PAIZI

SailWind Logic Tutorial

派兹

Contact Information

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

Email: market@pzeda.com

Phone: 0755-86703052

Website: www.pzeda.com

Table of Contents

Learning the User Interface Tutorial

The SailWind Logic user interface is designed for ease-of-use and efficiency. SailWind Logic is designed to meet the needs of the power user, while keeping the beginner in mind. SailWind Logic's interface and interaction are similar to other Windows™ applications. You can interact with SailWind Logic using the keyboard, menus, toolbars, and shortcut menus.

In this lesson:

- Keyboard command shortcuts
- Using the workspace
- Panning and zooming
- Selecting objects

Preparation

If it is not already running, start SailWind Logic, and click File > New.

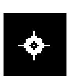
Keyboard command shortcuts

You can use keyboard shortcuts to start commonly used commands or to set options. You will use some of the shortcuts throughout the tutorial.

Keyboard shortcut types:

Type of shortcut	Description
Modeless Commands	Commands invoked through the keyboard. The available commands handle display options, design settings, and mouse click substitutions.
Shortcut Keys	Commands that change settings or execute commands without using the mouse. Windows shortcut keys, such as Alt+F to open the File menu, are also used.

Using the workspace



The default workspace measures 56 inches by 56 inches. The origin of the workspace, or 0,0 coordinate location, is represented by a large white marker. When you start SailWind Logic or open a new file, the origin is located at the lower-left corner of the sheet, at a medium magnification. The large white rectangle represents the area of the workspace occupied by the schematic sheet. For a size B schematic the area is 11 x 17. For a size C sheet it is 17 x 22, and so on.

Pointer position display

As you move the pointer around the workspace, its position, in absolute X,Y coordinates relative to the origin, appears on the Status Bar at the lower right corner of the screen. Throughout this tutorial you will refer to this coordinate data.

1. Place the pointer over the origin point and note the 0,0 reading on the Status Bar.
2. Move the pointer around the workspace and note how the X,Y coordinates change as the pointer position changes.

Set the sheet size

1. **Tools** menu > **Options** > **Design** tab.
2. In the Sheet area, locate the Size list. Leave the current sheet size and click **OK**.

Open a previously saved file

To make it easier to view changes in magnification, open the file named **preview.sch** in the \SailWind Projects\Samples folder.

Tip: If you have made any changes to the previous design, a dialog box appears prompting you to save the old file. Click **No**.

Panning and zooming

Several methods exist for controlling the centering and magnification of a design. In this exercise, you are going to use the mouse.

For two-button mouse operations, the Zoom icon enables and disables Zoom mode. Clicking Zoom from the View menu can also enable it. While in Zoom mode, the pointer changes to a magnifying glass. For three-button mouse operations, Zoom mode is always available using the middle mouse button.

Tip: Because some manufacturers give the center mouse button a specific function, you may need to de-activate default mouse button functionality before you can use these features in SailWind Logic. Change mouse functionality using the Control Panel.

Zoom in and out by placing the pointer at the center of the area and dragging in a specific direction. Pan and Zoom functions are also available using commands from the View menu, using the numeric keypad, and using the scroll bars. See *SailWind Logic Help* for more information on Pan and Zoom functions.

Practice zooming

The following procedure assumes you are using a two-button mouse, except where noted.

1. Click the Zoom button .
   Exception: If you are using a three-button mouse, skip this step.

2. Zoom in.
   a. Click and hold the left mouse button in the center of the area you want to magnify.

   Exception: If you are using a three-button mouse, click and hold the middle mouse button.

   b. Drag the pointer upward, moving the mouse away from you. A dynamic rectangle attaches to and moves with the pointer.
   c. When the rectangle encompasses the area you want to magnify, release the mouse button to complete the operation.
3. Zoom out. Repeat Step 2, but drag the pointer downward, moving the mouse towards you. A static rectangle, representing the current workspace view, appears with a dynamic rectangle representing the new view of the workspace.
4. Practice using Zoom mode to adjust the magnification.

   Tip: To re-establish the original view, click Sheet on the View menu.
5. Click the **Zoom** button again to end Zoom mode.
   Exception: If you are using a three-button mouse, skip this step.

Pan

Position the pointer in the direction you want to pan and click the middle mouse button. The new view changes to make the location of the pointer the center of the view.

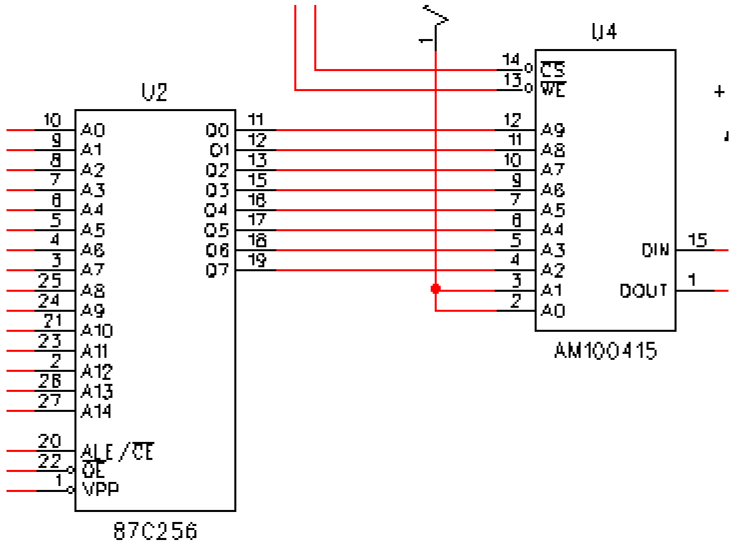
Selecting objects

When selecting parts, place the pointer on the outline of the part, and then click. Clicking inside the body of the part does not ensure selection, especially with larger parts.

Selection Filter

Since many types of objects make up a CAE Decal (schematic symbol) and many additional objects are found on a schematic sheet, a selection filter facilitates the selection of a single type of object.

1. Resize your view to fit an area around component U2 and U4 as shown below.



2. Right-click and click **Filter.**

   Alternative: Press Ctrl+Alt+F to open the Selection Filter.
3. In the Selection Filter click **Anything**. Notice that Parts and Nets are not selected but everything else is selected.
4. Close the Selection Filter.
5. On the schematic, click the connection segment attached to U4 pin 14. The whole net is not selected, just the one connection segment.
6. Right-click and click **Cancel** to deselect the connection segment.
7. Right-click and click **Select Connections**. Click and select the same connection segment. All connection segments leading to the tie dot are selected. Right-click and click **Select Net Instance**. All connection segments of this net instance are selected. Right-click and click **Delete**. The connection is deleted.

   You cannot delete individual connection segments. To delete nets or portions of nets you must select connections, net instances or an entire net and apply the delete command.

   Tips:

   - You can quickly change Selection Filter settings using the filter presets. With nothing selected in the design, right-click, and click one of the Select commands.
   - The Selection toolbar also contains buttons for Selection Filter settings.
   - Right click and click Cancel and Delete or use the Esc (cancel) and Delete buttons on the keyboard instead.
8. Do not save a copy of the file.

You completed the learning the user interface tutorial.

Creating Library Parts Tutorial

You create the library elements that compose a part type in the SailWind library using the Library Manager and the PCB Decal Editor.

In this lesson:

- Understanding the SailWind part type
- Creating a CAE decal
- Creating a new part type

Preparation

If it is not already running, start SailWind Logic.

Understanding the part type

Before you can add a part to a schematic, it must exist as a part type in the library. A part type is composed of three elements:

- The schematic symbol, or CAE decal as it is called in the library.
- A component footprint, or PCB decal as it is called in the library.
- Electrical parameters, such as pin numbers and gate assignments.

An example of a part type is a 7404.

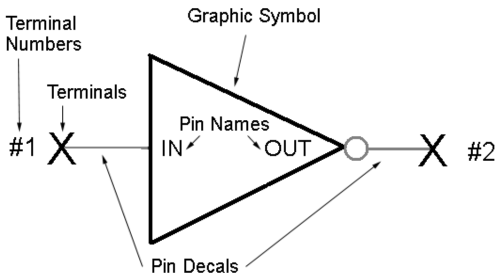
Element	Description
Part type name:	7404
CAE decal:	INV (inverter)
PCB decal:	DIP14 (14 pin dual in line package)
Electrical parameters:	Six logical gates (A through F) using 12 of the 14 pins with one power pin and one ground pin.

You can create a part type in SailWind Logic or in SailWind Layout; however, you can only create a CAE decal in SailWind Logic, and a PCB decal in SailWind Layout.

CAE decals

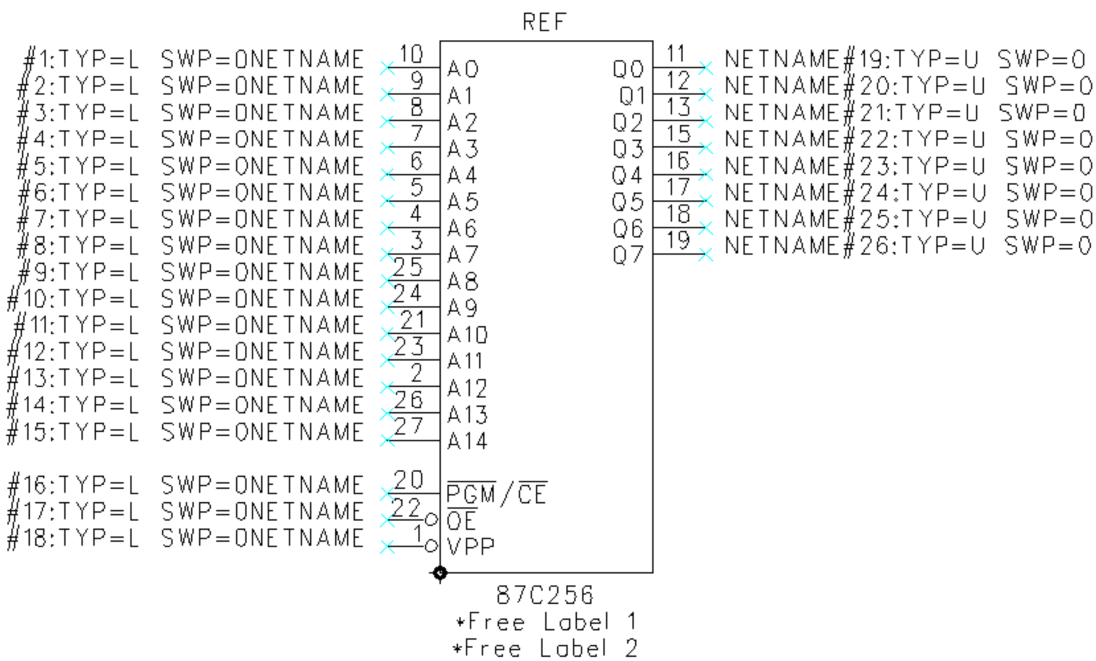
Each part added to a schematic must have a CAE Decal to graphically represent the logical function of the part. While some parts are represented entirely with a single CAE decal, others are composed of multiple instances of the same CAE decal, such as part 7404, the multi-gated inverter mentioned above.

A CAE decal is composed of pin decals attached to terminals placed at intervals around a graphic symbol. This entity represents all or part of the device. The terminals can be assigned pin names to represent the logical function of the pin, such as IN, OUT, or VSS. All elements of a CAE decal are created in the Part Editor.



Creating a CAE decal

In this exercise you will use the CAE Decal Wizard and other features to create a CAE decal for the 87C256 part type shown in the graphic below.



Create a CAE decal

1. Tools menu > Part Editor.
2. On the Part Editor toolbar click the **Edit Graphics** button  , and then click **OK** to confirm the creation of a new CAE decal with a default name.

Once you enter the CAE Decal Editor, several text labels and a CAE decal origin marker appear.

Each of these labels are placeholders for CAE decal-related text objects. The position of these labels on the CAE decal determines where the CAE decal text object appears in the schematic. The origin marker serves as the origin for moving or positioning the CAE decal on the schematic. Select the labels and move them to where you want.

Label	Placeholder
REF	The reference designator
PART-TYPE	The part type (7404, 74LS74, etc.)
*Free Label 1	The first displayed part type attribute
*Free Label 2	The other displayed part type attribute

Tip: An attribute is a specific type of part information that can be included in the part's library description and exported to a parts list. Examples are part manufacturer, package type, and part number.

Create CAE decals with the CAE Decal wizard

1. Decal Editor Toolbar button > CAE Decal Wizard button
2. In the Box Parameters area, type **800** in the Min Width box.
3. In the Left Pins area, select **PIN** as the Pin Decal, and type **19** in the Pin Count box.
4. Type **0** in the Right Pins, Upper Pins, and Lower Pins - Pin Count boxes.
5. Click **OK** to create the CAE Decal.

You now have the beginning of a CAE decal. This process creates a CAE decal with 19 terminals (the small X at the end of the pin decal) each having a PIN type pin decal.

See also: *SailWind Logic Help* for details on the relationship between terminals and pin decals

Add a new terminal

At this stage, the CAE decal is not complete. You must add output pins and modify some of the symbols on the input pins.

1. Add Terminal button **.**
2. Click the **PIN** decal from the Pins list, and click **OK**.
3. With the new terminal attached to the pointer, right-click and click **X Mirror**. This mirrors the terminal on the X-axis, or horizontally.
4. Position the terminal on the right side of the symbol, at X**1000**,Y**1900**, and click to place the terminal.

Add new terminals with step and repeat

After you add a terminal, SailWind Logic remains in Add terminal verb mode; the next new terminal attaches to the pointer. Instead of adding the next terminal as you did above, use the Step and Repeat feature to add several terminals quickly.

1. With the next new terminal attached to the pointer, right-click and click **Step and Repeat**.
2. Select **Down** as the Direction, type **7** in the pin Count box, and type **100** in the Distance box.
3. Click **OK**.
4. Right-click and click **Cancel** to exit the Add Terminal verb mode.

Modify terminals

Now that you have added the output pins, use the other terminal commands to modify the input pin arrangement and complete the CAE decal. The input pins consist of 15 address inputs, 3 control inputs and one extra input pin. The pin decals for two of the control inputs need modification and you must remove the extra terminal.

1. Change Pin Decal button 
2. In the Pin Decal Browse dialog box, click the **PINB** pin decal in the Pins list.
3. Click **OK**. You are now in a pin decal assignment mode. Any terminal that you select is assigned the PINB pin decal.
4. Select the two bottom input terminals on the left side of the part (terminals #**18** and #**19**) to change the pin decal on both from **PIN** to **PINB**.
5. Right-click and click **Cancel** to exit Change Pin Decal mode.
6. Right-click and click **Select Terminals**. Select the fourth terminal from the bottom (terminal #**16**).
7. Right-click and click **Delete**. The terminal numbers following terminal #16 are renumbered to accommodate the deletion.
8. On the View menu, click **Extents** to fit the completed decal in the view.

   Alternative: Press Ctrl+Alt+E to view Extents.

Assign pin numbers and pin names to a gate

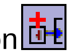
1. Set Pin Name button  **.**
2. In the *Start name of a terminal* dialog box, type **Q0** and click **OK**.
3. Select the top-most output pin (on the right side of the component) to assign it pin name Q0.
4. Select the output just below pin Q0 to assign it pin name Q1.
5. Select the remaining output pins, in order, to assign the names Q2 through Q7.
6. Right-click and click **Cancel** to exit the current naming mode.
7. Repeat steps 1 and 2, typing **A0** in the Start Name dialog box, and clicking **OK**.
8. Select the top-most input pin (on the left side of the component) to assign it pin name A0.
9. Select the input pin just below the A0 pin to assign it pin name A1.
10. Select the remaining input pins, in order, to assign the names A2 through A14.
11. Right-click and click **Cancel** to exit the current mode.

Two of the remaining pins have logical NOT pin name labels (bars over the name). To create this type of text, use the \ character. Assign the remaining pin names.

1. Right-click and click **Select Terminals**. Select the input pin below A14.
2. Right-click and click **Properties**.
3. In the Terminal Properties dialog box, in the Name box type **\PGM\ / \CE** being careful to not use spaces in the pin name.

4. Click **OK**.
5. Click in empty space to deselect the terminal. The remaining pins will be assigned a pin name later in this tutorial.

To finish defining the decal, set pin numbers for the terminals in the same manner as assigning pin names.

1. On the Decal Editing Toolbar, click the **Set Pin Number** button  .
2. In the Set Pin Numbers dialog box, accept the defaults and click OK.
3. Select pins, in the order shown in the 87C256 graphic at the beginning of this tutorial. Stop after you assign a pin number to pin Q2 and proceed to the next step.
4. Select pin Q3.

   Result: After you select pin Q3, it is assigned pin 14. Pin 14 was previously assigned as a signal pin for GND in the electrical parameters.
5. To change Q3 pin number to 15, select pin **Q3** a second time; it is assigned the next highest pin number.
6. Select pin **Q4** to assign it as **16**.
7. Continue assigning pin numbers to the remaining pins.

If you assign the wrong pin number, click the Set Pin Number button again, enter the new number and click the terminal.

Name the CAE decal and part type

1. File menu > Return to Part.
2. Click **Yes** to keep gate changes.
3. On the File menu, click **Save As**.
4. In the Save Part and Gate Decals As dialog box, double-click in the Name of Part box, and type **87C256**.
5. In the Names of Gate Decals box, double-click **NEW_PART** under the CAE Decal 1 column, and type **87C256**.
6. Switch the Library to ****Libraries\preview.
7. Click **OK**. Confirm the overwriting of the CAE decal if it already exists.
8. Click **OK** to the warning of no assigned PCB Decal. Confirm the overwriting of the Part Type if it already exists.

Creating a part type for the CAE decal

Now you will add electrical properties to the 87C256 part type and assign it a PCB decal.

Assign a family type

The first step in creating a part type is to assign the family type for the part.

1. Edit Electrical button > General tab.

2. In the Logic Family list, switch to **TTL** to assign it as the family type for the part type. This simply assigns a default reference designator prefix of U.

Note: The Pin Count displays only 26 pins.

Assign a PCB decal

The next step in creating a part type is to assign the 87C256 CAE decal and assign a 28-pin PCB decal.

1. PCB Decals tab.
2. In the Library list, switch to the **\Libraries\common** library.
3. Type **so*** in the Filter box, type **28** in the Pin Count box, and click **Apply**.
4. Click the **SO28** PCB decal, and click **Assign** to assign the PCB decal as the first (and only) PCB decal for the part type. This has also increased the Pin Count for the 87C256 part type.

Update the pin settings

1. Pins tab.
2. Examine Gate A, the first (and only) gate of the part type.
3. Scroll down the list of pins to Seq. number 17 and 18. These should be pins 22 and 1.
4. For **Seq. 17** (pin 22), double-click in the Name box, and type **\OE**.
   Press **Enter**.
5. For **Seq. 18** (pin 1), double-click in the Name box, and type **VPP**.
   Press **Enter**.

Tip: If you want to use pin swap, terminator assignment, and topology tools in SailWind Layout, the Swap (pin swap) and Type (pin type) values for gates must be set. You can leave them as they are for this exercise.

Assign signal pins

Next, assign the standard power and ground pins. These are called signal pins in SailWind Logic.

1. Scroll down the list of pins to the bottom.
2. For pin **14**, in the Pin Group column, click Unused Pin and change it to Signal Pin.
3. Repeat for pin **28**.
4. Double-click in the Name box for pin 14, and type **GND**.
5. Double-click in the Name box for pin 28, and type **+5V**.

Add user-defined attributes

Finally, you will add user-defined part type attributes.

1. Attributes tab.
2. Click **Add**.
3. Click **Browse Lib Attr**.

4. In the Browse Library Attributes dialog box, click **Description** and click **OK**.
5. Press **Tab** to switch to the Value cell, and type **32K X 8 BIT CMOS EPROM/LATCH**.
6. Click **Add**, click **Browse Lib Attr** again, and repeat the previous steps to add the following attributes and values:

Attributes	Values
Cost	(leave blank)
Part Number	87C256
Manufacturer #1	SIGNETICS
Manufacturer #2	(leave blank)

7. When you complete the addition of all attributes, click **OK** to complete the process of assigning electrical properties and close the Part Information dialog box.

Save the part type to the library

1. On the File menu, click **Save**.
2. On the File menu, click **Exit Part Editor** to exit from the Part Editor and return to the Schematic Editor.

Use the library manager

You can edit existing part types in the library from the library manager dialog.

1. File menu > Library.
2. In the Library list, switch to the \Libraries\preview library.
3. Click the **Parts** button.
4. In the Part Types list, click **87C256**. Notice the image in the preview window. It should be the CAE Decal that is associated with the Part Type.
5. Click **Edit** to bring the part into the Part Editor. Close the Library Manager. You can now edit the part type, its CAE decal, and its electrical properties.
6. On the File menu, click **Exit Part Editor**.
7. Do not save a copy of the file.

You completed the creating library parts tutorial.

Adding and Duplicating Parts Tutorial

Typically, schematic entry requires many add, copy, or delete part operations.

This lesson covers the following topics:

- Adding parts
- Deleting parts
- Duplicating parts

Preparation

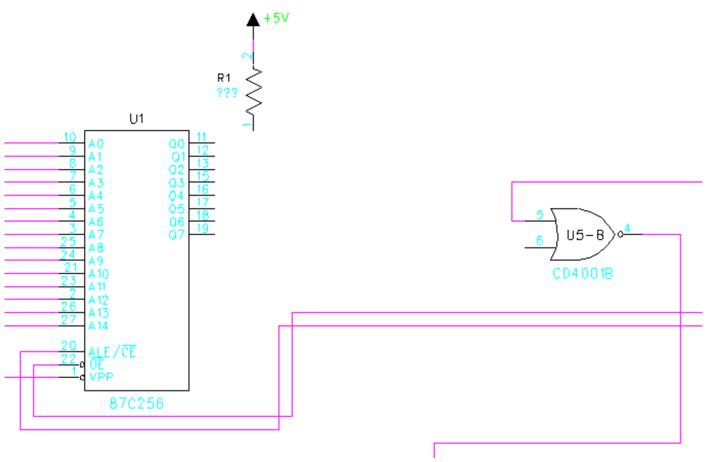
If it is not already running, start SailWind Logic and open the file named **previewstart.sch** in the \ SailWind Projects\Samples folder.

Adding parts

As you add parts, real-time component packaging is provided to automatically assign reference designators.

Resize your view

Before you begin, resize your view to fit an area around U1 and the B gate of U5, as shown in the following figure.



Add a part

Parts are added directly from the library to the schematic. In this exercise, you will add a new AM100415 part to the schematic using the Add Part command.

1. Add Part button  **.**
2. In the Add Part from Library dialog box, in the Filter area, select (All Libraries) in the Library list.

3. In the Items box, type **AM100*** and click **Apply** to perform a wildcard search of the libraries for all parts containing the first five characters, AM100. Results appear in the Items list.
4. Select **AM100415** from the list of parts. The part appears in the preview window.
5. Click **Add**. The AM100415 part attaches to the pointer in the design area.
6. In the Add Part from Library dialog box, click **Close**.

Place the part

Each time you add a new part it attaches to the pointer. Complete the part addition by positioning and placing it. For this exercise you need to place the part precisely.

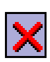
1. Position the part between R1 and U5B at **X6500**,**Y7900**. Use the coordinate display on the Status Bar as a guide. Do not use the D: (delta) coordinates.
2. Click to place the part on the schematic.

   Tip: You can see that U3 is the reference designator for the part. Even though the last reference designator number used for prefix U was 7, it was determined that designator number 3 was unused. Unused numbers are automatically assigned before resuming the numbering sequence.
3. Add more parts in an empty area of the design. When you finish placing symbols, press **Esc**.

Gaps created by deleting or renumbering parts are automatically filled in to minimize part usage. When you add a multi-gated part, unused gates in existing parts are used before a new package is created.

Deleting parts

To remove any multiple instances of AM100415:

1. Delete Mode button  **.**
2. Select a duplicate of the AM100415 part. If you cannot delete the part, right-click and click **Select Parts**.

   Tip: For best results, place the pointer on the part outline when selecting.
3. Repeat these steps to delete any additional parts you added, except U3.
4. Right-click and click **Cancel** to exit Delete mode.

   Alternative: Press Esc to cancel.
5. On the View menu, click **Redraw** to refresh the workspace display.

   Alternative: Press Ctrl+D to redraw.

Duplicating parts

You use the Duplicate command to duplicate parts on the schematic.

1. Duplicate Mode button  **.**

2. Select U3 to add duplicates of the AMD part. Like the Add Part command, the part attaches to the pointer and you enter Move Part mode.
3. Click to place several duplicates on the schematic. Right-click and click **Cancel** to exit duplicating U3.

4. On the Design Toolbar, click the **Select** button to exit Duplicate Mode.

Alternative: Press Esc to exit the mode.

5. Right-click and click **Select Parts**. Use Ctrl+click to select all additional parts you added except U3. Right-click and click **Delete**.

Tip: You can apply changes to multiple objects. All parts selected in the above example, were deleted. Other options for multiple selected parts include attribute additions, attribute visibility changes, and saving to the library. The options available for multiple selections vary according to the types of objects selected.

6. Do not save a copy of the file.

You completed the adding and duplicating parts tutorial.

Adding Connections Tutorial

In this lesson:

- Adding interconnects
- Modifying connections
- Adding connections to power and ground
- Adding connections across pages

Preparation

If it is not already running, start SailWind Logic and open the file named **previewpart.sch** in the \ SailWind Projects\Samples folder.

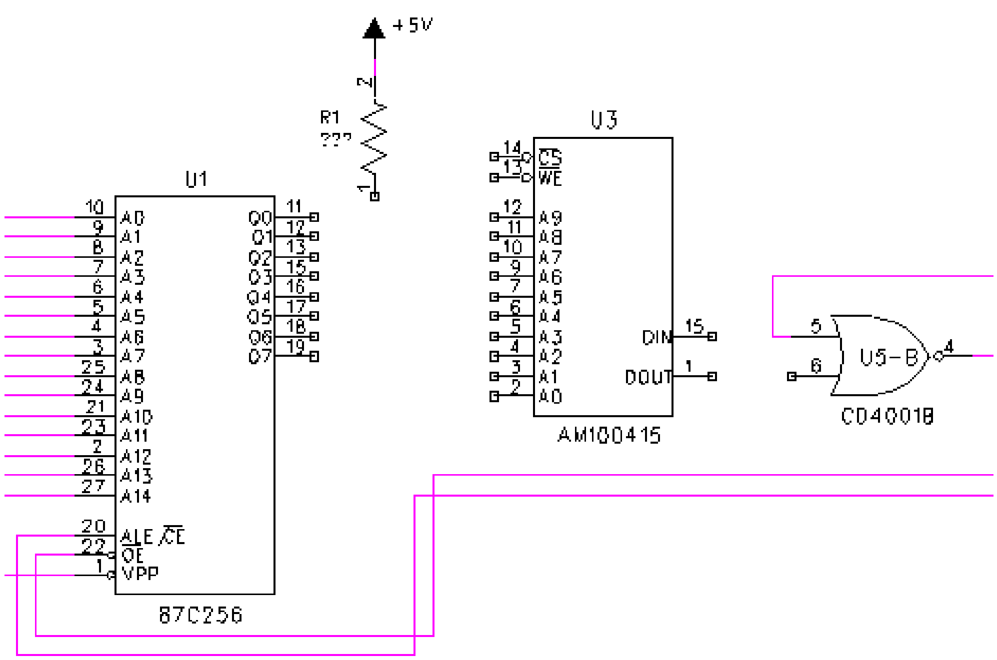
Adding interconnects

To interconnect parts, use the Add Connection command. Each new connection is automatically assigned a system-generated netname.

Add Connection command

Once you add a part, you add the interconnects to complete the schematic. To start a new connection, you will click on the end point of the pin. A connection line attaches to the pin and follows the pointer movement.

1. Add Connection button .
2. Zoom into the area shown below.



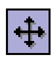
3. Select **pin number 1** on the right side of U3.
4. With the end point of the new connection attached, move the pointer horizontally to the midpoint between U3 pin 1 and U5B pin 6. Click to add a new corner.
5. Move the pointer and notice how the end of the connection remains attached to the pointer and a 90-degree corner is dynamically added as you move in a diagonal direction.
6. Click to add an additional corner above U5B.
7. Remove the last corner you added by right-clicking and clicking **Del Corner** or by pressing **Backspace**.
8. Position the pointer over U3 pin 15 and click to complete the connection. Right-click and click **Cancel** to exit the Add Connection mode.

Modifying connections

You can modify connections using either the Move or Delete commands.

Move a connection

Use the Move command to move the end point of the connection to another pin.

1. Move Mode button  .
2. With nothing selected, right-click and click **Select Connections**.
3. Click at the midpoint of the connection segment attached to U3 pin 15. The connection pulls off pin 15 and the end of the connection attaches to the pointer as you enter Add Connection mode.

   Tip: If you select U3 or a corner of the segment instead of U3 pin 15, press Esc to Cancel and try your selection again.
4. Move the path to U5B pin 6 and complete the connection.
5. Click the **Select** button  on the toolbar to exit Move Mode.

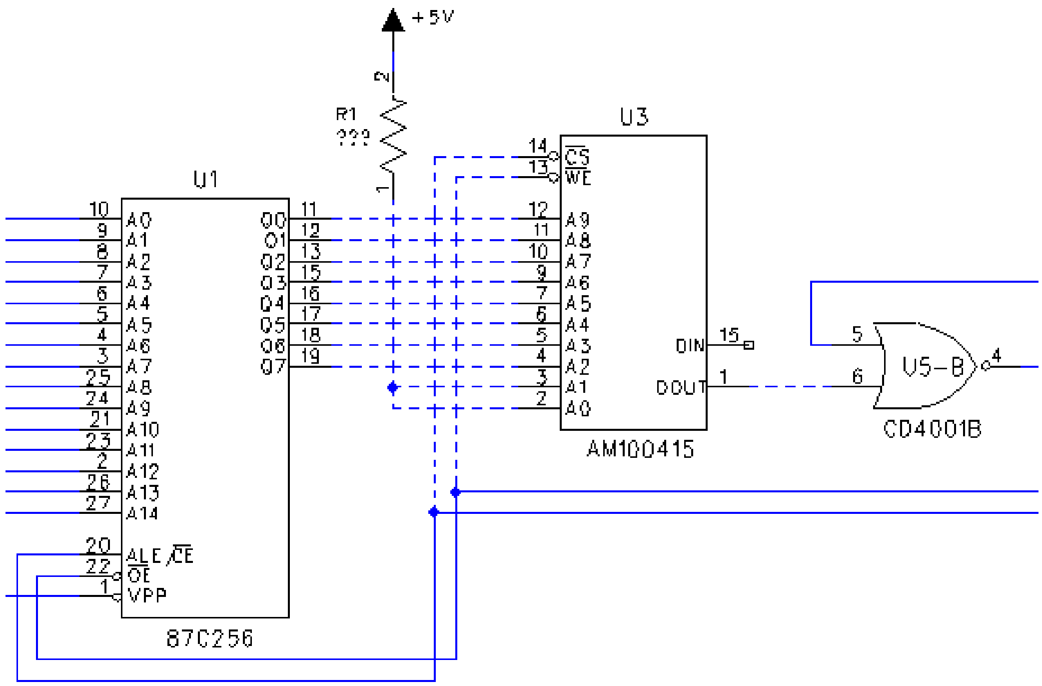
Delete a connection

Use the Delete command to remove the connection.

1. Right-click and click **Select Connections**.
2. Select the connection between U3 pin 1 and U5B pin 6.
3. Right-click and click **Delete**.

Connect U1 and U3

Connect U3 to the other parts (in dashed lines) in the following graphic using the Add Connection, Move, and Delete commands described in the above procedures.



Adding connections to power and ground

You can connect to power or ground with the same Add Connection operations, but you will terminate the connection using a special symbol. The symbol represents a connection to the specified net.

Create power and ground symbols in the Part Editor. Each symbol has a primary symbol and may optionally have several alternate symbols. In addition, you can associate each power or ground symbol (and its alternates) with a net and give each its own graphic symbol. This is useful when the schematic has many power or many ground nets.

Add a power connection

To complete the connections to U3, you must connect pin 15 of U3 to the +12V power net.

1. Add Connection button .
2. Select pin number **15** on U3.
3. Move the pointer horizontally and click to add a new corner.
4. With the connection attached to the pointer, right-click and click **Power**. A power symbol attaches to the pointer.

Choose an alternate symbol

To connect pin 15 to +12V, cycle through the alternate power symbols until you arrive at the +12V symbol.

1. With the special symbol attached to the pointer, right-click and click **Alternate**, until the netname **+12V** appears in the Status Bar.

   Alternative: Press Ctrl+Tab.

   The pin connection also appears in the Status Bar.
2. Click to complete the connection to +12V.

Connect resistors R5 and R2 to ground and power

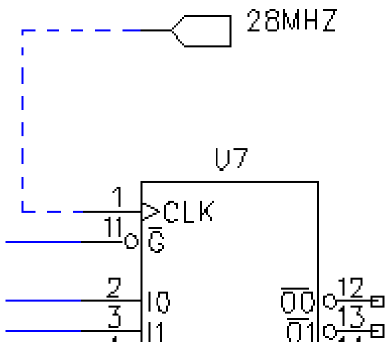
There are also two resistors on the schematic requiring power and ground connections.

1. Use the search modeless command s (search) to locate R5. Type **sr5** and press **Enter**.
2. Select pin number **2** on R5. A connection attaches to the pointer.
3. Right-click and click **Ground**. A ground symbol attaches to the pointer.
4. Position the symbol just below and to the right of R5 and click to complete the connection.
5. Locate R2 by typing **sr2** and pressing **Enter**.
6. Select pin number **2** on R2. A connection attaches to the pointer.
7. Before you add the connection to power, you must enable the display of the netname so it is visible when the power symbol is placed on the schematic. Right-click and click **Display PG Name**.
8. Right-click and click **Power** to connect R2.2 to power.
9. With the special symbol attached to the pointer, right-click and click **Alternate**, or press **Ctrl+Tab** repeatedly until the netname **+5V** appears in the Status Bar.
10. Position the symbol directly above R2 and click to complete the connection.

Adding connections across pages

Off-page symbols are added to connections to make indirect connections between net instances on the same sheet or across multiple sheets. Off-page symbols are added in the same manner as power and ground symbols.

In this section, you will add an off-page connection (in dashed lines in the graphic below).



1. Locate U7 by typing **su7** and pressing **Enter**.
2. Select pin number **1** on U7.
3. Move the pointer horizontally and click to add a corner in the connection. Move the pointer up and to the right just above the U7 reference designator.
4. Right-click and click **Offpage**.
5. Right-click and click **X Mirror**.
6. Click to place the off-page symbol, and
7. In the Add Net Name dialog box, type **28MHZ**, and click **OK**.
8. Do not save a copy of the file.

You completed the adding connections tutorial.

Adding Buses Tutorial

Buses are used to quickly add connections for a collection of sequentially named nets, such as an address or data bus.

In this lesson:

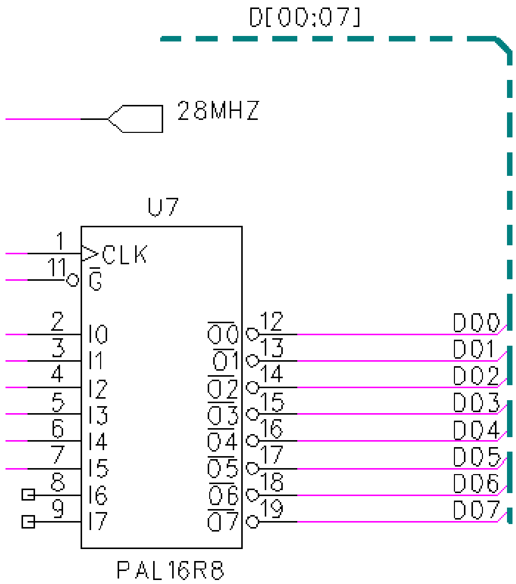
- Creating and connecting a bus

Preparation

If it is not already running, start SailWind Logic and open the file named **previewconnect.sch** in the \ SailWind Projects\Samples folder.

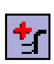
Creating and connecting a bus

In the following sections you will add a bus and connect U7 to the bus (in dashed lines in the graphic below). Buses are used to streamline net connections in a schematic for easy readability. Nets that commonly connect from one component to another are "merged" into a bus to make viewing of schematic connections clearer.



Create a bus

Create new buses by using the Add Bus command.

1. Click the **Add Bus** button  on the Design toolbar.
2. Using the coordinate display on the Status Bar as a reference, position the pointer at **X9400**, **Y4400**.
3. Position the pointer at **X8100**, **Y6200** and double-click, or right-click and click **Complete** to complete the definition of the bus.

4. In the Add Bus dialog box, click **Bit Format**.
5. In the Bus Name box, type **D[00:07]** to name the bus and click **OK**. The bus name label attaches to the pointer.
6. Position the label just above the horizontal segment of the bus and click to place it.
7. Right-click and click **Cancel** to exit Add Bus mode.

Bus naming conventions

The bus labeled in this exercise uses the Bit Format – a series of nets with consecutive net names. A Mixed Net bus type consists of individual nets that are not always sequential. It can even include a bit format bus along with other nets. Mixed Nets are added individually to the Bus Nets list in the Add Bus dialog box. A bit range for each net is optional.

Connect to a bus

Connect to a bus using the Add Connection command.

1. Add Connection button  **.**
2. Select pin number **12** on U7, and position the pointer over the vertical segment of the bus.
3. Click to add a connection to the bus.
4. In the Add Bus Net Name dialog box, click **OK** to accept the D00 net name.
5. Right-click and click **Move**.
6. Position the net name label over the connection, and click to place it.
7. Repeat steps 3 and 4 to connect pin number **13** on U7 to D01.

   Result: The net name assignment automatically increments, and the position of the net name label is preserved.
8. Click to accept the position of the D01 net name.
9. Right-click and click **Cancel** to exit the Add Mode.

   Alternative: Press Esc.

Use duplicate to step and repeat bus connections

You can use the Duplicate command to step and repeat connections and speed the process of connecting to a bus.

1. Duplicate Mode button  **.**
2. Select the midpoint of the D01 connection you added.

   Result: A copy of the connection attaches to the pointer.
3. Right-click and click **Step and Repeat**.
4. In the Step and Repeat dialog box, select **Down** as the direction, type **6** in the Count box, and type **100** in the Distance box.
5. Click **OK** to complete the connection of U7 to the data bus.
6. Press **Esc** twice to cancel and exit first Duplicate Mode and then Add Connection Mode.

Rename a bus, subnet, or net

You can rename a bus, subnets of a bus, or nets.

1. **Right-click** > **Select Anything** > select a connection (or bus) segment > **right-click** > **Properties**.
2. Change the name of the net or bus by typing a new name in the Net Name box or by switching to another net in the list.
3. Choose a Rename option to set how the rename is applied to the selected connection. There are two options:
   - **This Instance –** affects only the selected connection
   - **All Instances** – every connection instance of the named net in the design
4. Click **OK** to apply the rename.

   Tip: When you change a netname to a netname that already exists, you are prompted to confirm the combining of the nets. Click OK to combine the two nets into one net.

5. Do not save a copy of the file.

You completed the adding buses tutorial.

Editing Design Elements Tutorial

You can change objects, including their placement and properties, at any time in the design cycle.

In this lesson:

- Modifying schematic data
- Grouping objects

Preparation

If it is not already running, start SailWind Logic and open the file named **previewbus.sch** in the \ SailWind Projects\Samples folder.

Modifying schematic data

Use Properties mode to modify your data. You can modify text strings, bus names, reference designators, and most other types of data in your schematic. The properties feature is context sensitive; while you are in the Properties mode, select an object to view its properties.

In the following sections you will use Properties mode to modify various design objects in the schematic.

Rename a net

1. With nothing selected, right-click and click **Select Anything**.
2. Double-click the connection segment of the **28MHZ** net connected to the off-page symbol (above U7).
3. In the Net Properties dialog box, type **24MHZ** in the Net Name box.
4. Click **OK**.
5. Because this net already exists, the message "*Net <24MHZ> already exists - OK to combine nets?*" appears. Click **Yes** to combine the nets.
6. On the View menu, click **Redraw** to refresh the workspace display.

   Alternative: Press Ctrl+D to redraw.

Rename a part

You can use similar operations for renaming nets to rename a part; you select a part and change its reference designator property.

1. Right-click > Select Parts
2. Search for R500 by typing **sr500** and pressing **Enter**. Sheet 1 (Logic) is automatically changed to sheet 2 (Power) and the part is located.
3. Zoom in around R500 and double-click **R500**.

Tip: If the selection filter had been set to Select Anything instead of Select Parts, double-clicking the part would open the Part Properties dialog box and double-clicking the name label would open the Reference Designator Label Properties dialog box.

4. In the Reference Designator area, click **Rename Part**.
5. In the Rename Part dialog box, type **C10**, and click **OK**. R500 is renamed to C10.

Change a part type

The next step to updating the component change for the R500 part is to make it a capacitor. Use the Part Properties dialog box to change its part type to a capacitor.

1. In Part Properties dialog box, in the Part Type area, click **Change Type**.
2. In the Change Part Type dialog box, in the Filter area, in the Items box type **cap12***.
3. Switch the library list to **All Libraries**, and click **Apply**. All of the capacitors available in the libraries starting with cap12 appear in the Part Types list.
4. In the Part Types list, click **CAP1206** and in the **Apply update to** area click **This Part**.
5. Click **OK** to change C10 to a 1206 style capacitor.
6. Click **Close** to close the Part Properties dialog box.

Tip: Notice the capacitor is not oriented properly. Part symbols are built with alternate horizontal and vertical versions. You change the properties for the part again to set the Gate Decal to one of the alternate presentations.

Update the attribute values

To change the capacitance of multiple capacitors, follow these steps.

1. Click and drag a box around all of the capacitors including C10.
2. **Right-click** and **click Attributes.**

   In the Part Attributes dialog box, the Attributes area lists several Names and Values. Currently, the Value box next to the Value name contains ???.

   Tip: You may need to scroll down the list.
3. Double-click in the **???** Value box to edit it.
4. Type **.1 uf** in the Value box and press **Enter**.
5. Click **OK** to complete the edit and automatically update all of the CAP1206 capacitors to .1 uf.

To change all of the resistance values of a type of resistor, follow these steps.

1. Search for R7 by typing **sr7** and pressing **Enter**.
2. Click to select the resistor symbol.
3. Right-click and click **Attributes**.
4. In the Part Attributes dialog box double-click in the ??? Value box next to the Value name and type **10K** and press **Enter**.
5. In the Apply Update To area, click **All Parts This Type**.
6. Click **OK** to complete the edit and automatically update all resistors.
7. Click **Esc** to deselect the resistor.

Grouping objects

You can import and export groups of circuitry to and from other schematics. You can also group, duplicate, move, or delete blocks of circuitry within a schematic diagram. The first step is to define the group you'll work with.

Caution: Do not save the design with any changes you make in the following exercise.

1. **Right-click** > **Select Anything**
2. Click and drag a selection area that encloses the items that you want to group.
3. Right-click and click **Duplicate** to copy the defined group. The new duplicate attaches to the pointer.
4. Move the duplicate to any position on this page of the schematic, and then click to place it.

   Tip: You can save the duplicate to a file for use in other schematics. Select a group of objects, right-click and click Save to File. To paste a group into a design, on the Edit menu, click Paste from File, and select the group file.
5. Do not save a copy of the file.

You completed the editing design elements tutorial.

Assigning Constraints Tutorial

Assigning constraints to your design allows you to easily control and verify critical design areas. Constraint types include clearance, routing, and high-speed rules that can be assigned to nets, layers, classes (collection of nets), groups (collection of pin pairs), or individual pin pairs. You can also assign a default set of design rules that apply to all objects not having a unique rule.

In this lesson:

- Establishing the layer arrangement for the PCB
- Setting default clearance rules
- Setting net clearance rules
- Setting conditional rules

Preparation

If it is not already running, start SailWind Logic and open the file named **previewchange.sch** in the \ SailWind Projects\Samples folder.

Establishing the layer arrangement for the PCB

You can define layer arrangements for the PCB, including the number of layers, the nets associated with embedded plane layers, layer stackup, and layer thickness.

The tutorial design is a four-layer PCB with two internal plane layers supporting multiple plane nets. In this section, you will define the layer arrangement for the tutorial design using the layer definition tools.

Increase the number of layers

When you first start a design, the default is a two-layer PCB. To change the number of layers:

1. **Setup** menu > **Layer Definition**.
2. In the Layers Setup dialog box, in the Electrical Layers area, click **Modify**.
3. In the Modify Electrical Layer Count dialog box, type **4**.
4. Click **OK** to increase the number of layers from 2 to 4.
5. In the Reassign Layers dialog box, click **OK**.

Set layer arrangement and names

Once you set the correct number of layers, assign the layer types for each layer and enter layer names.

To set the first layer:

1. In the Layers Setup dialog box, select **Top** in the list of layers.

2. In the Name box, type **Primary Component Side**.
3. In the Electrical Layer Type area, click the **Component** layer type.
4. In the Plane Type area, click **No Plane**.
5. Set the Routing Direction to **Vertical**.

To set the second layer:

1. Select the second layer, **Inner Layer 2**, and rename it **Ground Plane**.
2. In the Plane Type area, click **CAM Plane**.
3. Click the **Assign Nets** button.
4. In the Plane Layer Nets dialog box, in the All Nets list, scroll down, click net **GND**, and click **Add** to associate the net with the Ground Plane Layer.
5. Click **OK**.
6. Set the Routing Direction to **Any**.

To set the third layer:

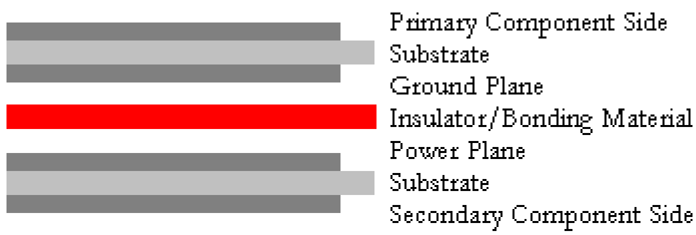
1. Select the third layer, **Inner Layer 3**, and rename it **Power Plane**.
2. In the Plane Type area, click **Mixed Plane**.
3. Click the **Assign Nets** button.
4. In the Plane Layer Nets dialog box, select nets **+5V** and **+12V** in the All Nets list, and click **Add** to associate the nets with the Power Plane Layer.
5. Click **OK**.
6. Set the Routing Direction to **Any**.

To set the last layer:

1. Select the fourth layer, **Bottom**, and rename it **Secondary Component Side**.
2. In the Electrical Layer Type area, click the **Component** layer type.
3. In the Plane Type area, click **No Plane**
4. Set the Routing Direction to **Horizontal**.

Set the layer stackup

A typical layer stackup for a four-layer PCB is composed of a pair of copper clad fiberglass substrates, with an insulator/bonding material in between. The insulator/bonding material is typically a resin sheet that bonds the substrate pairs together when a multilayered board is manufactured. This bonding material is also known as *Prepreg*.
Layer stackup:



Use the Layer Thickness dialog box to set the layer stackup values.

1. In the Layers Setup dialog box, click **Thickness**.
2. In the Layer Thickness dialog box, click **Weight(oz)** in the Copper Thickness Units area.
3. In the Layer list, for the **Primary Component Side**, in the Thickness box double-click and type **2** to reflect that it has a copper weight of two ounces.

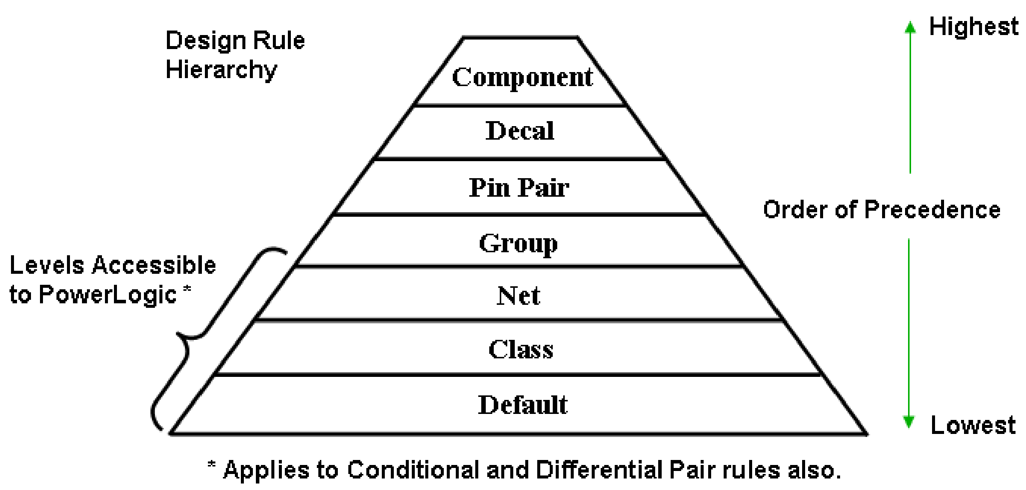
   Tip: 1 oz. of copper weight = .00135" of copper thickness.
4. For the **Ground Plane**, in the Thickness box double-click and type **1** to reflect that it has a copper weight of one ounce.
5. For the **Power Plane**, in the Thickness box double-click and type **1** to reflect that it has a copper weight of one ounce.
6. For the **Secondary Component Side**, in the Thickness box, double-click and type **2** to reflect that it has a copper weight of two ounces.
7. Double-click in the Substrate list box between Ground Plane and Power Plane and switch to **Prepreg** to establish this level as the insulator/
   bonding layer.
8. Double-click in the Thickness box for the Prepreg layer and type **35**.

   Tip: The Dielectric Constant is a value given for manufacturing materials, such as FR-4, to describe electrical characteristics.

13. Click **OK** to close the Layer Thickness dialog box.
14. Click **OK** to close the Layer Setup dialog box.

Setting default clearance rules

You can define clearance, routing, and high-speed rules for each level of the design rule hierarchy.



The Clearance area of the Clearance Rules dialog box contains a matrix of PCB design data. You can specify values for each or all data types in the matrix.

1. **Setup** menu > **Design Rules** > **Default** button > **Clearance** button.

2. Set a global default clearance value by clicking **All** in the upper-left corner of the clearance matrix.
3. In the Input Clearance Value dialog box, type **8** and click **OK**.

   Result: All matrix values change simultaneously.
4. In the Trace Width area, type **6** in the Minimum box, type **8** in the Recommended box, and type **12** in the Maximum box.
5. Type **12** in the Same Net and Other clearance text boxes, with the exception of Trace to Crn box. Set this box to **0**.
6. Click **OK**.

Set the default routing rules

To avoid routing on the plane layers, remove them from the selected routing layers as defined in the routing rules. The Layer Biasing area of the Routing Rules dialog box contains a list of selected routing layers. This list lets you specify which layers are permitted for routing.

1. In the Rules dialog box, click the **Routing** button.
2. In the Selected Layers list, select the **Ground Plane** and press and hold **Ctrl** and click **Power Plane** to add them to the selection.
3. Click **Remove** to prevent routing on the plane layers.
4. Click **OK** to close the Routing Rules dialog box.
5. Click **Close** to close the Default Rules dialog box.

Setting net clearance rules

You can assign net-specific clearances that take precedence over the default rules previously entered.

1. In the Rules dialog box, click the **Net** button.
2. Scroll through the Nets list. Ctrl+click **+5V**, **+12V**, and **GND**. The three selected nets appear in the *Selected* listing under the rule type buttons.
3. Click the **Clearance** button to set the same clearance rules for all three nets.
4. Set a global default clearance value by clicking **All** in the upper-left corner of the clearance matrix.
5. In the Input Clearance Value dialog box, type **10** as the global clearance and click **OK**.
6. In the Trace Width area, type **10** in the Minimum box, type **12** in the Recommended box, and type **15** in the Maximum box.
7. To complete the definition, click **OK**.
8. **Close** the Net Rules dialog box.

Setting conditional rules

When two nets require a specific clearance between each other (to avoid adverse affects on the circuitry), you must define a conditional rule. An example of a conditional rule might be the Underwriters Laboratories (UL) requirements of segregating primary, secondary, and ground nets when alternating current is directly connected to the PCB. You can assign conditional rules between most components of the design rule hierarchy. Conditional rules can exist between nets, nets and classes, classes and classes, nets and layers, and so on.

To assign a net-to-net conditional rule:

1. In the Rules dialog box, click the **Conditional Rules** button.
2. Click **Nets** in the Source Rule Object area.

   Result: A list of nets appears in the Source Rule Object list.
3. Select net **+5V**.
4. In the Against Rule Object area, click **Nets**.

   Result: A list of nets appears in the Against Rule Object list.
5. Select net **+12V**.
6. Click **Create** to define the conditional rule. The new condition appears in the Existing Rule Sets area.
7. In the Current Rule Set area, type **25** in the Object-to-Object box.
8. Close all of the open dialog boxes.

   Result: The rule you just created will keep all objects pertaining to the +5V and +12V nets 25 mils apart.
9. Do not save a copy of the design.

You completed the assigning constraints tutorial.

Creating Reports Tutorial

This lesson demonstrates report options available in SailWind Logic.

In this lesson:

- Checking a net statistics report
- Outputting an unused gates and pins report
- Compiling the bill of materials

Preparation

If it is not already running, start SailWind Logic and open the file named **previewrules.sch** in the \ SailWind Projects\Samples folder.

Creating a net statistics report

The Net Statistics report lists each net in the design. All pin connections for a net are listed and their status as sources, loads, or undefined pins. If pins are defined properly, this report can check the accuracy of a design by displaying questionable net connections.

1. **File** menu > **Reports**.
2. In the Reports dialog box, select the **Net Statistics** check box.
3. Click **OK**. A link to the NetStatistics.rep report appears in the Output Window. Click the link to open the report in the default text editor.
4. Examine the first net. The report indicates whether nets have a source and load. Nets marked [U-1] identify an undefined pin in this group.
5. Close the report file.

Outputting an unused gates and pins report

The Unused report lists any unused gates of a part or unused pins.

1. **File** menu > **Reports**.
2. In the Reports dialog box, select the **Unused check box**, and clear the **Net Statistics** check box.
3. Click **OK**. A link to the UnusedGatesPins.ref file appears in the Output Window. Examine and then close the report.

Tips:

- The Unused Gate List section of the report lists unused gates by part types.
- The Unused Pins List section lists unused individual pins by part types.
- To locate gates that should not appear as unused, use the S modeless command to search for parts by reference designator.

Compiling the bill of materials

The Bill of Materials looks into the part type data for every part in the design and arranges the information in a parts list format. You can customize the format of the report by entering unique column headers and specifying column width:

1. **File** menu > **Reports**.
2. Click the **Bill of Materials** check box and clear the **Unused** check box.
3. Click **Setup**.
4. In the Bill of Materials Setup dialog box, click the **Attributes** tab.
5. In the Field Header column, double-click **Reference**.
6. Type **Ref Des** in the box and press **Enter**.
7. In the Width column, double-click the value **15** next to the Part Name.
8. Type **12** in the box for a new column width and press **Enter**.
9. Click **OK**.
10. Click **OK** in the Reports dialog box. A link to the BillOfMaterials.ref file appears in the Output Window. Click the link to open the report. Review the contents of the bill of materials report, then close the report.
11. Do not save a copy of the file.

You completed the creating reports tutorial.

Linking to SailWind Layout Tutorial

There are two methods of forwarding SailWind Logic schematic data to SailWind Layout:

- Create a netlist that SailWind Layout can read and import it into SailWind Layout
- Use the SailWind Layout link to automatically manage the transfer of data between SailWind Logic and SailWind Layout.

The SailWind Layout link automatically passes design data between SailWind Logic and SailWind Layout and is the preferred method for synchronizing schematic and PCB design databases.

An additional benefit of the SailWind Layout link is inter-tool communication. Use this automation technology to *cross-probe* between linked applications. Cross-probing is having selections of nets, components, or pins made in one application result in the selection of the corresponding object in the linked application.

SailWind Logic is OLE Automation enabled. This capability allows you to develop custom applications using Visual Basic, Microsoft Visual C++, or other tools to extract specific data from a SailWind Logic database.

In this lesson:

- Creating a netlist
- Using the SailWind Layout Link

Preparation

If it is not already running, start SailWind Logic and open the file named **preview.sch** in the \SailWind Projects\Samples folder.

Requirement: This tutorial requires the sample libraries to be in their original search order. The preview library must precede the common library in the SailWind Layout search order or errors will result.

Creating a netlist

Creating a netlist is the basic method for passing schematic data into SailWind Layout to start the PCB design process. A netlist contains a list of the parts, their part types, and all of their net connections. Optionally, it can also include design rules and a layer stackup.

To create the netlist:

1. **Tools** menu > **Layout Netlist**.
2. In the Netlist to PCB dialog box, accept the default settings and click **OK** to create the netlist. A link to the Preview.asc netlist file appears in the Output Window. Click the link to open the file in your default text editor.
3. Review the content of the netlist file and close the file before proceeding to the next section.

Using the SailWind Layout Link

You can cross-probe between SailWind Logic and SailWind Layout with the features of the SailWind Layout Link. Use this feature to perform schematic-driven placement or post-design reviews of the layout.

1. **Tools** menu > **SailWind Layout**.
2. In the Connect to SailWind Layout dialog box, click **New** to start a new SailWind Layout session. It may take a moment for SailWind Layout to start.
3. Once SailWind Layout starts, the two applications are linked and cross-probing between SailWind Logic and SailWind Layout is enabled. The SailWind Layout Link dialog box also appears. Take a moment to rearrange the size and position of your SailWind Logic and SailWind Layout application windows so they each cover half of the screen.

   Tip: You may need to resize your views in SailWind Logic and SailWind Layout after rearranging the windows. Within each program, press **Home** to center the view in the rearranged windows.
4. To reduce the setup required to use SailWind Layout, you will import a file containing some basic PCB elements required for the tutorial.
   a. In **SailWind Layout**, on the **File** menu, click **Import**.
   b. Locate and import the logic tutorial.asc file in the \ SailWind Projects\Samples folder.

Send a netlist to SailWind Layout

Use the SailWind Layout Link to automatically generate and send a netlist to SailWind Layout.

- In the SailWind Layout Link dialog box, on the Design tab, click **Send Netlist** to generate a netlist from SailWind Logic and send it to SailWind Layout. After the process completes, all the components are positioned at the design origin, ready for placement.

  Tip: If a prompt appears asking “Schematic net list may have errors. Do you want to continue?”, click Yes.

Schematic-driven placement

Now that you have a SailWind Layout design with the parts and nets from the schematic, you will prepare the parts for placement. You will also put SailWind Layout in Move mode so part selections made in SailWind Logic result in movement of the selected part.

In SailWind Layout

Use the Disperse Components command to distribute the components around the outside of the board outline.

1. On the Tools menu, click **Disperse Components** and click **Yes** to confirm the dispersion.
2. Click the **Board** button  to fit the view to the board outline.

Enable the move component verb mode.

1. On the toolbar, click the **Design Toolbar** button to open the Design toolbar.
2. On the Design toolbar, click the **Move** button to place SailWind Layout in Move Component mode.

In SailWind Logic

Select the J1 component in SailWind Logic.

1. On the Selection toolbar, click the **Nothing** filter button, then the **Parts** filter button to place SailWind Logic in a mode that enables only component selections.
2. Select any pin of the J1 connector on the left side of the Logic sheet of the schematic.
3. Move your pointer to the SailWind Layout window. You are now moving J1 in SailWind Layout.

In SailWind Layout

Place the component in SailWind Layout.

1. Right-click and click **Flip Side**.
2. Position it at X1650,Y400 by typing **S 1650 400** and pressing **Enter**.
3. Press **Spacebar** to place J1.
4. On the Design toolbar, click the **Select** button to exit Move Mode

Multiple selections in SailWind Logic

You can also move multiple components sequentially by making group selections in SailWind Logic and applying the Move Sequential command in SailWind Layout

In SailWind Logic

Make a group selection in SailWind Logic.

1. With nothing selected, position the pointer in the upper-left corner of the schematic, then click and drag the pointer toward the lower-right corner of the schematic.
2. Release the mouse button when several components are enclosed in the selection rectangle.
3. Once you complete the selection, the components corresponding to your selections in SailWind Logic are selected in SailWind Layout.

In SailWind Layout

Move the components in sequential order.

1. Right-click and click **Move Sequential**.
2. Click **Yes to All**.

3. Click to place the first component.

   Result: The next component attaches to your pointer and can be moved.
4. Continue to place parts until no more components attach to the pointer.

Layout-driven selections

You can also drive selections in SailWind Logic from selections made in SailWind Layout.

In SailWind Logic

Enable SailWind Logic to receive selections made in SailWind Layout.

1. In the SailWind Layout Link dialog box, click the **Selection** tab.
2. Select the **Receive Selections** check box to enable SailWind Logic to receive selections made in SailWind Layout.

In SailWind Layout

Select the Y1 component in SailWind Layout.

1. Use the search and select modeless command to locate and select oscillator Y1 by typing **ssy1** and pressing **Enter**.

   Result: The Y1 component is selected in SailWind Layout and SailWind Logic.
2. Search and select C3 by typing **ssc3** and pressing **Enter**.

   This demonstrates how SailWind Logic responds to selections in SailWind Layout and automatically changes sheets to bring the part in view.
3. Search and select U6 by typing **ssu6** and pressing **Enter**.

   Since U6 is a multi-gated part, all gates of the part are selected in SailWind Logic. Each gate of the device is listed independently in the Search and Select box on the Selection toolbar.

In SailWind Logic

Choose a specific instance of U6.

1. Scroll through the items in the Search and Select list on the Selection toolbar and click **U6B**. The selection changes to only U6B. This is how you manage selections of multi-gated parts in SailWind Logic.
2. Do not save a copy of the file.
3. Close the SailWind Layout Link dialog box and SailWind Layout without saving any changes.

You completed the linking to SailWind Layout tutorial.

Updating a Schematic with Design Changes Tutorial

In the typical engineering environment, design modifications are documented by an Engineering Change Order (ECO). Implementation of these changes in the design may require pin and gate swaps, part deletions or additions, net deletions or additions, renaming of components, renaming of nets, or part type changes. SailWind Layout provides tools to implement these modifications quickly and records them accurately for documentation and backward annotation to the schematic.

ECO changes performed in SailWind Layout are recorded in an ASCII file with a *.eco extension. This file, in its native format, can be read into SailWind Logic to annotate the schematic with changes made in SailWind Layout.

In this lesson:

- Importing an ECO file
- Compare Files and Export an ECO file

Preparation

If it is not already running, start SailWind Logic and open the file named **preview.sch** in the \ SailWind Projects\Samples folder.

Importing an ECO file

Examine a few of the reference designators in the schematic. Now import an .eco file.

1. **File** menu > **Import**.
2. Click **No** if the message "Save file before loading" appears.
3. In the file type list, click **ECO Files (*.eco)**.
4. Select the **previewassy.eco** file in the \ SailWind Projects\Samples folder, and then click **Open**.

   Result: All reference designators on the schematic are updated. The schematic is redrawn and a message informs you that the ECO process is complete.
5. Click **OK**.

Compare files and export an ECO file

You use the Compare/ECO command to compare schematic and design files and create an ECO file for updating SailWind Layout.

1. Open the file named **previeweco.sch**. Do not save changes to the preview.sch file. An extra capacitor (C11) has been added to the tutorial design.
2. On the Tools menu, click **Compare/ECO**.
3. In the Compare/ECO dialog box, click the **Documents** tab.
4. In the Original Schematic Design to Compare area, click **Browse**. Select the **preview.sch** design and click **Open**.
5. In the Output Options area, click **Browse**. Set the output file to be named previeweco.eco in the \ SailWind Projects\Samples folder and click **Save**.
6. Click **Run**. A link to the previeweco.eco file appears in the Output Window. Review and then close the file. One or more ecogtmp.err files may open providing information about the files being compared. You can close the files.
7. Start **SailWind Layout** and open the file named **preview.pcb** located in the \ SailWind Projects\Samples folder.
8. On the **View** menu, click **Extents**.
9. On the **File** menu, click **Import**.
10. In the file type list, click **ECO Files (*.eco)**, select the **previeweco.eco** file in the \ SailWind Projects\Samples folder and click **Open**.
11. Click **OK**. A new capacitor C11 will be located near the origin.
12. Close SailWind Layout without saving any changes.

Automatic Forward Annotation Using SailWind Layout Link

You can use SailWind Layout Link to automatically forward annotate changes in your schematic to SailWind Layout with the click of a button.

1. Open the file named **previeweco.sch**. Do not save changes to the preview.sch file. An extra capacitor (C11) has been added to the tutorial design.
2. On the Tools menu, click **SailWind Layout**.

   Tip: If SailWind Layout is not already open, the Connect to SailWind Layout dialog box appears. Click **Open** to start a new SailWind Layout session with the original design. In the File Open dialog box, select the **preview.pcb** file and click **Open**.
3. In the SailWind Layout Link dialog box, click the **Design** Tab.
4. **Optional:** If you want to check the design differences before updating, click the Compare PCB button. The two versions are compared and differences written to *logic.rep* in the \ SailWind Projects folder. To see the report, click the *logic.rep* link in the Output Window.
5. On the **Design Tab:**

a. If needed, check the **Compare Design Rules** and **Show Net List errors report** check boxes.

b. Click the **ECO To PCB** button to send the changes.

Tip: While the SailWind Layout Link dialog box is open, you can cross-probe.

6. In SailWind Layout, a new capacitor C11 will be located near the origin.
7. Close SailWind Layout without saving any changes.

You completed the ECO tutorial.

Printing and Plotting Schematics Tutorial

This lesson demonstrates how to print schematics in SailWind Logic.

In this lesson:

- Choosing what appears in the printout
- Setting plot orientation and printing

Preparation

If it is not already running, start SailWind Logic and open the file named **preview.sch** found in \ SailWind Projects\Samples.

Choosing what appears in the printout

All items in the plot of a schematic sheet can be enabled or disabled for printing.

1. **File** menu > **Print**.
2. In the Print dialog box, click **Options**.
3. Ensure that both sheets in the design are in the Sheets to Print list. If not, click **Add All**. They are selected for plotting.
4. Clear the **Part Text** check box. This removes the pin names from all the parts in the plot.

Setting plot orientation and printing

Check the plot orientation for optimal coverage of your printed page. In the Options dialog box, the Preview area displays the current orientation setting of your printer. The white area represents your page, while the blue box represents the schematic and how it will display on your printed sheet.

1. In the Positioning group box, switch the **Orientation** options until the blue box best fits the sheet, and click **OK**.
2. Set up your printer as usual and click **OK**.
3. Do not save a copy of the file.

You completed the printing and plotting schematics tutorial.

Managing Multiple Sheets Tutorial

SailWind Logic offers many features to manage one page schematics or schematics with hundreds of pages. Use the sheet manger to add, delete, rename or change the order of sheets in a schematic. Create custom sheet borders and title blocks, store them in the library and add them automatically to each new sheet.

To manage large nets distributed over several sheets, add off page references for each instance of a net. Use the hierarchical features of SailWind Logic to organize your schematics with hierarchical blocks.

Use the copy, paste and save to file commands to copy and reuse portions of a schematic across pages or across different schematic designs.

In this lesson:

- Creating a custom sheet border
- Adding a sheet
- Sheet references
- Copying a portion of a schematic
- Hierarchical design

Preparation

If it is not already running, start SailWind Logic.

Creating a custom sheet border

When you start a new design, a default sheet border is added to each new schematic sheet. You have the option to create custom borders and title blocks and set them as the default border.

Modify the default border

1. **File** menu > **New**.
2. **Tools** menu > **Options** > **Design** page.
3. Ensure the Sheet Size is set to **B** and then click **OK**.
4. Zoom into the title block in the lower right corner of the border.
5. Right-click and click **Select Documentation**.
6. In the Title box, click **<Title>.**
7. Right-click and click **Properties.**
8. In the Field Properties dialog box, type your company name in the **Value** box and click **OK**.

9. Click in empty space to deselect the company name.

Save the customized border to the library

You can save the border to the library for reuse.

1. Right-click > Select Drafting Items.
2. Select a segment of the border.
   Result: The whole border and title block are selected.
3. Right-click and click **Save to Library.**
4. In the Save Drafting Item to Library dialog box, switch the Library to the **preview** library.
5. In the Name of Item box, type **MYBORDER**.
6. Click **OK**.

Set the default sheet border

You can change the current default border to a custom border.

1. **Tools** menu > **Options** > **Design** page.
2. In the Sheet area, click **Choose**.
3. In the Get Drafting Item from Library dialog box, in the filter area, switch the Library to the **preview** library.
4. In the Drafting Items list, click the **MYBORDER**.
5. Click **OK** to close the Get Drafting Item from Library dialog box.
6. Click **OK** to close the Options dialog box.

Adding a sheet

Add, delete, and modify sheets and sheet location using the Sheets dialog box.

To add a sheet:

1. **Setup** menu > **Sheets**.
2. Click **Add** to increase the sheets to two.
3. Click **Close** to exit the Sheets dialog box.

Tip: The Sheets dialog box contains buttons to move sheets Up and Down in the sheet order. A rename function also exists that renames sheets.

4. Note how the custom border was automatically added to the new sheet.

Tip: Setting a new default border affects only new sheets. It does not update the border of existing sheets. To change the sheet border on an existing sheet, delete the border and use the Add 2D Line from Library command on the Design toolbar to add a custom border.

Off page references

A SailWind Logic schematic database contains all sheets of the schematic. This feature provides for automated insertion of page references for all nets connected with off page symbols. You add the references to schematic when the off page reference feature is enabled.

Open a schematic

1. Open the file named **preview.sch** found in \ SailWind Projects\Samples.
2. Zoom into the upper left portion of the schematic around the A[00-14] bus.
3. On the **Tools** menu, click **Options**.
4. In the Off-Sheet Labels area, select the **Show Off-page Sheet Numbers** check box.
5. Click **OK**.

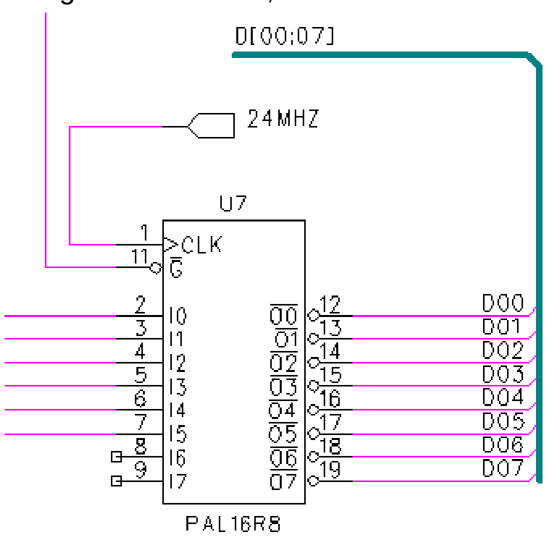
   Result: A [1] appears under the A[00:14] bus name. This indicates that another instance of the bus is also located on page 1 of the schematic.

Tip: You can create a report file that lists the locations of all nets in the design. On the File menu, click Reports, and select the Off-page check box.

Copying a portion of a schematic

You can copy and paste, or save to a file, a group of items for use on other sheets or other schematics.

1. **Right-click** > **Select Anything**.
2. Using area selection, create a selection box around U7 as shown below.



3. **Right-click** and click **Copy**.
4. In the Sheet list on the toolbar, select the **Power** schematic.
5. **Right-click** and click **Paste** and then click to place the object.
6. Zoom into the part and notice the nets on the left side have off-page symbols attached.
7. Do not save a copy of the file.

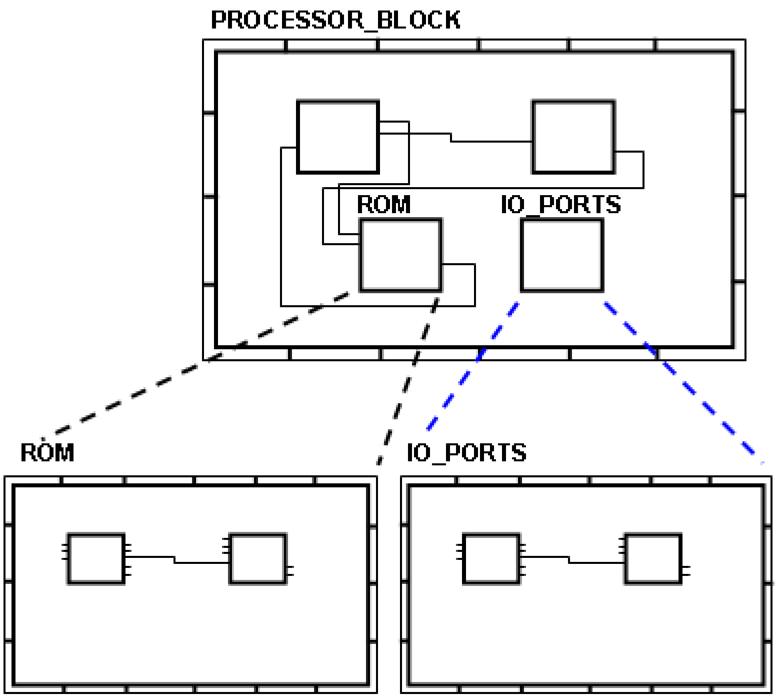
Tip: You use the right-click > Save to File command to save a portion of the schematic to a file and later use the Paste from File command on the Edit menu to add to a schematic.

Hierarchical design

In hierarchical designs, basic block diagrams are used to represent interconnect between logical functions of a design. Typically, these block diagrams are shown on the first page of the schematic to aid in the navigation among various functions in the schematic. The logical blocks represent the underlying schematic for that portion of the design.

In SailWind Logic you create and add logical blocks, called hierarchical symbols, and link them directly to a subset of the SailWind Logic schematic.

Hierarchical design example:



To create a hierarchical schematic in SailWind Logic, you either create the hierarchical symbols first using the Decal Wizard or select a sheet and automatically define a symbol to represent its content. See SailWind Logic Help for detailed steps in creating a hierarchical design.

You completed the managing multiple sheets tutorial.

Designing for Analog Simulation Tutorial

Analog parts require special analog attributes. The SPICE Netlist feature uses the analog attributes to build the SPICE netlist for the SPICE simulator. SPICE models can also be linked to parts to provide the SPICE netlister with required values.

In this lesson:

- Creating an RLC circuit
- Creating an opamp circuit
- Adding a model to the library

Restriction

This tutorial requires the SailWind SPICEnet licensing option. On the Help menu, click **Installed Options** to determine whether you can proceed..

Preparation

If it is not already running, start SailWind Logic.

Creating an RLC circuit

Build a resistor-inductor-capacitor circuit using analog parts. Add analog attributes to the parts for netlisting to a SPICE tool.

Start a new schematic

- Click the **New** button  . At the prompt, click **No** to saving the file.

Add the spice library

The parts for the design are located in the Spice library. Add the library to the Library manager.

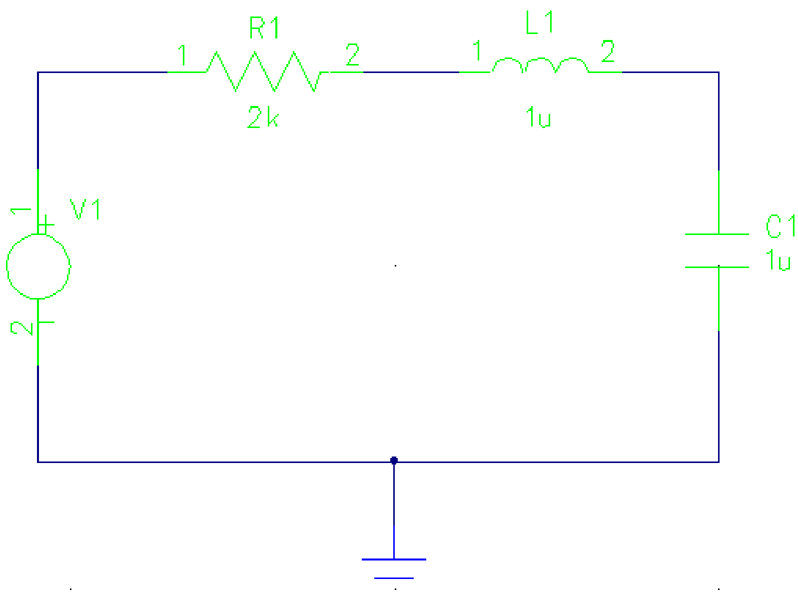
1. **File** menu > **Library**.
2. In the Library Manager dialog box, click **Manage Lib. List**.
3. In the Library List dialog box, click **Add**.
4. In the Add Library dialog box, select the **spice** library and click **Open**.
5. Select the **spice** library in the Library list, and click the **Up** button until the spice library is at the top of the list.
6. Click **OK**.
7. In the Library Manager dialog box, click **Close**.

Add the analog parts

Begin building the RLC circuit by adding the required parts to the schematic.

1. Add part button  **.**
2. In the Add Part Type from Library dialog box, in the Filter area, ensure that the **spice** library is listed in the Library list and that the Items box contains a "*" to show all library items. Click **Apply** if any changes were made in the filter.
3. Select the **CAP0805** capacitor and then click **Add**.

Circuit to be drawn



4. Click to place the capacitor in the schematic.
5. Press **Esc** to cancel adding additional capacitors.
6. In the Add Part from Library dialog box, repeat steps 3-5 for the following Part Types.

Part Type
IND-MOLDED
RES0805
VOLTAGE_SUPPLY*

   *If prompted, use V for the alpha-prefix.

7. When finished, click **Close** in the Add Part from Library dialog box.

Add connections to the analog parts

Connect all the components, using the picture in the previous section as a guide.

1. Add Connection button  **.**

2. Select a pin to start your connections, click to create corners, and select a pin to complete your connections.
3. Select the connection between the capacitor and the voltage supply.
4. Drag to create the power stub, moving the pointer down.
5. **Right-click** and click **Ground**.
6. Click to place the ground symbol.
7. Press **Esc** to exit Add Connection mode.

Label the connections

Label the connections between the components. Labels make the SPICE netlist easy to understand.

1. **Right-click** and click **Select Anything**.
2. Double-click the net between the power supply and the resistor.
3. In the Net Properties dialog box, select the **Net Name Label** check box.
4. Type **INPUT** in the Net Name box and then click **OK**.
5. Repeat steps 2-4 for the remaining nets. Use the net names in the following table.

Net Between the	Net Name
Resistor and inductor	MID
Inductor and capacitor	OUT

The ground connection is automatically labeled with the addition of the ground symbol.

Review analog attributes

Attributes are added to parts in the library or in the schematic. Examine the predefined SPICE attributes on the parts in the schematic.

1. **Right-click** > **Select Parts** > select a part > **right-click** > **Attributes**.
2. In the Part Attributes dialog box, examine the Sim.Analog.Order and Sim.Analog.Prefix attributes. All analog attributes contain the prefix Sim.Analog.
3. Click **OK** to close the Part Attributes dialog box.

Tip: A detailed list of the SPICE/Analog attributes can be found in Help. In the Topics tab of the Help, point to SPICE Netlist Attribute Glossary.

Set up the netlister

The SPICE Netlister creates a netlist format for several different SPICE tools.

1. **Tools** menu > **SPICE Netlist**.
2. In the SPICEnet dialog box, in the Select Sheets list area, select the **Sheet 1** check box.
3. In the Output Formats list, select the vendor format of the SPICE software you will use.
4. Click **Simulation Setup**.

5. Select the **AC Analysis** check box.
6. Click the **AC Analysis** button.
7. Ensure the Interval is set to 10 points per Decade.
8. Ensure the Frequency starts at 1Hz and ends at 1kHz, and then click **OK**.
9. Click **OK** in the Simulation Setup dialog box.
10. Click **OK** in the SPICEnet dialog box to create the netlist.

Checking the netlist

The netlist opens in the default text editor and can be viewed or edited before importing into SPICE software.

1. Notice the parameters from the AC Analysis.
2. Notice the connections and attribute values are listed beside the referenced parts.
3. Do not save a copy of the file.

Completing an opamp circuit

Complete another common analog circuit – the opamp circuit. Add an opamp from the library and examine the SPICE netlist.

Open the opamp circuit

- Click the Open button  and open the file named **opamp.sch** in the \ SailWind Projects\Samples folder.

Add the missing opamp

1. Add Part button  **.**
2. In the Add Part Type from Library dialog box, select the OP-471 part and then click **Add**.
3. Position the part over the open-ended connections in the schematic and click to place the gate.
4. Press **Esc** to cancel adding additional opamp gates.
5. Click **Close** in the Add Part Type from Library dialog box.

Add attribute values

Add net name values to the analog attributes of the amp.

1. With nothing selected > **right-click** > **Select Parts**.
2. Select the **U1-A** part.
3. **Right-click** and click **Attributes**.
4. Double-click in the **Value** box of the Neg attribute.
5. Type **OP_VIN** in the Value box.

6. Repeat steps 4 and 5 and use the following values for each attribute.

Attribute	Value
Out	VOUT
Pos	GND

7. Click **OK**.

Set up the netlister

1. **Tools** menu > **SPICE Netlist**.
2. In the SPICEnet dialog box, in the Select Sheets list, select the **Sheet 1** check box.
3. In the Output Formats list, select the vendor format of the SPICE software you will use.
4. Click **Simulation Setup**.
5. Select the **Transient** check box and clear all other check boxes.
6. Click the **Transient** button.
7. Ensure the following parameters exist.

Parameter	Value
Data Step Time	1ms
Total Analysis Time	10ms

8. Select the **Use Initial Conditions** check box and then click **OK**.
9. Click **OK** in the Simulation Setup dialog box.
10. Click **OK** in the SPICEnet dialog box to create the netlist.

Checking the netlist

Check the netlist for errors.

- Notice the line *** Could not open data file lm741n.mod**.

Tip: The amp in the schematic refers to an analog model and there is no model file in the library.

Adding a model to the library

Add a SPICE model to the lm741 part in the library to provide the netlister with essential values for simulation.

1. Using an Explorer window, navigate to your libraries folder. C:\<install_folder>\<version>\ Libraries\
2. Add a new folder entitled **spice**.
3. In the spice folder, add a new folder entitled **Analog Models**.
4. Copy the lm741n.mod file from the C:\ SailWind Projects\Samples folder and paste it into the Analog Models folder.

Run the SPICE Netlister

Create an error free SPICE netlist with the added model in the library.

1. **Tools** menu > **SPICE Netlist**.
2. Ensure your settings haven't changed.
3. Click **OK** in the SPICE Netlist dialog box to create the netlist.
4. Click **Yes** to overwriting the existing file.
5. Examine the netlist to see the additional SPICE model information.

You completed the creating library parts tutorial.