

# Constructing and Producing a Printed Circuit Board

Craig Zofchak

Design Team 8

11/07/2008

## **Abstract**

When a circuit in its development phase is finished and working correctly, it is then necessary to take the prototype and make it into a finished product. Moving the prototype from a breadboard to a strip board is cheap and fast, but can lead to inefficiency's in layout and human errors in solder connections. Strip boards are also very inefficient for mass production and have a cheap look. A more professional and robust layout option is to use a Printed Circuit Board. This guide will show an individual the steps to make a working circuit prototype into a manufactured Printed Circuit Board.

## **Keywords**

PCB, freeware, Soldermask, Silkscreen, DRC

## **Introduction**

Printed Circuit Board, or PCB, design requires the developer to have already completed an electronic circuit. This electronic circuit design will be what is transferred to the PCB. The PCB is essential in most designs because it is where every circuit component comes together and is ensured proper connection and functionality. The PCB also will greatly reduce the size of the circuit by allowing greatly reduced chip sizes and much closer interconnects. The board also is very rigid and can take any two dimensional shape, allowing it to fit perfectly inside its finished product. Because the PCB is the final product, and for most industry applications will be produced tens, if not hundreds, of thousands of times, care needs to be taken to ensure it is done correctly, efficiently, and has the greatest cost advantage.

## **Objective**

The Objective of this application note is to teach the reader how to successfully layout a PCB using freeware software from PCB123. This software design is then available to be sent to the manufacturer to produce the PCB for the customer. The user will learn every step needed to successfully layout a PCB accurately and efficiently, while minimizing cost.

## **Drawing the Schematic**

Start by installing PCB123 freeware. The software can be found at [PCB123.com](http://PCB123.com).  
*Note: Using freeware is much cheaper for laying out your PCB. It is usually provided by the manufacturer who will then produce your PCB for you. Freeware, such as PCB123, will work for most standard circuits and is good for one time use PCB manufacturing. Manufacturers will sometimes provide a more advanced version for a fee. For complex*

*circuits, or frequent use, advanced layout software is recommended. Many advanced layout programs will not require you to use the freewares manufacturer.*

When you have installed the software start by opening **PCB123 Schematic**. The program will open and appear like figure 1 below.

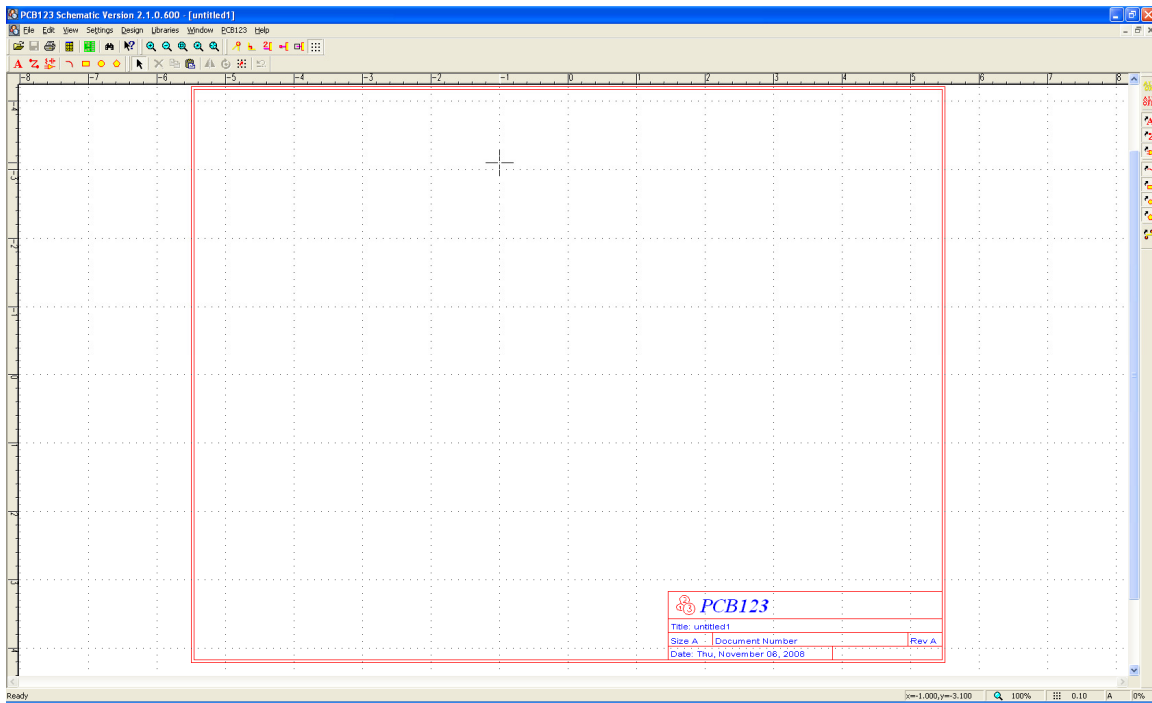


Figure 1.

The program that opens will allow you to layout your circuit's schematic and then cross check it with your own design. This will then allow you to map your schematic to the PCB layout view. Start by clicking The **Library Part** button (third from left on bottom).

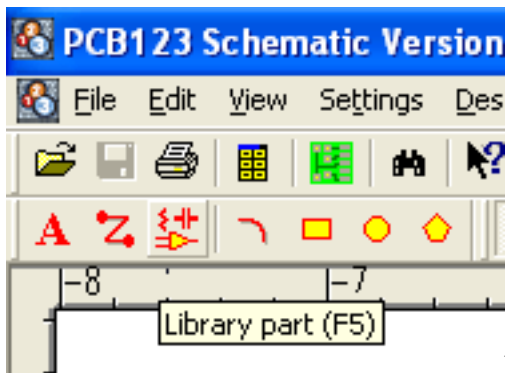


Figure 2.

In the library parts window click on the **Search** tab, enter a part from your design, and click the search button. Results should display like below in Figure 3.

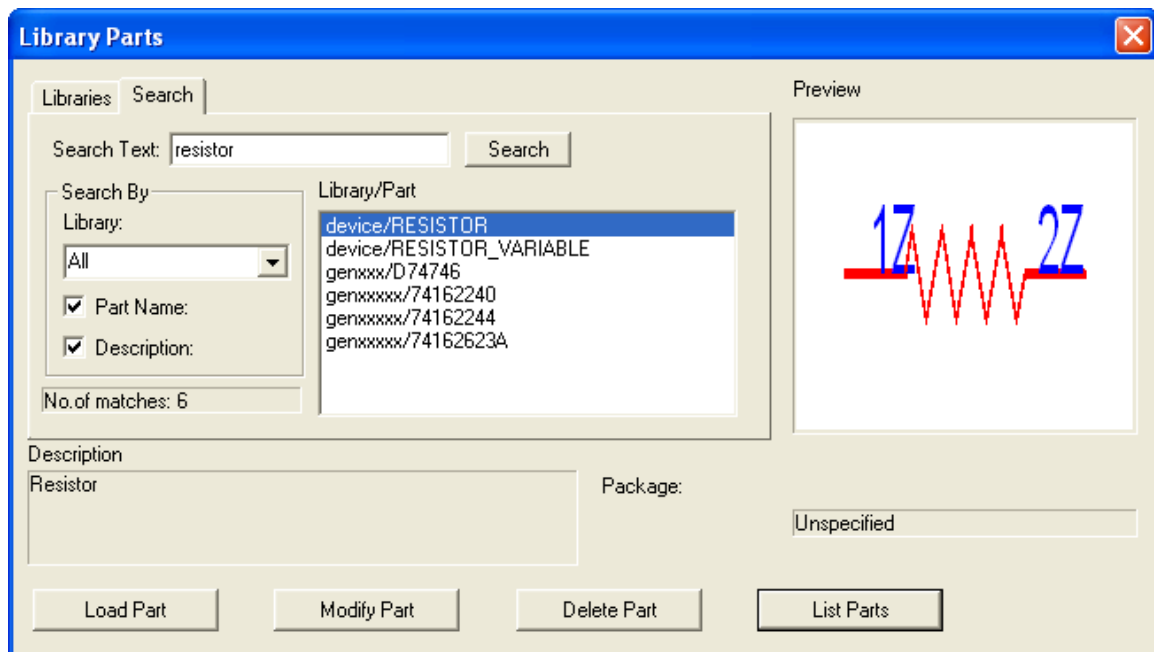


Figure 3.

Select the part you need from the Library/Part list and then click **Load Part**. The schematic view will redisplay. Click in the schematic where you would like the part to go and it will appear there. For multiple instances of the same part, clicking again will suffice. Repeat loading parts until all of the parts for your design are in the schematic view. The design should look like something below in Figure 4.

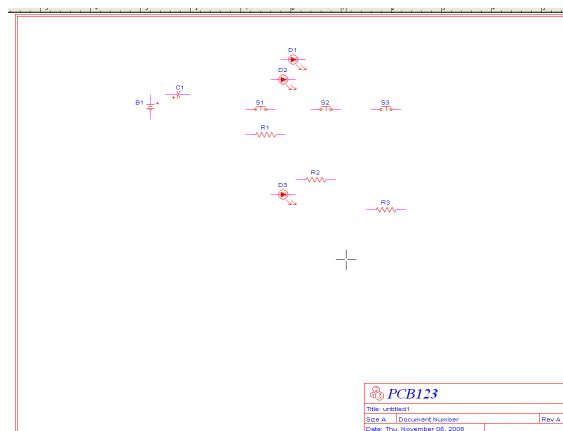


Figure 4.

To make the PCB more visually friendly you can click and hold on the parts and drag them to where they need to go. Once selected, parts can be rotated by clicking the **Rotate Selected Objects** button, or pressing the space bar. When completed your design could look something like Figure 5.

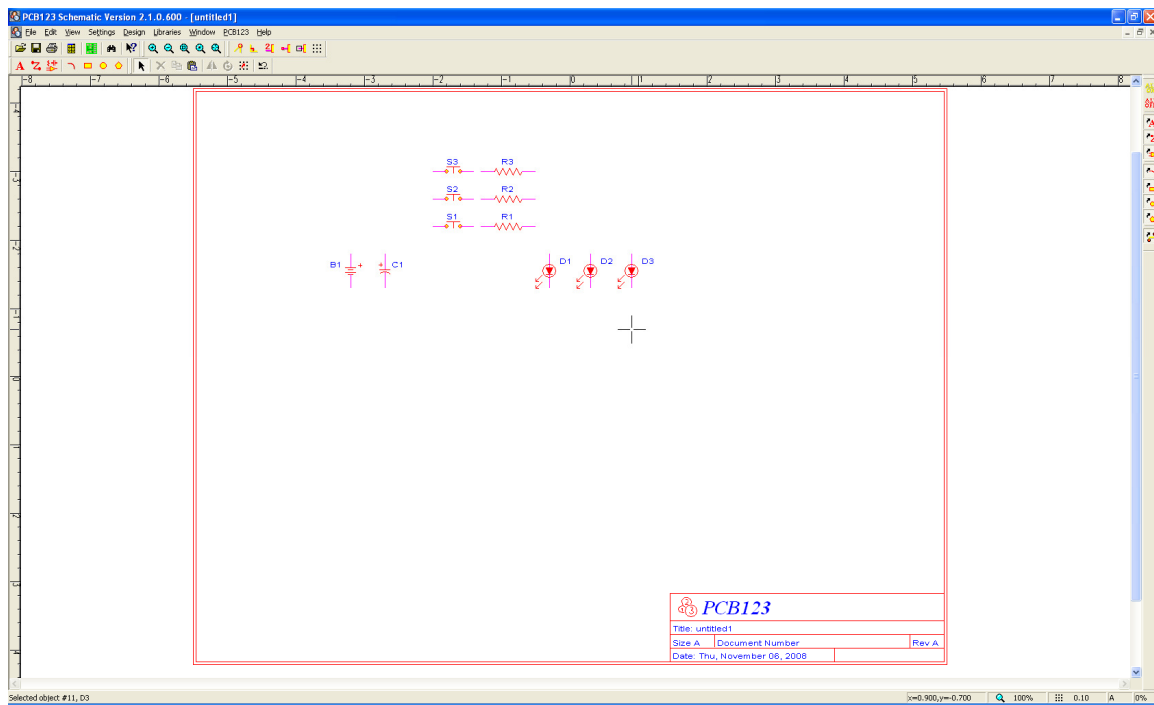


Figure 5.

The next step is to draw the wires to connect the parts together. Click the **Insert Wire** button to enable the wire connectivity. When Insert Wire is selected, click on the terminals of each individual part and drag to the interconnect or location you would like the wire to lead to. Continue this for each part until the design is completely wired together. When complete it should look like something below.

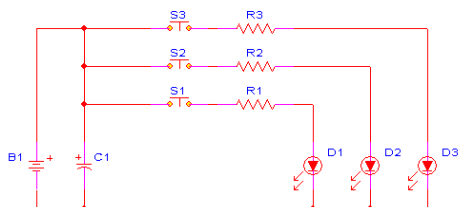


Figure 6.

It is important to check the schematic for loose connections. To do this go to **Design → Check Schematic**.

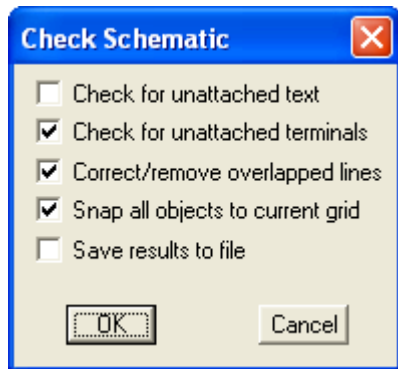


Figure 7.

Make sure the check boxes are selected for Check unattached terminals, and correct/remove overlapping lines are checked. Then push the **OK** button. Results will come back and should say there are zero errors. If this is not the case, go back and fix your schematic until it is error free. When this is done Save your design by clicking the **Save** button. Do not close your design after saving.

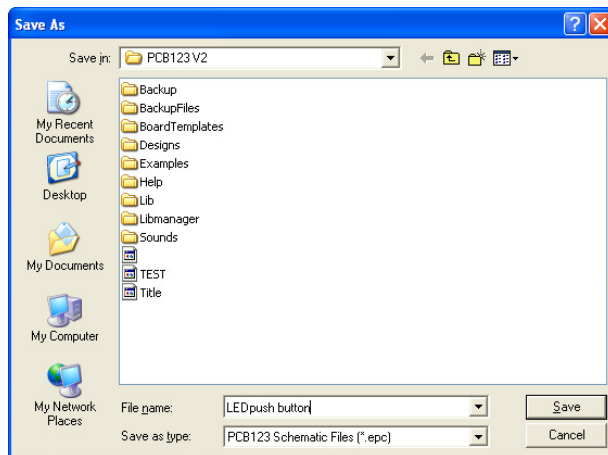


Figure 8.

## **Designing the Layout**

In the schematic view with the design still open, go to **Design → Convert to Layout**. Then when the pop up window appears click **Convert**. The new board wizard will appear.



Figure 9.

Enter a Board name in the blank and then click **Next**. Specify the width and height of your proposed board and then click **Next**. *Note: smaller boards tend to have lower prices.* Select the amount of layers needed for your design. *Note: Increasing layers will decrease your area, but will increase cost per board.* Two layers are usually suffice for most designs. Click **Next**. Select Soldermask, Silkscreen, and copper weight as desired. *Note: selecting a Soldermask, Silkscreen, Reducing board thickness, or increasing copper weight will increase cost. Some may require a Soldermask to select.* Select **Finish**. Click **OK**. If a device contains more than one possibly layout a window will pop up and prompt you to choose the footprint you would like in your design. Select the footprint

you would like and then click **Use this footprint** button.

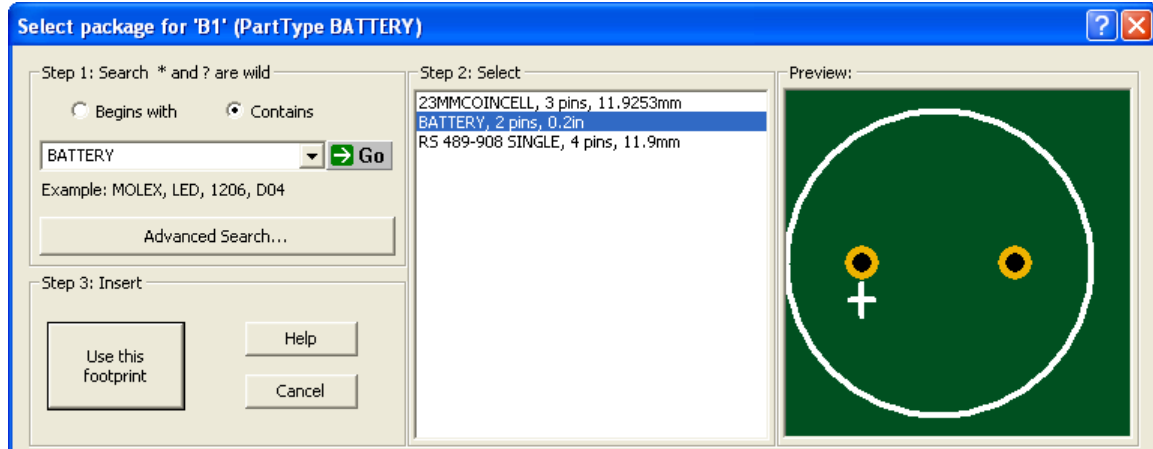


Figure 10.

Select **YES** to remember the part names. Click the **Done** button on the pop up window.

Your layout will look something like Figure 11.

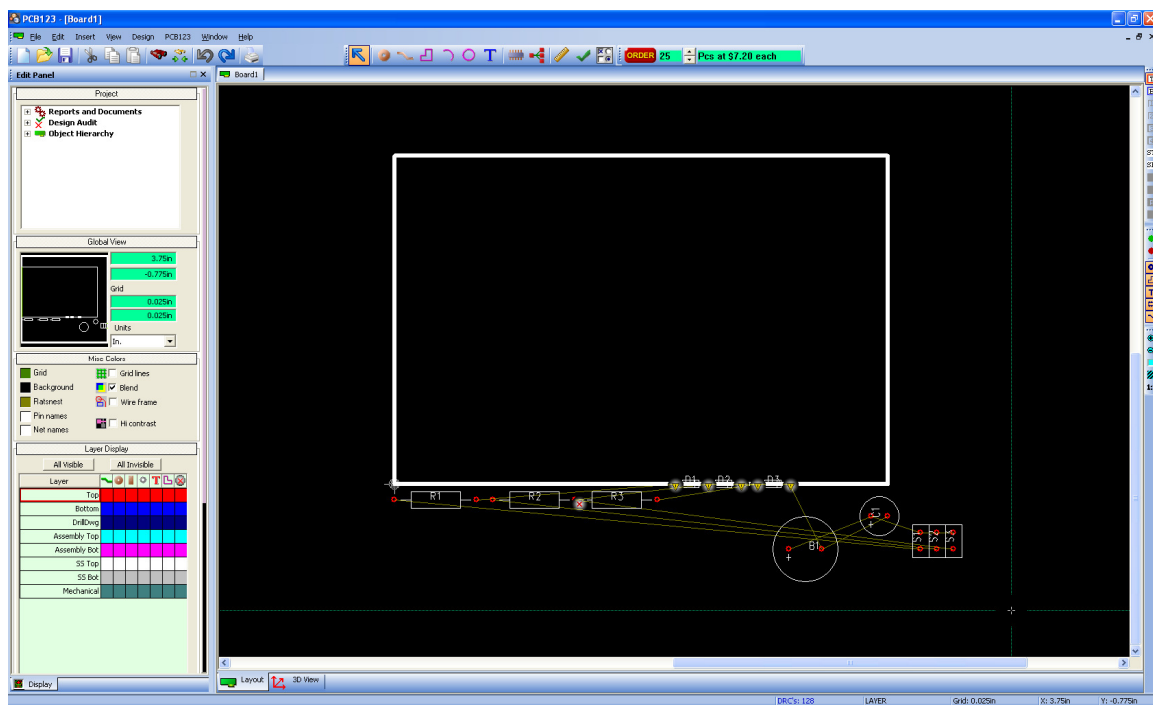


Figure 11.

The parts will need to be moved into the PCB box and laid out to minimize the area.

Click and hold on each part to drag it to the precise location inside the box. Make sure

that the interconnects remain when moving the parts. When the parts are laid out according to your specifications they should look something like Figure 12.

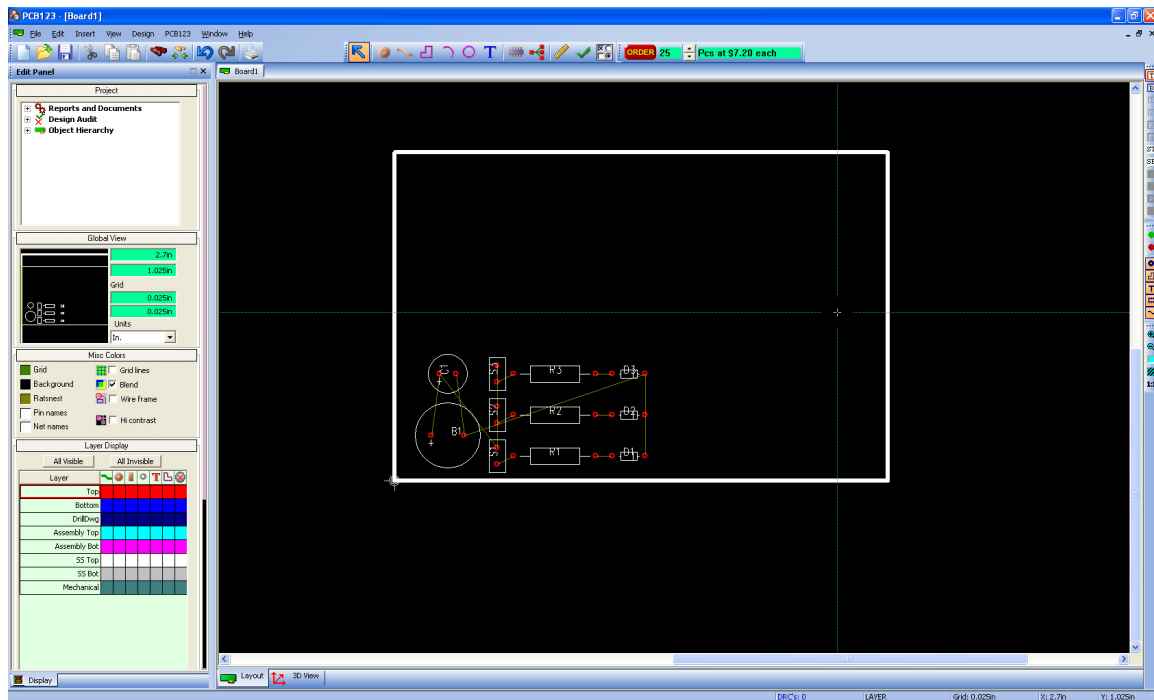


Figure 12.

In order to run the interconnecting wires between each piece it can be done by hand or more easily can be auto routed. Click **Design** → **Autoroute**. When finished there should be no errors and select **OK** to confirm. If there were errors there is a problem with your layout and most likely a part is placed on top of another one or parts are too close together. When finished the layout should look like Figure 13.

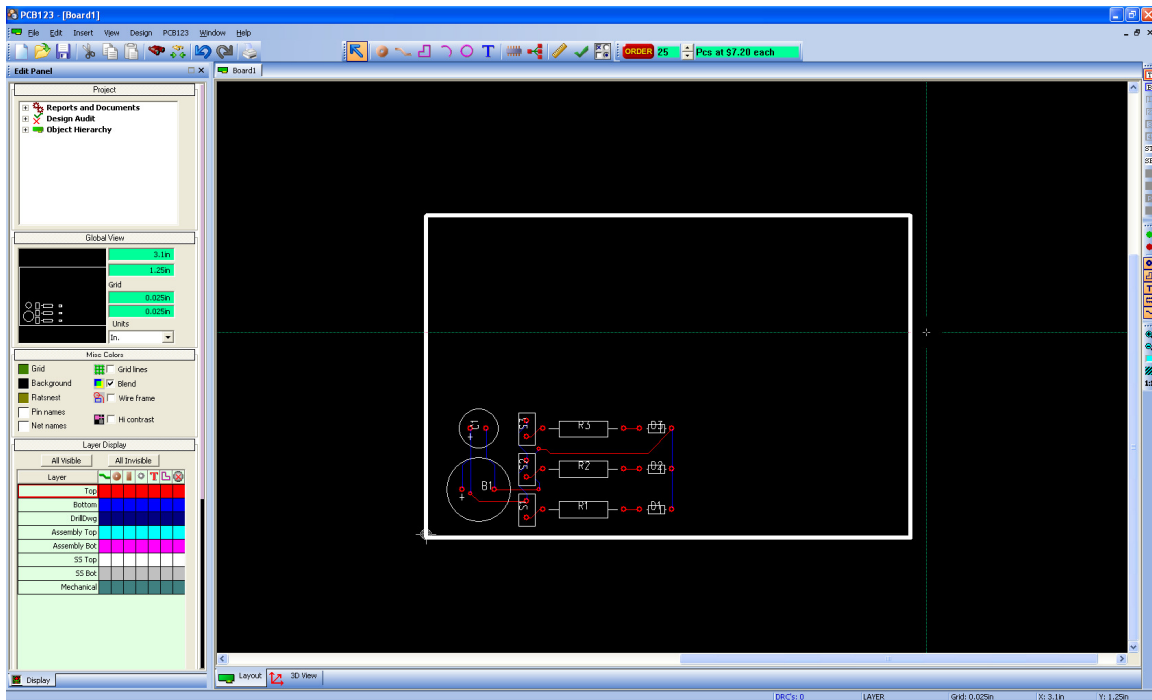


Figure 13.

To make sure all the Design rules have been met, Design Rule Check(DRC) should be run. Go to **Design → Run DRC**. Any errors and warnings should be unchecked to be excused if they are ok, although there should be no errors ideally before moving on. Click **OK**. Save your design. Do not close out of the window.

### Ordering the PCB

The next step is to order the PCB to be manufactured. In the Layout window go to **PCB123 → Place an Order**. A pop up window will appear and ask you to select a quantity. *Note: This works on economies of scale and ordering more PCB's will reduce your unit price.* From here on out continue through the order pop up window until the order has been placed.

### Results

Once ordering has been done the design will be sent to the manufacturer to be produced. When the design is done being manufactured it is then sent back to the user

competed. The Design board will look like the layout designed by the user. The user is then responsible for soldering any chips to the board to complete the PCB.

### **Conclusion**

You have now learned the basics on how to successfully create a PCB using layout software. Creating PCB's is necessary in almost all applications and is necessary in industry for completing products. More advanced PCB layout can be achieved using surface mount technology, amongst many other layout options. Many Manufacturers will mount these chips on the board for you and will ship the entire PCB ready to go right out of the box. Check with your manufacturer and software systems to check possibilities.

### **References**

[www.PCB123.com](http://www.PCB123.com)