net or electrical net length (total net length + unrouted length).

   Tip
   You can reset the current length value in the trace length monitor to zero by pressing Ctrl +Page Down while interactively routing.

4. Click **OK**. The estimated length appears next to the pointer when you begin actively routing a trace.

Creating Traces

Create traces by routing. Before routing, you must choose a starting point and initiate interactive routing. You can interactively route from copper polygons and net objects, such as a pins, virtual pins, vias, traces, trace segments, trace corners, T-junctions, unroutes, and end points of a trace.

For more information, see "Creating Differential Pair Traces" on page 335.

Procedure

1. In the Layer list in the standard toolbar, choose the layer on which the objects to route exist. As an alternative, use the L shortcut key to select a layer. You cannot interactively route on restricted layers. Set layer restrictions on the **Layers** tab or the **Layer Biasing** tab of the Design Properties dialog box.
2. Select the unroute to route, or the object from which to route. You cannot select a copper segment. You can select only the entire copper polygon. The routing start point is near the selection point.
3. Start interactive routing by right-clicking and clicking the **Interactive Route** popup menu item. Hatch outlines disappear automatically when you start interactive routing. SailWind Router also ignores stub length rules.

4. If in step 2 you selected a pin that belongs to more than one pin pair, you can change the pin pair that is selected for interactive routing. To select a different pin pair to route, right-click and click the **Switch Pin Pair** popup menu item.

5. Move the pointer in the direction you want to create a trace. For instructions to follow another shape with a trace, see "Following Shapes" on page 311.

6. If necessary, add the following while routing:

   - Corners on page 318 — Required if you are routing without dynamics. Optional when you route dynamically. Corners typically change the direction of a trace.

   - Vias on page 326 — To change layers while routing.

   - Test points on page — To make your design testable.

   - "Arcs" on page 321 — To create arc segments.

   - Length on page 312 — To meet length rules using accordions.

   As you move the pointer, guard bands may appear, indicating obstacles.

7. To change the trace properties such as width and layer while interactively routing, see "Changing Trace Properties While Routing" on page 340. The trace width may prevent you from routing between obstacles. If you selected the "Allow trace necking at pads" check box (Options dialog box > **Routing** category > **General** subcategory), you can automatically neck traces by clicking to add a corner. The trace width changes automatically to the minimum trace width. After you clear the obstacles, click again to return to the previous trace width.

8. End the trace as described in Ending Traces.

Ending Traces

SailWind Router provides three ways to end a trace while interactively routing.

Note:
Differential pairs require some specific steps. For more information, see Ending Differential Pair Traces.

Completing a Trace Connection

When the pointer is near a valid completion point, the pointer changes to a double circle. During rerouting, the pointer changes to a single circle to indicate a valid completion point.

Procedure

Click on the object where you want to complete the trace when a single or double circle is visible over the completion point. As an alternative, when the pointer is near the object at which you want to complete the trace, right-click, and click the **Complete** popup menu item.

Results

The trace completes, maintaining same net rules if possible. The trace width and length may also be adjusted. When you complete a trace, the autorouter passes may run automatically in the background; for example, if the object where you want to complete the trace is on another layer or far from the pointer. If a net has a length rule assigned, SailWind Router may add length automatically when you use the **Complete** popup menu item.

- If you encounter difficulty using Complete, you may need to switch to component rules before clicking **Complete**.

- If DRC Prevent mode is enabled and you receive an error message while trying to complete a trace, or if an operation does not complete, right-click and click the **Explain Last Error** menu item. DRC will explain the cause. Error markers appear if you have made them visible. If DRC Explain mode is enabled instead, the error is explained for you automatically and you can either accept the error (by right-clicking and clicking the **Continue with Errors** menu item) or you can reject it (by pressing the Esc key or right-clicking and clicking the **Cancel** popup menu item). For more information, see “Design Rule Checking Dialog Box” on page 405.

- If you are using DRC Prevent or Explain mode for clearance checking, you can temporarily turn off clearance checking in order to complete a trace. While interactively routing, right-click and click the **Check Clearance** popup menu item. You can then complete the trace, even if you are creating a clearance violation. With Check Clearance disabled, you can violate clearance boundaries and create clearance violations. When you click to complete an operation, error markers appear in the workspace at the location of each error and error messages appear in the spreadsheet window. You can continue with additional operations with Check Clearance disabled. To reactivate clearance checking during routing, you must right-click and click the **Check Clearance** popup menu item again.

- You can also disable or enable clearance checking during trace editing operations such as Move and Stretch. To temporarily disable clearance checking during these editing operations, right-click and click the **Ignore Clearance** popup menu item. To reactivate clearance checking during trace editing, right-click and click the **Ignore Clearance** popup menu item again. For more information, see “Edit Traces Using DRC Modes” on page 346.

Related Topics

Autorouting by Selection

Ending the Trace With a Via or Test Point

You have the option of ending a trace with either a via or a test point.

Procedure

1. Right-click, choose the **End Via Mode** popup menu item, then choose one of the following end via modes:

   - **End No Via** — End without a via.

   - **End Via** — End with a via of the current type, on the via grid, at the closest location to the pointer. Obstacles move to accommodate the via.

   - **End Test Point** — End with a via of the current test point type, on the test point grid, at the closest location to the pointer and assign it as a test point. Obstacles move to accommodate the via.

   SailWind Router uses the end via mode you choose until you choose a new mode.

2. Press the Ctrl key and click to end the trace.

Results

When you end a trace, a smoothing pass runs automatically if the “Smooth traces on complete” check box is enabled in the Options dialog box > **Routing** category > “General subcategory” on page 450.

If you end a trace with a via or a test point while the pointer is over a trace from the same net on another layer, the via connects to that trace. Exceptions:

- The via does not connect to the trace if via sharing is disabled for the net, and if the number of traces attached to the via is greater than two.

- The via does not connect to the trace if the trace is from a different subnet. In this case, when possible, the trace is plowed to make room for the via.

Ending the Trace Leaving it Incomplete

You can choose to end the trace to complete it later.

Procedure

Right-click, and click the **End** popup menu item. The end via mode must be set to **End No Via**.

Results

If you click the **End** popup menu item while the point is over a completion point, the trace is completed instead of ended.

When you end a trace, a smoothing pass will run automatically if you select the “Smooth traces on complete” check box on the Options Dialog Box, Routing Category, General Subcategory.

Related Topics

Creating Differential Pair Traces

Creating Traces Using Component Rules

Interactive Routing Workflow

Change Trace Width

Ending Differential Pair Traces

Routing to an SMD Pin

Routing from an SMD Pin

Vias

Autorouting by Selection

Edit Traces Using DRC Modes

Creating Traces Using Component Rules

If you start routing from a component, via, or trace segment with component rules, the component rules constrain the new trace. You can choose when to stop using component rules and start using trace rules for the creation of both traces and differential pairs.

Restrictions and Limitations

- When routing a differential pair from component pins, SailWind Router uses component rules automatically and you route the pair separately, just as if you clicked Route Separately.
- When routing a differential pair from fanout vias, SailWind Router uses trace rules and routes the pair together.
- When you route between component pins with differing component rules, you must create a small trace segment that uses trace rules between the two trace segments that use component rules.
- If you start routing from a trace segment or node that is part of a trace using component rules, the new trace uses component rules.
- If you start routing from a trace segment or node that is not using component rules, the new trace continues to use trace rules.

- If you start routing from a trace segment or node that is part of a composite rule trace, the new trace uses the composite rules.

Note:
A composite rule trace is a trace that is attached to a pin that is shared by two subnets. For example, if a trace belongs to more than one pin pair with different rules, the trace uses composite rules, which is defined as the most restrictive common setting for each pin pair (if one pin pair has a clearance value of 12 and the other has a clearance value of 20, the trace inherits a composite clearance value of 20).

- If you start routing from a via that is part of a composite fanout, the new trace uses the composite rules. If you switch rules, the trace uses trace rules for one of the subnets attached to the via.

Procedure

1. Start interactive routing from a component pin or trace segment that has component rules. Component rules take effect automatically, and a “CR” graphic appears at the pointer to indicate component rules are active. The graphic also displays the reference designator of the component that owns the rules.
2. Right-click, and click the **Switch Rules** popup menu item. The “CR” graphic is removed from the pointer, and a virtual point is added where the rule sets change. Trace rules are now active.

Related Topics

Ending Traces Using Component Rules

Ending Traces Using Component Rules

While routing interactively, you can switch to component rules to complete a trace.

Procedure

1. Right-click and click the **Switch Rules** popup menu item to stop using trace rules and start using component rules. Component rules take effect automatically, and a “CR” graphic appears at the pointer to indicate component rules are active.
2. Complete the trace using one of the methods described in “Ending Traces” on page 306.
SailWind Router maintains component rules from the last corner that you added to the completion point. The last added corner becomes a virtual point.

Related Topics

Ending Differential Pair Traces

Creating Traces

Creating Traces Using Component Rules

Creating Differential Pair Traces

Routing to an SMD Pin

You can route to an SMD pin and complete the routing with the placement of a via on the SMD pin.

Procedure

1. Set up SMD vias using the **Same net** tab of the Design Properties dialog box or the Component Properties dialog box.
2. While interactively routing, point to the SMD pin on another layer where you want to end the trace, press Shift and click to add a via at the SMD pin. As an alternative, while interactively routing, point to the SMD pin on another layer where you want to end the trace and double-click. The Route autorouter pass completes the trace and adds a via at the SMD. Skip step 3.
3. Right-click and select a completion method from the popup menu.

Related Topics

Properties Dialog Box, Same Net Tab

Routing from an SMD Pin

You can interactively route from a surface mount component pin.

Procedure

1. Set up Vias at SMD options using the **Same net** tab of the Design Properties dialog box or the Component Properties dialog box.
2. Interactively route from a component pin.
3. Add a via. A via of the current type appears at the SMD pin. For more information, see "Vias" on page 324.
4. Continue routing.

Routing From an SMD Pin on Restricted Layers

You cannot interactively route from a surface mount component pin on a restricted layer. However, the Fanout autorouter pass will create a fanout for you.

Procedure

1. Set up Vias at SMD options using the **Same net** tab of the Design Properties dialog box or the Component Properties dialog box.
2. Select the surface mount component or pin.

3. Right-click and click the **Fanout** popup menu item to run the Fanout autorouter pass on the selected component or pin. SailWind Router creates fanouts from the SMD, with vias at the end of the stubs based on your fanout settings.

4. Interactively route from the vias.

Following Shapes

Use Follow while interactively routing to easily create traces that follow other shapes (another trace, board outline, copper, or trace keepout).

Restrictions and Limitations

You cannot follow 2D Lines or objects on other layers.

Procedure

1. While interactively routing, right-click and click the **Follow** popup menu item.

2. Click to select the object to follow. To ensure you select the correct object, move the pointer over the object; the object the trace will follow is highlighted.

3. Click to indicate where on the object you want to start the trace. While in Follow mode, you can still back up, and use arcs; however, all other Interactive Routing options are unavailable.

4. Move the pointer in the direction you want to place the trace.

5. Click to finish following the shape and continue with Interactive Routing.

Results

The new trace appears along the outline of the selected shape. SailWind Router follows all clearance rules; if there is an obstacle along the way, the new trace goes around it and then back to following the shape at the first opportunity.

Pre-existing elements in the design, especially traces and vias, remain untouched; Follow mode works as if you had protected all nets are protected and disabled the Plower. When the Follows process encounters a trace or via from another net, it routes the trace around the obstacle.

Add Length to Traces

Use an accordion to add length to a trace or differential pair so that it meets length rules for the net, electrical net, or pin pair.

For more information about setting options for routing to length constraints, see "Options Dialog Box, Routing Category, General Subcategory."

Adding Accordions at Default Amplitude

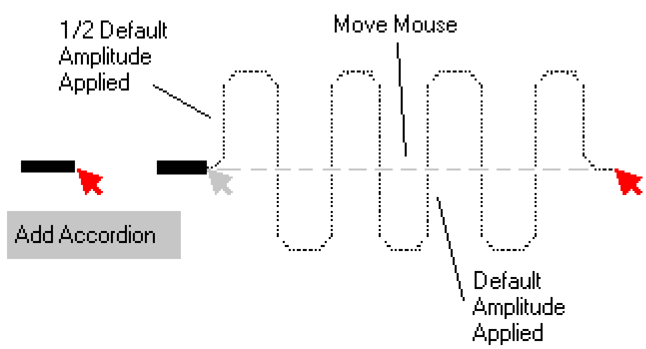
You can add length to your traces by adding accordions.

Procedure

1. While interactively routing, right-click and click the **Add Accordion** popup menu item.
2. Move the pointer in the direction you want to place the accordion. You can create accordions at any angle, depending on your routing angle.

   An accordion appears based on Minimum amplitude and Minimum gap settings (Options Dialog Box, Routing Category, General Subcategory). The amplitude of the accordion centers on the trace, as shown below.

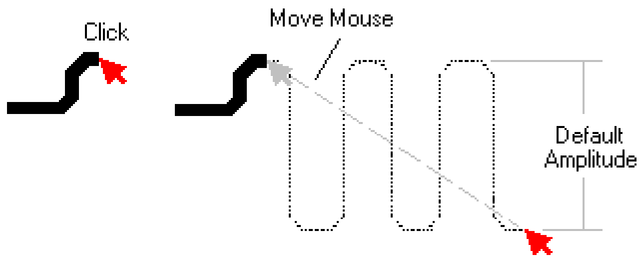
   Figure 73. Adding an Accordion at Default Amplitude

   The gap of the accordion is based on the Minimum gap setting (Options dialog box > **Routing** category > **Tune** subcategory).

   You can also add an accordion that uses the default amplitude, but does not center on the trace, as shown in below. Start the accordion, click to enter the first section, and move the pointer in the direction you want to place the accordion.

Figure 74. Adding a Non-centered Accordion



You can combine the accordion creation methods. For example, you can add an accordion that uses the default amplitude, and in one of the subsequent sections, click to redefine the amplitude. This is useful if you encounter an immovable obstacle.

3. Right-click and click the **Complete Accordion** popup menu item to end the accordion and continue routing. As an alternative, double-click to end the accordion.

4. Protect your accordions after you add them to prevent the autorouter (or other editing operations) from removing or adjusting them.

Adding Accordions with Custom Amplitude

Insert accordions and adjust the amplitude of individual sections as required. Create accordions at any angle depending on your routing angle.

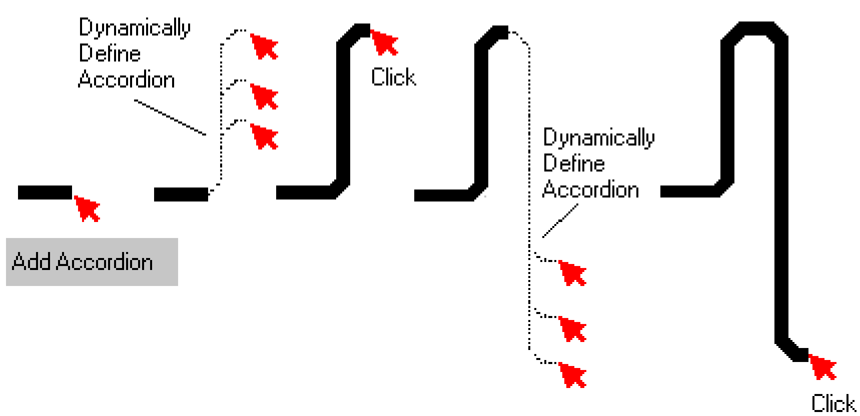
Procedure

1. While interactively routing, right-click and click the **Add Accordion** popup menu item.

2. To define the amplitude, move the pointer to define the highest point of the accordion, right-click, and click the **Add Section** popup menu item.

3. Move the pointer to define the lowest point of the accordion, right-click, and click **Add Section**.

Figure 75. Adding an Accordion with Custom Amplitude



4. Move the pointer in the direction you want to place the accordion.

5. Right-click and click the **Complete Accordion** popup menu item to end the accordion and continue routing. As an alternative, double-click to end the accordion. The gap of the accordion

is based on the Minimum gap setting in the Options dialog box > **Routing** category > **Tune** subcategory. You cannot adjust the gap while adding accordions.

6. Protect your accordions after you add them to prevent the autorouter (or other editing operations) from removing or adjusting them.

Add Accordions to Existing Traces

You can add an accordion if the existing trace does not meet length requirements after completion.

When you add accordions to existing traces, you have two choices:

- You can add an accordion so that just enough length is added to meet net length rules on page 315.
- You can specify which part of the trace on page 315 to convert to an accordion.

Adding an Accordion to Meet Length Rules

You can add an accordion to a trace, increasing its routed length so that it meets your length rules.

Procedure

1. Select the trace, trace segment, or trace path to which to add an accordion.

   Tip
   If you select a differential pair segment in the shoulder area, you can add an accordion to a single member of the pair. If you select a differential pair segment in the controlled gap area, you must add the accordion to both members of the pair.

2. Right-click and click the **Add Accordion** popup menu item. An accordion appears in the trace so that the net meets net length rules. SailWind Router generally adds the accordion to the middle of the selected trace or segment.

Adding an Accordion in a Specified Area

You can choose to specify which part of the trace to convert to an accordion.

Procedure

1. In the Selection Filter toolbar, click the **Path** button to enable path selection.
2. Click on the trace or trace segment to specify the start of the accordion location.
3. Click again to specify the end of the accordion. If you specify a differential pair segment in the shoulder area, you can add an accordion to a single member of the pair. If you specify a differential pair segment in the controlled gap area, you must add the accordion to both members of the pair.
4. Right-click and click the **Add Accordion** popup menu item. SailWind Router adds an accordion to the selected path within the trace or trace segment.

Related Topics

[Move Accordions]

[Deleting an Accordion]

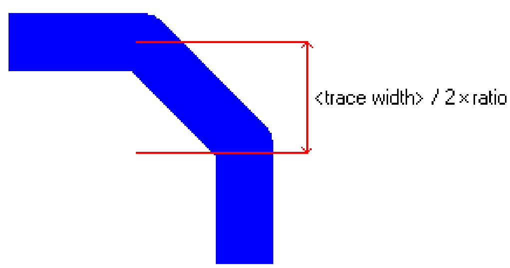
[Monitoring Trace Length]

Miters

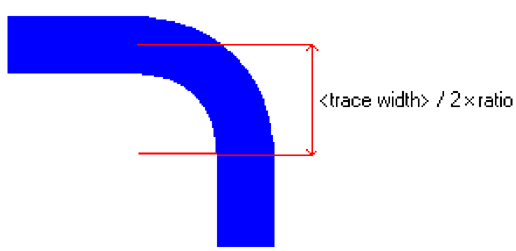
Use miters to specify values for replacing a corner with a diagonal segment or arc.

The miter ratio value, set on the **Routing** tab of the Options dialog box, specifies the miter ratio for accordion corners. SailWind Router calculates the radius using the following formula:

```
(<trace_width> / 2) x Ratio
```



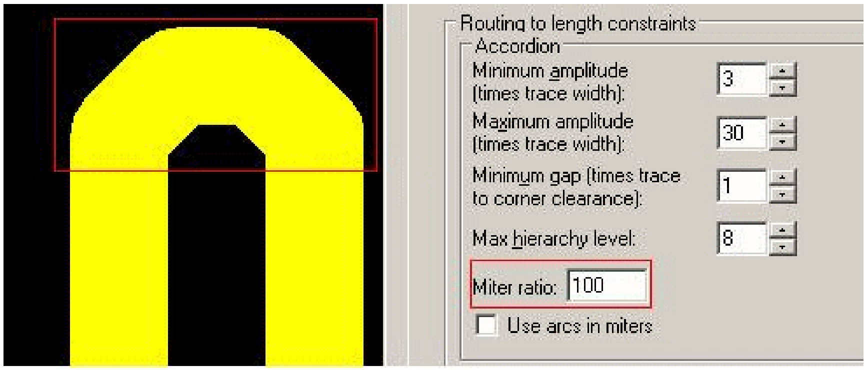
If “Use arcs” in the Miters area on the **Routing** tab is checked, the diagonal segment is replaced with an arc:



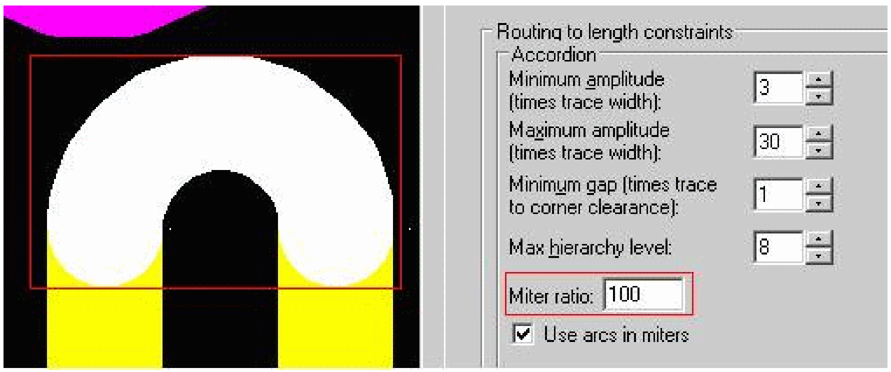
For example, if the trace width is 12 (set on the Trace Segment Properties dialog box), and you set the ratio to 1, the miter radius is 6 ((12 / 2) x 1 = 6). The miter segment length is about 8.49.

Accordions

When accordions use miters, the miter segment cannot be longer than the middle segment:

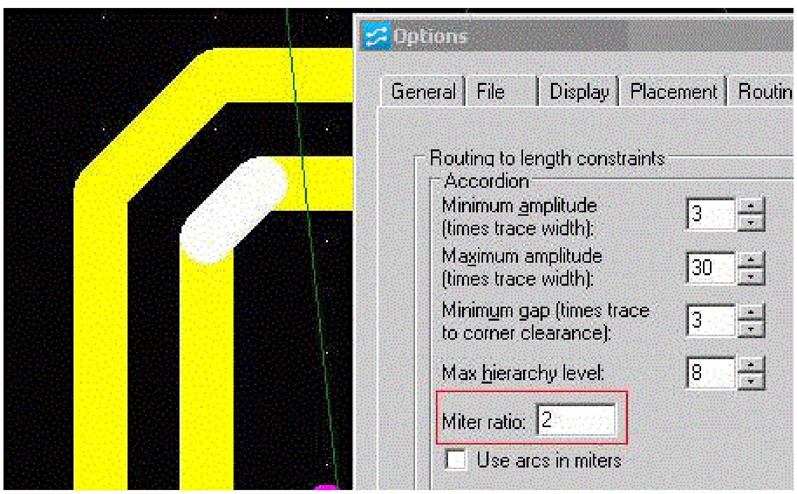


If accordions use arcs instead, the middle segment may be completely removed:



Differential Pair Miters

When you choose to add miters to differential pairs, SailWind Router generates miters for both parallel traces but calculates the miter size for the smaller (inside) miter segment based on the miter ratio setting.



Obstacles

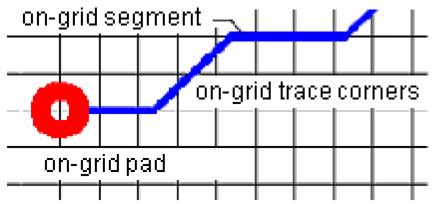
The Miter pass can create smaller miters if there are obstacles like pins or vias in the way.

Corners

SailWind Router places corners at the location that is closest to the pointer that meets current routing grid settings and design rules. When you select the “Snap Objects to Grid” check box for the Routing grid (on the **Grid** tab of the Design Properties dialog box), SailWind Router places corners on at least one grid coordinate.

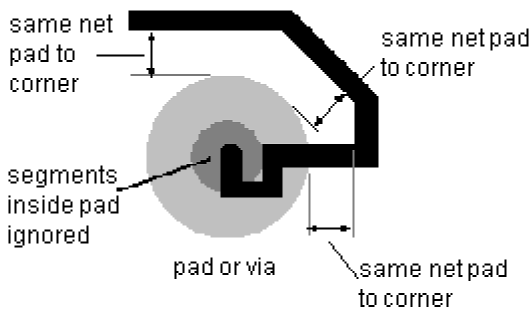
The following figure illustrates corner placement on a grid.

Figure 76. On-grid Trace Corners



When you place the first corner after exiting a pad, SailWind Router maintains first corner rules (Figure 77) unless you select the “Ignore first corner rules” check box on the **Pad Entry** tab of the Design Properties dialog box. This check box makes first corner rules soft.

Figure 77. First Corner Rule



Adding Corners While Interactively Routing

Adding Corners to Existing Traces

Adding Corners While Interactively Routing

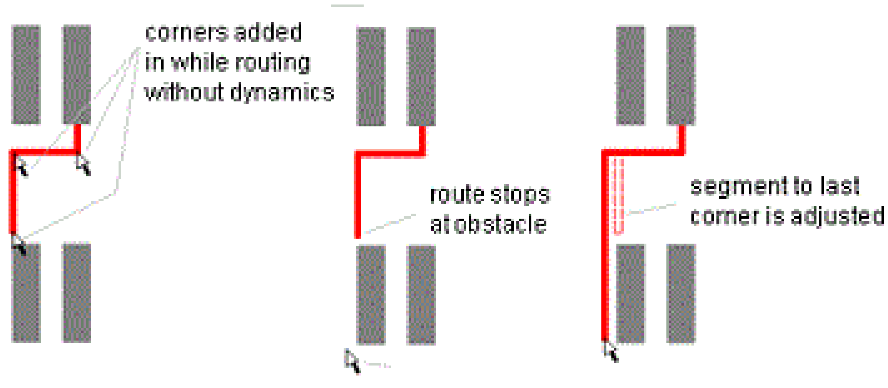
You can add corners when routing with or without dynamics.

Procedure

While interactively routing a trace, click to enter a corner. As an alternative, right-click and click the **Add Corner** popup menu item. SailWind Router removes any trace loops and adds the corner. If you click the **Add Corner** popup menu item when the end point of a trace is over a completion point, the trace completes.

Tip
If you add a corner, and the trace encounters an immovable obstacle (for example a component pin), the last-entered trace corner moves automatically to a new position that allows the trace to pass the obstacle, as shown below. This happens only when you route without dynamics and the “Adjust trace segments around obstacles” check box is selected (Options dialog box > **Routing** category > **General** subcategory).

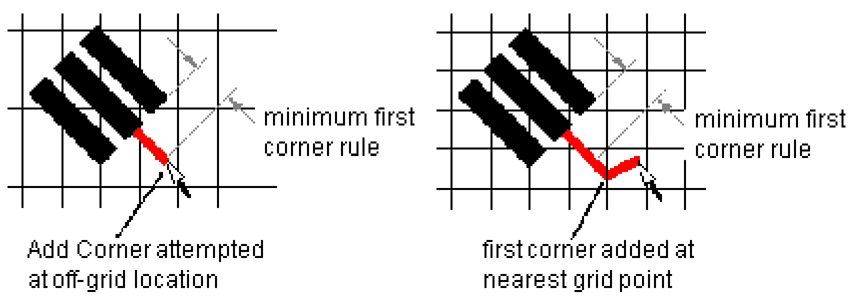
Figure 78. Interactive Routing—Avoiding Obstacles



When you dynamically route from an off-grid pad, SailWind Router adds corners at the point closest to the pointer and on the routing grid. When you route without dynamics from an off-grid pad, SailWind Router adds corners on a grid line, not at the grid intersection.

Pad entry settings are maintained when you route dynamically. Set pad entry on the **Pad Entry** tab of the Design Properties dialog box or Component Properties dialog box.

Figure 79. Routing from an Off-grid Pad



Adding Corners to Existing Traces

You can add corners to an existing trace if you have already routed a connection.

Procedure

1. Select a trace segment then right-click and click the **Add Corner** popup menu item. Alternatively, on the Route Editing toolbar, click the **Add Corner** button.

2. Move the pointer to the target location for the corner and click. A corner appears at the selection point. Grid lines also appear, indicating 45-degree angles from the end point of the trace. This makes it easier to place diagonal corners. When adding corners, SailWind Router does not smooth

adjacent segments, even if you have selected the “Smooth adjacent segments” check box (Options dialog box > **Routing** category > **General** subcategory).

Tip
Press and hold the Shift key while moving corners to temporarily override the current angle mode.

Related Topics

Moving Corners

Deleting Corners

Arcs

When routing interactively, you can add a free-drawn arc, or an arc of specified radius. This feature is also available when routing differential pairs. While routing a differential pair in arc mode, you cannot use the following commands: Add Via, Add Test Point, Add Accordion, Complete, Layer Toggle, Layer, Switch Rules.

Adding a Free-drawn Arc
Adding an Arc of Specified Radius

Adding a Free-drawn Arc

Add a free-drawn arc to a single trace, a differential pair, or an individual trace of a differential pair.

Procedure

1. While routing, click to locate the start point of the arc.

   Tip
   Ensure you protect arcs after adding them, as autorouting operations and plowing in interactive routing may remove unprotected arcs. To protect arcs as you add them, select the “Protect traces and vias when creating segments” check box in the Options dialog box > **Routing** category > **General** subcategory. As an alternative, while interactively routing, right-click and click the **Protect Traces and Vias** popup menu item.

2. Right-click and click the **Arc > Add Arc** popup menu item.
3. Move the pointer to draw the arc.
4. Click again to end the arc.

Adding an Arc of Specified Radius

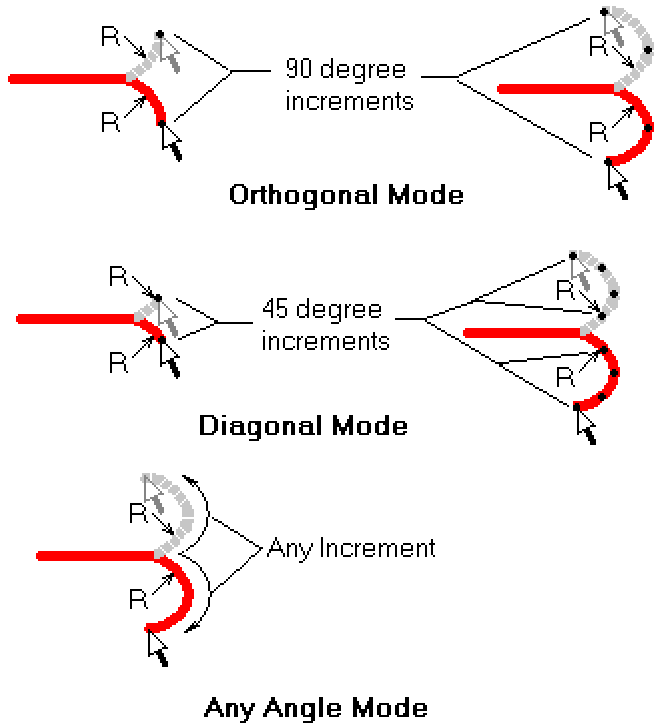
Add an arc of specified radius to a single trace, a differential pair, or an individual trace of a differential pair.

Restrictions and Limitations

- The end points of a fixed radius arc are restricted to those coincident with the routing angle and grid settings.
- You cannot create looping arcs. If you close the arc (make a circle), the circle is ignored and you continue to define the arc.

- The current routing angle determines the chord increment for the arc, as shown below.

Figure 80. Interactive Routing— Adding Arcs



Procedure

1. If necessary, set the desired radius using the “RAD” modeless command. If you are jointly routing a differential pair, the radius defines the smaller (inside) arc.

Tip
Ensure you protect arcs after adding them as autorouting operations and plowing in interactive routing may remove unprotected arcs. To protect arcs as you add them, select the “Protect traces and vias when creating segments” check box in the Options dialog box > **Routing** category > **General** subcategory. Alternatively, while interactively routing, right-click and click the **Protect Traces and Vias** popup menu item.

---

2. Click to locate the start point of the arc.

3. Right-click, and click the **Arc > Add Arc of Radius** popup menu item. The arc begins from the last entered corner, arc, or end point.

4. Move the pointer in the direction in which you want to create the arc. If you add an arc from a pad, or complete an arc at a pad, SailWind Router does not maintain pad entry and first corner rules.

5. Click to complete the arc and continue routing.

Related Topics

Moving Arcs

Deleting Arcs From Existing Traces

Vias

Add vias to route traces from the current layer to another layer in the design. The other layer then becomes the current layer. When you add a via, a via of the current type appears at a location that meets current via grid settings and design rules. SailWind Router plows movable obstacles to accommodate the new via.

If the DRC on page 405 mode for clearances is set to Prevent, and plowing is enabled, SailWind Router moves obstacles, creating room for the via without violating design rules.

SailWind Router uses the current via type for the trace in progress, provided the via biasing rules (set on the **Via Biasing** tab of the properties dialog boxes) allow it. If the current via type is not allowed for the trace, SailWind Router selects another valid via type. After using a new or different via, SailWind Router uses it for subsequent via additions.

When routing traces in the controlled gap area of a differential pair, SailWind Router adds vias on both traces. You cannot add vias while routing a differential pair in arc mode.

Via Mode
Setting the Via Type
Protecting Vias
Adding Vias on the Via Grid
Using Layer Change and Add Via to Change to a Specific Layer
Adding Vias While Interactively Routing
Adding Vias to Existing Traces
Adding Stitching Vias

Via Mode

The via mode determines what kind of via type to use when inserting vias.

SailWind Router selects vias based on several requirements, including:

- Which vias are allowed for the net or pin pair, as determined by via biasing rules.
- Which vias are legal for the layer change, according to the layer drill settings you set in the Pad Stacks Properties dialog box in SailWind Layout.
- If multiple vias qualify, which via has the smallest pad size but is also larger than the trace width.

Table 81. Setting or Changing the Via Mode

Modeless Command	For this mode	To limit the via type to
VA	Automatic	Automatic selection from a list of all available vias
VP	Partial	Partial via types that meet the requirements mentioned above
VT	Through	Through-hole via types that meet the requirements mentioned above

Setting the Via Type

When you add a via, its via type becomes the current via type and SailWind Router continues to use it when you add subsequent vias. When adding subsequent vias, if SailWind Router cannot use the current via type, or if it conflicts with the via mode, the via type is changed to “undefined” and then automatically selected based on the current via mode (VA, VT, or VP).

When adding a via, SailWind Router follows the settings on the **Via Biasing** tab of the following dialog boxes:

- Net Properties dialog box
- Pin Pair Properties dialog box
- Component Properties dialog box

Procedure

1. While routing a trace, right-click, choose the **Via Type** popup menu item, and click a via type from the list; for example, STANDARDVIA. The list contains the current via type, followed by a list of allowable via types in the design, followed by via mode commands.
2. To choose a via type that is not visible in the list, click the **Choose** popup menu item. The **Choose** options only appears on the menu if you have more than 12 available vias. The next via you add is of the new via type, if it meets via biasing rules and layer drill settings.

Results

If you have selected the “Turn on plower” check box (Options dialog box > **Routing** category > **General** subcategory) and you change the via type of an existing via, SailWind Router may plow traces and other movable obstacles to make room for the new via type.

Related Topics

Vias

Protecting Vias

Protect traces and vias as you add them to your design. Routing operations cannot rip up or otherwise modify the vias.

Procedure

1. Click the **Tools > Options** menu item; then, in the Options dialog box, click the **Routing** category > **General** subcategory.
2. Select the “Protect traces and vias when creating segments” check box. You can move protected vias if you select the “Shove protected traces when required” check box in the **Routing** tab of the Design Properties dialog box.

Adding Vias on the Via Grid

You can choose to add vias on a grid.

Procedure

1. With nothing selected in the design area, right-click and click the **Properties** popup menu item; then in the Properties dialog box, click the **Grid** tab.
2. Select the "Snap Objects to Grid" check box for the Via grid.

Tip
If you select the "Smooth adjacent segments" check box (Options dialog box > **Routing** category > **General** subcategory), SailWind Router smooths trace segments adjacent to the via.

Using Layer Change and Add Via to Change to a Specific Layer

Add Via automatically switches to the other routing layer (in a two-layer design) or to the other layer in the layer pair (for multiple-layer design). If you are routing on an unpaired layer, Add Via switches to the lowest numbered layer in the paired layer setting.

You can set layer pairs in the Options dialog box > **Routing** category > **General** subcategory. You can also combine a layer change operation with the Add Via command to change to a specific layer.

Procedure

1. Before adding the via, press Alt while scrolling the wheel button on the mouse.
2. This allows you to scroll through layers in the design. When you reach the layer you want to change to, add a via as described in the following sections.

Related Topics

Scrolling Through Layers Using the Wheel

Adding Vias While Interactively Routing

You can add vias while you route a trace interactively.

Restrictions and Limitations

You cannot add vias while routing a differential pair in arc mode.

Procedure

While interactively routing, press the Shift key and click. As an alternative, right-click and click the **Add Via** popup menu item.

Adding Vias to Existing Traces

You can choose to add vias to a trace that you have already routed.

Restrictions and Limitations

You cannot add vias or test points to protected objects. You can, however, add a via to a trace corner if at least one of the segments in the corner is unprotected.

Procedure

Select an existing trace, trace segment, trace corner, or T-junction, right-click and click the **Add Via** popup menu item. If adding a via to an existing trace changes a trace pattern, SailWind Router maintains first corner rules, which can result in changes to the trace pattern entering or exiting the via.

Adding Stitching Vias

You can also add vias to unroutes. Vias in unroutes are called *stitching vias*.

Procedure

1. Select an unroute, right-click, and click the **Add Via** popup menu item.

   > **Tip**
   > Protect vias from being moved or ripped up by clicking the **Tools > Options** menu item > **Routing** category > / **General** subcategory. Select the "Protect traces and vias when creating segments" check box. You can move protected vias if you select the "Shove protected traces when required" check box in the **Routing** tab of the Design Properties dialog box.

2. Move the pointer to the via location, and click to place the via. A via appears connectedly directly to the net. In addition to maintaining via biasing rules, adding a via to an unroute also maintains component rules. If the components pins attached to the selected unroute have mismatched via biasing rules, the via type is ignored.

Related Topics

Ending Traces

Via Mode

Setting the Via Type

Setting the Via Pair Pattern

Moving Vias

Deleting Vias

Showing Via Guides

Route to Coppers and Unroute Display

Both SailWind Layout and SailWind Router support routing to a copper. In some cases, however, SailWind Layout does not see that coppers are joined, and displays unroutes. In contrast, SailWind Router does not display unroutes and it does not add routes because it sees the objects as joined. SailWind Router's behavior saves in the routed design file where it can pass back to SailWind Layout.

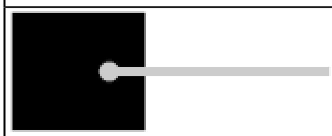
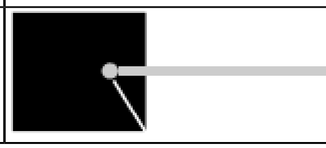
To display the design’s unroutes again in SailWind Layout, perform a Length Minimization or modify the plane net routing in SailWind Layout.

SailWind Layout and SailWind Router exhibit this behavior in certain instances.

Connecting Through a Via

You can connect to a copper from a different layer through a via. In the figure below, the trace and the copper are on different layers. SailWind Router does not show an unroute because trace and copper are considered connected. SailWind Layout, however, shows an unroute because it does not consider a connection from another layer.

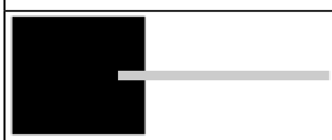
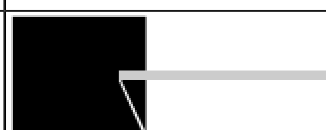
Table 82. Connecting to a Copper Through a Via From a Different Layer

SailWind Router does not show an unroute	SailWind Layout shows an unroute
	

Connecting an Open or Closed Copper

You can connect an open copper with closed copper on the same layer in SailWind Router. In the figure below, the trace (open copper) is on the same layer as the copper (closed copper). SailWind Router does not show an unroute because open and closed coppers are considered connected. SailWind Layout, however, shows an unroute because it does not consider a connection between open and closed copper.

Table 83. Connecting an Open or Closed Copper

SailWind Router does not show an unroute	SailWind Layout shows an unroute
	

Differential Pairs

SailWind Router fully supports the implementation of differential pairs.

Differential Pair Principles

Differential pairs are traces that SailWind Router routes side-by-side, separated by a fixed routing gap for as much of the overall length as practical. Differential pairs typically transmit two electrical signals that are driven 180 degrees out of phase from each other. The fixed routing gap contributes to the specific and uniform impedance required by differential signaling.

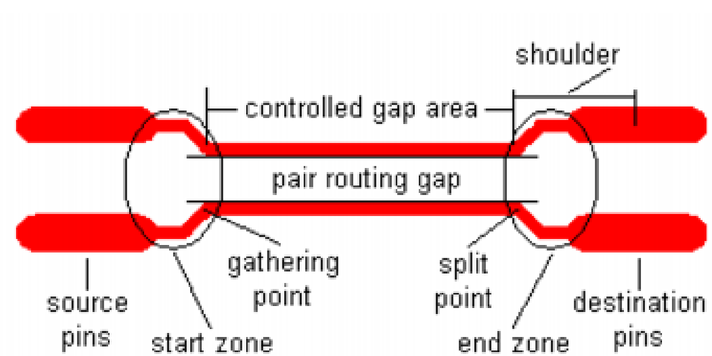
SailWind Router respects differential pair objects created by SailWind Layout.

This section introduces the routing elements, characteristics, and routing steps unique to differential pair routing.

Differential Pair Routing Elements

The figure below illustrates the various differential pair routing elements and how they fit together. The table contains a description of each routing element.

Figure 81. Differential Pair Elements



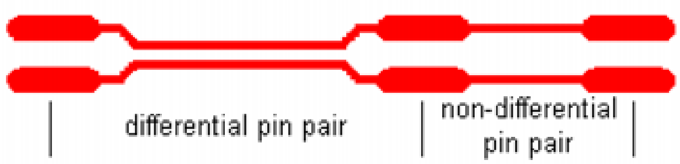
Tip
The source pins, destination pins, start zone, and end zone labels in the preceding graphic correspond to routing that starts at the left-hand set of pins and ends at the right-hand set of pins (the label positions reverse if the routing starts at the right-hand set of pins).

Table 84. Differential Pair Routing Elements

Routing element	Description
Controlled gap area	The part of the differential pair where the traces route in parallel, separated by the pair routing gap. The controlled gap area starts at the gathering point and ends at the split point.
Destination pins	The completion points of the differential pair.
End zone	The part of the differential pair between the split point and destination pins.
Gathering point	The point near the source pins where differential pair traces start to route at the pair routing gap.
Pair routing gap	The fixed edge-to-edge clearance between differential pair traces in the controlled gap area.
Shoulder	The trace segment between the source pin and the gathering point, or between the split point and the destination pin.
Source pins	The starting points of the differential pair.
Split point	The point near the destination pins where differential pair traces no longer route at the pair routing gap and where the traces route separately to the destination pins.
Start zone	The part of the differential pair between the source pins and gathering point.

A net may consist of differential pin pairs and non-differential pin pairs. The pair routing gap requirement does not apply to non-differential pin pairs.

Figure 82. Nets with Differential and Non-differential Pin Pairs



Defining Differential Pair Characteristics

Differential pairs typically route as impedance-matched traces, where each trace has the same specified impedance. The impedance is specified based on several factors and may depend on a particular IC manufacturer's recommendation or a standard imposed by the transmission medium through which the signals route; for example, LVDS and USB.

To yield differential pair routing with the desired impedance and signal propagation time, you supply the following differential pair properties to SailWind Router:

- Routing layer used to route the differential pair
- Pair routing gap

- Trace width
- Minimum and maximum trace lengths

You can use a high-speed analysis tool to calculate the set of differential pair property values that yield the desired impedance. To perform the calculations, you supply to the analysis tool the following layer stackup information for the board:

- Sequence of routing, dielectric, and plane layers
- Thickness of each layer
- The dielectric constant for each dielectric layer

In addition, you should learn the smallest edge-to-edge trace gap that the selected printed circuit board vendor can accurately produce without incurring extra cost. In theory, the smaller the gap, the higher the impedance.

Suggested Differential Pair Routing Practices

Some routing practices can produce differential pairs with incorrect impedance values, unmatched signal propagation delays, or elevated electromagnetic interference (EMI) emissions. You can promote good differential pair performance by observing the following routing practices:

- Route the differential pair on one layer
- Avoid changing the trace width
- Maximize the percentage of the overall length that is routed at the pair routing gap
- Minimize shoulder lengths
- Match shoulder lengths
- Use the 45 degree routing angle
- Route differential pairs perpendicular to traces on adjacent routing layers if no plane layer separates the routing layers

Routing Differential Pairs

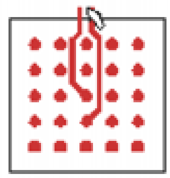
This section outlines the routing steps that are unique to routing differential pairs.

Routing From the Source Pins to the Gathering Point

The differential pair is routed from the source pins to the gathering point in as short a distance as practical while observing the design rules. The router also attempts to keep the differential pair traces together in the start zone. The gathering point can be set when the traces have sufficient clearance from source pins and other objects to be routed together at the pair routing gap.

This routing program attempts to match shoulder lengths. However the orientation of the routing relative to the source pins may produce unmatched shoulder lengths, as illustrated below.

Figure 83. Differential Pair with Unmatched Shoulder Lengths



To allow the differential pair traces to exit high-density components, such as BGAs, SailWind Router can route around obstacles in the start zone and reduce the trace width to the minimum trace width. The router is allowed to split the differential pair around objects in the start zone, even if you do not allow routing the differential pair around objects in the controlled gap area.

When routing the start zone interactively, you can exercise a high degree of control over the shoulder routing path by entering the route separately mode and routing the individual traces to the gathering point. At the gathering point, you can exit the route separately mode and route the traces together.

Routing in the Controlled Gap Area

Between the gathering point and split point, the differential pair traces route together at the pair routing gap and at a constant trace width. During autorouting, the traces do not split around obstacles in the controlled gap area unless you specifically allow it. When routing interactively, you can always split around obstacles in the controlled gap area.

Routing from the Split Point to the Destination Pins

The differential pair routes from the split point to the destination pins in as short a distance as practical while observing the design rules set for the pair, such as minimum length. After setting the split point, the router routes the traces separately to the destination pins.

SailWind Router can route around obstacles in the end zone, just as it can when routing from the source pins to the gathering point.

Routing Around Obstacles

During autorouting, you can allow the differential pair to split around small obstacles in the controlled gap area by selecting the “Allow pair to split around obstacles” check box on the Differential Pair Properties dialog box. When routing interactively, you can always split around obstacles in the controlled gap area.

To prevent the pair from splitting too far apart or for too long a distance, you can also specify in the dialog box the maximum obstacle size and maximum obstacle quantity properties. Obstacles in the start zone or end zone are not counted against the maximum obstacle quantity you set.

Restricting Layer Changes

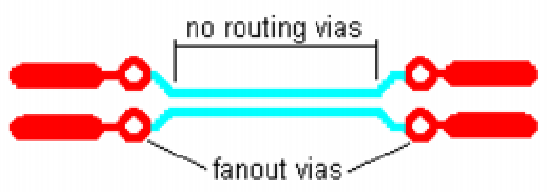
You can force SailWind Router to route the controlled gap area on one layer by selecting the “Restrict layer changes during autorouting” check box on the Differential Pair Properties dialog box. When you select this property, SailWind Router attempts to route the controlled gap area without vias and prevents you from interactively inserting vias into the controlled gap area.

To route the differential pair on one layer from a set of available layers, select the specific layers on the **Layer Biasing** tab in the Net Properties dialog box. The autorouter selects a layer from the set of available layers and routes the differential pair only on the selected layer. For example if you specify a set of four available layers, the autorouter selects one of the four layers and routes the differential pair only on the selected layer.

To route the differential pair only on a specific layer when using the autorouter, select only that specific layer on the **Layer Biasing** tab in the Net Properties dialog box.

You can add fanout vias to the differential pair routing, even when you choose to restrict layer changes. This behavior allows you to connect the differential pair to high-density BGA pins at the same time you route the controlled gap area on one layer.

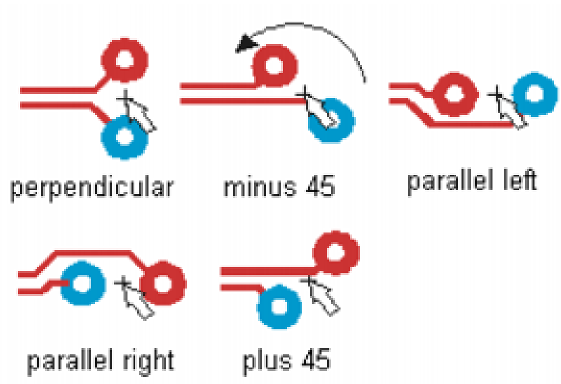
Figure 84. Differential Pair with Fanout Vias and No Routing Vias



Adding Via Pairs

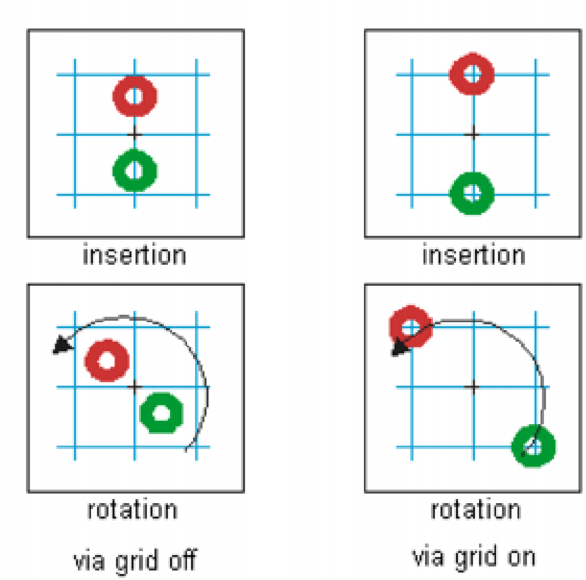
To change layers for the differential pair, SailWind Router inserts a pair of vias using one of the patterns illustrated below. (The via pattern names are based on the orientation of the via pair axis relative to the axis of the routing as it approaches the via pair.) The patterns keep the vias in a close, symmetrical pattern, and minimize the length of the connecting stubs. The perpendicular pattern is the default pattern, offering the best symmetry.

Figure 85. Via Pair Patterns



Via pairs snap to the via grid when you enable this feature. The following illustration shows the via pair reaction to the via grid.

Figure 86. Via Pair Reaction to Via Grid Setting

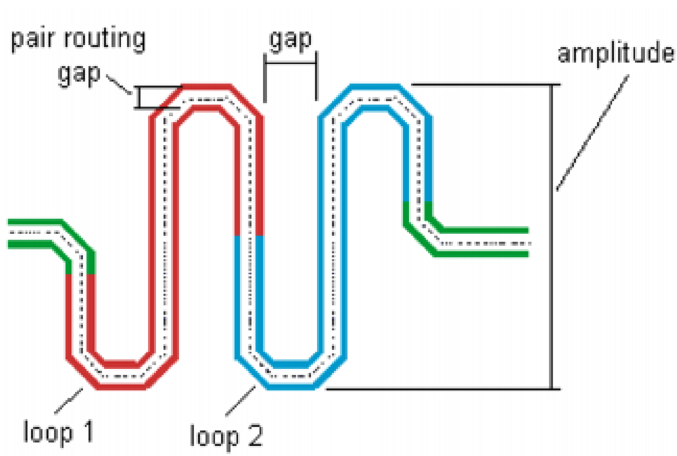


To add an individual via, route the traces separately. You can resume routing the traces together after placing both vias individually if both traces use the same routing layer. Or you can place the vias and complete the routing, and then you can interactively move the vias to the preferred locations.

Adding Length

To meet the differential pair's minimum length property, insert a symmetrical differential pair accordion into the controlled gap area. The autorouter can insert an accordion that meets the required length if the required space on the board is available, or you can interactively add an accordion.

Figure 87. Anatomy of a Differential Pair Accordion



For more information, refer to “Creating Differential Pair Traces” and “Ending Differential Pair Traces.”

Length Matching

If you run any autorouting operation or if you run the Tune pass type on a differential pair, the autorouter attempts to match the length of both pair members exactly. When possible, it adds length to the shorter pair member in the shoulder of the trace, without plowing or shoving obstacles.

The autorouter adds length by adding small accordions to the shorter trace. The autorouter attempts to add the accordions to the shoulder areas first. If neither shoulder area can accommodate the small accordion, the autorouter adds the accordion in the controlled gap area, as close as possible to the shoulder area.

Creating Differential Pair Traces

Create differential pair traces by routing interactively. Before routing, you need to choose a starting point for the differential pair and initiate interactive routing.

To route single traces, see "Creating Traces" on page 304.

Tip
You can add arcs while routing differential pairs.

Restrictions and Limitations

You cannot change the trace width while interactively routing differential pairs.

Procedure

1. On the main toolbar, click the **Route Editing** button. On the Route Editing toolbar, click the **Interactive Route** button; then select an unroute.

2. Define a differential pair either in SailWind Layout or in this program. For more information, see "Creating a New Object in a Secondary Group" on page 53.

3. On the standard toolbar, in the Layer list, choose the layer where the objects that you want to route reside. You cannot interactively route on restricted layers. Set layer restrictions on the **Layers** tab or the **Layer Biasing** tab of the Design Properties dialog box.

4. With the differential pair selected, right-click and click the **Interactive Route** popup menu item. You only need to select one of the unroutes in the differential pair. The traces route jointly. SailWind Router removes hatch outlines automatically when you start interactive routing.

5. Move the pointer to route the traces. As you move the pointer, guard bands may appear, indicating obstacles.

6. To route the traces individually, right-click and click the **Route Separately** popup menu item. After routing the trace to the point you want, right-click and click the **Switch Trace** popup menu item to route the other trace. After routing the other trace to the same point, right-click and click **Route Separately** (popup menu item) again to resume routing the traces jointly. As an alternative, use the Tab key to switch between members of the differential pair.

7. Click to anchor the gathering point, and start routing the controlled gap area. You have reached a valid gathering point when the traces route together at the pair routing gap and—if enabled—guard bands highlight any obstructions.

8. If necessary, add the following while routing the controlled gap zone:

    - Corners on page 318 — Needed if you are routing without dynamics. Optional if routing with dynamics. Corners change the direction of a trace.

    - "Arcs" on page 321 — To create arc segments.

- Splits on page 340 — Allow the traces to pass around both sides of an obstacle by temporarily exceeding the pair routing gap.
- Via pairs on page 336— To change layers while routing. To change via pair orientation, right-click, choose the **Via Pattern** popup menu item, and click the new via pattern.
- Test points on page — To make your design testable.
- Length on page 312— To meet length rules.

9. To change layers while routing, right-click, choose the **Layer** popup menu item, and click the layer on which you want to continue routing. To select a layer that is not listed, click the **Choose** popup menu item. The trace segment, from the last corner, moves to the new layer. Alternatively, use Layer Toggle to switch to the other layer in the layer pair automatically. You can also scroll through layers using the wheel button on your mouse.

   For more information, see “Scrolling Through Layers Using the Wheel” on page 126.

   If necessary, SailWind Router adds vias (of the current via type) or automatically removes them when you change layers. It does not remove vias that are also test points.

10. To change the routing angle while routing, right-click, choose the **Routing Angle** popup menu item, and click the new angle.

11. End the trace as described in “Ending Differential Pair Traces” on page 338.

Related Topics

Creating Matched Length Net Groups

Creating Matched Length Pin Pair Groups

Creating Traces

Creating Traces Using Component Rules

Ending Traces

Modeless Commands and Shortcut Keys

Setting the Via Pair Pattern

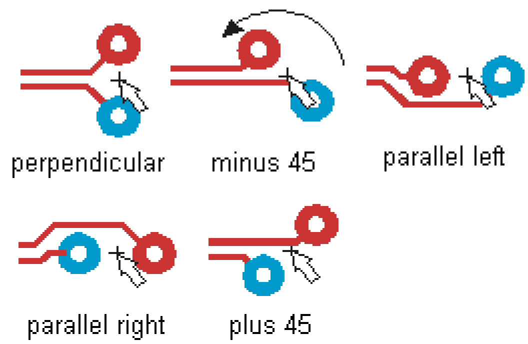
Setting the Via Pair Pattern

When you route differential pairs, the via pair pattern determines the orientation of the via pair axis relative to the axis of the routing as it approaches the via pair. You can choose to change the via pair pattern while routing interactively.

Procedure

1. Right-click, choose the **Via Pattern** popup memory item, and click the desired pattern.
2. A smoothing pass runs automatically after you change the via pair pattern. The following illustration shows the available via pair patterns:

Figure 88. Available Via Pair Patterns



Related Topics

Vias

Moving Vias

Deleting Vias

Ending Differential Pair Traces

You can end differential pair traces in one of three ways while routing interactively.

Completing Diff Pair Trace Connections
Ending the Diff Pair Traces With a Via Pair or Test Point Pair
Ending the Diff Pair Traces Leaving Them Incomplete

Completing Diff Pair Trace Connections

SailWind Router offers specialized commands for completing differential pair traces.

Restrictions and Limitations

You cannot use the Complete command to complete traces while routing a differential pair in arc mode.

Procedure

1. When you near the completion point, right-click and click the **Route Separately** popup menu item to route an individual trace to the destination pin. When you are using Route Separately, the pointer changes to a double circle when the pointer is near a valid completion point. During rerouting, the pointer changes to a single circle to indicate a valid completion point.

   Alternatively, when the pointer is near the object at which you want to complete the trace, right-click and click the **Complete** menu item, or double-click.

2. Click on the object where you want to complete the trace.

3. To route the other trace first, right-click and click the **Switch Trace** popup menu item. As an alternative, use the Tab shortcut key to switch traces. When the first differential pair trace is completed, the other trace is automatically selected for completion.

Results

If you have enabled Design Rule Checking (DRC) Prevent mode and you receive an error message while trying to complete a trace, or if an operation does not complete, right-click and click the **Explain Last Error** popup menu item. The DRC explains the cause. Error markers appear if you have made them visible. If you have enabled DRC Explain mode instead, the DRC explains the error automatically and you can either accept the error (right-click and click the **Continue with Errors** popup menu item) or you can reject it (press the Esc key or right-click and click the **Cancel** popup menu item).

If you are using DRC Prevent or Explain mode for clearance checking, you can temporarily turn off clearance checking in order to complete a trace. To do so, while routing, right-click and click the **Check Clearance** popup menu item. You can then complete the trace, even if you are creating a clearance violation. With clearance checking disabled, you can violate clearance boundaries and create clearance violations. When you click to complete an operation, error markers appear in the workspace at the location of each error, and error messages appear in the spreadsheet window. You can continue with additional operations with clearance checking disabled. To reactivate clearance checking during routing, right-click and click the **Check Clearance** popup menu item.

Restriction:
You can only use DRC with interactive routing, trace editing, and placement editing operations.

Ending the Diff Pair Traces With a Via Pair or Test Point Pair

You can choose to end traces with a via pair or test point pair.

Restrictions and Limitations

You cannot end any trace with a via or test point if the current trace segment is an arc. If you end a trace while in an arc, the trace ends with no via or test point, regardless of the End Via mode setting.

Procedure

1. Right-click, choose the **End Via Mode** popup menu item, and click one of the following end via modes:

    - **End No Via** — End without a via pair.

    - **End Via** — End with a via pair of the current type, on the via grid, at the closest location to the pointer. Obstacles move to accommodate the via pair.

    - **End Test Point** — End with a via pair of the current test point type, on the test point grid, at the closest location to the pointer; however, the via pair is also a test point. Obstacles move to accommodate the via pair.

    SailWind Router uses the end via mode you choose until you choose a new mode.

2. Press Ctrl+click to end the trace with a via or test point, or right-click and click the **End** popup menu item.

Ending the Diff Pair Traces Leaving Them Incomplete

You can choose to end the traces at an intermediate location and finish routing them later.

Procedure

1. Right-click and click the **End** popup menu item. Unless you are ending from an arc, you must set the end via mode to End No Via. (When ending from an arc, traces automatically end with no vias or test points.)

2. If you click **End** (popup menu item) while the point is over a completion point, the traces completed instead of end.

3. When you end the traces, a smoothing pass runs automatically if you select the Smooth traces on complete box on the Options Dialog Box, Routing Category, General Subcategory.

Related Topics

Ending Traces Using Component Rules

Creating Differential Pair Traces

Creating Traces Using Component Rules

Moving Vias

Deleting Vias

Autorouting by Selection

Splitting Differential Pair Traces

When routing a differential pair in the controlled gap area, you can split the traces to route them around an obstacle. You might split differential pair traces when you cannot route both traces together around an obstacle. SailWind Router always enables splitting in the start zone and end zone.

Restrictions and Limitations

Differential pairs can split around only small obstructions, such as through vias and through pins.

Procedure

1. While routing interactively, right-click and click the **Split Trace Segments** popup menu item.
2. The next click completes the split operation, allowing you to continue interactively routing the differential pair.

Related Topics

Creating Differential Pair Traces

Changing Trace Properties While Routing

If necessary, you can change trace properties while interactively routing.

Procedure

1. To change layers while routing, right-click, choose the **Layer** popup menu item, and select the layer on which to continue routing. The trace segment, from the last corner or from the beginning of the trace, moves to the new layer.

   To choose a layer that is not listed, click the **Choose** popup menu item.

   Alternatively, right-click and click the **Layer Toggle** popup menu item to switch to the other layer in the layer pair. You can also scroll through layers using Alt+wheel button.

   For more information, see “Scrolling Through Layers Using the Wheel” on page 126.

   If necessary, the software adds or removes vias (of the current via type) automatically when you change layers. The software does not remove vias if they are also test points.

2. To change the width of the traces while routing, right-click, choose the **Width** popup menu item, and click a width. To enter a width that is not listed, click **Set** (popup menu item).

   For more information, see “Change Trace Width” on page 353.

3. To change the routing angle while routing, right-click, choose the **Routing Angle** popup menu item, and click the desired angle.
4. To change between dynamic routing and manual routing, right-click and click the **Dynamically Route** popup menu item. A check mark next to this option indicates it is enabled.

5. To temporarily change plower settings to complete a trace, right-click and choose one of the following:

    - **Push Trace Behind** — Allows the plower to push traces behind the pointer as you interactively route.
    - **No Plowing** — Turns off plowing for the trace in progress.
    - **Plow After Click** — Reroutes obstacles after you add a corner or via. If you select Plow After Click, it only plows after you enter a corner or a via.
    - **Plow with Pointer** — Reroutes obstacles as you move the pointer. As an alternative, press and hold the Alt key while routing to temporarily switch between real-time plowing and guided plowing with the pointer. When using guided plowing, the potential path appears in green, but turns to yellow if the path encounters immovable objects. If your current trace colors are green or yellow, SailWind Router uses white and red instead.
    - **Rip Up Obstructing Traces** — Unroutes obstacles.

Chapter 20
Manual Editing

SailWind Router provides you with tools for editing traces manually after adding them to your design.

Setting Up Trace Editing

After you complete routing a design, you may want to edit traces or areas of the board. Before you start to edit the traces, set up editing options that help control trace editing.

For more information, see Options Dialog Box, Routing Category, General Subcategory.

Tip
You cannot edit protected objects, elements of physical design reuses, or jumpers.

Showing Clearances Around Obstacles

You can view the minimum clearance around obstacles with transparent, colored outlines that surround them on the display. These outlines, called *guard bands*, point out obstacles to objects as you route or edit them. Guard bands also show the minimum distance you must maintain between objects.

Procedure

1. Choose the **Tools > Options** menu item; then, in the Options dialog box, click the **Global** category > **General** subcategory.
2. In the “Display settings” area, select the “Show guard bands on object” check box.

Plowing Traces

If desired, you can enable SailWind Router to push and shove other traces while you make edits.

Procedure

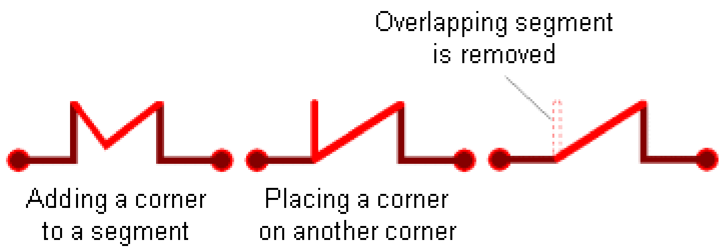
1. Choose the **Tools > Options** menu item; then, in the Options dialog box, click the **Routing** category > **General** subcategory.
2. In the Plower area, select the “Turn on plower” check box. SailWind Router maintains the Plower settings automatically during editing operations. For more information, see “Interactive Routing Setup” on page 291.

Remove Overlapping Segments

If you create an overlapping segment when editing a trace, SailWind Router automatically removes the overlap.

The following figure illustrates this principle:

Figure 89. Overlapping Segments Removed

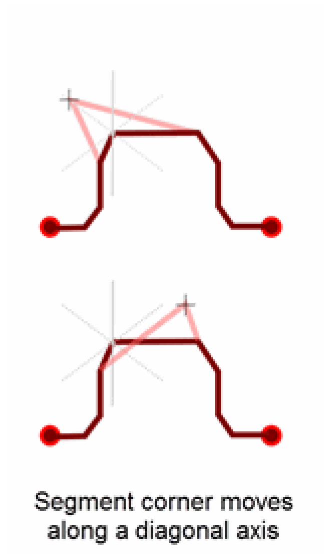


Changing the Routing Angle Mode

Editing operations automatically maintain the current routing angle. However, you can change the routing angle mode temporarily while editing a trace corner.

For example, if you add or move a segment corner, and the routing angle mode is Diagonal, the selected segment corner moves on a diagonal grid, and adjacent traces move in correspondence, as shown in the following figure.

Figure 90. Editing Traces— Routing Angle Mode



If desired, you can temporarily change the routing angle mode to Orthogonal to make an edit.

Restrictions and Limitations

SailWind Router does not maintain the routing angle when adding or moving a corner.

Procedure

1. Select a trace segment corner for editing, right-click, and choose the **Move** popup menu item.
2. While moving the corner, right-click and choose the **Routing Angle** popup menu option, then click the angle mode you want to use. As an alternative, use the AO, AD, or AA shortcut keys.

3. To temporarily change the routing angle, while editing, press Shift. This overrides the current routing angle. If you are in Orthogonal or Diagonal mode, you switch to Any Angle. If you are in Any Angle mode, you switch to Diagonal. When you release the Shift key, you return to the original routing angle mode.

Edit Traces Using DRC Modes

Editing operations automatically maintain current DRC settings. For example, if the DRC mode is set to Prevent, and an attempt to complete an editing operation creates a violation, you cannot complete the editing operation.

You can use three temporary DRC modes to help complete editing operations. During editing operations, you can access temporary DRC modes by use of a right-click menu.

The temporary DRC modes available for editing operations are as follows:

- **Ignore Clearance** — Temporarily disables clearance checking, which may make it possible to complete an editing operation.
- **Check Clearance During Move** — Checks clearance during editing. This option is mutually exclusive with the Check Clearance By Click mode.
- **Check Clearance By Click** — Checks clearance after you click to complete the editing operation; for example, when you click to indicate the new location of a moved trace. This option is mutually exclusive with the Check Clearance During Move mode.

In addition, you can right-click and choose the **Explain Last Error** popup menu item for additional information on any errors you encounter.

Related Topics

Setting Design Rule Checking

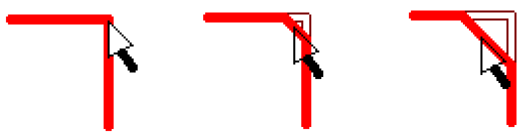
Stretch Traces and Arcs

The Stretch command shrinks and stretches trace objects, including arcs, trace segments, trace paths, and trace corners. SailWind Router pushes and shoves movable obstacles during stretch operations if you select the “Turn on plower” check box (Options dialog box > **Routing** category > **General** subcategory).

SailWind Router removes hatch outlines when you stretch traces.

When you stretch trace corners, the corners convert to miters, as shown in below.

Figure 91. Stretching Trace Corners



Note:
If a selected trace corner is also a virtual point, you cannot stretch the corner.

Stretching Trace Objects

You can stretch traces or traces segments.

Restrictions and Limitations

You cannot stretch a corner that has arcs attached to it.

Procedure

1. Select an object then right-click and click either **Stretch** or **Stretch Adjacent Segments** popup menu item.

   Restriction:
   The Stretch Adjacent Segments option appears only if you select a trace corner to move.

2. You can stretch trace segments or trace corners. Move the pointer to the new location and click to complete the stretch.

Stretching Arcs

You can stretch arcs or trace corners into arcs.

Procedure

1. Select an object then right-click and click the **Stretch Arc** popup menu item.

2. Move the pointer to the new location for the arc and click to complete the stretch. As a recommended practice, protect traces containing arcs before stretching them, as stretching a trace segment attached to an arc may remove arc corners.

Reroute Traces

You can change the pattern of an existing trace, or create a new pattern for a trace, a trace segment, or an unroute.

SailWind Router automatically removes hatch outlines when you reroute traces.

Rerouting with Interactive Routing

You can choose to reroute a trace interactively as one of the many editing options in SailWind Router.

Procedure

1. Select the trace, via, trace corner, or trace segment to reroute at the point from which you want to start rerouting.

2. Reroute the trace, creating a new trace as described in "Creating Traces" on page 304. If you have trouble completing the trace, check the following:

    - Trace sharing settings of the net to which you are routing in the "Allow junctions on" area (**Topology** tab of the Net Properties or Design Properties dialog box). If you disable the sharing settings, you may not be able to complete the trace.

    - The "Allow loops when rerouting" check box (Options dialog box > **Routing** category > **General** subcategory).

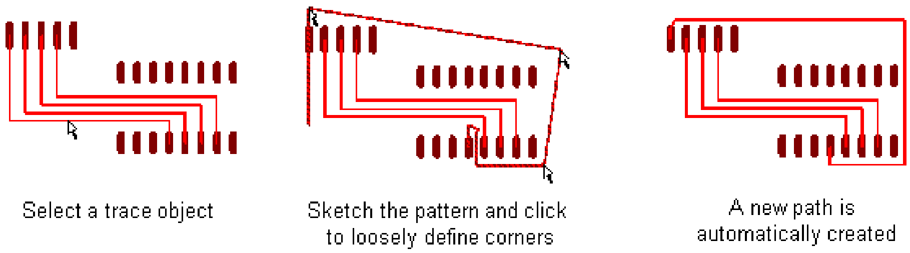
3. Double-click along another point in the same trace or net to complete the reroute. SailWind Router removes loops in the trace unless you select the "Allow Loops when Rerouting" check box (Options dialog box > **Routing** category > **General** subcategory). Clicking **Complete** when rerouting a previously complete trace has no effect. For more information, see "Design Rule Checking and Interactive Routing" on page 297.

Rerouting with Quick Route

The Quick Route feature lets you reroute a trace by defining a new trace pattern and then smoothing it. The Quick Route feature is useful when you make changes that result in a new path around an obstacle.

If you select the "Smooth traces on complete" check box in the Options dialog box > **Routing** category > **General** subcategory, the newly rerouted trace pattern is automatically smoothed upon completion. However, if the new trace pattern does not differ widely from the original pattern, the smoothed trace is likely to resemble the original. For this reason, Quick Route is most useful when creating a new path around an obstacle. An example of Quick Route rerouting appears in the following figure.

Figure 92. Quick Route Rerouting



Quick Route uses the push away feature. Traces are pushed away from obstacles regardless of whether you selected the “Push segments away from obstacles” box (Options dialog box > **Routing** category > **General** subcategory).

Procedure

1. Select a trace then right-click and click the **Quick Route** popup menu item. Alternatively, on the Route Editing toolbar, click the **Quick Route** button.
2. Move the pointer in the direction of the new trace pattern. You cannot change the layer of the trace or add vias while using Quick Route.
3. Click to enter any corners as you redefine the trace pattern.
4. When you get near the completion point, right-click and click the **Complete** menu item.

Rerouting with Split

Split divides trace segments into multiple segments by adding corners. The same segments or paths remain selected so you can add more corners.

You can split trace segments or trace paths.

Procedure

1. Select an object, right-click and click the **Split Trace Segments** popup menu item.
2. Click to indicate new corners.
3. Double-click when you finish redefining the trace pattern.

Trace Smoothing

Smooth trace patterns to remove unnecessary corners or reduce length. Trace smoothing frees up room on the board for additional traces and helps meet and maintain pad entry and first corner rules.

Smoothing is a single layer glossing operation. Smooth traces does not remove vias.

Smoothing converts arcs to line segments. Protect arcs before smoothing traces that contain arcs.

Smoothing Traces While Routing or Rerouting
Smoothing Traces While Plowing
Smoothing Existing Traces

Smoothing Traces While Routing or Rerouting

You can choose to smooth traces as the routing process completes them.

Procedure

1. Click the **Tools > Options** menu item; then in the Options dialog box, click the **Routing** category > **General** subcategory.

2. Select the “Smooth traces on complete” check box then click **OK**. Smoothing occurs as you complete traces during either interactive routing or editing.

Smoothing Traces While Plowing

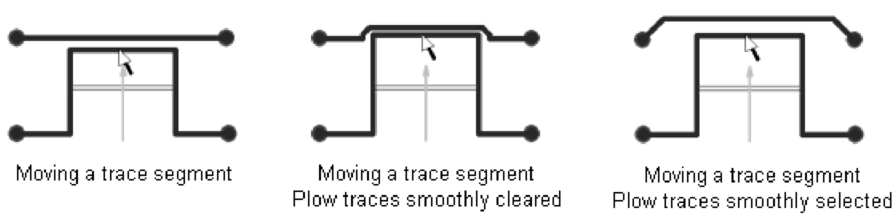
You can smooth obstructing traces as they are shoved during editing operations.

Procedure

1. Click the **Tools > Options** menu item; then in the Options dialog box, click the **Routing** category > **General** subcategory.

2. In the “Plower” area, select the “Plow traces smoothly” check box.

Figure 93. Plowing— Smoothing Traces



Related Topics

Interactive Routing Setup

Smoothing Existing Traces

You can smooth traces, trace segments, trace paths, and differential pair traces.

Procedure

1. Select a trace then right-click and click the **Smooth Trace Pattern** popup menu item.
2. If you select a differential pair, right-click and click the **Diff Pair Smooth** popup menu item. When smoothing differential pairs, select the entire differential pair. If you only select one of the traces in the pair, the single trace may be smoothed if the matching trace in the pair is not found, thus invalidating the differential pair.

Change Trace Width

You can change the width of traces both while interactively routing and after traces are complete. For example, you can change the width of a trace if its current width is an obstacle to completing the trace. By using a smaller trace width, you can complete the trace without violating design rules.

Restriction:
You cannot change the width of differential pair traces when routing in the controlled gap area.

Changing Trace Width While Routing Interactively
Changing the Width of Existing Traces

Changing Trace Width While Routing Interactively

If necessary, you can change the width of a trace while routing interactively. You can either select a new trace width from a list or type a specific width.

Procedure

1. While routing, right-click, choose the **Width** popup menu item, and either click a width or click **Set** to type a width.

   Alternatively, press the W key during interactive routing to specify a width to apply. The list of widths that appears includes the last widths set using the W shortcut key, and the minimum, recommended, and maximum widths (according to design rules).

   While routing with a reduced width, an indicator appears at the pointer. This indicator shows you when you can return to the previous trace width.

2. Click to continue routing the trace at the previous trace width.

3. If you are plowing while routing, objects move out of the way to accommodate a larger trace width.

Changing the Width of Existing Traces

You can change the width of traces, trace segments, trace paths, nets, and pin pairs.

Procedure

1. Select a trace that has a widths that you want to change.

2. Right-click, choose the **Width** popup menu item, and click either a width from the list or click **Set** to type a width. This list contains the recently used widths followed by the minimum, recommended, and maximum trace widths.

   Alternatively, click a trace and press the W key to apply a specific width.

Results

If the plower is enabled, movable trace objects move to accommodate the new width.

Converting a Trace to an Arc

You can convert the existing traces and trace corners to arcs. You can also change the radius of an arc.

Converting Existing Traces to Arcs

You can convert existing traces, trace segments, and trace paths to arcs.

Procedure

1. Select a trace segment then right-click and click the **Create/Modify Arc** popup menu item.
2. Move the pointer to define the radius of the arc.
3. Click to complete the arc. Alternatively, right-click and click the **Finish** popup menu item.

Converting Corners to Arcs

You can convert a corner to an arc by using the Stretch Arc command. The Stretch Arc command converts any trace segment that has convergent adjacent segments. You can create an arc with end points that are not tangent to adjacent segments. Adjacent segments stretch but retain their current angle. Using Stretch Arc automatically disables the “Push segments away from obstacles” check box (Options dialog box > **Routing** category > **General** subcategory).

Restrictions and Limitations

- You cannot use Stretch Arc on segments with vias at the end points.
- You cannot use Stretch Arc on segments that have an arc or a partially routed trace as an end point.

Procedure

1. Select a corner then right-click and click the **Stretch Arc** popup menu item.
2. Move the pointer to stretch the arc to the correct location.
3. Click to create the arc.

Changing the Radius of an Arc

You can adjust the radius of an existing arc. When modifying an existing arc, you cannot change the end points of the arc segment; they are fixed.

Procedure

1. Select an arc then right-click and click the **Create/Modify Arc** popup menu item.
2. Move the pointer to define the radius of the arc. You can also return the arc to a trace segment.
3. Click to complete the arc.

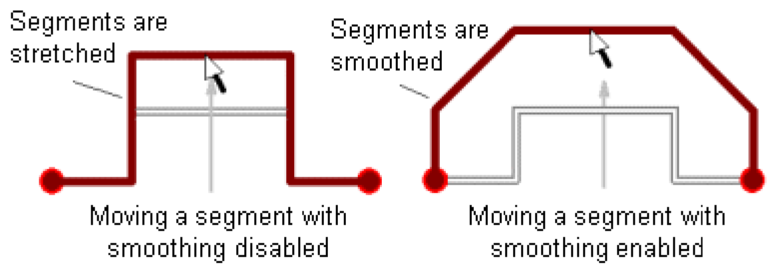
Moving Trace Objects

You can move trace objects, including trace segments, corners, T-junctions, vias, trace end points, and arcs. When you move a T-junction or a via, SailWind Router maintains pad entry rules, regardless of design rules or DRC modes.

If you select the “Smooth adjacent segments” check box (Options dialog box > **Routing** category > **General** subcategory), SailWind Router smooths adjacent trace segments as you move any trace object, except corners. When you move corners, it does not smooth adjacent segments.

When you move trace segments, adjacent segments do not move. Instead, SailWind Router adds new segments.

Figure 94. Moving Trace Segments



Procedure

1. Select a trace object then right-click and click the **Move** menu item.

   Tip
   Unlike SailWind Layout, SailWind Router maintains the angle mode.

   ---
2. To change the routing angle mode, right-click and choose the **Routing Angle** popup menu item; then click **Orthogonal**, **Diagonal**, or **Any Angle**.

   Alternatively, use the AO, AD, or AA shortcut keys.
3. Move the pointer to the new location for the object, and click to complete the move.

Related Topics

Moving Arcs

Deleting Arcs From Existing Traces

Moving Corners

Deleting Corners

Moving Vias

Deleting Vias

Moving Traces to Other Layers

Options Dialog Box, Global Category, General Subcategory

Moving Traces to Other Layers

You can move traces, trace segments, nets, paths, and pin pairs to other layers. When you select pin pairs or nets, all trace segments in the pin pair or net move to the new layer, regardless of their current layer. The same is true when you move multiple trace segments. If necessary, SailWind Router adds or removes vias of the current via type automatically when you change layers. SailWind Router does not, however, remove protected vias or vias set up as test points.

Restrictions and Limitations

You cannot move a protected trace or any portion of the protected trace.

Procedure

1. Select the objects to move.

   Tip
   Before changing layers, press Alt while scrolling the wheel button on the mouse to preview layers.

2. Right-click, choose the **Layer** popup menu item, and click to select a layer from the list or click **Choose** to move to a layer that does not appear in the list.

   The list contains the most recently visited, valid layers followed by other valid layers. If selected objects have conflicting layer biasing rules, invalid layers are not presented. If plowing is enabled, movable trace objects on the new layer are plowed to accommodate the new trace segments.

Results

When moving traces to other layers while the DRC mode on page 405 is set to Prevent:

- SailWind Router adjusts trace widths to support conditional rules on the new layer.
- Layer changes are cancel automatically if the layer change results in DRC violations.

Related Topics

Scrolling Through Layers Using the Wheel

Moving Trace Objects

Move Accordions

SailWind Router does not treat accordions as objects in the database; it treats them as a collection of trace segments. You can therefore move the trace segments, not the entire accordion.

For more information, see "Moving Trace Objects."

Deleting an Accordion

You can remove accordions while interactively routing the same way that you delete traces.

Procedure

While routing interactively, right-click and click the **Cancel** popup menu item or press the Esc key. While adding accordions, you can remove sections of the accordions by right-clicking and clicking the **Backup** popup menu item or pressing the Backspace key. If you click **Backup** after you complete the accordion, SailWind Router removes the entire accordion.

Related Topics

Add Length to Traces

Moving Arcs

When you move an arc, it does not modify the current shape; it modifies the position only.

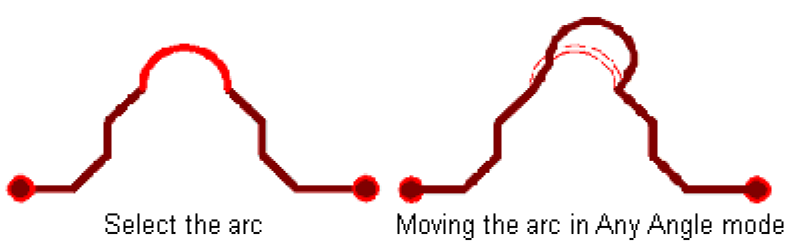
Procedure

1. Click on an arc then right-click and click the **Move** popup menu item. Alternatively, on the Route Editing toolbar, click the **Move** button.

2. To change the routing angle mode, right-click and choose the **Routing Angle** popup menu item; then click **Orthogonal**, **Diagonal**, or **Any Angle**.

3. Move the pointer to the new location for the arc and click to complete the move. If you select the "Smooth adjacent segments" check box (Options dialog box > **Routing** category > **General** subcategory), SailWind Router smooths adjacent trace segments as you move segments adjacent to the arc.

Figure 95. Moving an Arc



Deleting Arcs From Existing Traces

You can remove an arc from an existing trace and replace it with a trace segment.

Procedure

1. Click an arc to select it then press the Backspace key. Alternatively, right-click and choose the **Unroute** popup menu item.

   SailWind Router removes the arc from the trace, leaving an unrouted gap between trace segments.

2. Select one of the trace segments to either side of the unrouted gap and click the **Interactive Route** toolbar button or right-click and click the **Interactive Route** popup menu item.

3. Route a new trace between segments to complete the gap.

Related Topics

Deleting Existing Traces or Parts of Traces

Moving Trace Objects

Arcs

Options Dialog Box, Global Category, General Subcategory

Moving Corners

You can move corners from one place in the trace segment to another.

Restrictions and Limitations

- If the trace corner or T-junction is also a virtual point, you cannot move the corner or T-junction to a location that creates a subnet violation.

- When moving corners, SailWind Router does not smooth adjacent segments even if you select the “Smooth adjacent segments” check box (Options dialog box > **Routing** category > **General** subcategory).

Procedure

1. Click a trace corner to select it then right-click and click the **Move** popup menu item.

2. Move the pointer to the location at which to place the corner and click.

Deleting Corners

You can delete corners while interactively routing.

Restrictions and Limitations

You cannot delete corners or T-junctions that are also virtual points.

Procedure

1. Select the corner and press the Backspace key to remove the corner and unroute the adjacent trace segments.
2. Alternatively, to delete a corner without unrouting the trace, select the corner then right-click and click the **Delete** popup menu item. Alternatively, press the Delete key.

   SailWind Router removes the corner and replaces it with a straight trace segment. If the resulting trace segment encounters an obstacle, and you select the “Push segments away from obstacles” check box (Options dialog box > **Routing** category > **General** subcategory), the trace segment pushes away from the obstacle.

Related Topics

Corners

Deleting Existing Traces or Parts of Traces

Moving Trace Objects

Options Dialog Box, Global Category, General Subcategory

Move Test Points

You can move a test point via just as you can move a via.

For more information, see ““Moving Vias” on page 360.”

Deleting Test Points

If desired, you can delete a test point from a pin or a via. However, in doing so, you delete only the test point assignment, not the via to which it is assigned.

Procedure

1. Select a via or a pin that is a test point.
2. Right-click and click the **Properties** popup menu item.
3. Clear the “Serve as a test point” check box.
4. Click **OK**.

Deleting Test Point Vias

You can remove a test point via from the design.

Restrictions and Limitations

You cannot delete a test point if you selected the Preserve Test Points box in the Options dialog box > **Test Points** subcategory.

Procedure

Select a test point via then right-click and click the **Delete** popup menu item.

Related Topics

Ending Traces

Adding Test Points While Interactively Routing

Moving Vias

You can move unprotected vias. Trace segments attached to the via stretch as you move the via. When you move a via, SailWind Router maintains pad entry rules regardless of design rules or DRC modes.

Procedure

1. Click a via to select it then right-click and click the **Move** popup menu item.
2. Move the pointer to the new via location, and click to place the via.

Related Topics

Moving Trace Objects

Deleting Vias

You cannot delete protected vias or test points. You cannot use Delete to remove vias that are also virtual points: you must use the Unroute command instead.

Procedure

To delete vias during interactive routing, right-click and click **Backup**. To delete vias in completed traces, select a via then right-click and click **Delete**.

Results

If you assign a via as a test point, the test point also deletes when you clear the “Preserve test points” check box in the Options dialog box > **Test Points** category.

Related Topics

Adding Vias While Interactively Routing

Deleting Existing Traces or Parts of Traces

Moving Trace Objects

Options Dialog Box, Global Category, General Subcategory

Deleting a Trace While Interactively Routing

Delete a trace or part of a trace while routing.

Procedure

To remove the last entered trace, trace segment, corner, via, or accordion, right-click and click the **Backup** popup menu item. To exit interactive routing without completing any routing, press the Esc key. SailWind Router removes any trace segments and vias you added before pressing the key.

Deleting Existing Traces or Parts of Traces

Delete existing traces or parts of traces to reroute traces or segments.

Procedure

To delete traces or parts of traces, you must unroute the trace. See "Unrouting Nets and Trace Segments" on page 361. To remove trace corners, click to select a trace corner then right-click and click the **Delete** popup menu item. You cannot delete T-junctions or trace corners that are also virtual points.

Related Topics

Moving Arcs

Deleting Arcs From Existing Traces

Moving Corners

Deleting Corners

Moving Vias

Deleting Vias

Unrouting Nets and Trace Segments

Unroute a net or trace segment to delete it. You can unroute nets, paths, pin pairs, vias, and trace segments. You cannot unroute protected objects.

Prerequisites

To unroute traces, you must first select the "Allow trace unrouting" check box (**Routing** tab of the Design Properties dialog box).

Procedure

Select an object then right-click and click the **Unroute** popup menu item.

Tip
Unlike SailWind Layout, in SailWind Router, the Delete command removes a trace corner only. You must use the Unroute command in SailWind Router to unroute a trace.

Results

The unrouting command unroutes selected traces or traces connected to selected objects. The result of unrouting varies depending on the object you select:

- **Pin pair** — Unroutes the trace segment between the pin pair.
- **Net** — Unroutes the entire net trace.
- **Pin** — Deletes the trace segment entering the pin.
- **Component** — Deletes the trace segment entering each pin of the component.
- **Traces** — Unroutes the selected trace segment. When you unroute a trace, trace segment, or trace corner, adjacent vias are deleted.
- **Protected Objects** — Cannot be deleted. You cannot unroute protected objects. If you try to unroute them, an error message appears. You can click on the text link in the message to zoom to the trace, via, or jumper location. Use the object's Properties dialog box to unglue it. For more information see "“Design Rules” on page 154."
- **Objects with a No Rip Up Rule** — Cannot be unrouted. The no rip up rule indicates that the autorouter cannot rip up and reroute a routed net. You can set this option in the **Routing** tab of the Design Properties dialog box. You cannot unroute objects that have a no rip up rule. If you try to unroute them, an error message appears. You can click on the text link to zoom to the trace or via location. Use the object's Properties dialog box to enable rip up. For more information see "“Design Rules” on page 154."

Chapter 21
Verify the Design

SailWind Router provides multiple tools for checking a design for errors.

Checking a Design for Errors

Set up design verification to inspect your design for errors. You can then view the errors in the **Errors** tab of the Spreadsheet window or in the Error Properties dialog box.

Procedure

1. Click the **Tools > Options** menu item; then, in the Options dialog box, click the **Design Verification** category.
2. In the **Design Verification** category, select a scheme name from the Design verification scheme name list if you want to modify a predefined set of design checks. To create a new design verification scheme, set the individual design checks and properties you want, and then click **Save As**.
3. In the “Conduct checks” area, restrict what you want to check in the design if needed.
4. In the “Check design for” area, select the check boxes of the parameters you want to verify.

   Verification is based on the combination of your settings in both the **Design Verification** category and the **Fabrication** category of the Options dialog box.
5. When you finish making your verification settings, click **OK**.
6. On the Standard toolbar, click the **Design Verification** button.
7. If necessary, on the Design Verification toolbar, click the **Clear Errors** button to remove any error markers from the design.
8. On the Design Verification toolbar, choose the scheme you modified or created and saved in a previous step.
9. Click the **Verify Design** button to check the design. Error locations are flagged in the work area with round symbols indicating the error type. Errors also appear in the list on the **Errors** tab of the Spreadsheet window and in the Error Properties dialog box.
10. Correct any errors using Placement and Routing tools.

Related Topics

Verify Design Error Markers

Creating a Design Verification Scheme

Exporting the Design Verification Errors to a Report

Ignoring Errors

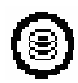
Options Dialog Box, Design Verification Category

Options Dialog Box, Fabrication Category

Verify Design Error Markers

SailWind Router can visibly mark errors in your design after performing design checks.

Table 85. Verify Design Error Marker

Marker	Description		
	Clearance		Testability
	Fabrication		High-speed
	Keepout		Board outline
	Assembly		Maximum via number **Tip** One marker is added for every net with too many vias.

Finding Errors

Use the error icon in the Type column of the Spreadsheet window for quick visual identification of the error.

Procedure

1. On the **Errors** tab of the Spreadsheet window, click the hyperlink representing the error explanation.

   The display zooms in and centers the corresponding error in the workspace.

2. Review the error information as follows:

- Expand the error description in the Type column to view additional error details.
- Use the Layer column to identify the layer on which the error is found.
- View the design rule violation details in the Rule Violated column. To view the entire error, resize the column by double-clicking the right-hand column separator.

Displaying Ignored Errors

If you have chosen to ignore errors, you can redisplay them.

Procedure

Perform one of the following:

- Click the **Display Ignored Errors** button on the Design Verification button
- Clear the “Ignore error during verification operations” check box in the Error Properties dialog box
- Clear the corresponding “Ignored” check box on the **Errors** tab of the Spreadsheet window

Related Topics

Options Dialog Box, Design Verification Category

Verify Design Error Markers

Ignoring Errors

Exporting the Design Verification Errors to a Report

You can create an error report of the Design Verification errors and store it in HTML format, letting you share the information with other members of the design team.

Procedure

1. Run Design Verification to display errors in the **Errors** tab of the spreadsheet window.
2. In the Spreadsheet window, click the **Errors** tab.
3. In the Spreadsheet window, click the **Export to HTML File** button.
4. When prompted, enter the name and location under which you want to store the file.

Related Topics

Finding Errors

Ignoring Errors

Ignore errors if they are known acceptable conditions.

Procedure

1. Click an error in the work area to select it then right-click and click the **Properties** popup menu item; then click the **Error** tab.

2. Select the "Ignore error during verification operations" check box to ignore the error.

3. Click **OK**.

Clearance Checks Using Hatch Outlines

To completely check clearances, any hatch outlines (plane pour and non-plane pour) created by pouring operations in SailWind Layout must also be visible in SailWind Router.

If hatch outlines are invisible in a design, they are ignored during clearance checking. If pour outlines are visible, they are not verified during clearance checking.

Visible plane pour outlines are not obstacles to interactive routing and route editing, but are obstacles during automatic routing.

You can toggle the visibility of hatch and pour outlines:

- If the pour outline is visible, typing "po" displays the hatch outline and hides the pour outline.

- If the hatch outline is visible, typing "po" displays the pour outline and hides the hatch outline.

To see changes in your design when you toggle hatch or pour outline visibility, ensure you assign a visible color to layers of copper items.

Related Topics

Options Dialog Box, Design Verification Category

Setting Colors for the Design

Creating a Design Verification Scheme

Use the **Design Verification** category to create a customized design verification scheme to reuse. A scheme is an easy way to save all your current settings on this tab. You can quickly switch among the schemes to run other checks without having to specify options individually.

Procedure

1. Choose the **Tools > Options** menu item; then, in the Options dialog box, click the Design Verification on page 425 category.

2. In the "Check design for" section, select the rules you want to add to the design verification scheme. A design verification scheme can consist of several design rules to check.

3. After making your design verification selections, in the “Design verification scheme” area, click **Save As**.

4. In the Save Scheme box, type the name you want to assign to the new design verification scheme, and then click **OK**.

Related Topics

Options Dialog Box, Fabrication Category

Chapter 22
Reports

SailWind Router allows you to generate a series of reports to help you analyze various characteristics of your design. Use these reports to review design statistics, perform a pre-routing analysis, and examine the testability of your final assembly.

Reporting Design Information

SailWind Router provides you with several types of reports for a given design.

SailWind Router reports are available from one of the following primary categories:

- **Design**— Provides information about autorouter-related design elements, such as the number of nets, connections, traces, vias, unroutes, and the percent of routed connections in the design
- **Prerouting Analysis**— Provides information about design properties that may impede autorouting, such as grid settings, nets that have a disabled routing status, or pin pairs that have a shove-protected status
- **Testability**— Provides information about which nets are inaccessible and why they are inaccessible.
- **Test Points**— Provides information about all test points in the design, such as the total number in the design, their location, probe nail diameter, the object and netname to which the test point is assigned.

You generate most reports from the Report commands on the **File** menu.

The Design Report

The Design report displays information about autorouter-related design elements, such as the number of nets, connections, traces, vias, unroutes, and the percentage of routed connections in the design.

SailWind Router stores this report, *DesignReport.txt*, in *\SailWind Projects*. To view the report, click the link to the report file, *\SailWind Projects\DesignReport.txt*, in the **Status** tab of the Output window.

The format of the design report follows. For more information, refer to "Generating a Design Report."

Design Report Format

The Design Report allows you to examine the statistics of your design in detail.

The design report is formatted as follows:

```
HEADER
SailWind Router Version         <version>
Report Name         (Design Statistics)
Design:         Filename of the open design
_________________________________________________________
Count
Nets:     Number of nets in the design
Connections:     Number of pin pair connections in the design
Traces:     Number of traces in the design
Vias:     Number of vias in the design
Unroutes:     Number of unroute instances in the design
Completion:     Percent of completed, routed connections in the design
_________________________________________________________
Trace Length     (design units)
Design Total:     Total length of all traces in the design
Layer:     Total length of traces on each layer of the design
Net:     Total trace length of each net in the design
```

Sample Design Report

A Design Report contains information to assist you in analyzing your design.

```
SailWind Router Version v2004 SPac1, Design Statistics
Design: \SailWind Projects\Samples\pwrdemoe.pcb
_________________________________________________________
Count
Nets:              40
Connections:       119
Traces:            132
Vias:              29
Unroutes:          17
Completion:        85.7
_________________________________________________________
Trace length (Mils)
Design
Total:         49599
Layer
TOP            22872
BOTTOM         26727
Net
```

```
$$$1906         1816
$$$1928         1795
$$$1951         1880
X               1880
```

Generating a Design Report

The Design report provides information about autorouter-related design elements, such as the number of nets, pin pair connections, traces, vias, unroute instances, and the percent of routed connections in the design.

Procedure

Choose the **File > Reports > Design** menu item.

Results

SailWind Router creates the *DesignReport.txt* file in the *\SailWind Projects* folder, and a link to the report appears in the **Status** tab of the Output window. Click the link to view the report.

Related Topics

Reporting Design Information

Generating a Prerouting Analysis Report

The Prerouting Analysis report provides information about design properties that may impede autorouting, such as grid settings, nets that have a disabled routing status, or pin pairs that have a shove-protected status.

Procedure

Click the **Tools > Pre-routing Analysis** menu item.

Results

Any errors or warnings are written to the **Status** tab of the Output window.

Related Topics

Reporting Design Information

Testability Report

The Testability Report allows you to assess the overall testability of your design and helps to locate any nets that may not have accessible test points.

The report provides test points results in two ways:

- By a number at the end of a inaccessible net, indicating the reason a test point is not added (shown in the **Status** tab of the Output window after each routing pass). To report with this number, select Generate Test Points on the **Test Points** tab of the Options dialog box.

- By a testability report. This report, *TestabilityReport.txt*, displays information about inaccessible nets. The reasons also appear so you can correct the problems. This increases the probability that your design is 100% accessible. This report is stored in *\SailWind Projects*. To view the report, click the link to the report file, *\SailWind Projects\TestabilityReport.txt*, in the **Status** tab of the Output window.

For more information, refer to "Creating a Testability Report."

Testability Report Format

The Testability Report uses a pre-defined format that allows you to view and analyze the testpoint data related to your design.

```
HEADER
System Name         (DFT Audit)
Report Name         (Board Report)
Board Name:         Name of the open design
Generation Date:         Date and time of the report generation

ELEMENTS:         Total number of elements used in the PCB.
         Elements can be components and/or vias.
Components:         Number of components used in the PCB
Vias:         Number of vias used in the PCB

SIDE:         PCB side selected for auditing:
         the bottom or both the bottom and top

PINS:         Total number of component pins

CONNECTED PINS:         Number of component pins connected to nets
Accessible Pins:         Number of accessible component pins
Inaccessible Pins:         Number of inaccessible component pins
Probed Pins:         Number of component pins connected to test
         probes(tester pins)
Nonprobed Pins:         Number of component pins not connected to test
         probes(tester pins)

UNUSED PINS:         Number of component pins not connected to nets
Accessible Unused Pins:         Number of accessible unused component pins
Inaccessible Unused Pins:         Number of inaccessible unused component
 pins
Probed Unused Pins:         Number of unused component pins connected
         to test probes (tester pins)

VIAS:         Total number of vias
Accessible Vias:         Number of accessible vias
Inaccessible Vias:         Number of inaccessible vias
Probed Vias:         Number of vias connected to test probes
         (tester pins)
```

```
Nonprobed Vias:        Number of vias not connected to test probes
         (tester pins)

NETS:         Total number of nets on the PCB, including
         nonconnected nets
Nets with Test Points:         Total number of nets in the PCB that are
         connected to test probes. This number is
         independent of the selected PCB sides.
Nets Without Test Points:         Total number of nets in the PCB not
 connected
         to test probes. This number is independent of
         the selected PCB sides. Before test probe
         (tester pin) placement, this number is equal to
         the number of nets on the PCB.
Accessible Nets:         Number of nets that are accessible to test
         probes (nail pins) from the indicated side of
         the board
Partially Tested Nets:         Number of nets that are not totally routed
 and
         the minimum test points specified for the net
         was not met
Inaccessible Nets:         Number of nets that are not accessible to test
         probes (nail pins) from the indicated side of
         the board
```

Table 86 lists possible reasons for inaccessibility.

Table 86. Inaccessibility Reasons

Reason	Explanation
No.:	Table entry number
Net Name:	inaccessible netname
Pins:	Component pins and vias on that net. Component pin names are specified as: Component Name. Component Pin Number. Vias are specified as Via Name and location.
Reason:	Reason for inaccessibility of each component pin on the inaccessible net. The reasons for inaccessibility follow.
Accessible from opposite side	The component pin is accessible from the other side of the board. Probe from the opposite side of the board in the **Test Points** tab of the Options dialog box.
SMD pin	The component pin is an SMD pin; therefore, it cannot be tested in the test point pass.
Covered pin	The component pin is covered (hidden) by another component; therefore, a test probe cannot reach it.

Table 86. Inaccessibility Reasons (continued)

Reason	Explanation
Pad too small	The pad size for the component pin or via is smaller than the specified minimum pad or via size; therefore, the available test probes cannot test this component pin. Change the minimum pad probing size in the **Test Points** tab of the Options dialog box.
Probing not allowed	Vias cannot be probed. Select Probe Vias in the **Test Points** tab of the Options dialog box.
Buried via	Buried vias that are not accessible from the bottom side (or top side if probed) are not accessible.
Outside board outline	Vias and component pins outside the board outline are not accessible.
Probe to Probe minimum distance	Component pins or vias under the specified minimum distance requirements for Probe-to-Probe clearance cannot be probed, even though other accessibility requirements are met.
Probe to Component minimum distance	Component pins or vias under the specified minimum distance requirements for Probe-to-Component clearance cannot be probed, even though other accessibility requirements are met.
Probe to Board minimum distance	Component pins or vias under the specified minimum distance requirements for Probe-to-Board clearance cannot be probed, even though other accessibility requirements are met.
Component Test Point Keepout Violation	Component pins or vias cannot be probed in the keepout area associated with component.
Board Test Point Keepout Violation	Component pins or vias cannot be probed in the keepout area defined at board level.
Drill Size is bigger than Pad Size	Component pins or vias have drill sizes larger than pad sizes. Pads cannot be probed.
Zero Nail Pin Count Specified for Net	DFT Audit Net Preferences (in SailWind Layout) specify that net has zero test points. You can also set this preference on the **Test Point** tab of the Net Properties dialog box.
Tooling Hole	Component pins are tooling holes as specified in DFT Audit preferences in SailWind Layout.

```
TEST POINTS:         Total number of test probes used to test both
         sides of the PCB. This number is independent of
         the selected PCB sides and provides the number
         of probes on the top side and the number on the
         bottom side.

NAIL SIZES TABLE:          Number of nail pins used in testing the
          selected sides of the PCB. In addition, the
```

```
           table shows the total number of nail pins
           used of each size and the total number of
           nail pins used in testing the selected sides
           of the PCB.
```

Sample Testability Report

Use the information in the Testability Report to review the testability of your design and identify any areas that might need attention.

```
------------------------------------------
SailWind Router Version v2004 SPac1 DFT Audit
Board Report
Board Name: \SailWind Projects\Samples\pwrdemoe.pcb
Generation date : Tue Jun 01 13:44:32 2004
------------------------------------------

ELEMENTS: 79
--------
Components: 23
Vias: 56

SIDE: BOTTOM
============

PINS: 212
----

CONNECTED PINS: 159
--------------
Accessible Pins: 154
Inaccessible Pins: 5
Probed Pins: 40
Non-probed Pins: 119

UNUSED PINS: 53
-----------
Accessible Unused Pins: N/A
Inaccessible Unused Pins: N/A
Probed Unused Pins: 0
Non-probed Pins: 53

VIAS: 56
----
Accessible Vias: 56
Inaccessible Vias: 0
Probed Vias: 0
Non-probed Vias: 56


NETS: 40
----
Nets With Test Points: 40
Nets Without Test Points: 0
```

```
Accessible Nets: 40
Partially Tested Nets: 0
===========================

TEST POINTS: 40
-----------
Top Side: 0
Bottom Side: 40

NAIL SIZES:
|------|------|------|------|
| Name | 100  | 75   | 50   |
|------|------|------|------|
| Top  | 0    | 0    | 0    |
|------|------|------|------|
|Bottom| 40   | 0    | 0    |
|------|------|------|------|
| Total| 40   | 0    | 0    |
|------|------|------|------|
```

Creating a Testability Report

The Testability report provides information about the testability of a board, including which nets are accessible and which are inaccessible.

Running the Testability report creates the file *TestabilityReport.txt* in the *\SailWind projects* folder. A link to the report appears in the **Status** tab of the Output window.

Procedure

Click the **File > Reports > Testability** menu item.

Results

The file TestabilityReport.txt is created in the \SailWind projects folder, and a link to the report appears in the **Status** tab of the Output window. Click the link to view the report.

Related Topics

Reporting Design Information

Testability Report

Test Points Report

The Test Points report displays information about all test points in the design, such as the total number, location, probe nail diameter, the object, and the netname to which each test point is assigned. SailWind Router stores the report, *TestPointReport.txt*, in *\SailWind Projects*. To view the report, click the link to the report file, *\SailWind Projects\TestPointReport.txt*, in the **Status** tab of the Output window.

The format of the test points report is described below. For more information, refer to "Creating a Test Points Report."

Test Points Report Format

The Test Points Report provides information describing the number and locations of test points in your design.

```
HEADER
Product     (SailWind Router)
Report Name     (Test Point List)
Design Name:     Name of the open design
______________________________________________________________
Test Points:     Total number of test points in the design
______________________________________________________________
X:     The X location of each test point
Y:     The Y location of each test point
Side:     The side of the PCB to which the test point is assigned
Probe:     The probe nail diameter
Object:     The component pin or via type to which the test point
     is assigned
Net Name:     The netname to which the test point is attached
```

Test Points Report Sample

Use the Test Points Report to view and analyze the information describing testpoint quantities and locations in your design.

```
SailWind Router Version v2004 SPac1 Test Point List
Design: \SailWind Projects\Samples\pwrdemoe.pcb
______________________________________________________________
Test Points: 40
______________________________________________________________
X          Y          Side       Probe      Object     Net Name
______________________________________________________________
950        900        BOTTOM     100        R3.8       $$$261
650        1100       BOTTOM     100        U5.32      $$$464
1150       1700       BOTTOM     100        U5.66      $$$1002
1150       1000       BOTTOM     100        U5.59      $$$1019
1050       1300       BOTTOM     100        U5.52      $$$1497
250        1800       BOTTOM     100        U5.9       $$$1562
0          0          BOTTOM     100        U2.1       $$$1879
200        725        BOTTOM     100        U1.12      SIG002
1225       725        BOTTOM     100        U4.10      SIG004
250        1600       BOTTOM     100        U5.7       SIGA
1025       425        BOTTOM     100        U4.3       X
1125       0          BOTTOM     100        U3.4       XX
```

Creating a Test Points Report

The Test Points report provides information about all test points in the design, such as the total number in the design, their location, probe nail diameter, and the object and net name to which the test point is assigned.

Procedure

Click the **File > Reports >Test Points** menu item.

Results

SailWind Router createst the file *TestPointReport.txt* in the *\SailWind Projects* folder, and a link to the report appears in the **Status** tab of the Output window. Click the link to view the report.

Related Topics

Reporting Design Information

The Routing Report

The Routing report displays information about the autorouting progress. SailWind Router stores the report, *RoutingReport.txt*, in *\SailWind Projects*. To view the report, click the link to the report file, *\SailWind Projects\RoutingReport.txt*, in the **Status** tab of the Output window.

The format of the routing report follows. For more information, refer to "Creating a Routing Report" on page 380.

Routing Report Format

The Routing Report uses a pre-defined format to organize data related to the routing of your design.

```
HEADER
SailWind Router Version <version>, Routing Report
Design: D:\design\demob1.pcb

SUMMARY
Number of passes:        Number of passes the autorouter performed
Routed:        Number of connections that were completely routed
Vias Total:        Total number of vias in the design, followed by
(added)        the number of vias added (+) or removed (-) in
        parentheses
Trace Length Total:         Total trace Length in the design, followed by
(increased)        the amount increased (+) or decreased (-) in
        parentheses
Test Points Total     :   Total number of test points, followed by how
(added)        many were added in parentheses
Total Time:        Amount of time the router spent routing

PASS DETAILS
Pass:        Pass number followed by the name of the pass,
        in parentheses, as it appears on the Strategy
        tab
Pins Processed:        For the fanout pass only. The number of pins
        the program attempted to fan out in the pass.
        The number includes successful and failed
        fanouts.
Fanouts Created:        The number of fanouts created in the pass.
Links Selected:        Number of unroutes to route in the pass. The
```

```
        number is determined by the routing order or
        by the selection.
Links Processed:        Number of unroutes the program tried to route
        in the pass. Includes complete and incomplete
        routes.
        More unroutes are processed than selected
        because pins not connected by an unroute are
        considered when routing. For example, a net
        consists of three pins (A,B, and C) and two
        unroutes (A to B and A to C). The router
        considers an unroute between B and C. Therefore,
        the routing report lists two unroutes selected
        and three unroutes processed.
Routed:        Number of unroutes routed in the pass.
Vias:        Number of vias in the design, followed by the
        number of vias added (+) or subtracted (-)
        in the pass in parenthses
Trace Length:        Total trace length, in mils. in the design,
        followed by the total length added (+) or
        subtracted (-) in the pass in parentheses
Test Points:        Total number of test points in the design,
        followed by, in parentheses, by the number of
        test points added (+) or subtracted (-) in the
        pass
Accessible Nets:        Total number of accessible nets in the design,
        followed, in parentheses, by the number of nets
        made accessible (+) or made inaccessible (-) in
        the pass
Time:        Amount of time the router spent on this pass,
        followed by the total time spent routing in
        parentheses
```

Tip
All items do not appear for every pass.

Sample Routing Report

Use the information in the Routing Report to analyze and assess the routability of your design.

```
SailWind Router Version v2004 SPac1, Routing report
Design: \SailWind Projects\Samples\pwrdemoe.pcb
=============================================================
Tue Nov 09 12:22:19 2002

Autorouting \SailWind Projects\Samples\pwrdemoe.pcb
=============================================================
Number of Passes:           0 of 2
Routed:                     102
Vias:                       29(+29)
Trace Length:               49599(+49599) Mils
Test Points:                40(+0)
```

```
Accessible Nets:           40(+0)
Time                       00:00:18
==============================================================
Pass:                      4 (Route)
Links Selected:            119
      Processed:           130
Routed:                    102
Vias:                      29(+29)
Trace Length:              49599(+49599) Mils
Time                       00:00:18(+00:00:18)
==============================================================
Tue Nov 09 12:27:32 2002

Autorouting \SailWind Projects\Samples\pwrdemoe.pcb
==============================================================
Number of Passes:          2 of 2
Routed:                    11
Vias:                      55(+26)
Trace Length:              78524(+28925) Mils
Test Points:               40(+0)
Accessible Nets:           40(+0)
Time                       00:00:19
==============================================================
Pass:                      4 (Route)
Links Selected:            17
      Processed:           27
Routed:                    11
Vias:                      55(+26)
Trace length:              78525(+28926) Mils
Time                       00:00:15(+00:00:15)
==============================================================
Pass:                      5 (Optimize)
Vias:                      55(+0)
Trace Length:              78524(-0) Mils
Rerouted:                  0
Time                       00:00:04(+00:00:04)
==============================================================
```

Creating a Routing Report

This report provides information about the autorouter progress during autorouting, such as the number of passes, connections completely routed, vias, and test points in the design.

Running the report creates the file *RoutingReport.txt* in the *\SailWind projects* folder. A link to the report appears in the **Status** tab of the Output window.

Procedure

Click the **Routing toolbar** button, then click the **Start Autorouting** button.

Results

SailWind Router creates the *RoutingReport.txt* file in the *\SailWind Projects* folder, and a link to the report appears in the **Status** tab of the Output window. Click the link to view the report.

Related Topics

Reporting Design Information

Testing Database Integrity

When a file loads, SailWind Router performs an integrity test on the database. SailWind Router bases its integrity test on the integrity test performed in SailWind Layout. Any integrity test error messages appear in the **Status** tab of the Output window. Each error links directly to the location of the error in the design. Click on the error link to zoom in on the error in the design. The **Status** tab displays up to 20 errors. All errors save to a report named *TestIntegrity.txt* located in *\SailWind Projects*. To view the report, click the link to the report file in the **Status** tab of the Output window.

Checking Database Integrity

You can run the same database integrity check at any time. The integrity check verifies the values within the database to be within an acceptable range. If the integrity check encounters any problems, the **Status** tab of the Output window displays the errors.

For more information, refer to "Creating an Integrity Test Report."

Integrity Test Report Format

The Integrity Test Report uses a pre-defined format to organize the information related to the database integrity of your design.

```
HEADING
Product     (SailWind Router)
Report Name     (Database Integrity Test Results)
Design Name:     Name of the open design
Errors:     Description of errors
```

Integrity Test Report Sample

Use the information in the Integrity Test Report to analyze and assess the database integrity of your design.

```
SailWind Router. Database Integrity Test Results for <\SailWind
 Projects\Samples\bench98ulrouted.pcb>

Trace of net AVDD placed on non routable layer
Trace of net AVDD placed on non routable layer
Trace of net AVDD placed on non routable layer
Trace of net AVDD placed on non routable layer
Trace of net AVDD placed on non routable layer
Trace of net AVDD placed on non routable layer
Trace of net AVDD placed on non routable layer
Trace of net AVDD placed on non routable layer
Trace of net AVDD placed on non routable layer
```

```
Trace of net AVDD placed on non routable layer
Trace of net AVDD placed on non routable layer
Trace of net TLCLK placed on non routable layer
Trace of net TSPCLK placed on non routable layer
Total problems found: 13
```

Creating an Integrity Test Report

The Integrity Test report verifies that the values within the design database are within an acceptable range.

Procedure

Click the **Tools > Integrity Test** menu item.

Results

SailWind Router creates the *TestIntegrity.txt* file in the *\SailWind Projects* folder, and a link to the report appears in the **Status** tab of the Output window. Click the link to view the report.

Chapter 23
Licensing

Your license for the software may be limited and may not enable all software features. Read the sections that follow to learn more about SailWind Router licensing and associated features.

You may work in a multi-user setting where multiple licenses are available of which some enable general features and other licenses enable advanced features. You can use the Installed Options dialog box on page to check which software features you are licensed to use, or to control which license or licenses you consume of the many licenses that are available when using the software. For example, if you have many suite licenses but limited atomic licenses of an advanced feature-set of the software, and you are not using those features, you can free up that license and allow it to be consumed by someone else who requires those software features.

Viewing a Node Locked License File

If you are using node-locked licensing, you can view the contents of a license file.

Procedure

1. Choose the **Help > Installed Options** menu item; then, in the Installed Options dialog box, click the **License File** tab.
2. On theLicense File tab on page 418, select the license file that you want to view from your list of license files.
3. Click **View** to display the contents of the selected file in the bottom portion of the screen.

Viewing the Floating License Status

If you use a floating license, you cannot view the license file that is in use, but you can view the status of the features associated with the server license.

Procedure

1. Choose the **Help > Installed Options** menu item; then, in the Installed Options dialog box, click the **License File** tab.
2. On the License File tab on page 418, select the server license file for which you want to display feature status.
3. Click **Status**. The feature usage information appears in the bottom portion of the screen.

Checking Out Suite Licenses

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

Restrictions and Limitations

To use this functionality, it is imperative that you remove any FlexNet Options files and/or batch script customization.

Prerequisites

This functionality is applicable only if you have floating licenses, and on your license server you have one of the following:

- A mix of different PADS Suite licenses. For example, PADS LS/padsls_c and PADS ES/padses_c.
- One or more PADS Suite licenses and one or more PADS Kit licenses. For example, PADS ES/padses_c and pwrshell and other similar features, as part of SailWind Layout 065 Kit.

Procedure

1. Choose the **Help > Installed Options** menu item; then, in the Installed Options dialog box, click **Suite Configuration**.
2. The check the status of all listed licenses, click **Check status**.
3. Select the license file to use from the list.
4. To determine which Suite licenses are being used each time you start SailWind Router, select the “Show this dialog box on program startup” and “Automatically check license status on dialog box startup” check boxes.
5. Click **OK**.

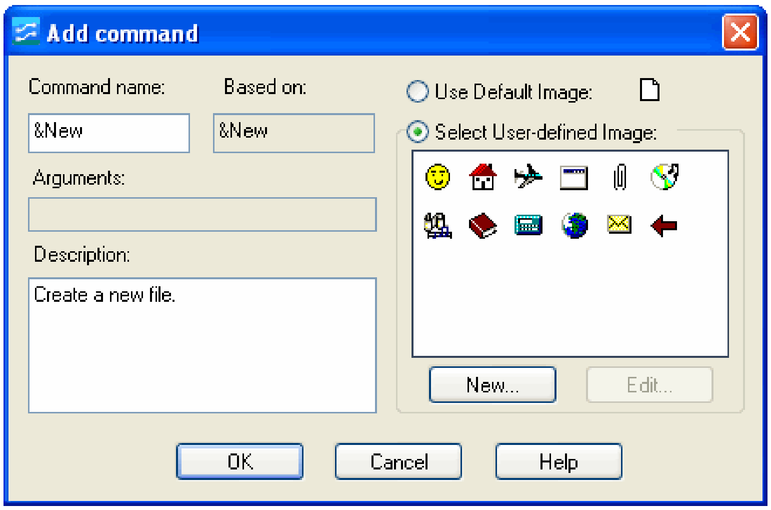
Chapter 24
SailWind Router GUI Reference

The various dialog boxes throughout SailWind Router provide access to tools and settings. Each dialog box provides a Help icon for accessing additional information.

Add/Edit Command Dialog Box

To access: Click **Tools > Customize** (menu item) > **Commands** tab > **New** or **Edit** button

Create commands that you can use as selections on menus or as buttons on toolbars.



Objects

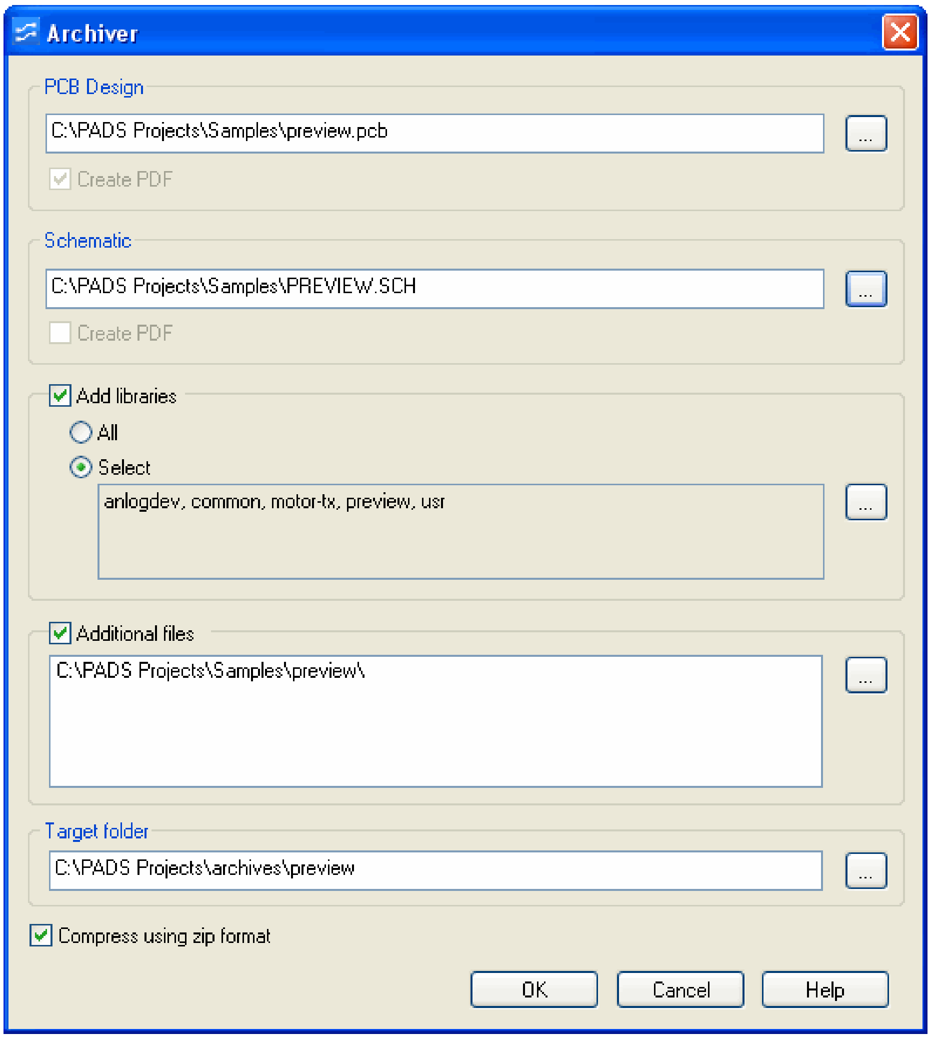
Field	Description
Command name	The name of the new command. **Tip** Type an ampersand before the letter you want to use as the Alt keyboard shortcut.
Based on	The command on which you want to base the new command.
Arguments	Any arguments for the new command. Tip: Use a space to separate arguments. If 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.
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.

Archiver Dialog Box

To access:

- Click the **File > Archive** menu item

Create archives of your schematics, designs, files and folders, and libraries.



Objects

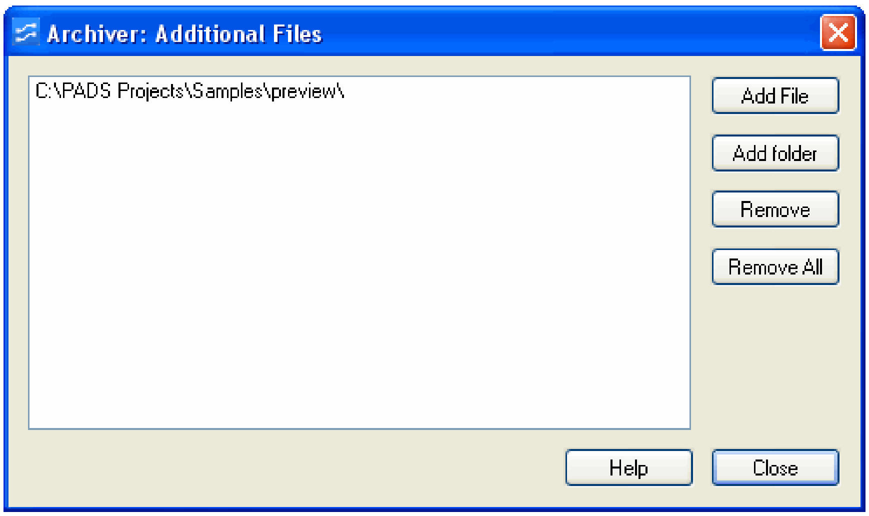
Field	Description
PCB Design	Specifies the location and name of the PCB design you want to archive. SailWind Router automatically populates this box with the information from the current design. To change the design, or if no design is open, type the location or click the browse button next to the box.

Field	Description
	Note: The Create PDF check box is unavailable in SailWind Router.
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 next to the box.
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 next to the box 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 next to the box. **Restriction:** If you clear the “Compress using zip format” check box, the target folder must be empty.
Compress using zip format	Creates a zip file when you select this check box. The resulting filename conforms to 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.

Archiver Additional Files Dialog Box

To access: Click the **File > Archive** menu item > select the Additional Files check box > click the corresponding browse button

Add files and folders to the design you want to archive.



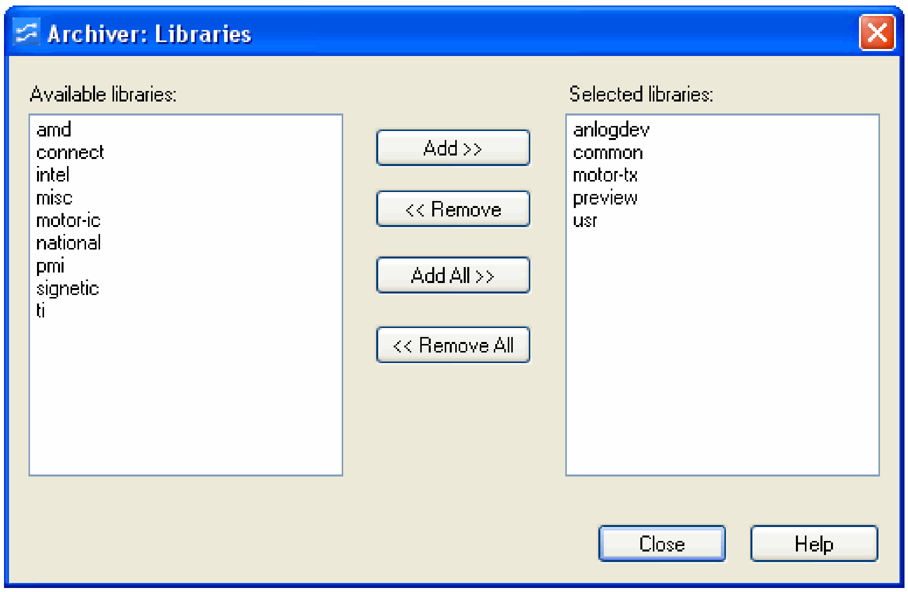
Objects

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.

Archiver Libraries Dialog Box

To access: Click the **File > Archive** menu item > select the Add libraries check box > choose the Select option > click the corresponding browse button

Add libraries to the design you want to archive.



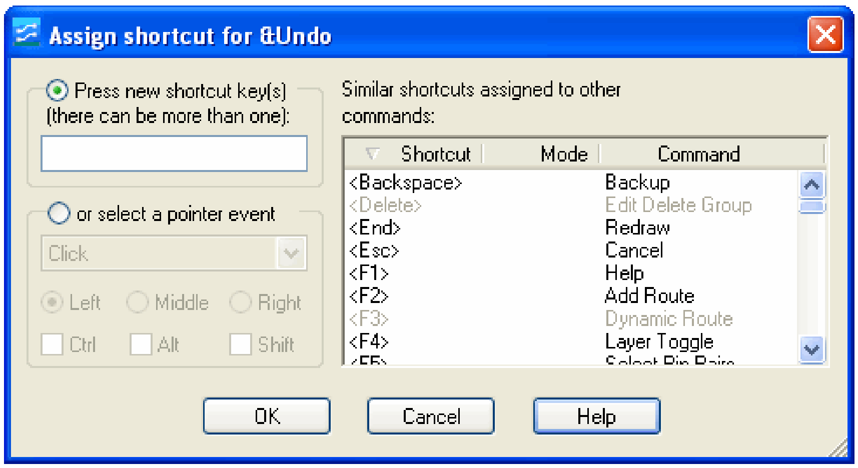
Objects

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.

Assign Shortcut Dialog Box

To access: Click the **Tools > Customize** menu item > **Keyboard and Mouse** tab > New button (in the Current Shortcuts area)

Create a new shortcut key.



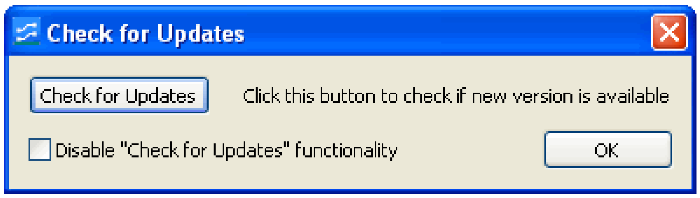
Objects

Field	Description
Press new shortcut key	Type the shortcut keys and the combination you want to use. For example, if you want to run a macro by pressing the Shift key and F9 key simultaneously, type “Shift+F9” in the box.
Select a pointer event	If you want to use a combination of mouse movements/button clicks instead of shortcut keys to initiate a command, specify the pointer/mouse combination here.
Similar shortcuts list	Lists the shortcut keys already assigned to other commands.

Check for Updates Dialog Box

To access: Click the **Help > Check for Updates** menu item

Manually check for a new version of SailWind, and disable or enable automatic checks.



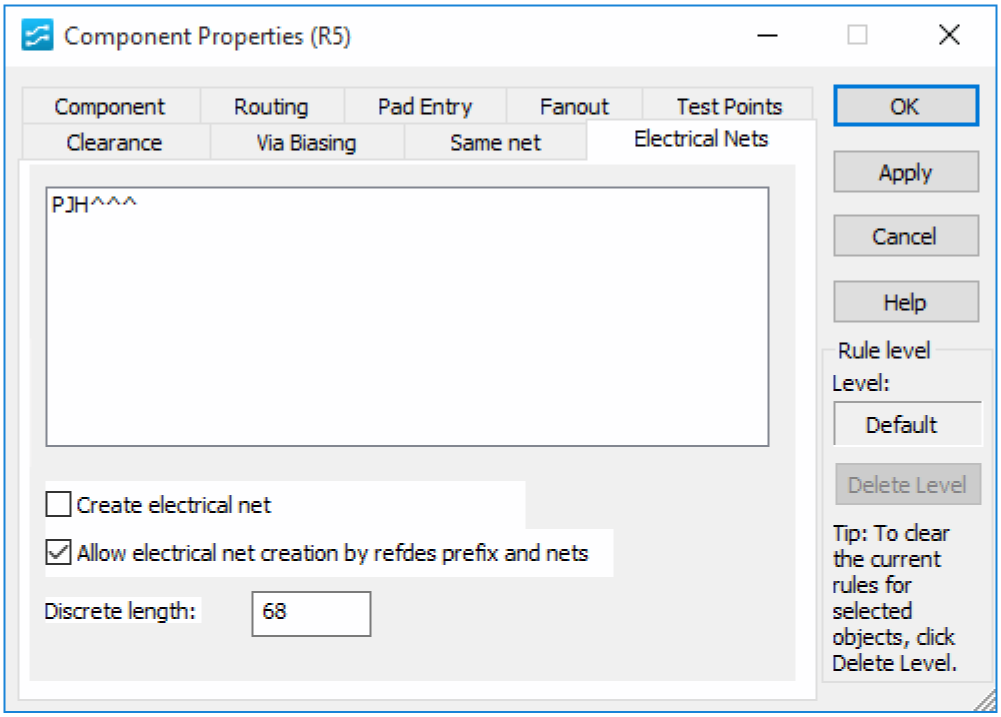
Objects

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. Select the check box to stop SailWind Router from automatically checking for a new version of the SailWind products; clear the check box if you want SailWind Router to automatically check for a new version.

Component Properties Dialog Box, Electrical Nets Tab

To access: Select a component > Right-click > **Properties** popup menu item > **Electrical Nets** tab

View and control the electrical net status of components.



Objects

Field	Description
List box	Name(s) of the electrical nets attached to or going through the component. An electrical net name has a suffix of 3 caret symbols (^^^).
Create electrical net	Select the check box to create an electrical net of the component's nets. **Tip** You can also affect this check box by the Create Electrical Net popup command for selected components. **Restrictions:**

<table>
<tr><th>Field</th><th>Description</th></tr>
<tr><td></td><td>

If a net of a selected component is excluded from electrical net creation, you cannot include it 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.)
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.

If you select the “Create electrical net” check box in the Net Properties dialog box for the component’s nets, you must also clear the “Allow electrical net creation by refdes prefix and nets” check box.

Clear the check box to remove the component from electrical net creation, that is, to prevent an electrical net from going through it.

Tip
You can also clear this check box by the Disable Electrical Net Creation popup menu item for selected components.

Clear both this check box and the “Allow electrical net creation by refdes prefix and nets” check box to prevent the selected components from creating electrical nets (that is, to prevent any electrical net from going through them).
</td></tr>
<tr><td>Allow electrical net creation by refdes prefix and nets</td><td>
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 setting 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 nets or by refdes prefix. The nets can still be made into electrical nets by the other components attached to them.

Tip
This check box is also cleared by the Disable Electrical Net Creation popup command for selected components.

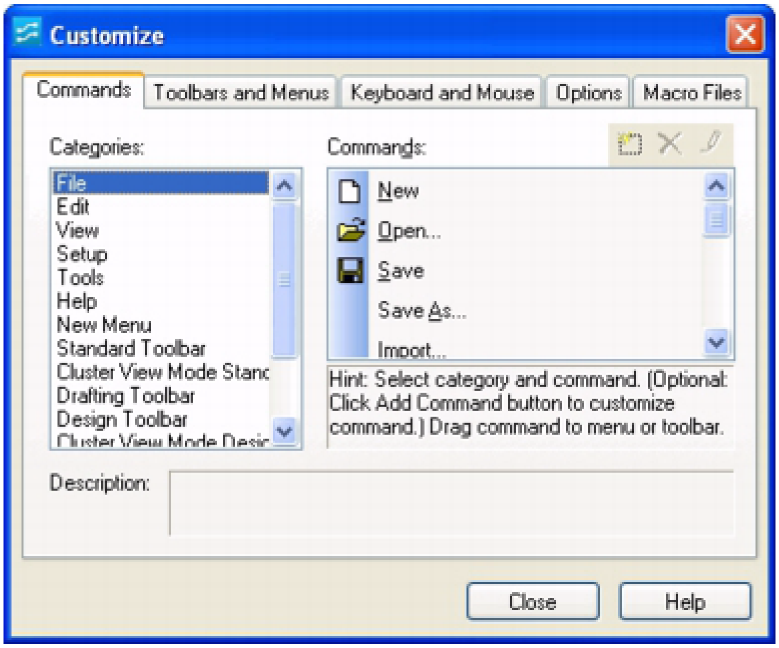
Restriction:
If you clear this check box, you must also clear the “Create electrical net” check box to prevent the selected components from creating electrical nets (that is, to prevent any electrical net from going through them).
</td></tr>
<tr><td>Discrete length</td><td>
Enter the length to be used for this component when calculating net length.

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, this value 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.
</td></tr>
</table>

Customize Dialog Box, Commands Tab

To access: Choose the **Tools > Customize** menu item > **Commands** tab

Add commands to menus or toolbars, or create custom menus.



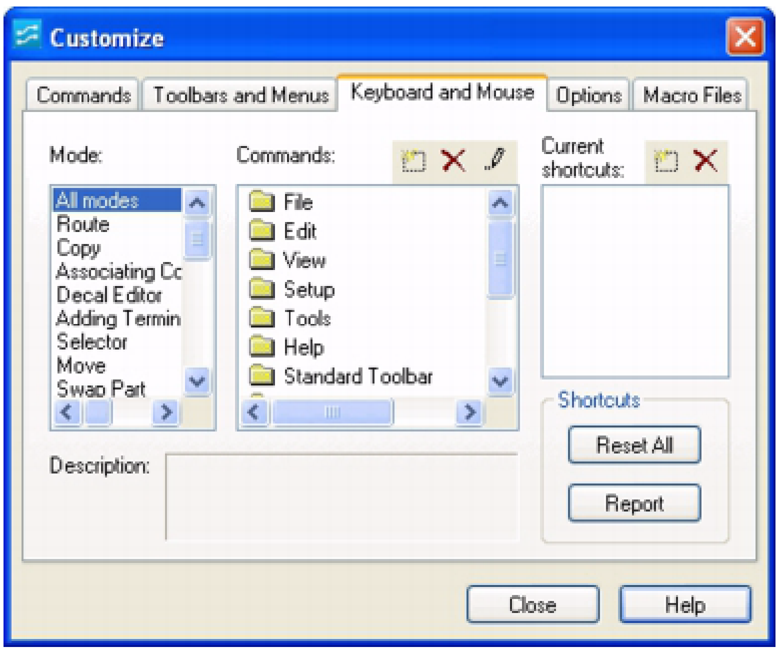
Objects

Field	Description
Categories list	Limits the display of available commands in the Commands list, according to category. For example, clicking the “Routing Toolbar” category displays only those commands found in the Routing toolbar (such as Start Autorouting, Stop Autorouting, and so on).
Commands list	Select from a list of commands available to add to a menu or toolbar.
Command toolbar 	Add a new command, delete a command you have added, or rename a command.

Customize Dialog Box, Keyboard and Mouse Tab

To access: Choose the **Tools > Customize** menu item > **Keyboard and Mouse** tab

Create and customize shortcut keys.



Objects

Field	Description
Mode list	Restricts the displayed list of commands.
Commands list	The list of commands available for which to assign a shortcut.
Command toolbar	Add a new command (opens the Add Command Dialog Box on page 388), delete a command you have added, or rename a command (opens the Edit Command dialog box on page 388).
Current shortcuts list	The list of shortcuts assigned to the selected command.
Current shortcuts toolbar	Add a new shortcut (open the Assign Shortcut Dialog Box), or delete a shortcut you have added.
Description	Lists what the selected command does.
Reset	Sets the selected toolbar or shortcut menu to the default settings.

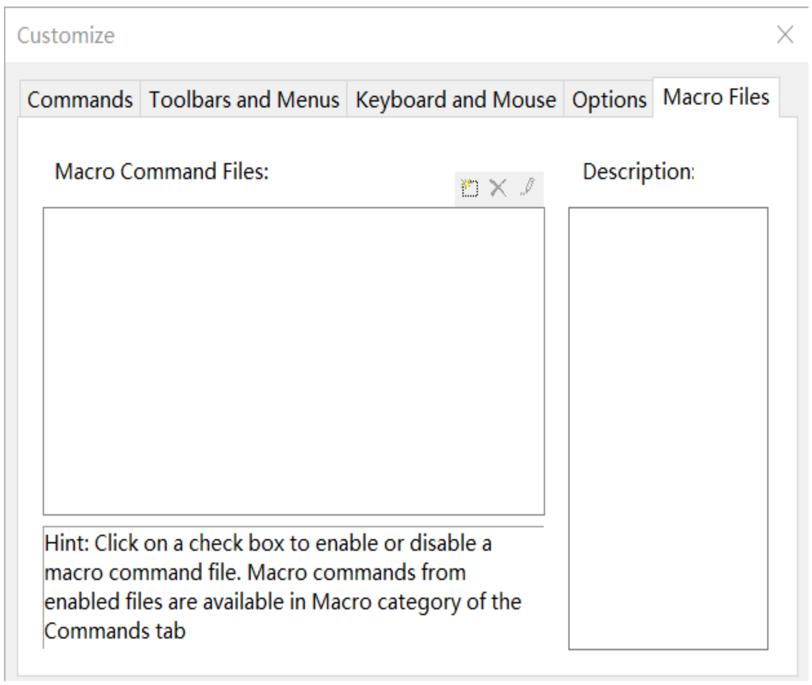
Field	Description
Report	Saves a report of all current shortcut commands.

Customize Dialog Box, Macro Files Tab

To access: Choose the **Tools > Customize** menu item > **Macro Files** tab

Create commands from macro files and add them to toolbars and menus.

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



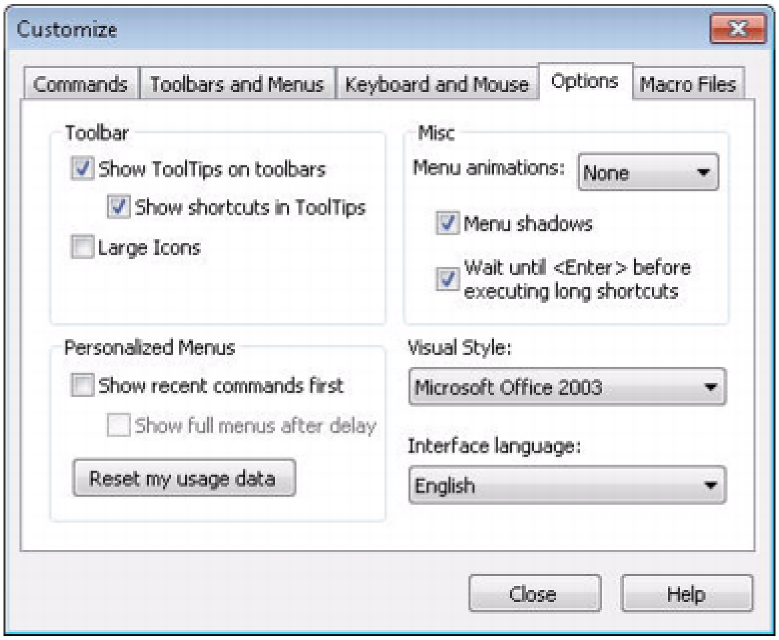
Objects

Field	Description
Macro Command Files list	Displays a 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.

Customize Dialog Box, Options Tab

To access: Choose the **Tools > Customize** menu item > **Options** tab

Customize the SailWind interface by changing the appearance of menus and toolbars.



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 Enter.
Show recent commands first	Displays your recent menu command selections at the top of the list.

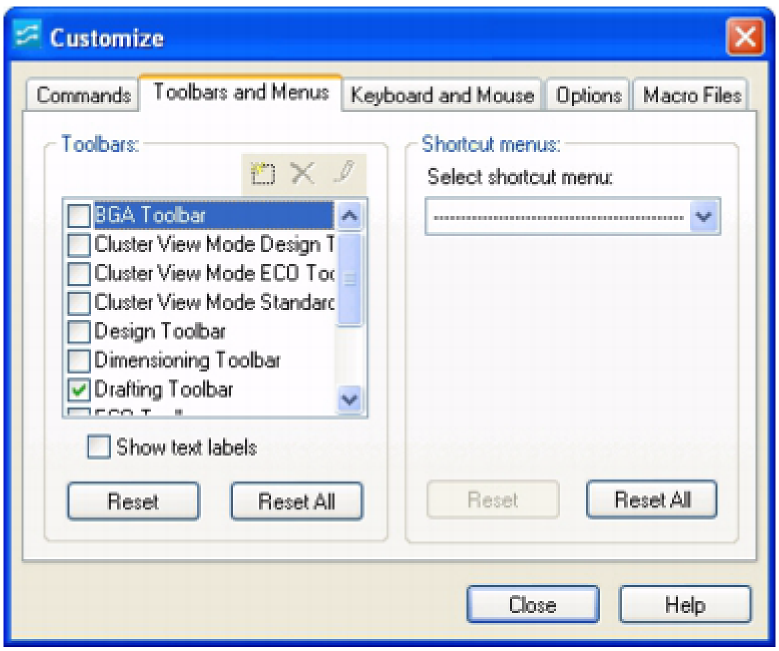
Field	Description
Show full menus after delay	Displays the full menu after a slight pause.
Reset my usage data	Restores the default set of commands to the menus and toolbars. **Tip** This option does not undo any explicit customizations you made.
Visual Style	Sets the look and feel of your toolbars and title bars.
Interface Language	Specifies the language for all dialog boxes and messages displayed: English, Japanese, Brazilian Portuguese, or Chinese Simplified.

Customize Dialog Box, Toolbars and Menus Tab

To access: Choose the **Tool > Customize** menu item > **Toolbars and Menus** tab

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.

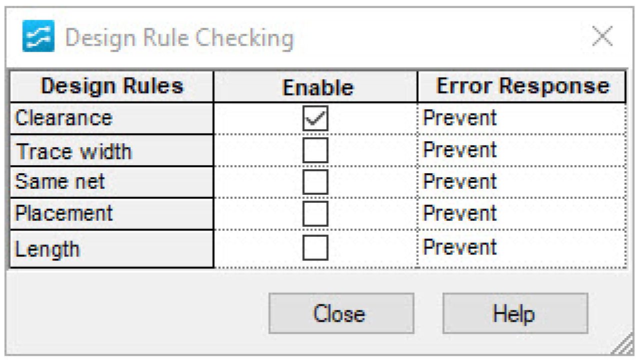
Field	Description
	Restriction: This option applies to SailWind Router only.
Reset	Sets the selected toolbar or shortcut menu to the default settings.
Reset All	Sets all toolbars or shortcut menus back to their default settings.

Design Rule Checking Dialog Box

To access:

- Choose the **Tools > DRC Settings** menu item
- **DRC Filter Toolbar > DRC Settings** button

Select which design rules to enable during DRC checking and choose the desired error response for each rule category.



Objects

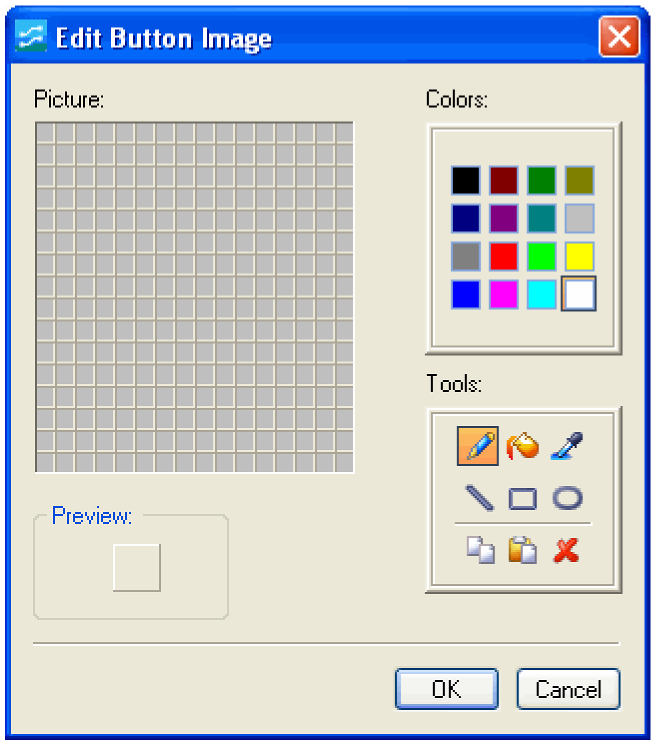
Column	Description
Design Rules	Shows the design rule categories available for design rule checking: • **Clearance** — Checks against all rules set on the “Properties Dialog Box, Clearance Tab” on page 468 • **Trace width** — Checks against Trace width rules set on the “Properties Dialog Box, Routing Tab” on page 517 • **Same net** — Checks against all rules set on the “Properties Dialog Box, Same Net Tab” on page 519 • **Placement** — Checks against the “Minimum spacing between components” rule set in the Component assembly area on the “Options Dialog Box, Fabrication Category” on page 433 • **Length** — Checks against any rules defined on the “Properties Dialog Box, Length Tab” on page 501 for all net objects in the design
Enable	When you enable DRC for a design rule, SailWind Router checks for design rule violations only for that design rule. When you clear the Enable check box for a specified design rule, the software ignores all the violations that occur for that design rule during interactive routing and route editing. To enable DRC for all the design rules, click the Enable title to select the entire column, and then select one of the check boxes in

Column	Description
	this column. If you need to disable the column selection, click a cell in another column. You can also select design rules on the DRC Filter toolbar. Click the button for each design rule you want to check.
Error Response	In the Error Response column, click the cell of the design rule for which you want to set an error response. Select the appropriate response for how you want SailWind Router to handle design rule violations when it encounters them during interactive routing, trace editing, and placement editing operations. Available error responses are as follows: • **Prevent**— Prevents design rule violations. If errors are encountered during routing and placement editing operations, the program returns to the state that existed before encountering the error and displays a brief message in the **Status** tab of the Output window and in the workspace near the pointer location. You can then manually initiate the Explain command to view the error details. • **Explain**— Allows the temporary creation of a violation. When an error is detected, the operation is suspended, error markers are created, and the errors are automatically explained in the Spreadsheet window. The system may let you accept the violation and continue the operation with errors or reject the violation and return to the state that existed before the creation of the error. • **Warn**— Creates error markers, warns, and continues with the next operation. The error is created in the database and marked with an error marker.
Error Response (Continued)	Note that you can set the error response for adjacent rules, non-adjacent rules, or the entire design rule set: • To set the same error response for multiple adjacent design rules, click the first design rule, press Shift, click the last design rule in the range, and select the error response you want from the list. • To set the same error response for multiple nonadjacent design rules, click any design rule, press Ctrl while you click other design rules that you want to select, and select the error response you want from the list. • To set the same error response for the entire design rule set, click the Error Response title to select the entire column, and then select the error response you want from the list.

Edit Button Image Dialog Box

To access: Choose the **Tools > Customize** menu item > **Commands** tab > **New** button > **Select User-Defined Image** option > **New** or **Edit** button

Create or edit button icons.



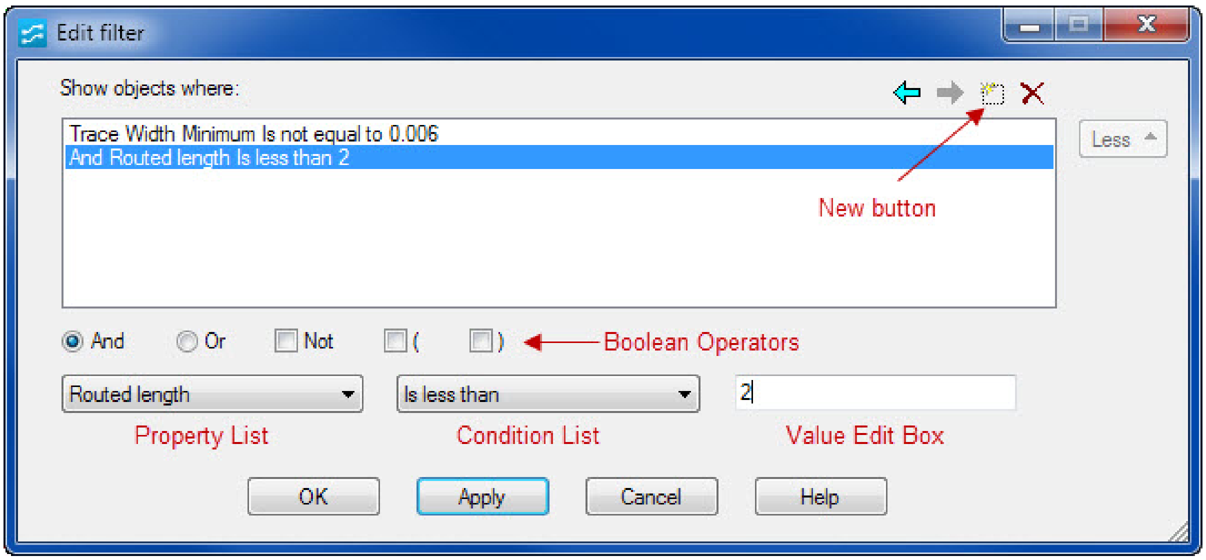
Objects

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 Filter Dialog Box

To access: In the Spreadsheet window, click the **Define Filter Settings** button. 

Use this dialog box to find objects in the currently selected tab of the Spreadsheet window.



Objects

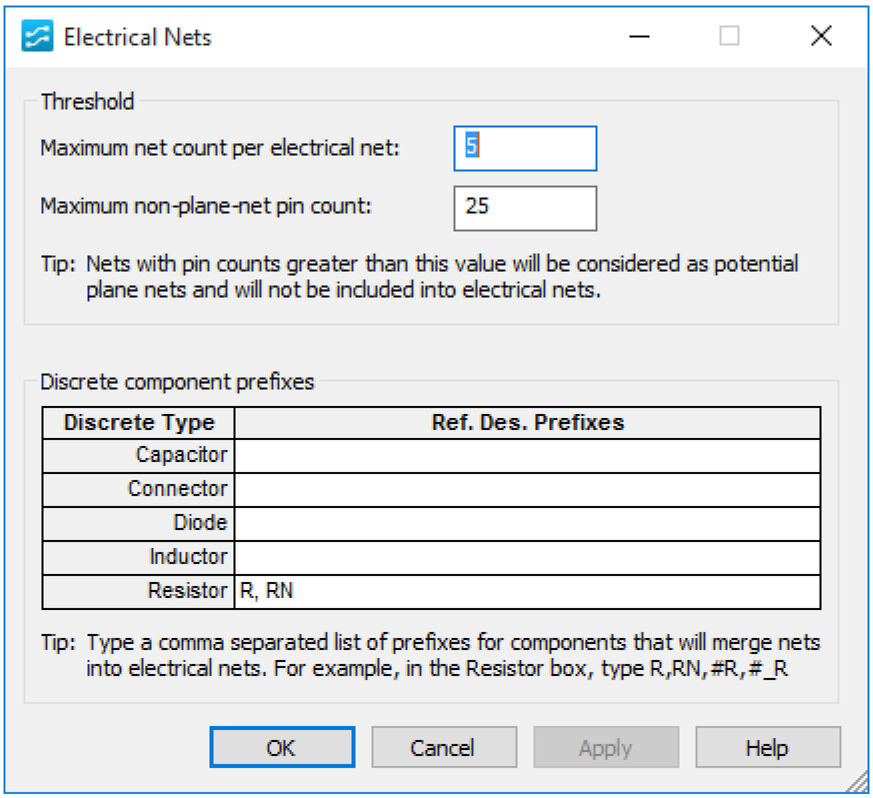
Field	Description
Show objects where:	Displays the current list of conditions that compose the search expression.
New button	Click to add a condition to the search expression
More button	Click to expand the dialog box to create a multiple-condition search.
Less button	Click to contract the dialog box to create a simple single-condition search.
Boolean Selectors	Set these appropriately to describe the boolean structure as you add additional conditions for multiple-condition searches.
Property List	Select the object property you wish to test against the condition.
Condition List Value Edit Box	Specify the condition and value to test the property against.

Electrical Nets Dialog Box

To access: Choose the **Setup > Electrical Nets** menu item.

Create and delete electrical nets by adding or removing the refdes prefixes of the associating components, that is, the components through which the electrical nets now pass (when you are deleting) or will pass (when you are creating).

Note:
Electrical nets are not available in *engineering change order (ECO) operations*.



Objects

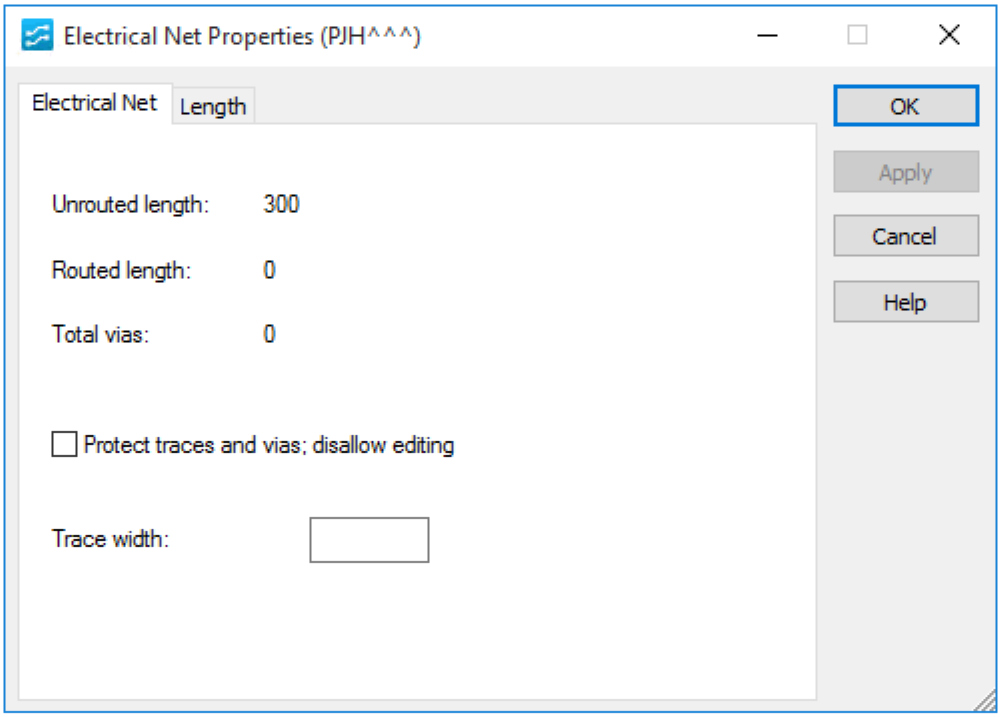
Field	Description
Maximum net count per electrical net	When creating electrical nets, it is possible that you may create electrical nets containing an unreasonable number of nets. To prevent this, specify the maximum number of nets allowed in an electrical net. Electrical nets having more than this limit are not created. **Tip** This limit also applies to the creation of electrical nets manually by component and by net.

Field	Description
Maximum non-plane-net pin count	When creating electrical nets, it is possible that you may include nets that are potential plane nets. (Plane nets are not allowed in electrical nets.) To prevent this, specify the maximum pin count for nets included in electrical nets. **Tips:** • This limit also applies to the creation of electrical nets manually by component and by net. • Virtual pins are not included in a net’s pin count.
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. **Tips:** • The discrete type categories are only for convenience; prefixes for any type of component can be entered in any field. • If a selected component has more than two pins, the following conditions must apply, or the component cannot 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. Delete refdes prefixes of existing associating components to remove them from electrical nets.

Electrical Net Properties Dialog Box, Electrical Net Tab

To access: Select an electrical net > right-click > choose the **Properties** popup menu item > **Electrical Net** tab

View and manage electrical net status.



Objects

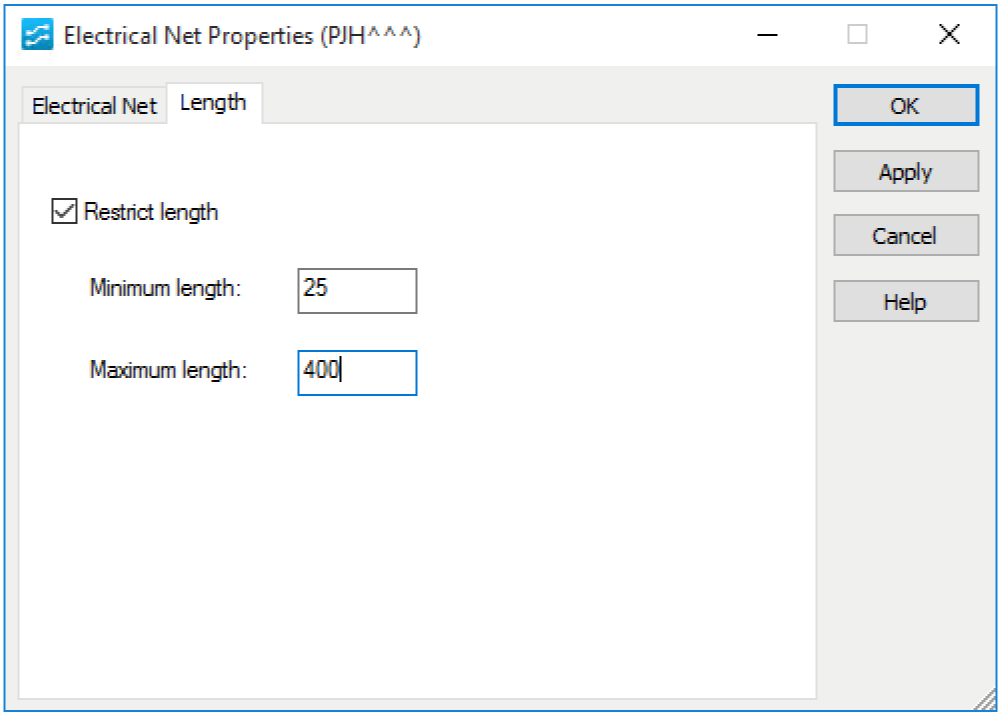
Field	Description
Unrouted length	The length of the unrouted part of the electrical net.
Routed length	The length of the routed part of the electrical net.
Total vias	The number of vias on this electrical net.
Protect traces and vias; disallow editing	Specifies to prohibit unrouting or moving a routed trace. **Tip** You can make protected traces visually distinguishable from unprotected traces. In the Options dialog box, in the Global category > General subcategory, select the “Distinguish protected traces and vias” check box. Protected traces will be displayed as outlines.
Trace width	Specifies the width of all traces on an electrical net.

Field	Description
	Restriction: If the electrical net has protected traces, these trace width properties cannot be modified.

Electrical Net Properties Dialog Box, Length Tab

To access: Select an electrical net > right-click > choose the **Properties** popup menu item > **Length** tab

Restrict electrical net length.



Objects

Field	Description
Restrict length	Specifies that you want to set the minimum and/or maximum length values.
Minimum length	Specifies the minimum value for the electrical net length. **Restriction:** Available only when Restrict length is selected.
Maximum length	Specifies the maximum value for the electrical net length. **Restriction:** Available only when Restrict length is selected.

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 Error 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 Customer Support for troubleshooting.

Objects

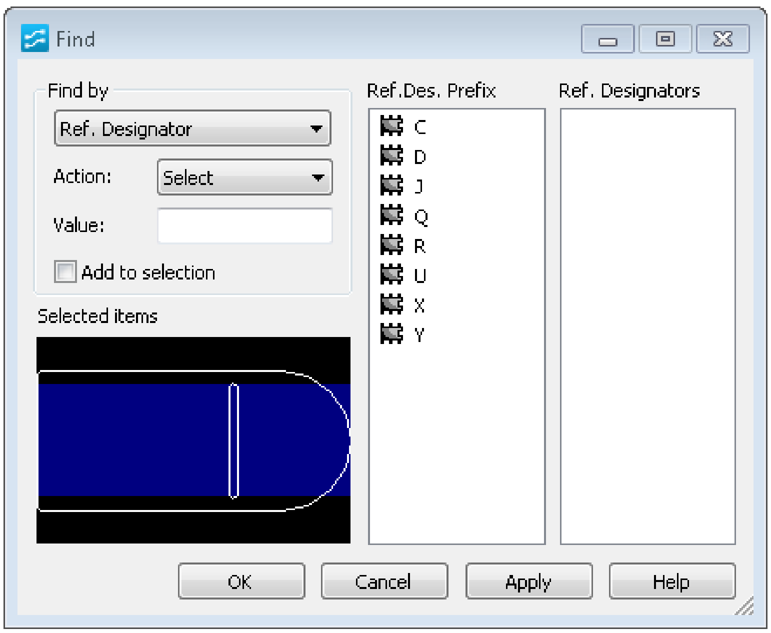
Name	Description
Comments box	You can describe what you were doing when the error occurred or add additional information that might help when investigating the crash.
Attach BMW data check box	You can include BMW data and your project files. This enables customer support to play back what you were doing in your design leading up to the crash. This check box is unavailable if the BMW feature is not enabled. See alsoCrash Detection
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 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.

Find Dialog Box

To access: Choose the **Edit > Find** menu item.

Find and select single or multiple objects by reference designator, part type, matched length pin pair group, or other attributes.

Note:
Find ignores the Selection Filter on page 137 settings and selects whatever you ask it to.



Objects

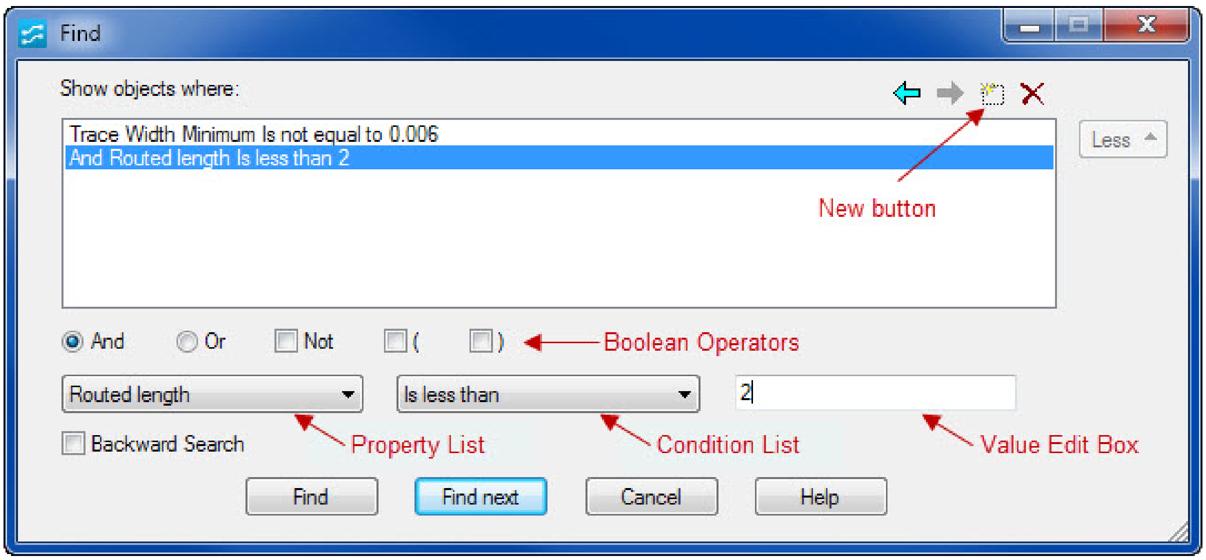
Field	Description
Object Type list	Specifies how you want to search. You can find by any of the following: • Reference Designators • Part types • Decals • Net classes • ML Net groups • Electrical Net • Nets • ML Pin Pair groups

Field	Description
	• Differential pairs • Via types • Jumper vias • Keepouts • 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.
Add to Selection	Selects the items that you find. Each selected item appears in the Selected Items preview area. **Tip** When you select objects with the Find dialog box, the shortcut menus change to the commands relevant for modifying the items.
Selected Items Preview area	Displays the selected items.
Find lists	The content and title of these lists change depending on what you select from the Find By list. For example, if you are finding by reference designator, the first list displays reference designator prefixes. When you select a reference designator prefix to search by, for example D, the second list displays all reference designators with the D prefix. You can further limit the search by choosing a specific D reference designator prefix to search by, such as D2.

Find Dialog Box, Spreadsheet Window

To access: In the Spreadsheet window, click the **Find**button.

Use this dialog box to find objects in the currently selected tab of the Spreadsheet window.



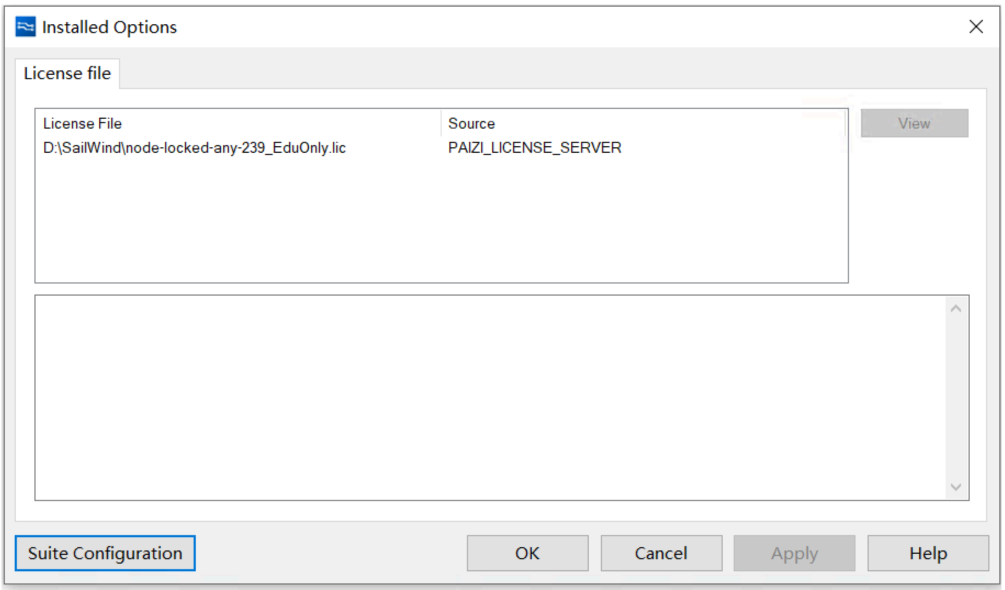
Objects

Field	Description
Show objects where:	Displays the current list of conditions that compose the search expression.
New button	Click to add a condition to the search expression
More button	Click to expand the dialog box to create a multiple-condition search.
Less button	Click to contract the dialog box to create a simple single-condition search.
Boolean Selectors	Set these appropriately to describe the boolean structure as you add additional conditions for multiple-condition searches.
Property List	Select the object property you wish to test against the condition.
Condition List Value Edit Box	Specify the condition and value to test the property against.
Backward Search	Select this check box to search backward from the bottom of the list.

Installed Options Dialog Box, License File Tab

To access: Choose the **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

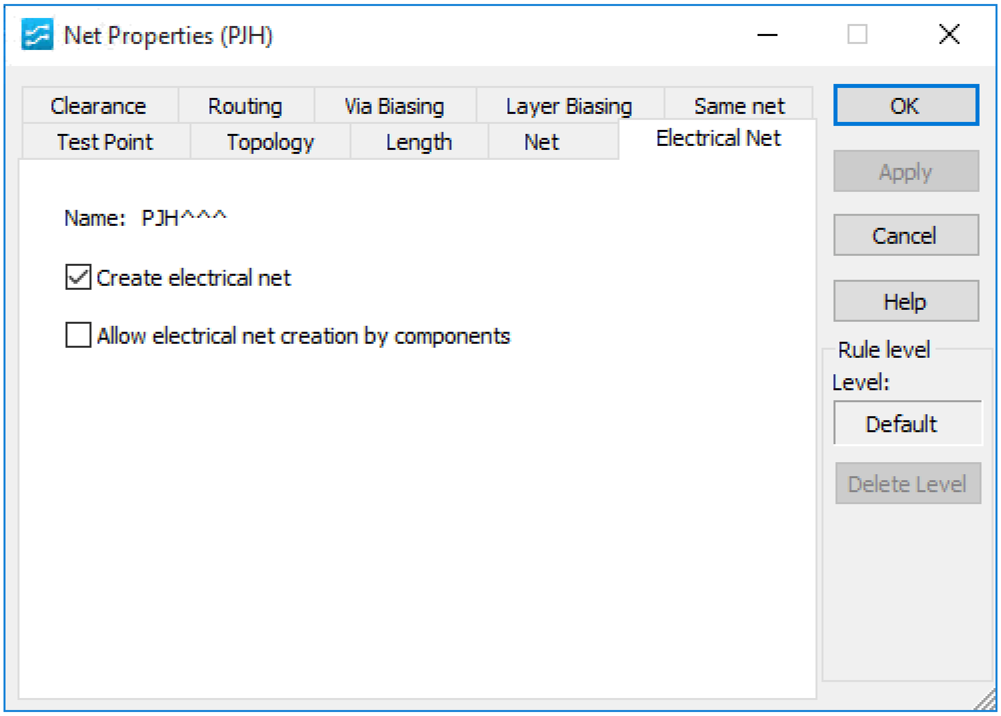
Field	Description
License File column	Displays the location of the license file(s) found on your computer.
Source column	Displays the source of the license.
View button	Click to display the contents of the license file in the License Information box. **Restriction:** Node-locked only.
Status button	Click 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.

Net Properties Dialog Box, Electrical Net Tab

To access: Select a net > Right-click > choose the **Properties** popup menu item > **Electrical Net** tab

View and control the electrical net status of nets.



Objects

Field	Description
Name:	Name of the electrical net of which the net is a part. An electrical net name has a suffix of three caret symbols (^^^).
Create electrical net	Select the check box to create electrical nets of the selected nets. **Tip** You can also set this check box (and create electrical nets) using the Create Electrical Net popup command for selected nets. Clear the check box to remove any selected net that is a part of an electrical net. Clear both check boxes to exclude the selected nets from inclusion in any electrical net.
Allow electrical net creation by component	Select the check box to allow the selected nets to be included in electrical nets by the following:

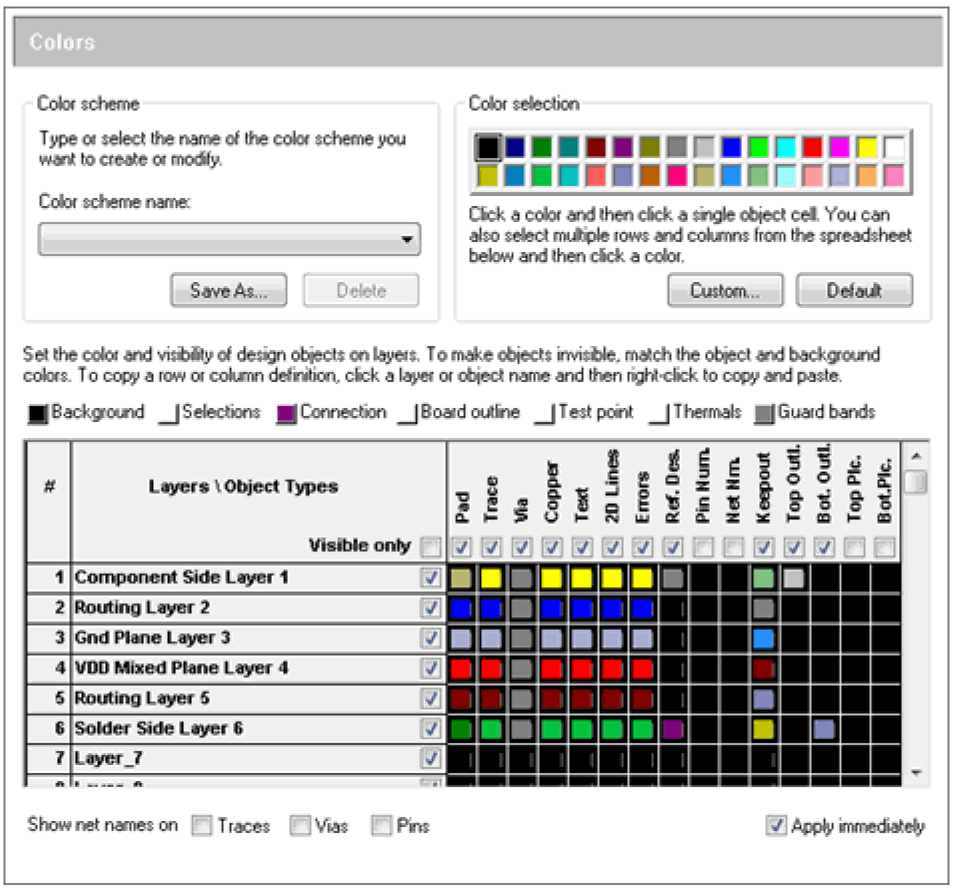
Field	Description
	• Specifying, in the Electrical Nets dialog box, the refdes prefix of the nets' associating components, or • Selecting the Create electrical net check box in the Component Properties dialog box. Clear the check box to disable electrical net creation by component (including by refdes 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.

Options Dialog Box, Colors Category

To access:

- Choose the **Tools > Options** menu item > Colors category
- Click the **Options** button > Colors category
- Press the Ctrl + Enter keys > Colors category

Control the visibility and display colors of design layers and objects. Save color schemes for reuse.



Objects

Field	Description
Display scheme area	
Color scheme name	Select a previously saved color scheme.
Save As	Click to type a name for the new color scheme created from your current settings. The new name appears in the Color scheme

Field	Description
	name list. The color scheme is saved in the .dsf file and is made active. For the location of the .dsf file and how to change the location, see File Locations Options on page 439.
Color selection area	
color palette	Click a color tile. Then in the Layers\Object Types table, click the tile for the object type in a layer.
Custom	Click to change the palette of colors from which you can select.
Default	Click to return the palette of colors from a customized palette to the default color palette.
global object tiles area	**Background** — Specifies the color of the design space background. Setting other objects to this color makes those objects invisible. **Selections** — Specifies the color of selected objects. **Connection** — Specifies the color of unrouted pin pair connections otherwise known as the ratsnest. **Board outline** — Specifies the color of the board outline and board cut outs. **Test point** — Specifies the color of test point symbols that overlay test points. These symbols are only visible on the layer of the board set for probing. **Thermals** — Specifies the color of thermal symbols on pads and vias that have a thermal connection. You can enable and disable the thermal connection in pad and via properties using the “Connect to plane with thermal” check box. **Guard bands** — Specifies the color of the shaded areas around objects that appears while interactively routing traces. The guard band displays the clearance you must keep from those objects. The width of the guard band is the clearance required between the object and trace being routed. The display of these guard bands can be turned on/off in the Options dialog box > **Global** category > General subcategory on page 441 with the “Show guard bands on object” check box.
Layers \ Object Types matrix	
Visible only	Select the check box to list only visible layers. A layer is visible if at least one tile is assigned a non-background color. You can use this setting to shorten the layers list to only those layers with which you are working.
Layers (rows)	The rows of layers lists the layers as you’ve named them in the Layers Setup dialog box in SailWind Layout. **Tips:**

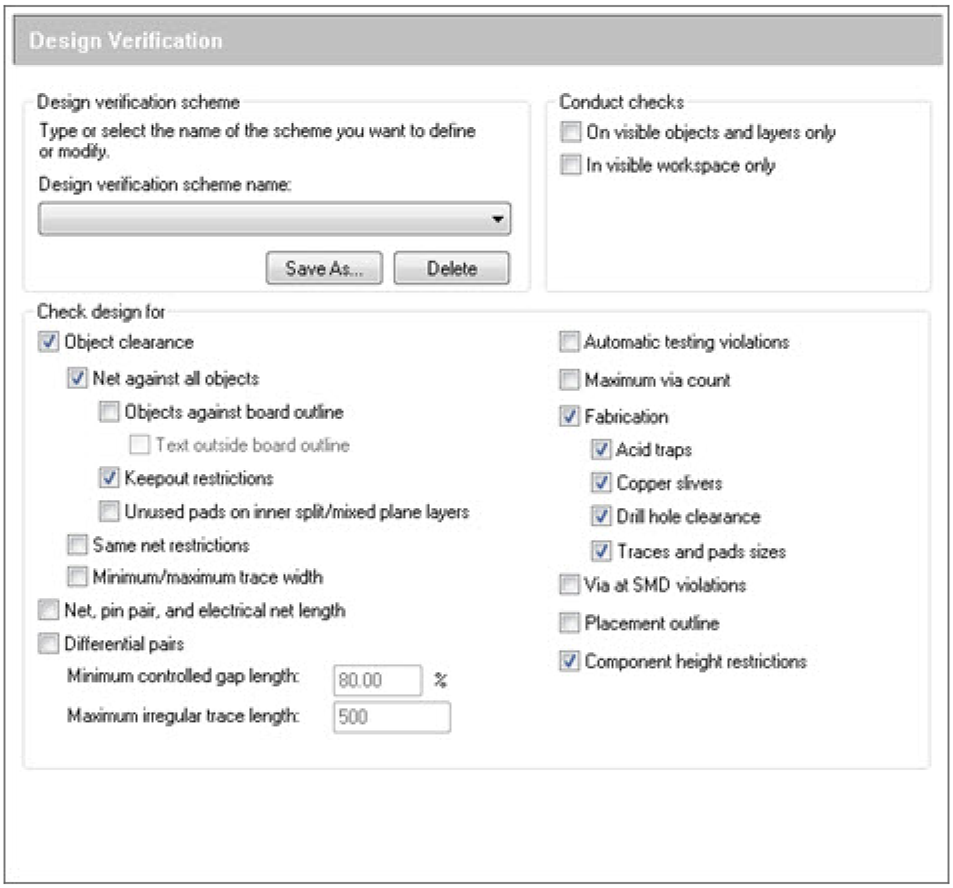
Field	Description
	• You can clear the check box of a row to make the layer invisible. • To make all items on a layer one color, click the number of the row to select all row objects and then click a color tile in the Selected Color area. You can also select multiple rows using Shift+click for a range or Ctrl+click for multiple selection.
Object Types (columns)	The columns list object types in the design.You can clear the check box of a column to make the objects invisible on all layers. • **Top/Bottom Outline Objects** — Used to control the visibility of component body outlines. Tiles in these columns associated with the Top and Bottom Component Side layers (Top Outl./Bot. Outl.) or Top/Bottom Silkscreen layers control the visibility of the component body outlines. • **Top/Bottom Placement Outline Objects** — Used to control the visibility of component placement outlines (derived from SailWind Layout Layer 20 outlines or auto-generated). Tiles in these columns associated with the Top and Bottom Component Side layers (Top Plc./Bot. Plc.) control the visibility of the component placement outlines. • **Pin Numbers and Net Names** — Sizing is controlled by the settings on the Display Options category on page 431. **Restriction:** Even if the Net Nm. 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.
Show net names on	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 Nm. column and give colors to color tiles in the column. **Tips:** • You can use the modeless commands NNT, NNV, and NNP to toggle these check boxes. • The sizing and frequency of net name placement is controlled by the settings on the Display Options category on page 431.
Apply immediately	SailWind Router applies to the design any color or visibility changes at the moment you make them in this dialog box; you do not need to click **Apply** first.

Options Dialog Box, Design Verification Category

To access:

- Click the **Options** button > Design Verification category
- Choose the **Tools > Options** menu item > Design Verification category
- Press the Ctrl + Enter keys > Design Verification category

Use the Design Verification category to set up design verification options.



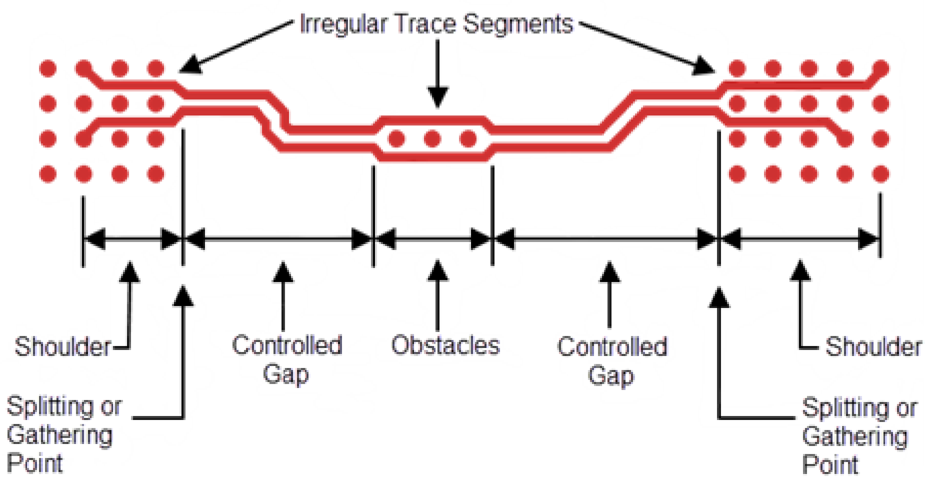
Objects

Name	Description
Design verification scheme area	Select a scheme name if you want a predefined set of design checks.

Name	Description
	Tip A scheme is an easy way to save all your current settings on this tab. You can quickly switch among the schemes to run other checks without having to specify options individually. Set the individual design checks and properties you want, and then click **Save As**. In the Scheme name box, type the name you want to assign to the new design verification scheme, and then click **OK**.
Conduct checks area	**On visible objects and layers only** — Select to avoid checking object and layers that have the Background color in the Layers/Object Types matrix in the Colors category. **In visible workspace only** — Select to avoid checking objects that outside the current view of the workspace.
Check design for area	
Object clearance	Checks the minimum allowable clearance for each object type, and the minimum and maximum trace width. **Tip** Set clearances on the **Clearance** tab of the Properties dialog box. Set trace width on the **Routing** tab of the Properties dialog box. **Net against all objects** — Checks clearance between each net or hierarchical level and any other obstacle type. Also checks clearance for each component or decal that has clearance rules set. **Objects against board outline** — Checks clearance between the centerline of the board outline and the edge of a board cutout or other object. **Text outside board outline** — Checks for text strings on electrical layers that are outside the board outline and flags all instances of off board text as errors. **Keepout restrictions** — Checks for keepout restriction violations. Does not check placement and height keepouts that are verified by Fabrication checking. **Unused pads on inner split/mixed plane layers** — Checks clearance for pads on inner split/mixed plane layers that do not connect to traces. **Same net restrictions** — Checks clearance between objects along the same net. **Minimum/maximum trace width** — Checks traces against the designated minimum and maximum widths.
Net, pin pair, and electrical net length	Checks the minimum and maximum length for nets, pin pairs and electrical nets. **Tip** Set lengths on the “Properties Dialog Box, Length Tab” on page 501.
Differential pairs	Checks against differential pair property settings—the minimum and maximum length, the gap, the number and size of obstacles, and also against the minimum controlled gap length, and maximum irregular trace length settings here.

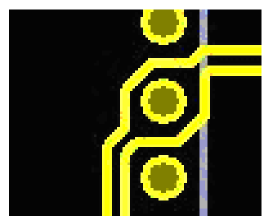
<table>
<tr><th>Name</th><th>Description</th></tr>
<tr><td></td><td>

Figure 96. Differential Pairs Checking



Checks all areas between the gathering point and the split point that are not routed at the fixed edge-to-edge route gap value. Does not check areas where the number and size of obstacles do not exceed the maximum values set. Differential pair traces should be at the gap before they split around an obstacle. They should also join back together at the gap.

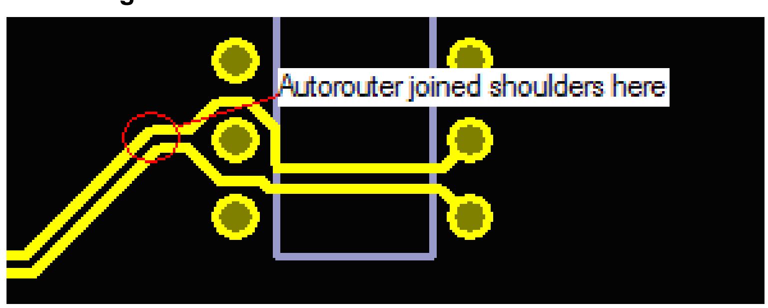
Figure 97. Differential Pairs - Splitting Around an Obstacle at the Gap



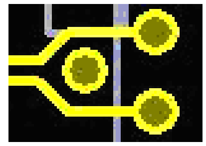
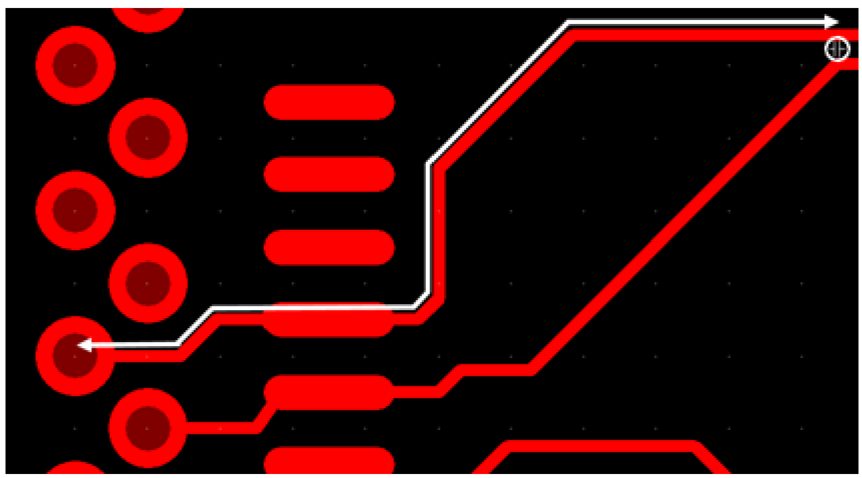
Tip

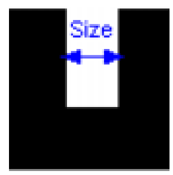
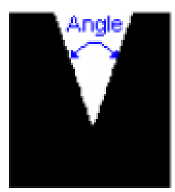
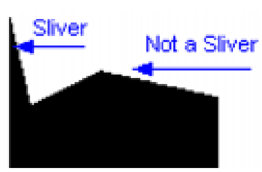
The shoulder area may be wider (from the autorouting point of view) than it appears to be in reality.

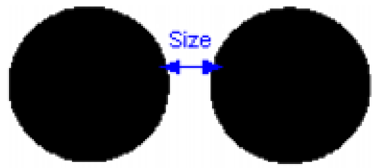
Figure 98. Differential Pairs - Wider Shoulders



</td></tr>
<tr><td>Differential pairs (continued)</td><td>Checks the number of routing obstacles routed around by each differential pair and reports if this number is over the specified value. Obstacles in the shoulder area are not counted toward the obstacles over the limit.</td></tr>
</table>

Name	Description
	Figure 99. Differential Pairs - Obstacle in Shoulder Area  Checks the maximum obstacle size routed around by each differential pair. **Minimum controlled gap length** — Set the percentage of the length of the diff pair that should have a controlled gap without obstacles. **Maximum irregular trace length** — Checks for trace segments not routed at the route gap and identifies any trace segments longer than the specified value that might affect the integrity of the differential pair. Irregular trace segments include shoulder traces in the start and end zones as well as trace segments in the controlled gap portion of the routing where the gap is wider than the value of the route gap. Reported lengths in error descriptions include trace segments to the center of pads and vias. In the following example, the shoulder area is extensive and bridges two components. The irregular trace length report includes the entire length shown with the white arrow.  **Tip** You set the gap between objects, restrict layer changes, minimum and maximum length, allow pair to split around obstacles, maximum number of obstacles, and maximum size of obstacles on the “Differential Pair Properties Dialog Box, Pair Tab” on page 511. **Restriction:** Maximum number and size of obstacles are not checked in the zone between the connection point and the shoulder of the differential pair.
Automatic testing violations	Checks for probe clearances, minimum pad sizes, SMD pin probing, test points on component pin on the component side, test point count per net setting, nail diameter settings, and test points on grid.
Maximum via count	Checks for the maximum number of vias on nets and pin pairs.
Fabrication	Checks the pad and trace size, drilling clearance, component assembly, acid trap size, and copper slivers. SailWind Router does not have CAM document capabilities required to process some fabrication checks. Full fabrication checking can only be

Name	Description
	performed within SailWind Layout. SailWind Router does not check the following: • Minimum solder mask slivers • Starved thermal checks • Silk screen over pads • Pad or drill to mask minimum annular ring • Connect to plane (has thermal where needed) • Solder bridge **Tip** Set fabrication rules on the "Options Dialog Box, Fabrication Category" on page 433. **Acid Traps** — Checks all visible electrical layers as defined by CAM documents. Maximum area size indicates the maximum area of the acid traps to flag. The area of pools that are flagged will be less than this value. **Figure 100. Acid Trap Maximum Area Size Checking**  Maximum angle between objects is an angle between 0 and 90 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 101. Acid Trap Angle Checking** 
Fabrication (Continued)	**Copper slivers** — Checks if there 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. Minimum width indicates the maximum width of copper slivers to flag. This flags slivers of a width less than this value. Minimum Copper checks all visible electrical layers as defined by CAM documents. **Figure 102. Sliver Checking**  Mask slivers in the solder mask layer are areas of the solder mask 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.

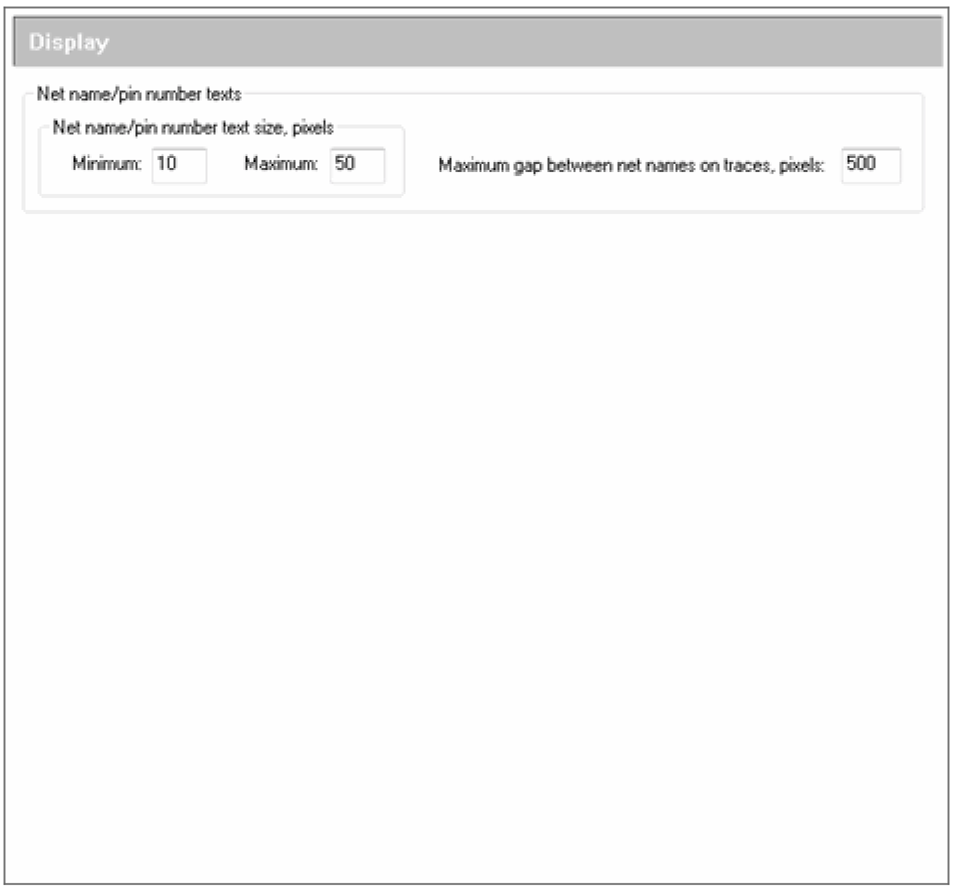
Name	Description
	Minimum Mask indicates the maximum size of the slivers to flag. This flags slivers of a width less than this value. Minimum Mask checks top and bottom solder mask layers, if visible as defined by CAM documents **Figure 103. Minimum Mask Checking**  **Drill hole clearance** — If drills are too close to one another, they may break out during board manufacturing. Drilling clearance checks the minimum distance between drill hole edges. It also checks the allowance for the plated drill.
Fabrication (Continued)	**Traces and pads sizes** — Checks minimum trace width and minimum pad size of electrical layers. Detects small electrical traces on the electrical layers in the design. Minimum Trace indicates the maximum allowable size of traces; traces with a width less than minimum trace will be flagged. This check runs on all visible electrical layers as defined by CAM documents. Detects small pads on the electrical layers. Minimum Pad indicates the maximum allowable pad size; pads with a diameter less than minimum pad will be flagged. This check runs on all visible electrical layers as defined by CAM documents.
Via at SMD Violations	Checks if the SMDs are to fit inside the pad, center within the pad, or be located at the end of the pad. The checking takes place only if the via outline intersects the SMD outline or is located completely inside SMD. The absence of a via at SMD is not considered an error.
Placement outline	Checks that the distance between component placement outlines complies with the “Minimum spacing between components” value set in the Options dialog box, Fabrication category. Measurements are made from the centerlines of the 2D lines that define the component placement outline. This check applies to all placement outlines (derived from SailWind Layout Layer 20 outlines or auto-generated).
Component height restrictions	Checks the maximum component height on the top and bottom.

Options Dialog Box, Display Category

To access:

- Choose the **Tools > Options** menu item > Display category
- Click the **Options** button > Display category
- Press the Ctrl + Enter keys > Display category

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



Objects

Field	Description
Net name/pin number text size, pixels	Type values for the smallest and largest text sizes you use in the net names. Net names change in size depending on the shape on which they are placed. The text remains in proportion to the boundaries of the object. For example, the net name will be smaller on a 10 mil trace than it would on a 50 mil trace.

Field	Description
Restriction: Net names will not appear on design objects, unless on the Colors Options category on page 422, you also select the check box for the Net Nm. column, give colors to color tiles in the column on the layers where you want the net names to appear, and select one or more of the “Show net names on” check boxes.	
Maximum gap between net names on traces, pixels	Type a value to set the gap at which the net names should be repeated at intervals along traces. The gap must be between 50 and 3000 pixels.

Options Dialog Box, Fabrication Category

To access:

- Choose the **Tools > Options** menu item Fabrication category
- **Options** button > Fabrication category
- Ctrl + Enter > Fabrication category

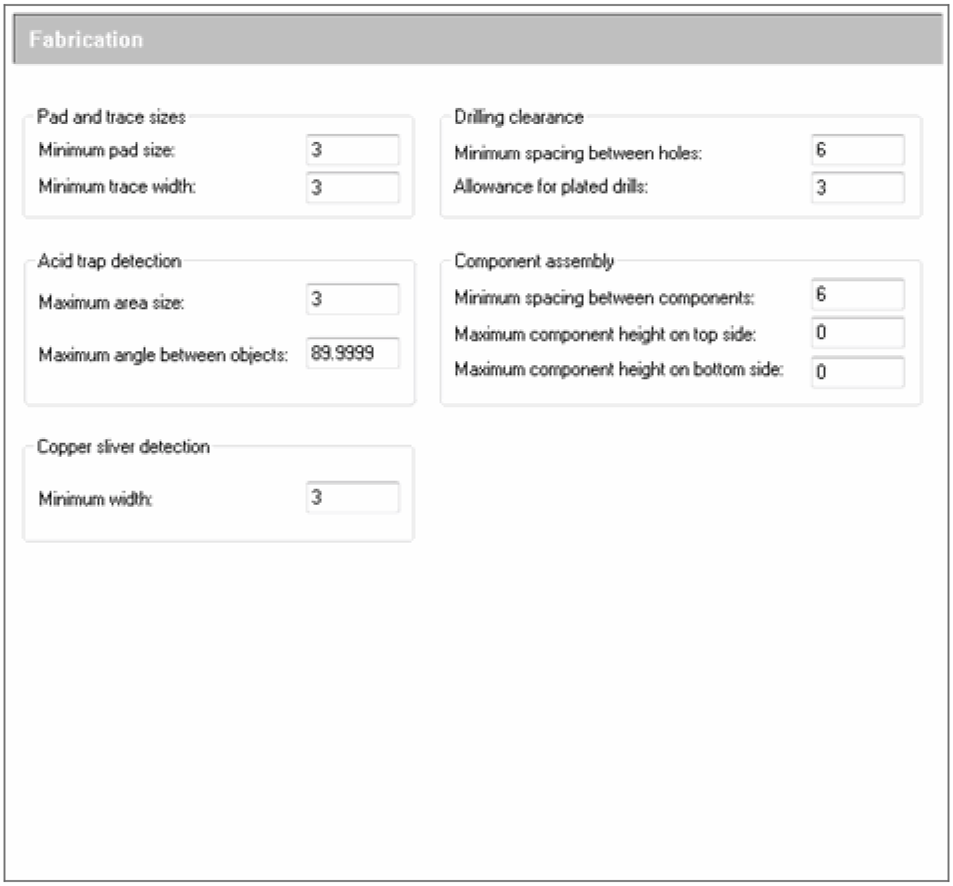
Use the Fabrication category to set design fabrication rules during design verification. Fabrication checks detect conditions that may result in errors in PCB fabrication or manufacturing.

Description

Restriction:
You cannot set fabrication rules at any level of the hierarchy except for component and drill clearances. You establish manufacturing clearances prior to performing the checks.

Restriction:
SailWind Router does not have the CAM document capabilities required to process some fabrication checks. Full fabrication checking can only be performed in SailWind Layout. SailWind Router does not check the following:

- Minimum solder mask slivers
- Starved thermal checks
- Silkscreen over pads
- Pad or drill to mask minimum annular ring
- Connect to plane (has thermal where needed)
- Solder bridge



Objects

Name	Description
Pad and trace sizes area	
Minimum pad size	Sets the minimum diameter of the pads allowed by the fabrication procedures.
Minimum trace width	Sets the minimum trace width allowed by the fabrication procedures.
Acid trap detection area	
Maximum area size	Specifies the maximum size of the acid traps to detect. It flags the small areas where the acid might pool up. The area flagged for a pool up should be smaller than this value.
Maximum angle between objects	Specifies areas of copper items (traces, pads, or any other objects that exist on the layer) with angles smaller than this value. Any of these items is flagged as an acid trap if it forms an angle smaller than the value specified here.
Copper sliver detection area	

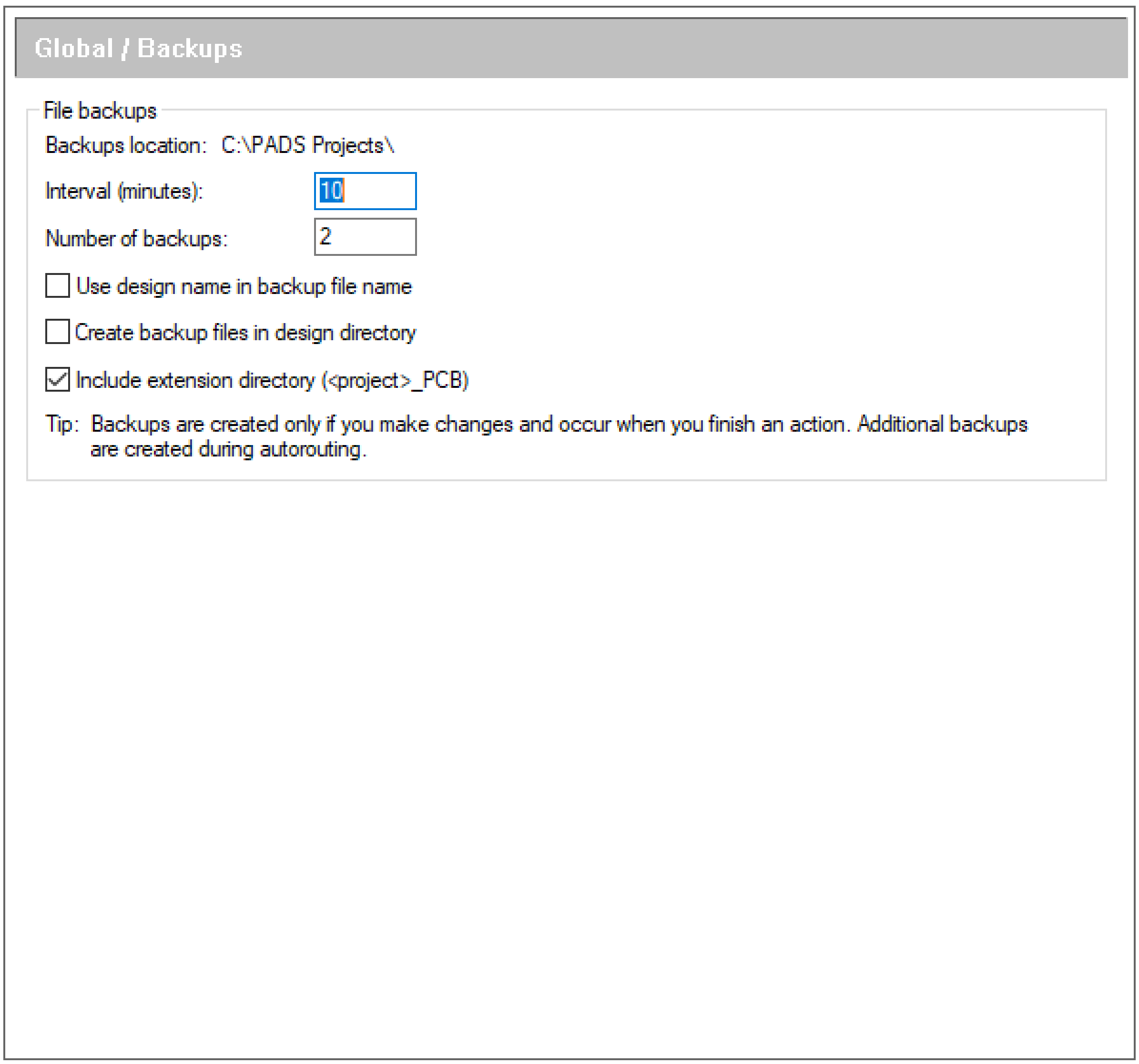
Name	Description
Minimum width	Specifies the minimum size of the copper slivers to detect as errors. Slivers with widths smaller than this value are flagged.
Drilling clearance area	
Minimum spacing between holes	Specifies the minimum value of the edge-to-edge spacing between two drill holes.
Allowance for plated drills	Specifies the amount of space added to the plated drill holes to be used for copper plating inside a drill hole. **Tip** You specify whether a via or pin is plated in Pad Stack Properties dialog box of SailWind Layout.
Component assembly area	
Minimum spacing between components	Specifies the minimum distance between component placement outlines. Measurements are made from the centerlines of the 2D lines that define the component placement outline. This check applies to all placement outlines (derived from SailWind Layout Layer 20 outlines or auto-generated). **Note:** Upon loading a design from SailWind Layout, this value populates with the value from the SailWind Layout “Body to body” clearance rule.
Maximum component height on top side	Specifies the maximum height for all components on the top side layer. **Tip** Use the **Clearance** tab of the Component or Decal Properties dialog boxes to set this value.
Maximum component height on bottom side	Specifies the maximum height for all components on the bottom side layer. **Tip** Use the **Clearance** tab of the Component or Decal Properties dialog boxes to set this value.

Options Dialog Box, Global Category, Backups Subcategory

To access:

- Choose the **Tools > Options** menu item > **Global** category > **Backups** subcategory
- Click the **Options** button > **Global** category > **Backups** subcategory
- Press the Ctrl + Enter keys > **Global** category > **Backups** subcategory

Set the location and make other settings for automatic backups.



Objects

Table 87. Options Dialog Box, Global Category, Backups Subcategory Contents

Field	Description
Backups location	Displays the backup file location. This location is set in the File Locations category on page 439. **Tip** Backup files have the *.bre* extension.
Interval (minutes)	Type the time in minutes between backups.
Number of backups	Type the number of backups to keep before deleting old ones. When the number of backups reaches this limit, the oldest existing backup is deleted whenever a new backup is created. For example, if the number of backup files is set to 2, only the last two backup files are kept, depending on the interval you set. **Tip** This limit can specify the maximum number of backups for all designs, or the number of backups for each design. For more information, see the "Table 88" table.
Use design name in backup file name	Check the check box to use the design name plus "Router" in the backup filename, as follows: *<design_name>_Router_<month_day_hour_minute>.bre* If this check box is not checked, backup files are named in this format: *Router_<month_day_hour_minute>.bre*
Create backup files in design directory	Select this check box to place your backup files in the same directory as the design. This overrides the location set in the File Locations category on page 439. **Tip** Clear the check box if you want your backup files in one common backup directory.
Include extension directory (<project>_PCB)	Select this check box to create a directory containing the 3D support files used in the design. The directory is named in this format: *<design_name>_PCB* This check box is checked by default. **Tip** Clear the check box if you do not want to create a backup directory containing the 3D support files used in the design. **Note:** If you have assigned 3D models to parts in a design you must keep the *<design_name>_PCB* folder and the *.pcb* file together in the same directory location or the next time you open the *.pcb* file the 3D models will not appear.

Table 88. File Backup Strategies

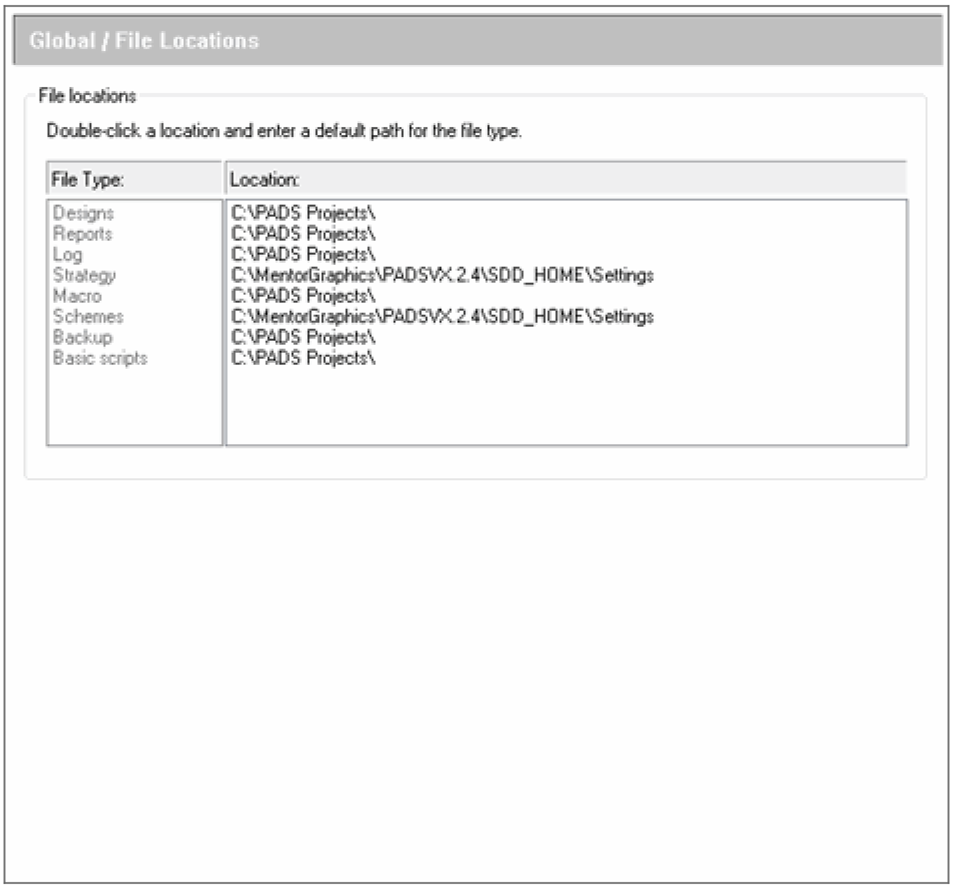
Backup Settings	Resulting Backup Behavior
☐ Use design Name in backup file name ☐ Create backup files in design directory	All backups saved in the default backup directory, and named *Router_month_day_hour_minute.bre*. “Number of backups” specifies the maximum total number of backups *for all designs.*
☑ Use design Name in backup file name ☐ Create backup files in design directory	All backups saved in the default backup directory, and named *<design_name>_Router_month_day_hour_minute.bre*. “Number of backups” specifies the maximum number of backups *for each design.*
☐ Use design Name in backup file name ☑ Create backup files in design directory	Backups of each design saved in the design directory, and named Router_*month_day_hour_minute.bre*. “Number of backups” specifies the maximum number of backups *for each design.*
☑ Use design Name in backup file name ☑ Create backup files in design directory	Backups of each design saved in the design directory, and named *<design_name>_Router_month_day_hour_minute.bre*. “Number of backups” specifies the maximum number of backups *for each design.*

Options Dialog Box, Global Category, File Locations Subcategory

To access:

- Choose the **Tools > Options** menu item > **Global** category > **File Locations** subcategory
- Click the **Options** button > **Global** category > **File Locations** subcategory
- Press the Ctrl + Enter keys > **Global** category > **File Locations** subcategory

Change file locations.



Objects

Field	Description
File locations area— In the Location section, double-click the location you want to change and type a new location.	

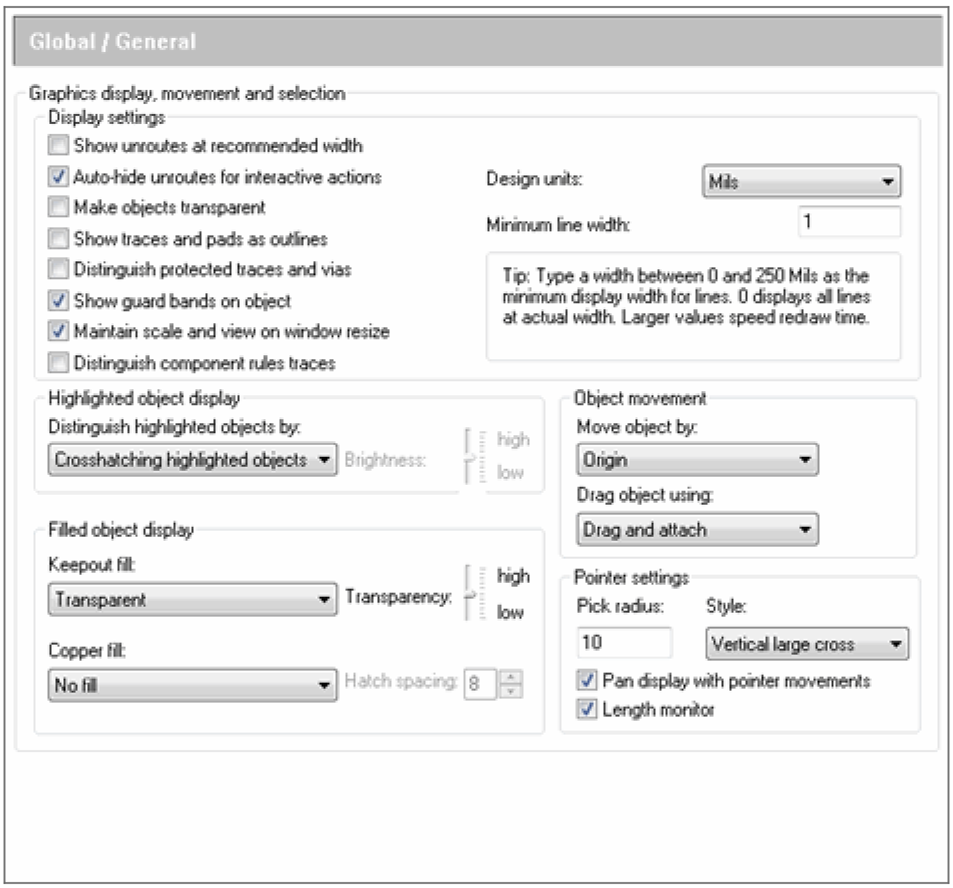
Field	Description
Alternative: In the Location section, double-click the location you want to change, and then click the **Browse** button to locate an existing folder. You can change to any location without compromising your data.	

Options Dialog Box, Global Category, General Subcategory

To access:

- Choose the **Tools > Options** menu item > **Global** category > **General** subcategory
- Click the **Options** button > **Global** category > **General** subcategory
- Press the Ctrl + Enter keys > **Global** category > **General** subcategory

Set user interface options that are global to the design process.



Objects

Field	Description
Display settings area	

Field	Description
Show unroutes at recommended width	Displays unroutes at the recommended width of a trace being routed. This option allows you to see the width of the trace before routing it. **Tip** You can set the recommended trace width on the Properties dialog box. In an open design with nothing selected, right-click, choose the **Properties** popup menu item, then open the **Routing** tab.
Auto-hide unroutes for interactive actions	Temporarily hides unroutes when you route or edit traces interactively.
Make objects transparent	Shows traces on multiple layers at once and makes all obstacles that may be hidden directly under the active layer visible.
Show traces and pads as outlines	Shows all objects as outlines instead of as filled objects to speed redraw time. Traces appear as two parallel lines separated by the defined trace width. This also makes it easier for you to see the true boundaries of overlapping objects.
Distinguish protected traces and vias	Displays normal unprotected objects as solid, and any protected objects as outlines. If you also select Show traces and pads as outlines, protected objects appear filled, and unprotected objects appear as outlines.
Show guard bands on object	Displays guard bands around objects during interactive routing. A guard band is a shape around traces and other obstacles. The width of the guard band is the required clearance between an object and the trace being routed.
Maintain scale and view on window resize	Maintains the proportional scale and the same area of the workspace, when you resize the window.
Distinguish component rules traces	Displays traces under component rules as outlines.
Design units	Select the design units you want to work with. All values in dialog boxes are displayed in these units, and any values you enter are interpreted in these units. **Tips:** • Values are converted when you change design units. • You can also use the following modeless commands to set design units: UM — Sets design units to mils. UMM — Sets design units to millimeters (metric). UI — Sets design units to inches. UUM — Sets design units to microns.
Minimum line width	Type a value to display lines with widths less than this value as centerlines, rather than as their actual width.
Highlighted object display area	

Field	Description
Distinguish highlighted objects by	Select the way you want the highlighted objects to appear in the design. You can identify objects in the open design by differentiating them from other design objects. **Exception:** You can only set the Brightness level if you selected Dimming other object color.
Filled object display area	
Keepout fill	Specifies how keepout areas appear on the screen. **Transparency** — Becomes available when you select the “Transparent” fill option. The “high” setting is equivalent to no fill and “low” setting is equivalent to solid fill.
Copper fill or Copper, copper plane fill	Specifies how filled solid copper objects appear on the screen. When the dynamic copper healing mode is disabled on the Flooding category on page 447 of the Options dialog box, the title of this list changes to “Copper, copper plane fill” indicating that the copper fill setting also affects the display of copper planes. **Hatch spacing** — Becomes available when you set the fill setting to a “hatch” setting (Orthogonal hatch, Diagonal hatch, Orthogonal crosshatch, or Diagonal crosshatch). The Hatch spacing value must be between 1 and 32 in pixels.
Object movement area	
Move object by	• **Origin**—The pointer attaches to the origin of the object. • **Cursor location**— The pointer attaches to a point that is set when you started the move and maintains its X/Y offset relative to the object during the move. Example: If you start the Move command with the pointer at X=200,Y=500 and the selected component at X=0,Y=0, moving the pointer to X=1200,Y=1500 moves the component to X=1000,Y=1000. • **Midpoint**— The pointer attaches to the center of a rectangle enclosing the object.
Drag object using	• **Drag and attach**— Select the object, drag the object slightly to attach it to the pointer, and then release the mouse button. Move the pointer to the new location and click to complete the move. **Tip** You can change the workspace view, start another command, or change the selected component properties, while the object is attached to the pointer. • **Drag and drop**— Drag the object to the new location. • **No drag moves**— Disables all drag moves.
Pointer settings area	
Pick radius	The Pick radius value must be between 1 and 100 in pixels. Larger values mean the pointer can be further away and still select an

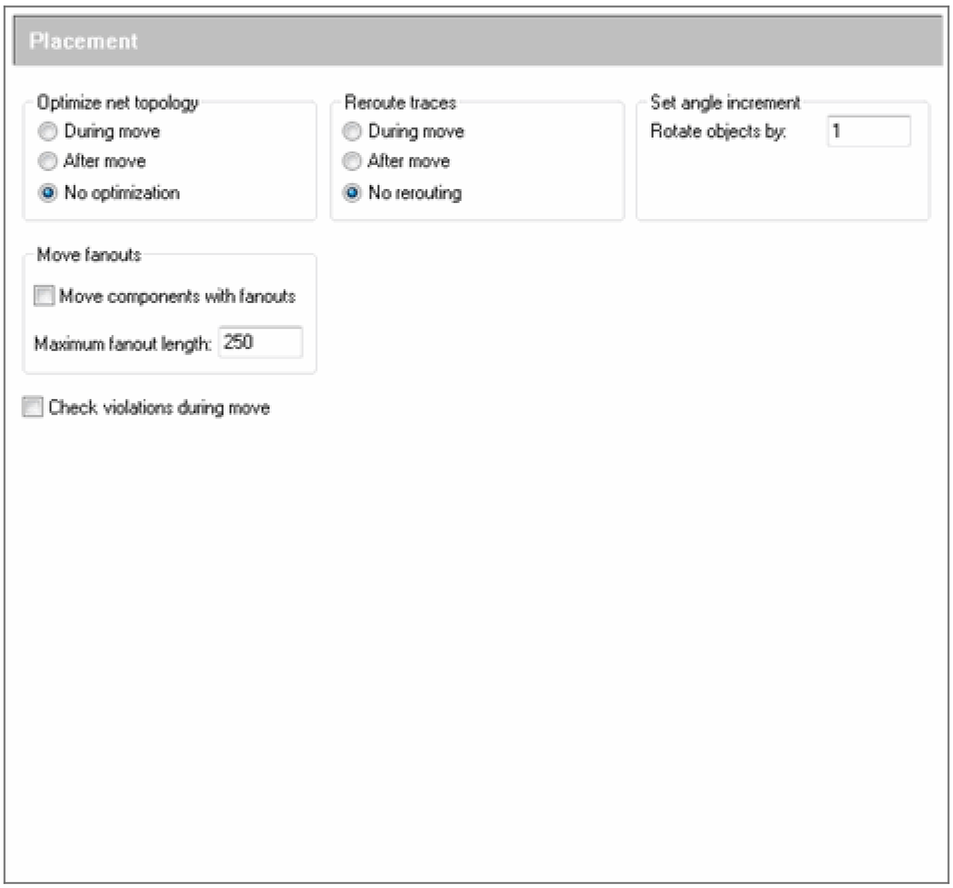
Field	Description
	object, but this can cause you to select other unintended objects within the same vicinity. **Tip** When many objects are in the same vicinity, you can cycle through them using the Tab key.
Style	Select a style of cursor from the list.
Pan display with pointer movements	Select the check box to pan the display area when moving objects with the cursor.
Length monitor	Select the check box to display net length values on the pointer.

Options Dialog Box, Placement Category

To access:

- Choose the **Tools > Options** menu item > **Placement** category
- Click the **Options** button > **Placement** category
- Press the Ctrl + Enter keys > **Placement** category

Specify the behavior of placement operations for components in a design.



Objects

Field	Description
Optimize net topology	Defines when to optimize the net topology. With net optimization enabled, the program tries to minimize connection lengths. You can optimize nets during component placement, after placing the component, or you can move the component without optimizing the net topology.

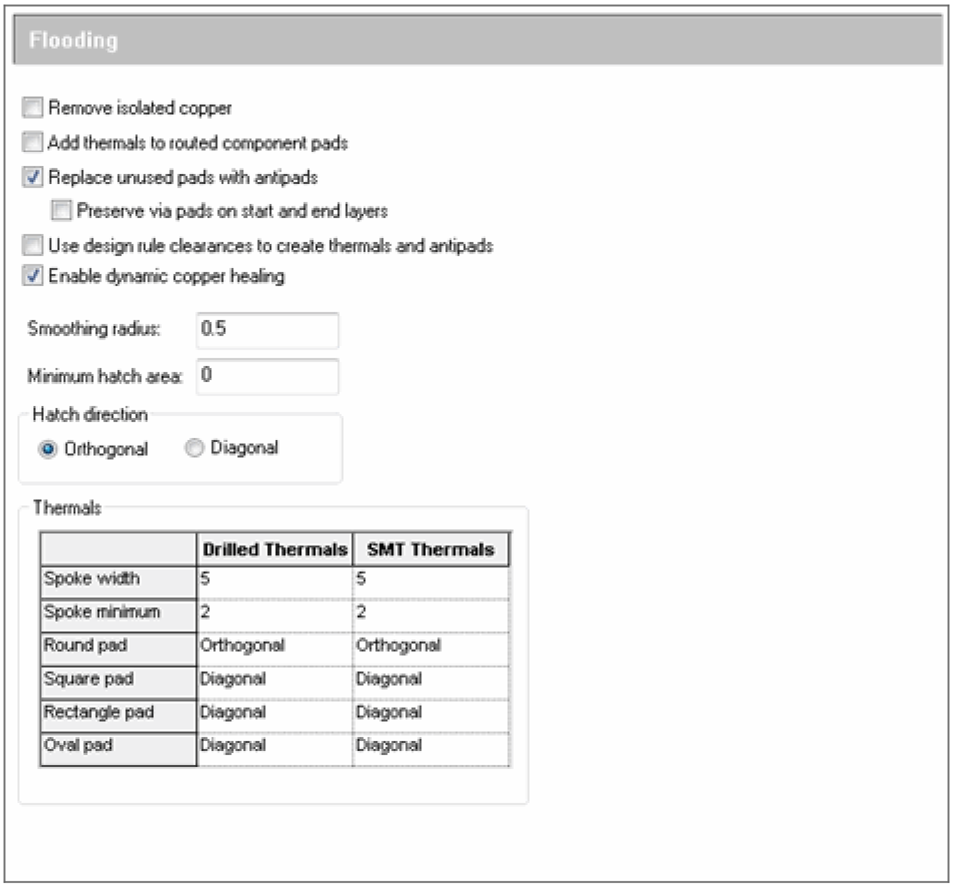
Field	Description
	See also "Setting Net Topology" on page 291
Reroute traces	Defines the rerouting behavior for placement commands when moving a component with connected traces. **During move** — Reroutes nets during placement. Traces that cannot be routed free of DRC violations during placement appear to be stretched and highlighted, and are rerouted again with batch routing tools after placement is completed. **After move** — Unroutes nets during placement, then reroutes them when placement is complete. **No rerouting** — Unroutes nets during placement and leaves them unrouted when placement is complete. When you disable trace rerouting, traces attached to components that are moved, rotated, spun, or flipped before the placement operation starts are unrouted. See also "Reroute Traces" on page 349
Set angle increment	Sets the minimum angle increments for the Spin command. Type a value between 0 and 360 degrees to determine the angle by which to rotate objects when using the Spin command. During the Spin command, the placement grid settings are ignored and angle increments are used instead. See also "Manipulating Components" on page 209
Move fanouts area	Select the check box to specify whether or not to move fanout traces and vias and the selected component as a group during placement operations Type the maximum length of fanout traces to move with the component.
Check violations during move	Select to check placement rule violations when moving or spinning a component. If body-to-body violations occur when moving or spinning a component, the neighboring components are highlighted, including placement keepouts or board outlines.

Options Dialog Box, Flooding Category

To access:

- Choose the **Tools > Options** menu item > Flooding category
- Click the **Options** button > Flooding category

Specify the behavior and settings for the representation of copper planes ("flooding") in a design.



Objects

Field	Description
Remove isolated copper	In plane areas, toggles the visibility of copper material that is not connected to a net.
Add thermals to routed component pads	Allows placement of thermals on routed pads. Thermals are normally generated only on unrouted pads, and vias.

Field	Description
	Tip Small leftover trace segments attached to a pad prevent the pad from receiving thermals if this check box is not selected.
Replace unused pads with antipads	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. **Note:** For CAM planes, this setting has an effect only in SailWind Layout. For partial vias, selecting the **Preserve via pads on start and end layers** check box prevents removal of the starting or ending pad if either is on a plane layer.
Use design rule clearances to create thermals and antipads	Uses the hierarchical “Pad to Copper” clearance rule for thermals and the “Drill to Copper” clearance rule for antipads. **Tip** Keep the following in mind: • 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/diameters/size is less than, or equal to, the inner width/diameter/size in the flood over settings.
Enable dynamic copper healing	Displays copper planes in *dynamic healing* mode, meaning edits you make to copper plane flooding—such as changes to clearances around pads, vias, and other objects—appear in real-time. Dynamic copper healing mode has the following effects: • Copper planes appear filled • Solid copper areas display according to the settings in the “Filled object display area” on page 443 on the Global category > General subcategory • PCB files saved from within SailWind Router retain hatch areas created in SailWind Router **Note:** When a design is first loaded and this option is not enabled, hatch areas created in SailWind Layout display according to the SailWind Layout settings. If this option is then enabled, fill settings from SailWind Layout are overridden by dynamic copper healing and they cannot be recalled. To reinstate the original settings, you need to reload the design from SailWind Layout.
Smoothing radius	Controls the radius of copper plane corners, according to the following: <corner radius> = <shape outline width> * <smoothing radius> You can set the smoothing radius parameter between 0 and 5.

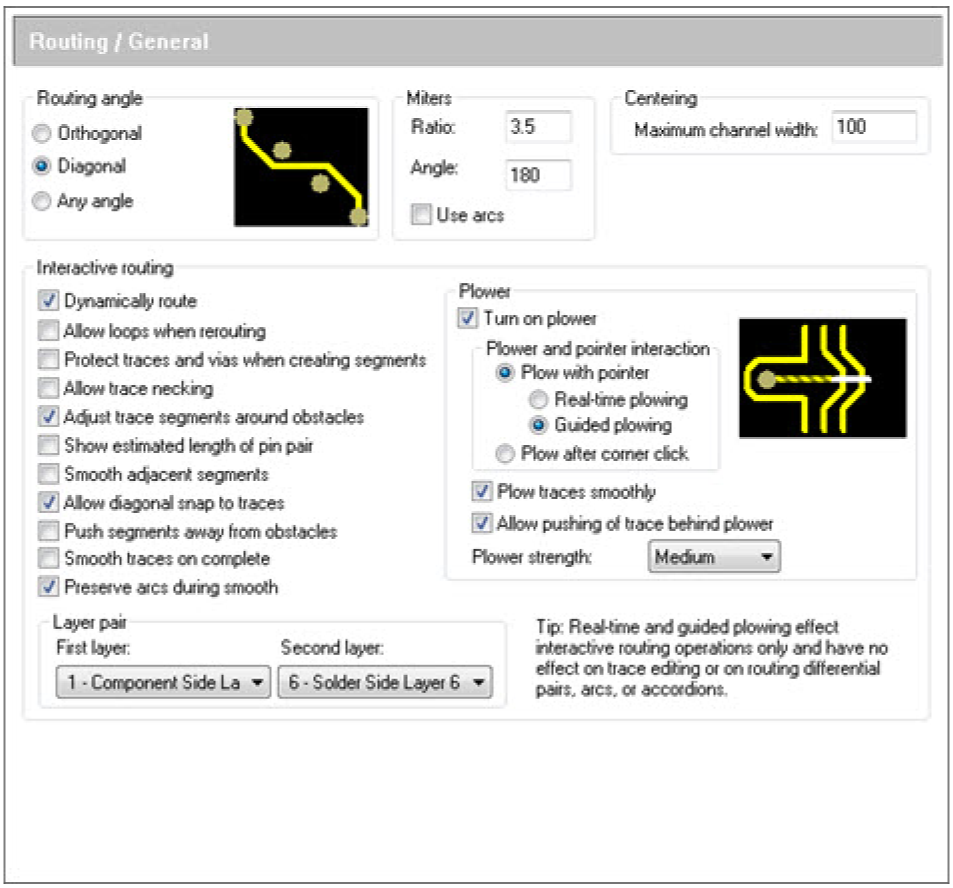
Field	Description
Hatch direction	Specifies the hatch orientation in the workspace. Choose one of the following settings: • Orthogonal — Hatching consists of vertical and horizontal lines. • Diagonal — Hatching consists of diagonal lines. You can change the hatch spacing using the Grid tab on page 491 of the Design Properties dialog box.
Minimum hatch area	Specifies the square root, in current design units, of the minimum hatch outline area (the smallest island area created by flooding). For example, if you do not want to display islands smaller than 9 square design units, type “3” into the box.
Thermals	Provides a table of thermal settings for through-hole and surface mount pad stacks. • **Spoke width** — Specifies the line width for the thermal relief in current design units. • **Spoke minimum** — Specifies the minimum number (1-4) of spokes. • pad shape (**Round pad**, **Square pad**, **Rectangle pad**, **Oval pad**) — Indicates the thermal relief options for a given pad shape. Choose a relief option by clicking the corresponding dropdown list: • **Orthogonal** — Creates orthogonal-shaped thermal reliefs. • **Diagonal** — Creates diagonal-shaped thermal reliefs. • **Flood Over** — Creates thermal reliefs that flood over the pad. • **No Connect** — Creates no thermal reliefs.

Options Dialog Box, Routing Category, General Subcategory

To access:

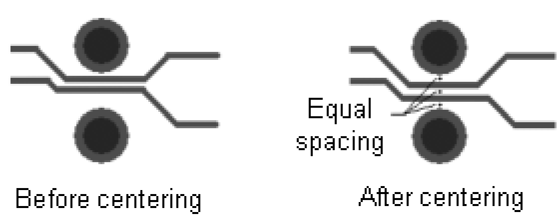
- Click the **Options** button > **Routing** category > **General** subcategory
- Choose the **Tools > Options** menu item > **Routing** category > **General** subcategory
- Press the Ctrl + Enter keys > **Routing** category > **General** subcategory

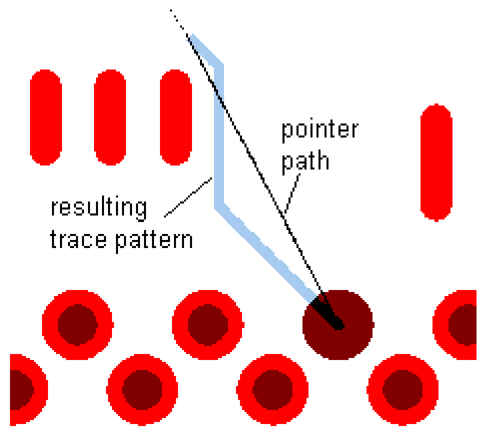
Specify interactive routing and autorouting settings.

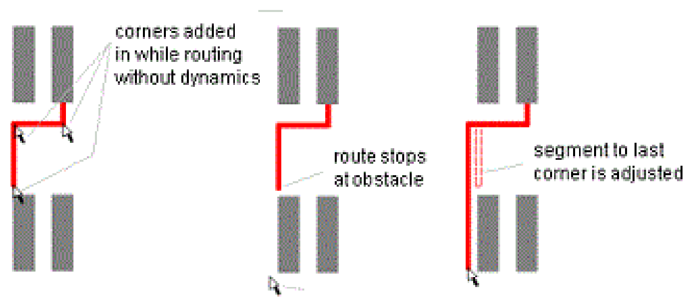
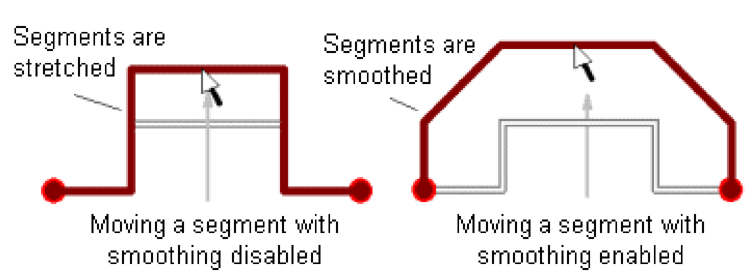


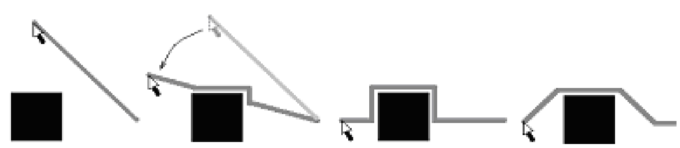
Objects

Field	Description
Routing angle area	Set the default angle used for interactive routing or the angle used globally for all autorouting.

Field	Description
	• **Orthogonal**— Adds trace segments at 90 degree increments. • **Diagonal**— Adds trace segments at 45 degree increments. • **Any angle**— Adds trace segments at any angle. **Exceptions:** • While you can change this routing angle during interactive routing, you cannot change the routing angle while autorouting. • Traces entering or exiting pads may ignore the routing angle to maintain other rules, such as first corner rules or grids.
Miters area	Use miters to specify values for replacing a corner with a diagonal segment or arc. These options also apply to the Miters pass of the autorouter. • **Ratio**— Specifies the size of a diagonal miter or the radius of an arc miter. Type the ratio value to define the ratio of the arc radius to the trace width. **Example:** For a 12-mil trace, a ratio of 1 produces a radius of 6; a ratio of 2 produces a radius of 12. See also Miters • **Angle**— Specifies the maximum corner at which miters or arcs are created. • **Use arcs**— Select to create miters with arcs for both regular and differential pair traces. **Tip** The Use arcs setting is used by the Miters Pass only. See also Converting Corners to Arcs
Centering area	**Maximum channel width**— Set a maximum channel width to limit the number of channels to center. Any channel larger than this width is not eligible for centering. Center is a pass of the autorouter that you can also interactively run on selected traces. Centering places traces equidistant from the pads of adjacent component pins or vias to evenly distribute any available space in the channel, as shown below. Centering does not violate design rules. **Figure 104. Centering Traces**  See also: Center Pass
Interactive routing area	

Field	Description
Dynamically route	Select to create trace patterns automatically as you move the pointer, as shown below. The software automatically adds corners, avoids obstacles, and creates an error-free trace pattern. If the “Dynamically route” check box is cleared, you must click to manually enter corners and define the trace pattern. **Tip** Dynamic routing is always enabled when you route differential pairs. **Alternative:** While interactively routing, right-click and choose the **Dynamically Route** popup menu item to temporarily turn dynamic routing on or off. **Figure 105. Dynamic Routing** 
Allow loops when rerouting	Select to allow a route loop. Add a second connection and the first connection will not be ripped up. When this check box is cleared, in interactively routing you are prevented from creating a loop in a trace. Any trace pattern you define that creates a loop is adjusted automatically and a more direct path is created instead.
Protect traces and vias when creating segments	Select to protect trace segments and vias as they are added. **CAUTION:** You should protect arcs because autorouting operations and plowing in interactive routing may remove unprotected arcs. Alternative: While interactively routing, right-click and choose the **Protect Traces and Vias** popup menu item.
Allow trace necking	Select to automatically reduce the trace width to the minimum trace width to escape the object from which you are routing when the recommended or current trace width is too large. If this check box is not selected, and the trace width is too large, then interactive routing cannot start. You will have to manually reduce the trace width to escape the starting object and then increase the trace to the desired width. **Tips:**

Field	Description
	• Set the Minimum, Recommended, and Maximum trace widths on the **Routing** tab of the Properties dialog boxes. • For best results, set the Minimum trace width for the net equal to or smaller than the smallest pad width for the pins in the net. • This feature is most useful when used with guard bands. See also Showing Clearances Around Obstacles
Adjust trace segments around obstacles	Select to allow trace segments to adjust around obstacles when not using dynamic routing. **For example:** If you add a corner, and the trace encounters an immovable obstacle (such as a component pin), the last-entered trace corner moves automatically to a new position that allows the trace to pass the obstacle, as shown in Figure 106. **Figure 106. Adjusting Around Obstacles** 
Show estimated length of pin pair	Select the check box to estimate the total length (total net length + unrouted length) of the current pin pair in the trace length monitor. If the check box is cleared, the monitor estimates the total length of the entire net (routed length + unrouted length). See also Monitoring Trace Length
Smooth adjacent segments	Select to smooth adjacent trace segments when moving a segment, arc, or adding or moving vias. If the check box is cleared, adjacent segments are stretched. **Figure 107. Stretched and Smoothed Adjacent Segments**  **Exception:** This option does not apply when adding or moving corners. See also Trace Smoothing for other smoothing options

Field	Description
Allow diagonal snap to traces	Select to automatically snap trace objects to a diagonal axis. Traces snap to a diagonal axis. This happens only when you modify trace objects and the routing angle is set to Diagonal or Any Angle. For example, when you stretch a trace segment and the trace is near a grid point, the trace automatically snaps to that grid point, creating a perfect diagonal. If the current routing angle is Orthogonal, the traces snap to 90-degree grid increments. Once an object snaps diagonally, click to place the corner. Objects snap when they are within one half the trace width of the diagonal axis.
Push segments away from obstacles	Select to let a trace "push away" from obstacles. **Tip** The distance that a trace segment is pushed away from an obstacle meets clearance rules. **Exception:** Trace segments do not push away from obstacles if you right-click and click **Ignore C during editing**, or if the DRC on page 405 mode for clearances is set to Prevent or Explain. **Figure 108. Pushing Segments Away From Obstacle** 
Smooth traces on complete	Select to run a smoothing operation on a trace when you end the trace.
Preserve arcs during smooth	Select to preserve arcs when you smooth a trace.
Layer pair area	Layer pair sets the first and second layer of the layer pair during routing. The layer pair increases your productivity when manually adding routes on multi-layer boards. When you add a via, the layer automatically switches to the other layer in the current layer pair. **Recommendation:** Set the layer pair to those layers upon which you expect to do most of your routing so you do not have to switch layers manually when you add a via. See also Setting Layer Rules
Plower area— Plowing moves unprotected traces, vias, and test points to make room for new traces and vias. Using plowing, you can continue routing without rerouting obstructing traces or stopping to move objects. See also Plowing Traces	

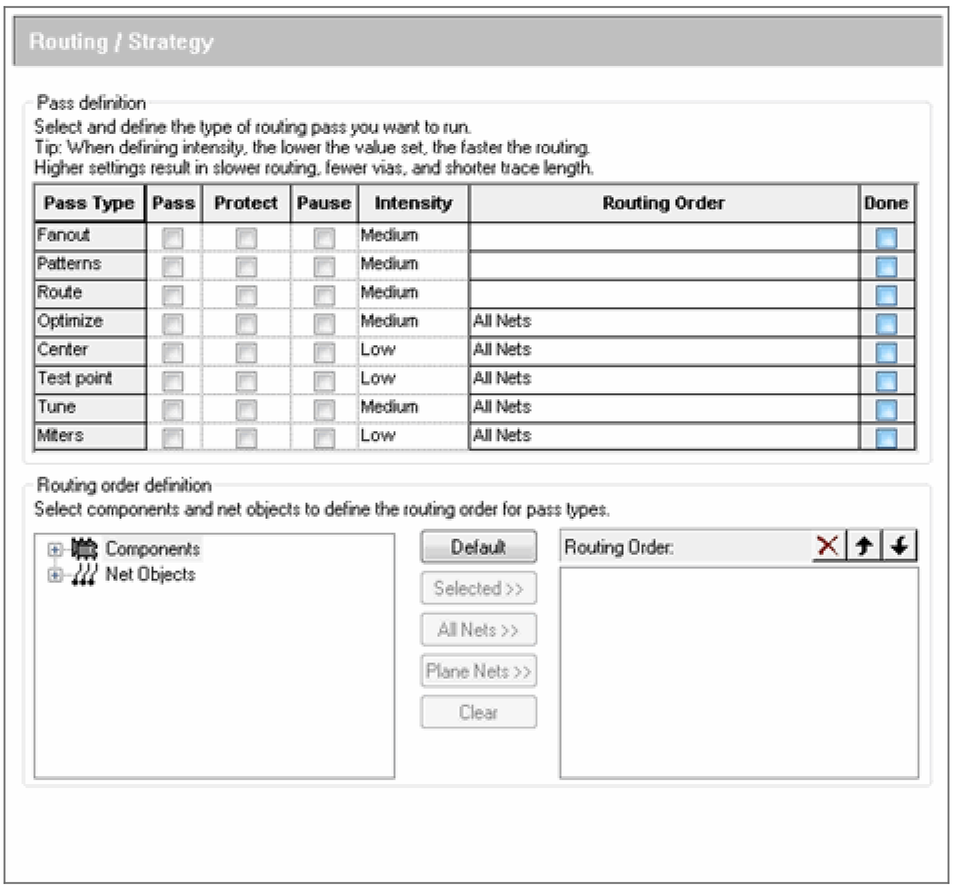
Field	Description
Turn on plower	Select to enable plowing.
Plower and pointer interaction	Choose when to plow obstacles: • To plow objects as you move the pointer, click **Plow with pointer**. • To plow obstacles immediately as the pointer passes them, click **Real-time plowing**. • To plow obstacles after the pointer defines a potential path and then reaches an open area, click **Guided plowing**. • To plow obstructing traces after you enter corners, click **Plow after corner click**.
Plow traces smoothly	Select to smooth obstructing traces as they are plowed, thereby enhancing the quality of trace patterns.
Allow pushing of trace behind plower	Select to push obstructing traces both ahead of, and behind the pointer as you route. Otherwise, traces are pushed ahead of the pointer.
Plower strength	In the Plower strength list, select the strength of the plower: Low, Medium, or High. Higher strength settings increase the depth of the plower so that more traces are plowed. Because more traces are plowed, additional time may be needed for each plow.

Options Dialog Box, Routing Category, Strategy Subcategory

To access:

- Choose the **Tools > Options** menu item > **Routing** category < **Strategy** subcategory
- Click the **Options** button > **Routing** category < **Strategy** subcategory
- Press the Ctrl + Enter keys > **Routing** category < **Strategy** subcategory

Define a strategy for autorouting a design. The strategy defines the sequential operations to perform during autorouting, including: passes that the autorouter should run and the order in which to autoroute components, nets, net classes, differential pairs, and matched length groups.



Objects

Field	Description
Pass definition area	

Field	Description
Pass Type column	Lists the types of passes that you can run on the design. The order of the passes is locked and proceeds from top to bottom.
Pass column	Select the check box of each pass type you want to run. You can run any combination of passes.
Protect column	Select the check box of those passes after which to protect the generated traces. This protects traces and glues vias that are completed during the pass.
Pause column	Select the check box of a pass if you want the autorouter to pause after a pass. To continue after the pause, on the Routing toolbar, you must click the Resume Autorouting button.
Intensity column	Select the appropriate intensity for the autorouting pass. Intensity determines the effort and time the router can spend on a pass. **Restriction:** You cannot set an intensity for the Center pass.
Routing Order column	Specify the order in which to autoroute components and nets (net classes, differential pairs, and matched length groups) for the selected pass. Once you select and activate the Routing Order cell for a pass, the Routing order definition area at the bottom of the dialog box becomes active. Use this area to create the order for autorouting that pass. **Tip** You can optimize routing by defining a routing order. For example, make a first quick trial pass at low intensity to see the resulting routing pattern of the autorouter. Look for components with many pins. Do any components have large numbers of pin nets that are unable to break out because of other connections that have already been routed in its breakout area? In this case, set the component at the top of the routing order and the nets of this component will be routed first by the autorouter ensuring a successful breakout and better overall autorouting completion.
Done column	A check mark is displayed in the Done column for each pass that completes. The check marks remain until you modify the autorouting strategy, or restart autorouting. This is a read-only column, you cannot manually mark a pass as complete.
Routing order definition area	
Components/Net Objects tree	Browse the tree to select individual nets or components to add the Routing Order list. Click **Selected** to add. When you add a component to the Routing Order list, all the nets attached to the component are routed at that stage of the routing order.

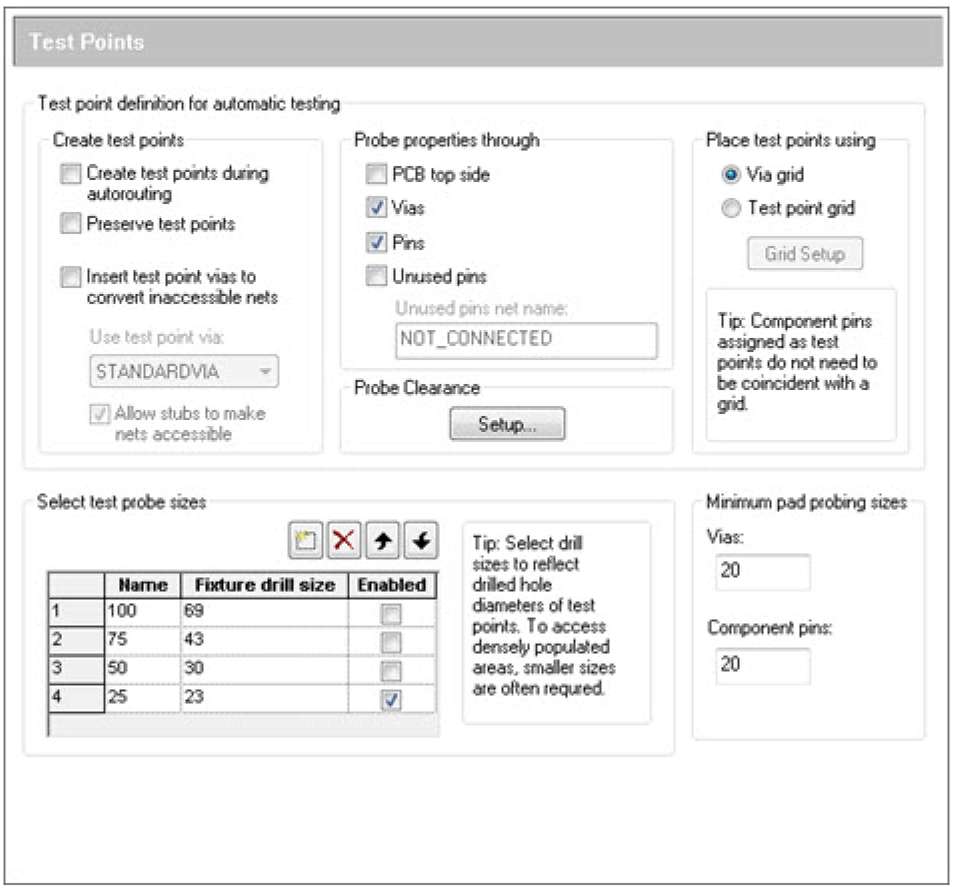
Field	Description
	Tip You can also add nets or components to the routing order list by using the following: • The Project Explorer—You add nets in exactly the same way as you add them using the Components/Net Objects tree. Select them in the Project Explorer, then click the Selected button in this dialog box. • The work area—Select nets or components in the design area, then click the Selected button in this dialog box. You may need to set the Selection Filter to select those design objects.
buttons	• Default—Click to clear the routing order list and add All Nets to the Routing Order list. • Selected—Click to add the object selected in the Components/Net Objects tree to the Routing Order list. You can also use this button to add objects selected in the design area or the Project Explorer. • All Nets—Click to add All Nets to the Routing Order list. • Plane Nets—Click to add all nets assigned to plane layers. • Clear—Click to remove all items from the Routing Order list.
Routing Order list	Use the pane buttons to remove or reorder the items in the Routing Order list. Items at the top of the list are routed first in the autorouting pass. Make sure to add All Nets to the bottom of every routing order list, otherwise the autorouter will only route the few items you may have added to the routing order.

Options Dialog Box, Test Points Category

To access:

- Click the **Options** button > Test Points category
- Choose the **Tools > Options** menu item > Test Points category
- Press the Ctrl + Enter keys > Test Points category

Use the Test Points category to define test point placement during autorouting, interactive routing, and trace editing. Test points allow probes of testing devices to make electrical contact with the board.



Objects

Field	Description
Creating test points area	
Create test points during autorouting	Select to insert test points automatically in the design during autorouting.

Field	Description
Preserve test points	Select to prevent existing vias assigned as test points from being reassigned, removed, deleted, shoved, or modified. **Exception:** This option does not preserve new test points on new traces until all routing passes are completed or if the routing process is stopped.
Insert test point vias to convert inaccessible nets	Select this check box to add test point vias for nets that are inaccessible for other test points. “Use test point via” dropdown list — If you choose to insert test point vias, also specify the type of via to insert. “Allow stubs to make nets accessible” check box — Select the check box to allow short trace stubs to the test point via when making nets accessible.
Probe properties through area	
PCB top side	Select to probe the board from the top side too. **Tip** Pins are always available for probing from the bottom side, whether or not PCB top side is selected.
Vias	Select to use vias as test points.
Pins	Select to use Pins as test points.
Unused pins	Select to assign test points for the unused pins in the design by selecting Unused pins and providing the net name for all unused pins in the design. **Unused pins net name**— When you specify to use unused pins as test points, you also specify the net name of the unused pins. This requires attaching a test point via, if it is an SMD pad, or assigning an unused component pin as a test point, if it is a through hole pin. For SMD pins, this is achieved by adding one single pin net to each unused SMD pin. The net is partially routed and terminated with a test point via.
Probe Clearance area	Click the **Setup** button to open the Test Points properties of the Design Properties. See also Properties Dialog Box, Test Points Tab
Place test points using area	Click an option to specify the grid type to use for test point placement: Via grid or Test point grid. **Grid Setup** — This button is only available if you select the Test point grid. Click to open the Grid properties of the Design Properties where you can set the Test Point grid values. See also Setting Grids
Select test probe sizes area	Use the buttons to add, delete, and move a probe size name. Click in a cell to change values.

Field	Description
	During the placement or assignment of test points, the values you specify in the Name cell are assigned as via or pin attributes. When you change these values, attributes for all vias and pins with these values are updated.
Field	Edit the name of the probe size to uniquely identify probe types.
Fixture drill size	Edit 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	Specify whether or not to use the associated nail during automated testing
Minimum pad probing sizes area	Minimum pad probing values for both Vias and Component pins ensure that there is sufficient pad area for probe contact. These values determine the minimum pad size to allow during test point assignment.

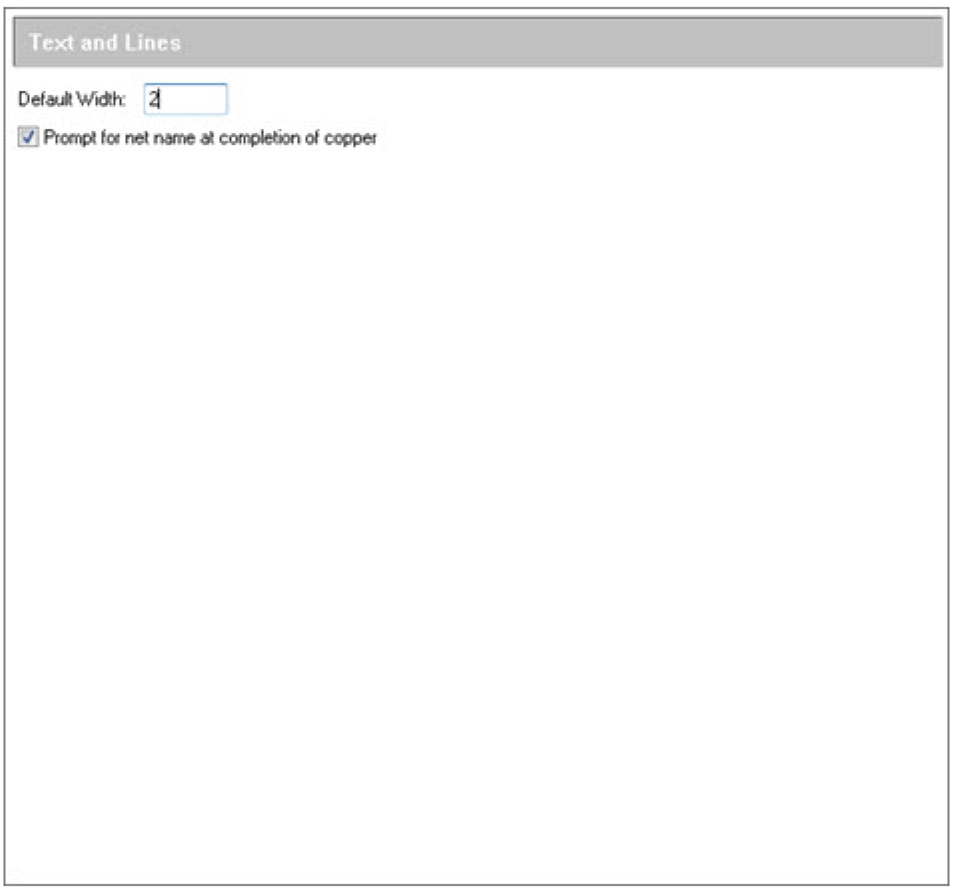
Options Dialog Box, Text and Lines Category

To access:

- Click the **Options** button > Text and Lines category
- Choose the **Tools > Options** menu item > Text and Lines category
- Press the Ctrl + Enter keys > Text and Lines category

Set the default width for border lines used in the drawing of copper planes.

Note:
You can change the border line width for individual copper planes by right-clicking and choosing the **Width > Set** popup menu item.



Objects

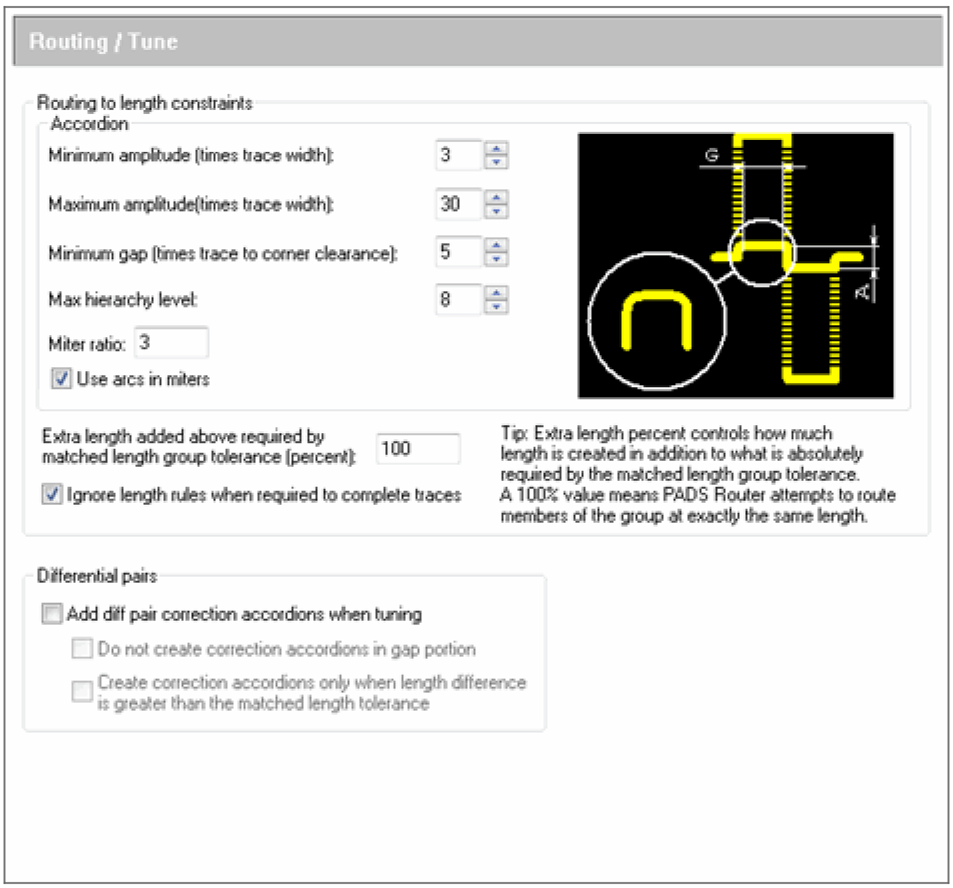
Field	Description
Default Width	Type a default width in the box. This becomes the default border width for all copper planes and shapes you draw using the Drafting Tools unless you specify otherwise.
Prompt for net name at completion of copper	When you select this check box, the Copper Plane Properties dialog box appears each time you finish drawing a copper plane. The Copper Plane Properties dialog box on page 475 provides a dropdown list for choosing a net to associate with a copper plane as well as other copper plane options.

Options Dialog Box, Routing Category, Tune Subcategory

To access:

- Choose the **Tools > Options** menu item > **Routing** category > **Tune** subcategory
- Click the **Options** button > **Routing** category > **Tune** subcategory
- Press the Ctrl + Enter keys > **Routing** category > **Tune** subcategory

Control the accordions used in length-controlled nets and differential pairs.



Objects

Field	Description
Routing to length constraints area	

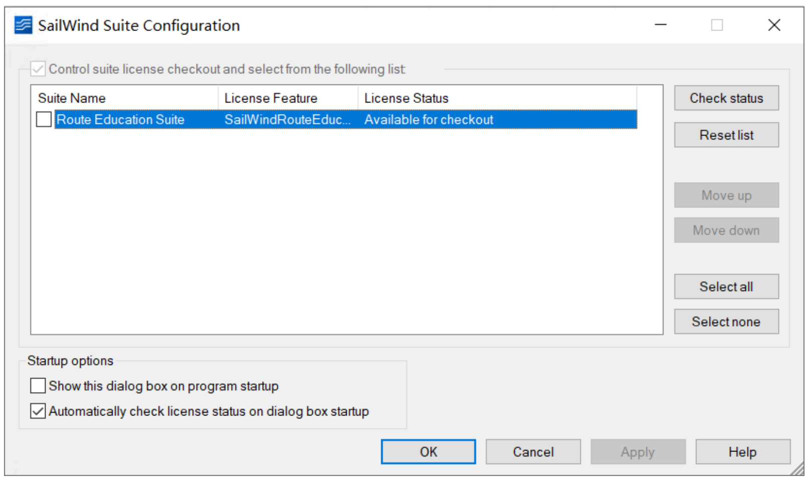
Field	Description
Accordion	• **Minimum amplitude (times trace width)**— Type a number to specify the minimum height for accordions. • **Maximum amplitude (times trace width)**— Type a number to specify the maximum height for accordions. • **Minimum gap (times trace to corner clearance)**— Type a number to specify the minimum distance between traces in the accordions. • **Max hierarchy level**— Type a number to specify how many levels of increased aggressive patterns can be used to complete the accordion. See also: "Maximum Hierarchy Level" in Tune Pass • **Miter ratio**— Type a number to specify the miter ratio for accordion corners. • **Use arcs in miters**— Select the check box to use an arc instead of a diagonal segment in the accordion.
Extra length added above required by matched length group tolerance (percent)	Type a number to determine 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	Select this check box if you need break length rules to complete traces.
Differential pairs area	
Add diff pair correction accordions when tuning	Select this check box to use an accordion to make differential pairs the same length. • **Do not create correction accordions in gap portion**— Select this check box if 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**— Select to ensure that correction accordions are not created when the differential pair net length difference is less than the tolerance of the matched length group.

SailWind Suite Configuration Dialog Box

To access:

- Choose the **Help > Installed Options** menu item > **Suite Configuration** button
- (Optional) Opens on program startup

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.
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.

Field	Description
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.

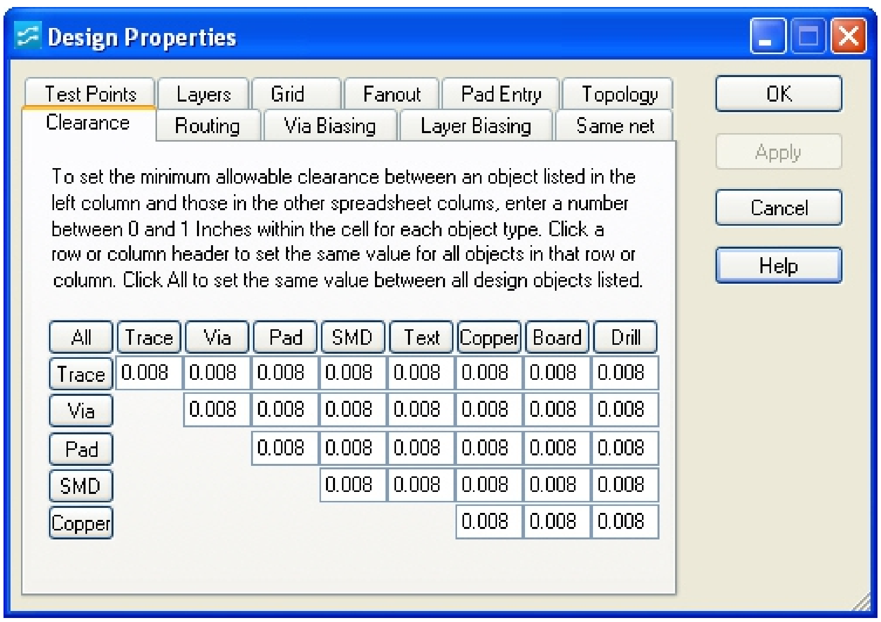
Properties Dialog Box, Clearance Tab

To access:

- Click the **Properties** button > **Clearance** tab
- Right-click > choose the **Properties** popup menu item > **Clearance** tab

View and modify minimum spacing clearances between design objects. If you select an object, you set the clearance properties for that object. If you select nothing, you set clearance properties as design defaults.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

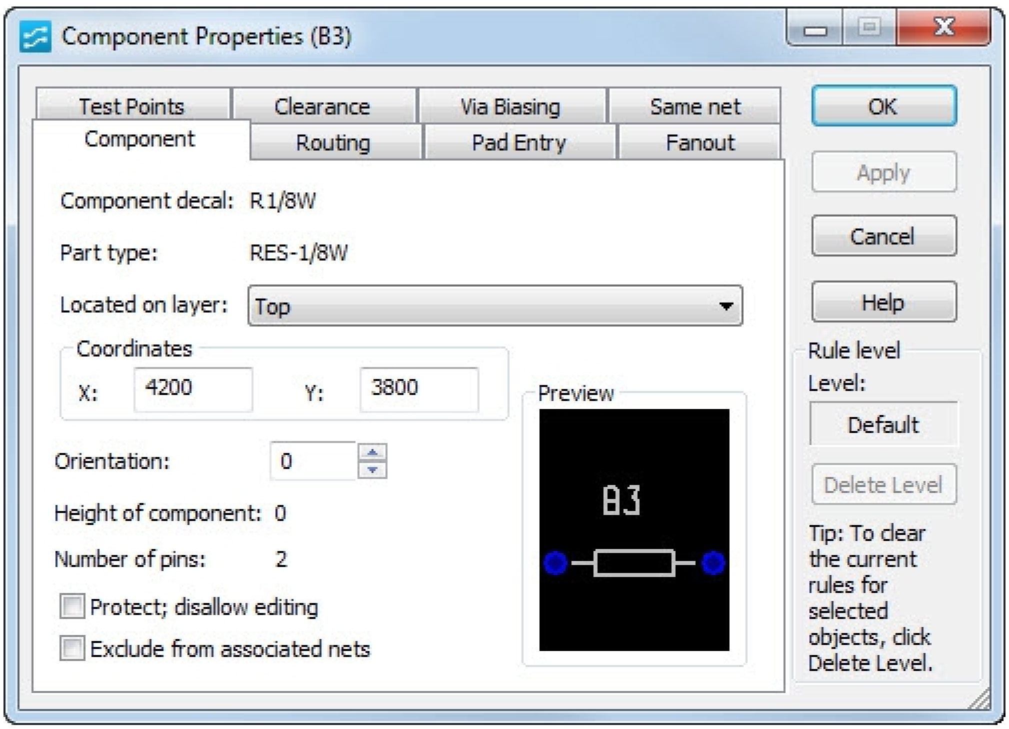
Field	Description
clearance matrix	Type a value in the box setting the permissible clearance between the object listed in the row heading and the object listed in the column heading. Alternative: To set the same value for an entire row or column, click on a column heading, row heading, or All. Type a value and click **OK** to apply the value.

Properties Dialog Box, Component Tab

To access: Select a component or pin > right-click > choose the **Properties** popup menu item > **Component** tab

View and modify the properties of a selected component or pin.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

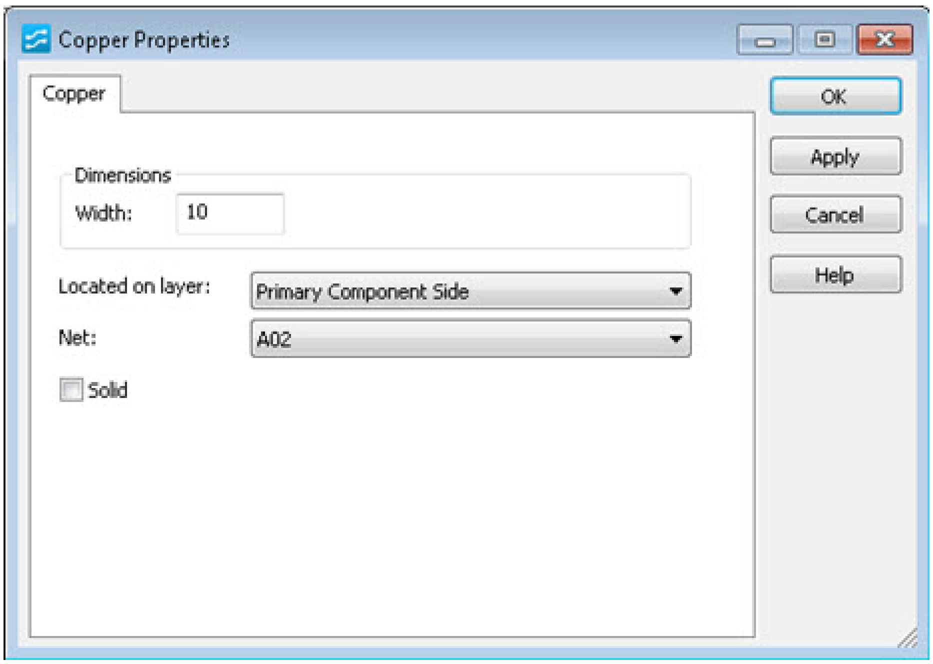
Field	Description
Component decal	Lists the decal associated with the selected component or pin.
Part type	Lists the part type associated with the selected component or pin.
Located on layer list	Specifies the layer where the component or pin is located.
X/Y	Specifies the X/Y coordinates of the component or pin.
Orientation	Specifies the rotation angle of the component or pin.
Height of component	Lists the height of the selected component or pin.

Field	Description
Number of pins	Lists the number of pins associated with the component.
Protect; disallow editing	Prohibits moving a component when selected.
Exclude from electrical nets	Specifies not to use the component or pin when creating electrical nets.

Properties Dialog Box, Copper

To access: Select a copper shape, right-click, choose the **Properties** popup menu item > **Copper** tab.

Set the layer location, line width, and the assigned net of a copper shape. Also specify whether or not the copper shape is filled solid or with a hatching pattern.



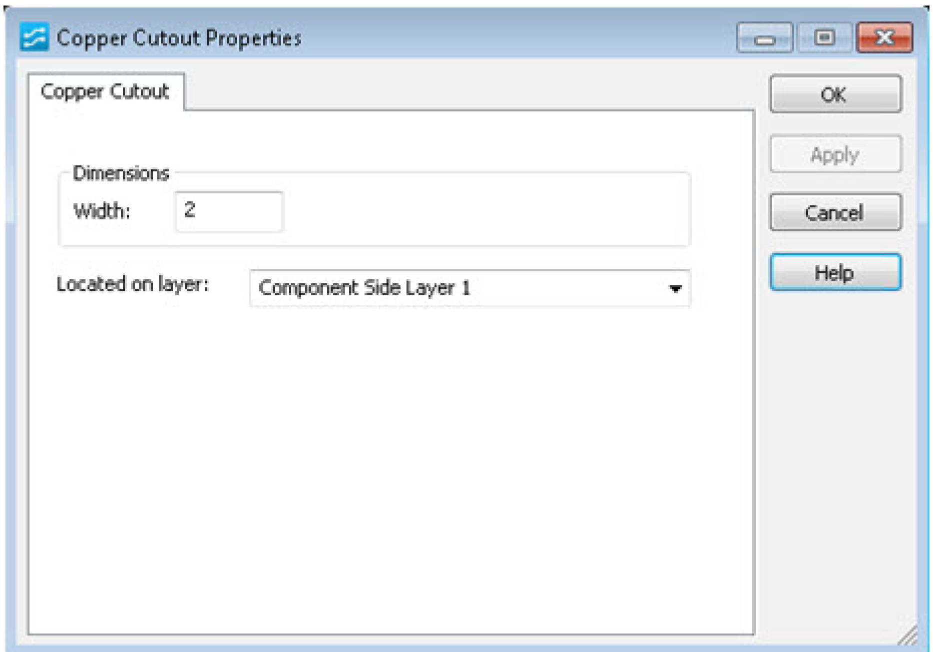
Objects

Field	Description
Width	Type a line width for the outline width of the copper shape and for the line width of the "hatching" pattern of the fill. If you do not enable the solid fill option, the shape fills with lines placed on the X and Y axes on the copper grid. In some cases, specifying a line width that is too wide would result in overlapping areas of the shape (known as a self-intersecting polygon). SailWind Router prevents you from specifying a width that results in such a self-intersecting shape.
Located on layer	Select the layer where you want the copper shape to be placed.
Net	Displays the assigned net for the copper shape.
Solid check box	Displays the copper shape as solid without regard to the line width or grid setting. When cleared, the copper fills according to the settings in the Options dialog box > **Global** category > "General subcategory" on page 441 whenever the Pour Outline mode is disabled.

Properties Dialog Box, Copper Cut Out Tab

To access: Select a copper plane cut out, right-click, choose the **Properties** popup menu item > **Copper Cutout** tab.

View information about the layer location and the dimensions of a copper cut out.



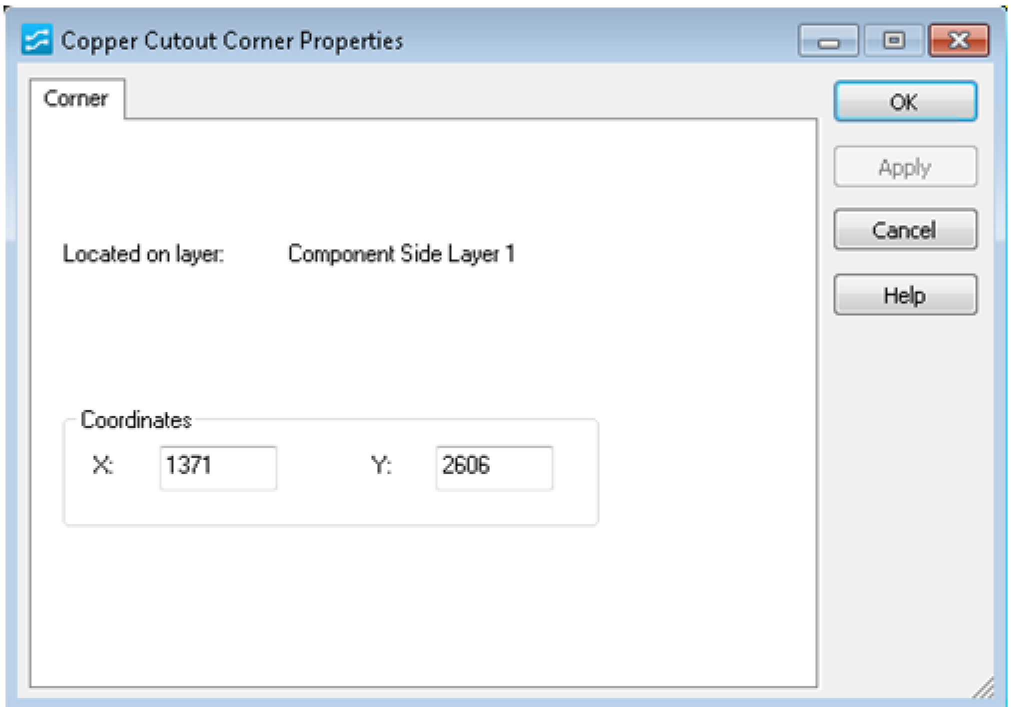
Objects

Field	Description
Width	Type a line width for the outline of the copper cut out. The line width applies to all lines used in the shape. In some cases, specifying a line width that is too wide would result in overlapping areas of the shape (known as a *self-intersecting polygon*). SailWind Router prevents you from specifying a width that results in such a self-intersecting shape.
Located on layer	Specifies the layer on which the copper cut out is located.

Properties Dialog Box, Copper Cut Out Corner

To access: Select a copper cut out corner > right-click > choose the **Properties** popup menu item > **Corner** tab

Set the coordinates of a copper cut out corner and view its other properties.



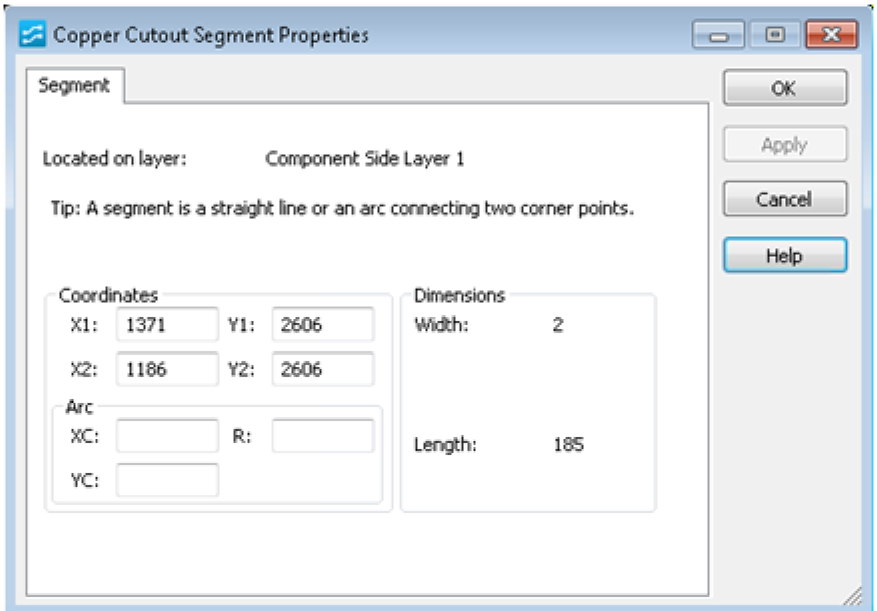
Objects

Field	Description
Located on Layer	Displays the layer on which the cut out corner is located.
X/Y coordinates	Specifies the X/Y coordinates of the corner. You can edit the coordinates by typing directly in the boxes.

Properties Dialog Box, Copper Cut Out Segment

To access: Select a trace segment of a copper plane cut out > right-click > choose the **Properties** popup menu item > **Segment** tab

Set the coordinates of the copper cut out segment.



Objects

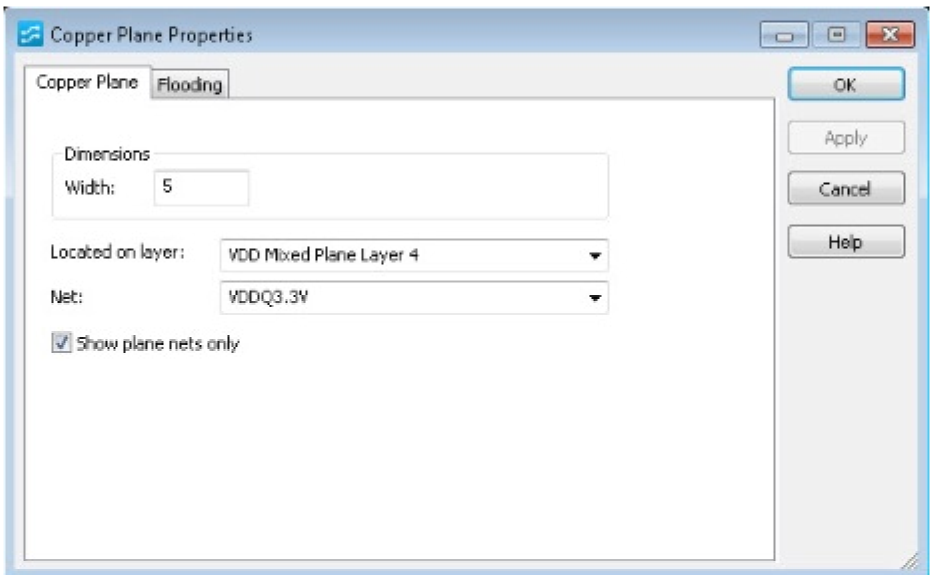
Field	Description
Located on layer list	Displays layer on which the copper plane segment is located.
Coordinates	Displays the location of the copper plane segment: X1, Y1, X2, Y2, arc center X (XC), arc center Y (YC), arc radius (R). You can edit these values by typing directly in the boxes. SailWind Router applies the change when you click **OK**. **Tip** Change an arc to a straight line by typing 0 in the “R” box. Change a straight line to an arc by typing a non-zero value in the Arc radius box.
Dimensions	Displays the line width and length of the copper plane segment. You can edit the width by typing directly in the box. SailWind Router applies the change when you click **OK**.

Properties Dialog Box, Copper Plane Tab

To access: Select a copper plane, right-click, choose the **Properties** popup menu item > **Copper Plane** tab.

View information about the layer location and the assigned net of copper planes.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

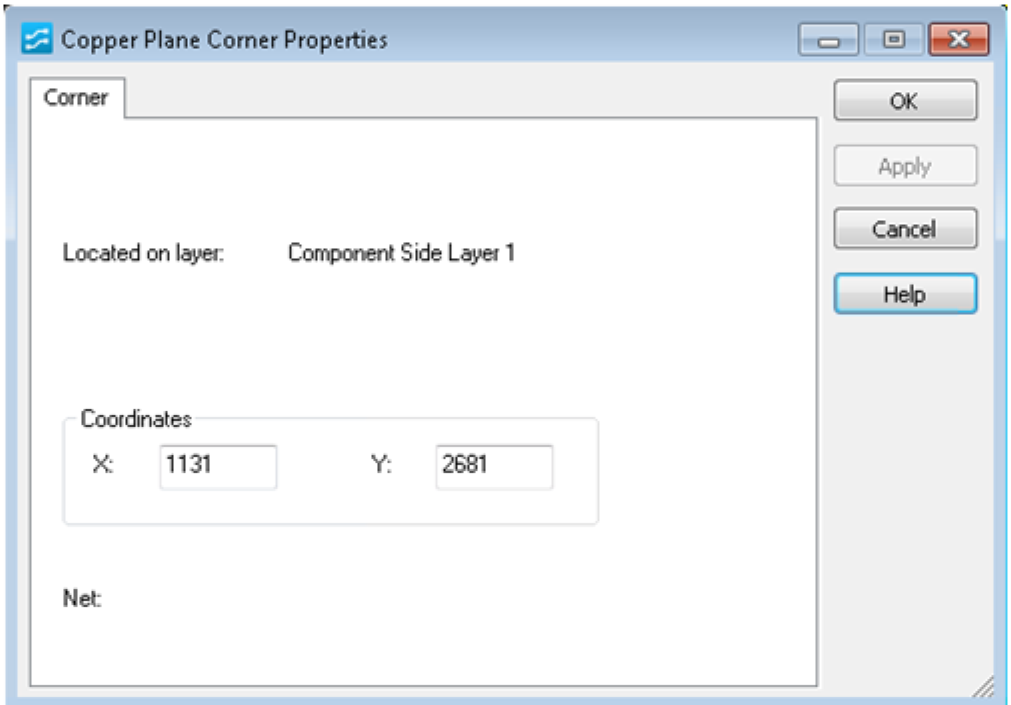
Field	Description
Width	Type a line width for the outline of the copper shape. The line width applies to all lines used in the shape, including borders and hatching. In some cases, specifying a line width that is too wide would result in overlapping areas of the shape (known as a *self-intersecting polygon*). SailWind Router prevents you from specifying a width that results in such a self-intersecting shape.
Located on layer	Specifies the layer on which the copper plane is located.
Net	Specifies the assigned net for copper planes. Choose a net from the dropdown list.
Show plane nets only	Select this check box to limit the display of nets in the Net list to only plane nets.

Field	Description
	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.

Properties Dialog Box, Copper/Copper Plane Corner

To access: Select a copper or copper plane corner > right-click > choose the **Properties** popup menu item > **Corner** tab

Edit the coordinates of a copper/copper plane corner and view its other properties.



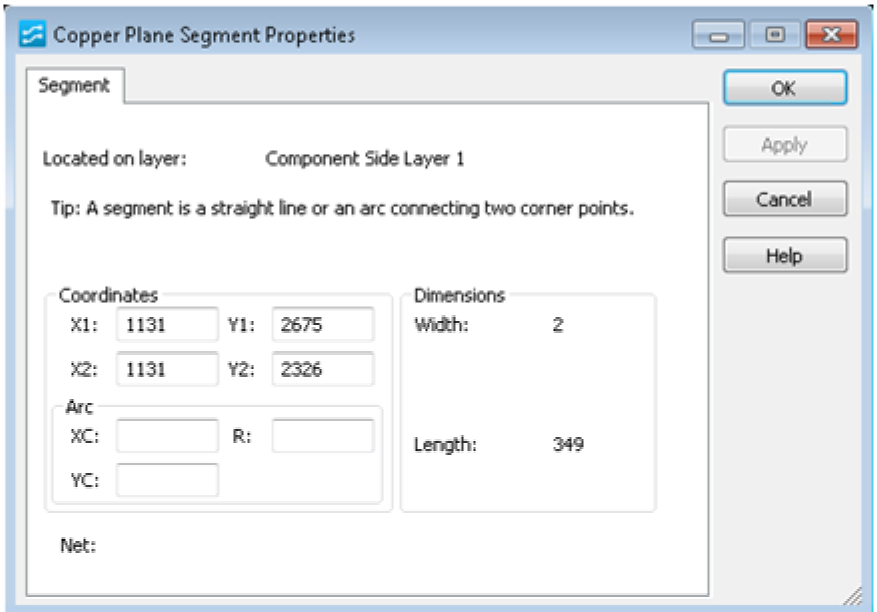
Objects

Field	Description
Located on Layer	Displays the layer on which the copper or copper plane corner is located.
X/Y coordinates	Specifies the X/Y coordinates of the corner. You can edit the coordinates by typing directly in the boxes.
Net	Displays the net associated with the corner.

Properties Dialog Box, Copper/Copper Plane Segment

To access: Select a segment of a copper or copper plane > right-click > choose the **Properties** popup menu item > **Segment** tab

Edit the coordinates of the copper/copper plane segment.



Objects

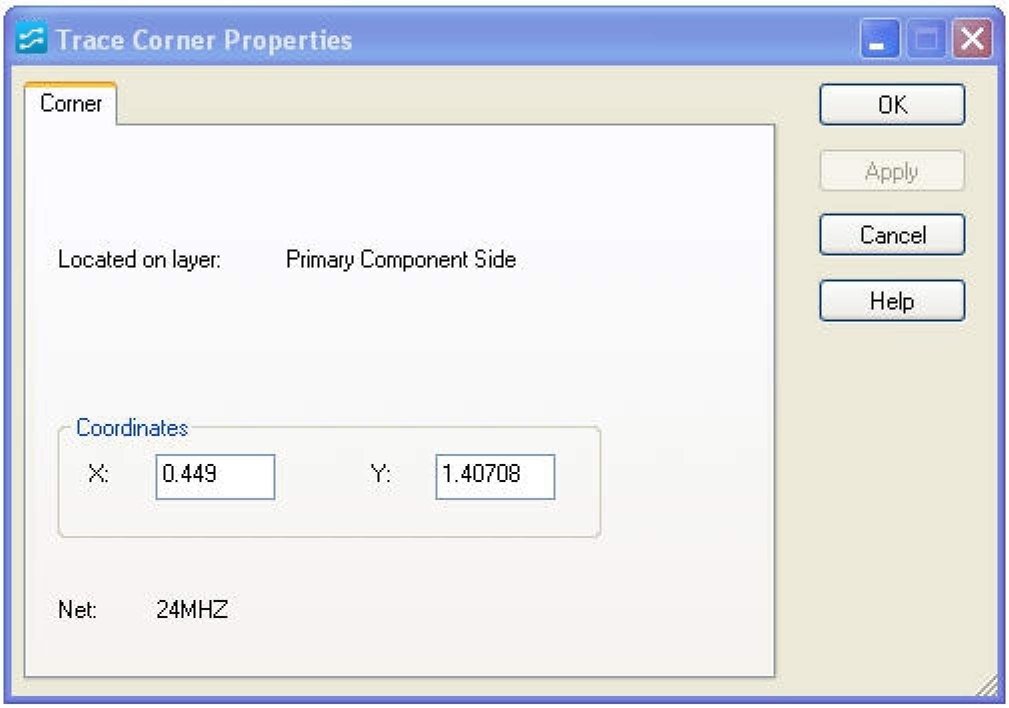
Field	Description
Located on layer list	Displays the layer on which the copper or copper plane segment is located.
Coordinates	Displays the location of the copper/copper plane segment: X1, Y1, X2, Y2, arc center X (XC), arc center Y (YC), arc radius (R). You can edit these values by typing directly in the boxes. **Tip** Change an arc to a straight line by typing 0 in the “R” box. Change a straight line to an arc by typing a non-zero value in the box.
Dimensions	Displays the line width and length of the copper or copper plane segment. You can edit the width by typing directly in the box.
Net	Displays the net name associated with the copper or copper plane segment.

Properties Dialog Box, Corner Tab

To access: Select a trace > right-click > choose the **Properties** popup menu item > **Corner** tab

Set the coordinates of a trace corner, and see which layer the corner is located on. Trace corners include T-junctions and trace end points.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Located on Layer	Lists the layer on which the trace corner is located.
X/Y coordinates	Specifies the X/Y coordinates of the corner.
Net	Specifies the net associated with this corner.

Properties Dialog Box, Drill Tab (Pin)

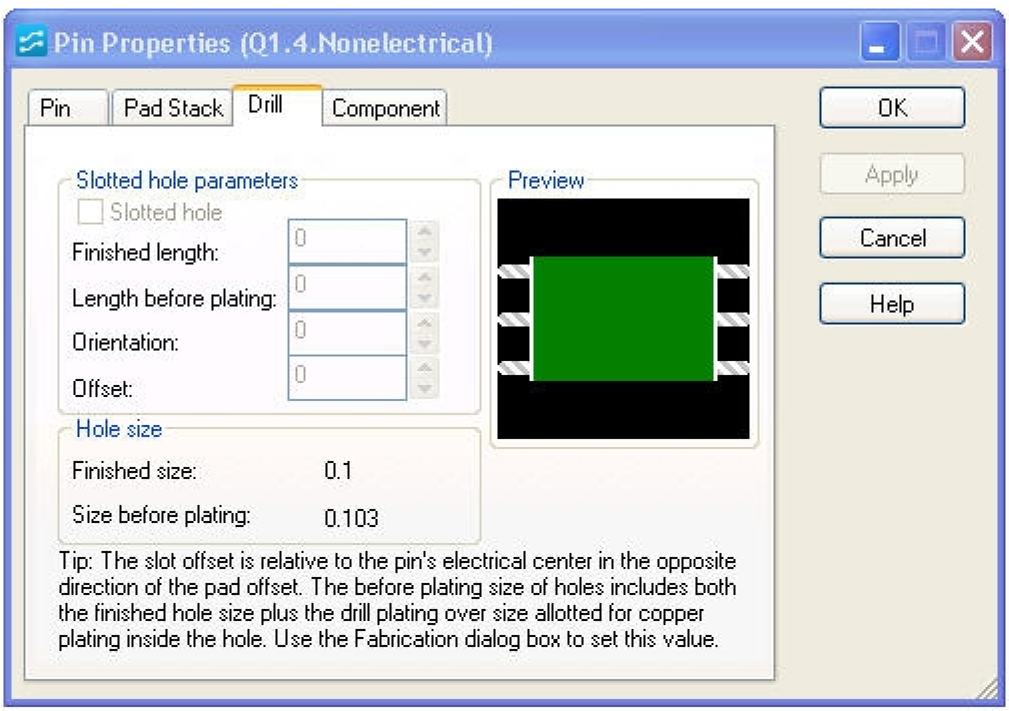
To access: Select a pin > right-click > choose the **Properties** popup menu item > **Drill** tab

View information about the pin pad stack, such as the finished drill hole size and the raw drill hole size. For slotted holes, you can view the finished length, length before plating, orientation, and offset.

Restriction:
This tab is for information only; these properties cannot be modified in SailWind Router. Use the Pad Stacks Properties dialog box in SailWind Layout to modify the properties displayed in this tab.

Tip
For plated holes, drill size includes global drill oversize value.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Slotted hole	Enables a slotted hole for the selected pin. See also: “Slotted Holes” in the *SailWind Layout Guide*.
Finished length	Sets the length of the slotted hole. This option becomes available when you select Slotted hole.
Length before plating	Sets the length of the slotted hole before plating. This option becomes available when you select Slotted hole.
Orientation	Sets the orientation of the slotted hole. This option becomes available when you select Slotted hole. **Tip** 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. This option becomes available when you select Slotted hole. The maximum amount of offset you can assign is one half of the slot's length. If you exceed this limit, the slotted hole display is suppressed in the Preview dialog box in CAM. **Tip** A custom thermal or antipad for a slotted hole has the same offset as the slot.
Hole size	Shows finished hole size and the size of the hole before plating. **Restriction:** You cannot edit these fields.
preview	Shows what the drill hole looks like.

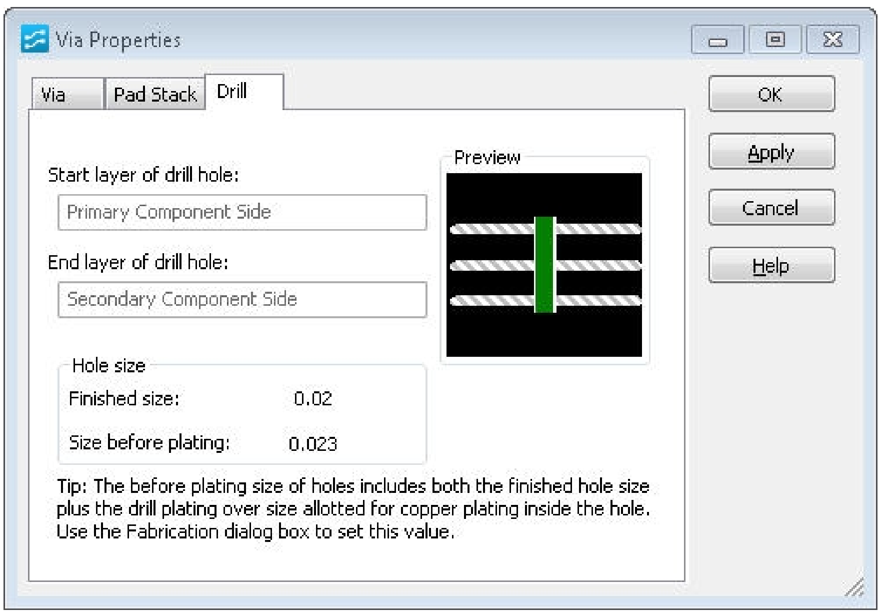
Properties Dialog Box, Drill Tab (Via)

To access: Select a via > right-click > choose the **Properties** popup menu item > **Drill** tab.

View the first and last layer the via connects, the finished drill hole size, and the raw drill hole size. The **Drill** tab is for information only; you cannot modify the properties in SailWind Router. Use the Pad Stacks Properties dialog box in SailWind Layout to modify the properties displayed in the **Drill** tab.

Tip
For plated holes, drill size includes global drill oversize value.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Start layer of drill hole	Shows the layer on which the drill hole starts. **Restriction:** You cannot edit this field.
End layer of drill hole	Shows the layer on which the drill hole ends. **Restriction:** You cannot edit this field.

Field	Description
Hole size	Shows finished hole size and the size of the hole before plating. **Restriction:** You cannot edit these fields.
preview	Shows what the drill hole looks like.

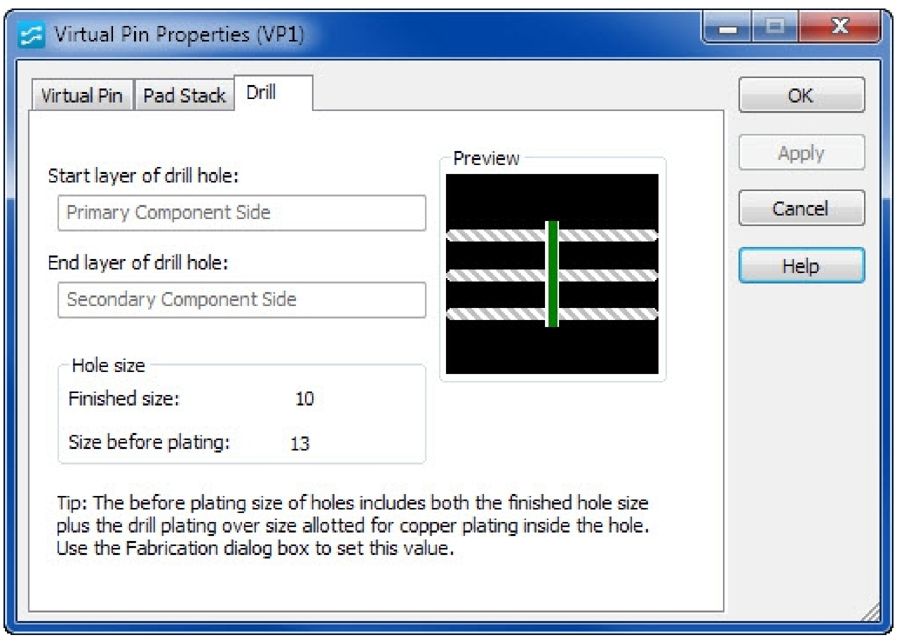
Properties Dialog Box, Drill Tab (Virtual Pin)

To access: Select a virtual pin > right-click > choose the **Properties** popup menu item > **Drill** tab.

View the first and last layer that the virtual pin connects, the finished drill hole size, and the raw drill hole size. This tab is for information only; you cannot modify these properties in SailWind Router. Use the Pad Stacks Properties dialog box in SailWind Layout to modify the properties displayed in this tab.

Tip
For plated holes, drill size includes global drill oversize value.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Start layer of drill hole	Shows the layer on which the drill hole starts. **Restriction:** You cannot edit this field.
End layer of drill hole	Shows the layer on which the drill hole ends. **Restriction:** You cannot edit this field.

Field	Description
Hole size	Shows finished hole size and the size of the hole before plating. **Restriction:** You cannot edit these fields.
preview	Shows what the drill hole looks like.

Properties Dialog Box, Fanout Tab

To access: Right-click > choose the **Properties** popup menu item > **Fanout** tab

Use fanouts to make the routing easier and to ensure connections are made. The program tries to place fanout vias according to the alignment settings on this tab. Fanouts are most useful for some complex SMDs, like BGAs.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.

Description

Figure 109. Fanout tab

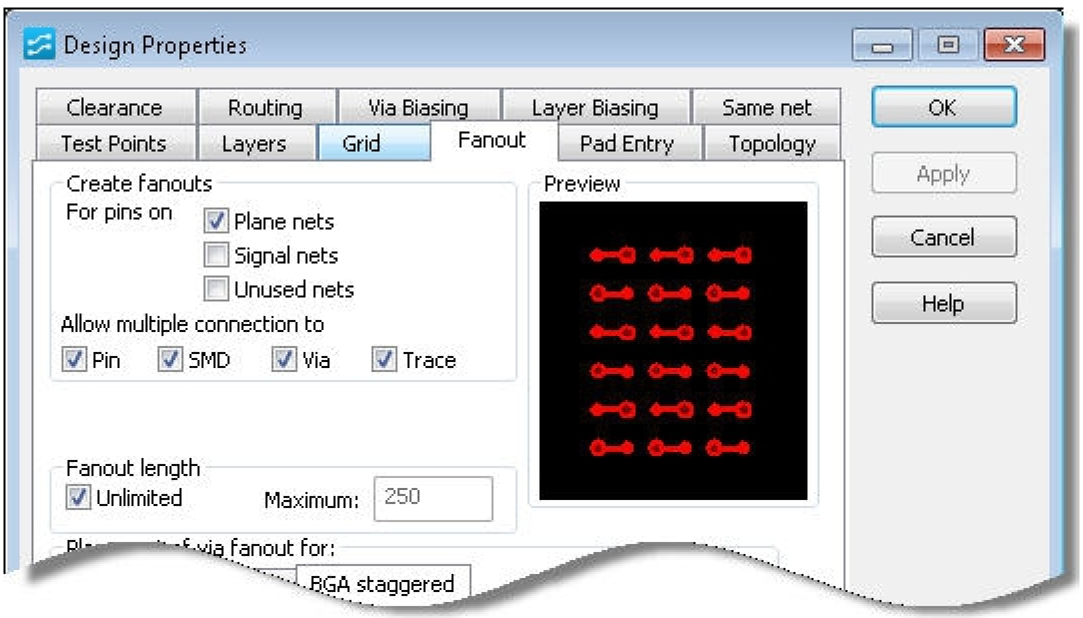


Figure 110. Fanout tab - BGA tab

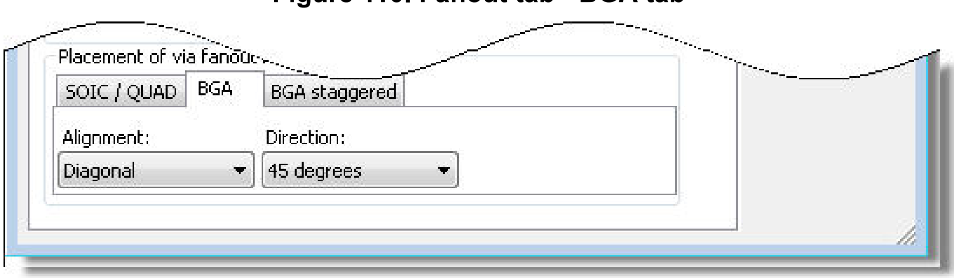


Figure 111. Fanout Tab - BGA staggered tab

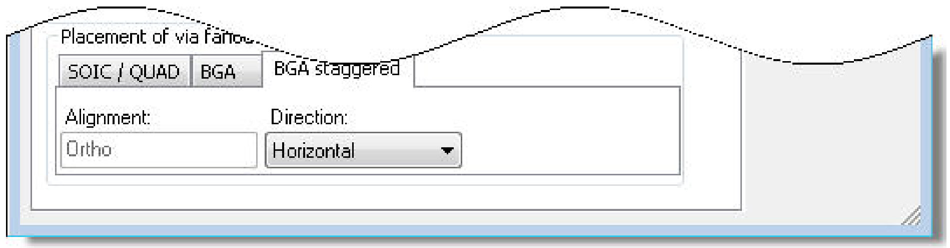
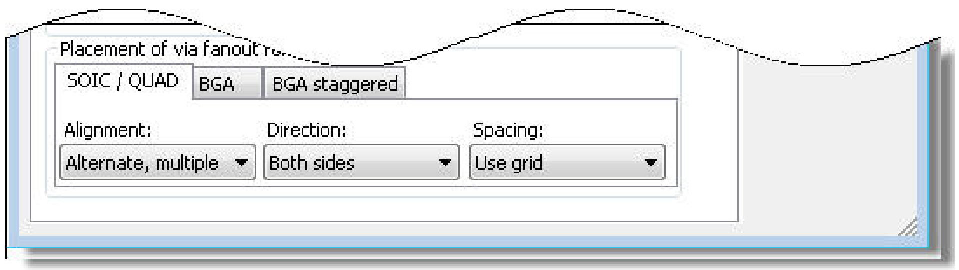


Figure 112. Fanout Tab - SOIC / QUAD tab



Objects

Table 89. Fanout Tab Contents

Field	Description
For pins on	Specifies the type of fanouts to create. For pins on: Plane nets, Signal nets, or Unused nets.
Allow multiple connection to	Specifies where to allow multiple connections: Pin, SMD, Via, and Trace.
Unlimited	Enables unlimited fanout length. **Tip** Clear the check box to specify a maximum length.
Maximum	Specifies the maximum fanout length allowed. **Restriction:** This box is available only when the “Unlimited” check box is clear.
BGA tab	
Alignment list	Specifies the via placement alignment: Quadrant, Diagonal, X-pattern.
Direction list	Specifies the via placement direction. The choices change depending on what you selected in the Alignment list:

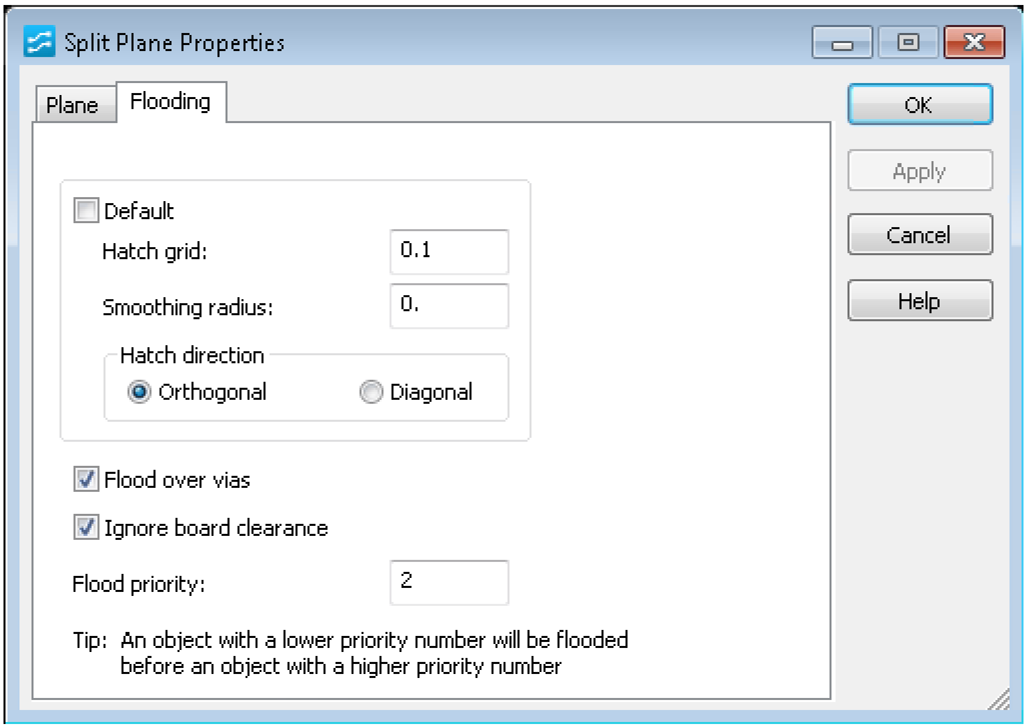
Table 89. Fanout Tab Contents (continued)

Field	Description
	• Quadrant— No choices. • Diagonal— 45 degrees, 135 degrees, 225 degrees, 315 degrees. • X-pattern— Clockwise, Counterclockwise.
BGA staggered tab	
Alignment	Displays the via placement alignment: Ortho
Direction list	Specifies the via placement direction: Horizontal, Vertical.
SOIC / QUAD tab	
Alignment list	Specifies the via placement alignment: Aligned, multiple rows; Alternate, multiple rows; Aligned, single row; Alternate, single row.
Direction list	Specifies the via placement direction: Inside, Outside, Both sides.
Spacing list	Specifies the via placement spacing: Use grid, One trace, Two traces.

Properties Dialog Box, Flooding tab

To access: Select a pour or a split plane > right-click > choose the **Properties** popup menu item > **Flooding** tab.

Use the **Flooding** tab to change the flooding behavior or appearance of the selected copper plane or shape. Changes you make appear when you enable the dynamic copper healing feature (**Options > Flooding** tab).



Objects

Field	Description
Default	Select this check box to use the default hatch grid, smoothing radius, and hatch direction settings on page 447 in the selected copper area. Otherwise, specify these settings if you clear the check box.
Hatch grid	Type a value for the hatch grid spacing. The value you type here supercedes the setting on the Grid tab on page 491 of the Design Properties dialog box.
Smoothing radius	Controls the radius of copper plane corners according to the following: <corner radius> = <shape outline width> * <smoothing radius> The smoothing radius parameter can be set between 0 and 5.
Hatch direction	Specifies the hatch orientation in the flooded area.

Field	Description
	• Orthogonal — Hatching consists of vertical and horizontal lines • Diagonal — Hatching consists of diagonal lines
Flood over vias	Allows copper flooding to extend over via thermals without affecting global settings in the Options dialog box.
Ignore board clearance	Extends flooding beyond the board boundaries such as edges and cutouts.
Flood priority	Establishes a priority order for flooding planes in the design. Type any whole number. A plane or object with a lower priority number floods before an object with a higher number. **Important:** SailWind Router does not route to overlapping portions of planes and pours that have the same priority setting.

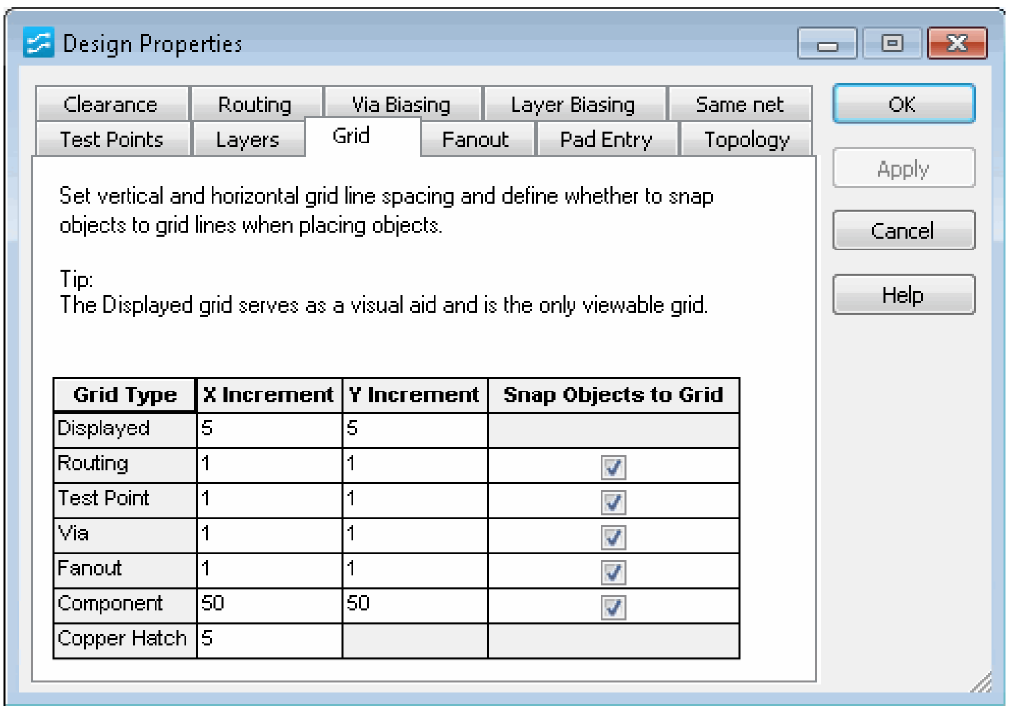
Properties Dialog Box, Grid Tab

To access: With nothing selected, perform one of the following actions:

- Double-click in the design area
- Right-click in the design area > choose the **Properties** popup menu item > **Grid** tab
- Click the **Properties** button > **Grid** tab

View and modify all grid settings in your design.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Displayed	Sets the visual dot grid. This grid can also be set using the GD shortcut command.
Routing	Sets the routing grid. This grid can also be set using the GR shortcut command.

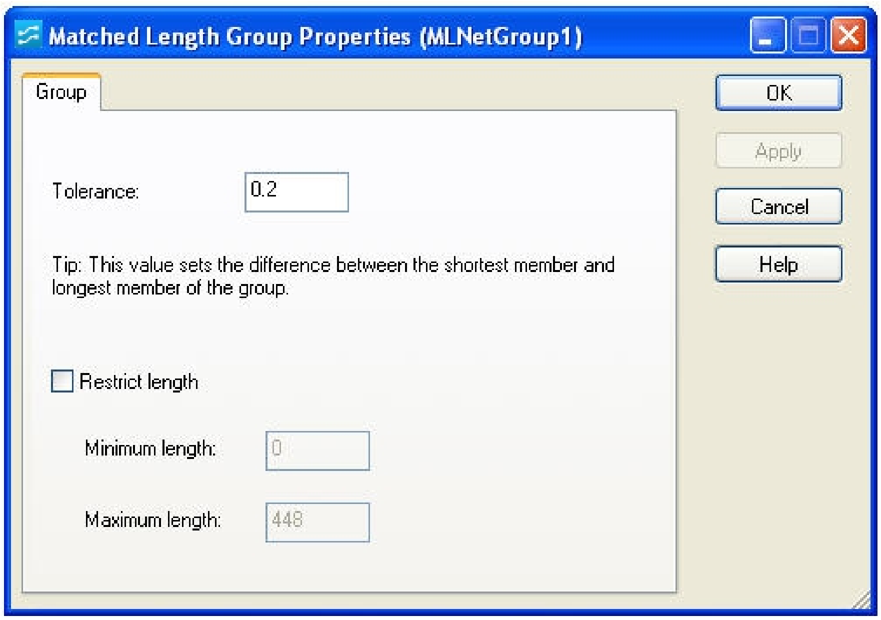
Field	Description
	Snap Objects to Grid— You can toggle this check box using the GS shortcut command. The GGS, GGM, and GGH commands require this check box to be selected for full functionality.
Test Point	Sets the test point grid. You can also set this grid using the GT shortcut command.
Via	Sets the via grid. You can also set this grid using the GV shortcut command.
Fanout	Sets the grid for fanout vias. You can also set this grid using the GF shortcut command.
Component	Sets the component placement grid. You can also set this grid using the GC shortcut command.
Copper Hatch	Specifies the distance between hatch lines of copper planes. The hatch grid, when combined with the line width, creates varying degrees of hatch through to a solid pattern.

Properties Dialog Box, Group Tab

To access: Select a group > right-click > choose the **Properties** popup menu item > **Group** tab

Set the tolerance between the members of the group and length restrictions for the group.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Tolerance	Specifies the difference between the shortest member and longest member of the group.
Restrict length	Restricts the length of the group to the range you specify in the “Minimum length” and “Maximum length” boxes. **Tip** Unlike SailWind Layout, this program counts overlapping trace segments in the trace length calculation.
Minimum length	Specifies the minimum length allowed for the group. **Restriction:** This box is available only when “Restrict length” check box is selected.

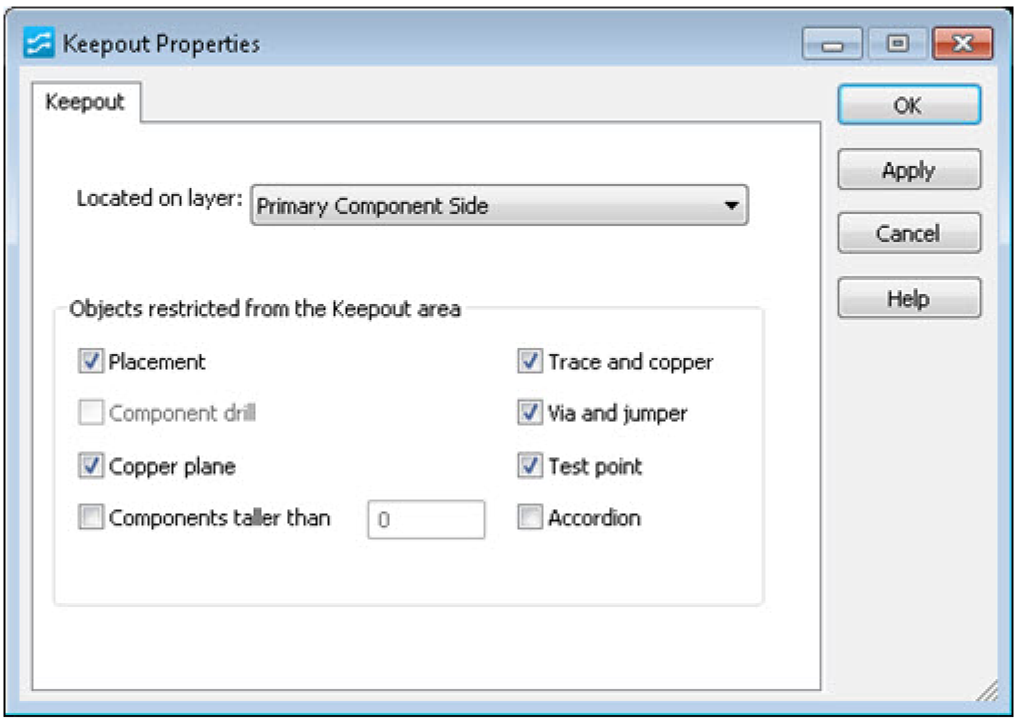
Field	Description
Maximum length	Specifies the maximum length allowed for the group. **Restriction:** This box is available only when “Restrict length” check box is selected.

Properties Dialog Box, Keepout Tab

To access: Select a keepout > right-click > choose the **Properties** popup menu item > **Keepout** tab

Set the layer on which a keepout is located and specify the objects to restrict from the keepout area.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see "Setting Design (Global and Default) Properties" on page 154 and "Setting Object Properties" on page 169.



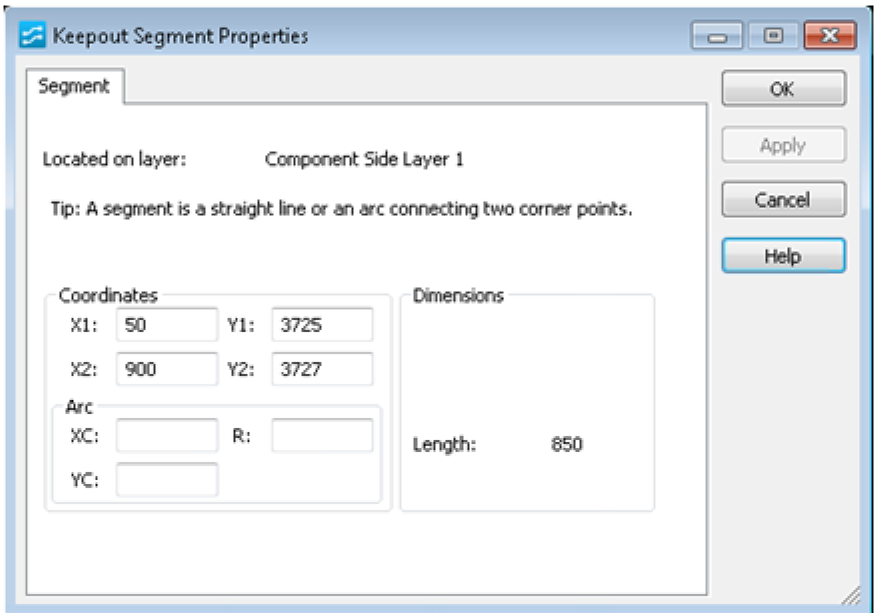
Objects

Field	Description
Located on layer	Select a layer from the dropdown list where you want the keepout to be located.
Objects restricted from the keepout area	Specify the objects that you want to restrict from the keepout area.

Properties Dialog Box, Keepout Segment

To access: Select a segment of a copper or copper plane > right-click > choose the **Properties** popup menu item > **Segment** tab

Edit the coordinates of the Keepout segment.



Objects

Field	Description
Located on layer list	Displays the layer on which the keepout segment is located.
Coordinates	Displays the location of the keepout segment: X1, Y1, X2, Y2, arc center X (XC), arc center Y (YC), arc radius (R). You can edit these values by typing directly in the boxes. **Tip** Change an arc to a straight line by typing 0 in the “R” box. Change a straight line to an arc by typing a non-zero value in the box.
Dimensions	Displays the length of the keepout segment.

Properties Dialog Box, Layer Biasing Tab

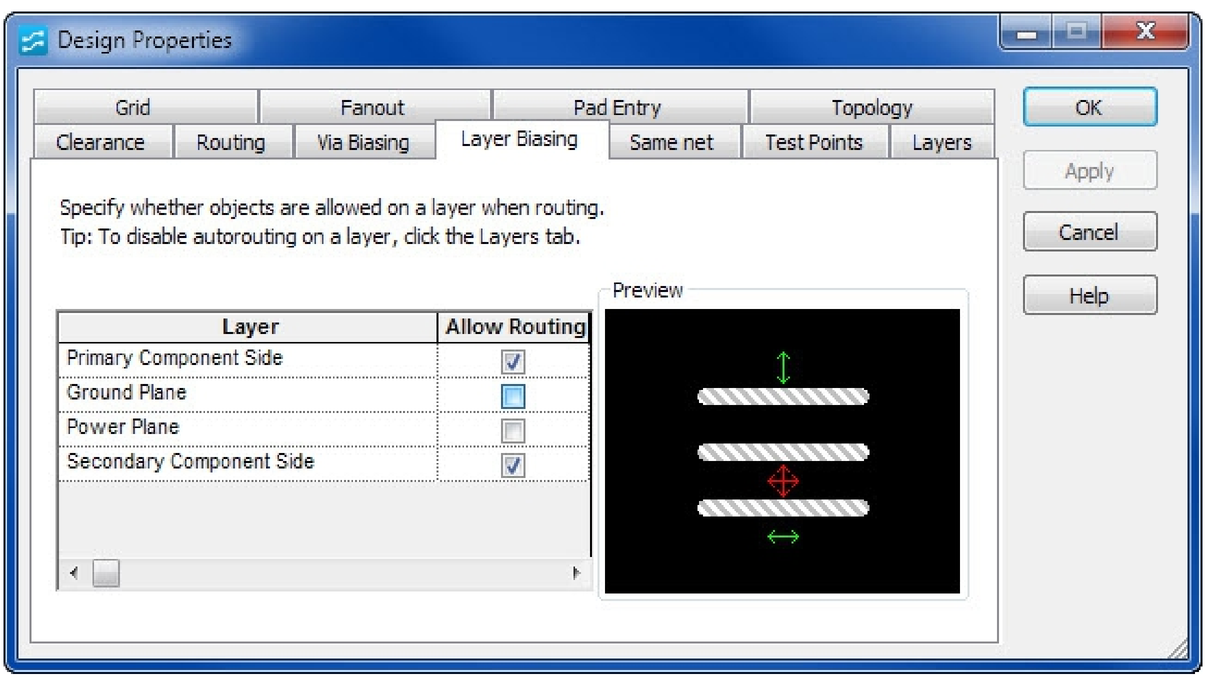
To access:

- Click the **Properties** button > **Layer Biasing** tab
- Right-click > choose the **Properties** popup menu item > **Layer Biasing** tab

Enable or prevent routing of net objects on a layer. Layer biasing provides a finer level of control over route location than the settings in the **Layers** tab, and restricts net objects to certain layers for routing. For example, you could use layer biasing to force SailWind Router to route power and ground nets on outer layers, and other nets on internal layers.

Restriction:
If a net object is selected, you specify the available routing layers for that object only. If a layer is selected, you can specify non-default availability for routing for any layer. If nothing is selected, you set the layer biasing properties as design defaults.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

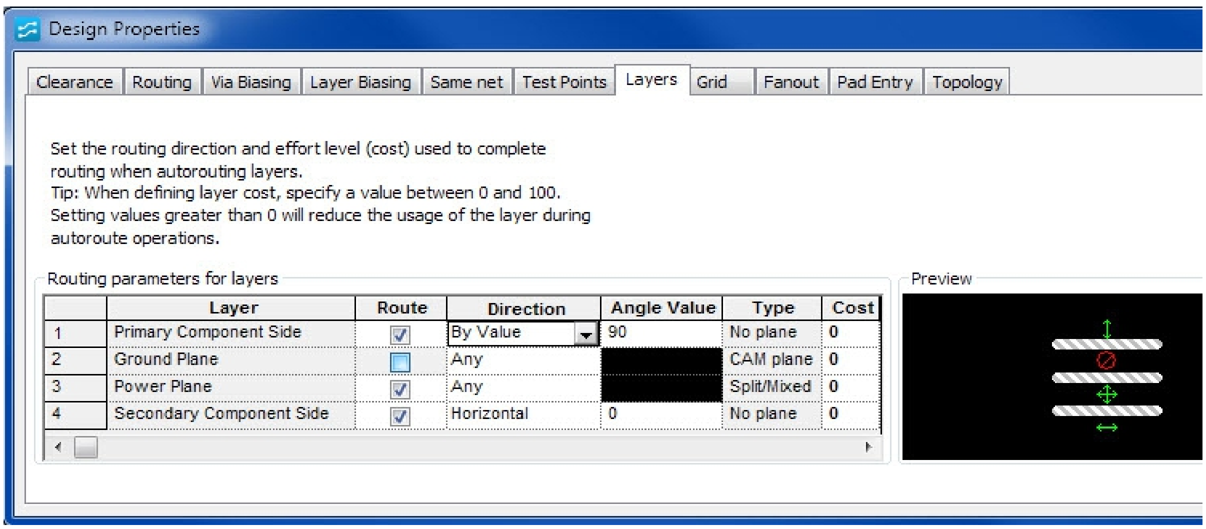
Field	Description
Layer	Lists the layers of the design.
Allow Routing	Enables the layer for routing. **Note:** If you select a net object, this check box enables routing on the layer for the selected object. Clear the check box to disable the layer for routing.

Properties Dialog Box, Layers Tab

To access: Select a layer > right-click > Choose the **Properties** popup menu item > **Layers** tab

Set routing parameters for a specific layer.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Layer	Lists the layers of the design.
Route	Enables the layer for routing. Clear the check box to disable the layer for routing.
Direction	Specifies the routing direction: Horizontal, Vertical, Diagonal (45), Diagonal (135), Any, or By Value. **Tip** If you select By Value, you must set the desired routing angle in the Angle Value column.
Angle Value	Specifies the desired routing angle. **Restriction:** This box is available only when you select “By Value” in the Direction column.
Type	Specifies the type of layer.

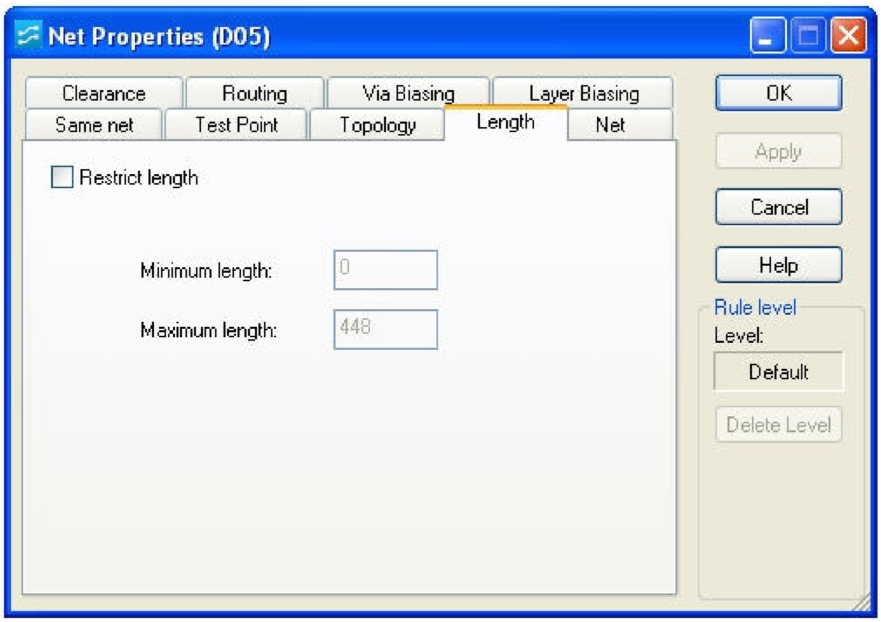
Field	Description
	Restriction: This value is not editable; it is set in SailWind layout.
Cost	Specifies the level of usage of the layer during routing. A higher number specifies *less* use of the layer.

Properties Dialog Box, Length Tab

To access: Select a net or a pin pair > right-click > choose the **Properties** popup menu item > **Length** tab

Restrict net or pin pair length.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

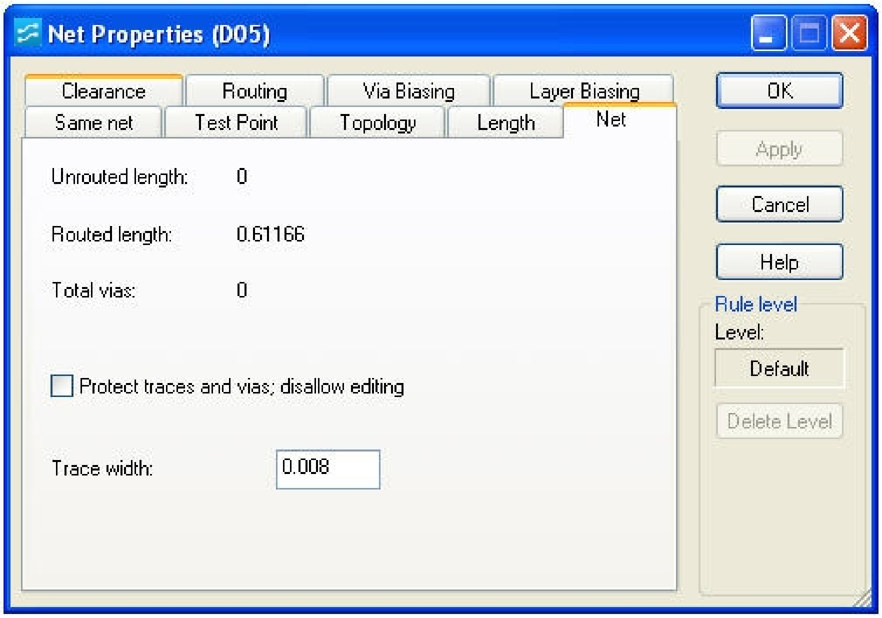
Field	Description
Restrict length	Restricts the net or pin pair length to the range specified by the “Minimum length” and “Maximum length” values.
Minimum length	Specifies the minimum value for the net length. You must select the Restrict length check box to make this box available. If the net is part of a matched length group or diff pair, you must open the Matched Length Net Group, Matched Length Pin Pair Group, or Differential Pair properties to change this value.
Maximum length	Specifies the maximum value for the net length. You must select the Restrict length check box to make this box available. If the net is part of a matched length group or diff pair, you must open the Matched Length Net Group, Matched Length Pin Pair Group, or Differential Pair properties to change this value.

Properties Dialog Box, Net Tab

To access: Select a net > right-click > choose the **Properties** popup menu item > **Net** tab

View net properties, protect traces and vias, and modify the width of all traces on that net.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Unrouted length	Displays the length of the unrouted net.
Routed length	Displays the length of the routed net. Includes half the Discrete length value of each connected pin of components that have a Discrete length assigned.
Total vias	Displays the number of vias on this net.
Protect traces and vias; disallow editing	Prohibits unrouting or moving a trace. **Tip** If a net has some traces that are protected and some that are not, this check box is unavailable.
Trace width	Specifies the width of all traces on a net.

Field	Description
	Restriction: If the net has protected traces, these trace width properties cannot be modified. **Tip** If a net has traces with different widths, the Trace width box is empty.

Properties Dialog Box, No Properties Tab

To access: with the Properties dialog box open, select an object with no properties, or two or more objects of different types.

Displays property information messages.

This tab appears with one of the following messages:

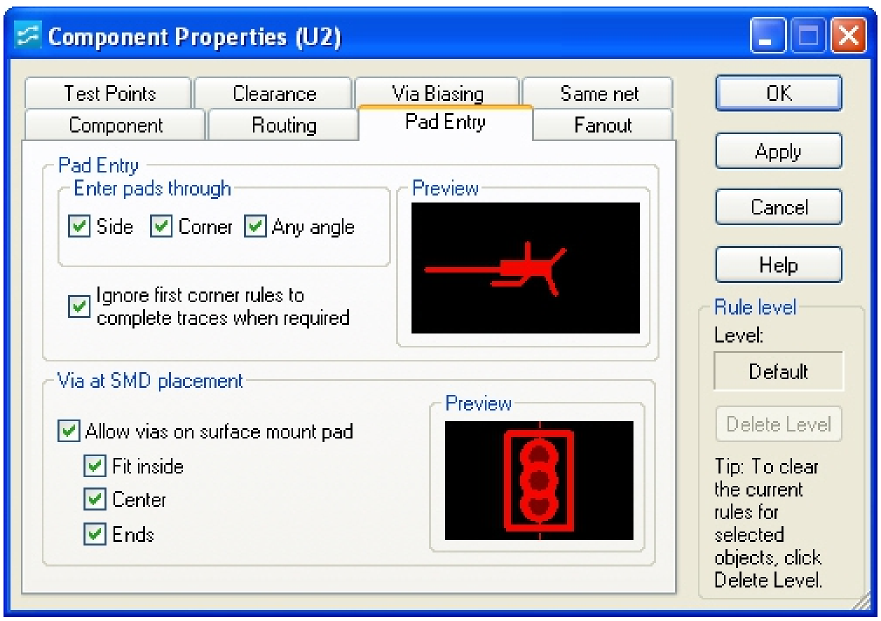
No properties available	Indicates there are no properties for the selected object; for example, an unroute.
Multiple selection	Indicates there are multiple object types selected; for example, a component and a net.

Properties Dialog Box, Pad Entry Tab

To access: Select an object > right-click > choose the **Properties** popup menu item > **Pad Entry** tab

Enable pad entry and exit methods, soften first corner rules, and allow vias in surface mount pads.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Enter pads through area	Select one or more methods where a trace is allowed to enter or exit a pad (Side, Corner, Any angle).
Ignore first corner rules to complete traces when required	Select to set soft first-corner rules to help complete traces.
Via at SMD placement area	**Allow vias on surface mount pad**—Select to allow vias at Surface Mounted Device (SMD) pads. Select any of these additional options: • **Fit Inside**— Fits vias entirely inside the SMD pad to prevent vias from intersecting and breaking the edge of the SMD pads. • **Center**— Places vias at the center of the SMD pad. **Exception:** For pins with associated copper, the via is placed at the origin of the pin, which may not be the center of the pad.

Field	Description
	• **Ends**— Restricts via placement to the ends of the rectangular or oval SMD pads. **Exception:** For pins with associated copper, the via is not placed in the SMD. **Tip** When using square pads, vias are placed at the midpoints of the sides of the pad. Round pads are ignored. **Result:** Depending on the option you select, the Preview section on the tab updates to show the placement of a via on an SMD pad. **Tip** To verify the presence of vias at an SMD pad, click the **Fanout** tab. The Preview changes when you enable or disable Vias at SMD and change the settings.

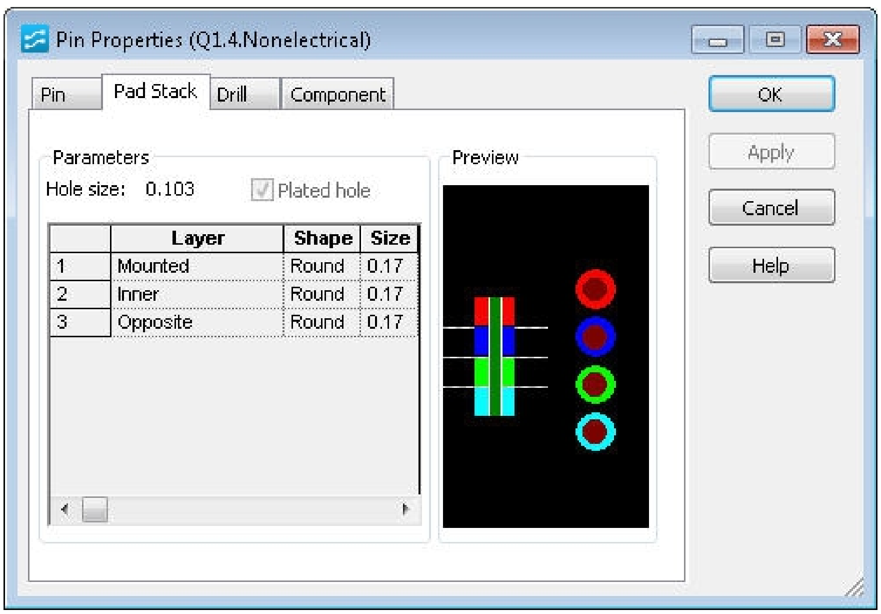
Properties Dialog Box, Pad Stack Tab (Pins)

To access: Select a pin > right-click > choose the **Properties** popup menu item > **Pad Stack** tab

View information about the pin pad stack, such as the layer, shape, size, length, orientation, offset, hole size, whether it is a plated hole, and the corner type and corner radius of rounded/chamfered pads.

Restriction:
This tab is for information only; properties cannot be modified. Use SailWind Layout to modify the information displayed in this tab.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Hole size	Displays the size of the pad stack hole. **Restriction:** You cannot edit this field.
Plated hole	Indicates if this is a plated hole. **Restriction:** You cannot edit this field.

Field	Description
Pad Stack table	Displays information about the pad stacks: layers, shape, and size. **Restriction:** You cannot edit this field.
Preview area	Shows what the pads stack looks like in your design.

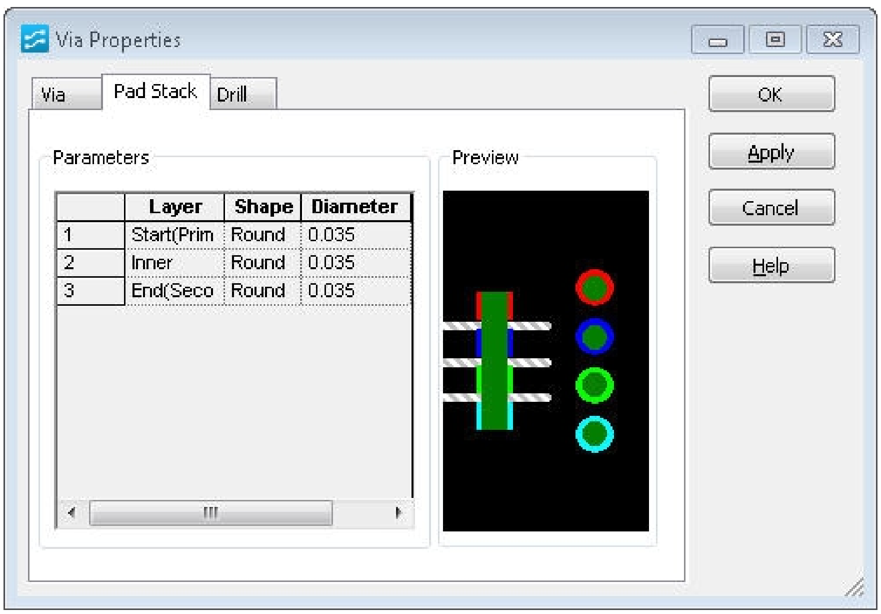
Properties Dialog Box, Pad Stack Tab (Vias)

To access: Select a via > right-click > choose the **Properties** popup menu item > **Pad Stack** tab

View information about the Via or Via type, such as the layers the via connects, the pad shapes and diameters, and the corner type and corner radius of rounded/chamfered pads.

Restriction:
This tab is for information only; the properties cannot be modified. Use the Pad Stacks Properties dialog box in SailWind Layout to modify the information displayed in this tab.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Parameters table	Displays information about the pad stack: layers, shape, and diameter. **Restriction:** You cannot edit this table.
Preview area	Shows what the pads stack looks like in your design.

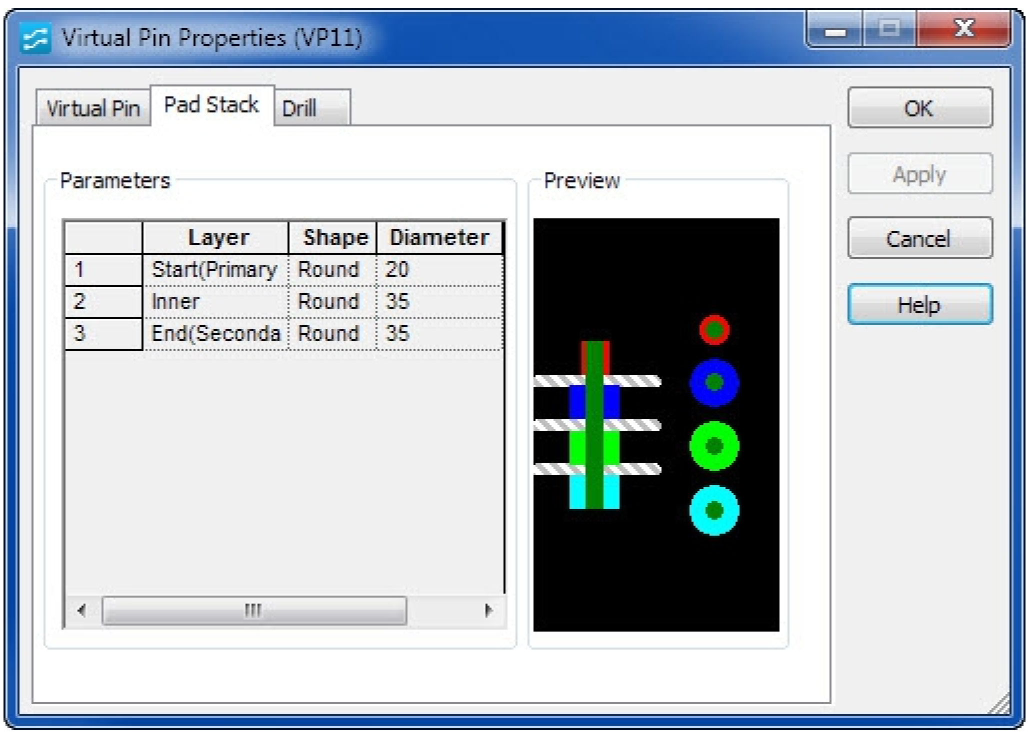
Properties Dialog Box, Pad Stack Tab (Virtual Pins)

To access: Select a virtual pin > right-click > choose the **Properties** popup menu item > **Pad Stack** tab

View information about the virtual pin or virtual pin type, such as the layers the virtual pin connects, the pad shapes and diameters, and the corner type and corner radius of rounded/chamfered pads.

Restriction:
This tab is for information only; you cannot modify the properties. Use the Pad Stacks Properties dialog box in SailWind Layout to modify the information displayed in this tab.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Parameters table	Displays information about the pad stack: layers, shape, and diameter.

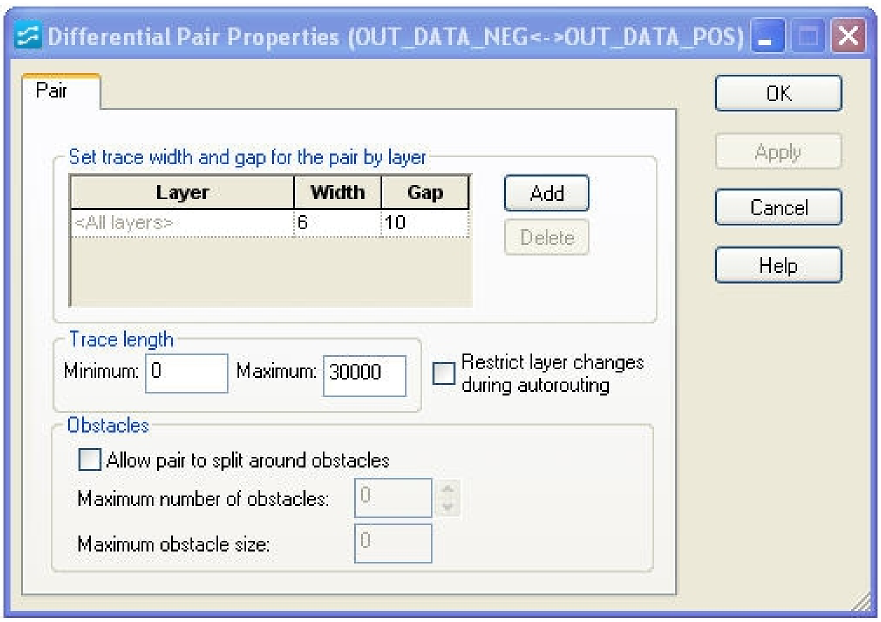
Field	Description
	Restriction: You cannot edit this table.
Preview area	Shows what the pads stack looks like in your design.

Differential Pair Properties Dialog Box, Pair Tab

To access: Select a differential pair > right-click > choose the **Properties** popup menu item > **Pair** tab

Set the gap, trace width, length, and obstacle handling for differential pairs.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and "Setting Object Properties" on page 169.



Objects

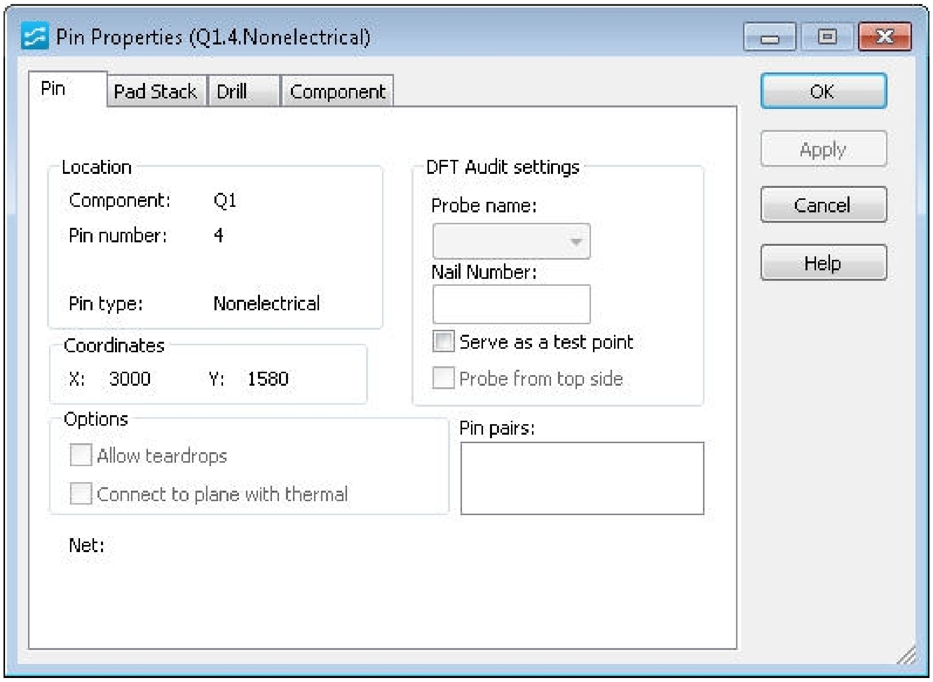
Field	Description
Set trace width and gap for the pair by layer table	Specifies the width and gap for each layer. **Tips:** • The gap rule overrides any other rule defining a clearance between the differential pair members. Therefore, the gap is the minimal clearance and must be provided when possible. • If you select multiple differential pairs, and a layer setting does not belong to all of the selected pairs, the Layer box is unavailable for that layer. If you enter a new value, it applies to the differential pairs that have this layer setting only. • If you select multiple differential pairs that have the same layer setting, but the Width and Gap values do not match, the Width

Field	Description
	and Gap cells will appear empty. You can, however, type a new value, and the new value will be applied to all selected differential pairs when you click **OK** or **Apply**.
Add button	Adds a row to the table where you can specify the width and gap for another layer.
Delete button	Removes the selected row from the table. **Restriction:** You cannot remove the <All layers> row.
Trace Length	Specify the minimum and maximum length allowed for a trace. Includes half the Discrete length value of each connected pin of components that have a Discrete length assigned. Restrict layer changes during autorouting — Specifies to force the pair to be routed on a single layer. This setting applies to autorouting and does not restrict splitting around obstacles when routing interactively.
Allow pair to split around obstacles	Specifies to temporarily exceed the pair routing gap.
Maximum number of obstacles	Specifies the maximum number of obstacles allowed when splitting the pair. **Tip** Obstacles in the start zone or end zone are not counted. This is the zone between the connection point and the shoulder of the differential pair. **Restriction:** This box is available only when you select the “Allow pair to split around obstacles” check box.
Maximum obstacle size	Specifies the maximum size allowed for an obstacle when splitting the pair. **Tip** Obstacle size in the start zone or end zone is not checked. This is the zone between the connection point and the shoulder of the differential pair. **Restriction:** This box is available only when you select the “Allow pair to split around obstacles” check box.

Properties Dialog Box, Pin Tab

To access: Select a pin > right-click > choose the **Properties** popup menu item > **Pin** tab

View and modify pin property information.



Description

You can modify only these options:

- Serve as a test point
- Probe from top side
- Probe name
- Nail number
- Connect to plane with thermal

Restriction:
All other options are for information only.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.

Objects

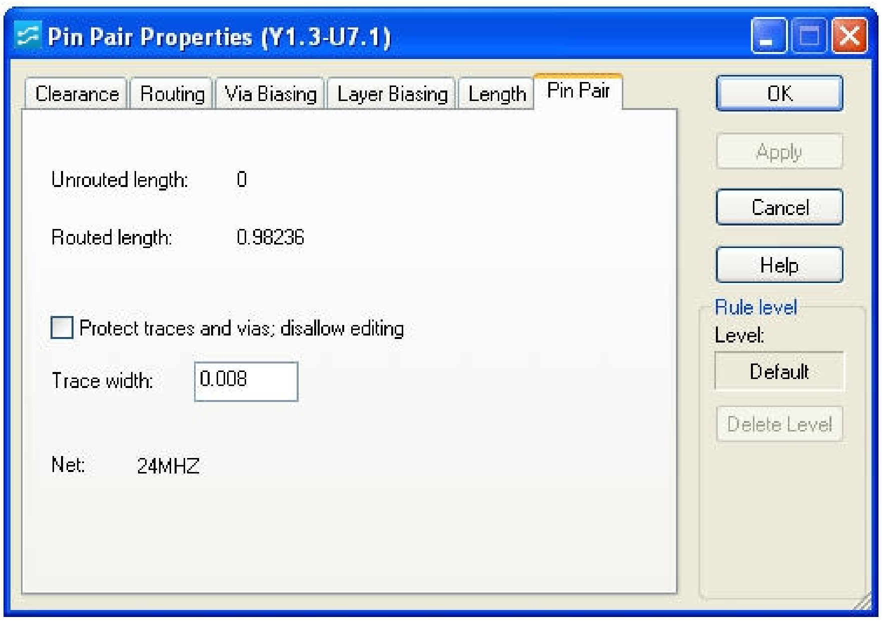
Field	Description
Location area	Displays the properties of the pin: the Component, the Pin number, and the Pin type.
Coordinates	Displays the X,Y coordinates of the pin.
Allow teardrops	Enables the use of teardrop shapes.
Connect to plane with thermal	Enables the pin to connect to a plane.
Net	Displays the name of the net(s) associated with this pin.
Probe name list	If you select the “Serve as a test point” check box, choose one of the available test probe names from this dropdown list. You can define probe names in the Options Dialog Box, Test Points Category.
Nail Number	Specifies a unique label to the probe.
Serve as a test point	Enables to pin to function as a test point. If you select this check box, also select the probe you want to associate with the test point by choosing it from the Probe name list.
Probe from top side	Specifies to make the test point pin accessible from the top side of the board. **Restriction:** The ability to probe from the PCB top side must be enabled (on the Test Points category of the Options dialog box) before you can make a test point pin accessible from the top side of the board.
Pin pairs	Displays the pin pairs associated with this pin.

Properties Dialog Box, Pin Pair Tab

To access: Select a pin pair > right-click > choose the **Properties** popup menu item > **Pin Pair** tab

View and modify pin pair property information. You can modify only the Trace width and Protect traces options. All other options are for information only.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Unrouted length	Displays the length of the unrouted pin pair.
Routed length	Displays the length of the routed pin pair. The displayed length includes half the Discrete length value of each connected pin of components that have a Discrete length assigned.
Protect traces and vias; disallow editing	Prohibits unrouting or moving a trace. **Tip** If a net has some traces that are protected and some that are not, this check box is unavailable.
Trace width	Specifies the width of all traces on the pin pair.

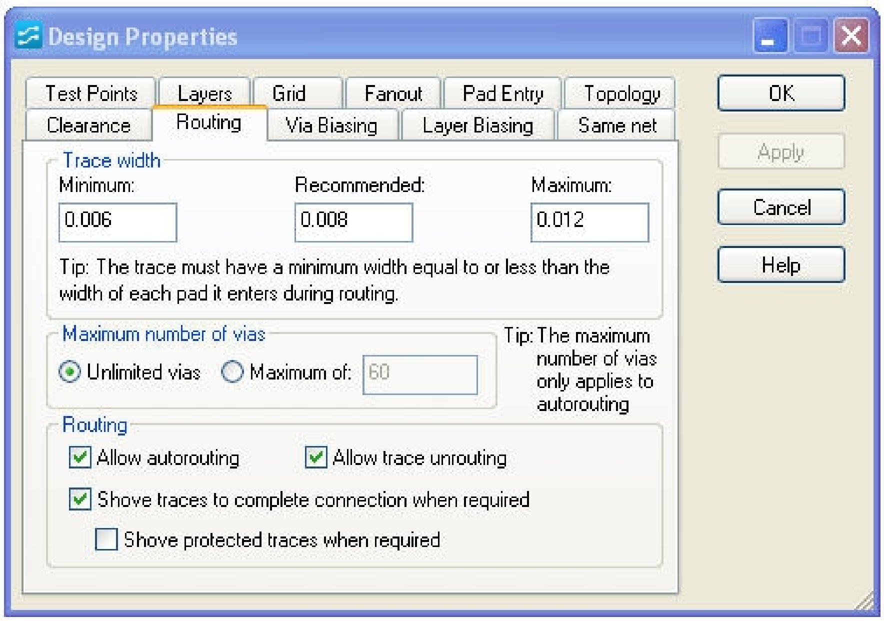
Field	Description
	Restriction: If the pin pair has protected traces, these trace width properties cannot be modified. **Tip** If a pin pair has traces with different widths, the Trace width box is empty.
Net	Displays the net for this pin pair.

Properties Dialog Box, Routing Tab

To access: Right-click > choose the **Properties** popup menu item > **Routing** tab

Set the default routing rules, including trace width and autorouting properties. If you select an object, you set properties for that object. If you select nothing, you set the routing properties as design defaults.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Trace width area	Specifies the trace width value you want: Minimum, Recommended, and Maximum boxes.
Maximum number of vias	Specifies the number of vias properties. • **Unlimited vias**— The autorouter can use as many vias as needed to complete traces. • **Maximum of**— The autorouter can use up to this number of vias to complete traces. Type a value from 0 to 50000. **Tip** An insufficient maximum number of vias might increase autorouting runtime and reduce completion rates. The autorouter considers this to be a hard rule. Interactive routing and design verification check this property.

Field	Description
Allow autorouting	Enables autorouting operations.
Allow trace unrouting	Enables you to unroute traces and delete trace corners.
Shove traces to complete connection when required	Enables the plower on page 299 to move traces aside to complete another trace.
Shove protected traces when required	Enables the plower on page 299 to move protected traces aside to complete another trace.

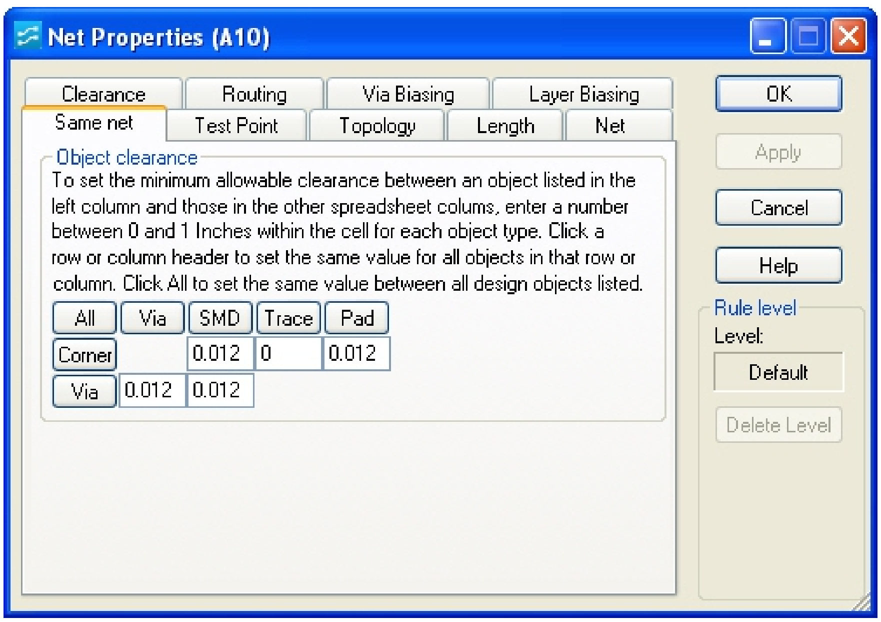
Properties Dialog Box, Same Net Tab

To access:

- Select a design object > right-click >choose the **Properties** popup menu item > **Same net** tab
- With nothing selected > right-click > choose the **Properties** popup menu item > **Same net** tab

Set values for edge-to-edge clearance to items that are in the same net. If you select an object, you set properties for that object. If you select nothing, you set the clearance properties as design defaults.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Object clearance matrix	Type a value in the box for which you want to set the clearance. Alternative: To set the same value for an entire row or column, click on a column heading, row heading, or **All**. Type a value and click **OK** to apply the value.

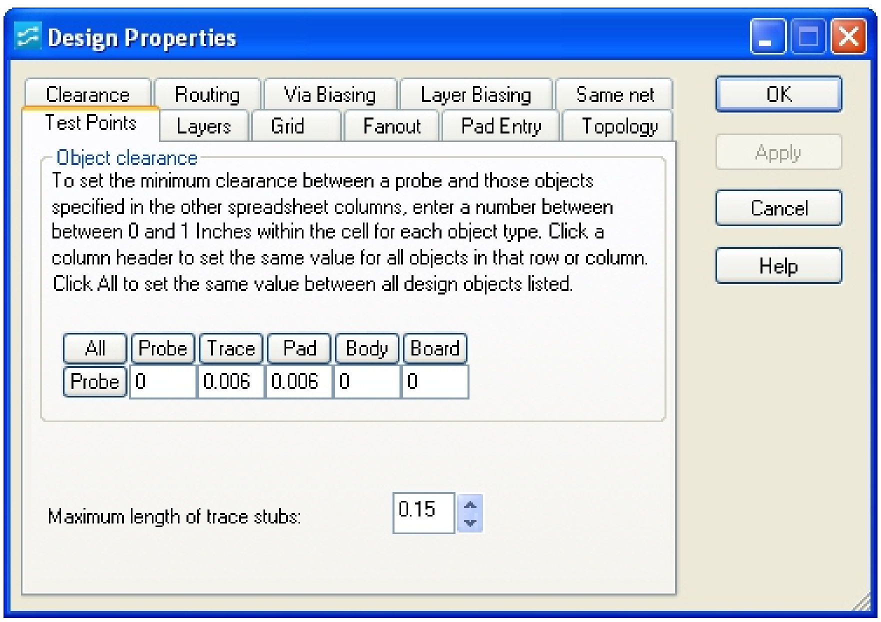
Field	Description
	Restriction: You are restricted to setting only clearances that apply to the type of object you have selected. For example, if you have selected a component and are viewing component properties, the Via-to-Via or Trace-to-Corner settings do not apply and they are unavailable.

Properties Dialog Box, Test Points Tab

To access: With nothing selected > right-click > choose the **Properties** popup menu item > **Test Points** tab

Specify minimum spacing clearances and set the maximum length of trace stubs created to gain access to a net on the entire design.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

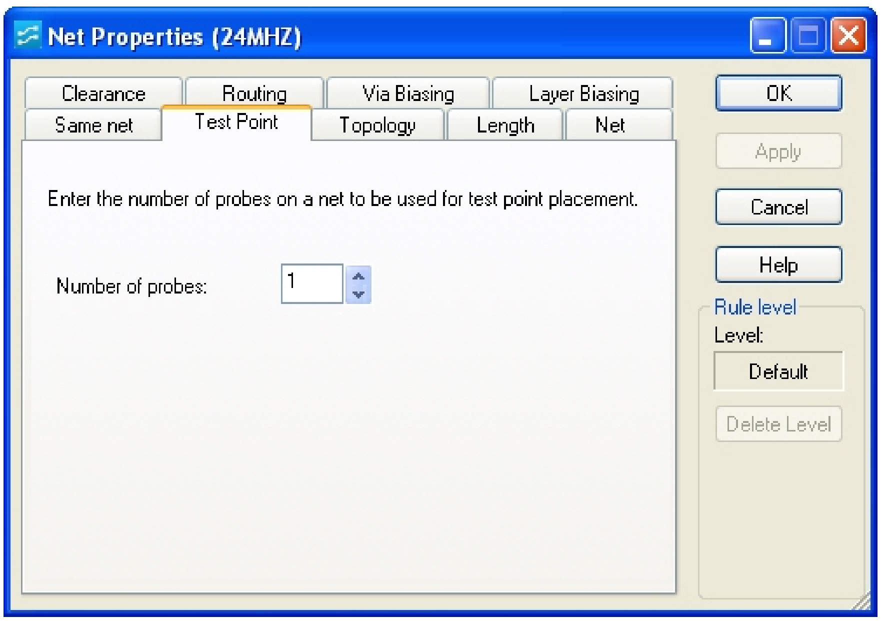
Field	Description
Object clearance matrix	Type a value in the box for the objects for which you want to set a clearance. Alternative: To set the same value for an entire row or column, click on a column heading, row heading, or **All**. Type a value and click **OK** to apply the value.
Maximum length of trace stubs	Type the maximum length value in current design units.

Properties Dialog Box, Test Points Tab (Nets)

To access: Select a net > right-click > choose the **Properties** popup menu item > **Test Points** tab

Set the minimum number of test points on a selected net.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Number of probes	Type a value (0 through 10)

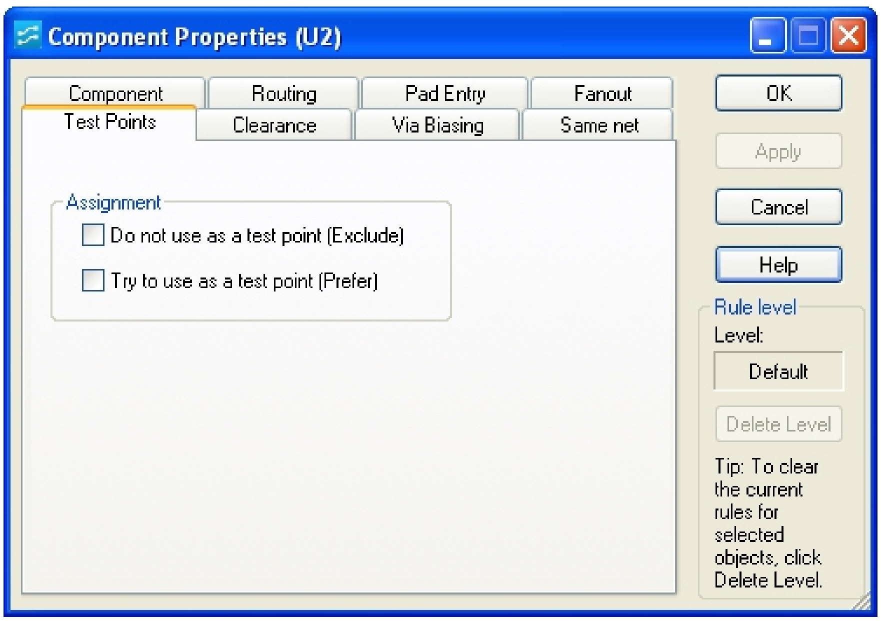
Properties Dialog Box, Test Points Tab (Component or Via Type)

To access:

- Select a component or via type (must be selected using the Project Explorer) > right-click > choose the **Properties** tab > **Test Points** tab

Use the **Test Points** tab to prevent or favor assigning test points to components or to via types. By default, SailWind Router makes all pins on a net available for test pin assignment and weighs them evenly as test point candidates.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Do not use as a test point (Exclude)	Select to prevent use as a test point. **Tip** To apply even weighting, clear both check boxes.
Try to use as a test point (Prefer)	Select to favor use as a test point. **Tip** To apply even weighting, clear both check boxes.

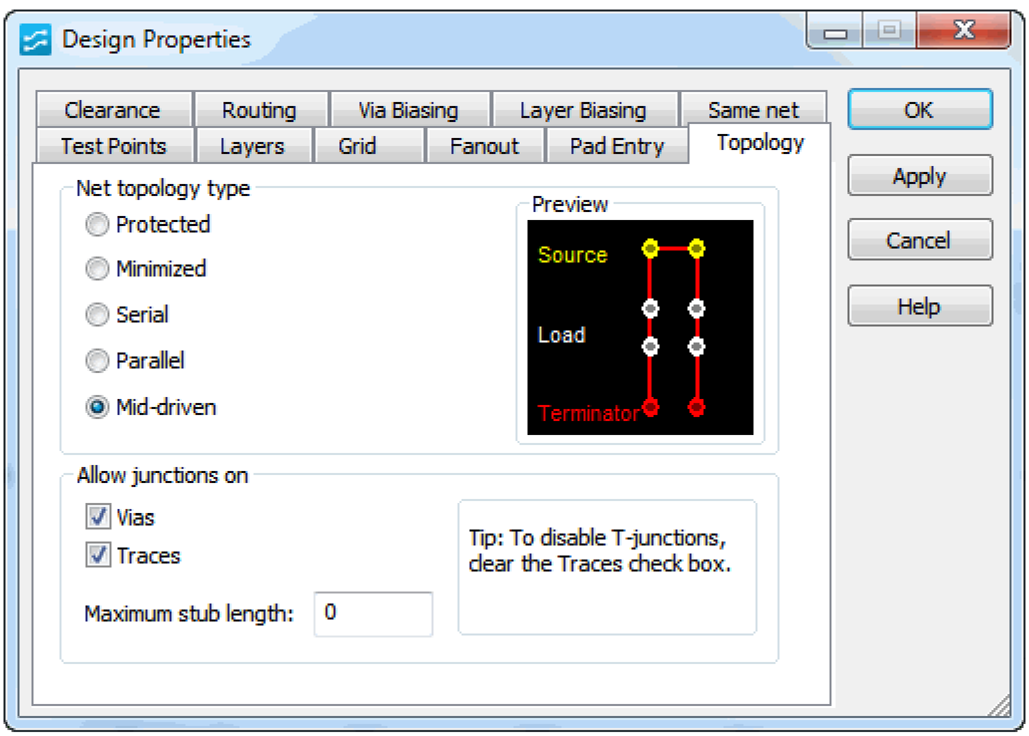
Properties Dialog Box, Topology Tab

To access:

- With nothing selected > right-click > choose the **Properties** popup menu item > **Topology** tab
- Select a net or net class > right-click > choose the **Properties** popup menu item > **Topology** tab

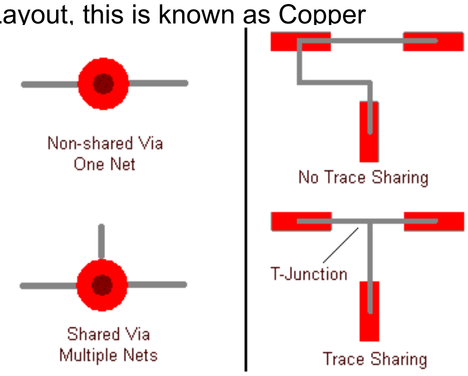
You can set different topology types for nets, which affects how they are routed whether you are routing interactively, autorouting, or editing traces. Topology types determine the order in which to connect pins in a net. Assign the default and net specific topology type and set the permissions for allowing junctions on nets while controlling the stub length.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Net topology type area	**Protected** — Do not change the order of the connectivity in this net. **Minimized** — Order the net by the shortest distance between pins. Net reorder or reconnect is permitted.

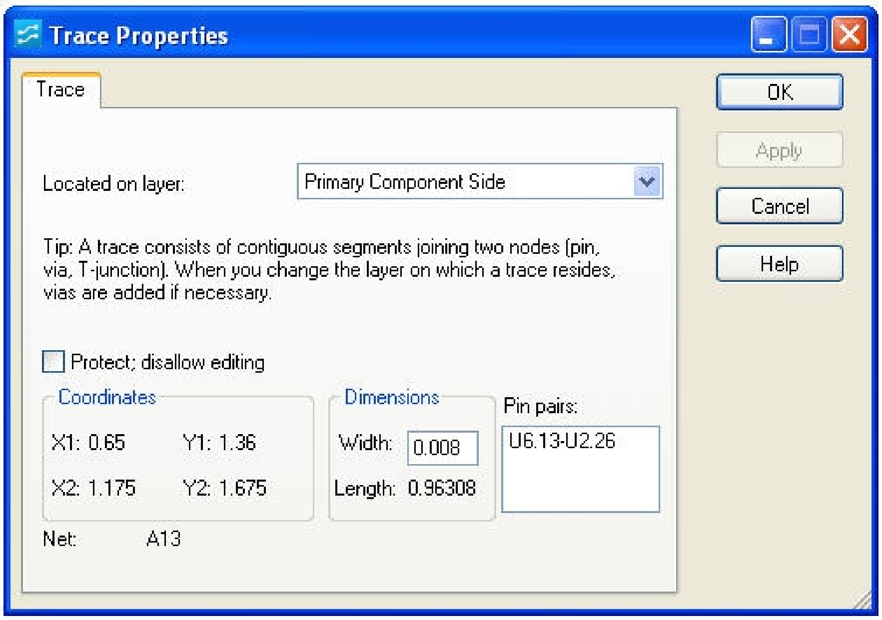
Field	Description
	Serial — Order the net in a series order from source pins to load pins to a terminator. **Parallel** — Same as “Serial” and “Serial Source” except order the net with parallel branches for each source-to-load connection. **Mid-driven** — Divide the net into two branches and order each branch in a source to load to terminator order.
Allow junctions on area	Control junctions on Vias and Traces. In SailWind Layout, this is known as Copper  Sharing. **Maximum stub length** — Type the maximum length of the trace that enters another to create a t-junction. Type zero to prevent t-junctions. This setting is not available if the topology type is set to minimized.

Properties Dialog Box, Trace Tab

To access: Select a trace > right-click > choose the **Properties** popup menu item > **Trace** tab

View and modify the width of the trace and the layer on which it is located, and protect the trace.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Located on layer list	Specifies the layer on which the trace is located. You can move the selected trace to another layer by choosing the layer from the dropdown list. Selecting the “Protect” check box disables the dropdown list.
Protect; disallow editing	Prevent unrouting or moving the trace, either manually or through autorouting.
Coordinates	Displays the location of the trace.
Dimensions	Specifies the width and displays the length of the trace. Length does not include any Discrete length.
Pin pairs	Displays the pin pairs associated with this trace.

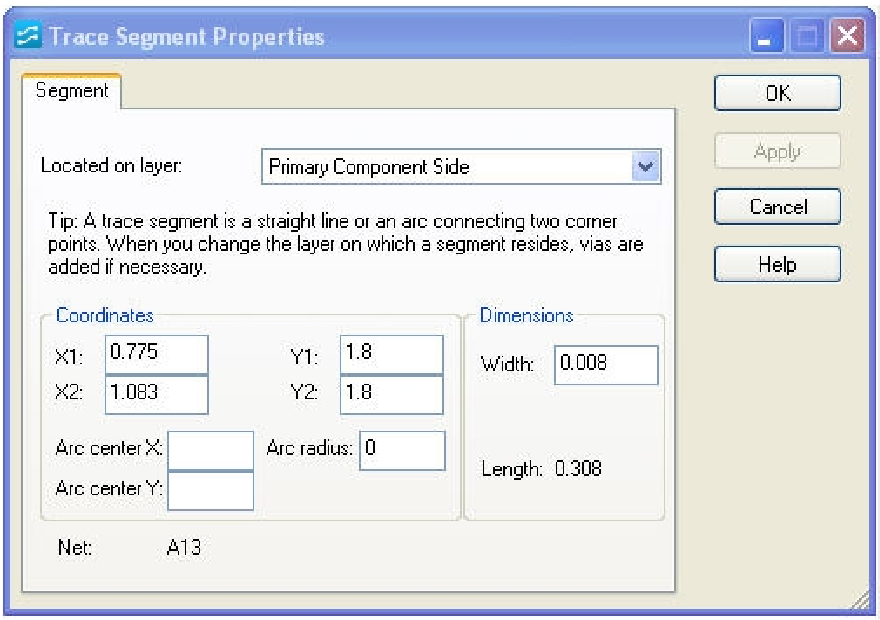
Field	Description
Net	Displays the net associated with this trace.

Properties Dialog Box, Trace Segment Tab

To access: Select a trace segment > right-click > choose the **Properties** popup menu item > **Segment** tab

Set the layer, coordinates, and dimensions of the trace segment.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

Field	Description
Located on layer list	Specifies the layer on which the trace segment is located. You can change layers by selecting another from the dropdown list and clicking OK. The segment moves to the new layer.
Protect; disallow editing	Prevents unrouting or moving the trace segment.
Coordinates	Displays the location of the trace segment: X1, Y1, X2, Y2, Arc center X, Arc center Y, Arc radius. You can edit these values by typing directly in the boxes. SailWind Router applies the change when you click **OK**. **Restriction:** If the segment ends on a pin, the corresponding X and Y coordinates are unavailable.

Field	Description
	Tip Change an arc to a straight line by typing 0 in the Arc radius box. Change a straight line to an arc by typing a non-zero value in the Arc radius box.
Dimensions	Specifies the width and displays the length of the trace. You can edit the width by typing directly in the box. SailWind Router applies the change when you click **OK**.
Net	Displays the net name associated with this trace.

Properties Dialog Box, Via Tab

To access: Select a via > right-click > choose the **Properties** popup menu item > **Via** tab

View and modify via property information.

Restriction:
You cannot change the size of vias or add additional vias in SailWind Router. Use the Pad Stacks Properties dialog box in SailWind Layout to do this.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.

Objects

Field	Description
Coordinates	Specifies the location of the via: X, Y. You can edit the location by typing directly in the boxes. SailWind Router applies the changes when you click **OK**. SailWind Router disables these boxes when you select the "Protect; disallow editing" check box.
Type lists	Specifies the type of this via. You can choose another via type by selecting from the dropdown list. SailWind Router disables this dropdown list when you select the "Protect; disallow editing" check box. **Tip** Only the via types that can be applied without causing connectivity errors or via biasing violations appear in the list.
Protect; disallow editing	Prohibits unrouting or moving the via. Selecting this check box disables all other controls on the tab except for the Stitching check box.
Connect to plane with thermal	Enables the via to connect to the copper plane.
Stitching	Enables the via to function as a stitching via (that is, a via in an unroute).
Net	Displays the net associated with this via.
Probe name list	Specifies the name of the probe. You can choose a probe from the dropdown list.

Field	Description
	The dropdown list becomes available when you select the “Serve as a test point” check box.
Nail number	Specifies a unique label to the probe. Type a label directly in the box.
Serve as a test point	Enables the via to serve as a test point when you select the check box.
Probe from top side	Makes the test point via accessible from the top side of the board. Select the “Serve as a test point” check box to enable this check box.

Properties Dialog Box, Via Biasing Tab

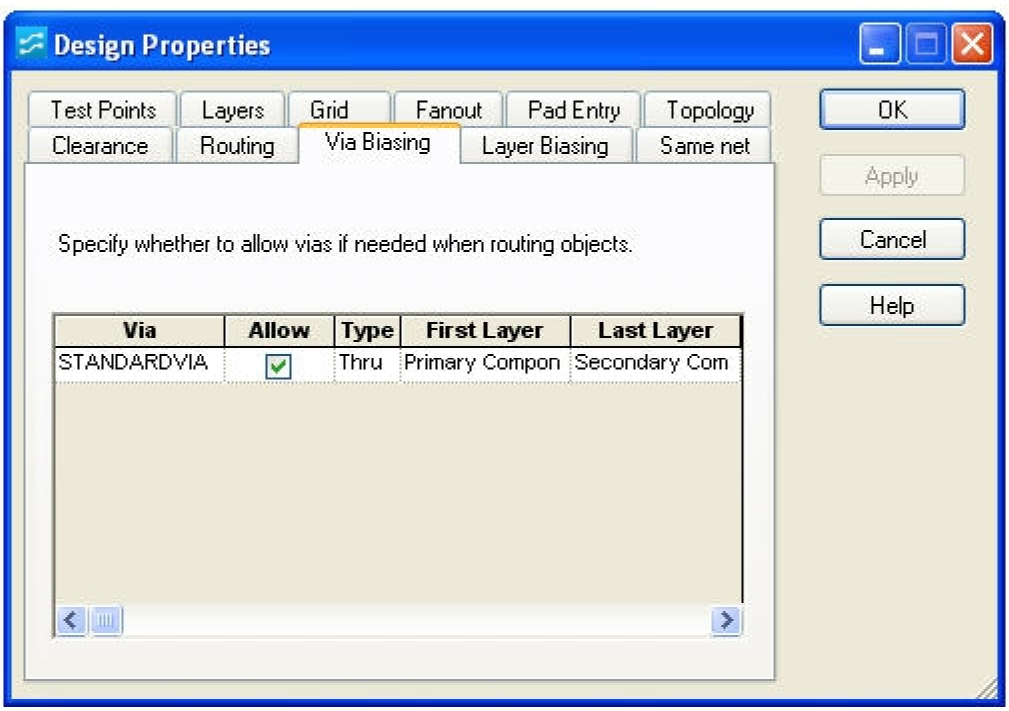
To access:

- Click the **Properties** button > **Via Biasing** tab
- Right-click > choose the **Properties** popup menu item > **Via Biasing** tab

Restrict the via types that SailWind Router uses during routing. If you select a net object, you specify the available vias for that object only. If you select nothing, you set the via biasing properties as design defaults.

Restriction:
You cannot change the size of vias or add additional vias in SailWind Router. Use the Pad Stacks Properties dialog box in SailWind Layout to do this.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

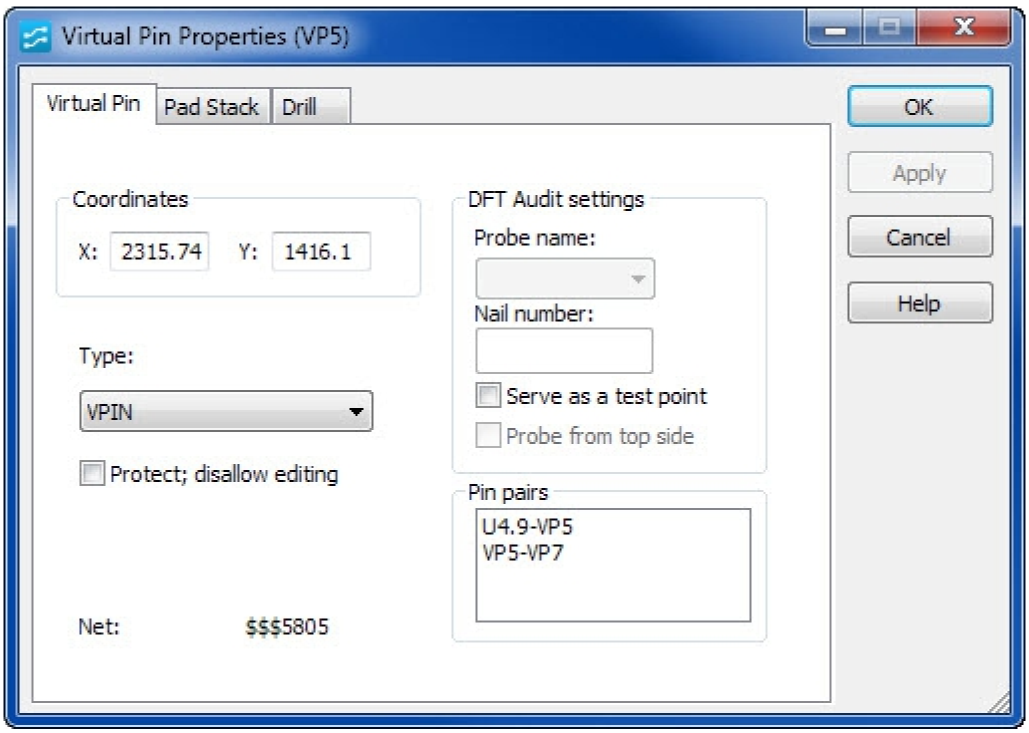
Field	Description
Via table	Lists the properties of each available via, including the name, type of via (for example, a through-hole or partial via type), and layers through which the via extends. Selecting the check box in the Allow column enables the via for routing.

Properties Dialog Box, Virtual Pin Tab

To access: Select a virtual pin > right-click > choose the **Properties** popup menu item > **Virtual Pin** tab

View and modify virtual pin properties.

Note:
Use the Properties dialog box for setting design and object properties. For more information, see Setting Design Properties on page 154 and Setting Object Properties.



Objects

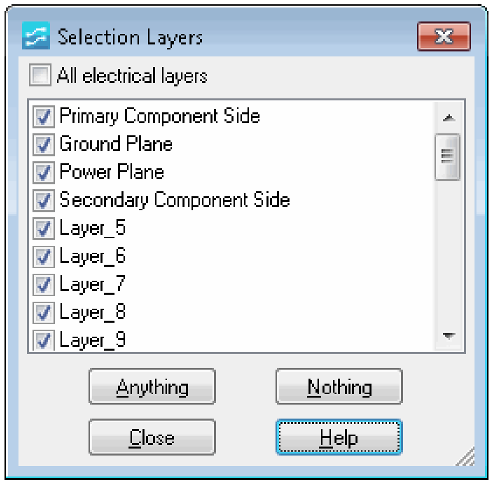
Field	Description
Coordinates	Specifies the location of the virtual pin: X, Y. You can edit the values by typing directly in the boxes. The virtual pin changes location when you click **OK**.
Type list	Specifies the type of this virtual pin. **Tip** Only the virtual pin types that can be applied without causing connectivity errors or via biasing violations are listed.
Protect; disallow editing	Prevents or prohibits moving the virtual pin.

Field	Description
	Selecting this check box disables the other controls on the tab.
Net	Displays the name of the net associated with this virtual pin.
Probe name list	Specifies the name of the probe. You can choose any available probe name from the dropdown list. This dropdown list is available only if you select the “Serve as a test point” check box.
Nail number	Specifies a unique label for the probe. You can edit the label by typing directly in the box. This box is available only if you select the “Serve as a test point” check box.
Serve as a test point	Enables the virtual pin to function as a test point.
Probe from top side	Enables access to the test point virtual pin from the top side of the board. This field is available only if you select the “Serve as a test point” check box.
Pin pairs	Displays the pin pairs to which the virtual pin belongs.

Selection Layers Dialog Box

To access: Click the **Selection Filter toolbar** button > **Layers** button

Use the Selection Layers dialog box to enable or disable the ability to select objects on certain layers.



Objects

Field	Description
All electrical layers	Enables object selection on all electrical layers. Clear the check box to disable object selection on all electrical layers. **Note:** This check box does not affect non-electrical layers such as the silkscreen layer.
Layer list	Lists all of the layers in the design. Select the check box to enable object selection on the corresponding layer.
Anything button	Enables object selection on all layers.
Nothing button	Disables object selection on all layers.

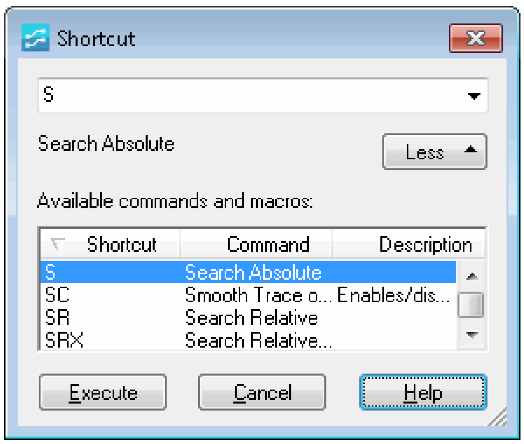
Shortcut Dialog Box

To access:

- Click the **Shortcut Dialog** button on the standard toolbar

- Start typing the shortcut sequence (Note that if you open the Shortcut dialog box by typing a shortcut, you cannot delete the initial character in the text box).

The Shortcut dialog box displays the shortcut key sequence as you type, allowing you to check its accuracy before starting the shortcut. Shortcut information and syntax, if available, display below the text box. Parentheses enclose optional arguments. Similar shortcuts display in the Available commands and macros box.



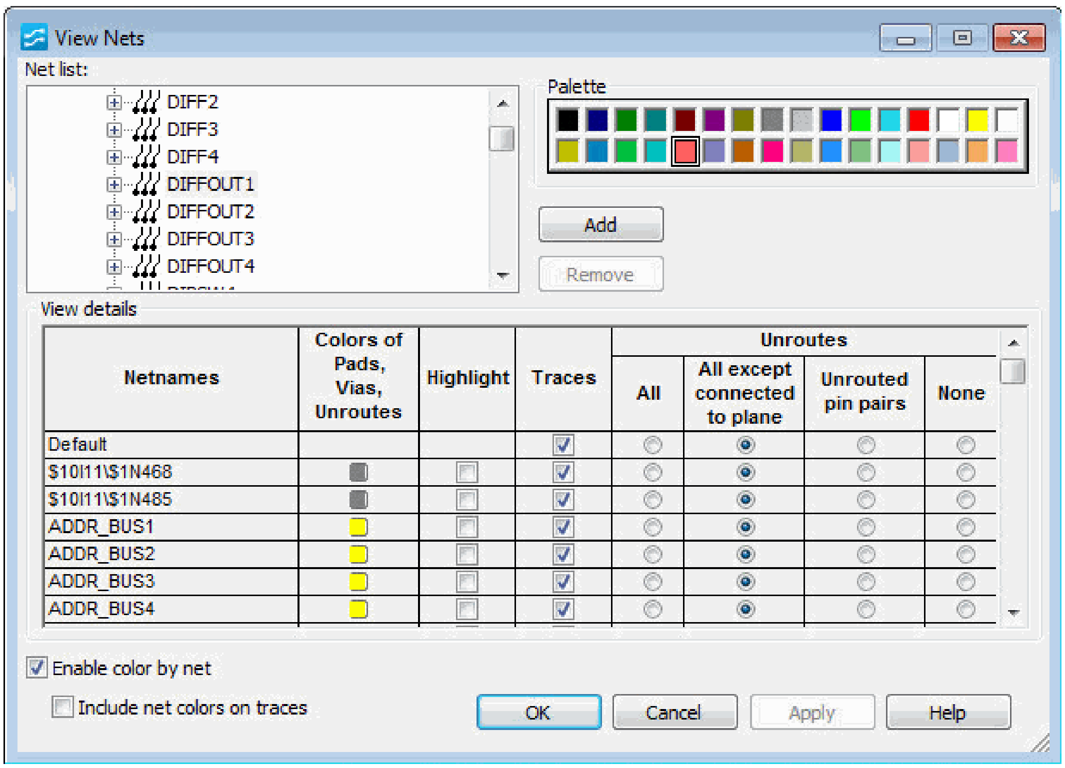
Objects

Field	Description
Shortcut text box	Displays the shortcut you type.
More/Less button	Displays or hides the Available commands and macros list.
Available commands and macros list	Displays the available commands or macros. The list updates based on what you type. **Tips:** • To re-sort the contents click a column header. • Use the wildcard character * to substitute for any number of characters, and use the wildcard character ? to substitute for one character. For example, to highlight all nets whose names start with the letter A, type H A*.
Execute button	Runs the selected command or macro.

View Nets Dialog Box

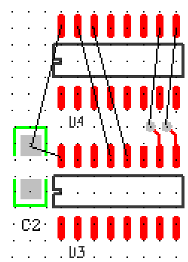
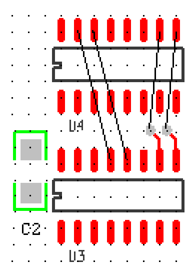
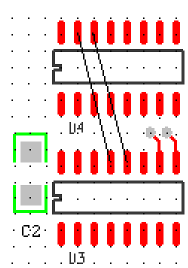
To access: Choose the **View > Nets** menu item

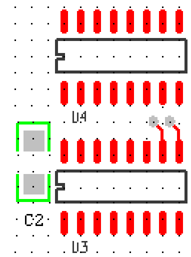
The View Nets dialog box lets you select and view nets. For each net, you can hide or show connections, routed paths, or unrouted paths. You can also specify color and show or hide various unroutes, making it easier to identify nets when you split planes. When you save the design, you also save any net color assignments you make.



Objects

Field	Description
Net List	Displays the Nets and Net Classes available. This list functions similarly to the **Object View** tab in the Project Explorer; however, it displays nets and net classes only. **Tip** Use Ctrl or Shift and click to select multiple nets.
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. Press the Ctrl key for multiple selections. Click **Remove** to move the net from the View List to the Net List.
View Details area	

Field	Description
Colors of Pads, Vias, Unroutes	Displays the assigned color. Click to apply the color selected in the Palette. Click again to return the color to its default. **Note:** Select the “Enable Color by Net” check box to enable this feature.
Highlight	Specifies that the net always appears in the highlight pattern that you set in the General category of the Options dialog box.
Traces	When you clear the Traces check box for a net, traces and unroutes associated with the net do not appear, and the unroute group becomes unavailable.
Unroutes	Enables filtering of the unrouted connections. • All  • All Except Connected to Plane  • Unrouted Pin Pairs 

Field	Description
	• None 
Enable Color by Net	Allows assignment of colors to pads, vias, unroutes, and coppers. You can also use the modeless command Y to toggle this setting—see the Status bar for confirmation. Selecting this check box also allows you to optionally select the “Include net colors on traces” check box to apply color assignments to traces. Clearing this check box returns colors to their default assignments (as assigned in the Options Dialog Box, Colors Category). **Note:** Re-selecting this check box restores any previously defined net color assignments.
Include net colors on traces	Shows traces using the assigned color. Selecting the “Enable Color by Net” check box enables this check box.

Chapter 25
Software Launch Options

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

You add command line switches to the Start menu shortcut or a Desktop shortcut.

Table 90. Start-up Options

Option	Description
full path to a file	Opens the specified design file when you start the program. Type the full folder path and filename. Use double quotation marks for folders or file names with spaces. For example: *“C:\SailWind Projects\Samples\preview.pcb”*
/BMW[=xx]	Opens the Basic Media Wizard. Use the Basic Media Wizard to start recording a session log or to convert the previous session log to media that can be replayed by Basic Log Test. To create session media files for the current session, use the BMW Table 30 on page 104. To use the BMW command line switch, type /BMW or /BMW=xx, where you replace xx with your initials.
/RUN=macro_name	Runs the specified macro when you launch the software. Make sure to type a space between “SailWindRouter.exe” and the “-run” command. For example, to run the macro “mymacro”, type /RUN=mymacro. Instead of using the / character before the run command, you can use a hyphen, as in -run=mymacro. When paths or filenames have spaces, you must place double quotes around the full path. For example: `/run=“C:\SailWind Projects\Samples\mymacro.mcr”`

Adding Start-up Options to a Shortcut

You can add various start-up options to a shortcut. Doing so allows you to start SailWind Router with certain designs already loaded, such as when you click a shortcut.

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 End, press Spacebar, 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 filename. 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\SailWindRouter.exe” “C:\SailWind
 Projects\Samples\preview.pcb”
```

Related Topics

Creating Macros

Editing BGA Fanout Controls With the SailWindRouter.ini File

In some PCB designs, you may have one or more BGA devices that do not conform to standard BGA definitions in SailWind Router and therefore require additional controls for a successful fanout. You can define additional BGA controls in the *SailWindRouter.ini* file to ensure SailWind Router creates a fanout for a device correctly.

BGA Commands in the SailWindRouter.ini File

You can designate a component as a BGA device through its decal name. SailWind Router automatically checks the name of each decal in the design. If SailWind Router finds any decal with the letters “BGA” (in order), it considers the component to be a BGA device.

Aside from decal names, you can add BGA control parameters to the *SailWindRouter.ini* file. This file resides by default in the *<install>/Settings* folder. Aside from decal names, you can add the following additional commands to the file using “[Fanout]” as the section header.

Table 91. BGA Control Parameters

Section	Type	Value Ranges	Parameter	Default Value	Description
fanout	Int	1-10	miminumfullnessgridrow	2	Number of minimum rows required for a component to be considered a BGA device.
fanout	Int	1-10	minimumfullnessgridcol	2	Number of minimum columns required for a component to be considered a BGA device.
fanout	String	1-99	maximumgridratio	40	Maximum ratio between rows and columns (number of rows divided by columns) required for SailWind Router to discard a component as a

Table 91. BGA Control Parameters (continued)

Section	Type	Value Ranges	Parameter	Default Value	Description
					BGA device. Anything less than this ratio is not a BGA.
fanout	String	1-99	odbminimumgridfullness	25	The percentage of fullness (with pins) required for a grid to be considered as a BGA device grid.

Example

The following example illustrates the additional BGA control parameters with the required section header in the *SailWindRouter.ini* file.

```
[Fanout]
minimumfullnessgridrow=2
minimumfullnessgridcol=2
maximumgridratio=40
odbminimumgridfullness=25
```

Checking for SailWind Updates

The SailWind products automatically check for a new software version when you launch an application. If a SailWind product detects a new version, it displays a tooltip in the system tray.

Downloading the Update

You can download an update if SailWind detects a new version is available.

Prerequisites

An Internet connection is required for the check.

Procedure

1. Right-click the icon in the system tray.
2. Click the **Open download page** popup menu item.
3. Follow the instructions on the download page.

Disabling the Check for Updates

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

Procedure

1. Choose the **Help > Check for Updates** menu item.
2. In the "Check for Updates dialog box" on page 394, select the "Disable 'Check for Updates' functionality" check box.

Checking for Updates Manually

You can check for updates manually if you disable the automatic check.

Procedure

1. Choose the **Help > Check for Updates** menu item.
2. In the “Check for Updates dialog box” on page 394, click **Check for Updates**.

Chapter 26
Customizing the Interface

You can customize the SailWind Router interface to suit your work style and needs. You can specify which toolbars to display, add items to toolbars and menus, and create custom toolbars, menus and shortcut keys.

You make customizations using the Customize dialog box. You can invoke the dialog box in two ways:

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

Your customizations save 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.

Migrating User Settings

You can use the User Settings Migration tool to extract your settings from one installation of SailWind Logic, Layout, and Router and import them into another installation or version.

For information on how to do this, see User Settings Migration in the *User Settings Migration Guide*.

Customizing Toolbars and Shortcut Menus

Choose the **Toolbars and Menus** tab on the Customize dialog box (**Tools > Customize** menu item, **Toolbars and Menus** tab) 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".

Creating a Custom Toolbar

Create a custom toolbar by creating a new empty toolbar and adding items (commands) to it.

Procedure

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

   The new, empty toolbar displays on the SailWind interface.
4. Drag the toolbar to the place on the SailWind interface where you want it.
5. To add items (commands) to your new toolbar, click the **Commands** tab and proceed as follows:

   a. In the Categories list, select a menu or toolbar name to display commands specific to that menu or toolbar. Alternatively, select "All Commands."

   Restriction:
   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.

   b. In the Commands list, select the command you want and drag it to the toolbar.

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

Related Topics

Removing Items from Toolbars and Menus

Editing a Custom Toolbar

Make edits to your custom toolbars as necessary using the Customize dialog box.

Procedure

1. Choose the **Tools > Customize** menu item.
2. In the Options dialog box, click the **Toolbars and Menus** tab.
3. Perform one of the following:

Desired Outcome	Steps
Showing or hiding a toolbar	1. In the Toolbars list, select the toolbar. 2. To display the toolbar in the interface, select the check box to the left of its name. To hide the toolbar, clear the check box.
Deleting a custom toolbar	1. In the Toolbars list, select the custom toolbar. 2. Click the **Delete** button.
Renaming a custom toolbar	1. In the Toolbars list, select the toolbar and click the **Edit** button. 2. In the Toolbar Name dialog box, type the new name and click **OK**.

Note:
You cannot edit or delete standard toolbars and menus.

Resetting Toolbars to Defaults

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

Procedure

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

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

Customizing a Shortcut Menu

You can customize an existing shortcut menu by adding, deleting, or rearranging its menu items. You can also reset a shortcut menu to its default state.

Note:
You cannot create a new shortcut menu.

Procedure

1. Click the **Tools > Customize** menu item; then, in the Customize dialog box, click the **Toolbars and Menus** tab.
2. In the Shortcut Menus area, select the shortcut menu you want to modify. The shortcut menu (called Popup Menu) appears.
3. You can now add, delete or rearrange its menu items (commands).
   - **To add an item**:
     i. Click the **Commands** tab of the Customize dialog box.
     ii. In the Categories area, select a menu or toolbar name to display commands specific to that menu or toolbar. Or click All Commands.
     iii. Drag the command to the shortcut menu.
   - **To delete an item**: Select the item and drag it outside the shortcut menu.
   - **To rearrange items**: Select an item and drag it to a new location on the menu.
4. When you have finished customizing the menu, close it (click X) or click **Close**.

Tip
If a command's operation does not match the context of the shortcut menu, the Customize operation lets you add the command to the menu, but the menu does not display the command (menu item). For example, in SailWind Router, the “Smooth in router” command is a verb mode command. If you add the “Smooth in router” command to the Interactive Routing shortcut menu, the command does not appear on the menu.

Resetting a Shortcut Menu

If you customize shortcut menus and want to change them back to their original menu items, you can reset one or all of them to default settings.

Procedure

1. Choose the **Tools > Customize** menu item and click the **Toolbars and Menus** tab.
2. In the Shortcut menus list, select the shortcut menu you want to reset.
3. In the Shortcut menus area, click **Reset**.

Tip
To reset all shortcut menus to their default settings, in the Shortcut menus box, click **Reset All**.

Related Topics

Customizing a Shortcut Menu

Creating Custom Commands and Menus

The **Commands** tab provides you the option of customizing commands or menus.

Creating and Editing a Custom Command

You can create custom commands from either commands that already exist or a macro command file.

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

Procedure

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

   Tip
   If you made macro commands (on the **Macro Files** tab) available as commands, the Categories list includes the Macro category and the Commands list includes the macros. For more information, see "Creating Commands from Macro Command Files" on page 563.

3. In the Commands list, select the command on which you want to base your custom command. Then click the **New** button. If you want to edit an existing custom command, select the command and click **Edit**.
4. In the Add Command dialog box (or Edit Command dialog box, if you are editing a custom command), specify or change the properties of your custom command:

   a. In the Command name box, type the name of the command.

   b. If the new command takes arguments, type the arguments in the Arguments box. Use a space to separate each one. If an argument contains a space, enclose the argument in quotation marks (" ").

   c. In the Description box, type a description of the custom command.

d. If an image is associated with the original command, select “Use Default Image” to use that same image with your custom command. Alternatively, select “Select User-Defined Image” to use a different image, edit an image, or create a new image.

e. Click **OK** to close the Add Command dialog box (or Edit Command dialog box) and return to the Customize dialog box.

5. When you finish with all customizations, click **Close**.

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

Deleting a Custom Command

You can remove any custom command you create.

Restrictions and Limitations

You can delete only custom commands (the commands you create). You cannot delete system commands.

Procedure

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

Related Topics

Adding Items to Toolbars and Menus

Creating Commands from Macro Command Files

Resetting Toolbars to Defaults

Creating a Custom Menu

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; then, in the Customize dialog box, click the **Commands** tab.
2. In the Categories list, select “New Menu.”
3. In the Commands list, select New Menu and drag it to the location you want as follows:

    - To create a top-level menu, drag the new menu to the Menu Bar.
    - To create a submenu, drag it over an existing menu name.
4. Click your new menu to select it. Then right-click and select **Button Appearance**.
5. In the “Button text” field, type the name for the menu and click **OK**.

   Leave the Customize dialog box open on the **Commands** tab.
6. In the Categories list, select a menu or toolbar name to display commands specific to that menu or toolbar, or select “All Commands.”

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

Adding Items to Toolbars and Menus

Add buttons to your custom toolbars and menus to populate them with commands and custom commands.

Procedure

1. Choose the **Tools > Customize** menu item and click the **Commands** tab.
2. In the Categories list, select a toolbar or menu name to display commands specific to that menu or toolbar, or select “All Commands.”

   Restriction:
   If you are working in a special mode in 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.

   Tip
   To remove an item from a toolbar or menu (while the Customize dialog box is open), click the item and drag it outside the toolbar or menu.
4. When you have finished adding commands, click **Close**.

Related Topics

Creating and Editing a Custom Command

Creating Commands from Macro Command Files

Removing Items from Toolbars and Menus

Moving Items on Toolbars and Menus

Moving Items on Toolbars and Menus

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

Moving Buttons on Toolbars
Moving Items on Menus

Moving Buttons on Toolbars

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

Procedure

If the Customize dialog box is closed, press and hold the Alt key; then drag the toolbar button to a new place on the same toolbar or to a different toolbar. If the Customize dialog box is open, click the toolbar button and drag it to a new place on the same toolbar or to a different toolbar.

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

Moving Items on Menus

You can rearrange the items on your custom menus.

Restrictions and Limitations

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

Procedure

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

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

4. Click **Close**.

Related Topics

Creating Custom Commands and Menus

Adding Items to Toolbars and Menus

Resetting Toolbars to Defaults

Removing Items from Toolbars and Menus

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

Procedure

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, you can still remove toolbar items by pressing and holding the Alt key then dragging the item outside the toolbar.

Tip
You can reset a toolbar or shortcut menu back to its default list of items. See "Resetting Toolbars to Defaults" on page 549.

Related Topics

Adding Items to Toolbars and Menus

Customizing Shortcut Keys

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

Creating a New Shortcut Key

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

Procedure

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

   Tip
   To replace an existing shortcut, select the existing shortcut and click **Delete** to remove it; then create a new shortcut for the command.

4. Above the Current shortcuts box, click the **New** button to open the “Assign shortcut” dialog box.
5. Select one of the following types of shortcut:

- To assign shortcut keys, select the “Press new shortcut key(s)” option, and then press the keys that you want to use. For detailed information about rules and restrictions for creating shortcut keys, see “Rules and Restrictions for Key Sequences” on page 559.

Tip
As you enter the new shortcut, similar shortcuts appear in the Similar shortcuts assigned to other commands box. This helps you to avoid creating a new shortcut that conflicts with an existing shortcut.

- To create a mouse action, select the “or select a pointer event” option, 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.

Results

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

Rules and Restrictions for Key Sequences

The first character of a shortcut key must meet certain requirements.

The first character of a shortcut key may consist of any item in the following list, plus Alt, Ctrl, or Shift modifiers:

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

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

Subsequent characters may consist of the following:

- Alphanumeric (a-z0-9)

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

Related Topics

Creating Custom Commands and Menus

Resetting Default Shortcut Keys

Listing Available Shortcut Keys

Listing Available Shortcut Keys

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

Procedure

1. Choose the **Tools > Customize** menu item and click the **Keyboard and Mouse** tab.
2. Click **Report** and then select or type the HTML filename, then click **Save**.
3. In the Customize dialog box, click **Close**.

Results

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

Related Topics

Creating Custom Commands and Menus

Creating a New Shortcut Key

Resetting Default Shortcut Keys

Expressions in Shortcut Keys

You can substitute a regular expression for characters in shortcut key command arguments.

The following table describes the use of expressions.

Table 92. Expressions in Shortcut Keys

Expression	Use to
*	Match any number of characters.
?	Match any one character.
[set]	Match any character in the specified set. **Tip** A set is composed of characters or ranges. A range has the form: Character Hyphen Character, such as A-Z or 0-9.

Table 92. Expressions in Shortcut Keys (continued)

Expression	Use to
	The minimum set of characters supported in a set consists of [0-9a-zA-Z_].
[!set] or [^set]	Match any character not in the specified set.
\	To suppress the special syntactic significance of the characters ` [ ] * ? ! ^ - \ ' within a set, and to match the character exactly.

The following table shows examples of regular expressions used in command arguments using the preview.pcb design, see Table 93.

Table 93. Shortcut Key Expression Examples

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

Deleting a Shortcut Key

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

Procedure

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

Related Topics

Creating a New Shortcut Key

Resetting Default Shortcut Keys

Resetting Default Shortcut Keys

You can restore all shortcut keys to their default settings.

Procedure

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

Assigning Shortcut Keys to Macros

You can create a shortcut key that executes a macro.

Tip
In order to assign a macro to a shortcut key, the macro command file (.mcr) must already exist. You can create a macro by recording it in a SailWind tool or scripting it in Macro language. For more information, see "Creating Macros" on page 60.

Procedure

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

   Tip
   To close the macro file or make it unavailable in the Customize dialog box, clear the check box next to the macro name.

4. To assign the macro to a shortcut key, click the **Keyboard and Mouse** tab.
5. In the Mode list, select "All modes."
6. In the Commands area, double-click "Macros" to display a list of available macros. Then select the macro you want.
7. In the Current Shortcuts area, click the **New** button. The software displays the "Assign shortcut" dialog box.
8. Select one of the following types of shortcut:

- To assign shortcut keys, select the "Press new shortcut key(s)" option, and then press the keys that you want to use. For detailed information about rules and restrictions for creating shortcut keys, see "Rules and Restrictions for Key Sequences" on page 559.

> **Tip**
> As you enter the new shortcut, similar shortcuts appear in the Similar shortcuts assigned to other commands box. This helps you to avoid creating a new shortcut that conflicts with an existing shortcut.

- To create a mouse action, select the "or select a pointer event" option, and then select a combination of list box options, mouse button events, and modifier keys.

9. Click **OK** to close the Assign shortcut dialog box.

Results

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

Creating Commands from Macro Command Files

You can add custom commands created from macro files to toolbars and menus.

> **Tip**
> To create a command from a macro command file, the macro command file (.mcr) must already exist. You can create a macro by recording it in a SailWind tool or scripting it in Macro language. For more information, see "Creating Macros" on page 60.

Procedure

1. Choose the **Tools > Customize** menu item and click the **Macro Files** tab.

2. Click the **New** button.

3. In the Open macro file dialog box, select the macro file you want to use as a command then click **Open**. The software loads the macro and makes it available for use as a command (and also selects the check box to the left of the macro name).

> **Tip**
> To close the macro file, or make it unavailable in the Customize dialog box, clear the check box next to the macro name.

You can use the macro as you would any other command, for example, to create a Commands list for the Macros category.)

Related Topics

Adding Items to Toolbars and Menus

Adding a Macro to a Menu

Adding a Macro to a Menu

Create a macro then add the macro command as an item on a menu or a button on a toolbar.

Tip
To add macro to a menu, the macro command file (.mcr) must already exist. You can create a macro by recording it in a SailWind tool or scripting it in Macro language. For more information, see "Creating Macros" on page 60.

Procedure

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

   Tip
   To close the macro file or make it unavailable in the Customize dialog box, clear the check box next to the macro name.

   ---

4. To add the macro to the menu, click the **Commands** tab.
5. From the Categories list, select "Macros."
6. In the Commands list, select the macro and drag it to the menu.
7. When you finish adding macros, click **Close**.

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

Customizing the Appearance of the Screen

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

Customize Display of Toolbars

You can add tooltips, display shortcut key information, or display large icons as ways of customizing the toolbar display.

Procedure

1. Choose the **Tools > Customize** menu item and click the **Options** tab.
2. Make one or more of the following selections:
   - Select the “Show ToolTips on toolbars” check box to display tooltips for toolbar buttons or clear the check box to hide the tooltips.
   - Select the “Show shortcuts in ToolTips” check box to display shortcut key information or clear the check box to hide it.
   - Select the Large Icons check box to display large icons or clear the check box to retain normal sized icons.

Tip
To show or hide a toolbar in the SailWind interface, use the **Toolbars and Menus** tab. See “Editing a Custom Toolbar” on page 549.

Customize Display of Windows

You can add animation, shadows, and delays to your window display from the **Options** tab of the Customize window (**Tools > Customize** menu item, **Options** tab).

You can make the following customizations to the display of windows:

- To change the way a menu is displayed, select an animation from the Menu animations list. For example, the “Unfold” animation displays part of the menu and an arrow you can click to display the rest.
- To show shadows on menu items, select the “Menu shadows” check box.

- To display your recent menu selections before other menu items, select the “Show recent commands first” check box.
- If menus display recent commands first, you can display the full list of menu items after a delay. Select the “Show full menus after delay” check box.
- If you have a shortcut made up of a number of keys, you can delay its execution until you press the Enter key by selecting the “Wait until enter before executing long shortcuts” check box.
- To delete the record of commands you have used and restore the default set of commands to the menus and toolbars, click **Reset my usage data**. This option does not undo any explicit customizations you have made.

Resizing the Layers List

You can change the width of the Layers list on the Standard toolbar.

Restrictions and Limitations

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

Procedure

1. Choose the **Tools > Customize** menu item to open the Customize dialog box.
2. Click on the Layers list box *on the toolbar.* (Ignore the Customize dialog box.)
3. Resize as needed.
4. Click **Close** in the Customize dialog box.

Organizing Windows

You can customize the way windows appear in your workspace.

Showing and Hiding Windows

Several windows display when you start SailWind Router for the first time. You can show and hide any of these windows.

Tip
You can also hide windows automatically. See "Hiding Windows Automatically" on page 567.

Procedure

To hide a visible window, click on its toggle switch on the **View** menu, or on the **Hide** command from the Window Position menu on the window's title bar. To show a hidden window, click on its toggle switch on the **View** menu.

Hiding Windows Automatically

You can set a window to hide automatically so that it appears only when you hover the pointer over the hidden window's tab.

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's 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, extending over 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 so that the thumbtack icon points downward again.

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 below it.

Restrictions and Limitations

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

Procedure

Double-click the window's title bar. The window detaches and you can move it to any part of the screen.

Tip
To undo the floating, see "Attaching Windows to the Current View" on page 569.

Attaching Windows to the Current View

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

Docking to the Last Location

You can attach or dock a floating window to its last docked location.

Procedure

Double-click the window's title bar. The window reattaches to the interface.

Docking to a New Location

You can drag a floating window and dock it from one location to a new one.

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 113:

   Figure 113. Window Dragging Graphic

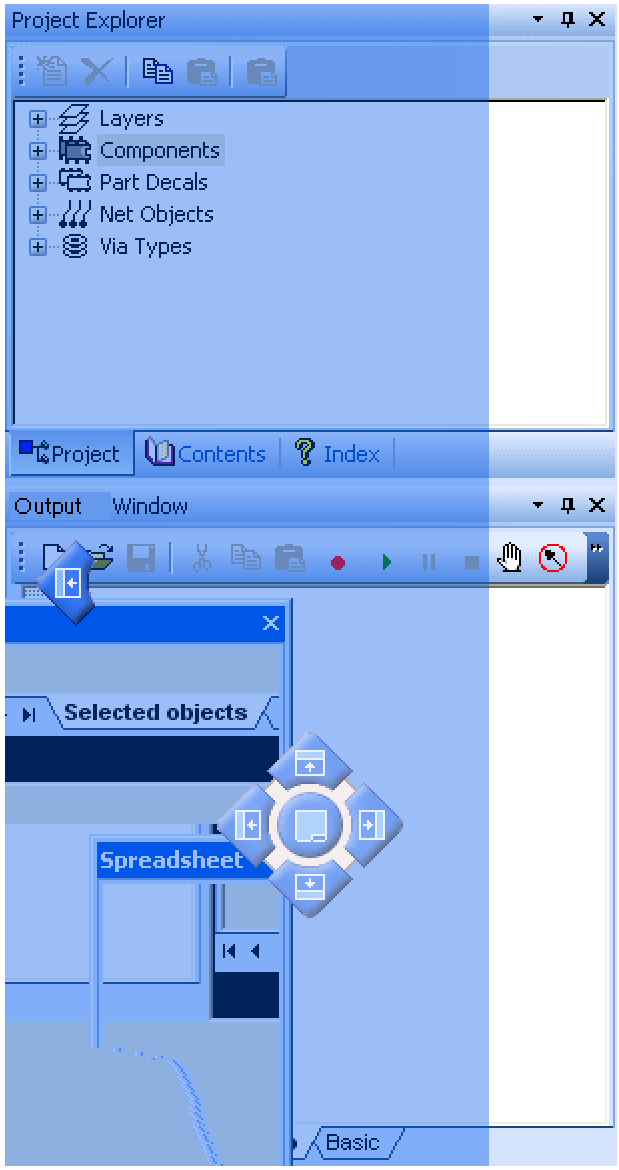
   

   > **Tip**
   > A similar group of arrows appears in a group near the center of the screen. Ignore that group of arrows for this procedure.

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

   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 114.

Figure 114. 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 Windows within Other Windows

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

Two Windows Sharing One Window Space
Creating Tabs Within Windows

Two Windows Sharing One Window Space

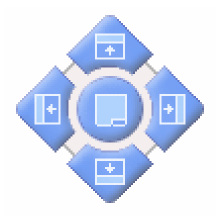
You can embed a window within another window's space.

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 you are dragging, as shown in Figure 115. Depending on the window you are dragging, the group of arrows may also have a tab graphic in the center.

   Figure 115. Dragging a Window— Arrow Group

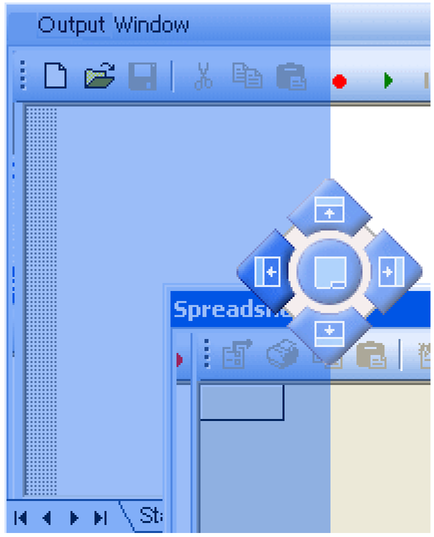
   Tip
   A similar group of arrows appears at the sides of the user interface. Ignore those arrows for this procedure.

   ---

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

   A transparent colored block appears along the side of the window you are dragging, as shown in Figure 116. This block indicates where the window will be docked when you release the mouse button. For example, if you hover over the left arrow, a block appears along the left side of the Project Explorer.

Figure 116. Dragging and Docking a Window



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

Results

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

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

Creating Tabs Within Windows

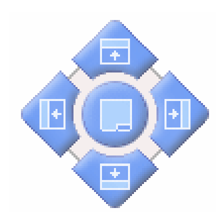
You can embed a window as a tab in another window.

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 you are dragging, as shown in Figure 117. Depending on the window you are dragging, the group of arrows may also have a tab graphic in the center.

Figure 117. Dragging and Docking a Window— Arrow Commands

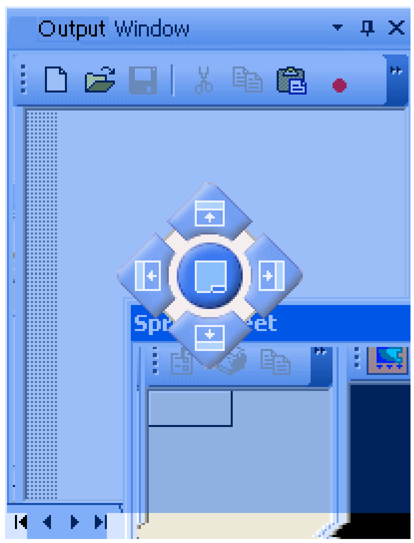


Tip
A similar group of arrows appears at the sides of the user interface. Ignore those arrows for this procedure.

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

   A transparent colored block appears over the window you are dragging, as shown in Figure 118. 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 118. Dragging a Window— Transparent Block

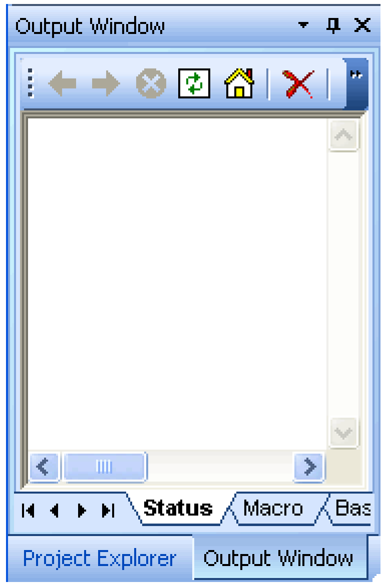


3. Release the mouse button while hovering over the tab.

Results

The window is embedded as a tab within a window, as shown in Figure 119. You can click each tab to access each window.

Figure 119. Window Embedded as a Tab



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

Managing Window Tabs

Some of the windows in the user interface contain tabs. You can reorganize the tab grouping to suit your needs or preferences.

Rearranging Tabs in a Window

You can change the order of tabs in a window.

Restrictions and Limitations

You can only move tabs that you have embedded in other windows. In windows that have tabs by default (such as the Output Window in SailWind Router), you cannot move the tabs. You can only rearrange them. For information on rearranging tabs, see "Rearranging Tabs in a Window" on page 575.

Procedure

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

Moving Tabs Between Windows

You can move a tab from one window to another.

Restrictions and Limitations

You can only move tabs that you have embedded in other windows. In windows that have tabs by default (such as the Output Window in SailWind Router), you cannot move the tabs. You can only rearrange them. For information on rearranging tabs, see "Rearranging Tabs in a Window" on page 575.

Procedure

1. Drag the tab to a new window. When you start dragging, the tab automatically behaves like a window.
2. Place the tab as you would a window.

Related Topics

Embedding Windows within Other Windows

Organizing Windows

Converting Tabs to Windows

You can create a new window from a tab.

Procedure

1. Drag the tab. When you start dragging, the tab automatically behaves like a window.
2. Release the mouse button. Make sure the pointer is not over any arrow graphics.

   You now have a floating window.
3. Place the tab as you would any floating window.

Related Topics

Organizing Windows

Crash Detection

Crash Detection provides the capability to capture and report unexpected terminations of the software.

If crash detection is enabled, the Error Detected dialog box on page 414 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 SailWind Software Customer Support. You can also attach feedback to this report, and optionally, the Basic Media Wizard (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 *SailWindRouter.ini* file (located in the *C:\<install_folder>\Settings* directory); 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.

Glossary

absolute coordinates

Coordinates of a location based upon their distance from the origin (coordinates 0,0) of the design area.

accelerator keys

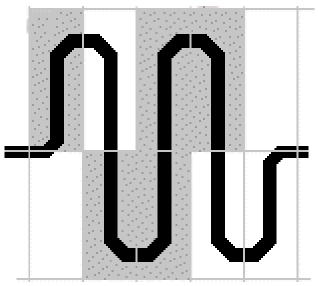
Key sequences used to invoke commands and change system settings without using the mouse. Accelerator keys are called shortcut keys in the SailWind product documentation.

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

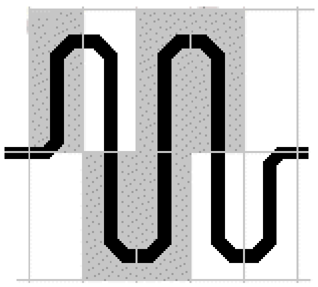
A trace pattern resembling a signal wave that adds length to traces. The trace patterns are contiguous and do not include layer changes.



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.

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.

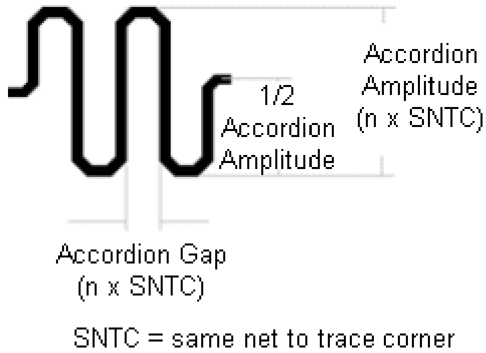
alphanumeric pins

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 option

The Base Option, in Assembly Variants, contains all of the common components in all of the existing variants; in other words, it contains a filtered database. If you uninstall or substitute components in a variant, they are removed from the Base Option. Therefore, the Base Option, because it contains only installed options, is also a subset of the raw database. You can use the Base Option to view all of the items in all of the variants, or the base of all variants.

The Base Option always exists; you cannot delete it.

base part

When making a union, the part type of the first selected part. Base parts can either be left in position and joined by secondary parts, or repositioned to imitate the first selected prototype part.

baseline dimensioning

A type of dimensioning in which a series of dimensions have a common start point, such as datum dimensioning.

basic units

A basic unit is the smallest unit of measurement in a SailWind 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

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

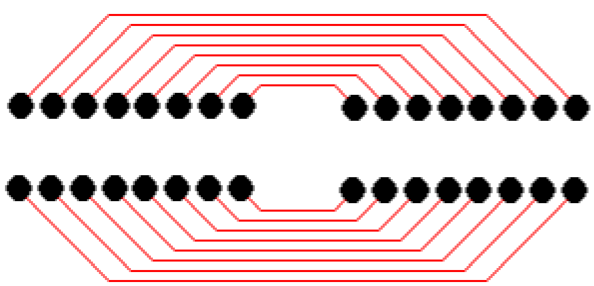
A series of connections that share a common use, such as memory array or data array, and are usually routed parallel to each other.

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

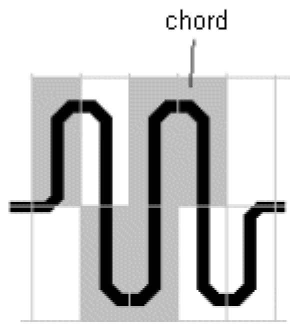
The packaging configuration in which a chip is bonded directly to a circuit board or substrate.

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.

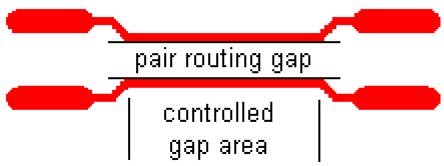
container application

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:

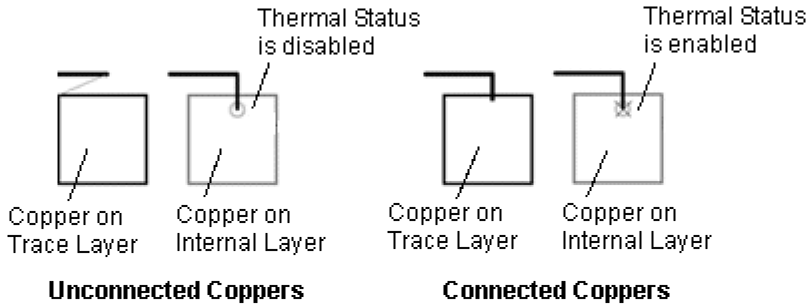
- Minimum/maximum length
- Matched length
- Differential pairs

converting database

The process that converts a non-native file, such as an *.asc* file or a *.dxf* file, to a 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 SailWind programs to reflect, in one SailWind application, selections made in another SailWind application.

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

D-codes

Specific numbers assigned to photoplot machine apertures for program identification. D-CODES are included in the aperture table.

decal

The physical representation, or footprint, of a part.

decal copper

Open, closed, or associated copper produced within the physical representation of a component.

decal text

Documentation text produced within the physical representation of a component.

default component

The original component, before being replaced in the current assembly variant. The default component is always in the raw database, but not necessarily in the Base Option.

See also *active component*

default layer mode

A layer mode in which a design can consist of up to 30 electrical layers, or a combination of electrical and nonelectrical layers. You change from default layer mode to increased layer mode by clicking the Max Layers button in the Layers Setup dialog box.

default.asc

The ASCII file accessed for new file creation. This file provides startup design information such as grid sizes, default colors, or other information.

default.cam

A file usually found in the *C:\<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.

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

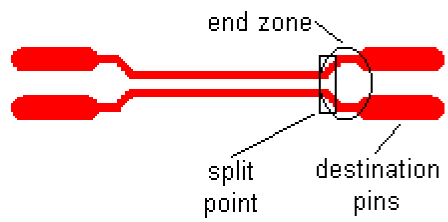
The mode initiated in the routing shortcut menus that, while routing, ends a partial route without a via.

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.

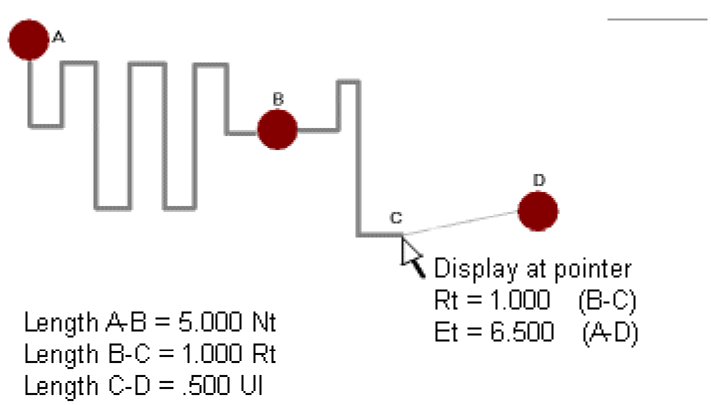
Esc

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 SailWind 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.

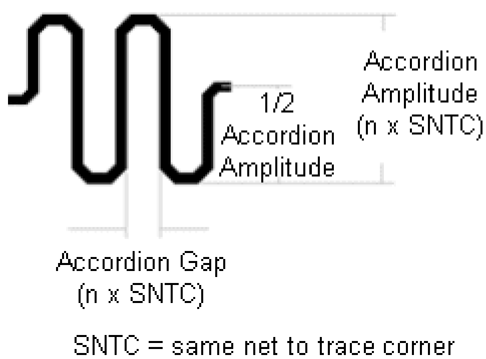
free disk space

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

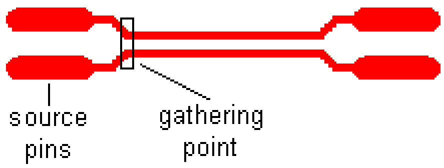
Gate swapping involves exchanging the entire element for a like element.

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 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 SailWind 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

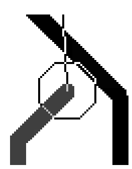
A collection of pin pairs that share common design rules.

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.

installed options

SailWind product features that you have bought and installed as part of the software package.

instruction pointer

A small yellow arrow in the Output window gutter that indicates the current line in a script or macro.

insulator

A material used to inhibit heat or electrical properties, such as current flow.

integrity check

A database check runs whenever a *.job*, *.dxf*, or *.asc* file loads. You can also initiate an integrity check while you are working. 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, Fanout Rules, Pad Entry Rules. The Latium checks include:

- component clearance rules
- component routing rules
- differential pair rules
- via at SMD rules

layer biased net

A layer biased net is any net with a design rule specifying a layer bias to one or more, but not all, electrical routing layers

layer pair

The assignment of two routing layers to switch between using the Layer Toggle command. On two layer boards, the toggle is automatically set between 1 (top) and 2 (bottom).

layer toggle

To switch between layer pairs while routing.

layers

A standard CAD database feature that separates graphical information into sheets of similar information such as dimensions, construction lines, or text. For PCB applications, this 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

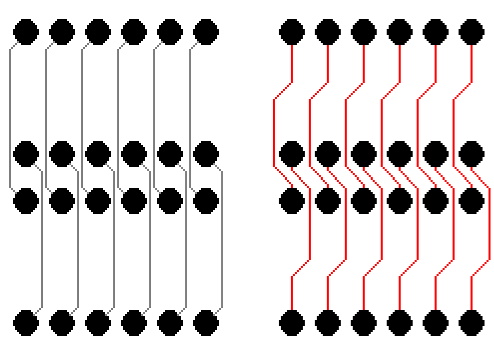
There are three material conditions when creating component decals. They are Maximum (providing the most robust solder joint), Nominal (providing a general purpose solder joint) and Minimum (providing the least possible solder joint for very dense designs).

MCM

An acronym for multichip module.

memory pattern

A collection of routes between memory devices that form a distinctly repeatable pattern.



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 in the *SailWind Layout Guide*

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.

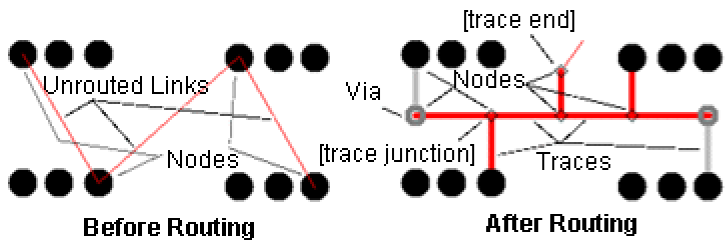
next.log

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

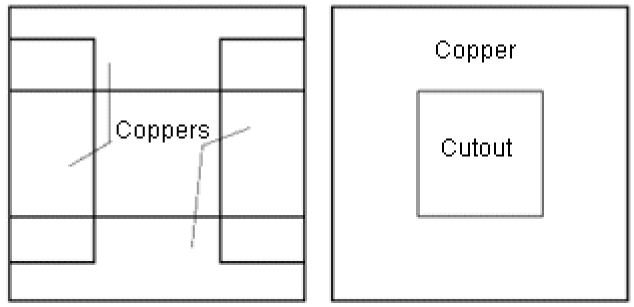
Rearranging placed parts and/or swapping pins and gates on parts to minimize trace lengths and reduce the number of vias required for routing.

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.



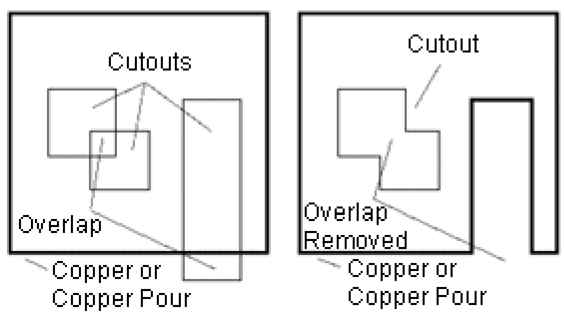
Overlapping Coppers

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.

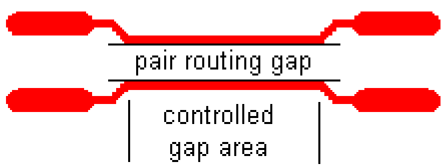
Pad oversize is measured from the center of the pad, not the perimeter. For example, if you have a 3 mil oversize, the measurement is actually1.5 mils in each direction from the center of the pad.

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.

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 of SailWind Layout or Via Biasing properties on page 169 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.

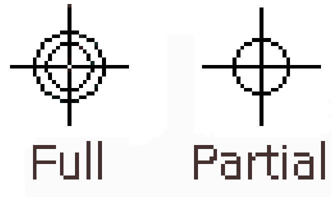
route

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).



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*	• *patterns pass*
• *fanout*	• *route pass*
• *miters pass*	• *test point pass*
• *optimize 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.

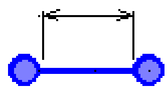
rules set

A specific set of user-assigned nondefault rules such as pin pair, groups, or classes.

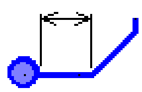
same net checking

Checks clearances between objects along the same net, as specified in the Clearance Rules dialog box. Object to object checking includes:

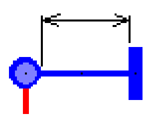
- Pad edge to pad edge.



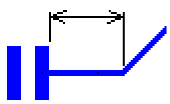
- Pad edge to inside corner of trace.



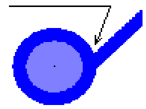
- SMD edge to pad edge.



- 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

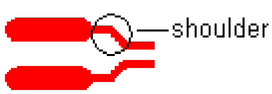
A key sequence that starts a command directly from the keyboard and without navigating through menus.

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

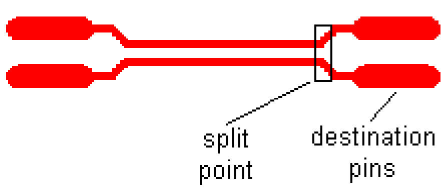
The command that creates a new corner at the pick point of the selected trace, enabling it to be rerouted.

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

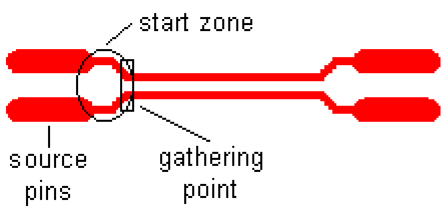
A program used to recommend set statement settings to properly adjust port access times for a security key.

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.

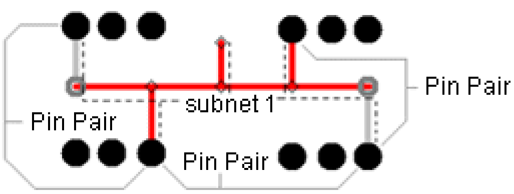
subnet

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 120. One subnet in a net

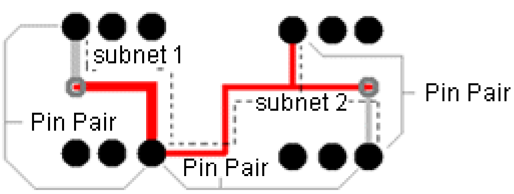


subnets

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 121. 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 SailWind 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

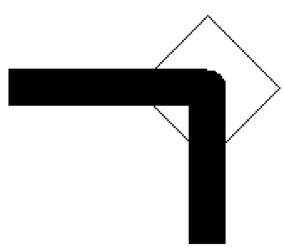
A trace that branches into another.

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.

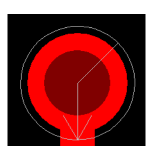
terminator

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

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.

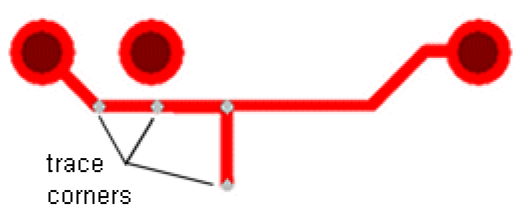
If pin pair rules are reported, the estimated total length of the pin pair is shown; otherwise, total length for nets is reported.

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.

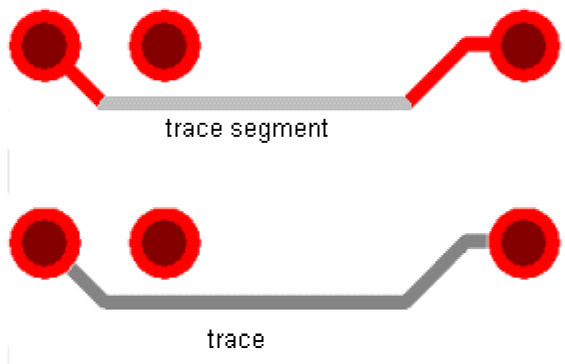


trace paths

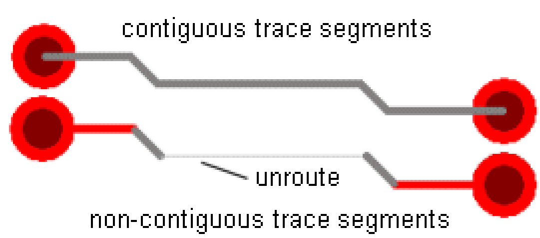
A continuous sequence of trace segments in the same trace on the same layer. Paths start and end at nodes, and cannot pass through a node.

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.

By default, TrueLayer mode is enabled. To disable it, use the /NTL command-line switch. See Software Launch Options.

TTL

Acronym for Transistor-Transistor Logic.

tune pass

This autorouting pass adjusts the length of length-controlled traces. The pass examines trace lengths for only completely routed nets or pin pairs. The pass analyzes the current length of each net or pin pair if length rules and length control are enabled, based on the following conditions:

- If the cumulative length of the adjacent trace segments is within the range of minimum and maximum trace length, the tune pass skips the trace and does not adjust it.
- If the trace is longer than the maximum trace length, the tune pass rips it up and places it in a queue for routing.
- If the trace length is less than the minimum trace length, the tune pass changes the length by adding accordion patterns.

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 *#unique_1218* 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

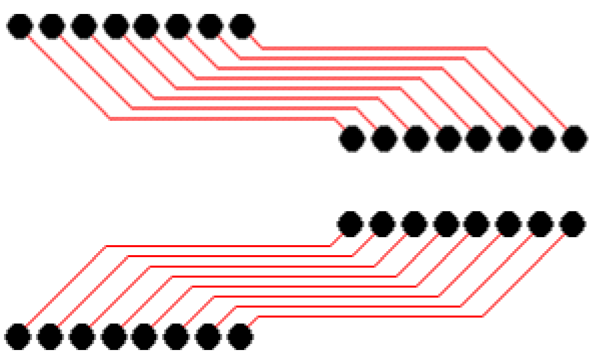
The actual work area where a design is created.

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.