

MicroSim PCB Layout Tutorial

By

Brett Marshall & Mike Parker

Prepared for

Dr. Thomas Stewart

PURPOSE

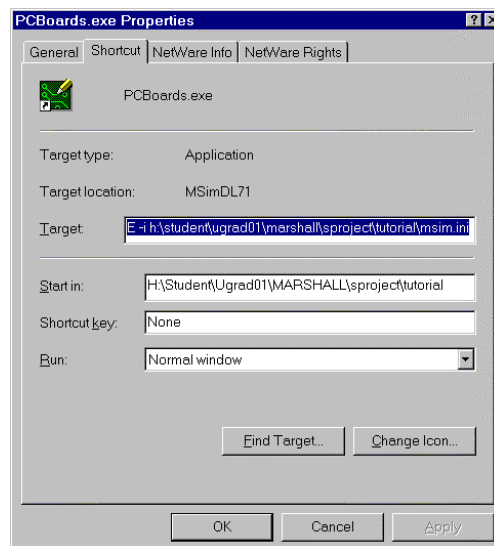
The purpose of this tutorial is to show the user how to use the MicroSim PCB Layout software to create a board layout and generate the files needed to manufacture the boards. This tutorial will explain how to create the shortcuts needed to run the PCB Layout and Schematics software, how to create a footprint from a copy or from scratch, create the corresponding schematic symbols, and how to perform the autoroute function.

GETTING STARTED

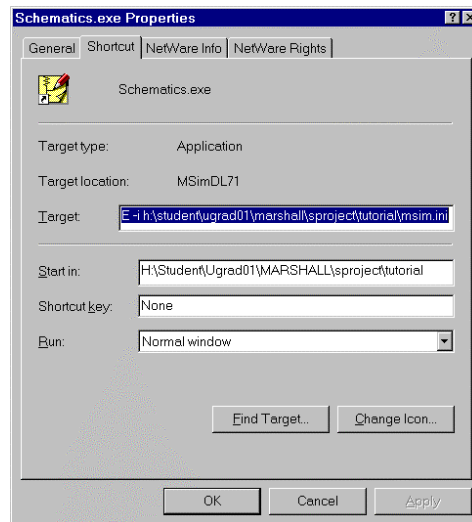
Before attempting to use the PCB Layout software, become familiar with the PCB Layout User's Guide. The user's guide is called Pcburd.pdf and can be found at g:\MsimDL71\Docs or by clicking on the start menu\EE Applications\MicroSim DesignLab Documentation\MSimPCB User's Guide. Shortcuts for the MicroSim Schematic and PCB programs must be created and stored in a dedicated directory. To avoid software problems all file and folder names created while using the MicroSim software should be eight characters in length or less.

To create the necessary shortcuts:

1. Open Windows Explorer
2. Go to C:\Windows
3. Select the *msim.ini* file
4. Copy it to a dedicated folder
5. Go to g:\MSimDL71 folder
6. Use the CTRL key to select the PCboards.exe and the Psched.exe files
7. Copy these files to the dedicated directory
8. Select the PCboards.exe file
9. Right click and select *Create Shortcut*
10. Select the new shortcut and right click
11. Select *Properties* from the menu
12. Click on the *Shortcut* tab
13. Change the Target to G:\MSimDL71\PCBOARDS.EXE -i h:\full path name to dedicated folder\msim.ini



14. Then repeat the procedure for Schematics
15. Change the Target to G:\MSimDL71\PSCHED.EXE -i h:\full path name to dedicated folder\msim.ini

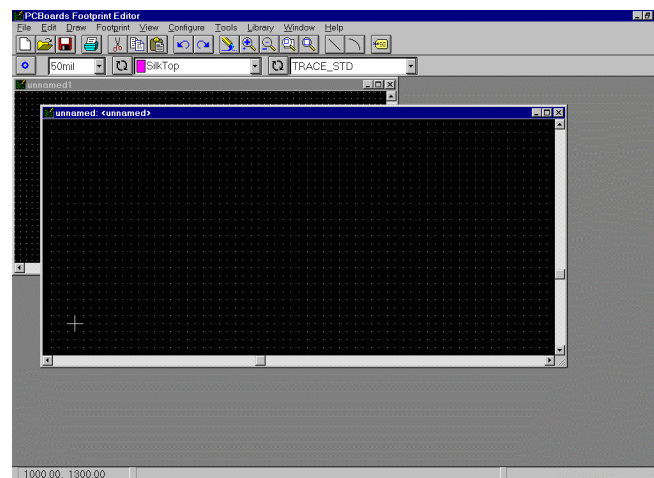


16. Delete the original copies
17. Rename the shortcuts Schematics and PCBboards

Once this is done, you will be able to launch PCB Layout and Schematics from your dedicated folder. Launching from your folder will allow the programs to find the libraries you create by reading the modified Msim.ini file. If these programs are launched from the start menu, an error will be generated stating that the program cannot find the customized part libraries.

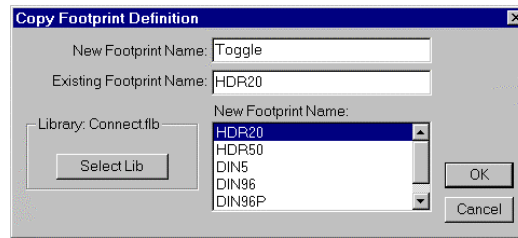
CREATING CUSTOM FOOTPRINTS

If you plan to use a component that is not included in or you cannot find in the Microsim libraries, you will have to create a custom footprint for your component. The footprint must be created before you try to develop the schematic version of your circuit. To create a custom footprint, launch PCB Layout from the shortcut you created in the dedicated folder. Once the PCB Layout program is open, go to the **Library** pull-down menu and select **Footprint Editor**. A new window is opened. Now that you are in the Footprint Editor, you can create your own footprint by starting from scratch or by copying an existing footprint and modifying it.

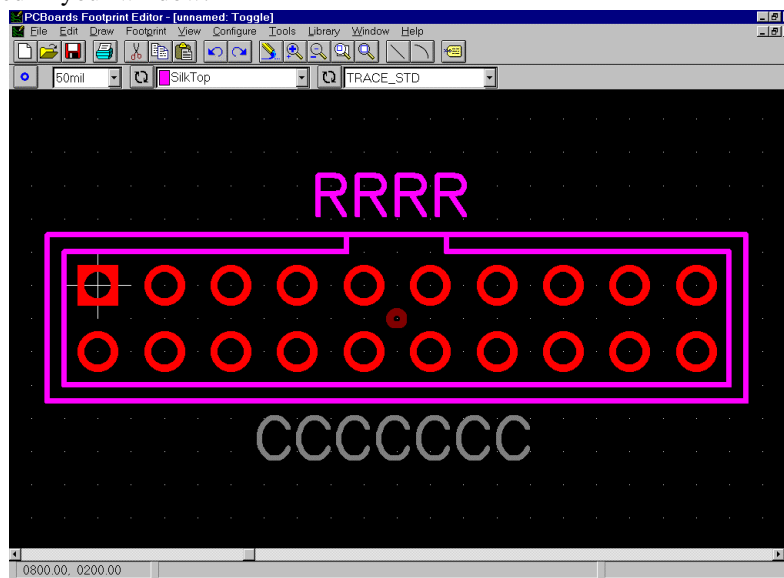


Footprints Using “Copy”

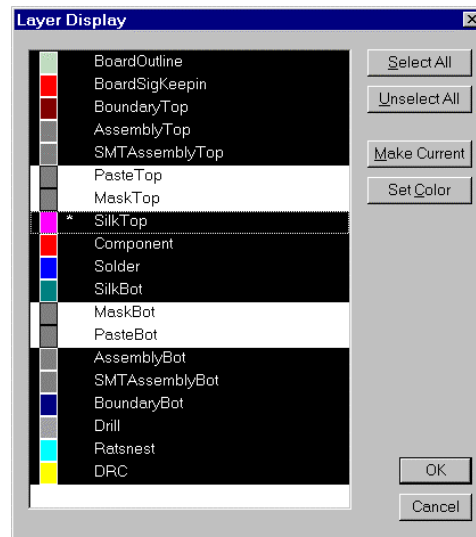
To create a footprint from an existing footprint, open the Footprint Editor and select *Copy* from the *Footprint* pull-down menu.



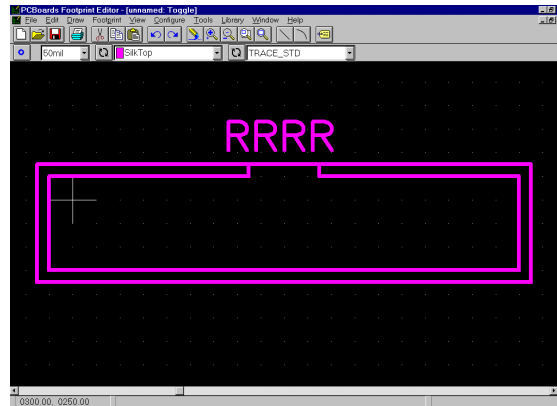
Then click on *Select Library* and browse the libraries (G:\MSimDL71\lib) for an existing part. Select the “Connect” library. Select HDR20, change the New Footprint Name to Toggle, and hit OK. The existing footprint is placed in your window.



Select *Layer Display* from the *Configure* menu and then click on *Unselect All*. While the layer display window is open, highlight the silk top layer and then click on the *Make Current* button and then OK.



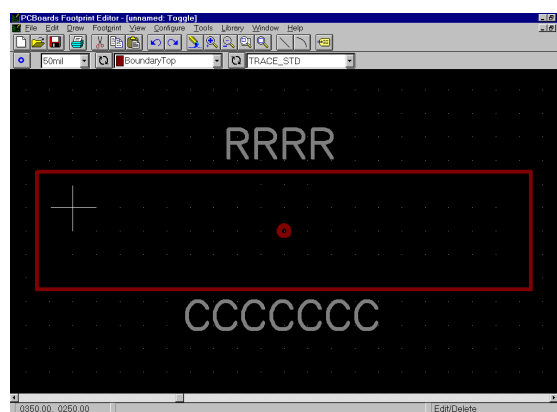
The silk top layer boundary and REFDES should be the only items on the screen.



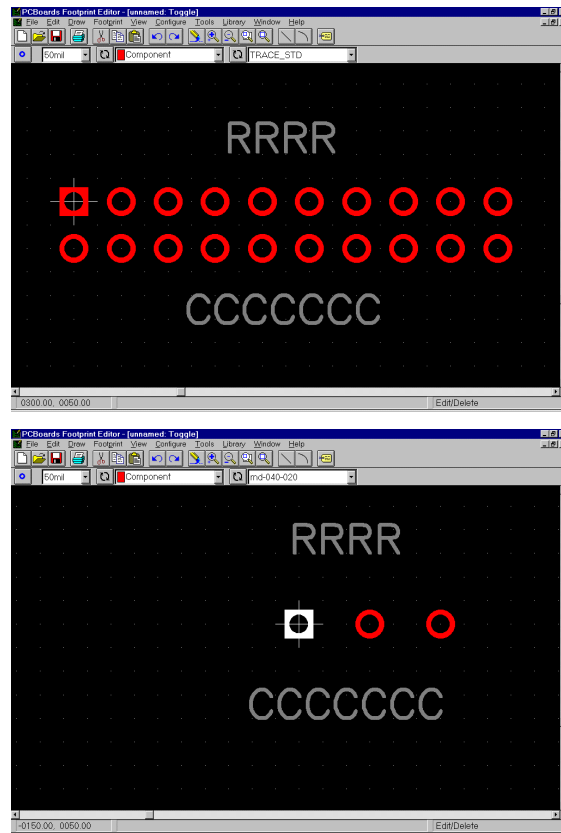
Delete the boundary and leave the REFDES. Then select the Assembly Top layer from the toolbar and delete the boundary only.



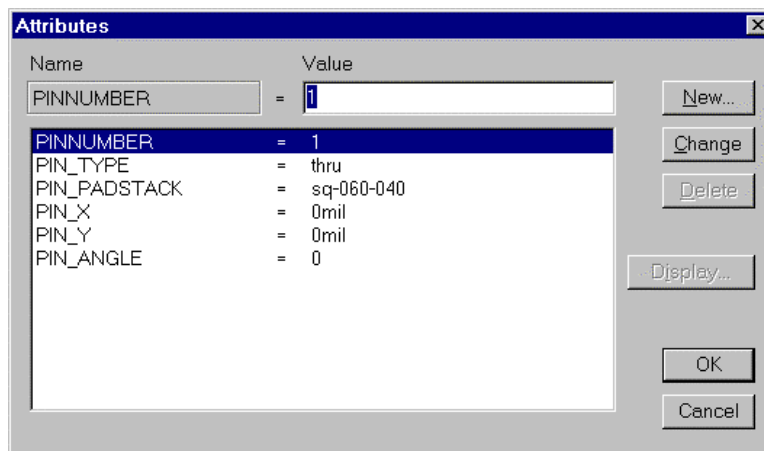
The next layer is the Boundary Top layer. Select the boundary and delete it.

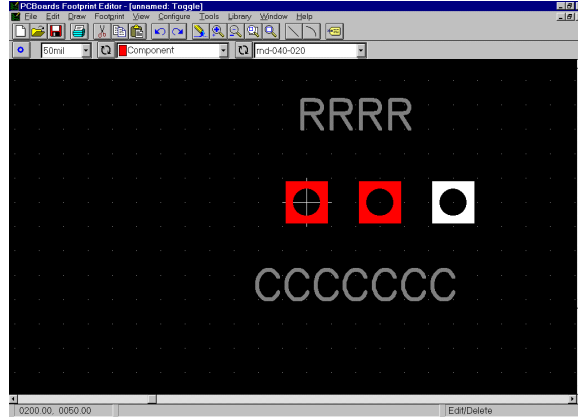


Now select the Component layer from the toolbar and adjust the pin separation and number of pins to meet the specifications of your component. For the toggle switch, you need three pins about 150mil apart.



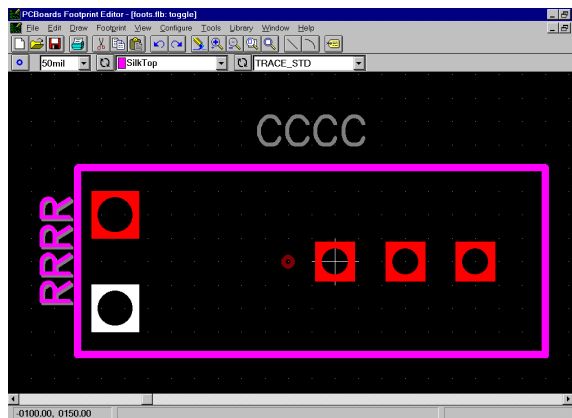
Double click on the individual pins and make sure that the pins are labeled as you need them to be. This is important when you create a part in the Schematic Capture program; the pinout of the footprint must be the same as the pinout of the schematic symbol. When you double click on the pins, an attribute window pops up that will allow you to change the pin number and the size and shape of its padstack. To see a list of available padstacks, go to the window in the toolbar that says **TRACE_STD**. To the left of this box there is a button with two circular arrows, click on this button until the box says **rnd-040-020**. Next, click on the arrow to the right of the box and the list of available padstacks appears. Find one that will work for your application and then double click on the pin you want to change and enter the size and style you want the padstack to be changed to. For the toggle, make the padstacks **sq-060-040**.





Once the pins are located properly, follow these steps:

1. Go to the **Configure** menu and select **Layer Display** (or hit F3). Select **Unselect All** and then hit OK. Return to the layer display box in the toolbar and select **Component**, this should show all of the pins.
2. Select the **Silk Top** layer. Go to the **Draw** menu and select **Rectangle**. Click on the point that you want to be the first corner of the rectangle and then the second corner. There should now be an elastic line. Use this to complete the rectangle. Make the rectangle large enough to add two mounting holes for the switch. The dimensions of the switch were taken from the parts catalogue. The silk top rectangle does not have to be the same as the other layers. The silk top layer is everything that will be printed on your board in white paint.
3. Select the **Assembly Top** layer and repeat the procedure from the silk top. The assembly top rectangle should contain all of the component pins within it.
4. Select the **Boundary Top** layer and draw another rectangle over the assembly top rectangle.
5. Go to the **Draw** menu and select either **Machine Placement Center** or **Machine Center by Coord**. Machine Placement Center will allow you to manually place the center and the Machine Center by Coord will allow you to type in the coordinates for the center of your footprint.
6. Now go to **Add Via** from the **Draw** menu.
7. Place two vias about 200 mils apart (If you were actually going to use this footprint on a production board you would need to know exact placement of pins and dimensions)
8. Double click on the pins and change them to square vias

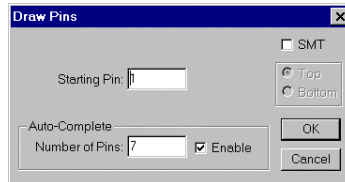


9. Go to **File/Save As** and type in the name of the component, browse for your dedicated folder, and enter a name for your custom library.

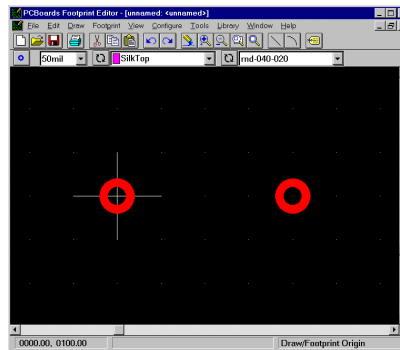
Footprints From Scratch

To create a footprint from scratch, go to the **Footprint** pull-down menu and select **Set Name**. Name your component (keep the name 8 characters or less). Then complete the following steps.

1. Change the grid spacing to the desired pin separation by clicking on the arrow next to the box that says “50mil.” If your pins should be 100 mil apart then you would select 100mil from the pull-down menu.
2. Go to the **Draw** menu and select **Pins**. A window will appear and ask you for the number of pins to draw and if you want the auto-complete function enabled. Enter the number of pins your component has and make sure the enable box is checked. Click on OK and a pencil appears on the screen. Use the pencil to place the first three pins in the pattern you want and the auto-complete will add the remaining pins following the same pattern. For the power header, you need two pins.

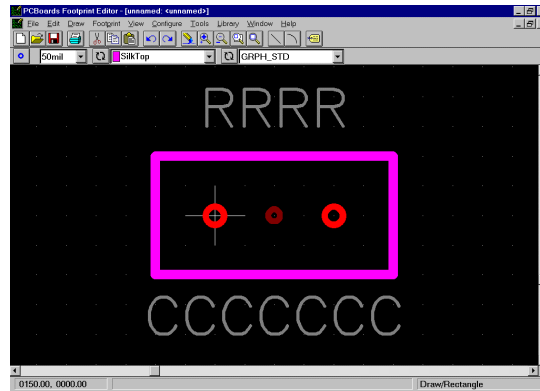


3. Go to the **Draw** menu and select **Footprint Origin**. Place the cross hairs over Pin One and click. The rest of your footprint will be oriented using Pin One as the origin.



4. Go to the **Configure** menu and select **Layer Display** (or hit F3). Select **Unselect All** and then hit OK. Return to the layer display box in the toolbar and select **Component**, this should show all pins.
5. Select the **Silk Top** layer. Go to the **Draw** menu and select **Rectangle**. Click on the point that you want to be the first corner of the rectangle and then the second corner. There should now be an elastic line. Use this to complete the rectangle. The silk top rectangle does not have to be the same as the other layers. The silk top layer is everything you want to be printed on the board in white paint.
6. While in the silk top, go to the **Draw** menu and select **REFDES Template**. Place the REFDES near the boundary of your component. Then double click on the REFDES and click on the display button. Take note of the numbers present and highlight the Assembly Top layer. Insert the numbers into the appropriate categories. Click on **Assign to Layer** and then OK.
7. Select the **Assembly Top** layer and repeat the procedure from the silk top. The assembly top rectangle should contain all of the component pins within it.

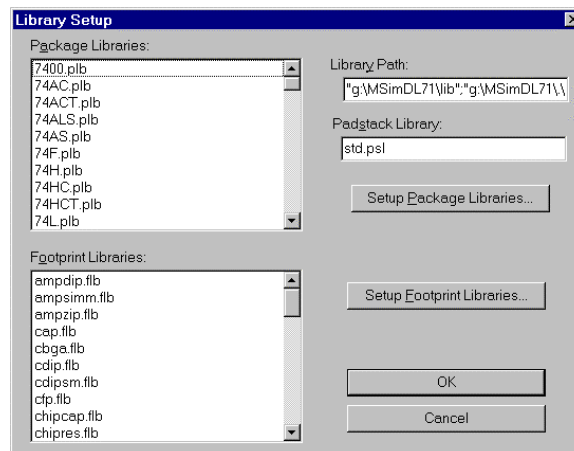
8. Go to the **Draw** menu and select **TYPE_NAME Template**. Place it near the boundary of your component.
9. Select the **Boundary Top** layer and draw another rectangle over the assembly top rectangle.
10. Go to the **Draw** menu and select either **Machine Placement Center** or **Machine Center by Coord**. Machine Placement Center will allow you to manually place the center and the Machine Center by Coord will allow you to type in the coordinates for the center of your footprint.



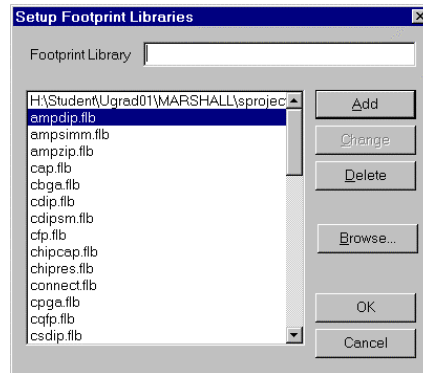
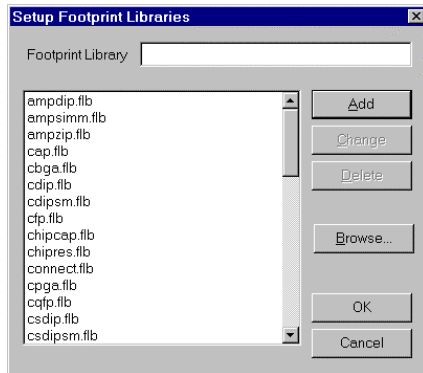
Now you need to save this footprint to the library that you created for your custom footprints. To do this, go to **Footprint\Set Name** and enter the name of your component. Then go to **Footprint\Save to Library** and browse for your dedicated folder. Enter the name of your custom library. Once you have completed these steps, you should have a footprint for a two pin header and a three pin toggle switch with two mounting holes in your library. To edit your custom footprints you have to open the Footprint Editor then open your library. Once your library is open, go to **Footprint\Get**. A list of the footprints in your library should appear. Select the footprint of the component to be edited. Then make the necessary changes, and save the footprint.

To add new custom footprints to your library you should open the Footprint Editor and then select **File\Open** from the pull-down menus. Browse through the folders until you find the dedicated folder that contains your customized library. Once you have found your library select it and click OK. The window should now say the name of your library and then “:unnamed1” in the blue title bar. From this point, you can either copy or create a new footprint in the same manner as described previously. Save the footprint in the same manner as for the power header. This will place the new footprint into the library without replacing the footprints already contained in your library.

To be able to use your new footprints in the Schematic capture program, or in PCB layout, you must open the PCB layout program and select **Setup** from the **Library** menu.

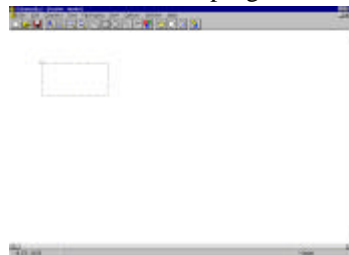


Select **Setup Footprint Libraries** from the new window, and then browse the files until you find your library. Once you have found your library, highlight it and click on the **Add** button and then **OK**. If you have done it correctly, your library name will appear at the top of the Footprint Libraries window. If it does, click **OK**, if it doesn't, try again.

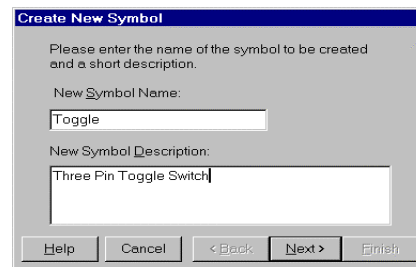
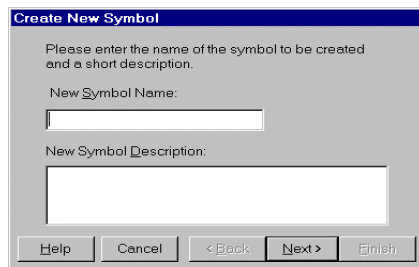


CREATING SCHEMATIC SYMBOLS

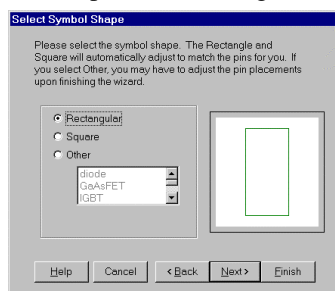
The next step is to create schematic symbols to go along with your footprints. Use Windows Explorer to open the dedicated folder and launch the schematics program. Then go to **File\Edit Library**.



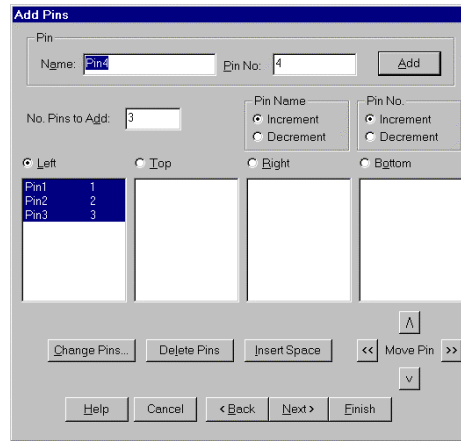
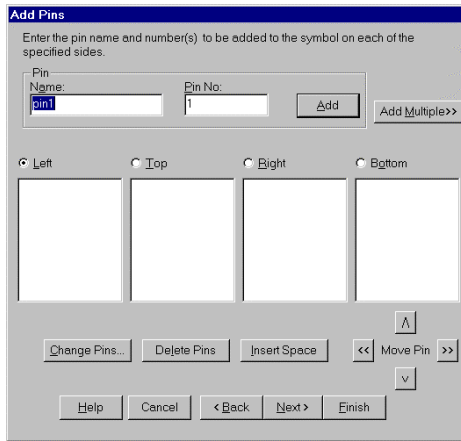
The Editor window appears as shown. Go to **Part\Wizard** and follow the steps. The first step is to name the new symbol. We used the same name as our footprints. In the Symbol Description box, type a brief description that will be shown when you are selecting parts later.



Next, the wizard will ask for the symbol shape. We used a generic rectangle for all of our parts.

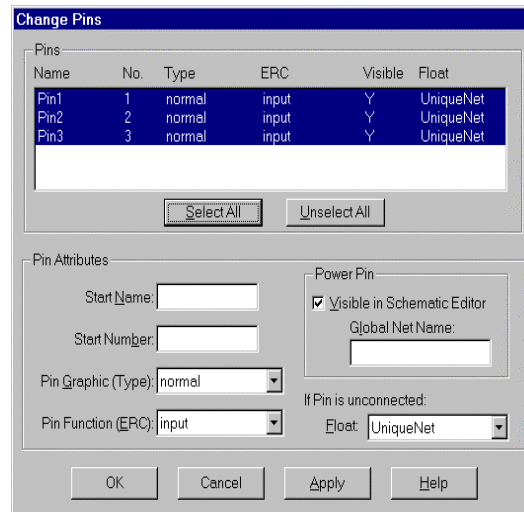
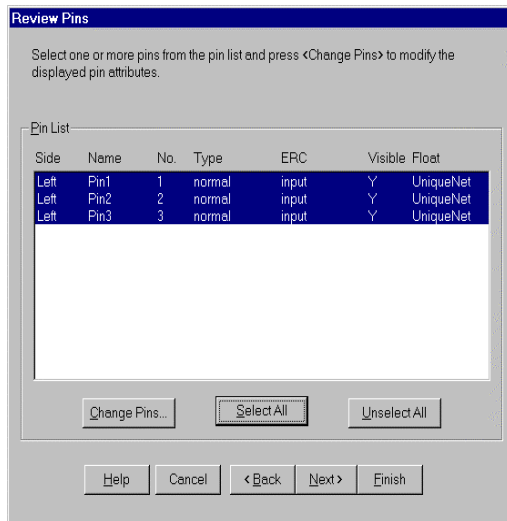


Then it will ask you to add pins. There are two ways to do this. In the current window you can add individual pins, which would be fine for a three pin toggle switch, but not for a complex part with many pins. The second choice is to use the **Add Multiple** button. This window allows you to add as many pins as you want at once.

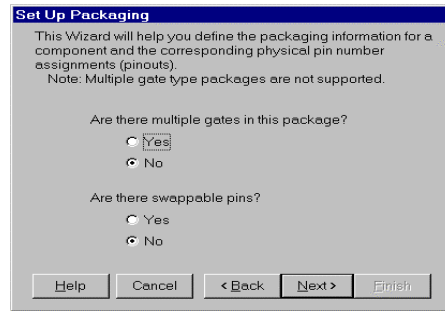
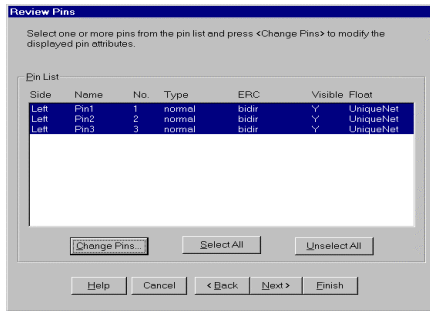


Once the pins appear, they can be moved to other sides of the symbol by highlighting the pins to be moved and then clicking on the arrows in the lower right corner of the window. The pin order can also be rearranged by highlighting a pin number and then using the arrows to move the pin into the correct position.

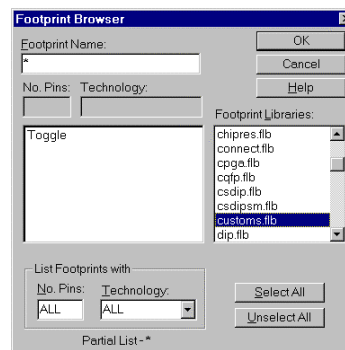
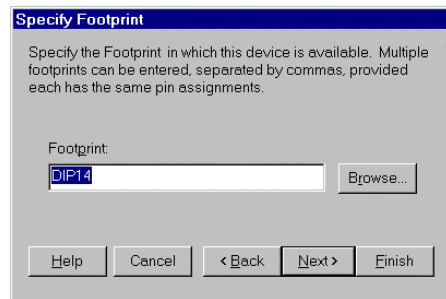
The next screen will let you select the type of pins to be added. For our circuit, we didn't need to do any simulation so we chose bi-directional pins for all of our symbols. To do this, click on **Select All** and then **Change Pins**. Another window will open. **Select All** again and then change the Pin Function box to "bidir."



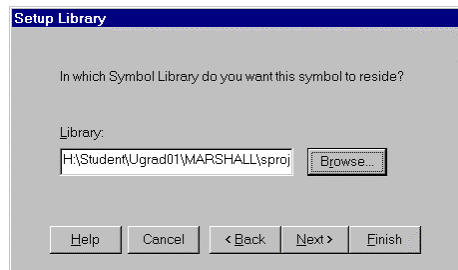
When you click OK, the Review Pins window will return showing the change in the ERC column. Then click OK, and the Setup Packaging window appears. We never changed this, click **Next**.



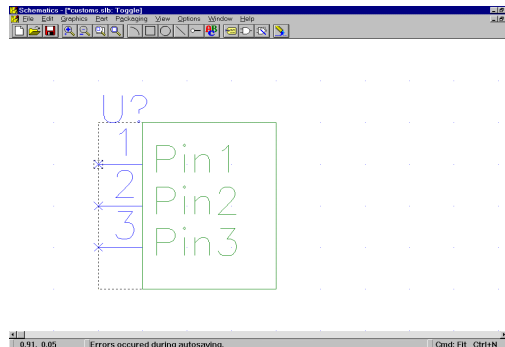
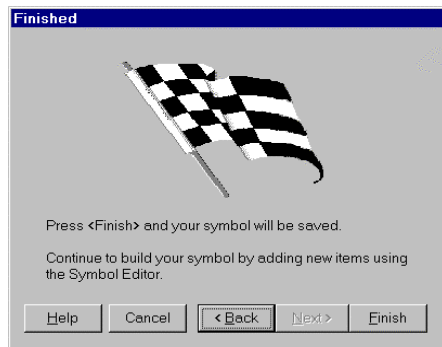
The next step is to specify which footprint to use for the new symbol. You should be able to browse and find your custom footprints. For the inverter, we created a symbol because in the schematics program the inverter is an individual inverter instead of a chip. The inverter is a standard pinout, so we used the DIP14 footprint. For the toggle, we browsed to find the toggle footprint.



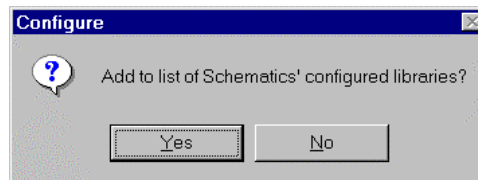
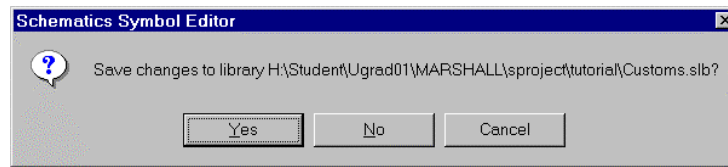
The next step is to save the new symbol library. The window will ask where you want to save the library. Browse to make sure that you are in the dedicated folder and then name your symbol library and save it.



For the next step, browse until you find the library you just created and click *Next* for the remaining windows until you reach the end.

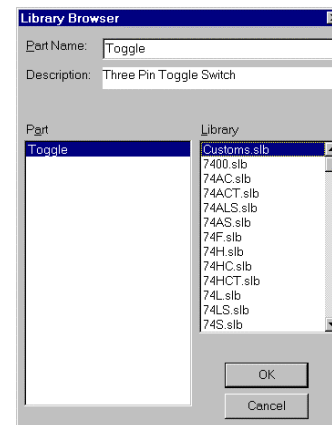
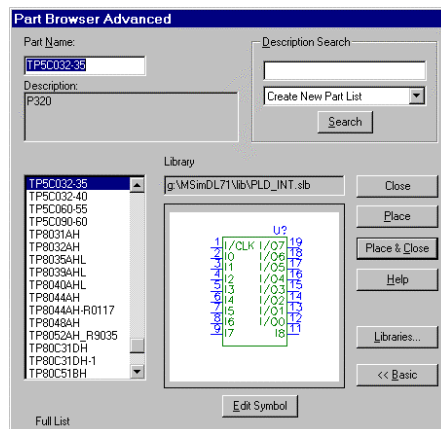
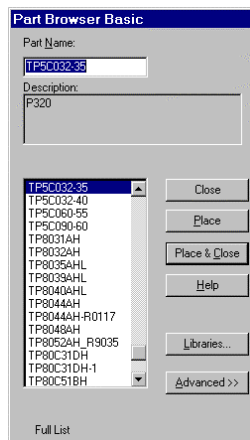


Once you have completed the symbol, close the Library Editor and click Yes to both windows that appear. If you don't add the library to the configured libraries, you won't be able to place your parts into the circuit.

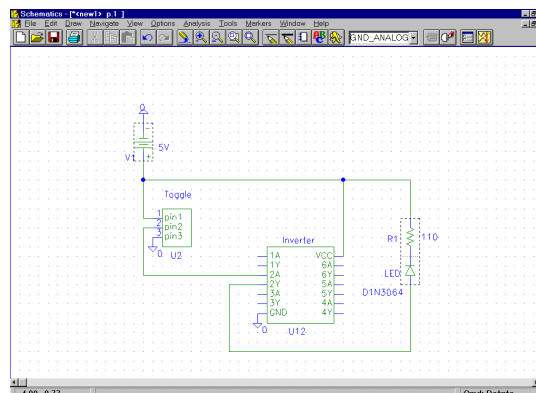


USING SCHEMATIC CAPTURE

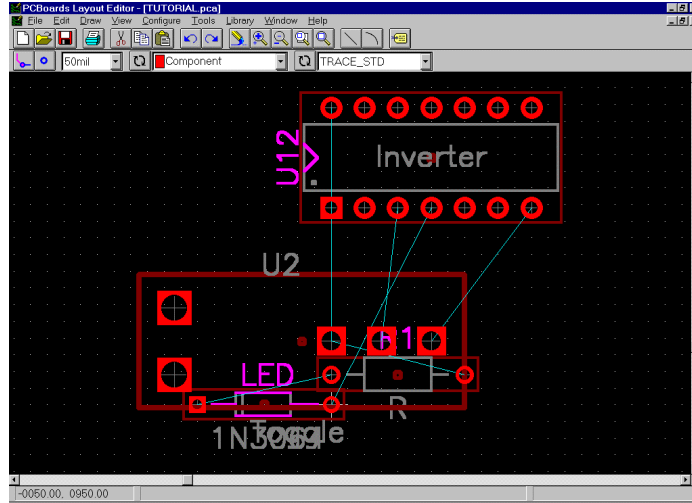
Now you should be able to use the Schematics program to connect all of your components. To insert parts, go to **Draw\Get New Part**. The basic browser will open. For convenience, click on the **Advanced** button. This allows you to see the part before placing it into your circuit. Select the part you want and click on the **Place & Close** button. If the part is one that you have created, click on the **Libraries** button and find the name of your library.



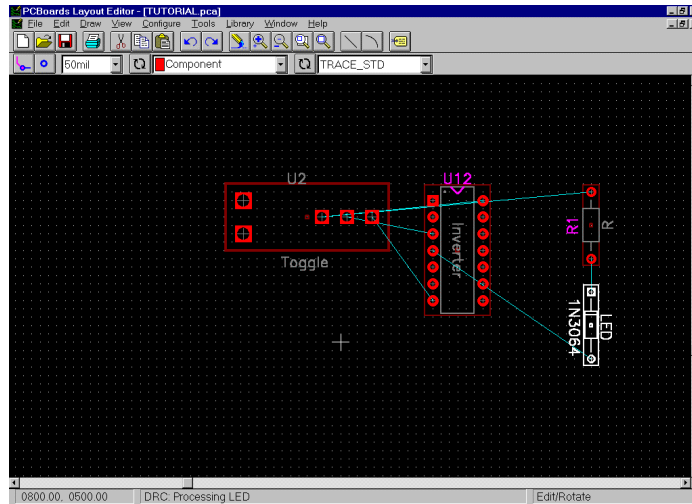
Once you have all of your components placed where you want them connect them in the appropriate way. The placement of the components is not critical because the PCB Layout placement will not be the same initially.



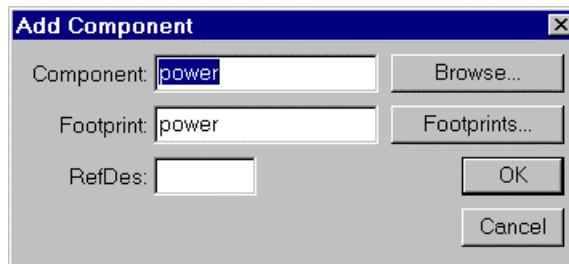
When everything is connected, save the file into your dedicated folder. Then go to **Tools\Run PCBords**. A warning appears asking if you want to “package” the schematic. Click **No**. Then it will ask if you want to update the netlist. Click **Yes**. If you have any Vcc or gnd components connected in the schematic a warning will be given stating that the simulation components were ignored. The PCB Layout software will be opened and your circuit will appear in the “ratsnest” form.



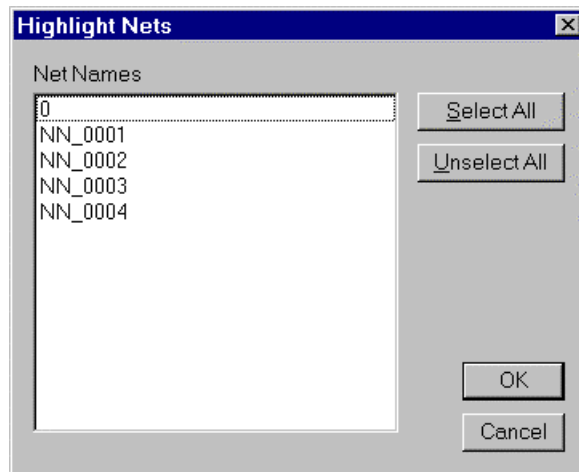
From here, you have to separate the components. If this was for a real board layout, the spacing would have to be exact, but for this example, we just separated them enough to see the connections clearly.



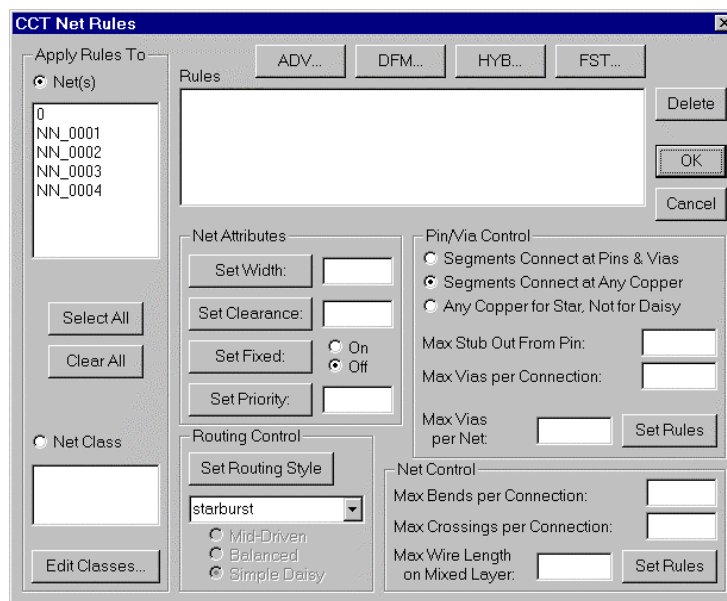
Next, we added the two pin header for the power and ground connections. To add the component go to the **Draw** menu and select **Add Component** and type the footprint name into the box.



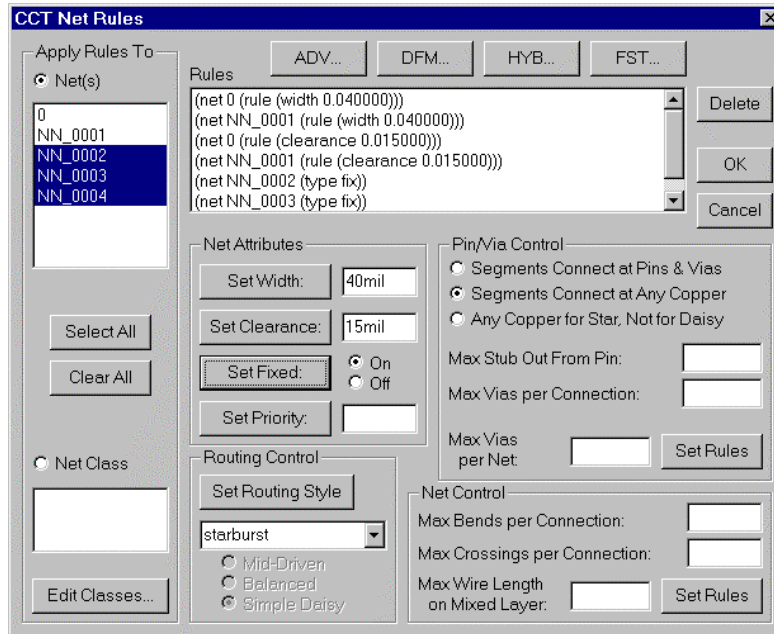
The two pin header footprint should appear. Place it where you want and then connect the pins to the corresponding component pins using the **Draw\Connection** option. Now you can prepare the circuit for autoroute. For our example we will make the power and ground traces thicker than the signal traces. To do this you need to know which “net” contains your power and ground connections. To find out which “nets” you need go to **View\Highlight Net**. This window allows you to highlight the connections one net at a time until you find the power and ground nets.



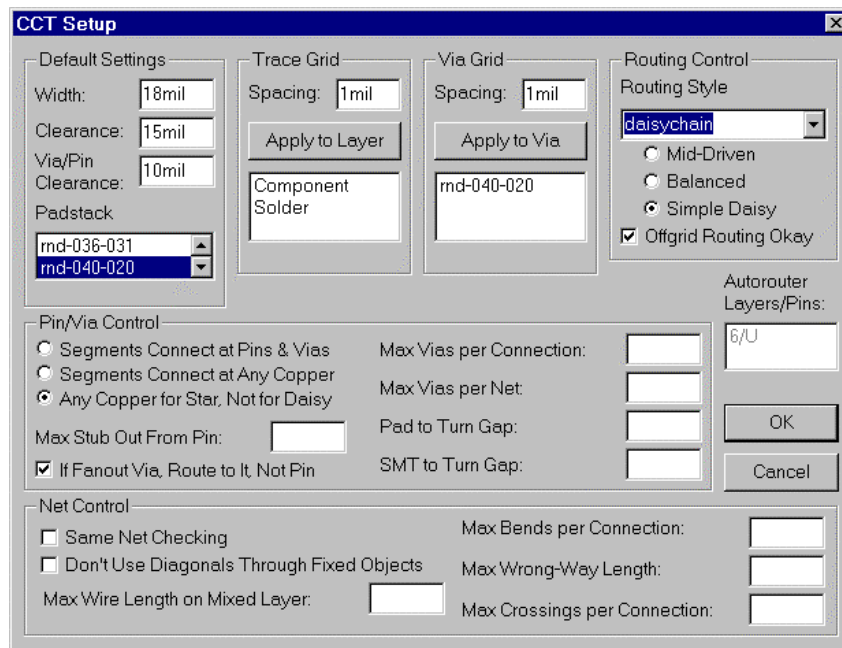
When you find the nets you need, remember them and click on the **Tools** pulldown menu. Go to **CCT: Net Rules**. This window lets you set the properties of individual nets and to select which nets will be autorouted.



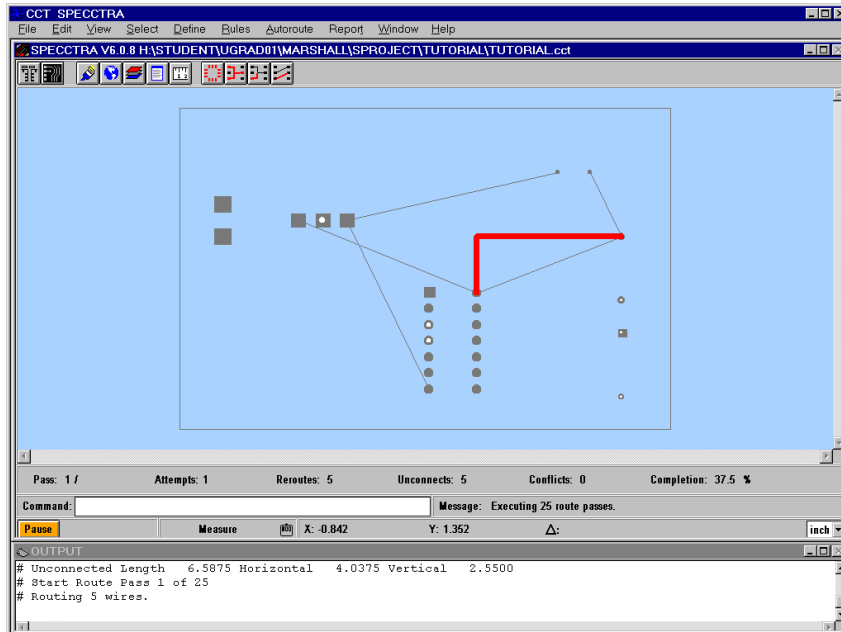
For our example, the power and ground nets were 0 and NN_001 so we highlighted them in the box on the left, then entered “40mil” into the Set Width box, and then clicked on **Set Width**. We then entered “15mil” into the Set Clearance box and clicked the button. In order to only autoroute the power and ground paths, we have to select all other nets and select the **On** circle and then click the **Set Fixed** button. The selections we made will autoroute the power and ground paths with 40mil thickness and 15mil separation, but will leave all other traces un-routed. The rules we set should appear in the Rules box. Click OK.



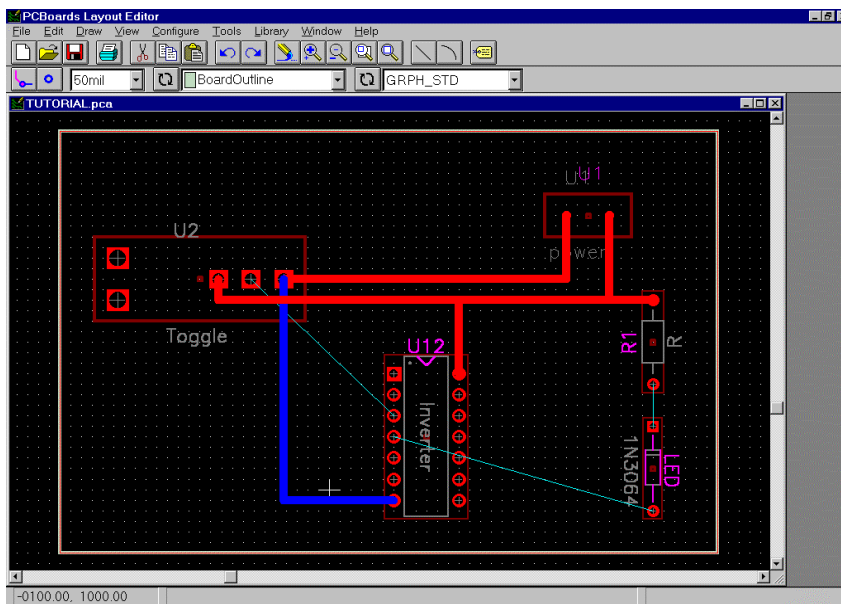
The next step is to select **Tools\CCT: Setup**. This window allows you to set the trace width and spacing of the paths to be created and to select the connection style, starburst or daisychain. Since we have set the trace width and spacing of the paths that will be routed, we only need to select the style. For the power and ground paths that are connected one pin to the next, the daisychain option is best.



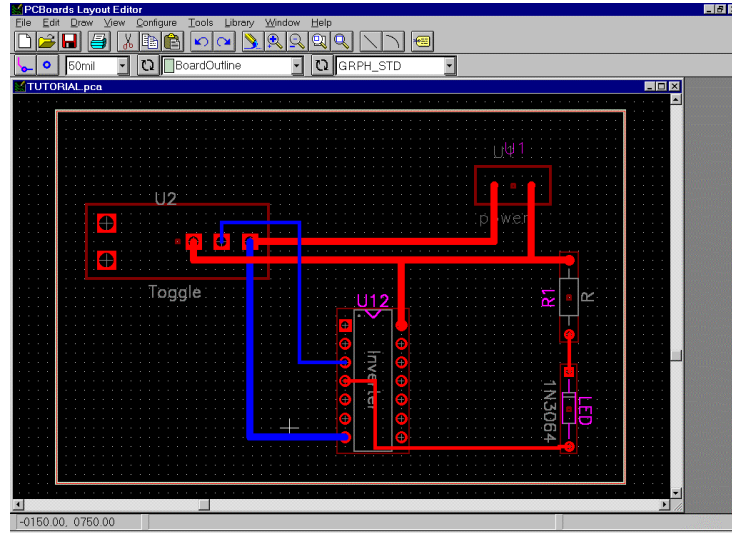
Once this is done you are ready to autoroute the board. Go to **Tools\CCT: Autoroute**. A new program window is opened and the program begins changing the thin connections to the thick traces.



It will continue to try different paths until there are no conflicts and then it will return you to the PCB layout software.



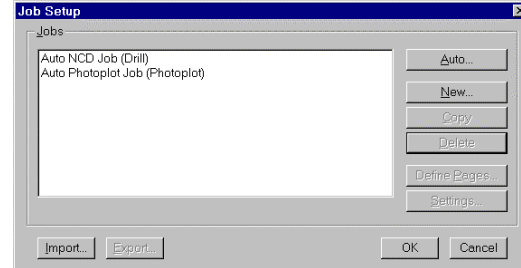
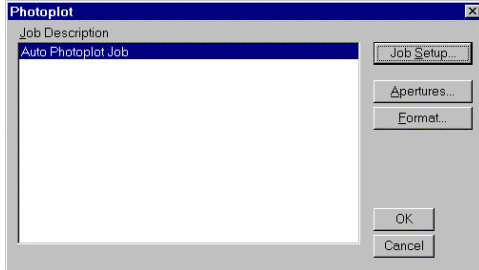
Now the power and ground traces are done. The red traces are on the top layer of the board and the blue ones are along the bottom side. The signal traces are still in the ratsnest connection form. To perform the autoroute on these connections, return to the *CCT: Net Rules* window and delete the rules previously set. Then select the power and ground nets and use the *Set Width* button to fix the nets. This will allow you to do the autoroute again without changing the power and ground traces. Then open the *CCT: Setup* window, set the trace width and separation to the desired amount, and change the style to starburst. Then run autoroute again. If the trace separation is too close there is a greater chance of shorts occurring on the board.



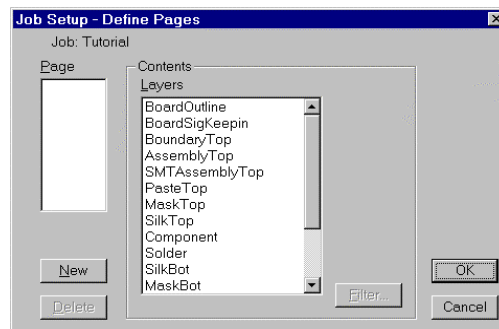
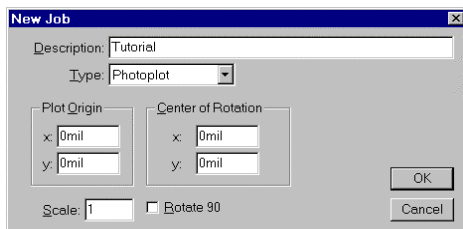
Now the board layout is complete.

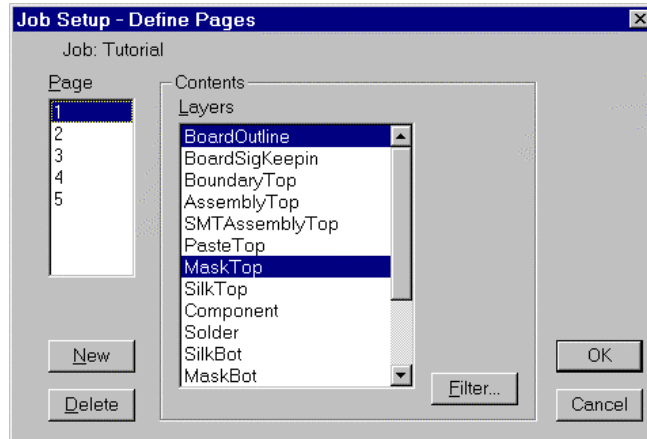
PREPARING FILES FOR THE PCB HOUSE

The files the manufacturer will need depends on the board house, but most will ask for the gerber files, the aperture list, and the NC drill file. To create these, start with the gerber files. Click on the **File** pulldown menu and select **PhotoPlot\Plot**. A window will open that says Auto Photoplot Job. You don't want that. Click on the **Job Setup** button. Click on the **New** button.

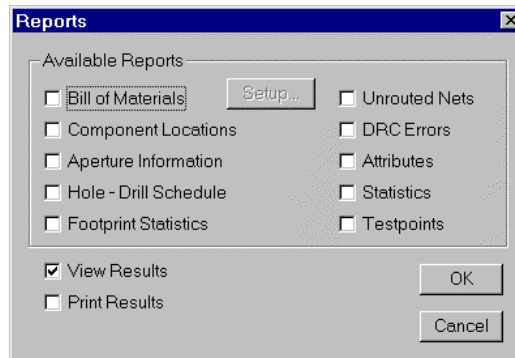


The New Job window lets you pick the type of new job. Select **PhotoPlot**. The next window lets you Define Pages. Each page will have its own gerber file. We defined five pages using the Board Outline layer in all of them. Highlight the Board Outline and the Mask Top then click **New**. Next, highlight the Board Outline and the Silk Top and click **New**. Repeat this process for Component, Solder and Drill layers. You should end up with five pages. Click OK.





Then make sure that the job you created is highlighted in the following windows and click OK. The gerber files should be created. Check your folder to see if there are any “.gxx” files. Next return to the **File** menu and select **Reports**.



Click on **Aperture Information** and OK. There should now be a “.apr” file in your folder. We also went to the **PhotoPlot** menu and selected **Apertures** then clicked on **Auto** and OK. We weren’t sure we needed this, but we did it just in case it made a file that they needed. Then we selected **NC Drill** from the **File** menu and clicked on **Output**. Then clicked OK and a “.d01” file was in our folder. The last file we needed was a readme text file; the manufacturer requested this file and told us what to say in it. We created it and then zipped all of the files to send off.

