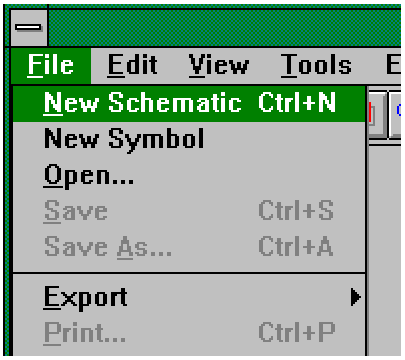
Tutorial - New File

Welcome to the Schematic Plus 0.6 tutorial. As you can see your windows have been arranged so as to maximize the effectiveness of this tutorial. If you wish to exit the tutorial at anytime, reselect Help - Tutorial from the Schematic Plus 0.6 menu. You may need to scroll this tutorial in order to read all of the text.

This tutorial will describe any actions you may need to perform. These actions should be applied to the Schematic Plus window and not the Tutorial window. Also, the actions will be summarized in color below the selection.

This tutorial will step you through the process of creating a schematic and performing a PCB layout. For the first step, you should create a new window. This is performed by selecting File-New as shown below, from the Schematic Plus menu.



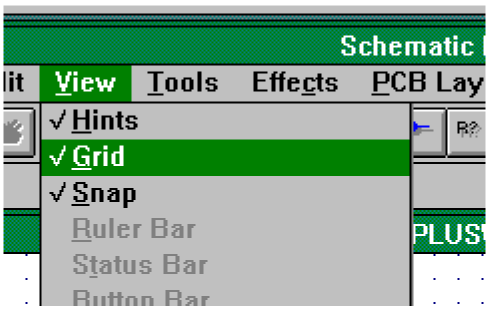
- Select File from the menu.
- Select New Schematic from the submenu.

During the tutorial, when you finish performing the actions, select the browse forward button in the Tutorial window. This is the button with two rightward pointing double arrows **>>**. For example, this needs to be done now.

Tutorial - Grid and Snap

Now that we have a new window, we can begin creating a schematic.   Later we will create symbols. Notice the caption of the window (Schematic -- No Name).   This lets us know that the file has never been saved and is currently un-named.

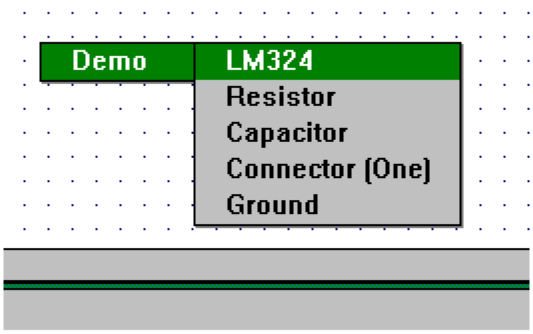
Lets now make sure that grid and snap are on.   Although this is not necessary, it makes drawing go much easier.   When you turn on the grid, you will see the grid appear in the new window.   When you turn on snap, you will only see the button stay pushed (or the menu checked).   At this time, you will need to make sure that both the grid and snap menu items are checked.   If not, select the one(s) that are unchecked.   You may not be able to see the buttons due to the way the window is arranged.



- Verify that the grid and snap menu items are checked.
- If not, select the appropriate one from the menu.

Tutorial - Place a Symbol

We are now ready to place our first part. Lets start with an op-amp (LM324). There are several ways to select a symbol (or part or component). Selecting Tools-Symbol from the menu will bring up a file browse dialog box where a symbol (*.sym) can be searched for. If the symbol is in our library, we can place it by clicking the right mouse button anywhere in our New File window. We then will see a list of libraries and can drag to the desired library where our symbol is stored. Releasing on our symbol will now allow it to be placed. Clicking the left mouse button places the symbol. We want to place it so that it is centered vertically and as far to the right as possible, horizontally.

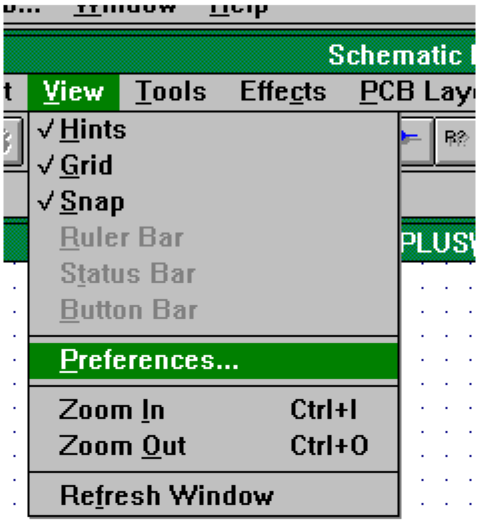


- Click and hold the right mouse button inside the New File window.
- Drag the mouse pointer to the library named Demo Library.
- Drag the mouse pointer to the right and release on the symbol named LM324 Op-Amp.
- Move the cursor to the right of the New File window and center it vertically.
- Press and release the left mouse button.

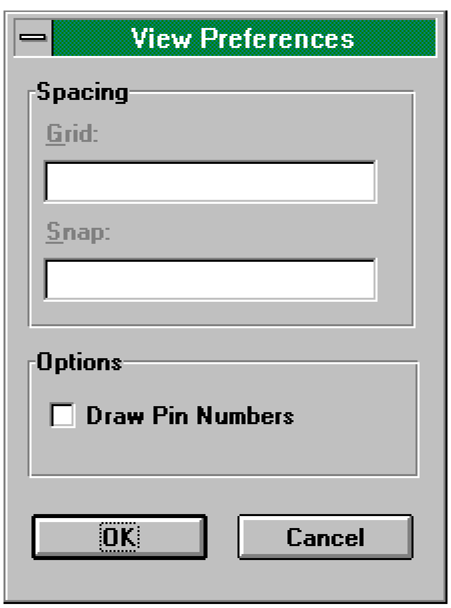
The symbol is now placed. Notice that the designation starts at U1. Since there are 4 LM324s in this package, the next 3 placed would also be named U1.

Tutorial - Viewing Pin Numbers

Now that we have a symbol placed in our new schematic, we may want to see what the pin numbers now. Normally we would want these off since these are really only used for the netlist generation. Lets see how to turn them on.



- Select View-Preferences from the menu.

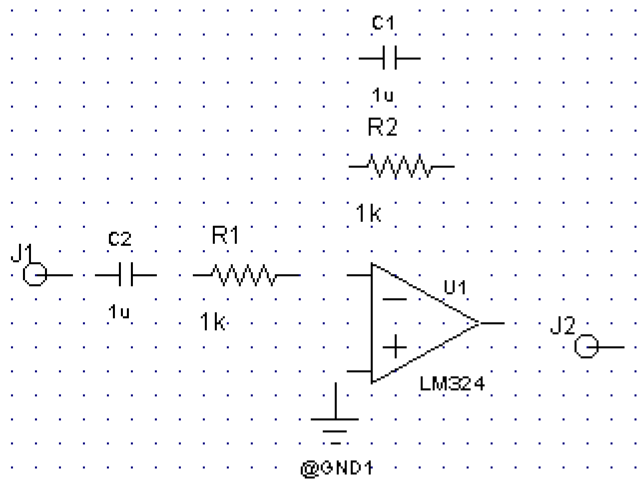


- Check the box and select OK to see the pins.
- Select View-Preferences
- Uncheck the box and select OK to hide the pins.

Tutorial - Placing Other Symbols

Now we will place the rest of the symbols for this tutorial. We will need the following: 2 resistors, 2 capacitors, 2 one-pin connectors, and 1 ground. These parts are selected the same way (right mouse button) and are found in the library as such:

resistors: Demo Library - Resistor
capacitors: Demo Library - Capacitor
connectors: Demo Library - Connector - One Pin
ground: Demo Library - Ground



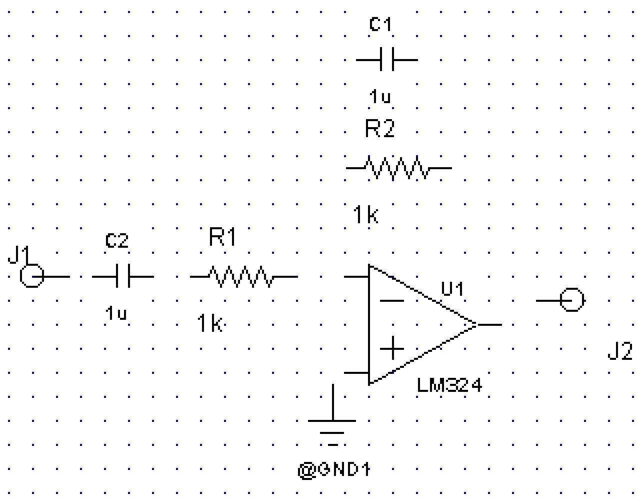
- Place the components to create the above schematic.

NOTE: You may need to move the components once placed. If this is the case, perform the step below as needed.

- Click and hold the left mouse button on the symbol to move.
- Drag the symbol to the new location.
- Release the left mouse button.

Tutorial - Rotating Symbols

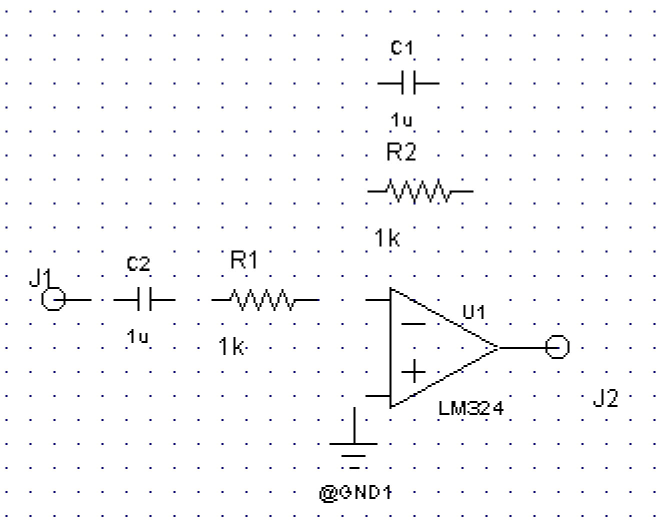
We will now attempt to rotate J2. This is performed by selecting the desired symbol (J2) and selecting rotate from the menu. We will also deselect the object by clicking in an unused area. Notice J2 is now misaligned. We will need to correct in the next step.



- Click and release the left mouse button on J2.
- Press CTRL-R twice.
- Click and release the left mouse button anywhere in the schematic where a symbol is not.

Tutorial - Moving Symbols

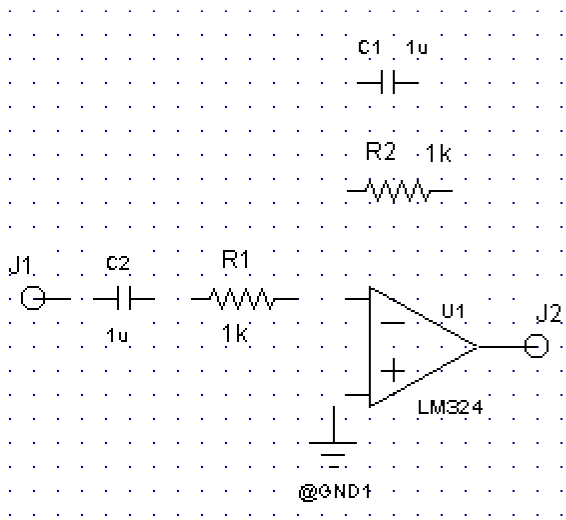
All symbols should be placed and in the proper position, with the exception of J2. We will now move that symbol so that it is in alignment as shown below.



- Click and hold the left mouse button on the symbol J2 (not the text itself).
- Drag the symbol to move it.
- Release the left mouse button to place the symbol.

Tutorial - Moving Parameters

We now would like to move some of the parameters to clean up the schematic. The parameters are attributes of a symbol that are changeable. R1 is a parameter along with 1k. To move these, we perform the same steps as moving symbols, except we select the parameter instead. Clean up the parameter to look something like this.



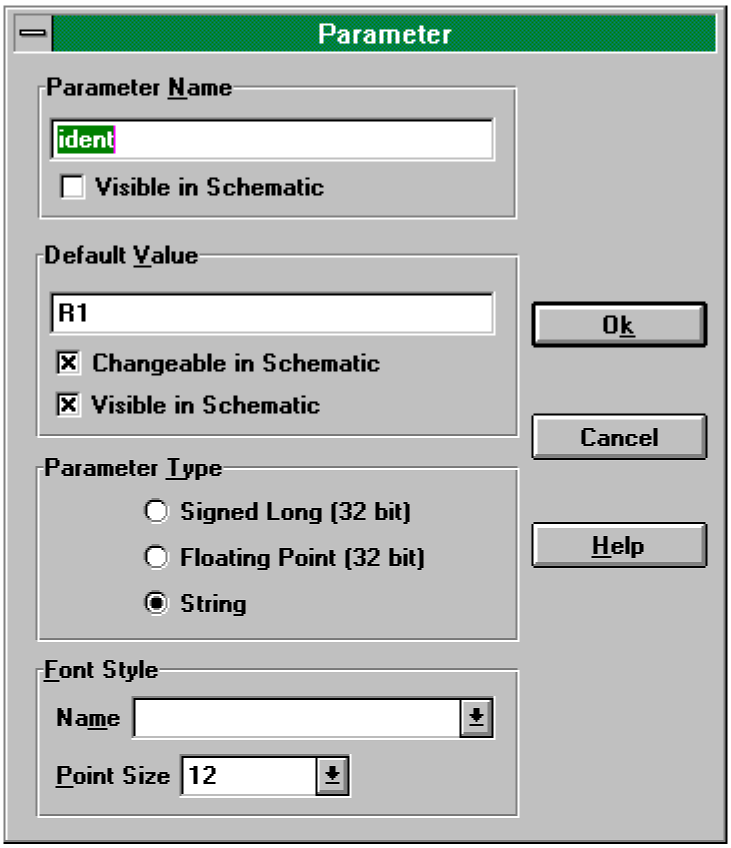
- Click and hold the left mouse button on a parameter.
- Drag to position the parameter.
- Release the button.
- Repeat for all parameters needed.

Tutorial - Changing Parameters

Lets next change some of the parameter values.   We normally do not want to change the identifiers (designators).   We may, however, want to change the part value.   Lets change the resistors to 47k and the capacitors to 0.1u.   To do this, we double click on the parameter with the left mouse button.   We then can change any of the information in the dialog box.

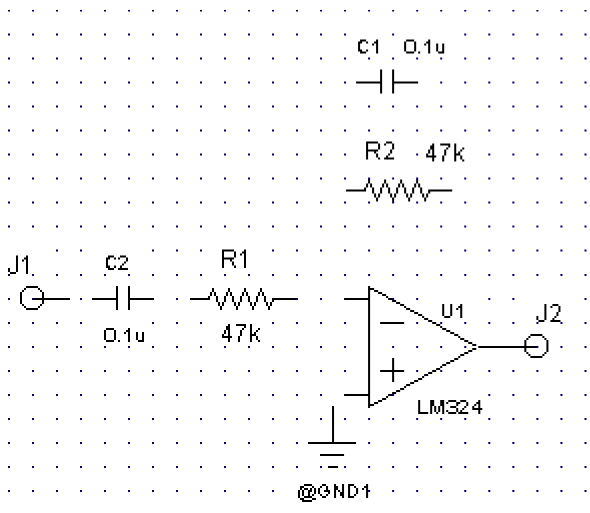
- Double-click the left mouse button on the parameter of 1 for C1.

We should now have a box that looks like this.



- Change the 1 in Default Value to 0.1u.   Do not put in the quotes.
- Select Ok.
- Repeat to change C2 to have a value of 0.1u also.
- Repeat for R1 and R2 to have a value of 47k.

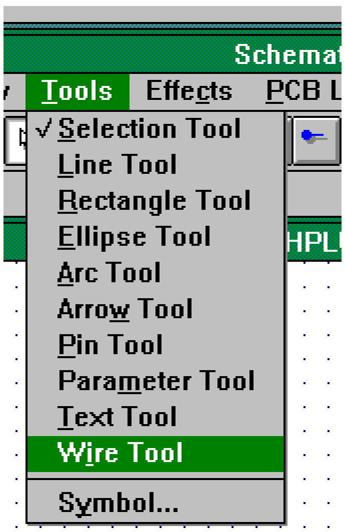
We should look like this now.


C1 0.1u
R2 47k
J1
C2
0.1u
R1
47k
U1
J2
LM324
@GND1

Tutorial - Wire Tool

Now, we need to connect the parts with wires. Before we do that, we need to understand tools. It is easy to visualize a bag of tools from which we can only pick one. This is the way Schematic Plus now works. We have a line tool, a pin tool, a parameter tool, a wire tool, etc. We have been working all along with the selection tool which allows us to move, resize, delete, and edit objects. We can tell which tool is currently in our hand by three methods. The first is the cursor. If our cursor is the standart windows pointer, we are using the selection tool. If it is a pencil, we have one of the graphics tools. If it is a solder iron, we are using the wire tool. We can also tell by the button bar as to which tool we are using. The current tool has its button depressed. We can see this now by the selection tool button being depressed. Finally, we can look under the Tools item in the menu and one tool will have a checkmark beside it.

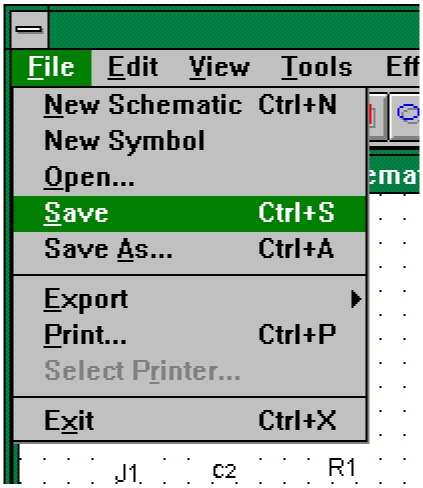
We now want to select the wire tool.



- From the menu, select Tools-Wire Tool.
- Click on any symbols pins to start the wire.
- Click to set a corner of the wire.
- Click on any symbols pins or on another wire, to end the wire.
- Place all wires as shown.

Tutorial - Saving the File

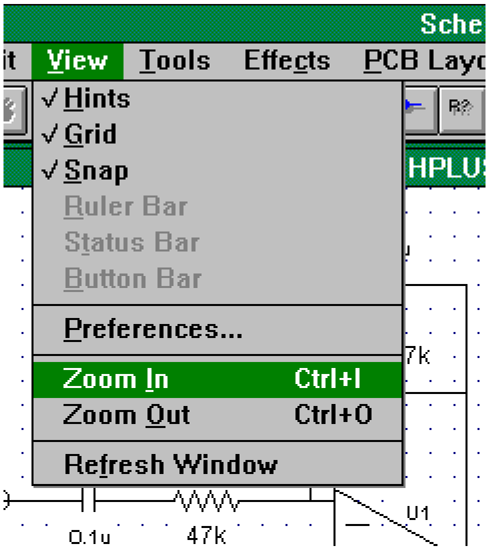
Now, we would like to save the schematic. This ensures that in the event of some unforseeable error condition, we are protected. Saving the file is as easy as selecting File - Save from the menu.

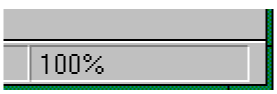


- From the menu, select File-Save.
- Enter a file name.
- Select Ok.

Tutorial - Zoom

Sometimes, it is desirable to view the workspace with a variable magnification. This can be performed by selecting View-Zoom In or View-Zoom Out from the menu. We can also quickly see the magnification by looking at the right side of the status bar.

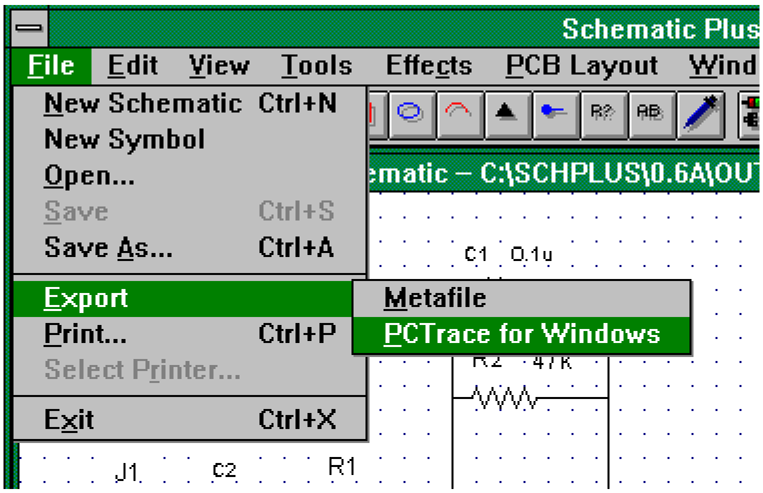




- Select View-Zoom Out from the menu.
- Notice how the status bar changes.

Tutorial - Creating the Netlist

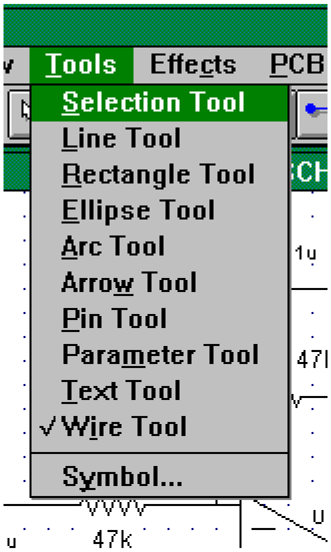
Now, we would like to create a net list for import into PC-Trace. This allows us to perform a PCB layout. Currently, Schematic Plus does not capture netlist errors. These are captured by PC-Trace. Creating the netlist is as simple as selecting File-Export-PC-Trace for Windows and selecting a filename.



- Select File-Export-PC-Trace for Windows from the menu.
- Select a filename.

Tutorial - Deleting Items

Suppose we made a mistake and would like to delete an item.   First we get the selection tool:

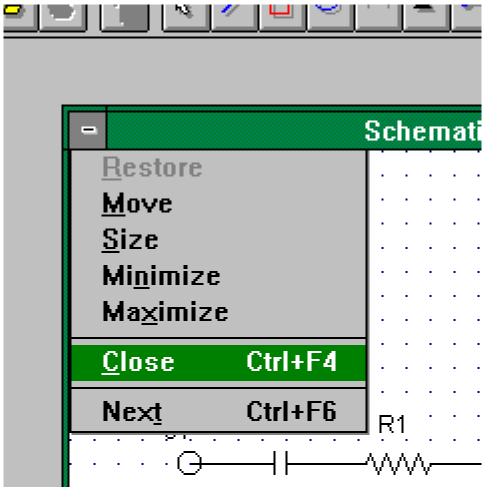


Next, we click on the item(s) we are interested in deleting.   Multiple selection is performed by holding the CTRL key while clicking.   Finally, we select Edit-Delete from the menu.

- Select Tools-Selection Tool from the menu.
- Click the left mouse button on the LM324 symbol.
- Select Edit-Delete from the menu.

Tutorial - New Symbol

We last saved the schematic and do not want to keep the changes (deleted LM324). First, we would select the system menu box of the schematic window:



Next we would select Close. We then would be asked if we want to save the file.

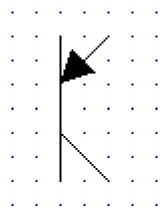
- Click the left mouse button on the system menu.
- Click Close.
- Click No.

Now, lets create a symbol. There are several things which must be done when creating a symbol. It is important that these steps are followed and that none is omitted. First lets get a symbol window.

- Select File-New Symbol from the menu.

Tutorial - Graphics

For our symbol, lets first draw the graphics. The graphics are just a visual representation of the symbol and in no way affect the behavior. Keep in mind that pins are a special graphic which are very important. We will create the NPN transistor as shown below.

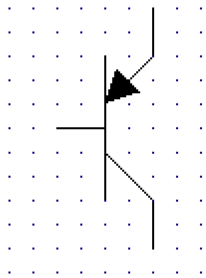


- Select the Line Tool.
- Draw the lines as needed.
- Select the Arrow Tool
- Draw the arrow as shown.

NOTE: To resize an object, select the object and drag the control points (black solid rectangles).

Tutorial - Pins

Now that we have the graphics, we need to add some pins.   Pins have two purposes beyond what graphics can do.   First, the are points where wires can connect to.   Wires can only connect to pins and other wires.   Second, they are used to resolve connections in the netlist generation for PC-Trace.   Pins have two ends, one of which is the actual connection point.   This point appears as an X when selected but disappears when de-selected.   Add three pins for the base, collector, and emitter connections as shown.   NOTE: The start point of the pin should be away from the part.



- Select the Pin Tool
- Click and hold the left mouse button near where the base pin should be.
- Drag to connect to the vertical line.
- Release the left mouse button.
- Repeat for the emitter and collector pins.
- Select the selection tool and move the pins if needed.

Tutorial - Symbol Parameters

For symbols which exactly one physical package, the only other step is setting up some parameters. We select PCB Layout-Part Parameters from the menu and fill in the information. The identifier string is the desiginator which will be used for the part. For resistors, this is usually R. The identifier is not limited to one character but must be only characters. There is an exception for net symbols which is described in the help file. Description is for your information and for PC-Trace. Symbols Per Package tell how many of these symbols will make up one physical package. An example is the LM324 which has 4 op-amps per package. Finally, the Definition File is the name of the .def file used by PC-Trace. The .def describes the physical layout of the part. See the PC-Trace tutorial for information on how to create this if one does not exist.

- Select PCB Layout - Part Parameters from the menu.
- Type a capital Q for the Identifier.
- Type NPN Transistor for the Description.
- Leave the Symbols Per Package set to one.
- Type tran.def for the Definition File.
- Select Ok.

Tutorial - Moving the Identifier

Selecting View-Refresh from the menu causes the Identifier Q to becomes visible. This can now be moved to the desired position. It may also be resized so as to allow space for the number which will later be appended to it when it is placed in a schematic.

- Select View - Refresh from the menu.
- Set the tool to the Selection Tool.
    - Click and hold on the Identifier.
    - Drag to position.
    - Resize if desired.

Tutorial - Pin Numbers

For every symbol created, we must go in and edit the pin numbers. This is performed by double clicking on each pin and modifying the Primary Pin Number to that desired. If we have multiple Parts Per Package (greater than 1), we must add Alternate Pin Numbers. For instance, if our Parts Per Package was set to 4, we must add 3 Alternate Pin Numbers. The symbol LM324 can be opened at a later time to examine how this works. For now, change the base pin to 1, the collector to 2, and the emitter to 3.

- Double click on the base pin.
- Set the Primary Pin Number to a 1.
- Select Ok.
- Repeat for the other two pins setting them to 2 and 3 as stated.

Tutorial - Adding Text

Suppose we would like to add text to the part such as 3904. This is as simple as selecting the Text Tool, dragging a bounding rectangle near the graphics, and double clicking to edit the text and font.

- Select the Text Tool
- Drag a box near the graphics of the symbol.
- Double click the box.
- Type in the Text 3904 and select the font information.
- Select Ok.
- Select the workspace, away from the graphics to de-select the text.

Tutorial - Saving the Symbol

Now, just as we saved our schematic, lets save the symbol. Select File-Save from the menu and fill in the appropriate name. It is a good idea to save new symbol files in a directory with the existing symbols.

- Select File-Save from the menu.
- Change directories to the symbol directory (preferably {schplus directory}\sym\)
- Enter npn for the symbol name.
- Select Ok.

Tutorial - Adding a Library

Now that our symbol is created, lets add it to the library.

- Select Edit-Library Editor from the menu.
- Select the New button in the Library section.
- Enter My Parts for the Library Name and select Ok.
- Change directories to the LIB directory. This should be ({schplus directory}\lib\).
- Enter the name myparts.lib and select Ok.
- Select New in the Symbol section.
- Enter NPN Transistor for the Symbol Name.
- Change directories to the SYM directory. This should be ({schplus directory}\sym\).
- Select the file npn.sym and select Ok.
- Select Close to close the Library Editor.
- Click the right mouse button on an empty schematic to see your new symbol.

Remember, a symbol can be added to the schematic without adding it to the library. Just select Tools-Symbol from the menu and search for the symbol to add.

Tutorial - Special Items

As stated earlier, sometimes a symbol needs to be created which has multiple symbols in one physical package. There are a few special steps and considerations when creating these parts. Also, sometimes a net symbol is desired. This type of symbol has no physical part but is used to show connections to common nets (such as the Ground Net). The included ground.sym is a net symbol.

For multiple symbol parts, remember the following:

- Enter the number of parts per package in the Part Parameters dialog box.
- For every pin in the symbol, add alternate pins. Alternate pins map part numbers (in a package) to different pin numbers.

For symbol nets:

- If the first character of the Identifier is @, the symbol is a net symbol. When used, with the net list, this symbol will not show up as a part (the .def file entry is ignored) but instead all connections to any of these parts are considered connected together.

Pin Numbers can be turned on for verification purposes in the View-Preferences menu item.

IMPORTANT: This version of Schematic Plus is completely new. SAVE OFTEN!!! Always double check your PCB Layouts and you must route power and ground manually for multiple part packages.

Tutorial - Complete

Congratulations, you are on your way to creating professional looking schematics and PCB layouts. Thank you for your interest and time.   For more information, see the on-line help, see the file README.WRI, register to receive the manual, and call the BBS for the latest updates.

Steve Poulsen