McCAD™
EDS-Plus _____ *"The Designer's Tool"*

PCB Project Quick Start

for

Beginners

"Quickly Master the Basics"

VAMP INC

6753 Selma Ave.
Los Angeles, CA 90028
www.McCAD.com
TEL (1-323-) 466-5533 • FAX (1-323) 466-8564

Copyright

Technical Support

Technical support is available to licensed users of McCAD tools by calling, writing, faxing or E-mail:

Vamp Inc.
6753 Selma Ave.
Los Angeles, CA 90028
TEL (323) 466-5533
FAX (323) 466-8564
E-mail: support@mccad.com
WEB: http://www.McCAD.com

Before contacting Technical Support the user should be familiar with the Microsoft Windows OS or Mac OS procedures and should have read this manual. Also, please itemize all questions on paper before calling.

Technical Support personnel will respond to questions regarding the operation and use of VAMP Inc. software ***only***.

Extended Software Support

After the initial 30 day support period, Extended Software Support is available for an annual fee. It entitles the subscriber to updates, E-mail and phone support. For additional information contact VAMP Inc.

VAMP Inc.
6753 Selma Avenue
Los Angeles, California 90028
USA
Phone: (323) 466-5533
FAX: (323) 466-8564
E-mail: sales@mccad.com
WEB: http://www.McCAD.com

Software Requirements

To use this documentation you will need to install McCAD EDS Plus on your Computer. This software is available from http://www.McCAD.com. It is available for either MS Windows or MacOS operating systems. Download whichever archive suits your needs.

Almost all of the example demonstrations can be done with the FREE LITE versions of the EDS Plus tool set. If you have the tool set already installed and licensed for PRO mode, then you will be able to run some of the more demanding examples.

About the Screen Graphics

This manual was created using the Macintosh version of the EDS Plus tool set. The graphics in the MS Windows OS will look very similar and so we have not included those to reduce the redundancy of information as much as possible.We have tried to focus the screen graphics within this manual as much as possible to avoid the cosmetic differences of each OS's windowing display. The procedural information presented is the same for both operating systems. Differences are noted where relevant.

Chapter 1
Simple PCB Project

What follows is a simple example of a project starting with a schematic created in McCAD Schematics Plus, moving that project to the PCB layout environment and then having that layout partially auto-routed in McCAD Trailblazer.

This tutorial will provide you with the a starting foundation of how the system works. We will not cover all aspects or features available in the respective tools which we will be using. Once you have completed this tutorial you should be able to begin your own projects.

This tutorial assumes that you have properly installed all of the tools that will we used. If not, install the tools before proceeding.

The figures you will see for the most part come from the Macintosh versions of the applications. The Windows versions will look almost identical.

Introduction

The McCAD EDS Plus design system is comprised of independent modules which work together. These modules are the Schematic Plus, PCB-ST and Gerber Translator. We will also demonstrate how McCAD Trailblazer can be used to auto-route a layout to completion.

As part of this tutorial all of the files used in this tutorial can be found in the PCB Tutorial Folder which is included with this tutorial archive.

Creating the Schematic

Any project must have a complete schematic. All the symbols must be properly attributed with the appropriate information before moving on to the next step in the design phase. For this tutorial we will be creating a simple OpAmp AM Modulator and putting it on to a PCB layout. If you have already done the Simulation Quick-Start Tutorial you will recognize most of the circuit.

The following circuit is based on the Op-Amp circuit which was created in the simulation examples. Either duplicate that project and reedit it to match the circuit below, or start a new project and create the following circuit as follows.

Creating a Project

1. Create a folder directory labelled ***MyPCBProjectFolder***. This is where you will put the various documents which you will be creating with this tutorial. You may compare them with the files (found in PCB Tutorial Folder) which are included with this archive.
2. Begin by launching the McCAD Schematics Plus module.
3. In the File Menu select ***New -> Project***.
4. In the dialogue window which appears label the project as ***PCB Project Example.prx*** . Be sure to place your project file in the ***MyPCBProjectFolder.***

5. You will next be presented with the Settings for your project. Click OK for now. This will cause an empty project to appear in the upper right of your screen.

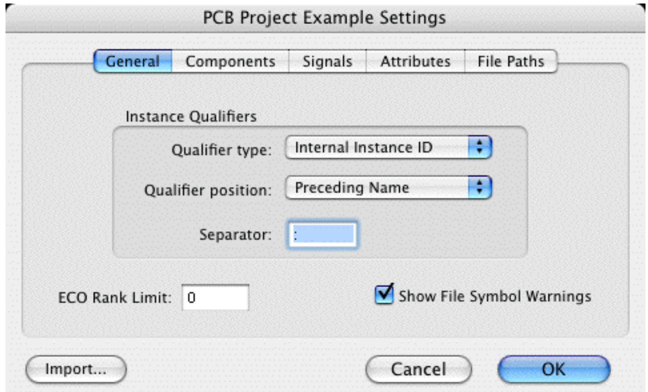


Figure 1-1 • Project Settings Panel

Creating Sheets & Adding them to the Project

6. In the **File** menu select ***New->Schematic Sheet***. A blank untitled schematic sheet will now appear.
7. In the **File** menu select **Save** and then title the drawing as ***PCB_Example.shx*** . Be sure to place your sheet file in the ***MyPCBProjectFolder.***
8. In the Project window click on the add sheets button at the top.

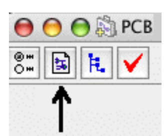


Figure 1-2 • Add Sheet to Project Button

9. A dialogue window will appear. Locate the sheet file ***PCB_Example.shx*** you created earlier and select OPEN**.** This sheet will now appear in the project window.

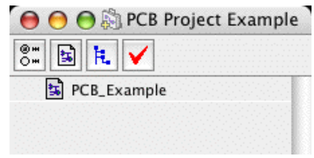


Figure 1-3 • Project with sheet added.

10. At the top of the schematic sheet make sure that the reveal tab is in pointing in the bottom direction.

Reveal Control

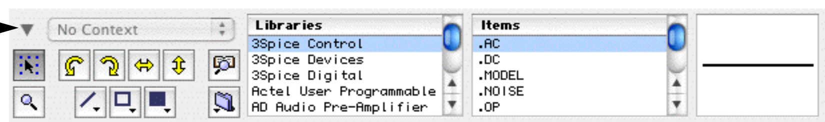


Figure 1-4 • Schematic Reveal Control

Drawing the Circuit

11. Locate the symbols needed for the following schematic. The symbols can be found in the 3Spice and Connectors library.

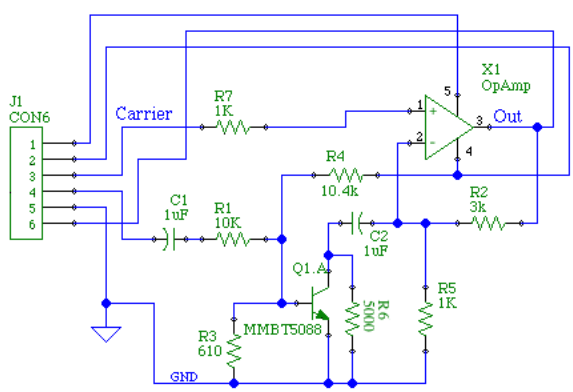


Figure 1-5 • Op-Amp Based AM Modulator Circuit.

12. Complete the wiring of the circuit as shown in Figure 1-5.
    - R1 -> 10K
    - R2 -> 3K
    - R3 -> 610
    - R4 -> 10.4K
    - R5 -> 1K
    - R6 -> 5K
    - R7 -> 1K
    - C1 -> 1uF
    - C2 -> 1uF

At this point you have a wiring diagram of the circuit. However before you can proceed to a layout of a PCB you will have to assign the footprints that all of the components will be using.

✎IMPORTANT NOTE;

Many of the symbols in the schematic libraries do not have footprints assigned. This is because the possible variants are enormous. Therefore as you start to use the system you will either be creating your own custom libraries or tailoring the existing libraries based on the exact devices you will be using.

13. Double-Click on R7. The will reveal the resistor symbol's attributes

Click On First Free Line

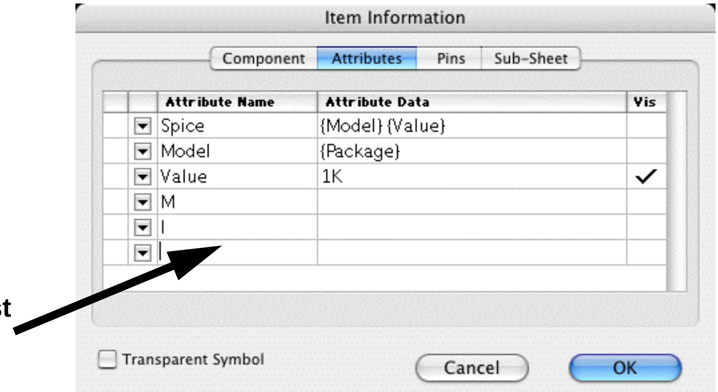


Figure 1-6 • Resistor Attributes Panel

14. In the column **Attributes Name** click on the first free line.
15. Type "**Footprint**" . In the column Attribute Data we will type the actual footprint specification.

At this point we must specify a footprint. PCB-ST affords two methodologies for doing this. The first is the use of the Macro-Generation and the second is a Specific Footprint Library Name located in the PCB-ST's List of library footprints. Will will use both methods so that you can see how both work.

In the figure below you will find a summary of both footprint specification techniques.

PCB-ST PARTS LIST REPORT FIELDS

	1	2	3	4	5			
FOOTPRINT CALL-OUT	PART NAME	REF. DESIG-NATOR	VALUE	*Reserved*	FOOTPRINT NAME			
	TAB	TAB	TAB	TAB				CR
	1	2	3	4	5	6	7	8
PATTERN GENERATION	PART NAME	REF. DESIG-NATOR	VALUE	*Reserved*	PADS	OD	ID	
					VDIP	PIN COUNT	ROWSPACE	
					HDIP	PIN COUNT	ROWSPACE	
					VDBS	PIN COUNT	ID	
					TOS	PIN COUNT	ID	
					VSMDIP	PIN COUNT	ROWSPACE	
					HSMDIP	PIN COUNT	ROWSPACE	
					VSOT	PIN COUNT	ROWSPACE	
					HSOT	PIN COUNT	ROWSPACE	
					CCP	STYLE #		
					FLTPK	STYLE #		
					HEDGCON	PIN COUNT	PIN SPACE	PINWIDTH
					VEDGCON	PIN COUNT	PIN SPACE	PINWIDTH
					ARRY	X-PINS	GRID	Y-PINS
					EURO	TYPE	ID	PADS
					PADPR	OD	ID	PADSPACE
	TAB	TAB	TAB	TAB	TAB	TAB	TAB	CR

Figure 1-7 • McCAD PCB-ST Parts List File Format

In the first example we will use ST's internal macro generators. In the above table we will look at **columns 5 thru 8** in the lower half of that table. For a simple two hole resistor we could use a pair of pads separated by a specified distance. The macro name we will use is PADPR.

16. In the Attribute Data column adjacent to the Attribute field Footprint you just created type the following:

```
PADPR\t60\t32\t400
```

This specifies a 0.060" round pads, 0.032" drill hole and a center to center distance of 0.400" .

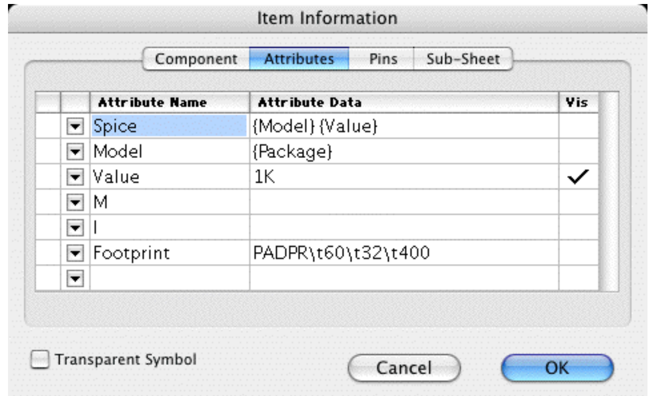


Figure 1-8 • R7 Attributes Panel with Footprint data

✎Important Note:

*The " **\t**" represents the TABs used as field separators as shown in the previous table. These are required.*

17. For the sake of simplicity we will use the same footprint string specification for all of the resistors and capacitors in this project. Therefore access all of the Attribute Panels for the resistors and capacitors and use the same attribute & footprint specification shown.
18. Access the **Attribute Panel** for the Op-Amp. Create the **Footprint** attribute and in the Attribute Data column type:

```
\F.DIP-8
```

***DIP-8** is the actual name of a footprint that can be found in the external footprint libraries shipped with the PCB-ST system. The "**\F.**" is an optional prefix. It signals the PCB-ST to search the external libraries first.*

19. Access the Attributes Panel for the transistor. Create the **Footprint** attribute and in the Attribute Data column type:

```
\F.TO92A
```

***TO92A** is the actual name of a footprint that can be found in the external footprint libraries shipped with the PCB-ST system. The "**\F.**" is an optional prefix. It signals the PCB-ST to search the external libraries first.*

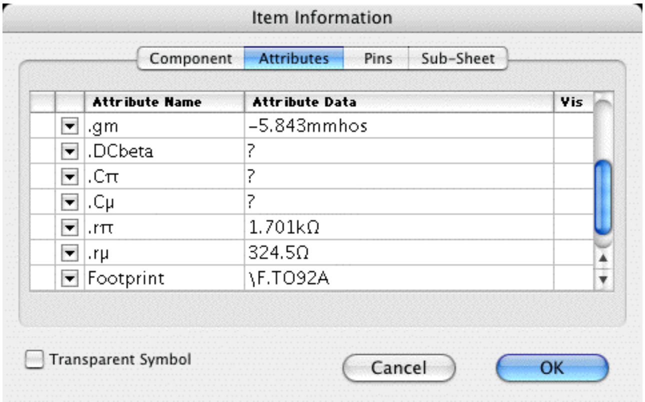


Figure 1-9 • Transistor with Footprint Attribute Specified

20. Access the **Attribute Panel** for the J1. Create the **Footprint** attribute and in the Attribute Data column type:

```
\F.MyOwn6Pin
```

`MyOwn6Pin` *is the actual name of a footprint that will be found in the external footprint libraries of PCB-ST system. This will be a Custom Footprint which we will create later in this tutorial. The "****\F.****" is an optional prefix. It signals the PCB-ST to search the external libraries first.*

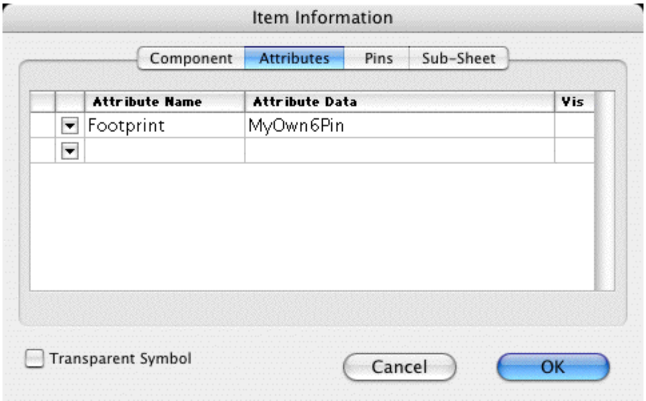


Figure 1-10 • Footprint Specification without the Prefix

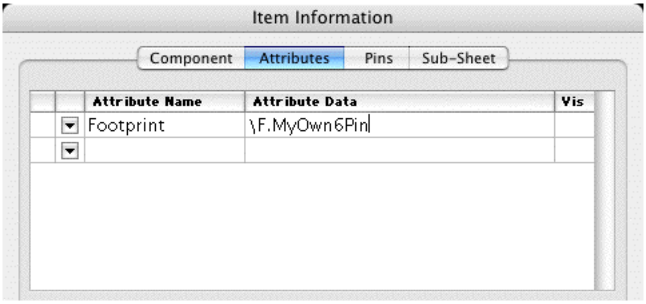


Figure 1-11 • J1 Footprint specification with Prefix

Either Footprint specification will work with the later allowing faster processing in large designs.

We have completed the creation of the schematic at this point. Before proceeding to the PCB Layout we will extract a standard PCB-ST parts file from the schematic.

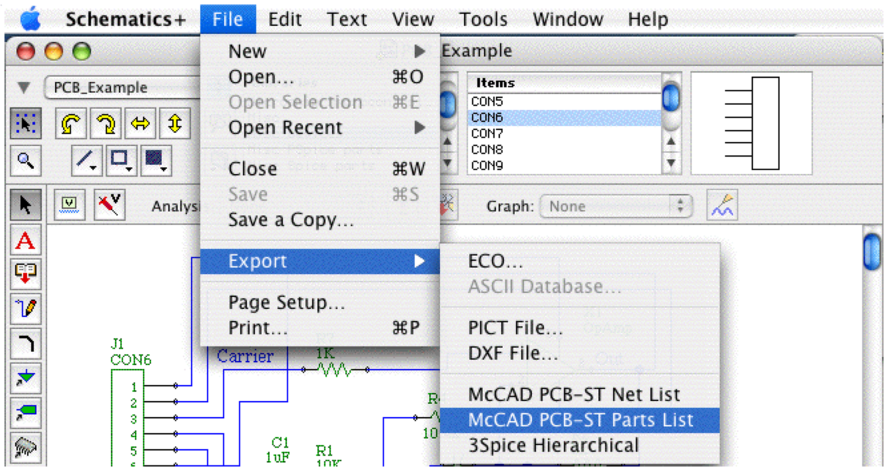
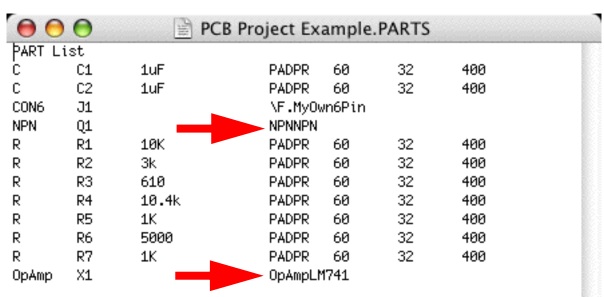


Figure 1-12 • Exporting an Parts list

This is not normally necessary but is handy in determining if you have all of your components properly attributed. In the following two figures you will first see an incomplete parts list (it was extracted before all components were properly attributed) followed by a figure showing a complete parts list record.

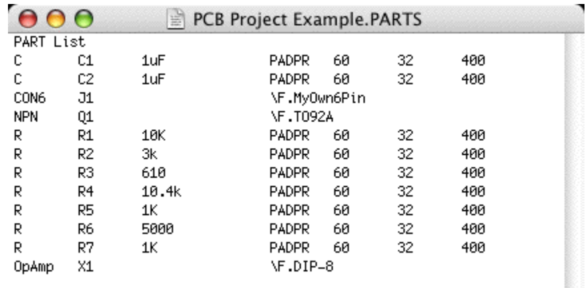


PCB Project Example.PARTS

PART List

C	C1	1uF	PADPR	60	32	400
C	C2	1uF	PADPR	60	32	400
CON6	J1		\F.MyOwn6Pin			
NPN	Q1		NPNNPN			
R	R1	10K	PADPR	60	32	400
R	R2	3k	PADPR	60	32	400
R	R3	610	PADPR	60	32	400
R	R4	10.4k	PADPR	60	32	400
R	R5	1K	PADPR	60	32	400
R	R6	5000	PADPR	60	32	400
R	R7	1K	PADPR	60	32	400
OpAmp	X1		OpAmpLM741			

Figure 1-13 • Incomplete Part List.. with Errors



C	C1	1uF	PADPR	60	32	400
C	C2	1uF	PADPR	60	32	400
CON6	J1		\F.MyOwn6Pin			
NPN	Q1		\F.TO92A			
R	R1	10K	PADPR	60	32	400
R	R2	3k	PADPR	60	32	400
R	R3	610	PADPR	60	32	400
R	R4	10.4k	PADPR	60	32	400
R	R5	1K	PADPR	60	32	400
R	R6	5000	PADPR	60	32	400
R	R7	1K	PADPR	60	32	400
OpAmp	X1		\F.DIP-8			

Figure 1-14 • Complete Parts List... No Errors

To complete the schematic portion of this project we now need to extract an **ECO** document. This stands for **E**ngineering **C**hange **O**rder. This document is used to synchronize the Schematic and Layout environments.

21. In the File Menu select **Export ->ECO**.

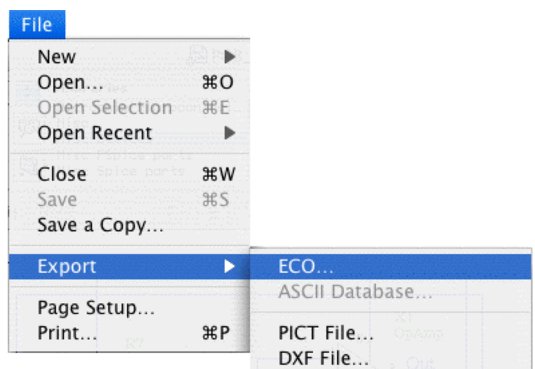


Figure 1-15 • Exporting an ECO document

22. Save your Project and close the Schematics Plus application.

Creating the PCB Layout

In this portion of the Tutorial Project we will take the ECO document created in the Schematic Plus module and bring it into a blank layout design window. We will build a custom footprint in the ST footprint library. We will arrange the footprints in preparation for layout.

PCB-ST System Architecture

Before actually beginning the design we will momentarily divert the discussion to a very brief description of how the design environment is structurally set up. It is very important to understand this as it provides tremendous flexibility when compared to other systems and failing to grasp its method of use may lead to problems with some of your own designs in the future.

The layout environment is made up of 32 individual layers. Each layer behaves like a single sheet of transparent material upon which you may place or draw graphics. Thus the environment is simply a stack of 32 transparent sheets. Clicking in the overhead tool bar on a specific layer button informs the software that this is the sheet you will be working with and therefore you want it to become active. The sheet which currently active will have its layer button blink. When a screen is redrawn, the currently active layer is always painted last thus moving it to the top of the working stack on your computer screen.

To produce the individual production layers of a PCB, these individual transparent layers are arranged in fabrication stacks known as **Fabrication Tooling** definitions. These definitions are created using the **Create Fabrication Tooling** command found in the **Art** Menu. In the dialogue which appears you can specify which transparent layers are to be combined to produced the necessary Photo tooling for each of your PCB's production layers. The definitions found in this dialogue represent the backbone definitions for your design and these definitions are also instrumental in determining how your mouse clicks will be interpreted while you are designing your board.

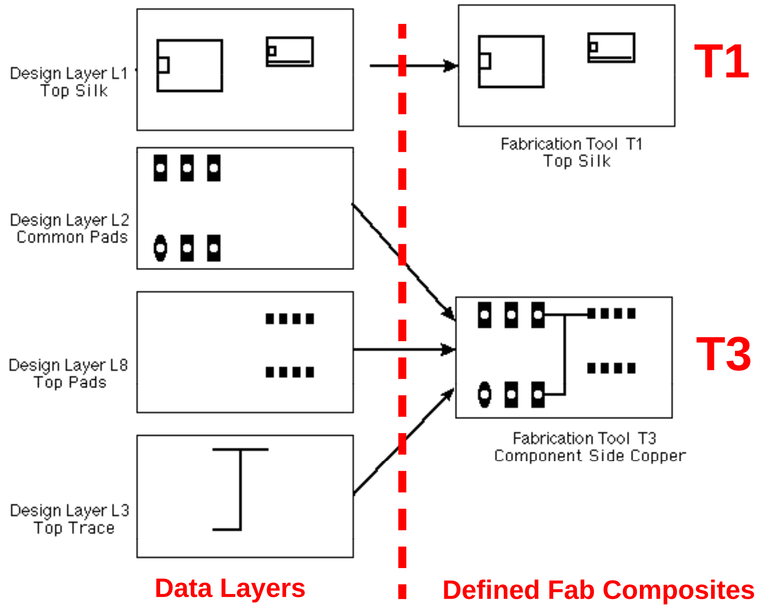


Figure 1-16 • ST System Architecture

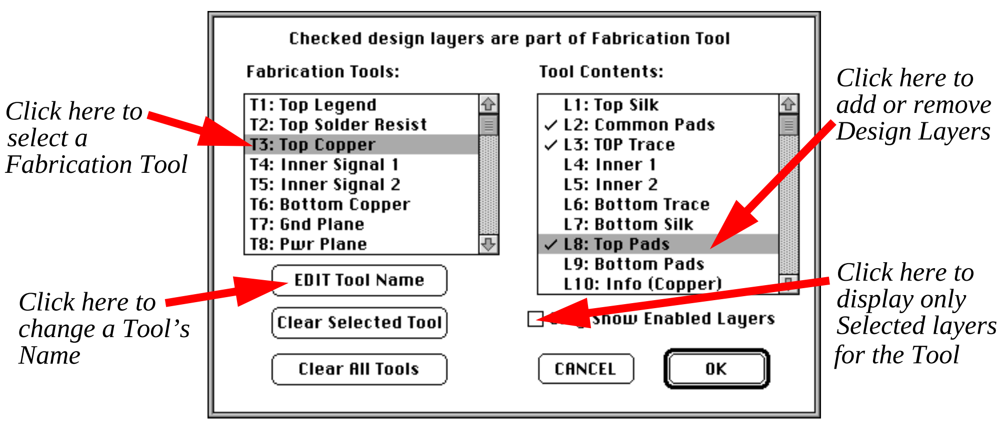


Figure 1-17 • ST Tooling Dialogue

For our current project we will use the default tooling setup as shipped with the PCB-ST software. This configuration will meet the needs of most designers.

Defining the PCB board's perimeter

1. Launch the PCB-ST Module. You will be presented with a blank untitled design window.
2. In the top level tools pallet on the left select the Board Layout Button.

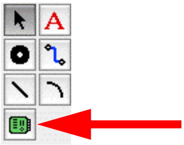


Figure 1-18 • Selecting Board Definition Tools

3. In the new tool pallet, select the Perimeter Line tool.

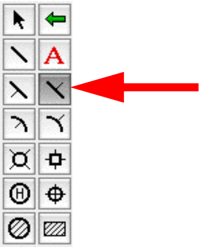


Figure 1-19 • Selecting the Perimeter Line Tool

4. Carefully draw a rectangular boundary for the PCB design which is 3" by 2".

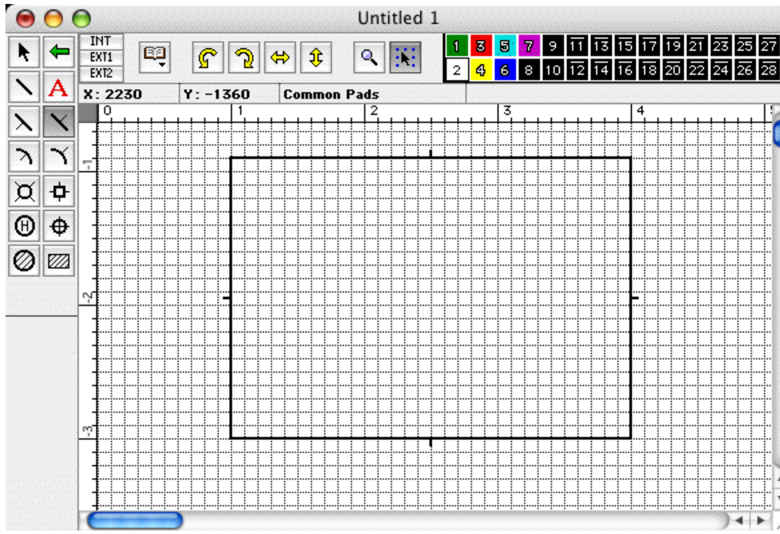


Figure 1-20 • Board Perimeter Defined

Importing the Schematic's ECO

5. In the **File Menu** select **Import -> ECO**. You will be presented with an OPEN dialogue. Locate the ECO you created in the Schematics portion of this tutorial and select OPEN.

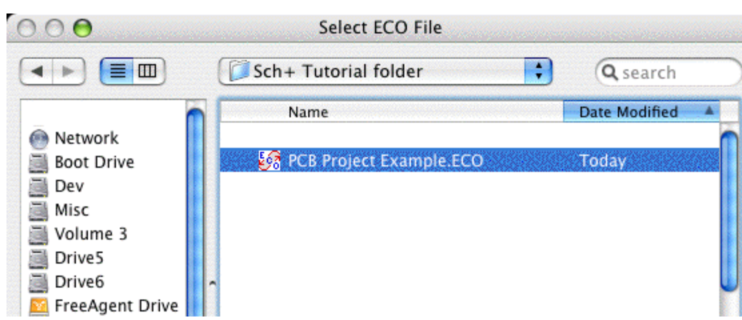


Figure 1-21 • Locating the ECO Document

6. You will next be presented with the ECO Import Options.

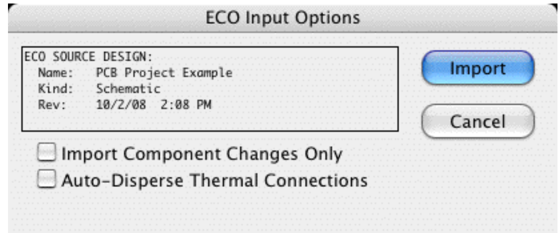


Figure 1-22 • ECO Import Options

7. **Continue** by selecting the **Import** button.

PCB-ST will compare the incoming ECO file with its empty condition in the new window. It will scan its current list of libraries looking the necessary footprints specified in the ECO document. These footprints will be placed at the bottom of the design work surface.

8. Once the import is complete scroll the view downwards to reveal the footprints that have been pulled from the ST libraries and placed at the bottom.

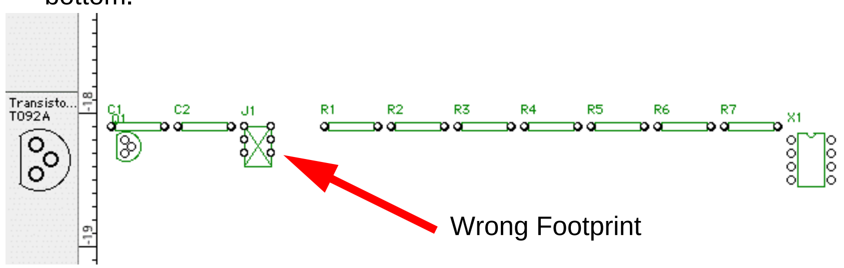
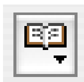


Figure 1-23 • J1 .. Missing Footprint Symbol

You should see that most of the desired footprints have either been created or pulled into the work area. However you should note that there is a problem. J1 does not have the correct footprint. A symbol similar to one you see for J1 will be used anytime the correct footprint for the requested footprint can not be located within the ST's currently known list of footprints. In this particular instance the footprint did not exist and so it must be created before we can proceed with our design.

Creating a needed footprint

9. We will create this needed footprint in the Library Editor window and save it into a new Library file created just for this exercise.



10. In the Library Access button found in the top tool bar click with your mouse. A pop-up menu will appear and within that pop-up select **NEW..** which will allow you to create a few library file which will hold the footprint we are about to create.

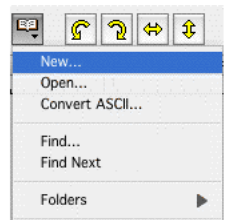
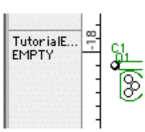


Figure 1-24 • Creating a New Library File

11. This cause a PUT dialogue to appear. Give this file the name TutorialExample.plb and save the file to the same directory as the application.

✎Important Note:

Saving to same directory as the application is important for this exercise. Later you can read in the main manual about relocating the library if you decide to keep it.



12. The system will open this file and scan for the first footprint in the list. Since this is a new library it will be empty.
13. With your mouse click below where it says EMPTY. This will cause the menus to focus towards the library editor.
14. In the **File** menu select the **NEW Library Footprint...** command.

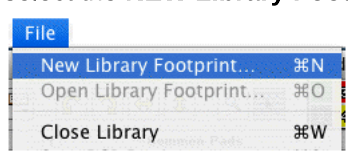


Figure 1-25 • Opening an New Library Editor

15. A new Library Editor Window will now appear.

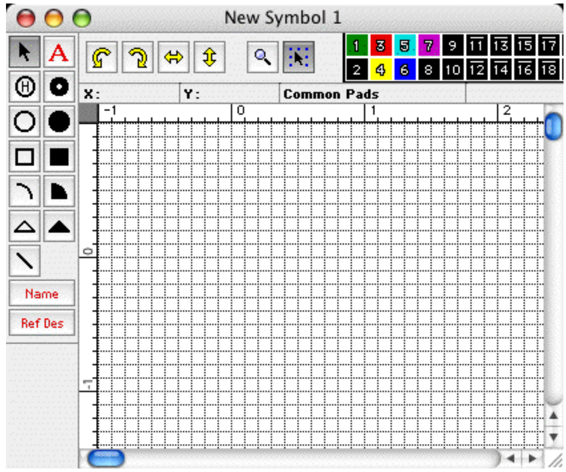


Figure 1-26 • Untitled Library Editor Window

This editor window will be very similar in design and function to that of the main design window, however its primary use is solely for the creation of footprints.

16. Select Data Layer 2 in the top tool bar. Then select the round doughnut pad in the left tool palette. Starting at location 0,0 place a total of six pads in any arrangement you wish. You can copy the figure below if you wish.

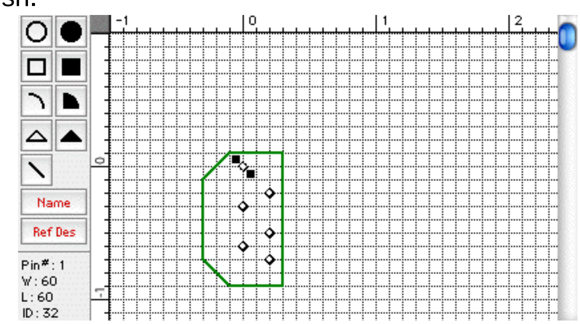


Figure 1-27 • Placed Pads and Silk Outline

17. Select Data Layer 1 and using the line tool from the palette on the left draw an outline of the connector.

✎**Note:**

Assuming that you placed the pads consecutively they should be numbered 1 to 6 as needed in the schematic. You can check the pin numbering by first selecting the pointer from the tool palette and then switching back to Data Layer 2. When you click on each pad individually the information about each pad will display on the left below the tool palette. If the numbering is incorrect double-click on the pad itself to bring up its labeling dialogue and relabel the pad as needed.

18. Once you are happy with the footprint select **SAVE** from the **File** menu.
19. In the labeling dialogue which appears enter **MyOwn6Pin** so that it will match the name we used in the schematic we created earlier.

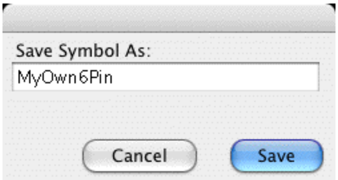


Figure 1-28 • Labeling the PCB Footprint

20. Close the Library Editor Window.

Having created the needed footprint we can now return to the main design window. There are several ways to replace the problem footprint in our design window. In this example we will simple delete the footprint and re-import the ECO we did earlier.

21. With the pointer select the bad footprint. It will highlight. Then hit the **Delete Key.**
22. Go back to step step 5. on page 21 and import the ECO again.

23. Your updated ECO import should look similar to the figure below.

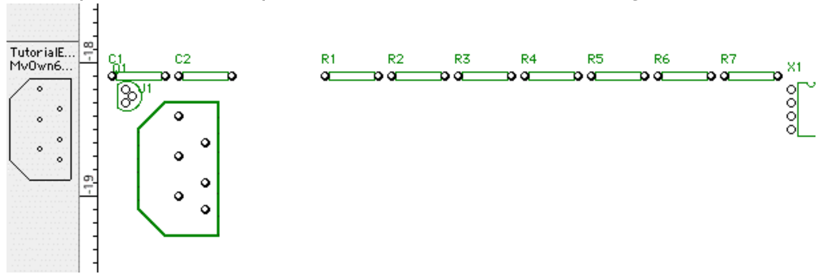


Figure 1-29 • PCB Footprints after last update.

24. Reposition your view so that you can see the perimeter of the board we created earlier.

✎ **Note:**

At this point all of the footprints are marked as having been un-placed; meaning that they are not in position. Once you drag or move these from their initial position they will lose their un-placed status.

Arranging the footprints

We will next position the footprints into their final positions within the board perimeter we created earlier. This can be done a number of different ways, however for this demonstration we will use the most simple method which will also demonstrate the dynamic reconnect of the nets as well.

25. In the **Utilities** menu in the **Dynamic Place** submenu select the **Get Unplaced Device**. You can also use the keyboard short cuts. Using this command will get the first un-placed device from the software's internal list and attach it to the tip of the mouse pointer.

    CTRL U (MS Windows)
    CMD U (MAC OSX)

26. Move the mouse pointer to the desired location. While attached to the pointer you can use the rotation buttons in the top tool bar to change the orientation of the device that is attached. You will also note that the rat lines will rubber band as you move the footprint. While moving the footprint the system will recompute the shortest path to any connection

point that is a member of that same net. This can be used to help you determine the best placement to minimize copper track cross-overs.

27. Click the mouse when you are ready to anchor the footprint at the desired location. Repeat the previous two steps until you have placed all of the devices as shown in the following figure.

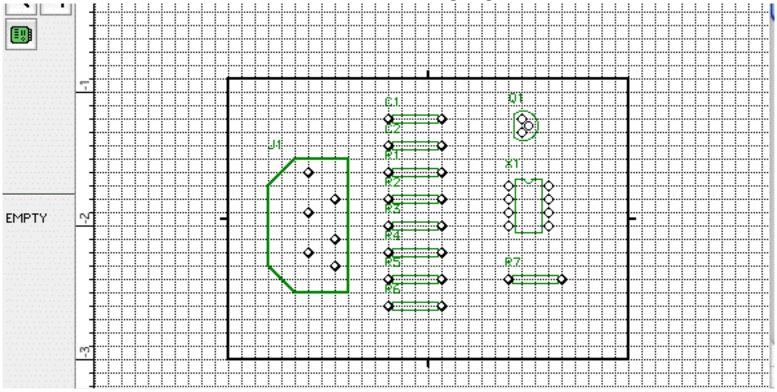


Figure 1-30 • Placed Footprints

28. Refresh the screen as needed using the **Redraw Screen** command in the **Layout** menu.

Manually Routing the Interconnects

Once you have the footprints in position, you are ready to begin interconnecting the various pins. The Net List has been imported through the ECO process and the Rats Nest which was created from it, resides on layer 20 which is currently in the NoDraw mode and thus is not being displayed.

29. To enable the display of the imported nets list we shall use a shortcut technique. Move your pointer over the data layer button (in the top toolbar) which contains the rats nest; usually layer 20. While depressing the OPTION Key (Mac) or CTRL Key (Windows) click the mouse button once. You should observe that the over-score line in the button will disappear. When the over-score line is present this indicates that that layer will not be drawn when the screen is refreshed.

30. Refresh the screen display.

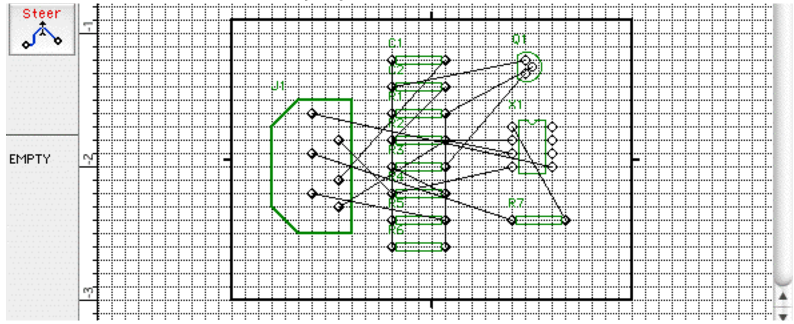


Figure 1-31 • Layout with Rats Nest layer enabled

Before beginning the routing let's quickly review the various cursors that the system will display.

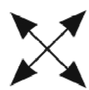
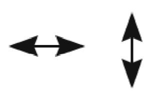
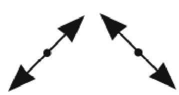
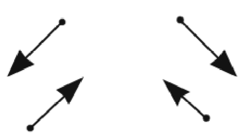
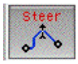
	Dragging Trace Junctions & Vias ***Un-Constrained***
	Dragging Segments ***Constrained***
	Resizing 45° Segments ***Constrained***
	Dragging Segment Intersections Donut Star Pointer ***Un-Constrained***
	Creating 45° Segments ***Constrained***

Figure 1-32 • Wire Movement Cursor Types

Clicking on a rat line or existing piece of copper track will cause one of these movement cursors to appear indicating which way you have available movement. You will observe that some operations will automatically have imposed directional constraints.

✎Important Note:

*Remember that when you click on either a rat line or copper track the data layer associated with the item **MUST** be currently active otherwise nothing will happen. The currently active layer is the one with the blinking button in the top tool bar.*



STEERing Switch

Since we will next be demonstrating the use of **STEER**ing there are two additional Cursors which the system will display depending the mode we are in.

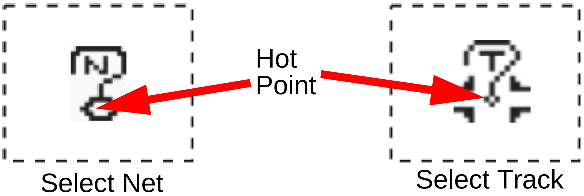


Figure 1-33 • STEERing...Net and Track Finding Cursors

Initially the system will ask you for the Net you wish to work with. That cursor will have the **“N”** in it. Click on a Device pin or copper track will identify the signal you plan to work. The entire net will high light and the cursor will change to one containing the letter **“T”**. The high lighted net will have one or more rat segments. Move the hot point over the desired segment and click the mouse button to select the segment for manipulation. Once the segment is located by the system, the cursor will change yet again to indicate that you have un-constrained movement.

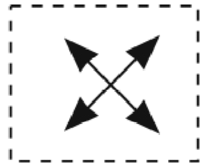


Figure 1-34 • Un-Constrained Steering Cursor

You will also note that both ends of the rat line are anchored to their source and terminus as well as the mouse's hot-point. The rat will rubber band with the movement of the mouse and every time you click the mouse you will anchor a copper track leg on the currently active data layer. While in this mode you can change data layers while routing the path to the terminus. If you change layers the system will automatically insert vias as needed. Upon reaching the terminus the cursor will switch back the STEERing **"T"** cursor so that you can select the next track in the net if there is one.

To stop or begin working with a different net, hit the ESC Key and you will see the STEERing **"N"** cursor appear allowing you to select a different net.

31. Select Data Layer 3.
32. Select the STEERing Switch in the routing Tool Bar on the left.
33. Move the pointer to the lower-left pin on J1. This will high-light the rat line.
34. Next click in the highlighted rat line on the end to the J1 pin.
35. Route the path as shown.

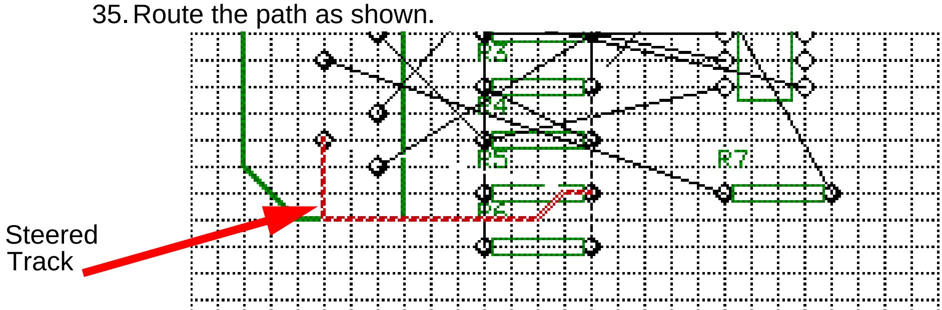


Figure 1-35 • STEERing First Highlighted Net.

36. Hit the ESC key so that we can now select a different net.
37. Repeat this STEERing sequence with other rats. Use the following figure as a guide.

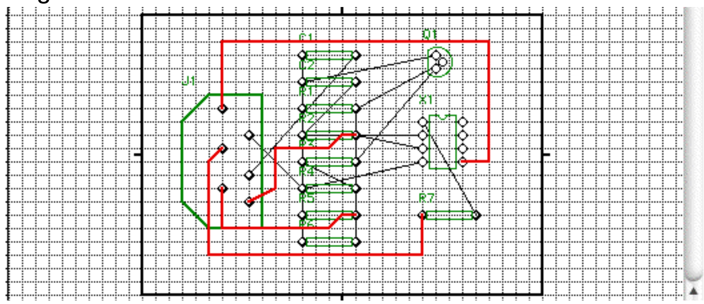


Figure 1-36 • More STEERed Signals

✎ Note:

If you decide to switch trace layers to avoid a collision with another trace the software will automatically insert a via for you at the last anchor point prior to making the layer switch.

You may continue manually routing this design until completed. However we will take this opportunity to demonstrate taking this partially routed design into the optional autorouting software module to have this tool finish the routing of the board for you. If you do not plan to use any autorouting capabilities then complete the manual routing of this design and then proceed to **Power Planes & Thermal Pad Set-up on page 35**.

To use the autorouting environments you will need to export a router document.

38. In the File Menu select Export-> Trailblazer.

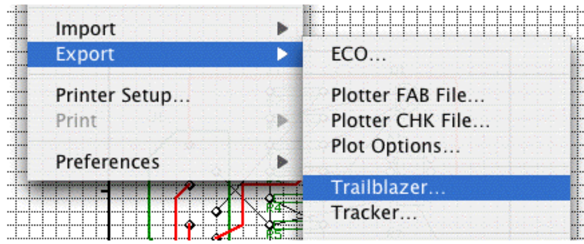


Figure 1-37 • Exporting Trailblazer Document

39. In the file Put Dialogue which appears, label the file as **"PCB_Tutorial.tbs"** . This document will be brought into the autorouting environment in the next section.

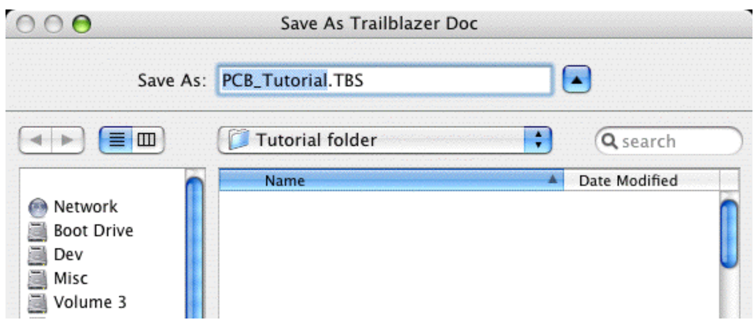


Figure 1-38 • Labeling the Trailblazer File.

Using the Optional Auto-Routing Tools

The McCAD environment offers you a choice of two autorouting tools. They have basically the same interface and only differ in capabilities and level of robustness. Trailblazer is the most capable will be demonstrated next.

40. If you have not already done so, install McCAD Trailblazer on your machine.
41. Launch the McCAD Trailblazer application by double-clicking on the icon.

42. As the application launches you will we presented with a open file dialogue. Locate the file **"PCB_Tutorial.tbs"** which was saved earlier, select it and then open it.

43. The file will open and display your partially routed design.

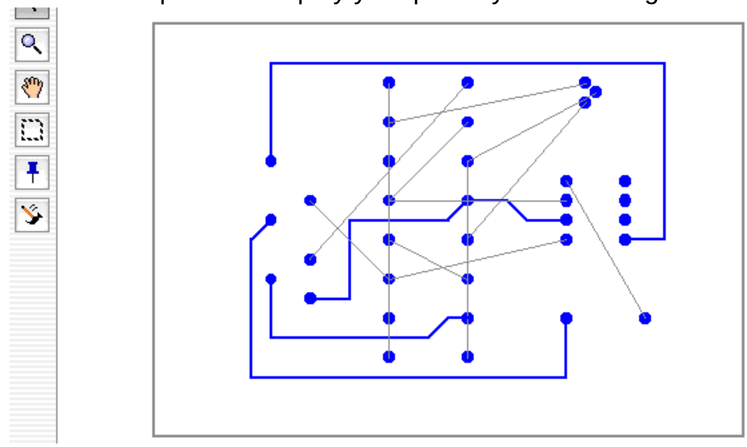
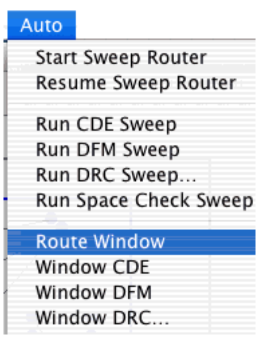


Figure 1-39 • Partially routed design.



At this point since we are dealing with a very simple and straight-forward layout, we will turn the routing over to the Autorouter engine. With more complex and higher density designs you have the option of doing some pre-route optimizations and set-ups which will improve the finished results. You should review the Trailblazer manual at the appropriate time when you are working at that level.

44. In the **AUTO** Menu select the **Route Window** command. The routing will begin and complete quickly. You will be presented with a completion dialogue which you will close by clicking the OK button.

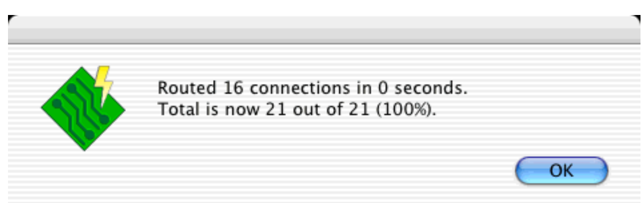


Figure 1-40 • Routing Completion Dialogue

45. Upon screen refresh your finished design should appear.

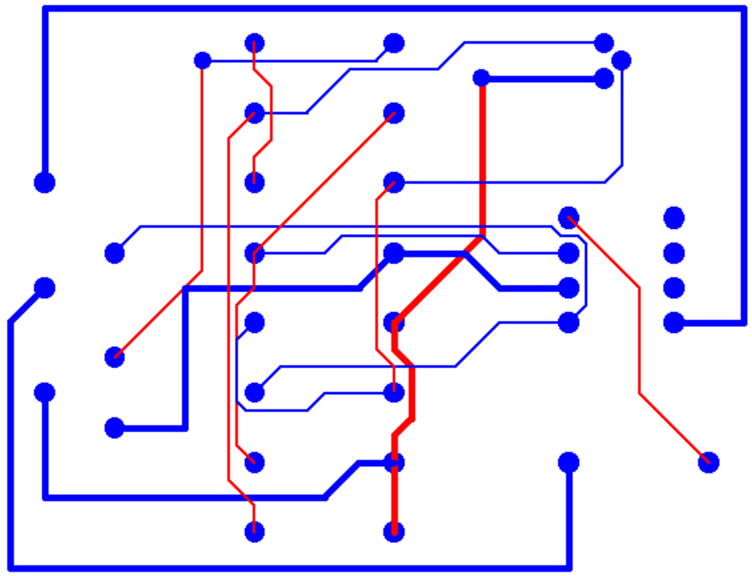


Figure 1-41 • Autorouted Design

For those who have a n enabled version of Trailblazer you would save the file for return to PCB-ST environment. Those using Trailblazer in Demo-Mode we have provided a copy of this saved file so that you can continue with the balance of this tutorial.

Importing the Routed Data File

We are returning to the PCB-ST environment at step 39. on page 32 of this tutorial and resuming the completion of this project.

46. In the File Menu select **Import->Trailblazer**. In the Open dialogue locate the **"PCB_Tutorial_rtd.tbs"** . This is the routed design file which we have provided for those who only have the demo-mode version of Trailblazer. Import this file.

Similar to importing an ECO, bringing in the routed track data from Trailblazer, you are presented with some options.

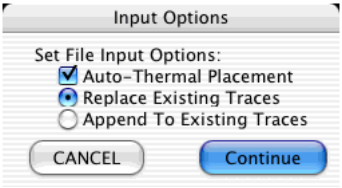


Figure 1-42 • Import Options Dialogue.

We will continue however we did not set up for thermal planes earlier and so thermals will not be automatically created and deposited even though there is a check mark present in the dialogue. We will manually disperse the thermals later.

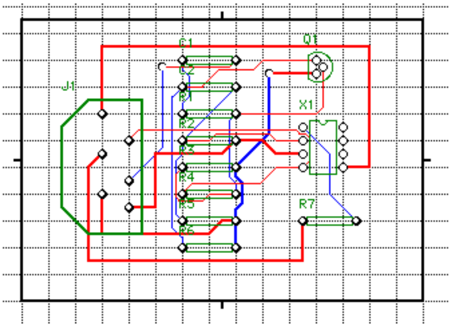


Figure 1-43 • Trailblazer Route Data Imported Back to ST

Power Planes & Thermal Pad Set-up

As mentioned previously we had not setup out project to automatically handle the dispersal of Thermal pads (as needed) prior to importing either the ECO or the return of routed track data. Since we will be demonstrating Split Planes in this tutorial we will now show how to do the setup as well as create the planes.

To use the automatic thermal dispersal feature the PCB-ST system requires us to first specify the data layers which will be associated with those signals which will be connected to an appropriate plane via a thermal type pad. The association is accomplished by assigning the same name to a data layer as the name of the signal. For example, a specific data layer would be renamed to “GND” thus linking it to the signal net “GND”. During the dispersal process the software would then know to place all thermal pads belonging to that net on that data layer at the appropriate locations.

47. In the **ART** Menu select “**Assign Layer Attributes..**”.

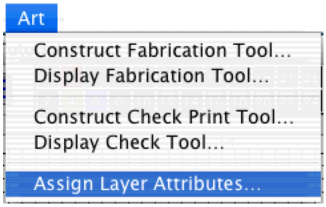


Figure 1-44 • Reassigning a layer name

48. In the dialogue that appears scroll down on the left to Layer 14 and select it. Then click on **Edit Layer Name**.

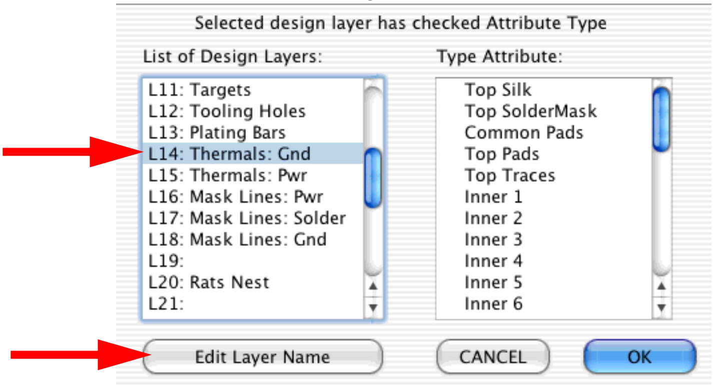


Figure 1-45 • Selecting Data Layer 14 for Name Change

49. In the next dialogue type **GND**. Then click OK

Figure 1-46 • New Layer Label

50. Close the data layer attributes dialogue.

✎ **Note:**

Having set the Name of the Data Layer any time we import an ECO, a netlist or manually disperse a net the system will automatically handle the creating of thermal pads, providing that all the conditions are met.

At this point we will also reset the color for data layer 14. Doing so will help to quickly identify the presence of the thermal and its potential. (One can assign a different color for each potential.)

51. In the **Layout Menu** select the "**Layer Display Settings...**". In the dialogue which appears select layer 14 and in the color palette on the right assign the light cyan color to this layer.

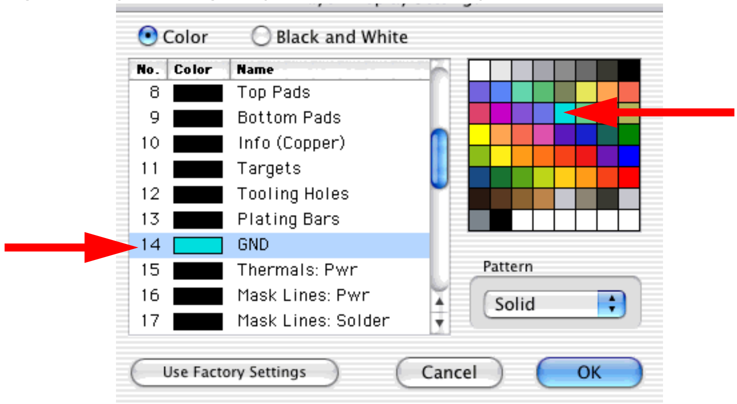
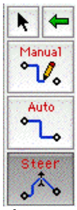


Figure 1-47 • Assigning a Layer Color

52. After assigning the color click OK to close this dialogue.

Manual Thermal Pad Dispersion



Routing Tools

Now that we have completed the setup for thermal pads we will use the automatic feature found in the software to place thermals as needed. In this simple project we have only a single net upon which we will be dispersing thermals. It's labelled **GND**.

53. In the tools palette on the left navigate to the trace routing tool palette. This will cause the Nets Menu to appear in the Menu Bar at the top.
54. In the **Nets Menu** select **"Disperse Thermals on Net...."**. In the dialogue which appears type GND for the name of the net.

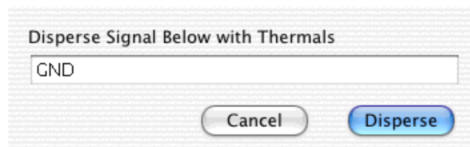


Figure 1-48 • Specifying a Signal for Dispersion

55. Upon clicking on the Disperse button the software will locate the GND signal and then locate a layer bearing the same name. If found (we relabels a layer earlier) it will proceed to place thermals on that data layer.

Preparing for a Split -Plane

We will demonstrate the necessary steps to split a plane even though in this specific project it is not truly necessary since we have only one plane signal; **GND**.

56. In the Data-Layer Button bar click layer 18 to make it the active layer.

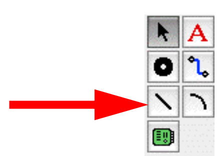


Figure 1-49 • Selecting Simple Graphic Line

57. Select the Simple Graphic Line tool. Set the line thickness to 0.020".
58. Draw a line similar to the one shown.

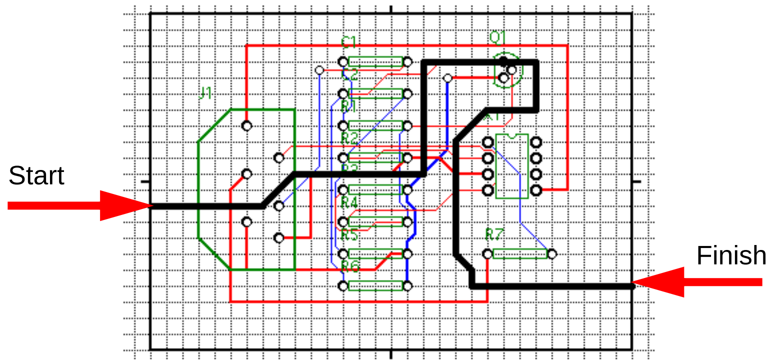


Figure 1-50 • Drawing the Split Line on Layer 18.

Making The Power Planes

The finished Plane is usually a combination of a number of masks. The number of masks needed to produce a single Plane depends upon the complexity of what is needed. The masks are produced using the **Make Special Tooling** command found in the **Edit Menu**. In the following exercise we will make a single plane which is split.

59. Select the **Make Special Tooling** command to bring up the dialogue.

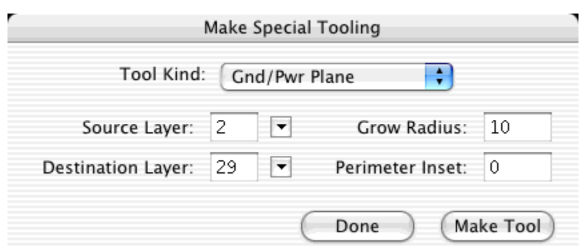


Figure 1-51 • Special Tooling Dialogue

60. Make sure that you have selected **Gnd/Pwr Plane** for the Tool Type. The source layer is **2** and the destination for the Mask is layer **29**. The Grow Radius (or Gap) should be set to 10 mils.

61. Select the **Make Tool** button.
62. The mask generator will begin but first it will ask you to confirm that you want the area around the feed-through vias to be cleared. You should answer **ADD**.

Figure 1-52 • Confirming Via Clearance additions

63. When then mask is finished you will be notified.

Figure 1-53 • Completed Mask Info Dialogue

64. Select **Continue**. This will return you to the **Make Special Tooling** dialogue so that we can continue constructing the plane.
65. Next we need to create a mask containing the clearances for the top side traces. Change the Source layer to 3 which is the top trace layer.
66. Select the **Make Tool** button.
67. The mask generator will begin again, but first it will ask you to confirm that you want the area around the feed-through vias to be cleared. This time you will answer **Don't ADD**. This is because we already have a mask which contains the clearances. Adding any more just wastes memory.
68. Select **Continue**. This will return you to the **Make Special Tooling** dialogue so that we can continue constructing the plane.
69. Next we need to create a mask containing the clearances for the split line we drew earlier on data layer 18. Change the Source layer to 18.
70. Select the **Make Tool** button.

71. The mask generator will begin again, but first it will ask you again to confirm that you want the area around the feed-through vias to be cleared. This time you will answer **Don’t ADD**. This is because we already have a mask which contains the clearances. Adding any more just wastes memory.

At this point we have three separate masks residing on data-layer 29 all on top of one another making it difficult to see the desired result.

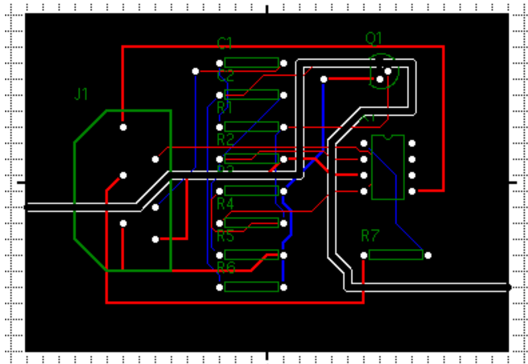


Figure 1-54 • Un-merged Masks

These individual masks will be combine into a single finished composite mask.

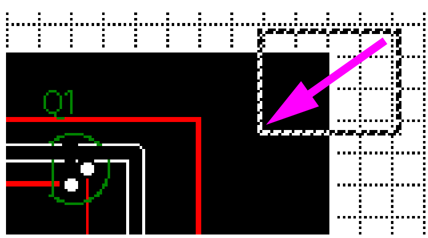


Figure 1-55 • Selecting the masks with a area marquee

72. Make data-layer 29 the active layer.

73. Using the mouse in pointer mode, begin outside the mask perimeter. Depress the mouse button and drag diagonally into the body of the

mask and then release the mouse button. It is not necessary to enclose the entire mask area.

✎Warning;

***Avoid** falling into the trap of using the **Select All** or **Select All Visible** commands. Following the steps described above will avoid accidentally including inappropriate features into a merged mask and possibly damaging your work to this point.*

74. In the **Edit** Menu select the "**Merge Objects...**" command. You will receive a warning dialogue which give you the chance to cancel this operation.

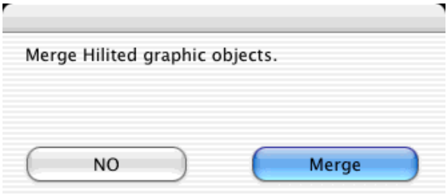


Figure 1-56 • Merge Mask Confirmation

75. Click the **Merge** button.

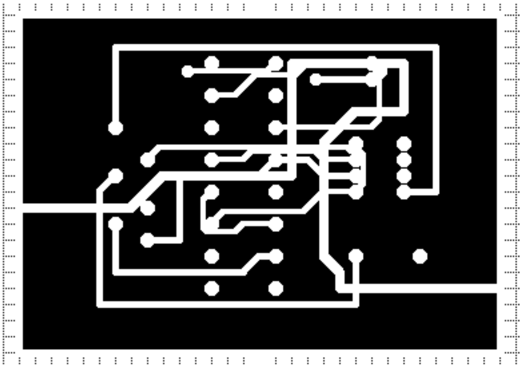


Figure 1-57 • Multiple Masks Merged into single mask

76. To improve clarity, use the technique shown earlier to reassign layer 29's color to a Light Brown.
77. Disable layer 18 so that it will not be drawn to the screen.

78. Make data-layer 2 the active layer and select the redraw command to refresh the screen.

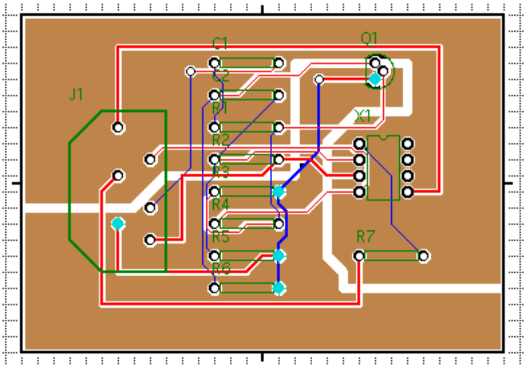
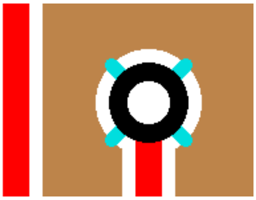


Figure 1-58 • The finished design.



Thermal Pad

The design is nearly complete. You can zoom in and examine the area around the thermals. Please note the spokes which contact the plane. You can change the spoke orientation through a Global preference located in the Design's Preference dialogue.

Making the Drill Template

In preparing the design for manufacture you will need a drill list. This is made from one or more drill templates contained in your design. The number of templates will depend on the overall complexity of your project. Most projects usually only need one single template. In this project the design is very simple and thus we need to create just a single template.

1. In the **Edit Menu** bring up the **Make Special Tooling** dialogue once again.

Figure 1-59 • Special Tooling... Drill Template

2. In thus design all of the drill holes are located on layer 2. The Drill template will be placed on its default layer 26. Select the **Make Tool** button.

Figure 1-60 • Adding Vias to Through-Hole Template

3. Click the **ADD** button to add the vias to the drill template. If vias are found you will be notified.

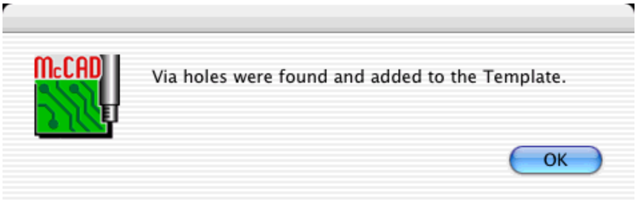


Figure 1-61 • Drill Vias Found

4. You will next be asked if you wish to keep any previously created Drill Templates. In this case since we have not created any previous drill templates either choice will yield the same result.

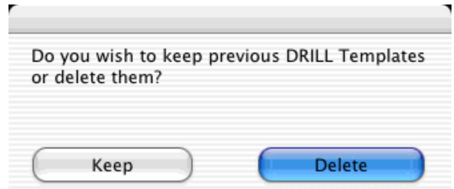


Figure 1-62 • Option to Keep Previous Templates

5. The next dialogue confirms what we have done. Hit the DONE button since we will not need to create any more Drill Templates.

Figure 1-63 • Notification of Template placement

6. Save your PCB design file and quit the application.

You are now done with the design portion of this project. We will next begin the production of the manufacturing documents.

Manufacturing Outputs

Depending upon what you board fabricator requires the PCB-ST system will produce either Gerber and Excellon format document or the integrated design file format known as GenCAD (aka, GenCAM, IPC 2581 subset). The latter can be found in the **FILE** menu under **Export -> Drill & Other.** We will discuss the Gerber documents as they are more commonly used, however the integrated file approach may become more dominant in the coming years.

McCAD-Gerber2

Making Gerber Plot Files

1. Launch the Gerber Translator Utility.
2. If a the file Get Dialogue does not appear go to the **File** Menu and select **OPEN PCB-ST File**. Locate the design file you save in the previous section and open it.
3. Your design file should appear in a window.

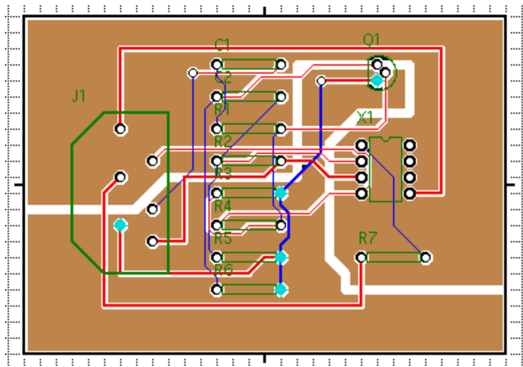
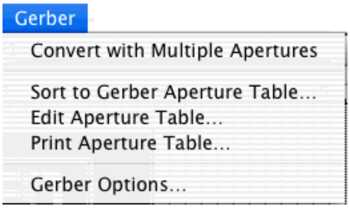


Figure 1-64 • Simple PCB Design Project

4. In the Gerber Menu select the **"Sort To Aperture Table..."** command.



This a very important step as it will insure that an efficient Gerber file will be created. The software will thus scan your design and build a list of required graphic features needed to efficiently reproduce your design using the 274X Gerber standard. The table will be organized based on frequency of feature use.

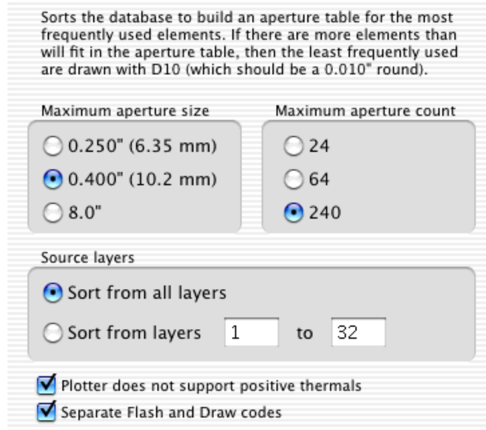


Figure 1-65 • Gerber Sorting Dialogue

5. Prior to the completion of the Scan and Sort operation you will presented with options dialogue which controls the results of the sorting process. The default settings are what you will be using most of the time. Click **Proceed**.
6. You can view the results of the sorting by selecting “Edit Aperture”
7. In the **Gerber** Menu select the **“Convert with Multiple Apertures..”** to begin the Gerber file creation process.
8. You will be asked to confirm whether the Aperture Table (a.k.a., Aperture Wheel) is appropriate for this project. Since we just did a sort we answer YES to continue. You would answer NO if you needed to load a specific predefined table or you had forgotten to do the sort.
9. The next dialogue allows you to position the Gerber Image. You would do this if you were trying to combine a number of Gerber images into

a single Production Gerber document. In most cases you will just leave the offset at 0, 0 an click OK to continue.

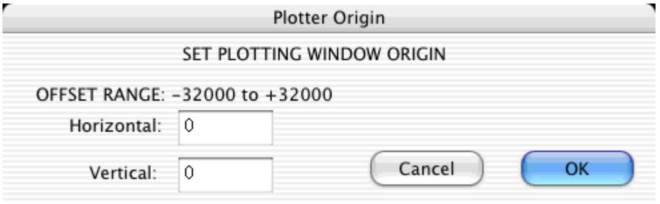


Figure 1-66 • Gerber Output Image Offset Option

10. The next dialogue allows you to select the Gerber Fabrication Negatives (Tools) you wish to create.

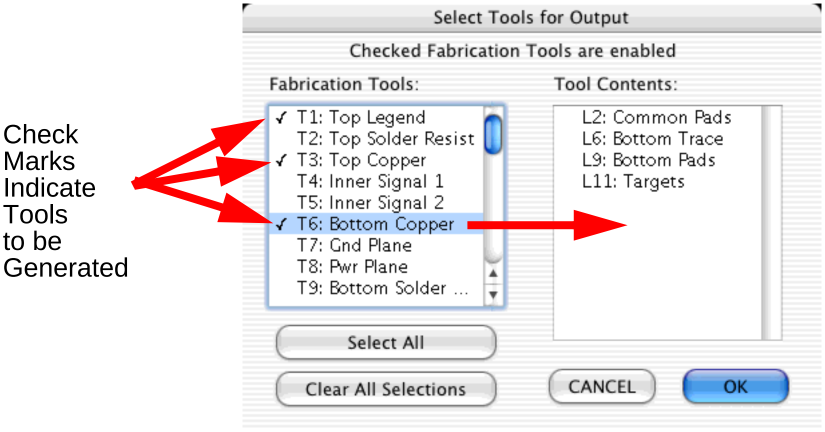
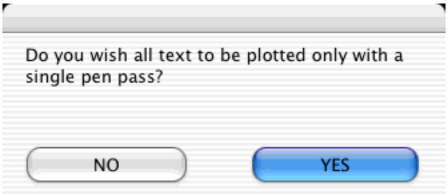


Figure 1-67 • Selecting the Gerber Tools

11. When you select each tool on the left the data-layer contents that make up that selected tool are displayed on the right. This allows you to check before committing to the Gerber file creation. Once you have selected the tools you will need and a check mark appears next to those desired tools click OK.

12. Before producing the output you have one more option to select.



Standard Gerber does not support a character set. Therefore all text strings must be drawn by the Gerber generator. If in your design you marked any text as BOLD the generator will do a second pass to fatten up the text marked as BOLD. Doing this is not recommended for small text. Therefore this dialogue allows you to limit text drawing to a single pass regardless of whether it is marked BOLD or not.

13. Select the YES button for this demonstration. The Gerber files will now be created. and placed where you have directed the system.

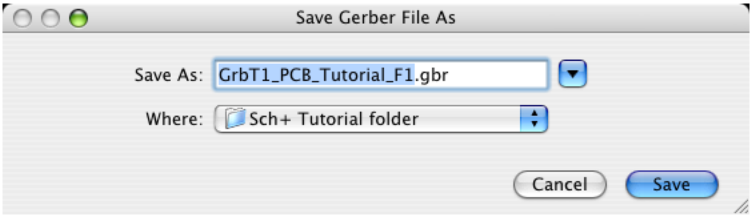


Figure 1-68 • OS System Appropriate PUT dialogue

Making the Excellon Drill files.

The production of the Excellon Drill files can be done either in the PCB-ST design module or, as we will now demonstrate, in the Gerber translation utility.

14. In the **Utility** Menu select **“Export Drill & Other...”**.

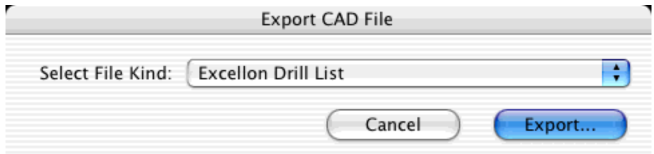
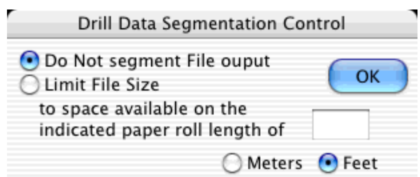


Figure 1-69 • Selecting the Excellon Drill List

15. Select the Excellon Drill List in the Pop-Up Menu and then click **EXPORT..**
16. In the next dialogue specify how you want the file labeled and where it is to be put.



17. Next a file segmentation dialogue will appear. Simply click OK. Segmentation may only have to be done on very large designs which have upwards of ten thousand holes. Your fabricator will tell you when he needs this to be done.

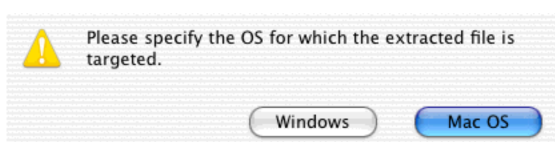


Figure 1-70 • Selecting OS Target Format

18. It is best to select Windows in most cases since most Fabricators have Windows OSes and some of their drill handling software my cause them problems if you send it to them in MAC OS format.
19. Finally there are two Excellon Formats. Select # 2 in most cases since it is a newer format.

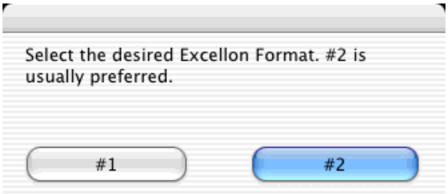


Figure 1-71 • Selecting Fabricator's Excellon Format.

20. Take the Gerber and Excellon document which are created and place them into a compressed archive so that you can send them to your board fabricator.
21. Close all McCAD Applications.

This completes the tutorial.