



Complete Tutorial (Includes Schematic & Layout)

Download

1. Go to the "**Download Free PCB123 Software**" button or [click here](#).
2. Enter your e-mail address and for your primary interest in the product. (*Your information is kept private.*)
3. After you submit this information, we will send a link to your e-mail for you to download the free software.

Typical download time is:

- 10-to-15 minutes for a modem
- >30 seconds for DSL or cable modem

Want to know who else is using PCB123? [Click here](#).

Installing Software

1. Click on the software link in your e-mail and the downloading process will begin.
2. A dialog box will then ask you where you would like to install the software.
3. Go through the quick installing procedure, which includes a standard EULA (end user licensing agreement)
4. You are now ready to begin designing your printed circuit board!!

PCB123 Testing Tutorial

There are two applications included in our product that we will be using in our tutorial.

- **PCB123Schematic:** You can create an easy to read, one dimensional schematic diagram of a functional circuit and from this diagram a netlist can be generated. This netlist is basically a description of all the parts in your diagram and describes how these parts are connected to make your circuit. The netlist is useful for PCB123Layout to start with.
- **PCB123Layout:** You can layout and design physical characteristics of your schematic with PCB123Layout. Although you can design circuit boards without a schematic diagram, it is needed to run the batch routing routines in the PCB123Layout application. It is also a good idea to start with a schematic diagram to aid in the verification that all your circuits are connected properly on your final printed circuit board layout.

If you see problems or omissions from this document please let us know ASAP
support@pcb123.com

PCB123Schematic - "Creating the Schematic Drawing"

This tutorial will show you how to design the schematic drawing in *Figure 1*. You can look at this sample file in the following directory: *Examples/Decade.epc*.

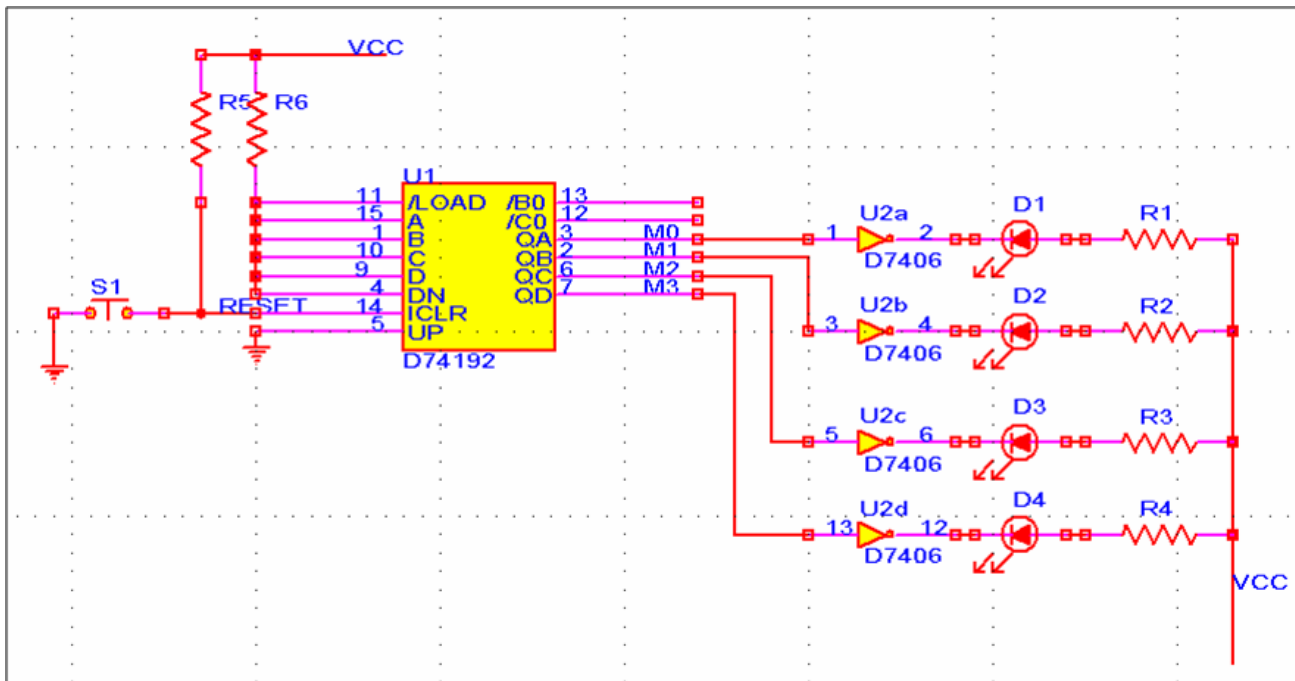


Figure 1: Example Finished Tutorial Project: Analog-to-Digital Converter Interface

Schematic Design

Schematic Diagram: A schematic diagram is a visual representation of an electronic circuit.

PCB123 Schematic is a computer-aided design environment used to help the user in the designing of an electronic circuit. This process, called **schematic capture**, is not just a simple drawing of a circuit. It provides information about the connectivity of all the parts that make up this circuit. This computer-based information is then used to create a **netlist** program. This netlist is then used in performing electrical design checks and to transfer the circuit connections and its component packages to the in the next step of the process of laying out the circuits as a printed circuit board (PCB).

This process of capturing a design of an electronic circuit involves the placing and arranging of schematic symbols, which fully or partially represent the electronic components that will make up the circuit, and then placing connecting lines to describe the continuity of the circuit.

Connectivity of Schematic Objects

Connectivity is the electrical links between objects in the schematic design area. Objects with electrical properties and their own functionalities are:

- Schematic symbols
- Wires
- Junctions
- Net labels

Schematic symbol parts: A schematic symbol is a representation of an electronic component.

These objects usually have multiple connection points or terminals that describe a pin location on the actual component. Each terminal has a pin number corresponding to the pin of the physical component and may have a label to describe this pin. A connection can be made to a terminal by drawing a wire to the terminal end or anywhere along its length. If a connection is made along its length a junction will be placed at the intersection.

Note: There are also special schematic symbols that will let you easily connect a network to ground.

Note: Hidden terminals are another way to show connectivity through a schematic symbol. This method connects hidden terminals to special net identifiers VCC, +v, -v, and gnd automatically through the design of the symbol part.

Wire Object: Wires are the main object used to define connectivity in a diagram. They can be arranged end-to-end creating a path from terminal to terminal. The main connection points of a wire are at its ends, but connections can also be made anywhere along its length. Junctions are used at these intersections to show they are connected.

Junctions: Junctions are used to show connectivity at intersections of wires or terminals. These are automatic. If a wire ends on another wire there will be a junction. If two wires cross but do not terminate at their intersection, there will not be a junction and therefore will not connect.

Net labels: Net labels are used to assign **net names** to electrical nets. They show connectivity not by physically joining networks, but by assigning common net names to different networks on the schematic. The netlister will interpret these networks and combine them into one network.

Starting a new Schematic Diagram

1. Open up the PCB123 schematic application.

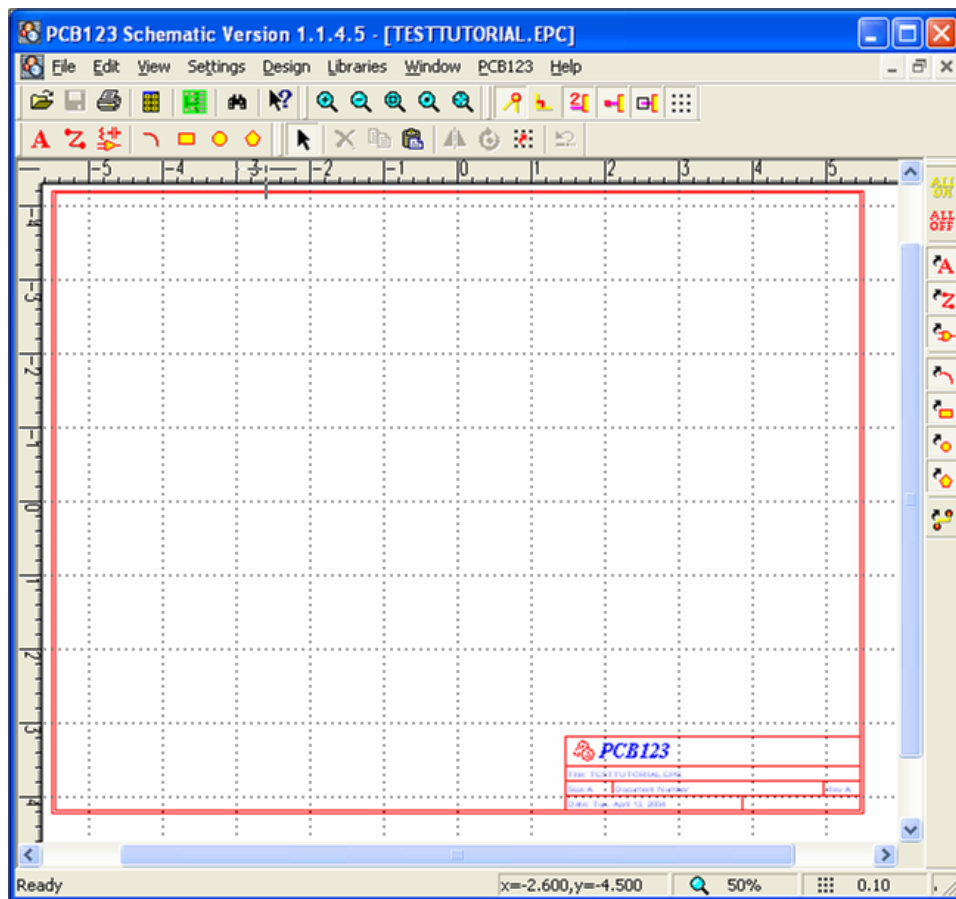


Figure 2: Initial Screen of the PCB123Schematic Application

2. Save the new diagram by clicking **File | Save As...** and name it '**TestTutorial.epc**'.
3. Notice that the Title in the title block, located at the bottom right of the page frame, will also change to reflect new file name '**Title: TestTutorial.epc**'.

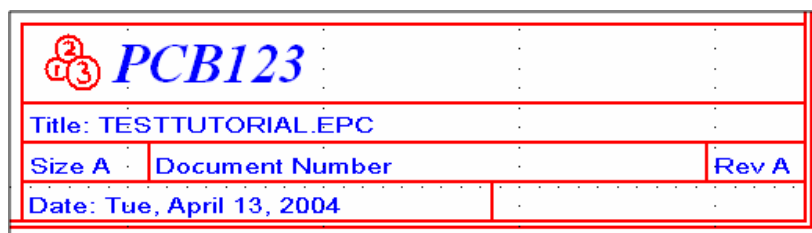


Figure 3: Title Block

4. You can edit the information in the title block by going to **Settings | Page Setup | Schematic Page Setup**.

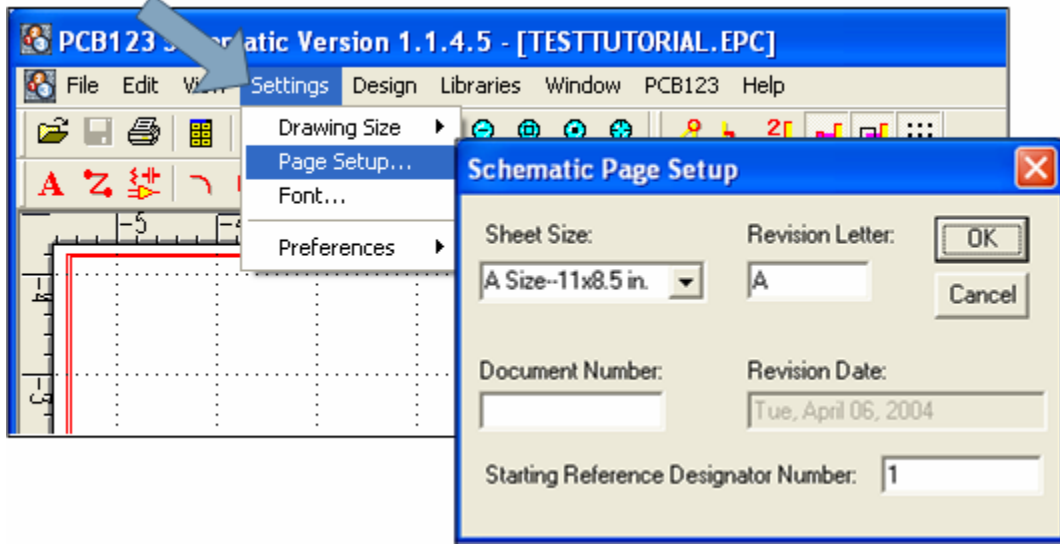


Figure 4: Changing Project Information

When you are designing, the snap-to-grid is always on.

Inserting and Editing Schematic Parts

There are 3 different ways of adding parts to your schematic diagram.

- Library Parts Dialog
- Using the Parts Bin
- Duplicating Existing Parts

Library Parts Dialog

1. Select **Edit | Insert Object | Library Part** from the menu or click the button  from the **Edit** toolbar to open the **Library Parts** dialog:

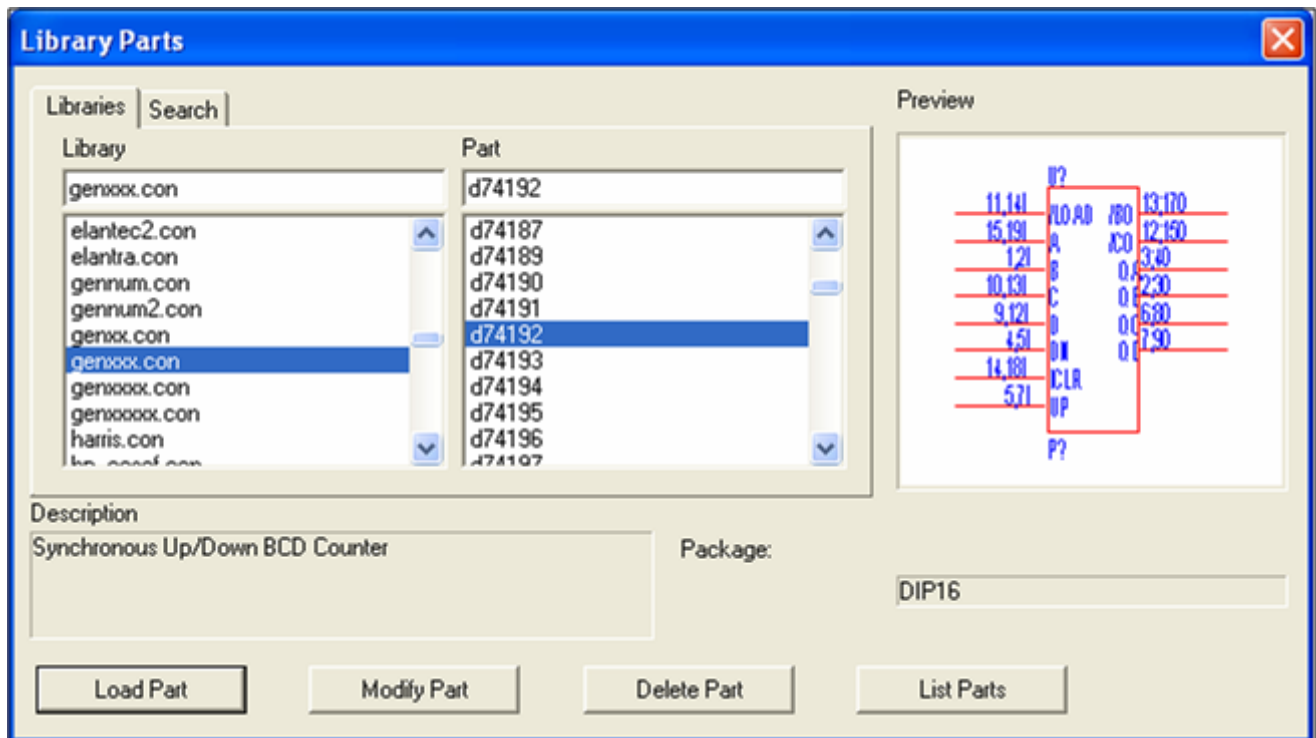



Figure 5: Library Parts Dialogue

2. Select **genxxx.con** from the **Library** list.
3. Select then **D74192** from the **Part** list.
4. Click the **Load Part** button.
 - Position the cursor inside the drawing border. (**Note:** You may have to zoom out to see the drawing border with the title block in the lower right hand corner.)
 - Left-click on the work area to place the part. (**Note:** By pressing the left-mouse button and holding it down you can drag the mouse to precisely place your part.)
5. Place these additional parts using this method:

Quantity	Library	Part Name
4	GENXX.con	D7406
1	DEVICE	Push_button

Using the Parts Bin

Common and frequently used parts can be accessed quickly from the Parts Bin. To display the Parts Bin, click on the  from the **Main** toolbar.

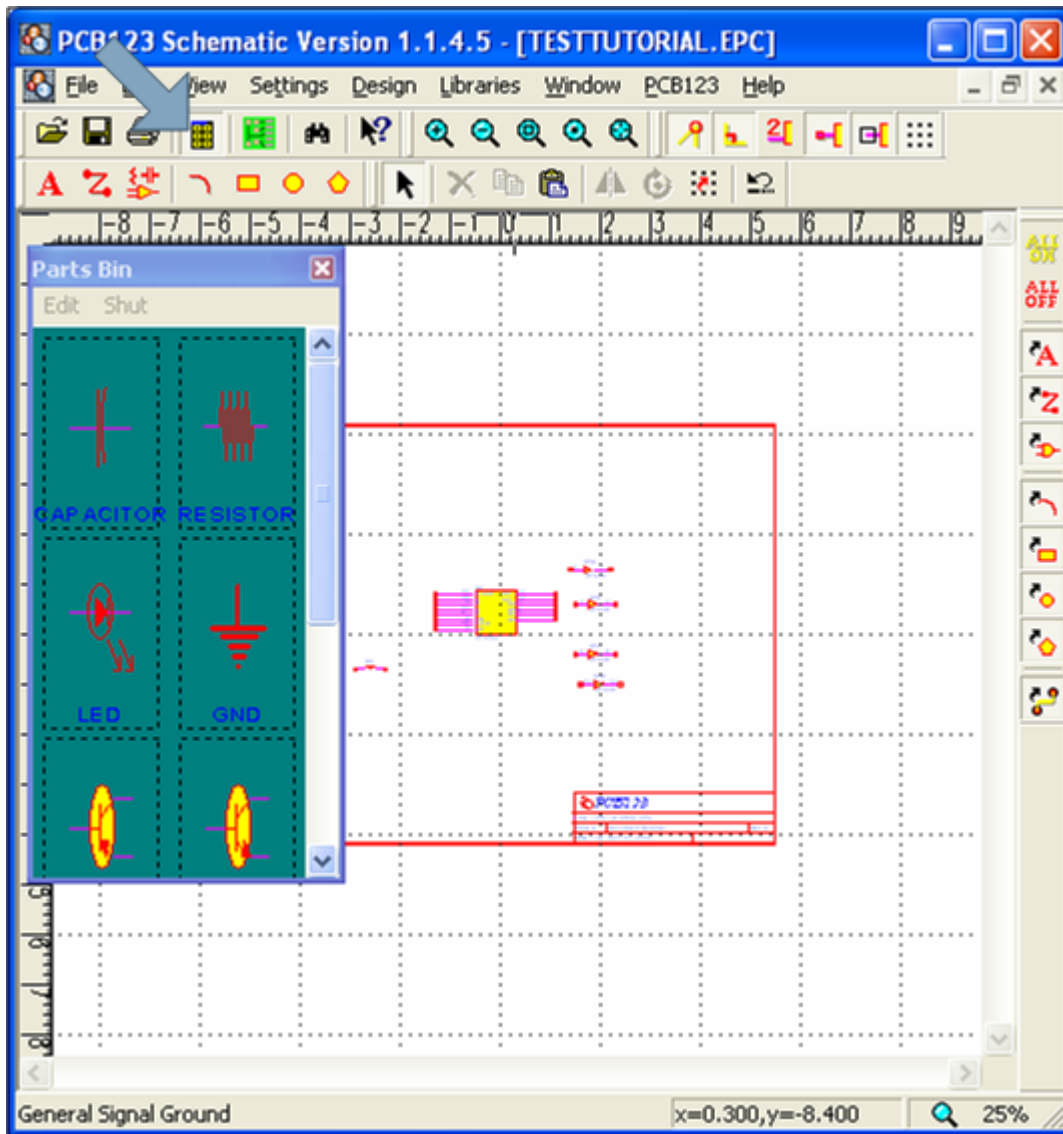



Figure 6: Part Bin Dialog

You will see a collection of 12 commonly used parts which can be modified to hold your favorite parts. To insert a part, simply select and drag the part onto your diagram.

Using the “Parts Bin” method, insert the following parts (if you can’t find the parts in the “Parts Bin”, look in the Library Parts dialog).


Quantity	Type	Library	Part Name
2	GND (ground)		
1	resistor	DEVICE	resistor
1	LED	DEVICE	LED

Duplicating Existing Parts

The “Duplicate Existing Parts” method consists of holding down the **SHIFT** key while holding down the left-mouse button and dragging a part (you have to be in Select Mode , which takes you out of the other modes). When you release the mouse, a copy of the part will appear at the location of the cursor.

Now use the “Duplicate Existing Parts” method to create duplicates from existing items on the diagram:

Quantity	Type	Library	Part Name
5	resistor	DEVICE	resistor
3	LED	DEVICE	LED

 Top of Page

Assigning Unique Reference Numbers

Once all parts are placed in the design, all reference designators for the resistors and LED’s need to be reset (renumbered).

1. To reset the reference numbers click **Design | Reset Ref Designators.....**and a dialog box will appear.

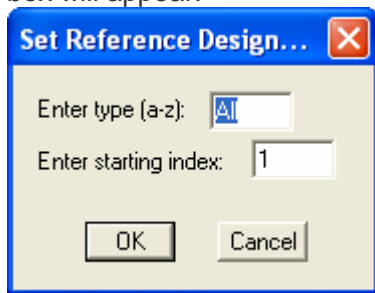


Figure 7: Reset Reference Numbers

2. Clicking OK will renumber **all** reference numbers starting with 1.
3. To reset only one type of item, enter its initial in the **Enter Type (a-z)** field. Example: to reset only resistors, type “R”, then OK.

Arrange Objects to Match Example

Arrange all the objects in the diagram to match the example in *Figure 1*. Refer to the section *Editing Objects* for help in arranging them. (The LED's were mirrored to achieve their appearance.)

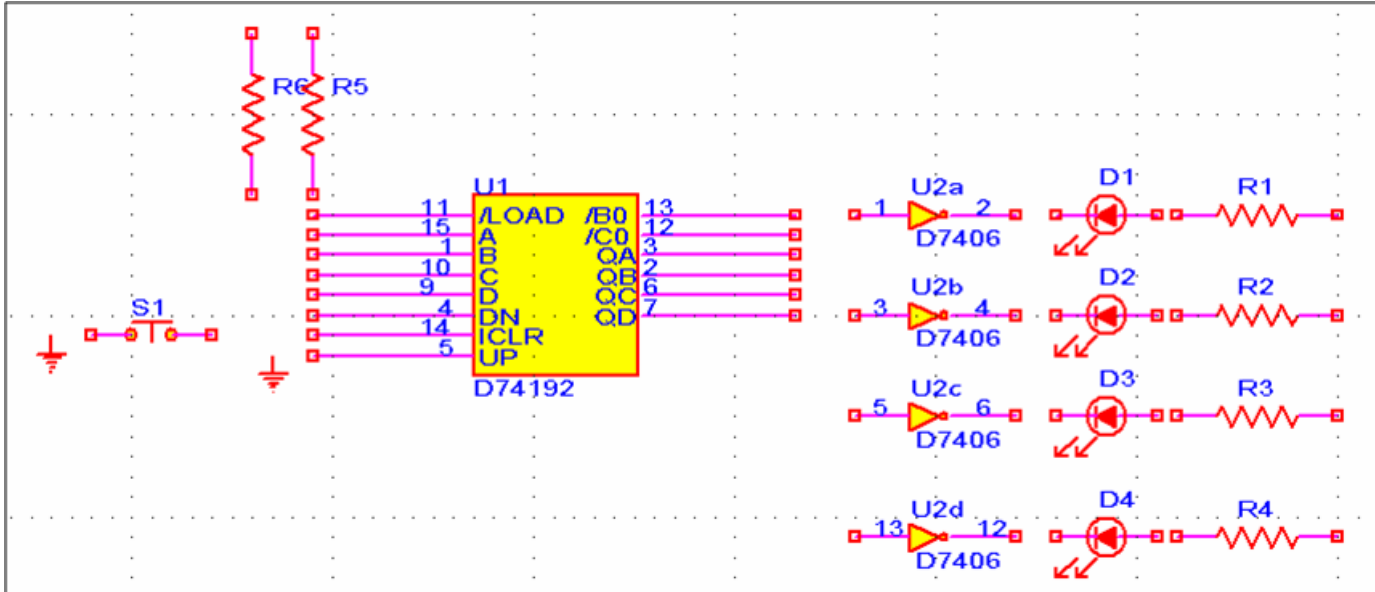



Figure 8: Objects Before Schematic Wiring


Editing Objects

Select Mode: To select an object, you must first be in Select mode, which is the default mode.


Select objects by clicking on the  icon in the toolbar, or use the Escape-key to get to Select mode, then left-mouse clicking on the object. You can select multiple objects by holding down the control-key while selecting objects.

Deleting: To delete a part, go into Select mode, select the object and hit the delete key.

Escape-key: To escape from other modes, hit the Escape-key.


Mirroring: The  button flips the object horizontally on the diagram.

Moving: To move parts once they have been placed, simply select the part by holding down the left-mouse button and then drag to a new position.


Rotation: You can rotate most objects as needed with the  button. If parts of the object do not rotate correctly, try mirroring.

Rubberband Mode: When Rubberband Mode is enabled, connectivity of the schematic is maintained when parts or wire ends are moved. Wires will stretch to maintain their connection to terminals of part when they are dragged to new positions in the drawing area.

Wiring the Schematic

To wire the schematic you will need to make simple terminal-to-terminal connections using the wire object . Each connection is a series of wire objects, segments, that are independently selectable and editable.

Note: A general rule is to stick to only horizontal and vertical wire layouts. The following steps describe how to insert a wire segment.

1. Select **Edit | Insert Object | Wire** from the menu or  from the **Edit** toolbar.
2. Position the cursor where you want the wire segment to begin.
3. Push and hold down the left-mouse button.
4. Drag the mouse to the position where you want to end the wire segment.
5. Release the left-mouse button.

Wiring Information

Junctions: automatically inserted by the program where a wire segment is started or ended in the middle of another wire (see *Figure 9*).

A junction will **not** be placed if:

- **A:** two wires cross each other without terminating at the intersection
- **C:** two wires connect at their ends only, such as turning a corner

A junction **will** be placed if:

- **B:** there is an accidental termination in one of the wires, at an intersection, but it is hard to see unless you click on the wire.
- **D:** there is a termination at an intersection.

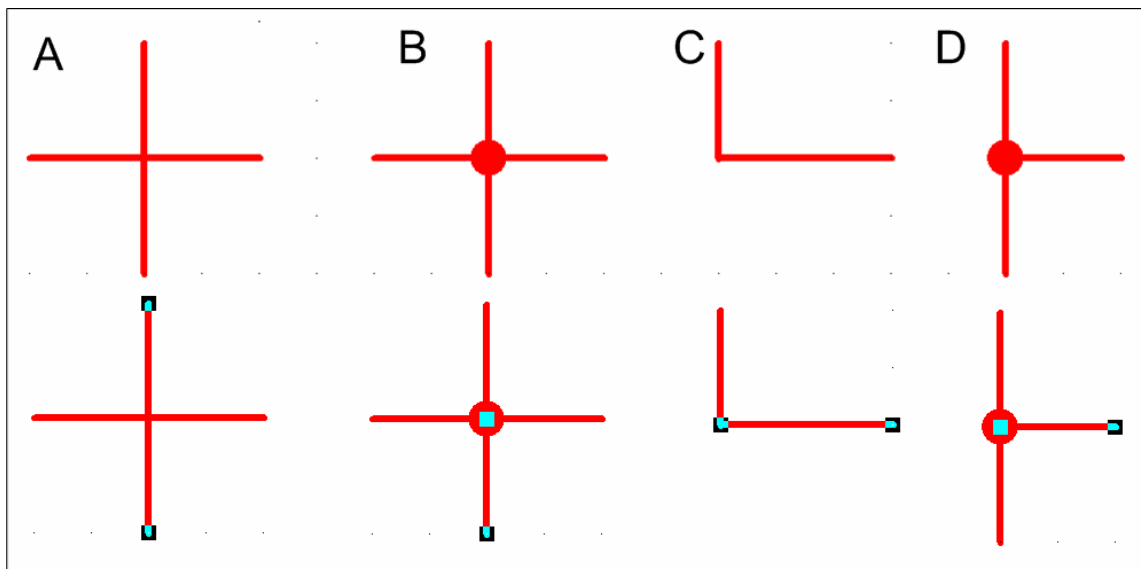


Figure 9: Junction Examples and Troubleshooting

Segments: segments can be a selectable part of a connection.

Connections: Connections are a collection of segments between objects.

Adding/Modifying Net Names


The netlister will assign each net a unique **net name**. However, you can manually name nets so that their functions within the associated circuit are easier to understand or remember over long periods of time. For example, you may want to call a clock wire "CLOCK", and a set of memory address lines "A0"-"A11".

If you assign two separate nodes to the same net name, they will effectively be connected electrically in the netlist.

In our schematic diagram there are seven nets that have been manually assigned. For this design, the only net that is required by the netlist to be assigned is the VCC. (See Note below)

Use the **Text** object to add net names to the network. You should be in snap-to-grid mode, with the cross-hairs cursor showing. Place the cursor on any vertical, or horizontal line, or device terminal, which belongs to the net. Click, and type the name; push the "**Esc**" key to end the name.

Follow these steps to assign new net names:

1. Go to the menu, and select **Edit | Insert Object | Text** or click  from the **Edit** toolbar.
2. Locate the cursor on one of the wire segments of the net to be assigned and left-click your mouse.
3. Enter the new net name.
4. In the project, you need to enter **VCC** and **Reset** in the locations similar to *Figure 10*. Editing the VCC in this design will effect 3 locations.

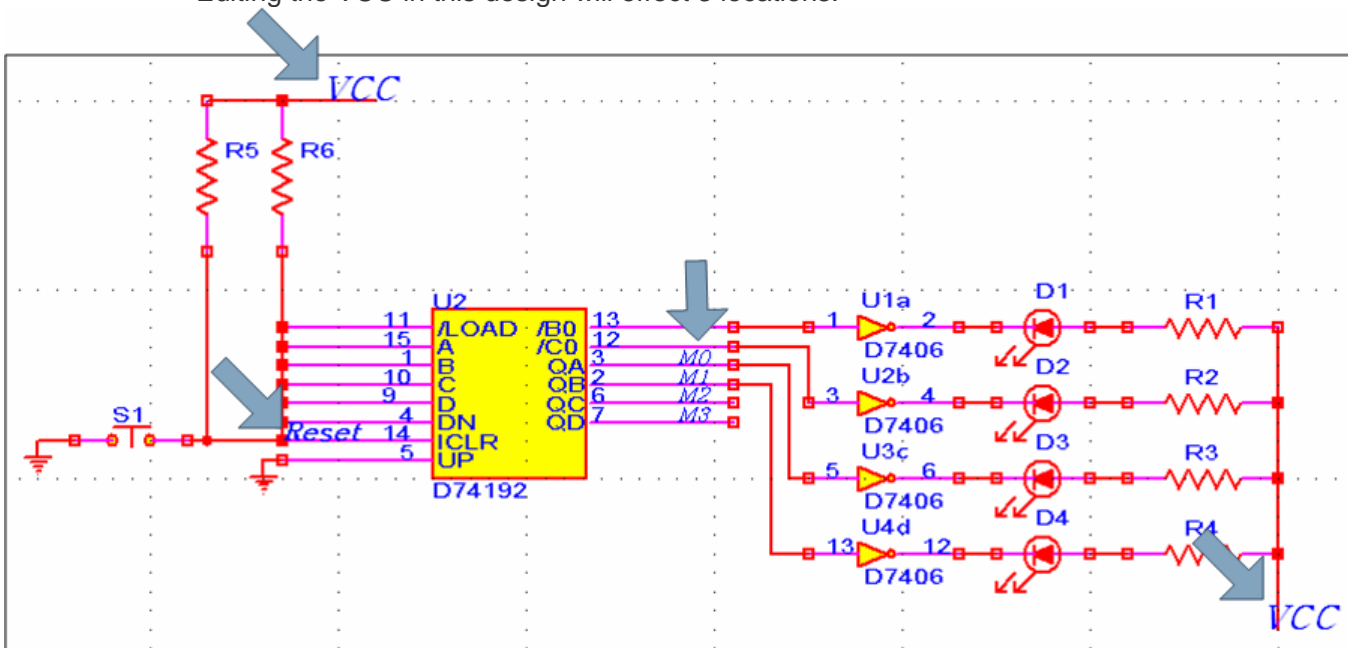


Figure 10: Entering Net Names

NOTE: VCC is a standard net name, which refers to the digital IC +5 volt power. Although these terminal pins are not displayed in the schematic for each IC component, they are taken in account by the netlister. When the netlist extracts the netlist it connects any terminals connected to a net named VCC to the IC +5 volt power. The same is true for signals attached to the ground symbol (GND part). They are automatically connected to the IC ground terminals by the netlister.

It is not necessary to have net names on all nets in a schematic; the netlister will add default names.

The Text object location must be within five screen pixels of the wire being assigned.

Net names are limited to 8 or less characters and cannot contain the space character. (For example, do not use CLK IN, instead use CLK_IN)

Caution: Do not end the net name with a dash character '-'. It will cause problems with the netlist format when it is read into the Layout application.

Finishing the Schematic

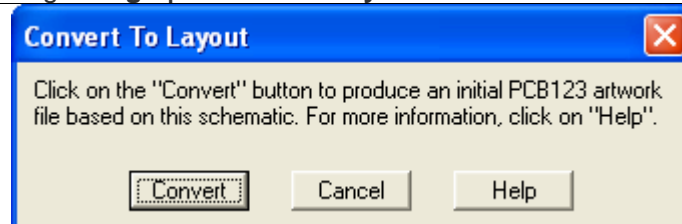
All essential parts have been done to generate a netlist at this point. To save the schematic diagram select **File | Save** from the menu.

Convert To Layout

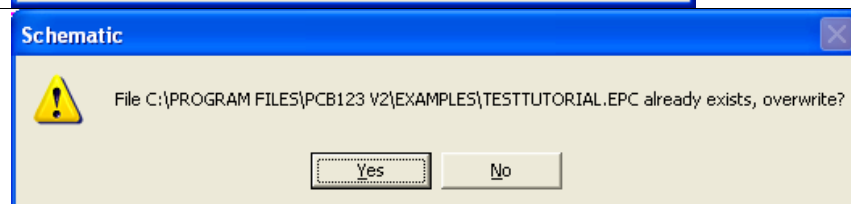
Creating a Netlist and Auto Place Parts

While still in PCB123Schematic you can go to the PCB123Layout application in one simple step by going to the menu and selecting **Design | Convert to Layout**. You will see the following dialog:

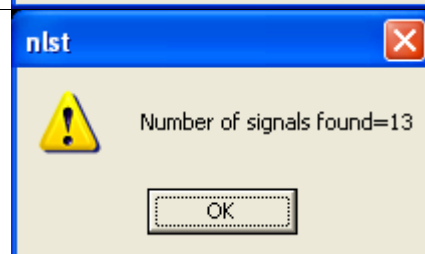
1. Click Convert.



2. Click Yes



3. Click OK.

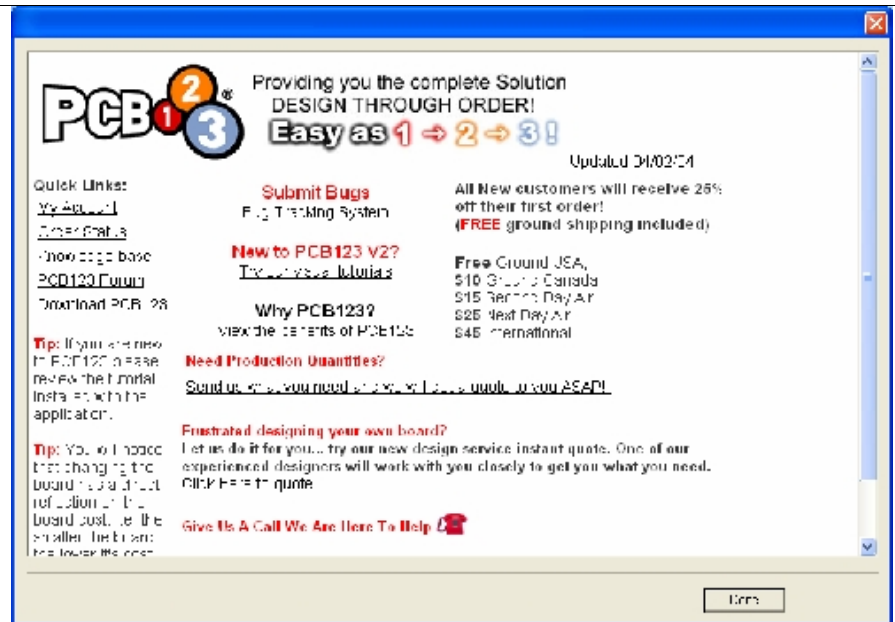


4. A new netlist will be generated in PCB123Layout.

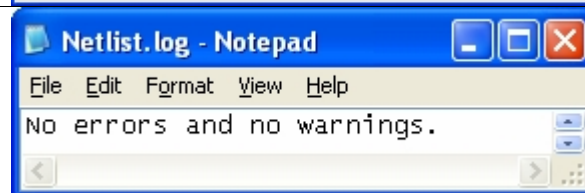
5. The PCB123Layout application launches

6. All components in your design will be placed in Layout application.

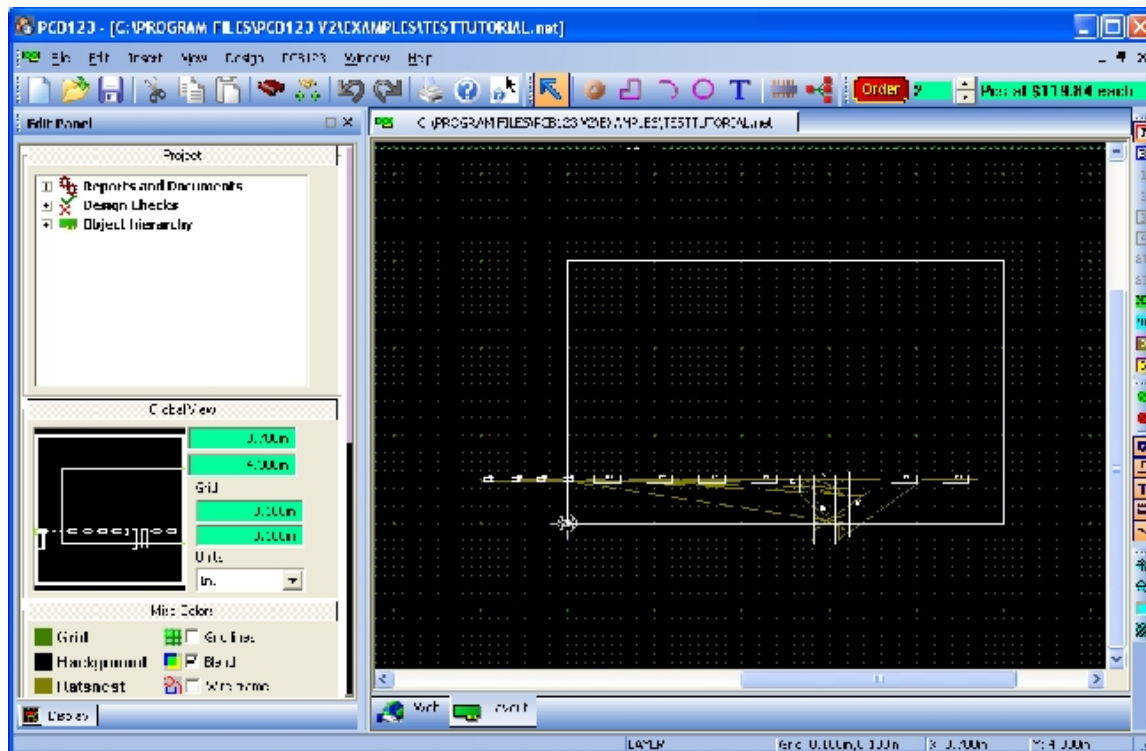
7. A Layout information dialog will be displayed, Click **Done**



8. You will see this report or you will see an errors report.



9. This is your circuit in PCB123Layout, save it, then proceed.



Board Configuration

To edit the Board Configuration:

1. Go to the Standard Menu
2. Select **Design | Board Configuration**

The Board Configuration dialog will be displayed. Select the board configuration you need, then click OK.

The screenshot shows the 'Board Configuration' dialog box with the following settings and annotations:

- Board Name:** TESTTUTORIAL.net
- Netlist file name(s):** (empty field)
- Board Size:** X 5.03, Y 3.03, 15.21 Sq.In
- Layers Required:** Four Layer Board (selected)
- Power Planes:** Use Inner 1 as a plane layer (checked), Use Inner 2 as a plane layer (checked), Use Inner 3 as a plane layer (unchecked), Use Inner 4 as a plane layer (unchecked)
- Solder Mask:** (checked) Solder Mask (Selecting Solder Mask will enable Silk Screen)
- Silk Screen:** Top (selected)
- Material Thickness:** 0.062" (selected)
- Finished Copper Weight:** 1-oz (selected)
- Pricing:** Quantity: 2, Unit Price: \$145.39, Silk Screen Charges: \$0.00, Subtotal: \$290.77
- Note:** Cost does not include shipping and applicable taxes.
- LEAD TIME:** Orders in by 8am Pacific Time will ship in 4 business days. (Example: Orders in by 8am Monday, will ship Friday.)
- Check if you do not want this screen displayed every time you open a new board:** (unchecked)
- OK** button

Blue arrows point to the following elements:

- Board Size fields
- Layers Required section
- Power Planes section
- Solder Mask checkbox
- Silk Screen section
- Material Thickness section
- Finished Copper Weight section
- Pricing section

Figure 11: Configuring Your Board

PCB123Layout - "Laying Out Your Board Design"

Before you continue, you may want to edit your grid's visibility, which can aid you when it comes time to route your board.

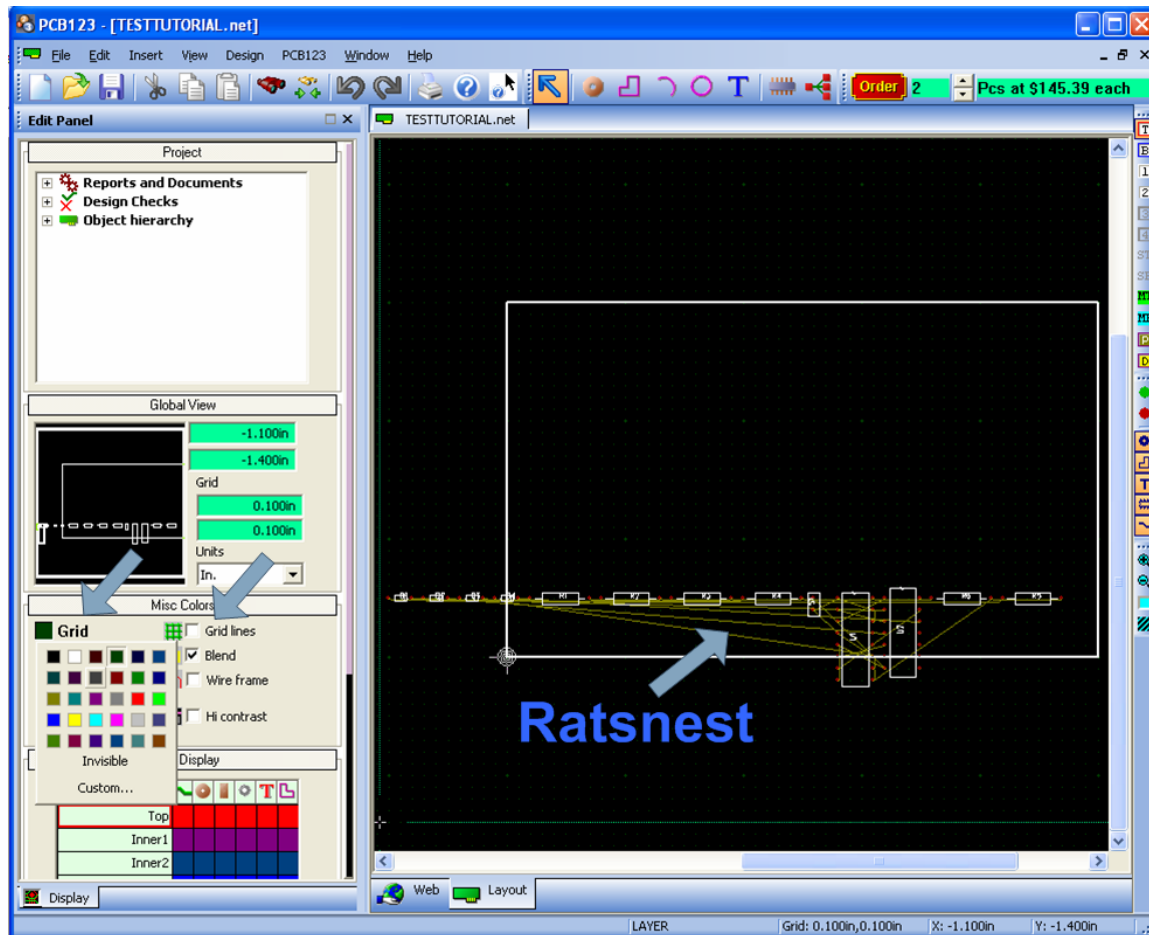




Figure 12: Modifying the Grid

Rearranging the Components

The goal in rearranging components on the board is to move and orient the components in such a way as to insure the shortest possible routing of each net.

1. Make the Ratsnest invisible to make it easier to rearrange components. Since we are getting ready to place the board, we don't care about selecting text or doing any routing.
2. Turn these off by toggling the  button and the  button on the right-hand toolbar.

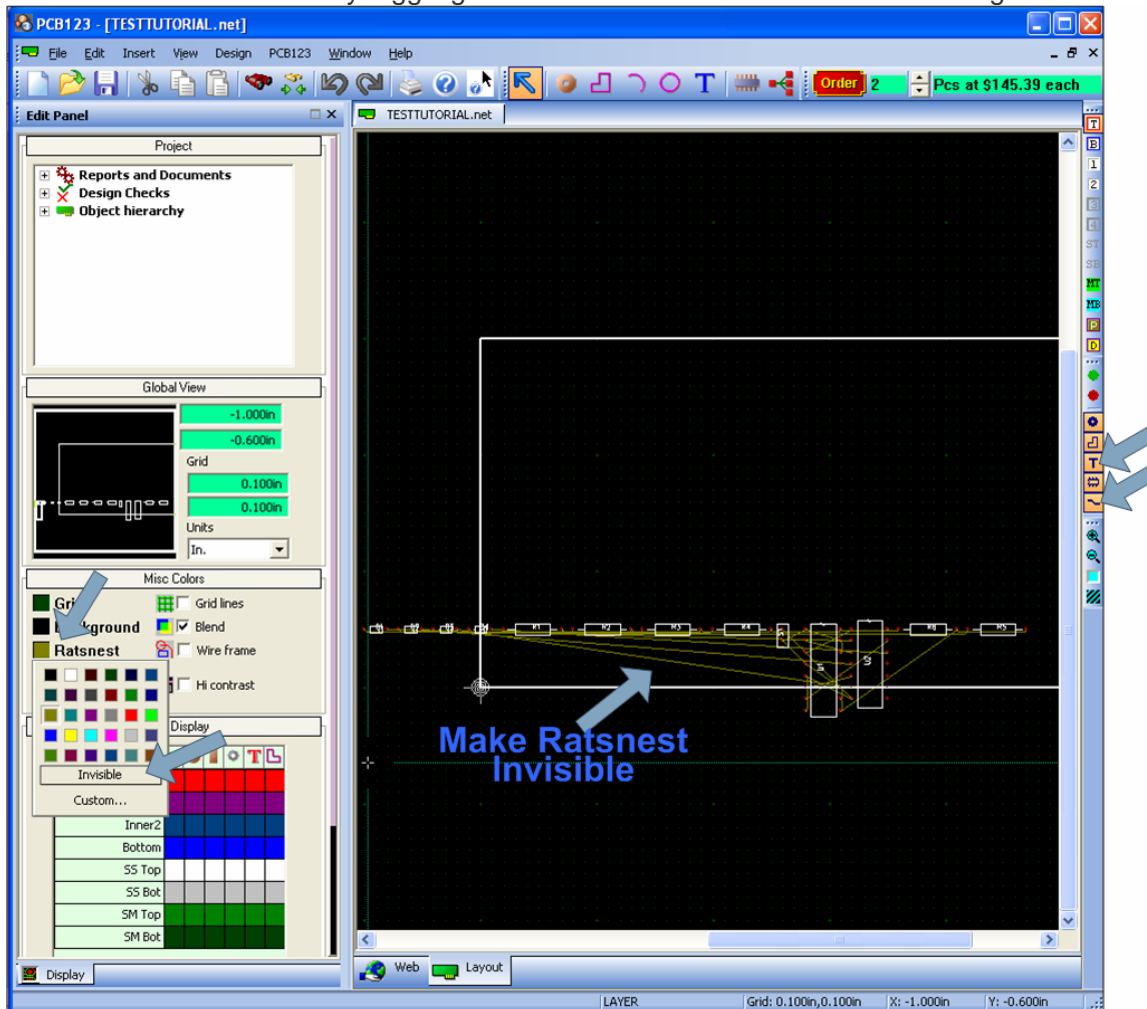


Figure 13: Making the Ratsnest Invisible

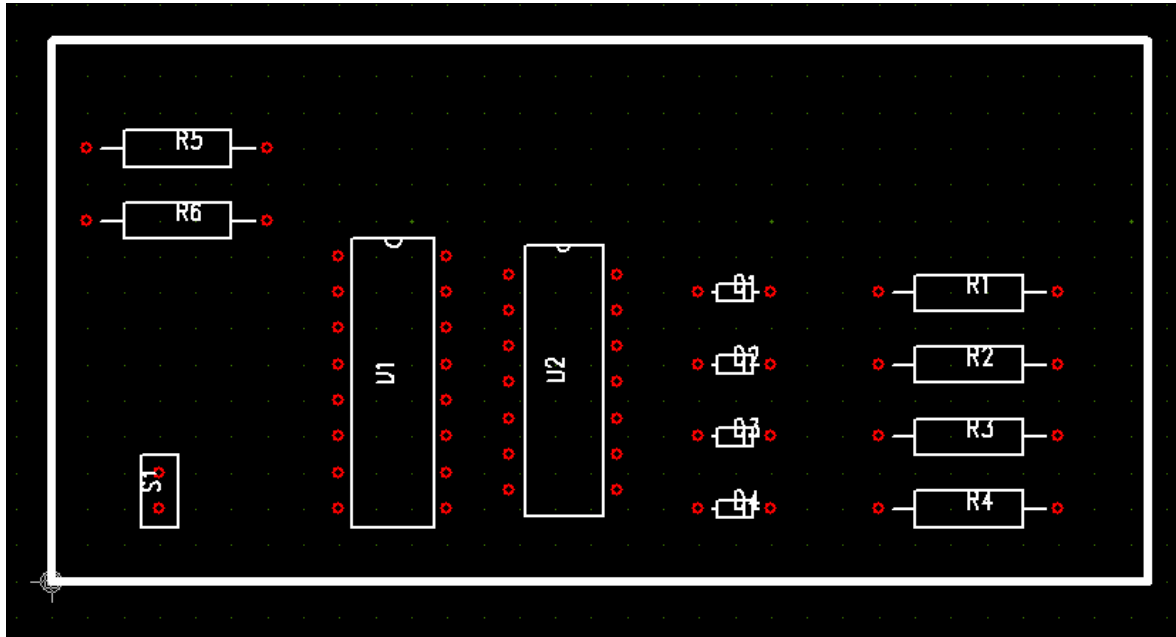
1. As a general rule, you should place the component with the most connections in the center of the board.
2. Next, select the component with the next greatest connections and place it. You may have to rotate the component to get the right fit. Continue this process until all components are placed.

Editing the Board Outline

The Board outline describes the general shape of your board. Board Outlines can be generated,

or modified with these three tools: Polygon , Arc  and Circle .

To edit the shape of your board, doubleclick anywhere on the board outline, and drag it to the desired position.



re 14: Rearranged Components and Board Outline

Figur

Route the Board

Routing is the process of determining the path of routes that electrically connect components together. In our tutorial these connections are determined by the netlist generated by the schematic drawing supplied by the tutorial. By displaying the **Ratsnest** you can view these connections.

Manual Routing vs. Autorouting

The process of manually routing connections is to route the path of each connection one by one. These paths may go from one layer to another through **vias**. The goal here is to route all connections required by the netlist using the shortest path possible, while using a minimal amount of vias. Depending on your design this could be a very time consuming process. Fortunately this process can be made shorter and easier by using the integrated autorouter. The autorouter automatically routes all connections in the netlist.

Authorouting

This section demonstrates autorouting the board, or creating connections between components the easy way.

1. First, press the '**G**' key on your keyboard to invoke the **Grid** dialog. Enter a value of 0.025 to establish your routing grid.
2. Go to the menu and select **Design | Autoroute** and watch it start generating **routes**.

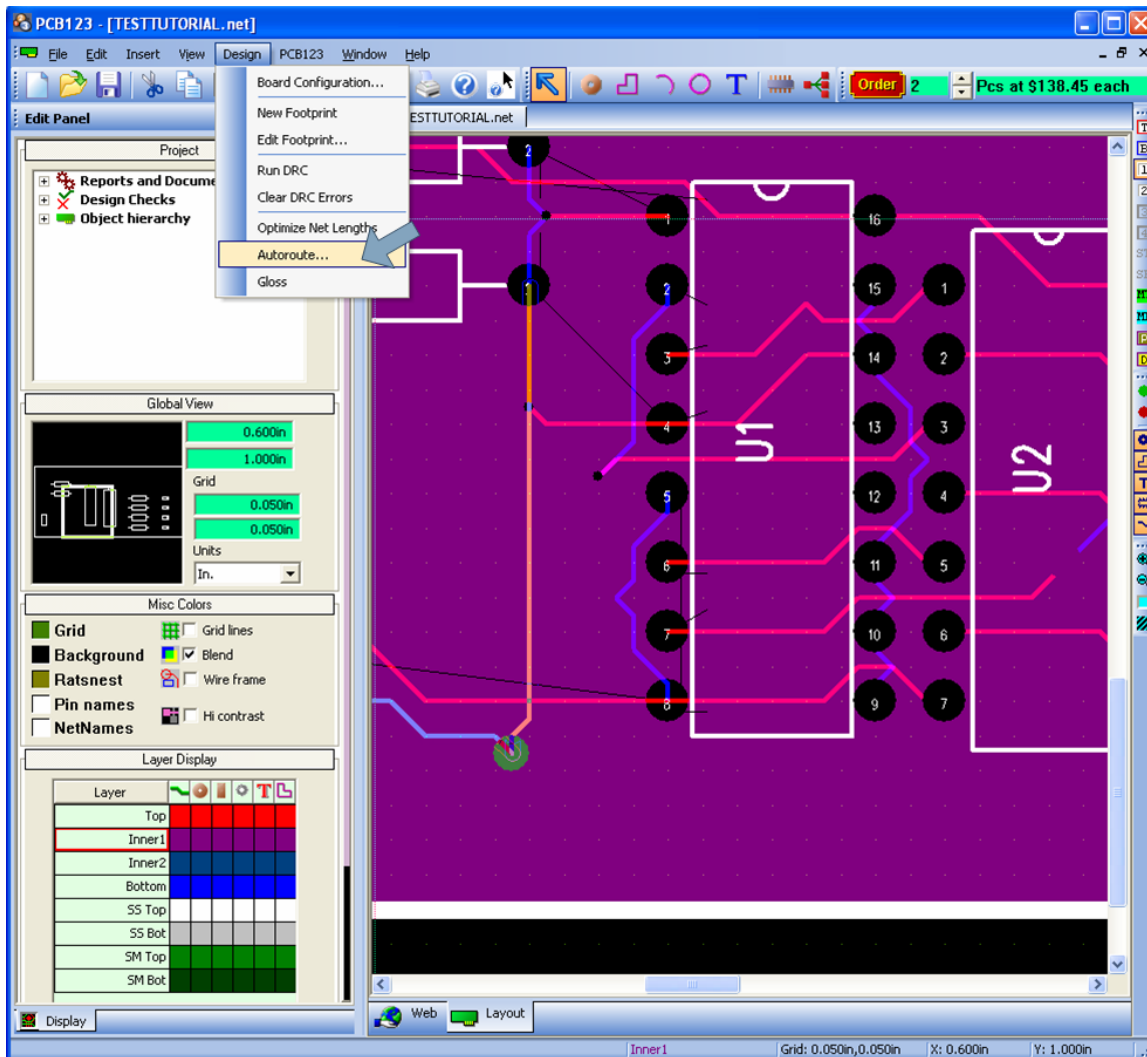


Figure 15: Autorouting Your Routes

[Top of Page](#)

At the end of autorouting, if there are any connections the autorouter was not able to route they can be routed manually. To do this you may have to move traces and vias. The following are some common operations for inserting and modifying routes and vias.

Manual Routing

As you mouseover the route, the cursor will dynamically change, and the route will highlight.

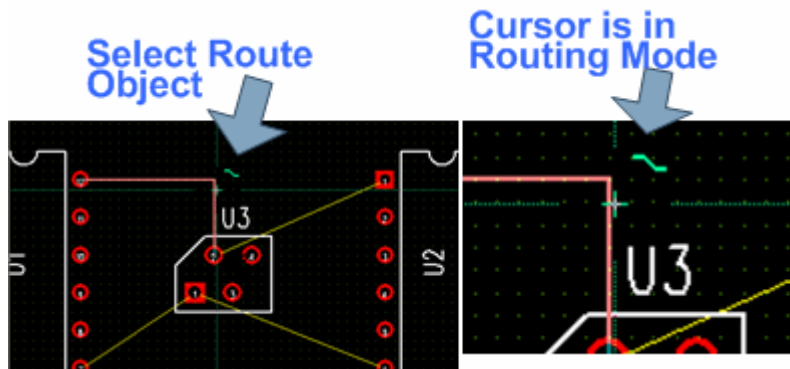


Figure 16: Cursor Modes: Routing Mode

Click on the route to re-arrange it.

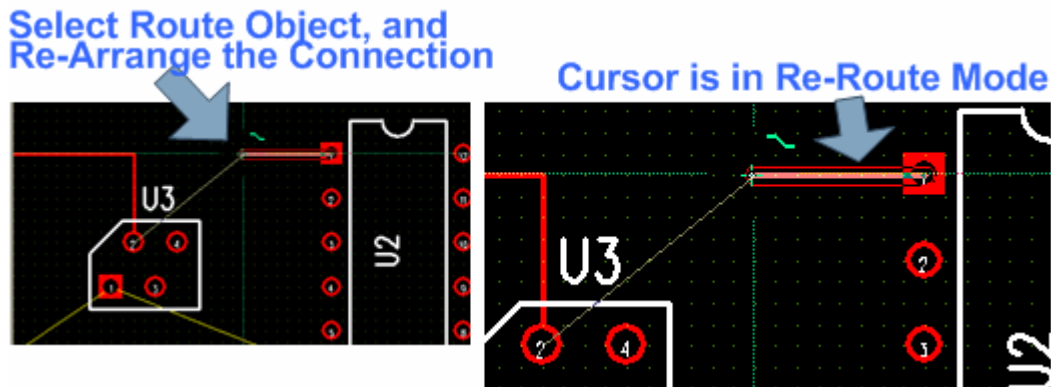


Figure 17: Cursor Modes: Re-Route Mode

A spacing violation is when you drag a route too close to another one.

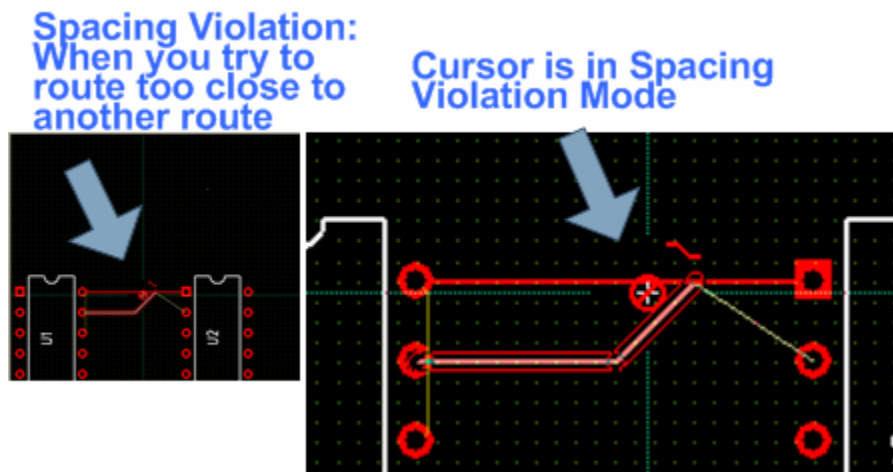


Figure 18: Cursor Modes: Spacing Violation Mode

Manual Routing Tips

Move corners: Just click on a corner of an object and drag it to where you want. You can fold consecutive corners over each other to optimize segments of routes away, giving the required clearance around an object.

Move segment: Press the left-mouse button down and drag a segment to where you want it. Release the left-mouse button to drop the segment.

Reroute segment: By single-clicking on the middle $\frac{3}{4}$ of a segment, you can dynamically 'ripup' and 'reroute' that segment. The benefits of this operation will soon become apparent as it lets you dynamically create in free form mode, including adding vias by simply changing layers as you go. A via is always inserted at the last corner clicked, not necessarily under the cursor. Rerouting also allows you to walk a route backwards, removing segments as you go. It is a very quick way to edit and modify routing.

Routing: clicking on an unrouted (ratline, connection, airline, etc.) you simply create a connection by clicking corners. If you change layers in the midst of creating a connection, using the **Z-key**, a via (a connection between layers) will be inserted at the last corner made.

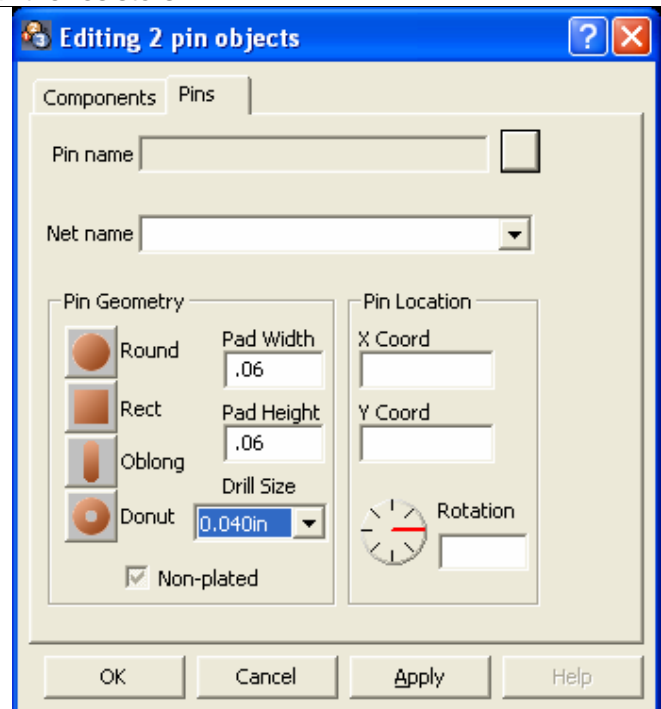
Quick-Complete: Doubleclicking on a board outline or route will cause it to complete the connection, or complete the board enclosure.

Segments: Segments snap to 45 degree increments unless you hold down the **Ctrl-key**. The **Ctrl-key** is an override and will allow you to route at any angle as long as it is held down.

Changing Pad/ Hole Sizes of a Placed Part

You may need to change the pad and/ or hole sizes of parts placed in your design. In our example design we will change the pad and hole sizes for all the resistors.

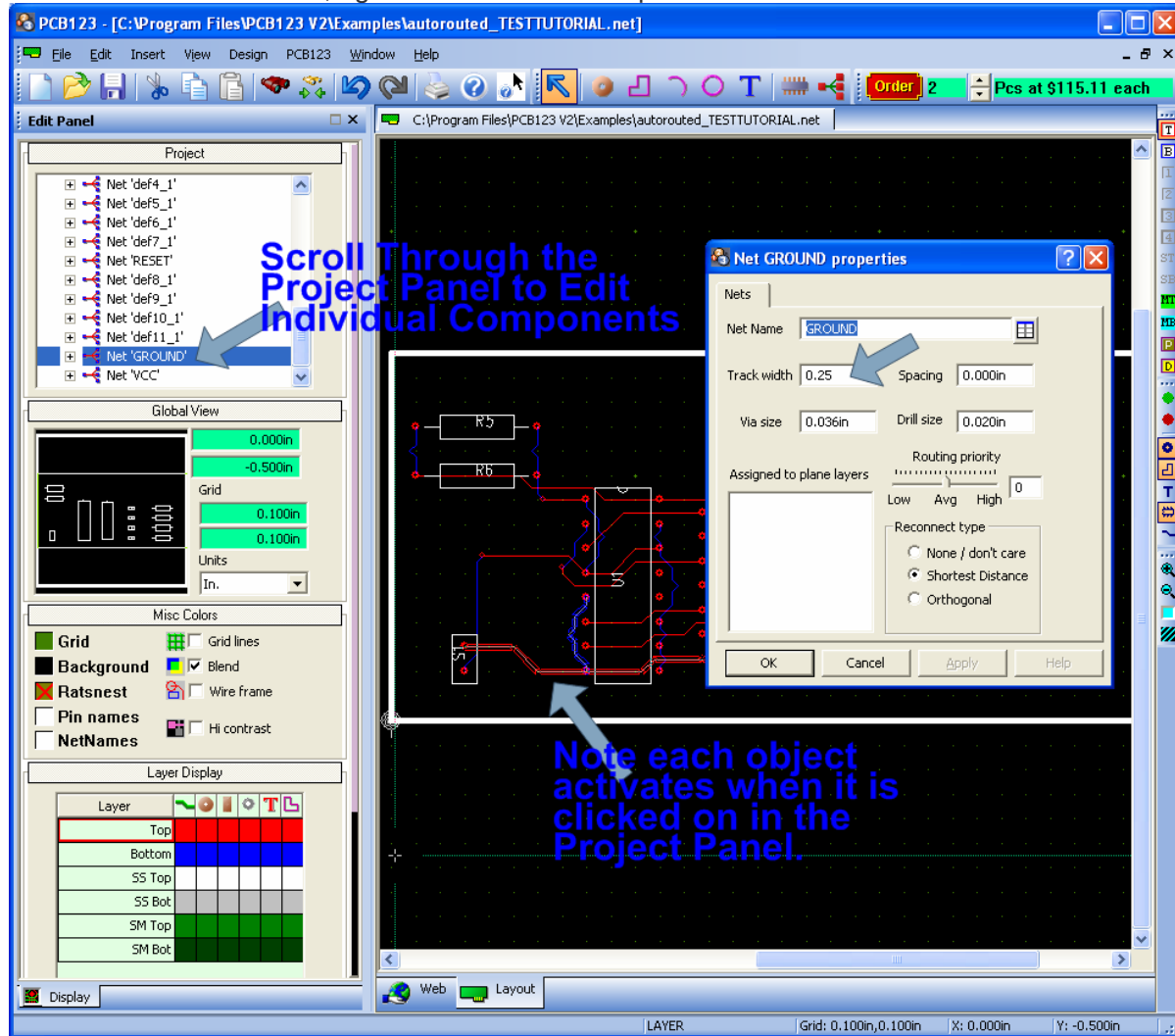
1. Select R1.
2. Right-click on R1 to bring up the **Properties** dialog.
3. Click the Pins tab.
4. Select the Pad shape: Round
5. Set the Pad Width and Pad Height to: .060", and the Drill size to .040"



Changing Trace Widths of VCC and Ground

Next, we will change the trace widths of our VCC and Ground nets. Follow these steps:

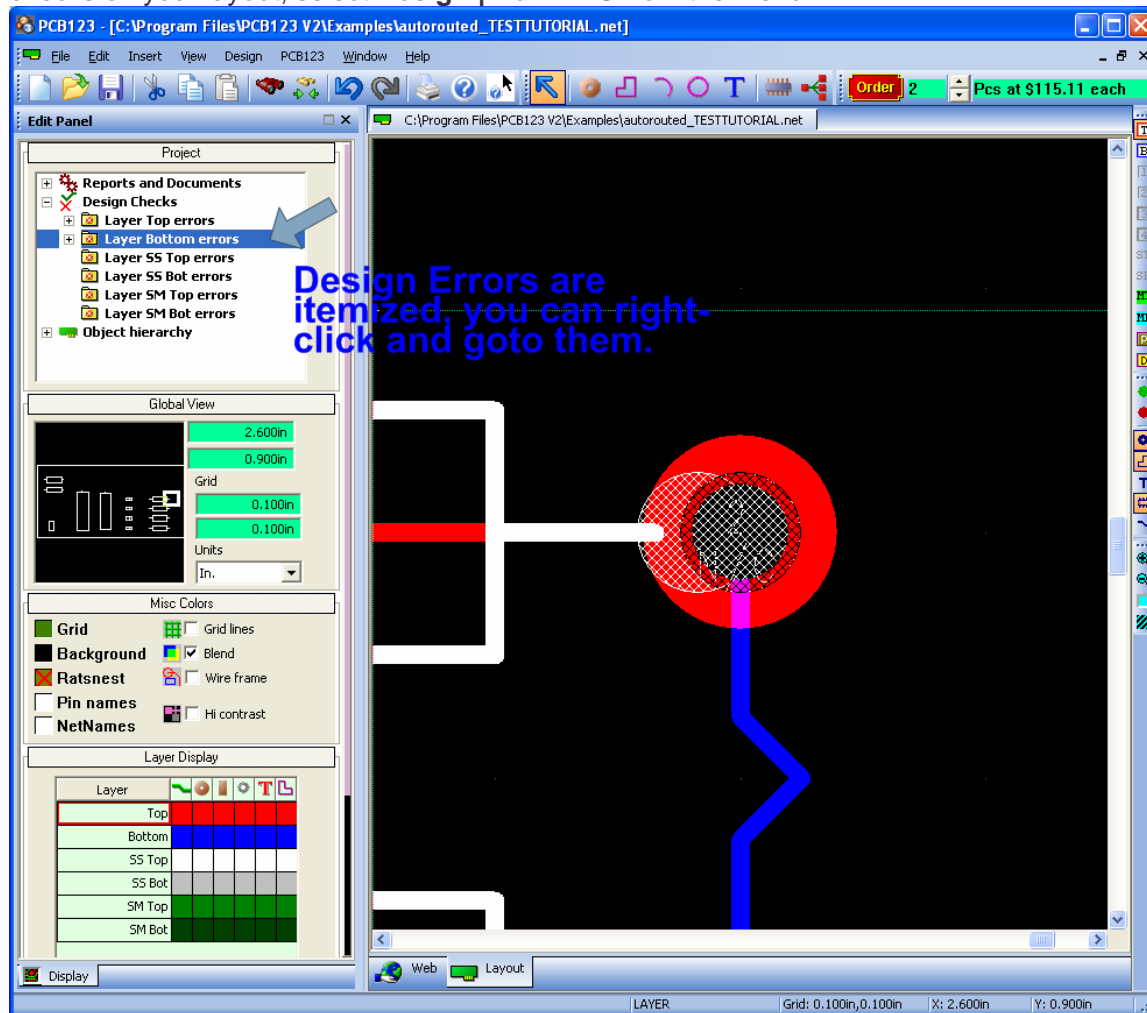
1. Scroll through the Project Panel and note how any object selected highlights on the screen. You can edit their properties from the Project Panel.
2. Select the Ground, right-click and select Properties.



3. Change the trace width to 0.25".
4. Now repeat the steps and select the VCC net instead of the Ground.

Design Checks

The design checks are used to point out any possible manufacturing problems in your design. If any errors are flagged they will need to be fixed before boards can be ordered. To run Design checks on your layout, select **Design | Run DRC** from the menu.



After you edit the errors, you can perform a **Design | Clear DRC Errors**, then repeat the process.

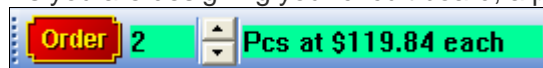
Saving The Board and Giving It a Name

After completing the board, select **File | Save** from the menu. For new artwork, as in this case, you will be prompted for a file name. Enter the file name, the **.123** extension will be added.

Order Information

Pricing

As you are designing your circuit board, a pricing toolbar is always on your screen.



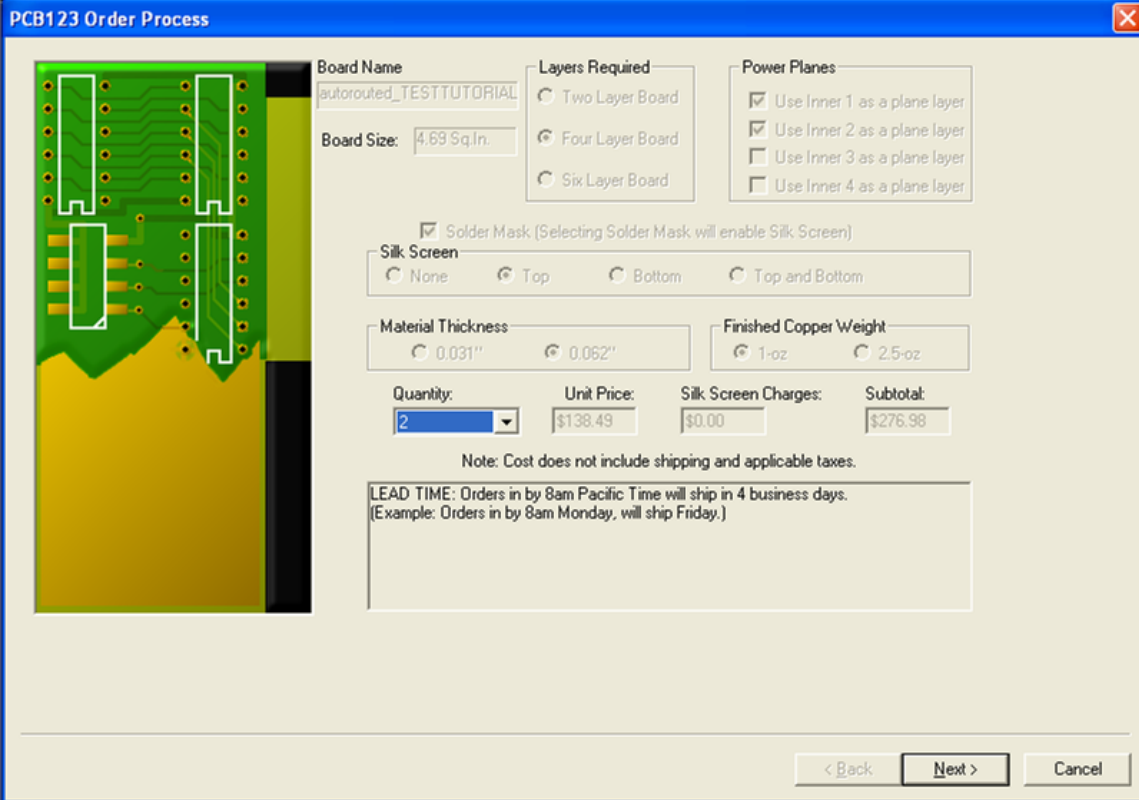
You will instantly see if any changes you have made will affect the board price.
Board Cost Factors are:

- size of the board – (the smaller the board, the lower the cost)
- soldermask
- backside silkscreen
- quantity

You can modify your pricing by changing your menu options: **Design | Board Configuration**.

Product Quantity Adjustment

Once you select **Order**, you will see the screen below:



The screenshot shows the 'PCB123 Order Process' window. On the left is a 3D visualization of a green PCB with yellow components. The main area contains the following fields and options:

- Board Name:** autorouted_TESTTUTORIAL
- Board Size:** 4.69 Sq.In.
- Layers Required:**
 - ☐ Two Layer Board
 - ☒ Four Layer Board
 - ☐ Six Layer Board
- Power Planes:**
 - ☒ Use Inner 1 as a plane layer
 - ☒ Use Inner 2 as a plane layer
 - ☐ Use Inner 3 as a plane layer
 - ☐ Use Inner 4 as a plane layer
- ☒ Solder Mask (Selecting Solder Mask will enable Silk Screen)
- Silk Screen:**
 - ☐ None
 - ☒ Top
 - ☐ Bottom
 - ☐ Top and Bottom
- Material Thickness:**
 - ☐ 0.031"
 - ☒ 0.062"
- Finished Copper Weight:**
 - ☒ 1-oz
 - ☐ 2.5-oz
- Quantity:** 2 (dropdown menu)
- Unit Price:** \$138.49
- Silk Screen Charges:** \$0.00
- Subtotal:** \$276.98

Note: Cost does not include shipping and applicable taxes.

LEAD TIME: Orders in by 8am Pacific Time will ship in 4 business days.
(Example: Orders in by 8am Monday, will ship Friday.)

At the bottom are buttons: < Back, Next >, and Cancel.

On this screen you may change the quantity. If you modify quantities you will be able to see the price change immediately. If you need to change the rest of the board configuration, go to the menu and select **Design | Board Configuration**.

We offer the following:

- 2 and 4 layer boards, with or without soldermask.
- Silkscreen legend is available only with soldermask.
- Top silkscreen is included with soldermask, back silkscreen is an additional charge.
- 2 ounce OR .031 material are also available for 2 layer boards with soldermask.


These options are run every Wednesday at no extra charge.

Login Screen

Below are the screens you will be taken through to process your order. Fill in your information and you will be on your way.

Once your info is entered, the majority will be populated automatically. You will need to enter your payment information each time, as we do not store this information for your security.

PCB123 Order Process


Let Me In! 

Enter your e-mail address:

☒ I am a new customer.

☐ I am a returning customer,
and my password is:


☒ Remember me on this computer?

 VeriSign
SECURE SITE
PCB123

Forgot your password? [Click here](#)

Account Screen

PCB123 Order Process

Account Information 

NOTE: If you already have an account with us, please login at the [login page](#).

Your Personal Details

First Name: required

Last Name: required

E-Mail Address: required

Company Details

Company Name: required

Your Credit Card Billing Address

Country: required

Street Address: required

address:

City: required


State: required

-- Select Above -- required


Zip Code: required

Shipping Address Information

PCB123 Order Process

Ordering Online is Easy! 

Please Select Shipping Priority [1 shipping | 2 payment | 3 confirmation | 4 finished!]

 FedEx or UPS, your choice, same price! \$15 UPS 2nd Day Air


Ship To	Billing Address
<input type="radio"/>	Your Name Company Address Street City State, Zip


Ship To	Alternate Shipping Address
Your Address Book has currently no entries!	

[1 shipping | 2 payment | 3 confirmation | 4 finished!]

Payment Information

PCB123 Order Process

Payment Method 

 **SECURE SITE**
PCB123 [1 shipping | 2 payment | 3 confirmation | 4 finished!]

Methods

Credit Card

Credit Card Holder:


Credit Card Type: Visa

Credit Card Number:

Credit Card Expire Date: January / 2004

Billing Address	Shipping Address
Your Name Company Address Street City State, Zip	Your Name Company Address Street City State, Zip

Shipping Method

 FedEx or UPS, your choice, same price! (UPS 1DA) \$25.00USD

Confirm Order

PCB123 Order Process

I'm Ready to Purchase!

PCB123

Please Confirm Your Order

[1 shipping | 2 payment | 3 confirmation | 4 finished!]

Board Name:

Your File Name

Board Size (x,y):

3.08,1.53

Number of Layers:

4

Solder Mask:

Yes

Material Thickness:

0.062 in

Number of Holes:

57

Board Size (square inches):

4.69

Power Planes:

2

Silk Screen:

Top

Copper Weight:

1-oz

Quantity	Unit Price	Total
2	138.55	\$277.10USD

Silk Charges:

\$0.00USD

Build-Total:

\$277.10USD

25% Off First Time Order Discount:

\$69.28USD

Sub-Total:

\$207.83USD

Tax 0.00%:

\$0.00USD

UPS 1DA Shipping:

\$25.00USD

Total:

\$232.83USD

Billing Address

Delivery Address

Checkout Confirmation

PCB123 Order Process

Your Checkout Has Been Processed!

PCB123

Congratulations, your order has been successfully processed!

You can check your board status or view your order history at any time by going to your 'My Account' page on the Website and by clicking on 'History'.

You are almost done!

Click on the next button to send your board files.

Thank you for your order!

[1 shipping | 2 payment | 3 confirmation | 4 finished!]

Next >

File Transfer Screen



Transferring the following PCB123 files:

TUTORIAL

FILE TRANSFER COMPLETED!

Your PCB123 file transfer has been completed!

You will receive a confirmation e-mail shortly. If you do not receive this e-mail within 15 minutes please contact sales@pcb123.com and reference your Order No.: 133SMI35

Thank you,

PCB123 Team

Finish

[Top of Page](#)

Shipping Methods

As you fill out your information you will be asked for your choice of ship method. Once your order is completed your boards will be shipped. UPS orders will have the tracking number sent to the supplied e-mail address the evening the boards ship.

You will have the convenience of checking the status of your order online. During the build process your order will show as "Processing". Once the order is completed and shipped, the status will change to "Shipped".

Order History



Order No. **43HJUK02** Order Status: **Pending**

Board Name:	Your File Name	Number of Holes:	57
Board Size (x,y):	3.08,1.53	Board Size (square inches):	4.69
Number of Layers:	4	Power Planes:	2
Solder Mask:	Yes	Silk Screen:	Top
Material Thickness:	0.062 in	Copper Weight:	1-oz
Qty.	Unit Price	Total	
2	138.55	\$277.10USD	