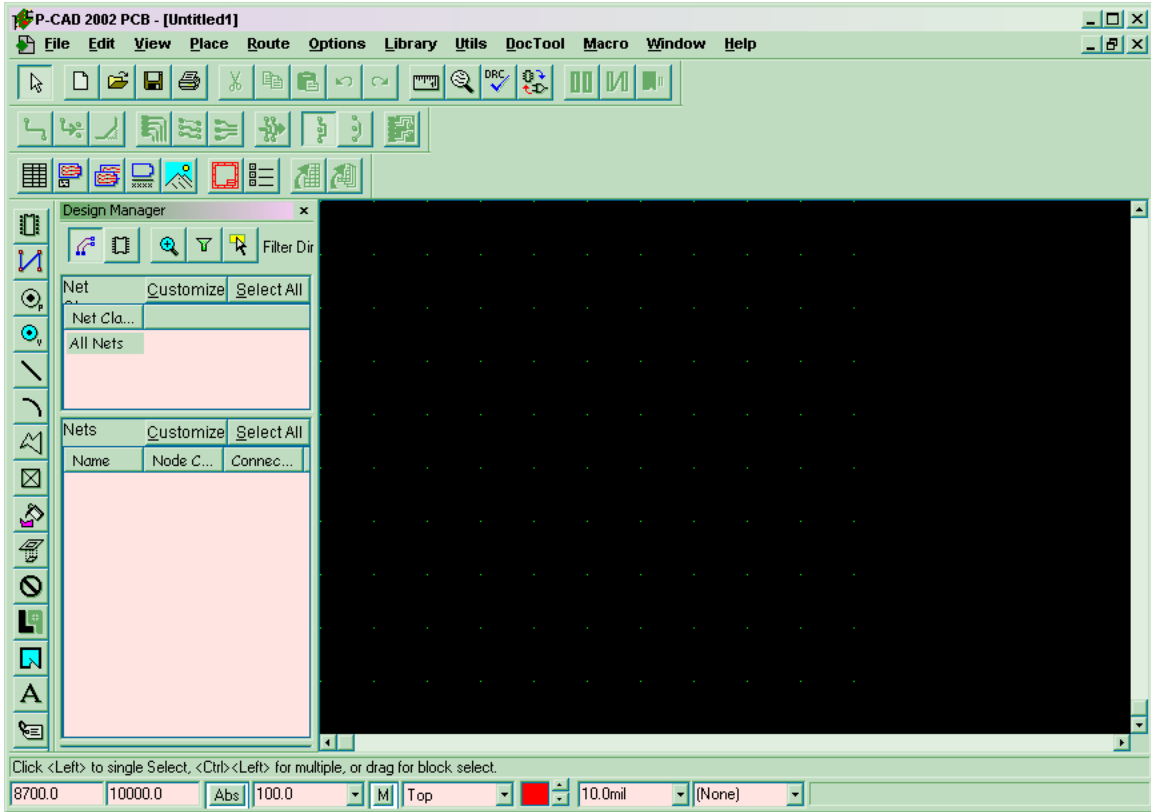


## Getting started in the PCB Editor



by  
Lori Zukerman  
of the EE/CAD group

1. Initial Setup .....	3
1.1 Copying Drawing Formats .....	3
2. Starting your PCB .....	3
2.1 Open a setup drawing .....	3
2.2 Environment setup .....	4
3. Moving around in the editor window.....	5
3.1 Zooming .....	5
3.2 Panning or moving .....	6
4. Starting your PCB .....	6
4.1 Design Info .....	6
5. Laying out your PCB .....	7
5.1 Placing your Board outline .....	7
Existing outlines .....	7
Creating an outline from scratch.....	7
5.2 Reading in the Netlist file.....	9
Library setup.....	9
Workspace size.....	9
The netlist.....	9
5.3 Component placement .....	10
5.4 Setting up DRC.....	11
5.5 Routing the board .....	13
Four Routing Tools.....	13
5.6 Autorouting .....	13
6. Post processing (or checking your work).....	15
6.1 Design Rules Check .....	15
6.2 Board Identifiers.....	16
6.3 Internal Plane borders.....	16
6.4 Gerber file labels .....	16
6.5 Fabrication drawing .....	16
6.6 Assembly drawing .....	17
7. Generating Gerber and NC Drill files.....	17
7.1 NC Drill file.....	17
7.2 Generating Gerber files .....	17
7.3 The Read Me file .....	18
8. Printing.....	19
8.1 Setting up and Plotting a mirrored bottom assembly view .....	19
9. PCB layout start to finish.....	20
10. Tips, Tricks & Hot Keys.....	20
11. General Rules & Guidelines .....	21

## 1. Initial Setup

### 1.1 Copying Drawing Formats

Create a folder for a copy of the setup files on your machine in a location such as \program files\PCAD 2002\pcb formats. Now, let's copy the provided format setup files to your machine for your use. Double click on "group on jlabgrp" (typically your M drive). Find the folder EECAD and double click, find PCAD and double click, then PCB Formats. Copy all three folders to your machine. Most likely you will primarily use the full size formats for your work. The half size formats were created to allow us to plot our PCB documentation at twice scale but that is rarely done these days. The board out lines folder contains commonly used PCB board outlines complete with targets, layer nomenclature and dimensioning.

## 2. Starting your PCB

### 2.1 Open a setup drawing

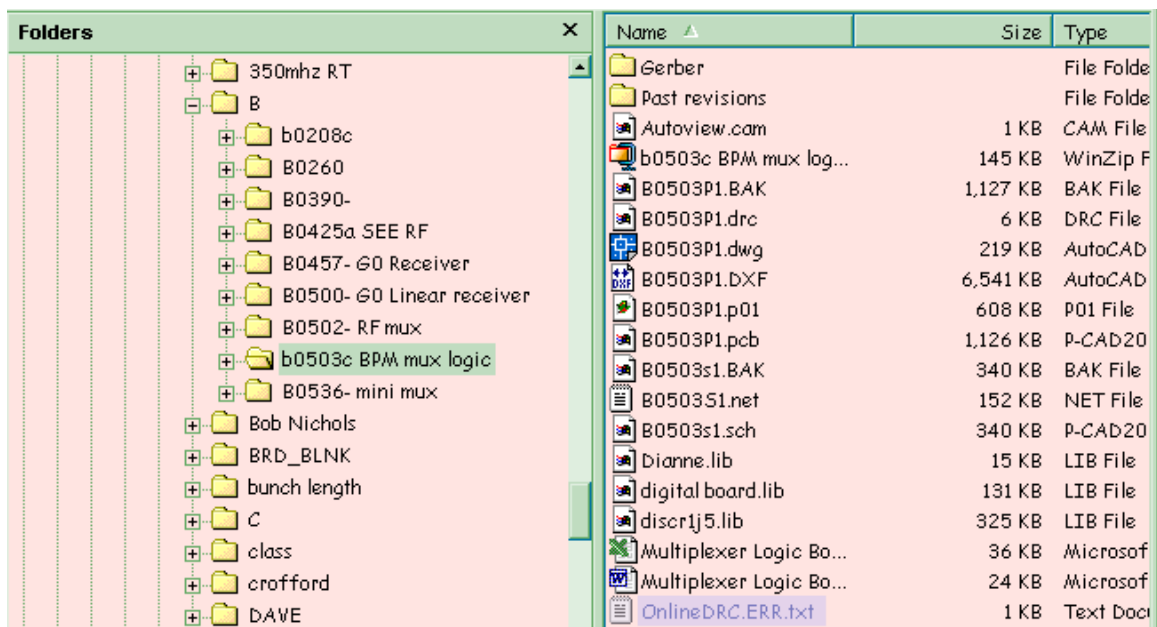
Okay, let's start a PCB. Open the PCAD PCB editor. The first thing to do is open a setup file. Select File OPEN and find your drawing setups directory. Select the size you think will be appropriate to contain your PCB. The setups correspond to the following sizes:

B\_?lyr\_g is 11" x 17"  
 C\_?lyr\_g is 17" x 22"  
 D\_?lyr\_g is 22" x 34"  
 E\_?lyr\_g is 34" x 44"

We recommend that you use B (preferred) or C size. B size sheets can be printed on almost any printer here and still be very legible. 3U cards will fit on a B, while 6U will fit on a C. For backplanes, you may have to use a D size.

The names above contain a ? to represent the number of layers. This is why we have multiple setups for each size. Each format has been structured to have print & gerber setups for the corresponding number of layers. It is in your best interest to start your PCB with one of these setups. They will save you a LOT of work later on.

Click File-Save AS and save this file as a different name into a different directory. We suggest that you make a separate folder for each project you work on and that you name the file with a descriptive name. Please don't call it PCB. A more appropriate name would include system and function, like BPM mux.pcb. A project directory might look something like this:



We suggest that you store your work in an area that gets backed up regularly. The EECAD group has a common area for file storage and would be glad to provide you with your own directory to store your projects in. Contact Lori @ 7332 or [beckett@jlab.org](mailto:beckett@jlab.org) if you'd like some space in this area.

## 2.2 Environment setup

Now let's make sure everything is as it should be.

Open the **View** menu pull down.

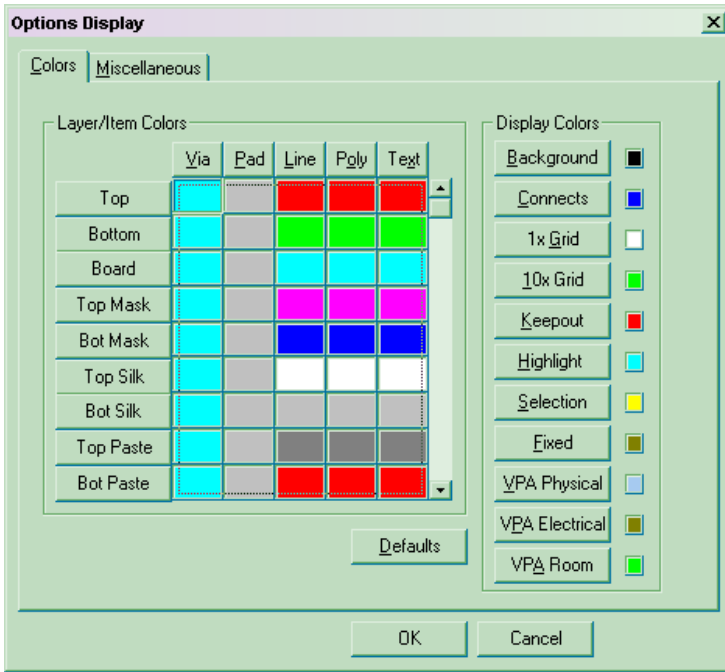


Verify that **Snap to Grid** has a check mark next to it. If not, click on it to make it active.

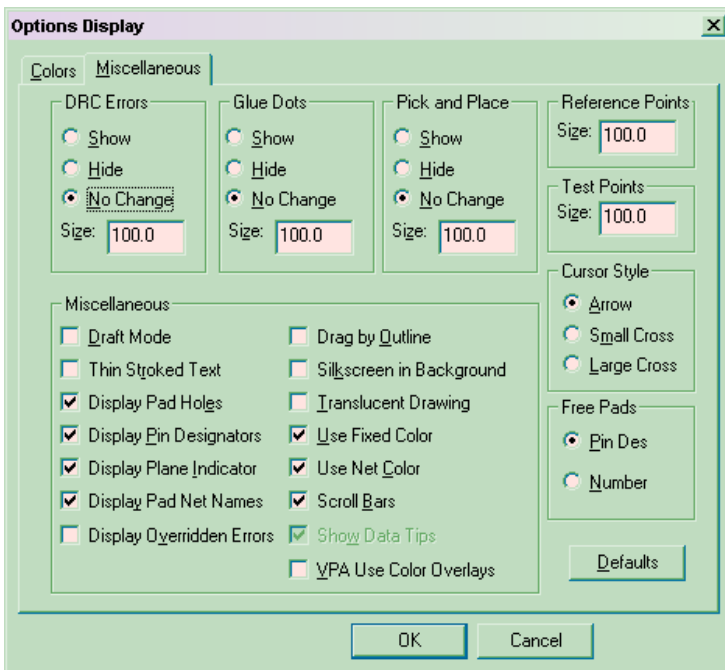
If you only remember a few things from this document, this is probably the MOST important thing to remember. If you don't snap to grid, it will be very difficult to connect your components and to make changes later.

Verify that your Grid is set to 50 mils. Look at the very bottom of your editor window. The third box from the left that has numbers in it is your grid size. If this does not say 50, click the arrow to the right of the number and select 50 from the pull down. Your grid may need to be set lower if you are placing SMDs. A typical grid in this case might be 5 mils.

Next, click on **Options-Display**



Here you can control & change the colors your PCB is displayed in. Some people, like myself, might not like red & green for Top & Bottom. Using this you can change your color options for just about anything.



The Miscellaneous tab allows further control of how & what's displayed. If working with SMDs, you might choose to Hide Glue Dots and Pick and Place indicators. You can also change your cursor style here or through the keyboard by using the X key.


### **3. Moving around in the editor window**

#### **3.1 Zooming**

One of the easiest methods of zooming in and out of your drawing is with the PLUS and MINUS keys on your keyboard.

You can also zoom in or out by going to the **View** menu and selecting either **Zoom in** or **Zoom Out**, then clicking on the area you want to zoom from or to.

You also have the option to do a **Zoom Window** by either selecting it from the **View** menu or clicking on

the  button. Then Left click and drag a window around the area you want to zoom in to.

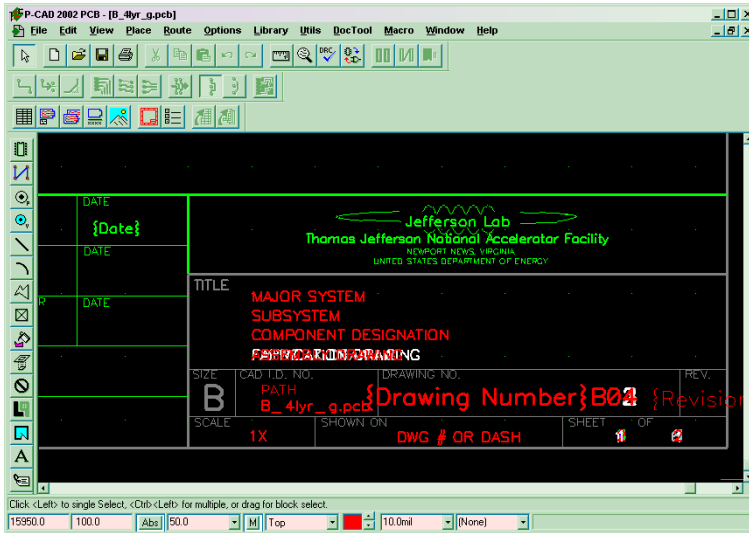
### 3.2 Panning or moving

You can use the scroll bars on the right or bottom to pan across your drawing. Or you can move your cursor to an area and hit the “C” key. This re-centers the view with your cursor location at the center of the window.

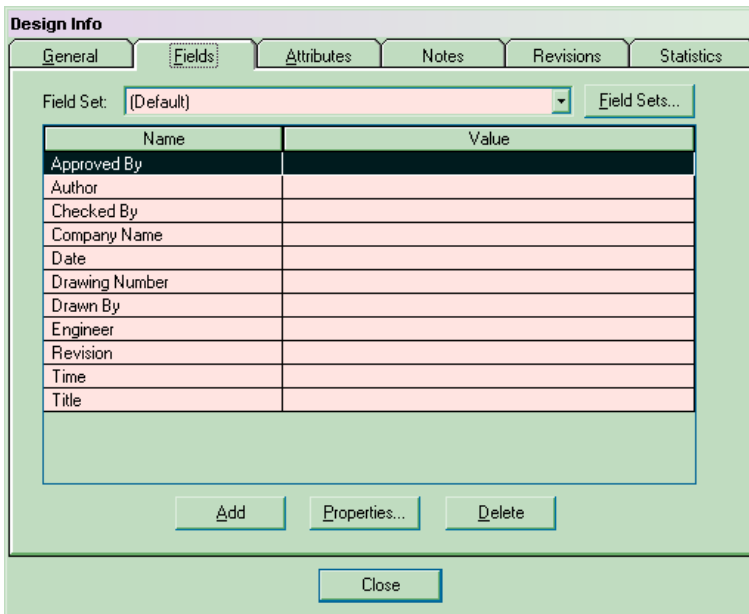
## 4. Starting your PCB

### 4.1 Design Info

Using the PLUS key, zoom in to your title block area.



Here you can see the numerous *FIELDS* that have been placed in the title block. They look like this {Title}. These Fields can only be edited through the **FILE- Design Info** selection which we will select now.



Double click on the value box for the Field you wish to modify to open the dialog box. The following Fields will need to be modified:

- Date
- Drawing Number
- Drawn by
- Engineer
- Revision

Additionally, you need to edit the text that says Major System, Sub System, Component Designation and the path strings that indicate where this drawing is stored. The EECAD group will be glad to help you with naming your drawing if you're unsure how to do this. We are also the ones that you would call for drawing numbers (or you can wait if you plan to submit your design to us).

## **5. Laying out your PCB**

A good PCB layout has...

- components oriented in only two of the four possible directions (exceptions can be made)
- components placed at 0 or 90 degrees (no components shall be placed at 45 degrees!)
- uniform and generous component spacing, if possible
  - (remember they have to be built & worked on)
- all the components lying flat (no antennas!)
- filter caps placed next to the power pins on their associated ICs.
- pull up/down resistors located adjacent to their associated component
- a minimum trace width spacing between parallel traces equal to or greater than the width of the traces
- internal power/ground planes or decent width on the power & ground traces
  - (no 10 or 15 mil traces here! 25-30 mils or greater if possible)
- the voltages run to the filter caps, then to the IC.
- in the event of internal planes, the power strap on filter caps then a trace run to the IC
- planes backed off from the edge of the board (50 mils minimum)
- pin one indicators for ICs, connectors and sips
- same style pads for like components
  - (all ICs should have the same shape pads, i.e. round & square or oblong & rectangular)
- the name, drawing number and revision level silk screened onto the board
- no fonts below 60 mils on the silkscreen (font should be relative to board density)

### **5.1 Placing your Board outline**

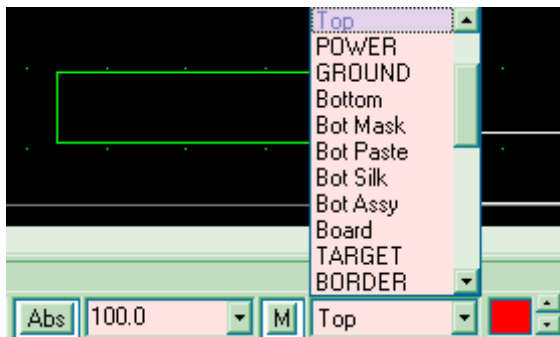
Before you can read in your netlist information to start your board, you need to place a board outline in your workspace. We have a number of common ones available, which you should've copied to your computer in an earlier step. We will talk about copying one of these existing outlines into a format and creating an outline from scratch.

#### **Existing outlines**

Since you opened a setup & saved it in an earlier example, we'll assume you still have that open. Now open one of the board outlines in the board outlines folder. Make sure your grid is set to either 50 or 100 mils. Click on the **Edit** pull down menu and click on **Select All**. Right Click and click on **Copy**. Click on the **Window** menu pull down and switch to your saved format. Make sure the grid here is set to either 50 or 100 mils. Click on the **Paste** icon and click and drag within your window area to place the copied outline.

#### **Creating an outline from scratch**

Maybe one of the existing outlines doesn't suit your needs because you're making a board to fit into a special enclosure. Here is what you do to create your special board outline.



From the layer pull down at the bottom of your design window, select the Board layer to make it current.

Verify that your grid size is set to 100 mils. (Unless you have an odd, tight area that you are fitting this board into, try to keep the size of your board defined to one decimal place. i.e. 5.5 instead of 5.475)

Make sure your line width is set to 10 mils.

Select **Place- Line** and draw the outline of your PCB. If you look to the lower right of your editor window, you will notice a line of changing text that will read something like this:

Ortho=Any dX= 500.0, dY=200.0

This shows that your orthogonal snap is set to any, you have traveled 500 mils in the X (horizontal) and 200 in the Y (Vertical). This can be very helpful in drawing lines of certain lengths. *One important item to keep in mind, your board outline must be closed.*

Once you have your outline drawn, you need to think about how the board will be mounted in the enclosure. Most likely, you will need to add mounting holes to your board so the board can be mounted in some fashion, for example on stand offs. If it is a small board, typically you'd want to add three to four holes for mounting. For larger boards, you would want to add additional mounting to support the board and prevent sagging, warping or flexing. Go to **Options- Pad style** to select a mounting hole. Your selection will depend on what size hardware you choose to mount your board with and whether or not you want your mounting holes grounded. For no grounding, select a pad style defined as the same size as the mounting hole such as P:MX110Y110D110A. If you intend to ground, select a pad style that is bigger than the hole size, such as P:MX150Y150D110A. Double click on the pad style you choose to make it current. Then from the left hand toolbar, click on Place- Pad and place pads for your mounting holes.

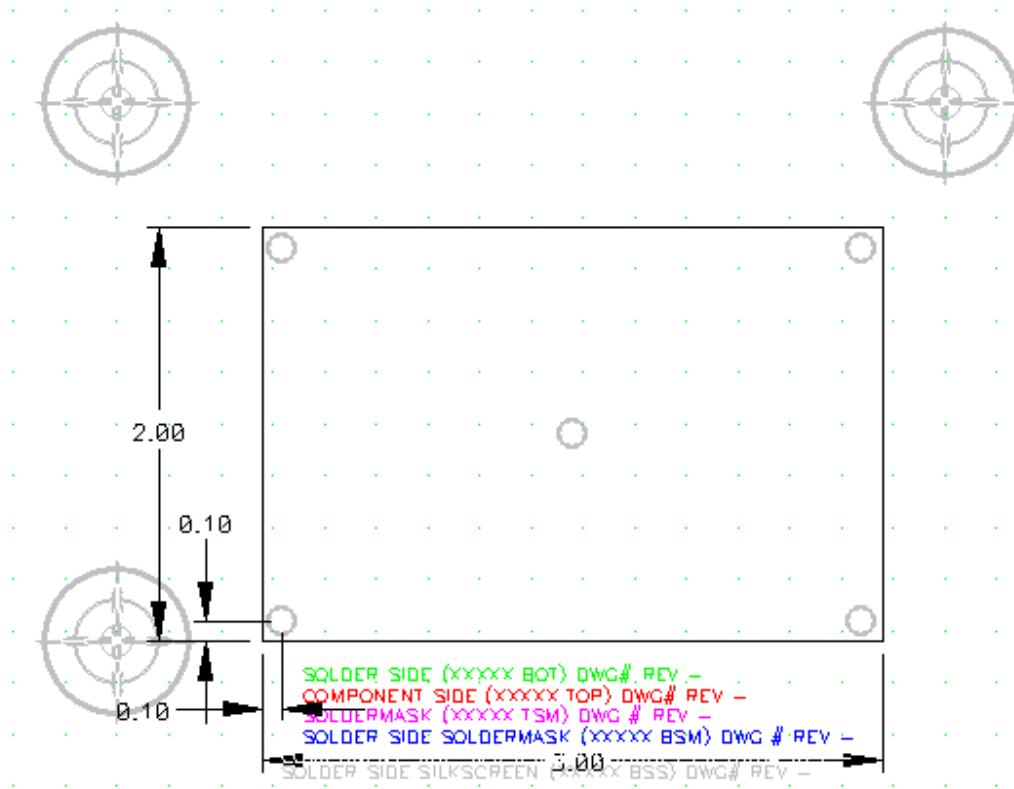
Add your targets by opening one of the existing board outlines and copying a target from it into your current drawing. Do likewise for the text located under the board outline. (These will be used later to label your Gerber layers.)

Now, you need to dimension your outline. Click on the Dimension button. 

Click on the **Layer** selector and change it to Fab. Click on **Precision** and change that to 2. Change your **Text Style** to something appropriate to the size of the format. For a B size, this should be T:H120Y10. Since our **Orientation** is set to Horizontal, this is where we'll start. Click **OK**, then select the left side of your board outline. Now, click on the mounting hole you placed earlier that is the closest to the lower left hand corner of the board, Click & drag to place your dimension and locate the text. (Play with locations as you drag to get a feel for how it looks and behaves.) Next place a dimension for the overall board width by clicking on the left outline and the right outline. Right click, then left click to bring up the **Place- Dimension** menu again. Change the **Orientation** to Vertical and click **OK**. Now, place the vertical dimension for the mounting hole and for the board height.

When you're done, you should have something that looks like this:





You should always dimension height and width plus the location of one hole, typically located close to the lower left corner of the board.

Now that we have our board outline defined, it's time to read in our netlist.

### 5.2 Reading in the Netlist file

If you worked through the schematic handout, you'll recall that towards the end of it we generated a netlist from our schematic. The netlist contains all the point to point connection information along with the component information. We will now read that schematic into our PCB drawing.

#### Library setup

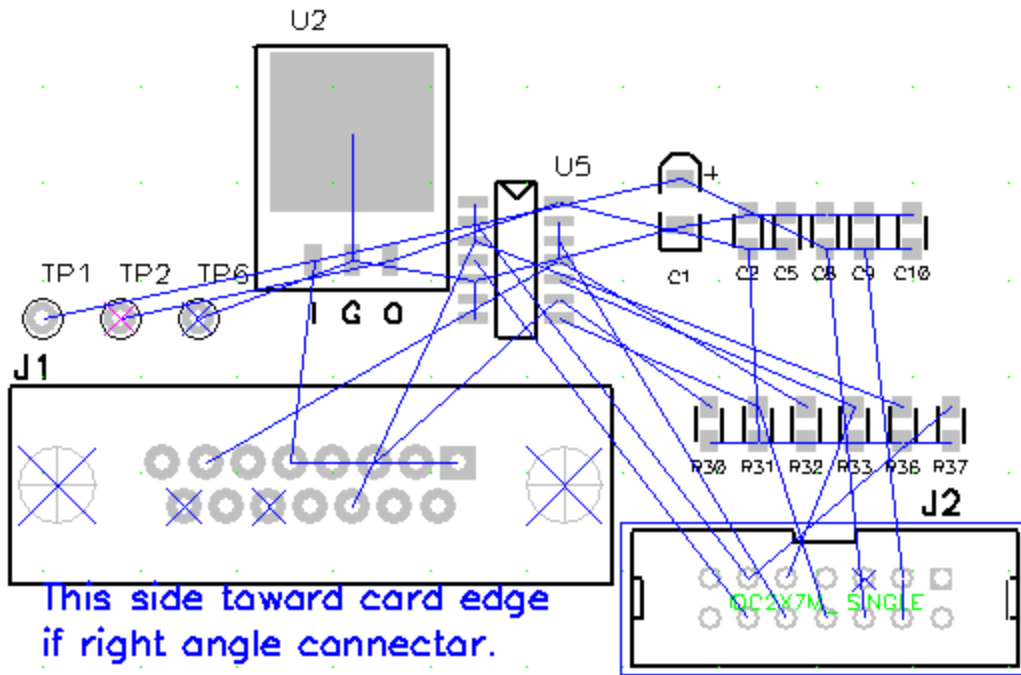
The first thing you need to do is open the library file or files that correspond to your PCB. Hopefully you have followed the steps detailed in "Getting Started in the Schematic Editor" for archiving a library for your project. Select **Library-Setup-Add** and find the library you created earlier for the board you are getting ready to start.

#### Workspace size

Your overall workspace size will need to be increased briefly. Please remember to change it back once you get all your components relocated within your PCB outline. Click on **Options- Configure** and double the size of the height of your workspace.

#### The netlist

To read in your netlist, click on **Utils- Load Netlist**. A dialog box will appear. Verify that the netlist format is the same as what you generated in and that the name of the netlist file is correct, then click **OK**. If you are working with formats that were built for PCBs with greater than 2 layers, you will be prompted during the load to assign nets to certain internal planes. This would be were you tell the system which voltages are placed internal. If there are no problems, your components will be placed at the top of your format with "ratsnets" (rubberband or connection) lines indicating the net connections. It will look similar to this:



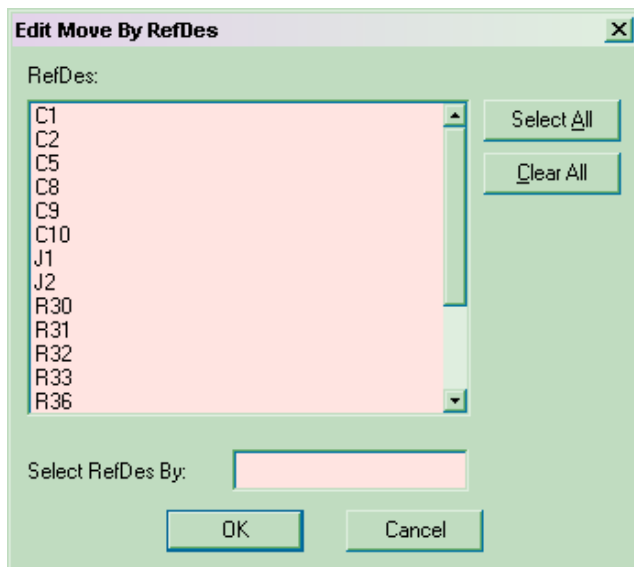
**5.3 Component placement**

Usually at this point I try to move all my components so that they are within the drawing borders but still outside the PCB outline. Then I set my workspace size back to normal.

Okay, now comes the time to place your components within your PCB. This can be done a number of ways.

The first way is the old-fashioned and perhaps the hardest way. Sitting down with the schematic in front of you and hunting for your parts, clicking on them and dragging them to where you want to place them. This can be rather time consuming, so we recommend one of the following three ways.

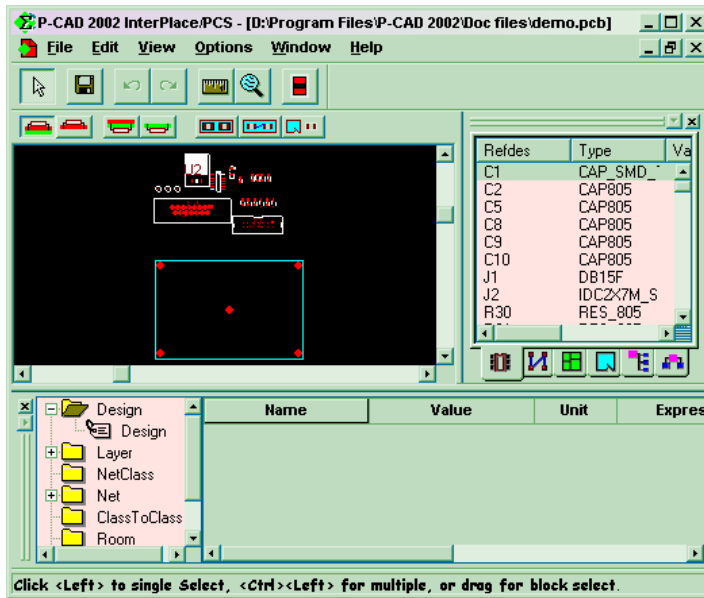
The second way, is to open your schematic in the schematic editor and select the portion of circuit you wish to place by doing a selection window. Right click and select **Highlight**. Now, return to the PCB and those components will be highlighted and you can select & place them where you wish. Return to the schematic and **Unhighlight** the components you just placed. Then select and highlight the next set of components you wish to place. Continue this process until all components are placed.



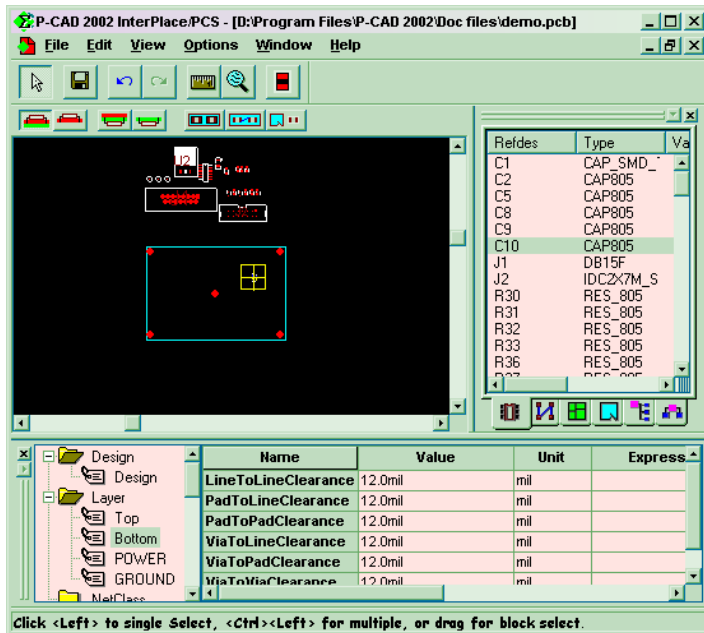
The third way, is through **Edit- Move by RefDes**

Highlight the component you wish to move and click OK. Or click Select All and it will allow you to place each component in order without having to restart this command each time

The fourth way is through an interactive placement tool called InterPlace. It can be accessed through **Utils-P-CAD InterPlace/PCS**, which will open a new window: Click on **File- Load PCB Design** and notice how the windows all change.



As you can see, it has loaded a simple version of your PCB outline & components into one window. The window on the right is a listing of all the components by Reference designator and also includes type and value information (which helps in finding all those filter caps). You can click and drag from the listing into the window with the PCB outline to place your components. You can access additional information through this right hand window. If you click the second tab there on the bottom, it brings up a listing of all the nets. With this you can turn them on, off or highlight them. Having your signal nets on helps greatly with placement.



You'll notice that we have additional information in the bottom two windows. This allows you to view things like DRC by layer (as shown here), net constraints, net classes and additional component information.

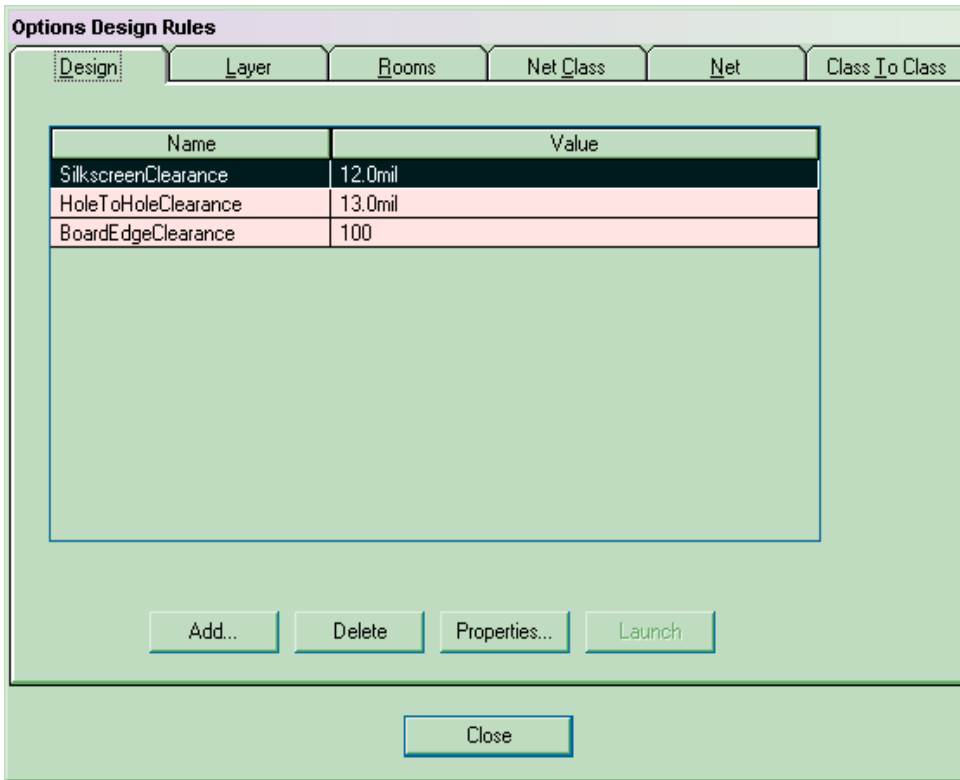
You can also combine the second method with this method if you find that helpful.

Once you finish placing your components, you will have to update your PCB and close this window. Click on **File- Update PCB**, then **File- Close**.

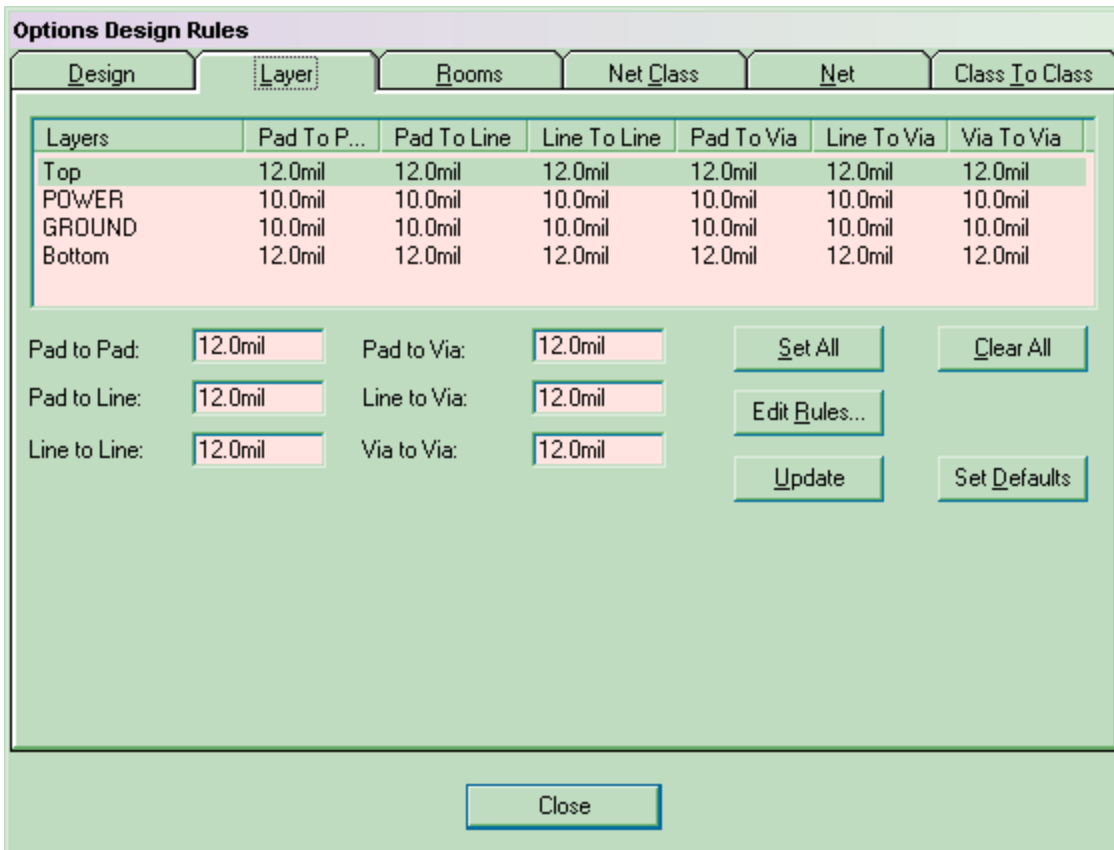
Remember that when placing components, you can use your "R" key to rotate and the "F" key to flip. Please keep in mind that if you flip a component in the PCB editor that it physically moves from the top side to the bottom side of the board.

#### 5.4 Setting up DRC

Before you route your board, you need to set up your design rules. Select **Options- Design Rules** to open the dialog window. There are two tabs that should be reviewed by all users. The first is the **Design** tab.



Here is where you would define your rules for Silkscreen clearance, general hole to hole clearance and board edge clearance. If your board looks to be a tight board, you will want to set your Silkscreen clearance much, much lower to prevent DRC from flagging every piece of silkscreening.



The **Layer** Tab lets you define your design rules per layer. For example, maybe you have SMD RF components on the top and wish the pad to pad and pad to via clearance to be greater than on the bottom. You would change those values and click **Update**.

Click **Close**, then click the DRC icon to make DRC active.

### **5.5 Routing the board**

As with placement, there are numerous ways to route your board. The first is to route it manually. Some boards have to be routed manually, controlled impedance boards for example. This method is more time consuming, but for a seasoned designer it will result in a cleaner, better route. A seasoned designer can, in most cases, manually route a board in less time than it would take to Auto-route and clean up. (You should **always** clean up an autorouted board.) You can use Highlight Net and control the display of the nets to help you with your routing. You may also choose to manually route certain nets before turning one of the Auto-routers loose on your board.

While routing, hitting the **L** key will cycle through your layers. The **G** key will cycle through your grids. Pressing the **O** key cycles through orthogonal modes (allowing you to snap your traces @ 45 or 90 degrees) **Backspace** will undo your last mouse click when routing traces.

Interactive Routing is similar to manual with the exception that you don't have to pick every corner of every segment like you do with manual route. In Interactive, you select your net and it will automatically place bends and corners in as you stretch your trace closer to its destination.

### **Four Routing Tools**

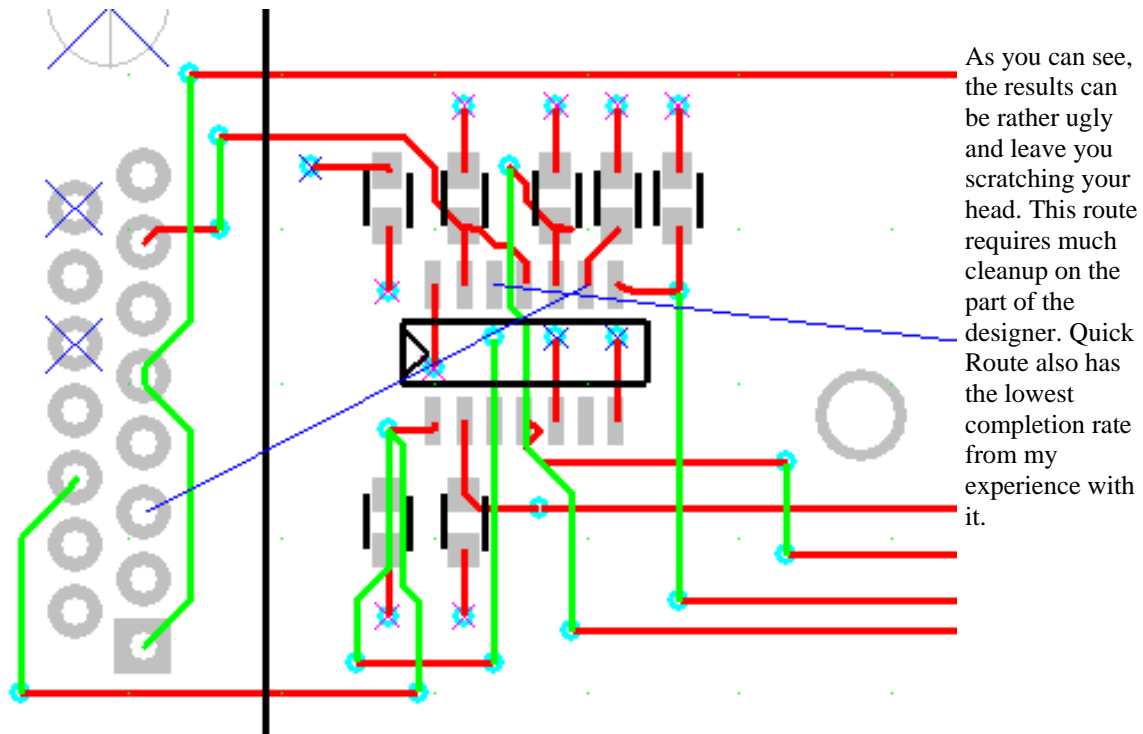
There are four routing tools we will quickly discuss before moving on to talk about the auto routers.

1. The **Route- Miter** tool. converts 90-degree corners on routed connections into 45-degree mitered corners or 90-degree arcs. The 90-degree corner must be true horizontal and vertical segments and have the same line width.
2. The **Route- Bus** tool allows you to specify several connections to be guided simultaneously as a bus.
3. The **Route- MultiTrace** tool is designed to automatically route several connections. In contrast to Bus Route, MultiTrace routes each connection individually allowing each trace to find its optimum path.
4. The **Route- Fanout** tool is used for the systematic placement of traces from a PCB component. These traces can then be used by Route Bus as a start or termination point. This can be extremely helpful on surface mount devices.

### **5.6 Autorouting**

When you choose the **Route- Autorouters** command, the Route Autorouters dialog appears. The Route Autorouters dialog lets you choose the autorouter you want to use to route your designs and to set autorouter options for the selected autorouter. You can also start and restart the autorouting process from this dialog. Each router has different parameters that can be manipulated. As with anything, time & experience will help you to configure these routers for top performance.

**Quick Route**- this is the default router. It is a very basic, down & dirty router that will get your connections from point A to point B with straight vertical connections and plenty of vias. If you have a simple board to route, we would recommend doing it manually/interactively as opposed to using this router.



As you can see, the results can be rather ugly and leave you scratching your head. This route requires much cleanup on the part of the designer. Quick Route also has the lowest completion rate from my experience with it.

**P-CAD Shape Route**- is an easy to use, powerful, high quality, shape based autorouter. Based on the latest in routing algorithms, P-CAD's Shape-Based Router uses polygonal shapes to provide virtual gridless autorouting (similar to Specctra) and the ability to specify different routing angles for each layer of your PCB design.

Optimized for today's dense SMT-based designs, P-CAD's autorouter easily handles mixed metric and imperial pin spacings and double-sided component placements. Since it is a more complex router, there are many more variables that can be changed prior to routing. This router will route cleaner than Quick Route with fewer vias. The major drawback to this router comes when doing an ECO. It can be more time consuming to make changes to a board routed "gridless" than if you routed the board on a 25 mil grid. If you want to use an autorouter, this is the one we recommend and we will touch on a few important things to do to get optimum results. If the board includes routing that you do not want the router to rip up, make sure the traces involved are fixed, using the **Edit- Fix** command in the PCB Editor. These manual traces can include fully routed nets, nets with some of the connections routed, and nets with partially routed connections, such as fan outs and breakout routing.

When you start this router, it will open a new window. Select **Options- Autorouter** to make parameter changes. It is important to note that the Router analyzes your board database during loading and automatically set up the routing parameters and the routing passes. We recommend using the selected parameters and routing passes unless you have a specific reason to change them.

#### Prior to Routing-

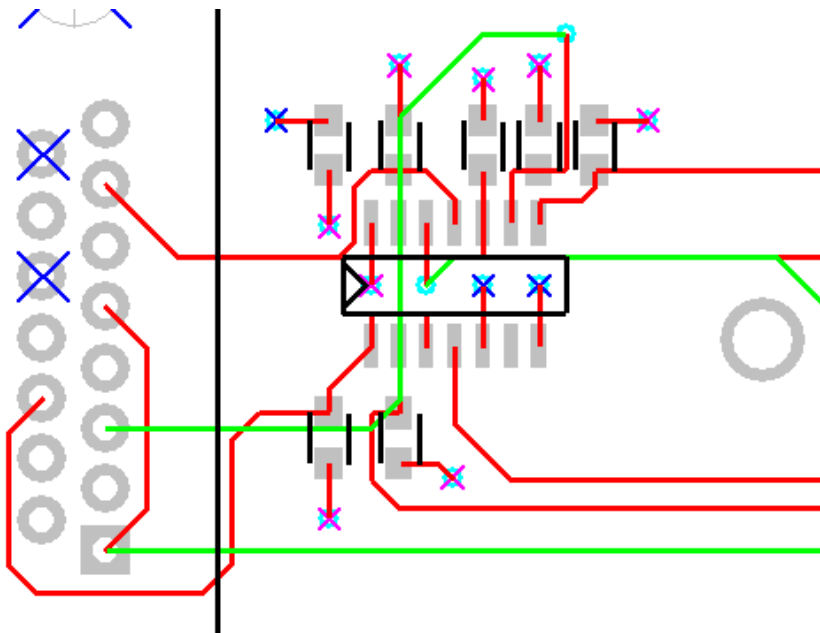
Perform a **Pre-Route Synopsis** prior to starting the autorouter.

Use the **Density Map** feature to check for areas of the board that may prove difficult to route.

Both features are found under the **Reports** pull down.

To start autorouting select **Tools Start Autorouter**. The cursor will change to an hourglass, indicating that routing is in progress. The Status Line at the bottom of the display provides real time status on the routing progress.

The first stage of the autorouting process is analyzing the board and selecting a suitable routing strategy. The routing strategy is selected to suit the device technology used on the board and the routing parameters that have been defined. The strategy will automatically select and schedule the various routing passes, the active routing pass is shown on the status line. If there are any problems detected during the initial analysis the router will stop and a message will be displayed.



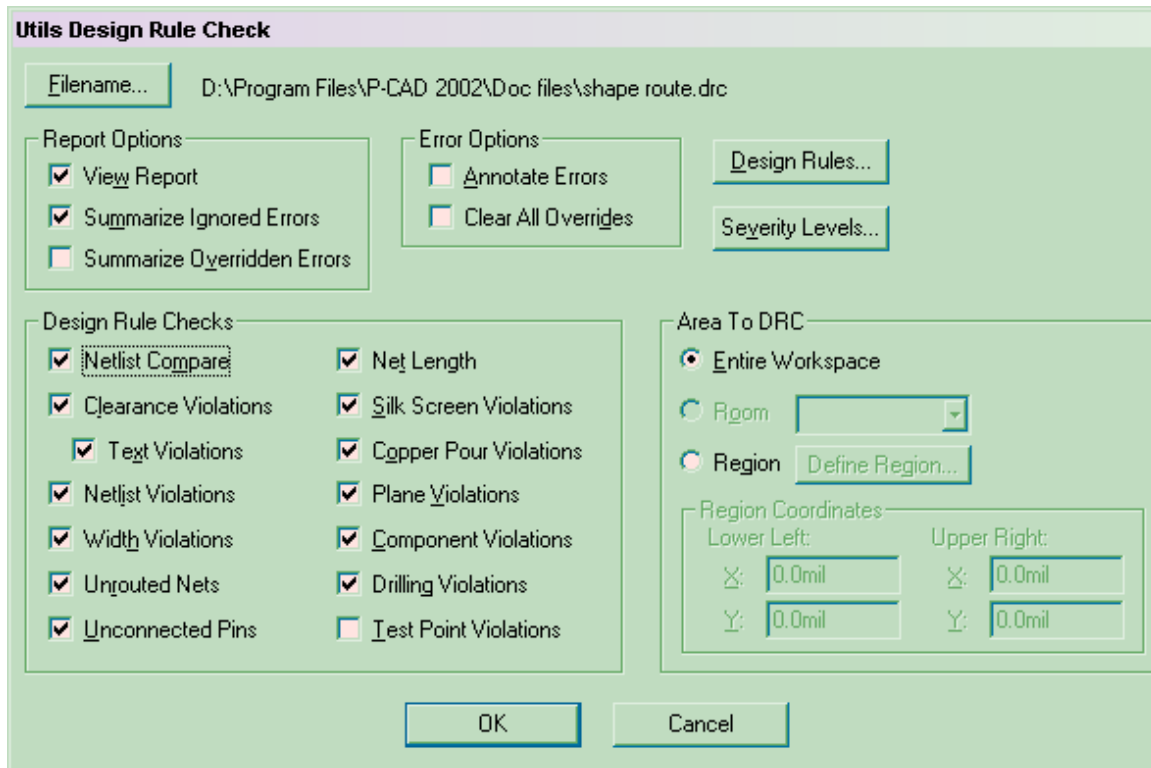
This is the same section of the board that was routed earlier with Quick Route. You can see that this route is much cleaner, has a minimal number of vias and was routed completely. Plus it was routed in about 20 seconds. You will still need to review the routed traces as there will probably be a few areas in need of clean up.

At this time, we do not have licensing for either of the Pro routers. The only current license for the Spectra router is held by the Fast Electronics Group. They can be contacted to set up a time to route your board should you need to use the Spectra router. They can also advise you on how to set up your "Do" (route parameters) file.

## 6. Post processing (or checking your work)

### 6.1 Design Rules Check

Okay, so you've routed your board, what now? Well, the first place to start is to check your work. We use Design Rule Checking for this. Select **Utils- DRC**.



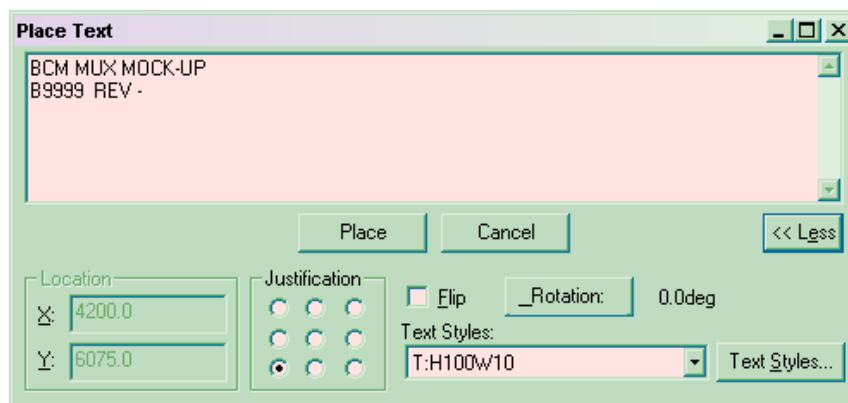
For the most part, it doesn't hurt to check everything under the **Design Rules Checks** section. You can turn off Copper pour violations and Plane Violations if you don't have either, but it doesn't hurt to leave them on. *Please make sure you are doing a **Netlist Compare***. This will verify that the netlist you ran off of the schematic matches the routed board. Also, please make sure to run **Unrouted Nets**. This will flag anything you may have forgotten to route. Clearance Violations is also an important one as it will flag any clearance violations you may have between your traces. If you check the Annotate Errors box, DRC will place little indicator symbols to indicate the locations where it found problems. If you click on any of these symbols, it will open a box that will detail the problem. Fix your problems and continue to run DRC until it comes up clean.

Now that the board has been checked and any errors cleaned up, we have a few things left to do before we can generate gerber.

## 6.2 Board Identifiers

First thing is you need to add text to your board

Make sure your active layer is set to Top Silk then click on the Place Text icon. A



Try to use a decent size font so that your text stands out from the Reference Designators on the board. You want to type in the name of the board, possibly preceded by the system or sub-system, the drawing number and the revision level. Place this in an open area on your board taking care to avoid pads & holes if possible.

## 6.3 Internal Plane borders

If you have internal plane layers and haven't done so already, add a 50 to 100 mil line around the edge of the board to prevent your plane from running all the way up to the edge and possibly shorting to other planes.

## 6.4 Gerber file labels

Next, we need to update the text we copied & placed underneath our board outline. An example piece of text might be: Solder Side (XXXXX.BOT) DWG # REV -. Update the text in the parenthesis with the name of your PCB file, (B9999P1.BOT). Change DWG # to reflect the artwork number of the board and update the revision level if appropriate. These lines of text will label your Gerber data and help you and the board house know what's what.

## 6.5 Fabrication drawing

Next we need to complete our formatting for our fabrication drawing. We do this now because this drawing will be one of the Gerber files we send to the board house.

First, we need to update the notes. Double click on the notes on the right hand side of the drawing to select and edit them. Note three should be updated if you are making a backplane (either .093 or .125) or if there is a special need to have it a different thickness. In most cases, leave it at .063. Next, update note four if you desire more or less copper thickness. 1/2 to 1 oz is pretty common. Use 2oz if yours is a high voltage board. If you have components on both sides, you will need to update note seven to reflect that. Note nine should be updated with the name of the board and the drawing number for the artwork. Click OK to finish editing. Now we need to place our drill table. There are two ways to do this. One uses a utility program. The other is through Document tool box and since it is the easiest method we will discuss it. First off, we need to assign symbols to our various hole sizes. We do this by clicking on the printer icon, then selecting the **Drill Symbols** button. Click on the **Automatic Assign** button, click **Close** and **Close**. Click on the



**DocTool** menu and select **Place Table**. Select Drill Table and define your line width (10 is fine) and your text font (125 or 100 if you have a large number of different size holes). Clicking the Advanced button will preview your table so you can make changes prior to placement. Add a column for TOLERANCE. Don't worry about the width as you can adjust it physically in the table. Enter +/- .003 for all sizes less than .055. +/- .004 for all sizes .055 to .124 and +/- .005 for all sizes over .125. Use your arrow keys to move from row to row. Do not hit ENTER! This will place your table and we don't want to do that until we're done adding our tolerances. Hit Enter or click OK to place your table once you have added all your tolerances.

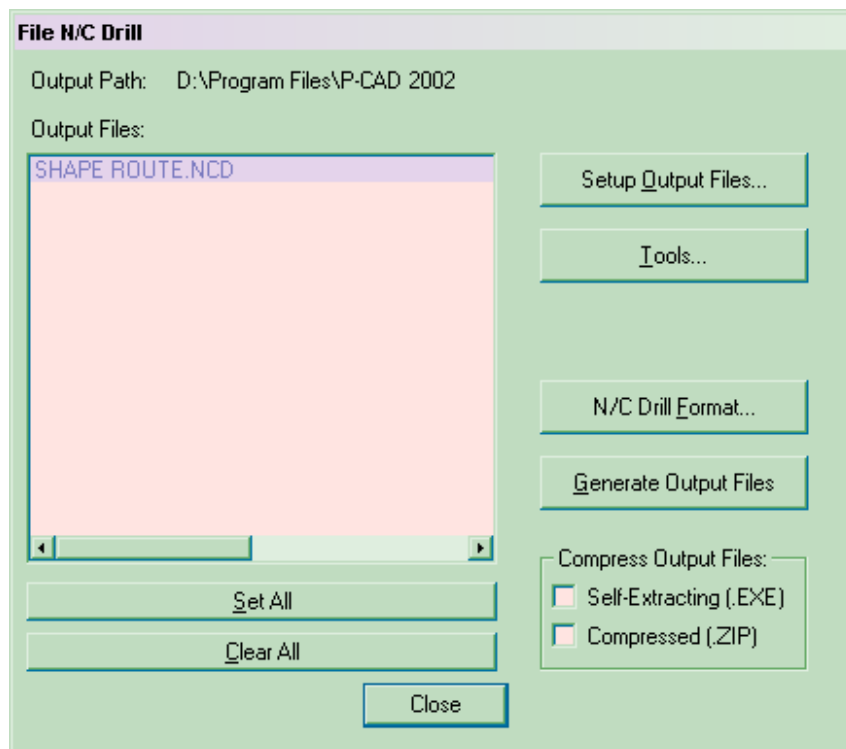
### **6.6 Assembly drawing**

Update the notes for the assembly drawing by clicking on the notes located at the bottom of the drawing. Add the schematic drawing number to note one in place of the xxxxs.

## **7. Generating Gerber and NC Drill files**

### **7.1 NC Drill file**

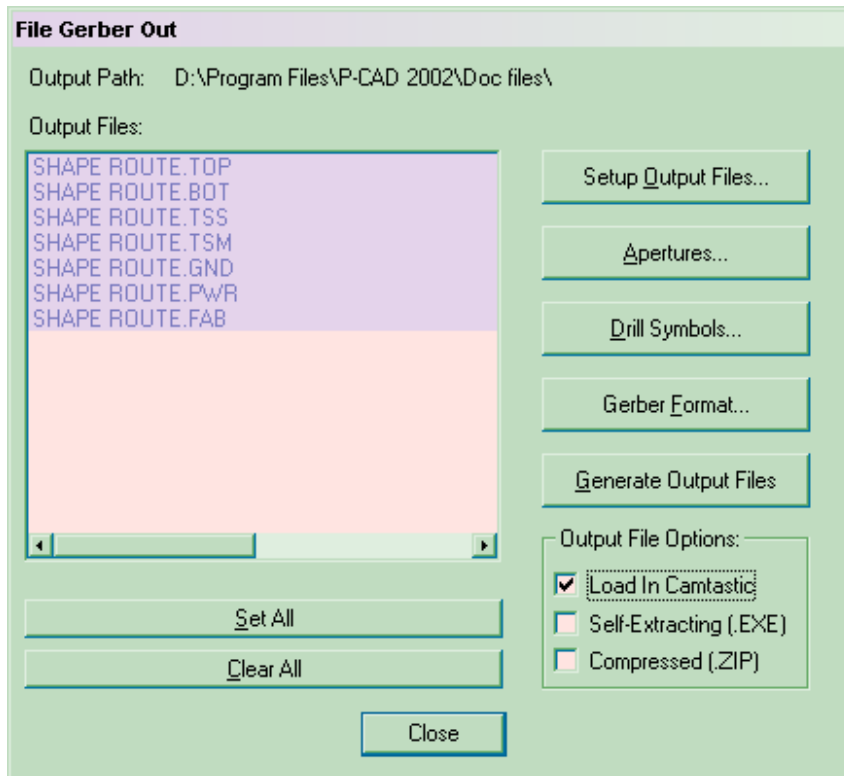
Before we generate our Gerber, we'll generate our NC Drill file. This is the file the board house uses to drill your board with. Click on **File- Export- N/C Drill**



Click on **Tools...** and click **Auto** to auto-assign tools to your hole sizes. Click **N/C Drill Format** and make sure the following are selected: Inches, ASCII None and Leading, click **Close**. Click on **Setup Output Files** and make sure to change your output path to the directory that your board lives in. (Ultimately, it would be great to write this & the gerber files into a directory within your board directory called Gerber.) Now, click on **Generate Output Files** to generate your file.

### **7.2 Generating Gerber files**

Okay. We've done all our prep work & now's the time to generate the all important Gerber files. These are the files the board house will make your board from. We generate these files in RS-274X which embeds the aperture information in the file so there is no need to generate a separate aperture list. Select **File- Export- Gerber**



The first thing to do is assign aperture, so click on **Apertures** and click **Auto** and **Close**. Next click **Gerber Format** make sure the output is in Inches and that the RS-274X box is checked. Make sure that the file listing is correct for your layer stackup/board configuration. If not, Click on **Setup Output Files** to modify it. You can also elect to automatically load your Gerber into Camtastic by checking the box as I have here. Camtastic is the tool we use to check our Gerber data. It is highly recommended that you check your files.

Take a deep breath, and click **Generate Output Files**.

Once it is done, a Camtastic window will open displaying your Gerber files. Uncheck the check boxes next to each layer then start turning individual layers on until you have reviewed them all. If everything looks good, then we now move to the read me file that you will include in your zip file.

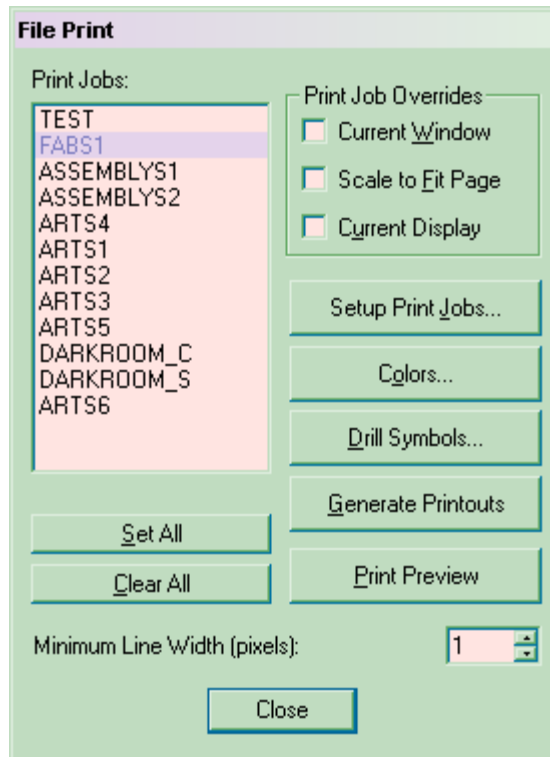
### **7.3 The Read Me file**

It is recommended that you include a read me file with the data you send to the board house. A template can be found at <\\jlabgrp\ecad\pcad\templates>. Update it to the requirements of your board. There are a number of items in italics that you need to update such as board name, physical size, total number of holes, your name & contact information, etc.

Create a ZIP file containing the N/C drill file, your Gerber files and the read me file. Name it something descriptive like the name of the board & revision level so you & the board house know what it is.

## 8. Printing

Printing in the PCB editor is very similar but there is one major difference from the schematic editor. You can set up Print Jobs in the PCB editor for things you would always want to print, like your assembly drawing for example.



We have multiple Print jobs set up in the JLAB supplied formats to save you the time & trouble of doing it. We have jobs set up for our standard outputs, fabrication, artwork & assembly drawings. The last two may have to be modified based on your board configuration.

Some people question the value of hard copy fab & artwork files. These drawings have been used to help re-create boards that have had their files either lost or destroyed. The artwork file can also be helpful if you need to modify an existing board. Fab drawings can be used by our buyers to get quotes. So, you see there are many reasons to create hard copies of these files.

Assembly2 is for your BOM unless you have a board with components on both sides. If that's the case, copy sheet 2 to sheet 3 and make sheet 2 the view of the bottom side components. I have written a procedure to help you create a view of your PCB from the bottom so that it will print out correctly. See the instructions below.

### 8.1 Setting up and Plotting a mirrored bottom assembly view

1. Under OPTIONS- DISPLAY change the following:
  - a. Uncheck the Display Plane Indicators box
  - b. Uncheck the Display Pad Holes box
  - c. Click the Hide button for Glue Dots and Pick & Place
2. Turn off all the layers not associated with the view you want to create, in this case everything except BOARD, BOT ASSY, BOT SILK and your format layers. Make BOT ASSY your current layer.
3. Under DOCTOOL, make sure Mirror On Copy is checked by selecting it.
4. Click & drag a window to select your board including the board outline.
5. Select EDIT- COPY
6. Select EDIT- PASTE Special then click OK to accept the choice.
7. Place your mirrored assembly view pretty much over the existing view.
8. Set up a print job for this view (should be sheet two of your assembly moving the BOM to sheet three). This view would include the following layers: BOT ASSY, BORDER and TITLE.

## **9. PCB layout start to finish**

1. Create a directory for your new project (on a drive that gets backed up regularly)
2. Open a JLAB format
3. Save the opened format into your new project directory giving it a new & descriptive name
4. Update title block information
5. Copy in an existing board outline or draw a new one
6. Increase the size of your workspace
7. Associate the libraries you wish to use using **Library-Setup**
8. Load your netlist
9. Place your components within your board outline
10. Route your board
11. Run Design Rule Check
12. Place your identifying text on the silkscreen layer
13. Add plane layer borders if applicable
14. Update labels for gerber
15. Update fab and assembly drawing notes
16. Place Drill symbol table
17. Generate N/C Drill file
18. Generate Gerber files
19. Update Read Me file
20. Create a Zip file of the N/C drill file, Gerber files and the Read Me file.
21. Plot vellums for signatures

## **10. Tips, Tricks & Hot Keys**

While in the Edit mode:

- **Shift-Left click** allows you to sub select any portion of a part.
- **CTRL-Left click & drag** allows you to copy & paste the object.
- Once an object is selected, **Right Click** brings up a sub-selection menu.
- **Left click- delete** to delete an object.

**C**- re-centers your View window to where you have your cursor located.

**+**- Zoom In

**-**- Zoom Out

**X**- changes the style of cursor you see

**U**- Undo

**O**- cycles through orthogonal modes

**R**-rotates components or sub-selected text

**F**- flips components or sub-selected text to the opposite side of the board

**G**- cycle through your grids

**W**- cycle through your line widths

**M**- starts the Macro recorder

**E**- plays a temporary Macro

**Z**- activates the Zoom Window (Remember, when using a window to select, left click and DRAG to draw your box.)

**Backspace**- undo your last mouse click when placing wires.

**Shift +Select**- allows you to sub-select component text so you can move & rotate it independent of the component.

**F1**- Help menus.

**11. General Rules & Guidelines**

Grid size should be set to 100 or 50 mils for parts placement & 25 mils for routing; 5 mils is acceptable for placing & routing surface mount components.

Components should be oriented vertically or horizontally, NEVER diagonally.

Filter caps placed adjacent to the power pin on the IC.

Voltages routed to the filter caps first, then to the IC.

Traces should be kept as short as possible.

Spacing between traces should be equal to or greater than the trace width.

Internal planes backed off from the edge of the board.

Pin one indicators for ICs, connectors and sips included on silkscreen.

The name of the board, base drawing number and revision level included on the silkscreen And should be larger than the reference designators.

Reference designator fonts should be legible and as large as possible. Try to use a font that is 90 to 100 mils tall unless your board is very dense. Then, no smaller than 60 mils.

Silkscreen orientation should always be either 0 or 90 (right reading) degrees.

Silkscreen should never cover a hole or pad.