

PCB123 Version 2 Tutorial

From Netlist to PCB in 10 Minutes



A tutorial designing a circuit board with: 4 layers, 2 outer routing layers, internal power and ground planes; and is 3.7 by 2 inches in size.

Example Board





Table of Contents

PCB123 Version 2 Tutorial From Netlist to PCB in 10 Minutes	1
Installation of PCB123 Software	2
Tutorial	3
Importing the Netlist	3
Moving Components into the Board Outline	7
Component Selection	9
Component Placement	10
Assign Power and Ground Nets to Plane Layers	11
Edit Connection Properties	11
Route the Board	14
Manual Routing vs. Autorouting	14
Autorouting	14
Manual Routing	16
You're Finished!	17
Appendix A: Anatomy of PCB123 Layout Interface	18
Standard Menu	18
Insert Toolbar	19
Projects Panel	20
Layers Toolbar	20
Selectability Toolbar	20
View Tools	21
Global View Panel	21
Misc. Colors Panel	21
Layers Panel	22
Bottom Panel	22
Board Configuration Dialogue	23
Appendix B: Cursor Modes	25
Appendix C: Tips	30
Object Selection	30
"Modeless" Functionality	30
Mouseover Focusing	30
Left-Mouse Click	30
Ctrl-Left-mouse Click	30
Area Selection	30
Object Creation	30
Escape Mode	30
Undo/Redo Mode	30
Editing the Board Outline	31
Hot Keys	31
Appendix D: Glossary	32

Installation of PCB123 Software

1. Extract the zip file into a directory of your choosing, and make sure you check the **Use Folder Names** button.
2. Navigate to the **Tutorial** directory (under your installation directory) and launch the program by clicking on **PCB123.exe**.
3. Depending on your screen resolution and preferences, you may have to make some adjustments to the toolbar positions.

Tutorial

For the tutorial, we are going to import two schematic pages by importing two netlists, **gates1.net** and **gates2.net**. These schematic pages come from PCB123 schematic packages.

Importing the Netlist

Import a **parts** list and **netlist** from a multi-page schematic.

1. Click on the **Netlist Browser** button, and navigate to the **Designs** folder (Figure 1).

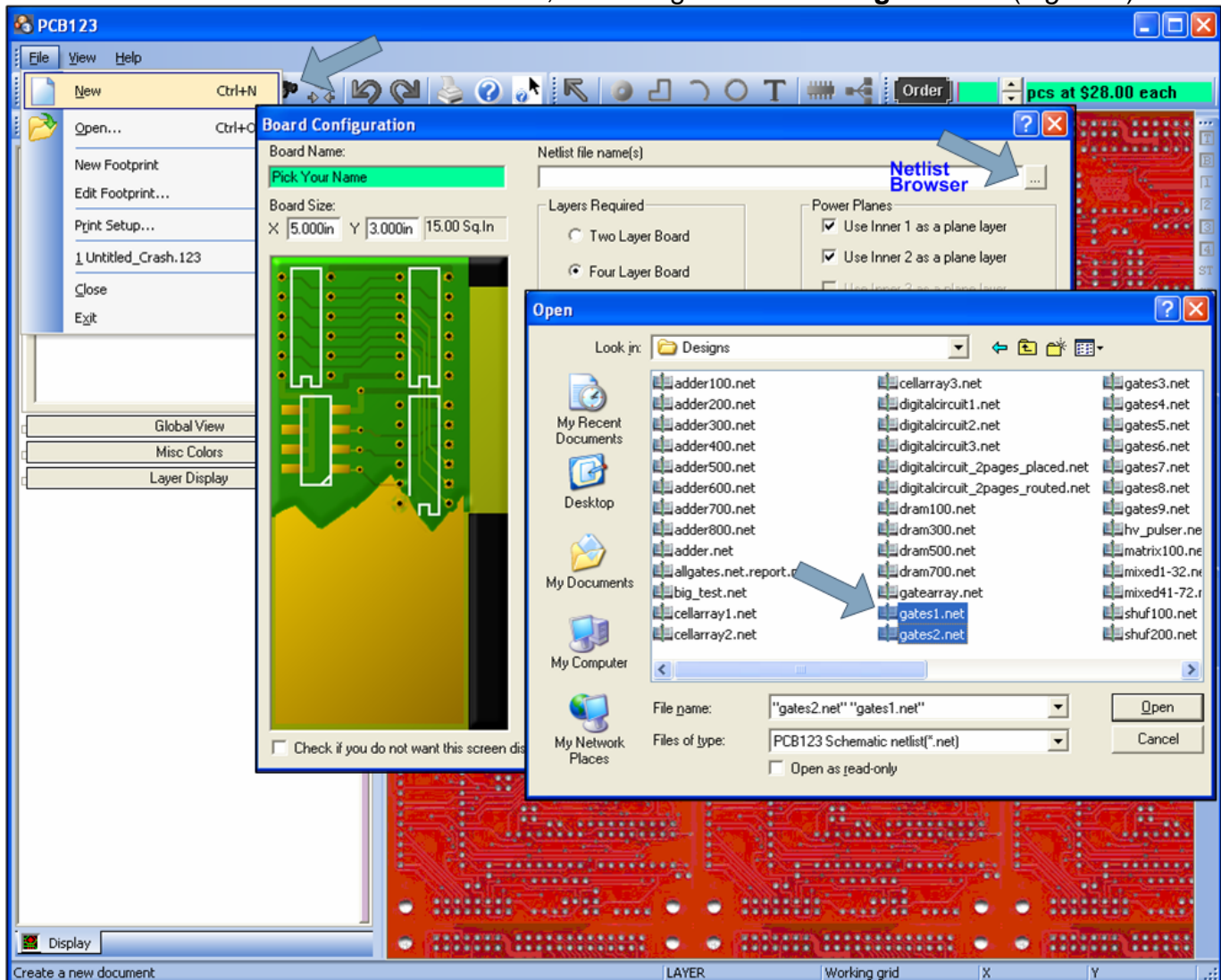
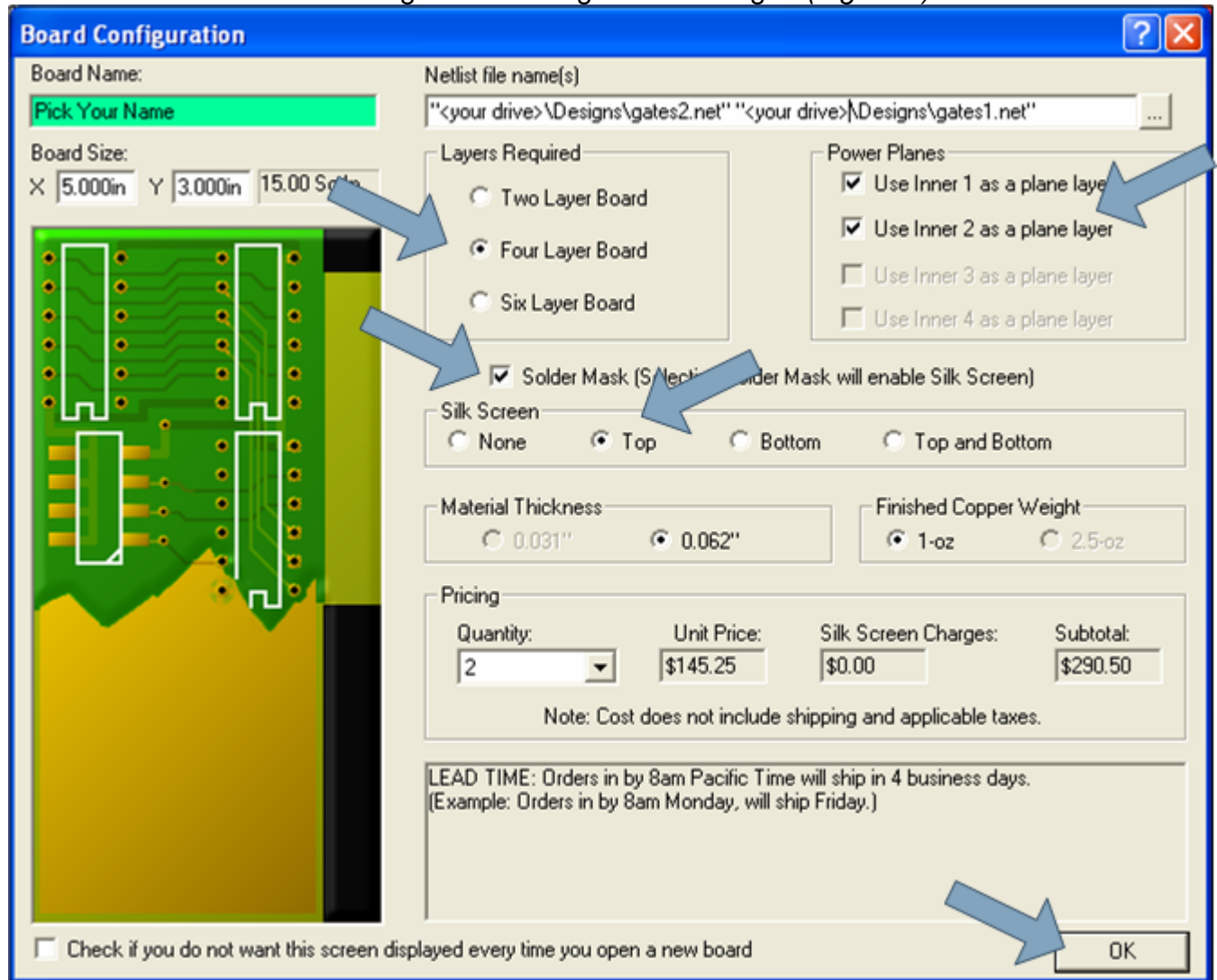


Figure 1: Importing Netlists

2. You should see several **.net** files in the file selection box. Select files **gates1.net** and **gates2.net** (by holding down the **shift** or **Ctrl**-key while selecting these files) and then click **Open**.

3. You should see the resulting Board Configuration dialogue (*Figure 2*).



Board Configuration

Board Name:

Netlist file name(s):

Board Size: X Y 15.00 Solder

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

Pricing

Quantity:	Unit Price:	Silk Screen Charges:	Subtotal:
<input type="text" value="2"/>	<input type="text" value="\$145.25"/>	<input type="text" value="\$0.00"/>	<input type="text" value="\$290.50"/>

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

OK

Figure 2: Board Configuration Dialogue

- Click Layers Required: **Four Layer Board**
- Click Power Planes: **Use Inner 1**
- Click Power Planes: **Use Inner 2**
- Click **Solder Mask**
- Click Silk Screen: **Top**
- Click **OK**

4. **(Note:** you will only see this window if the software doesn't already have the existing part.) The first thing you should see is the library parts browser popup (*Figure 3*) asking you to select a package for **DIP14**.

(The schematic makes reference to a **DIP14** package but the libraries do not contain a generic **DIP14**. The browser displays anything that closely matches **DIP14**.)

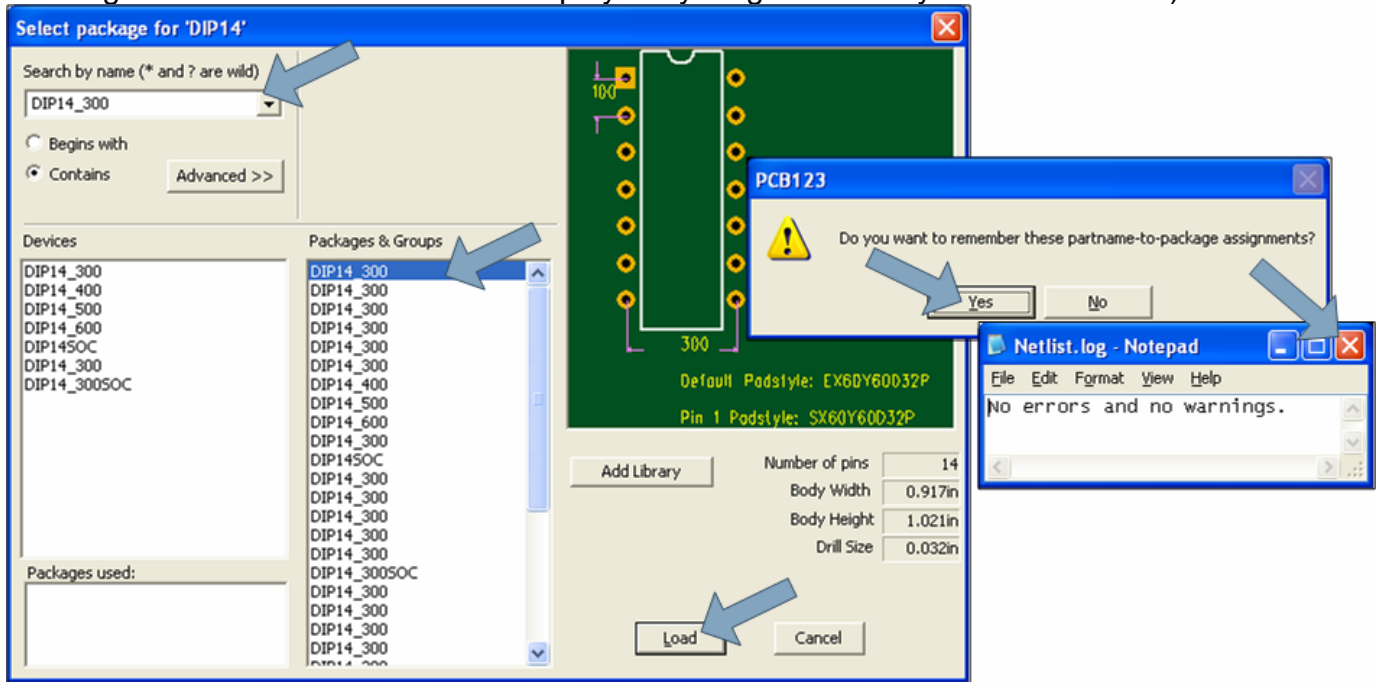


Figure 3: Select Package

5. Select the **DIP14_300** package and then click the **Load** button.
6. The program will then ask you if you want it to remember the partname-to-package assignments (that **DIP14** maps to **DIP14_300**). Click **Yes** to enter an alias in the file named **Global.alias**. Future imports referencing **DIP14** will automatically resolve to **DIP14_300**.
7. The program will do an error check. After a moment, a text window will appear, displaying either:
 - Message "No errors found"
 - A detailed list of errors, if any were detected
8. Close the text window.

8. Perform a **Zoom Entire** by hitting the **Home** key (*Figure 4*) on your keyboard (make sure **Num-Lock** key is off).
9. Save your file, **File | Save As**, to protect your project.

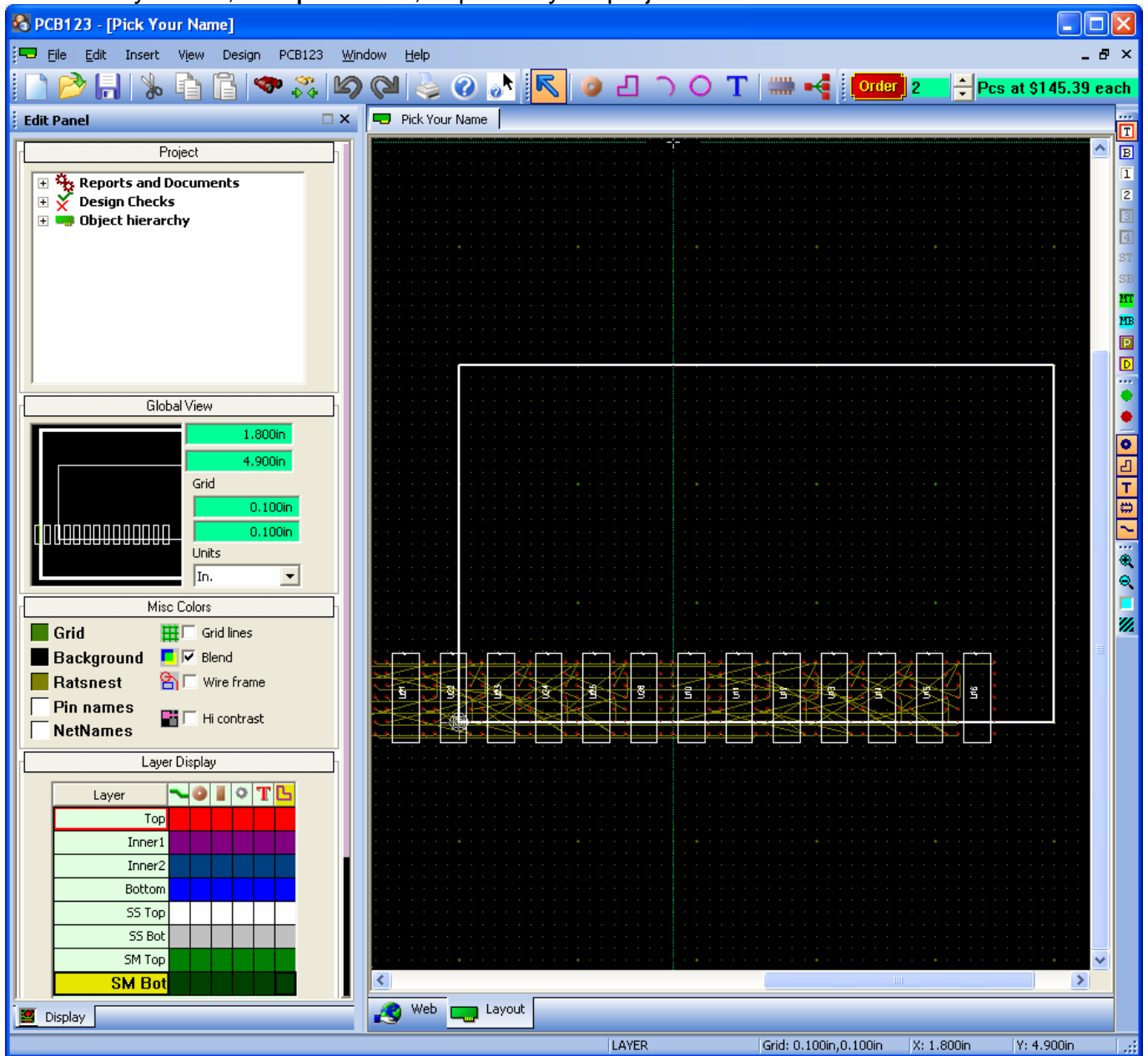



Figure 4: Initially Loaded Components with Zoom Entire

Moving Components into the Board Outline

If you have components outside the board outline, you can grab the objects and center them inside it (Figure 5).

1. Click the “All on”  button, which lets you select everything, and drag them to within the board outline.

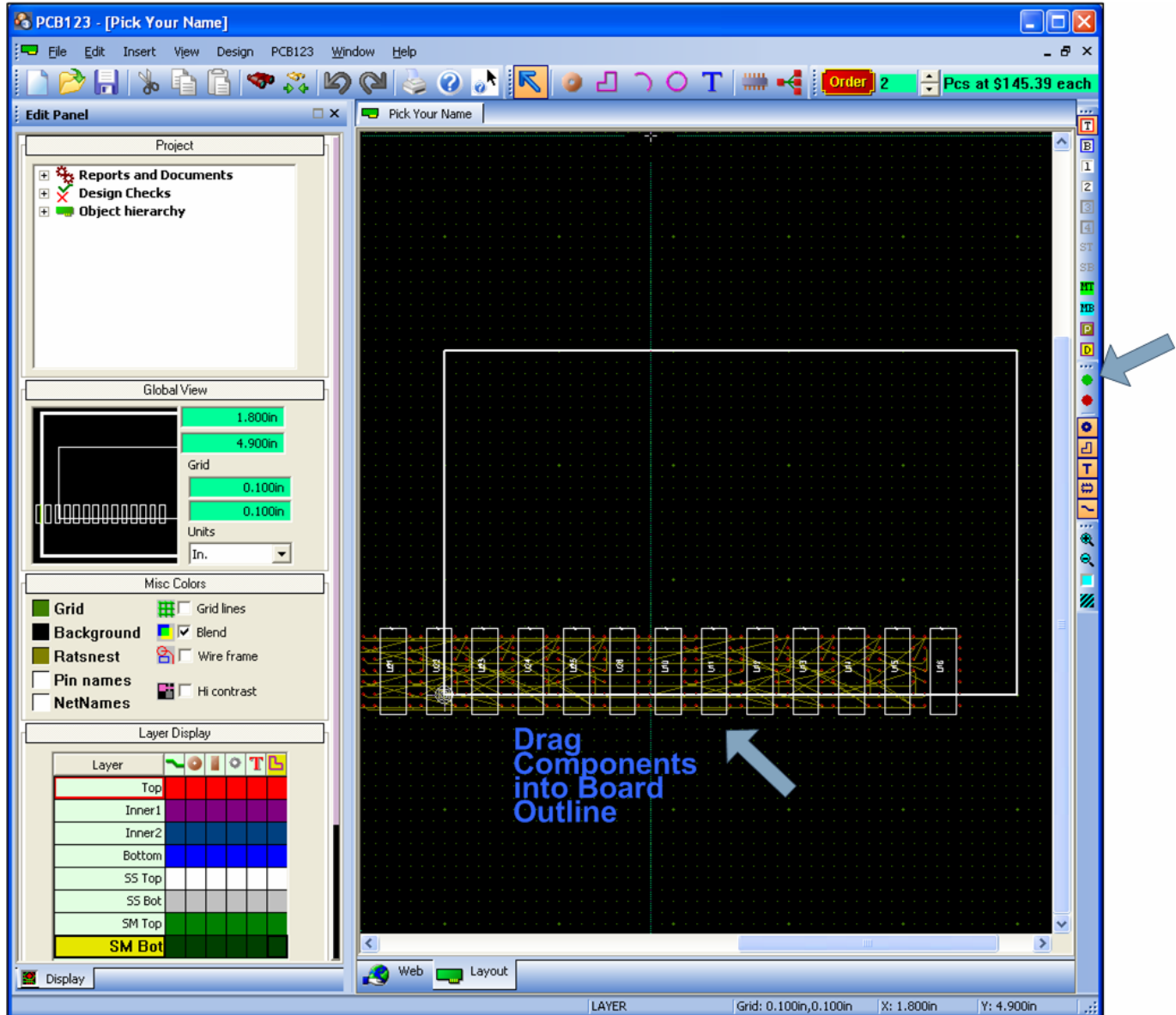




Figure 5: Moving Components into Board Outline

2. The **modeless** behavior of the software can be helpful, but until you are used to the precision picking of objects, it is best to *filter* the selection set. By default, all objects are active for selection.
Since we are getting ready to place the board, we don't care about selecting text or doing any routing.
3. Turn these off by toggling the  button and the  button on the right-hand toolbar.
4. To simplify the screen, we will first turn off the 'Ratsnest' by clicking on the Ratsnest

color palette (Figure 6) on the left side of the screen and selecting **Invisible** from the palette.

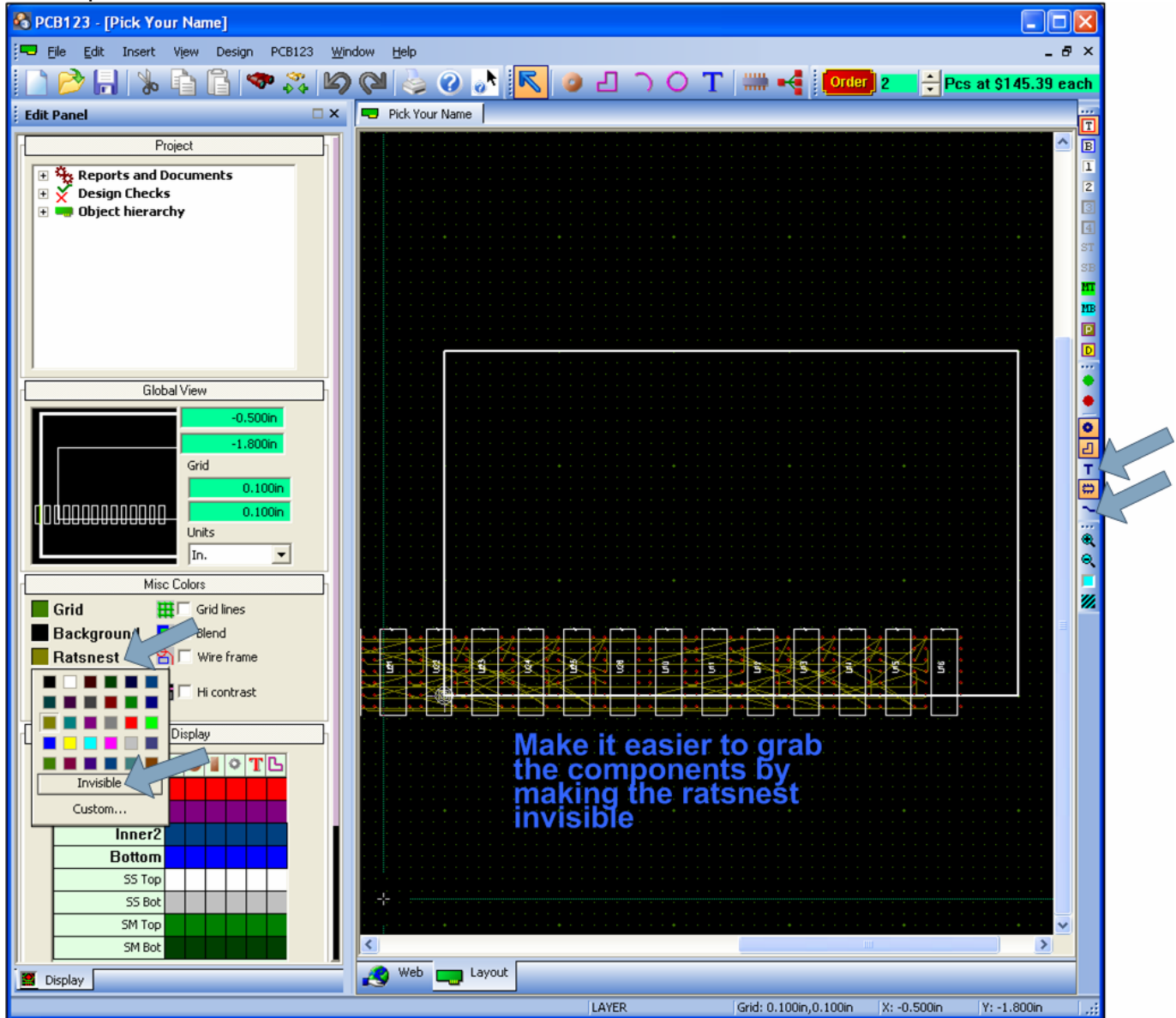


Figure 6: Setting Ratsnest Invisible for Moving Components

Component Selection

Now that you have set **Ratsnest** to **Invisible** mode, when you point at a component pin or outline you will notice the component becomes highlighted (*Figure 7*), telling you that it is the object under focus. Also note that the component becomes featured in the **Projects** Panel.

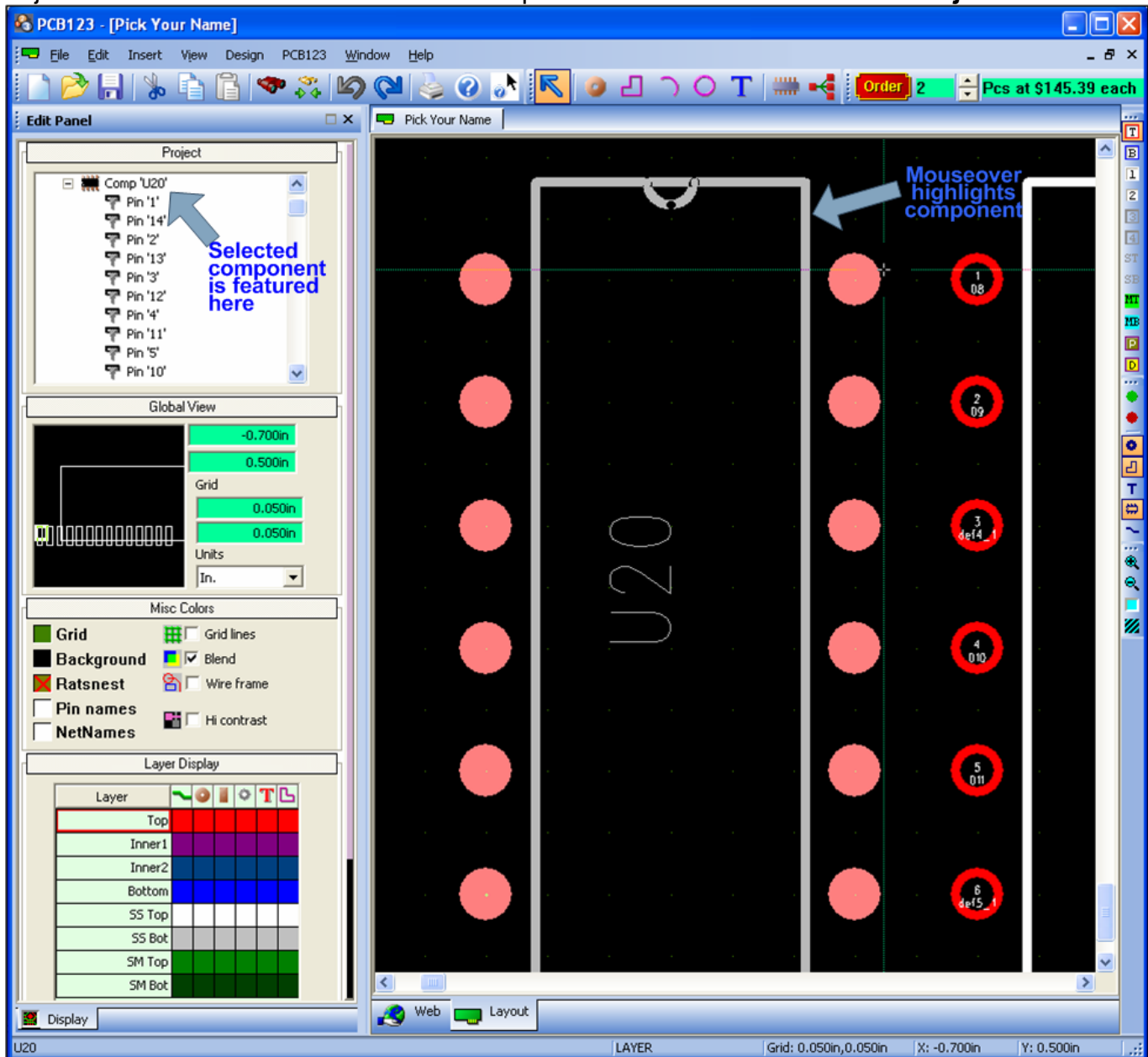


Figure 7: Demonstrating Component Selectability

Component Placement

You can arrange all of your components as you please. The goal in rearranging components on the board is to move and orient the components to insure the shortest possible routing of each net.

Note the placement of the numbered components (*Figure 8*). This is important to keep routes from getting trapped and not have enough space.

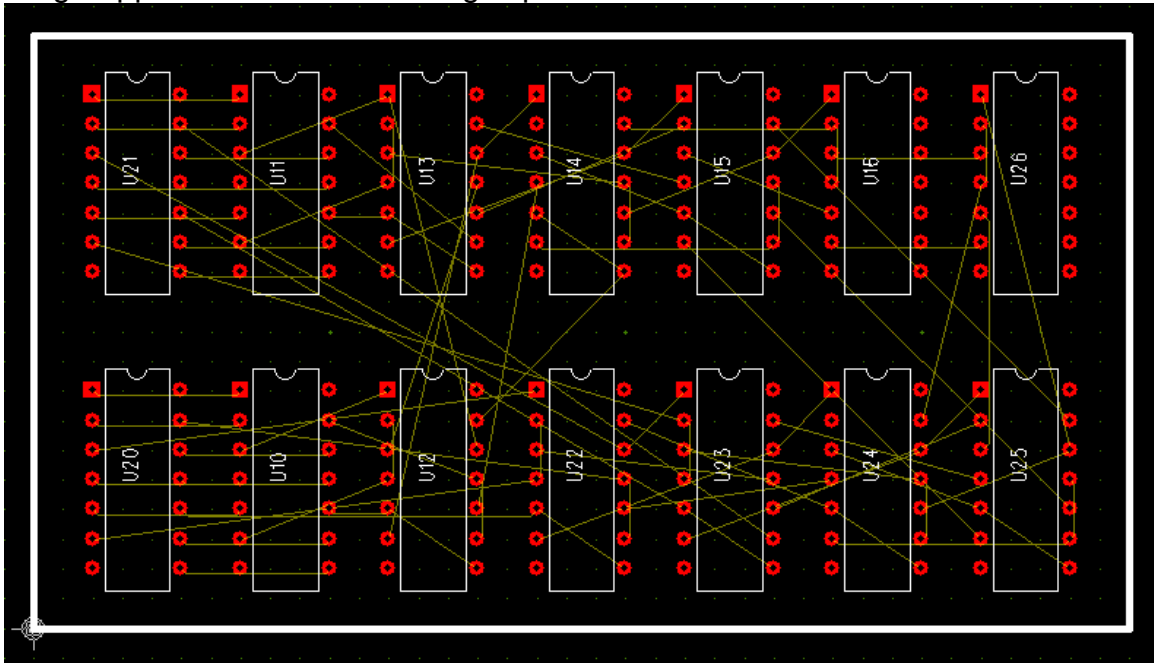


Figure 8: Example Component Placement

To display all the **net** connections, display the **Ratsnest** from the **Misc. Colors** panel. What you should see are wires showing you all the circuit connections between all your components in the netlist. As a general rule you should place the component with the most connections in the center of the board. Next, select the component with the next greatest connection and place it. You may have to rotate the component to get the right fit. Continue this process until all components are placed. To clear the **Ratsnest**, make it **Invisible** from the **Misc. Colors** panel.

Dynamic Component Movement: If you click on the component, it will attach to the cursor, and you can move it around.

Dynamic Nets Movement: You may also notice the **nets** going to the component dynamically change to reflect the shortest path for that net.

Dynamic Component Functions: By right-mouse clicking on the selected component, you can:

- Pop up menu options for the component.
- Rotate by pressing the 'R' key on your keyboard. Move the component to where you want and click again to drop it.

Assign Power and Ground Nets to Plane Layers

Edit Connection Properties

Zoom in to component **U20** on the board.

Using the Projects Panel (Figure 9)

1. Select **Net 'Ground'**, right-click on **Track**, select **Properties**, check "Assigned to Plane Layers" **Inner 1**.
2. Select **Net 'VCC'**, right-click on **Track**, select **Properties**, check "Assigned to Plane Layers" **Inner 2**.

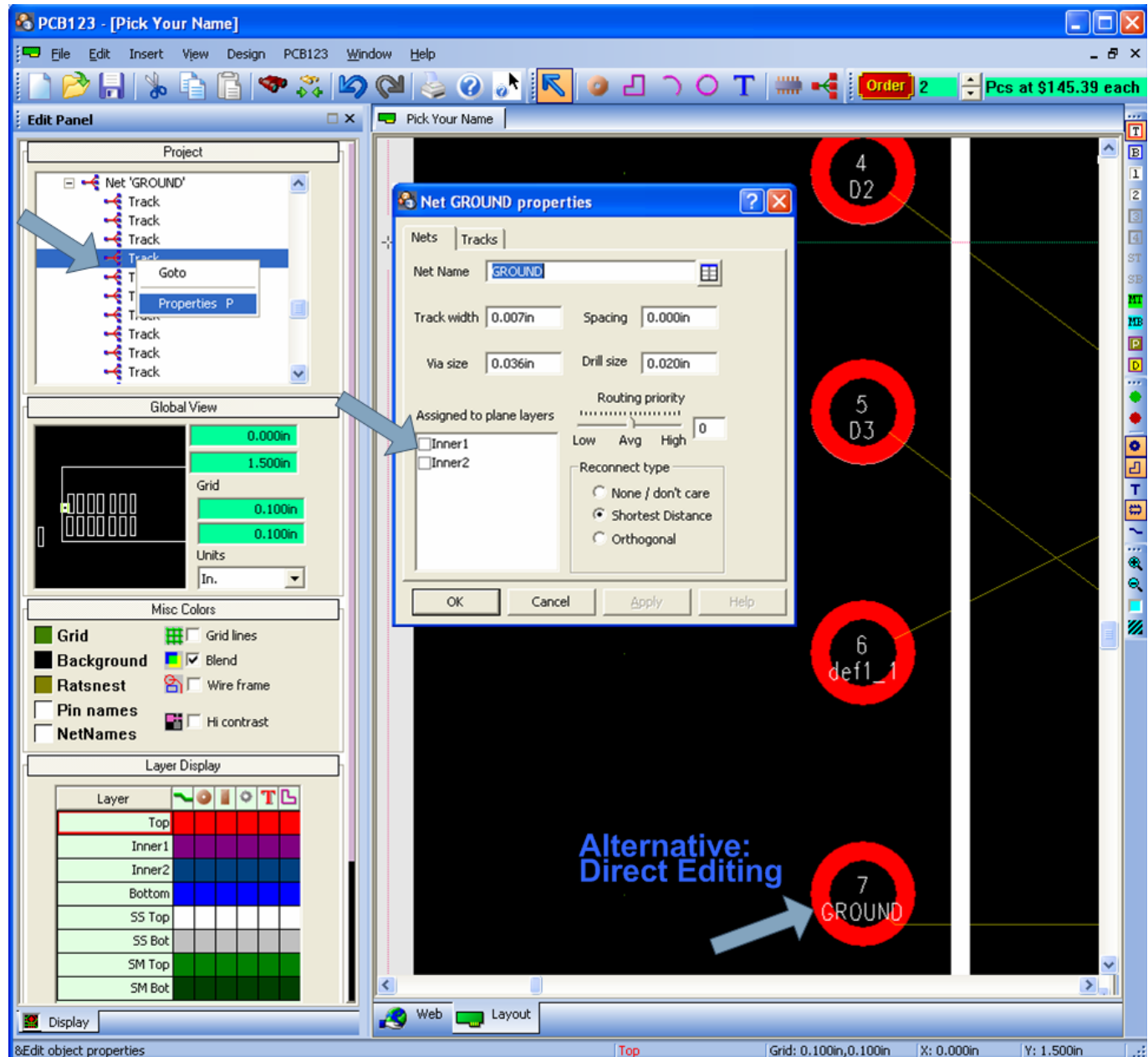





Figure 9: Edit Connection Properties

Alternative: Direct Editing of Connections (*Figure 9*)

1. Move your cursor over the yellow connection leaving **pin 7** of **U20**, right-mouse click and select **Properties** to edit the yellow connection's attributes.
2. You should see the same **Net Ground** properties window as in Figure 9.
(If you do not see the Net Ground properties window, press the **escape** key, and go back to the beginning of Section: *Assign Power and Ground Nets to Plane Layers*, on page 11, and make sure the **Ground** connection is highlighted when you right-mouse click.)
3. Check the box labeled **Inner1** and press **OK**.
4. Repeat this operation on the connection leaving **pin 14** of **U20**, and select **Inner2** for it.

Results of Editing Connections

You will notice that the connections disappear from the screen. Where did they go? To find out, change the active layer to another layer, **Inner1**, by either clicking on the  button on the right side toolbar, or the  row on the left panel. This will move you to the **Inner1** level of the board.

What you are looking at is a positive image of the ground plane and if you look at **pin 7** you should see the pad has turned into a thermal relief  (Figure 10).

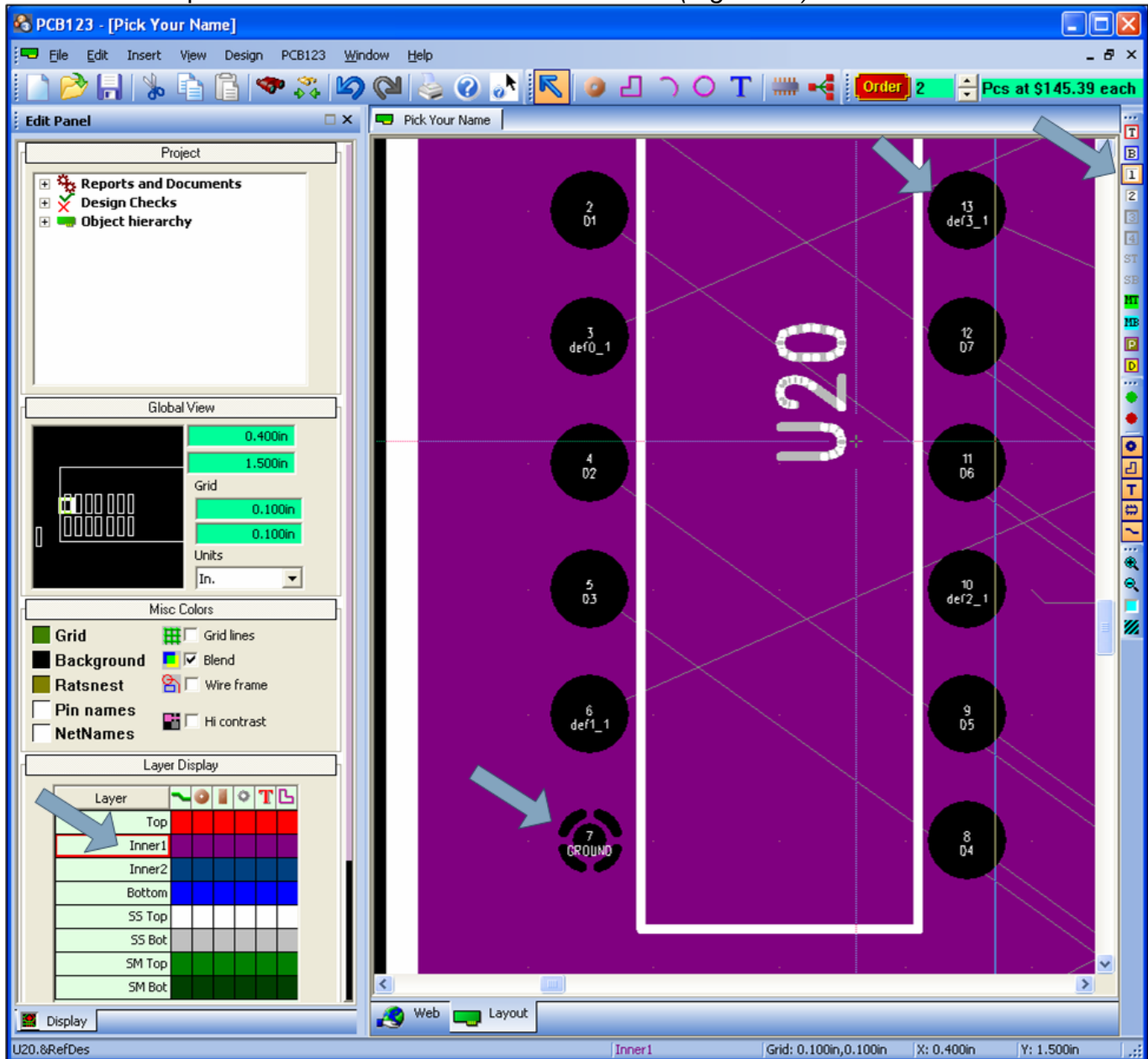


Figure 10: Thermal Relief and Different Layers

The connection is now tied to the *plane*.

Move back to the **Top** layer  or  for the next step, "Route the

Board”, page 14.

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 batch autorouter. The autorouter automatically routes all connections in the netlist.

Autorouting

This section demonstrates autorouting the board (*Figure 11*), 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 **Design | Autoroute** and watch it start generating **routes**.

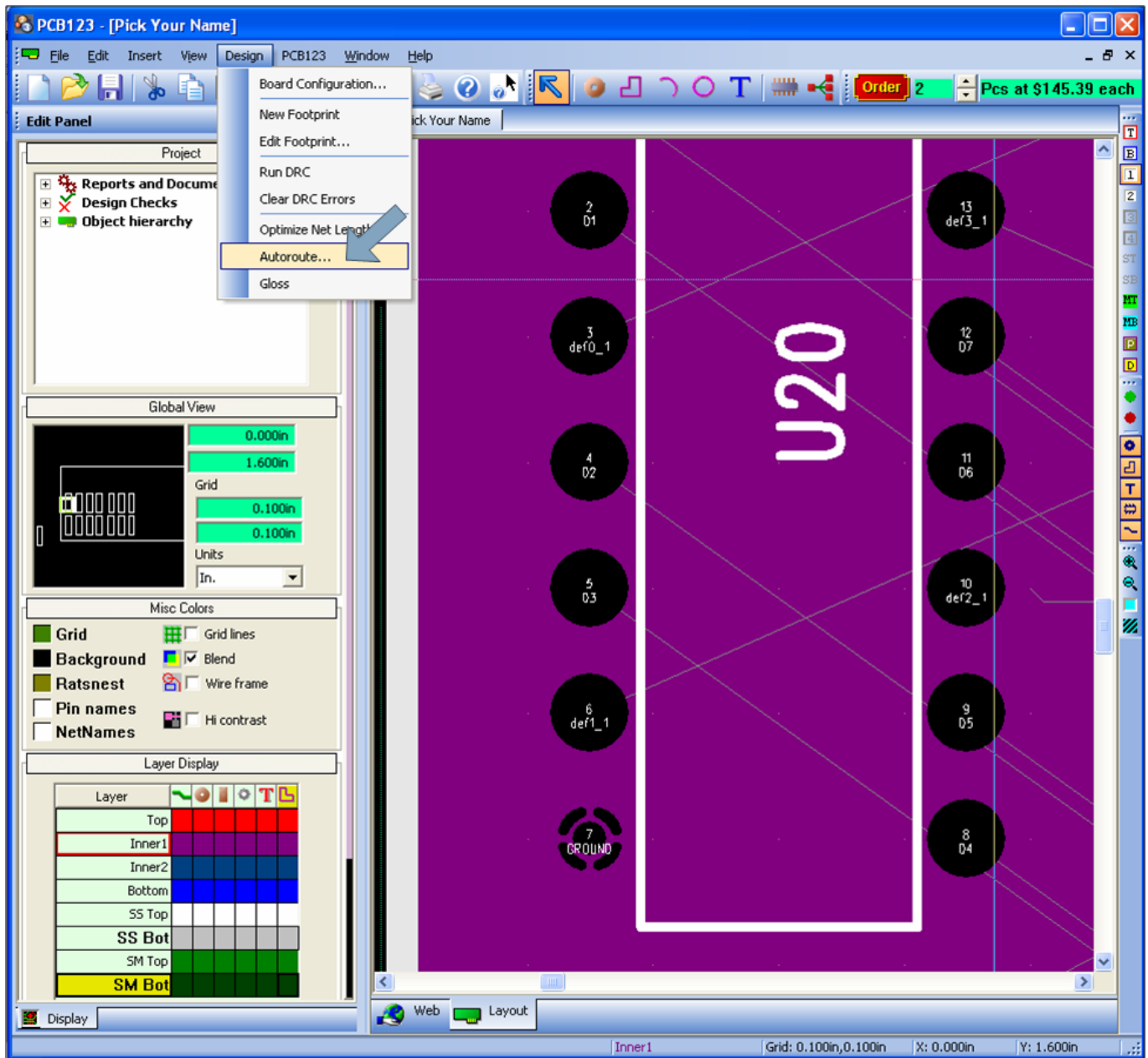


Figure 11: Autorouter

3. Stupid router! Now you have to clean up after it and dig in the toughies yourself. We suggest you zoom to a comfortable zoom factor, put on some nice music and actually have some fun manually routing the rest. You will find the manual routing tools quite enjoyable to work with.

Manual Routing

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

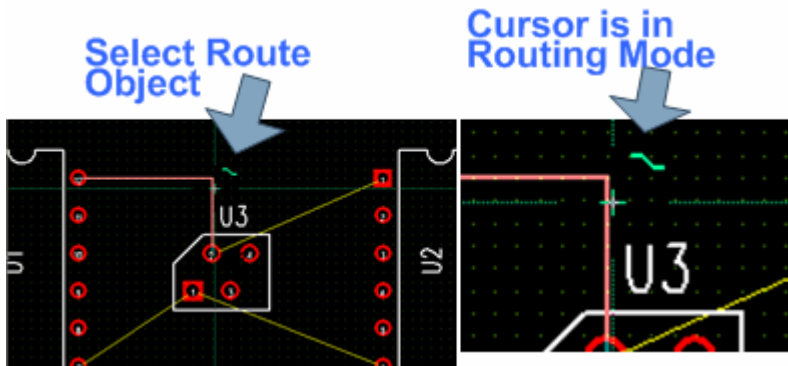


Figure 12: Cursor Modes: Routing Mode

Click on the route to re-arrange it.

Select Route Object, and Re-Arrange the Connection

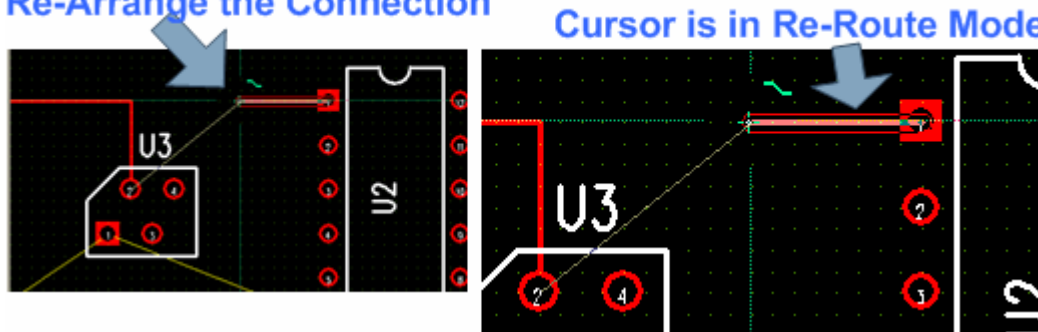


Figure 13: Cursor Modes: Re-Route Mode

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

Spacing Violation:
When you try to
route too close to
another route

Cursor is in Spacing
Violation Mode

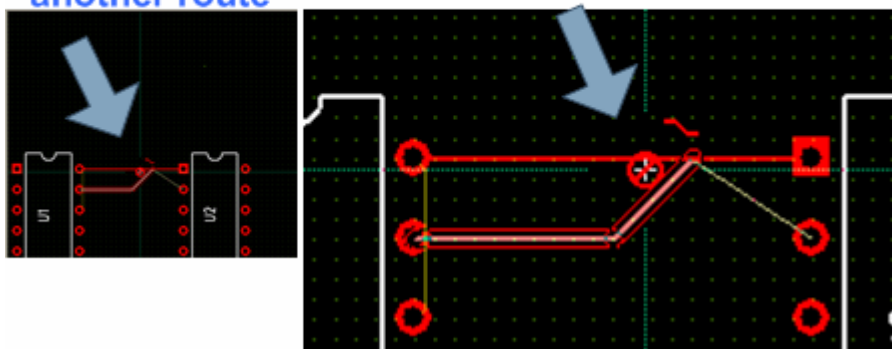


Figure 14: Cursor Modes: Spacing Violation Mode

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.

You're Finished!

Once you are done, we can manufacture your board!

Note: Make sure you save your latest work before sending it.

See typical example results at: <http://www.pcb123.com/pcb123manufacturing.php>.

We offer free assistance online or by phone:

(503) 829-9108 x236

(800) 228-8198 x236

Online Chat and Email form: <http://www.pcb123.com/pcb123supportfaq.php>

Appendix A: Anatomy of PCB123 Layout Interface

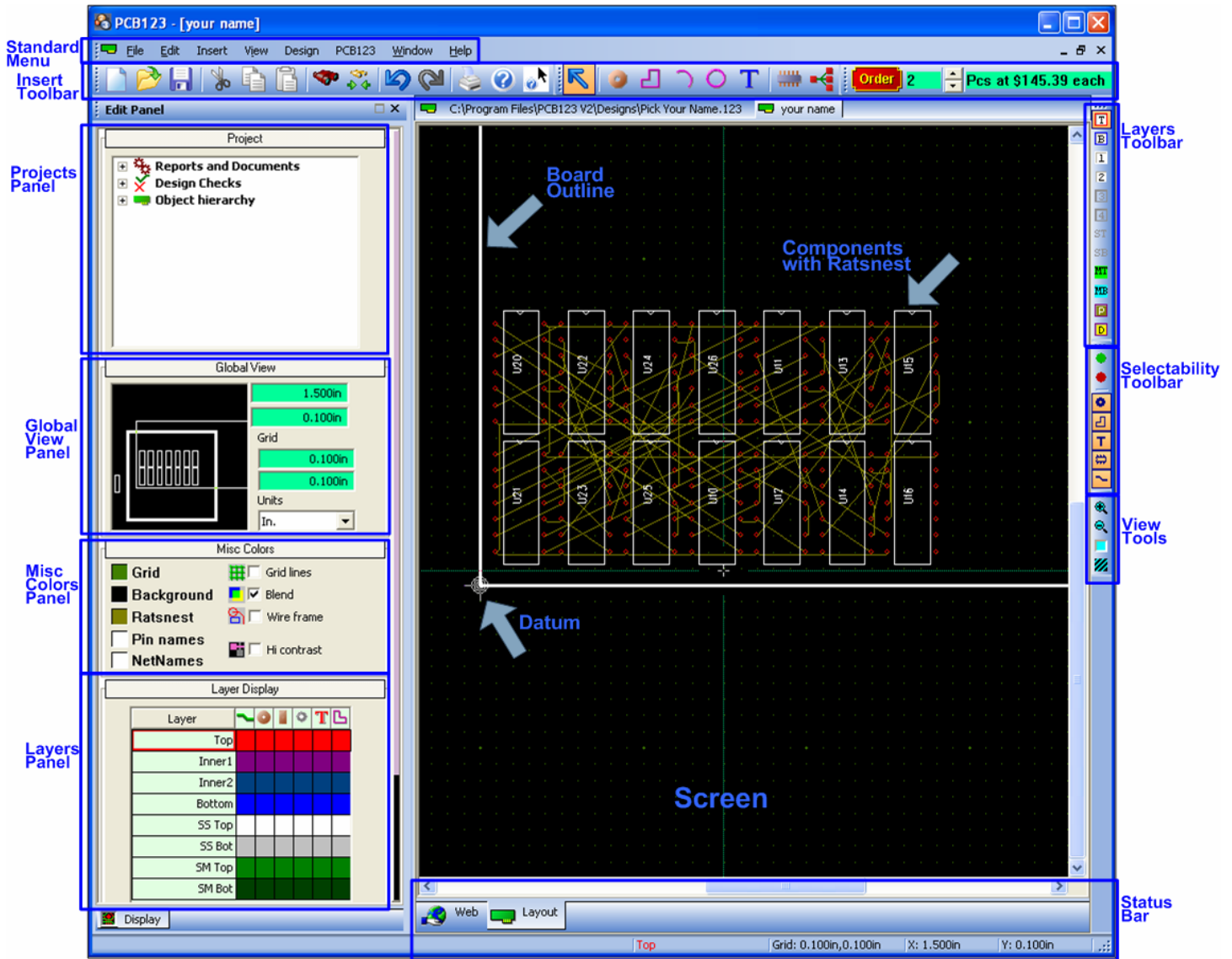
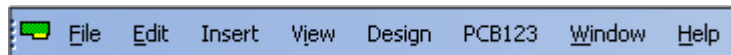


Figure 15: Anatomy of PCB123 Interface

Standard Menu



File	Edit	Insert	View
New	Undo	Select	Status Bar
Open	Redo	Add Pin	Edit Panel
Close	Cut	Add Polygon	Toolbars >
Save	Copy	Add Arc	Standard Toolbar
Save As	Paste	Add Circle	Add Object Toolbar
Export>	Delete	Add Text	PCB123 Toolbar
Simple Netlist	Find	Add Component	Layers Toolbar
BOM	Bring to Cursor	Add Connection	Selection Toolbar
Backup Now	Line Width		Zoom Toolbar
Print	Grid		Zoom >
Print Preview	Selection >		Zoom In PgUp
Print Setup	Select Pins		Zoom Out PgDn

File	Edit	Insert	View
Artwork Print Generate Gerber Files <Previous files> Exit	Select Polygons Select Text Select Components Select Tracks Select Anything Select Nothing Properties		Zoom All Home Pan Up UpArrow Pan Left LeftArrow Pan Right RightArrow Pan Down DownArrow Redraw End Center * Detach Cursor Tab Layer > Top 1 Bottom 2 Inner 1 3 Inner 2 4 Inner 3 5 Inner 4 6 Silkscreen Top Shft+1 Silkscreen Bottom Shft+2 Solder Mask Top Ctrl+1 Sold Mast Bottom Ctrl+2 Refresh All

Design	PCB123	Windows	Help
Board Configuration New Footprint Edit Footprint Run DRC (<i>Run Design Rule Check: looks for traces and spaces that are insufficient</i>) Clear DRC Errors Optimize Net Lengths Autoroute Gloss (<i>Cleans routes after using autorouter, but can create shorts.</i>)	Place an Order My Account PCB123 Website Report a Bug Download Library Parts	New Window Cascade Tile Arrange Icons	Help Topics Check for Updates About PCB123

Insert Toolbar

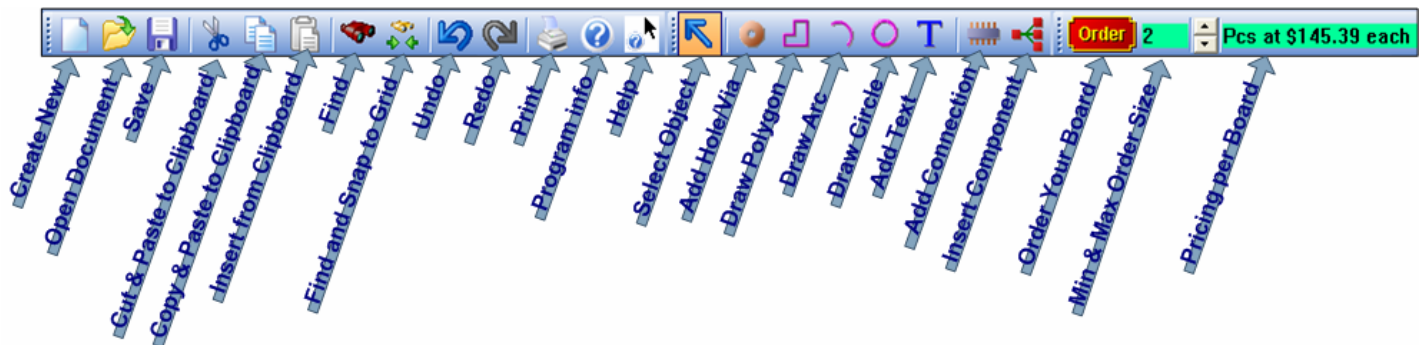


Figure 16: Insert Toolbar

Projects Panel

You can use the Projects Panel as a quick access tool to drill down through a hierarchical list of all objects on the board.

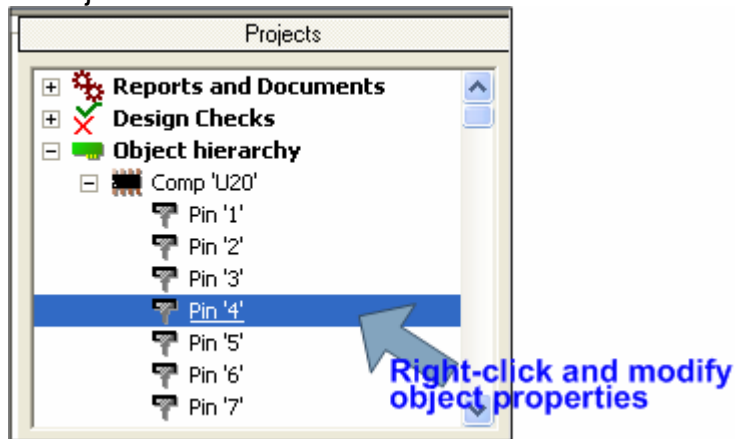


Figure 17: Projects Panel

Layers Toolbar



Figure 18: Layers Toolbar

Selectability Toolbar

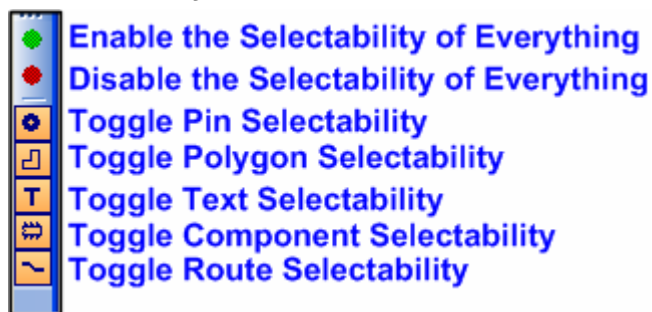


Figure 19: Selectability Toolbar

View Tools



Figure 20: View Tools

Global View Panel

This provides a birds-eye view of the Screen.

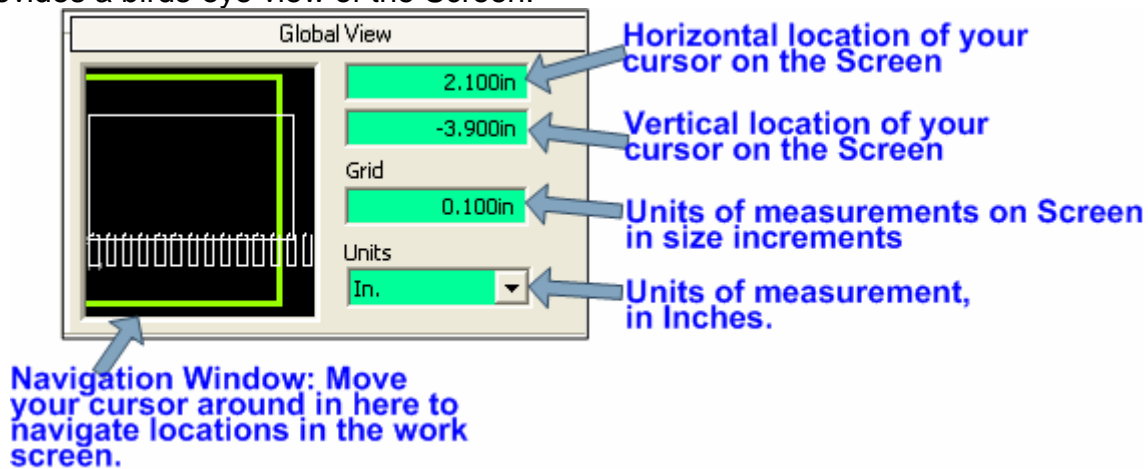


Figure 21: Global View Panel

Misc. Colors Panel

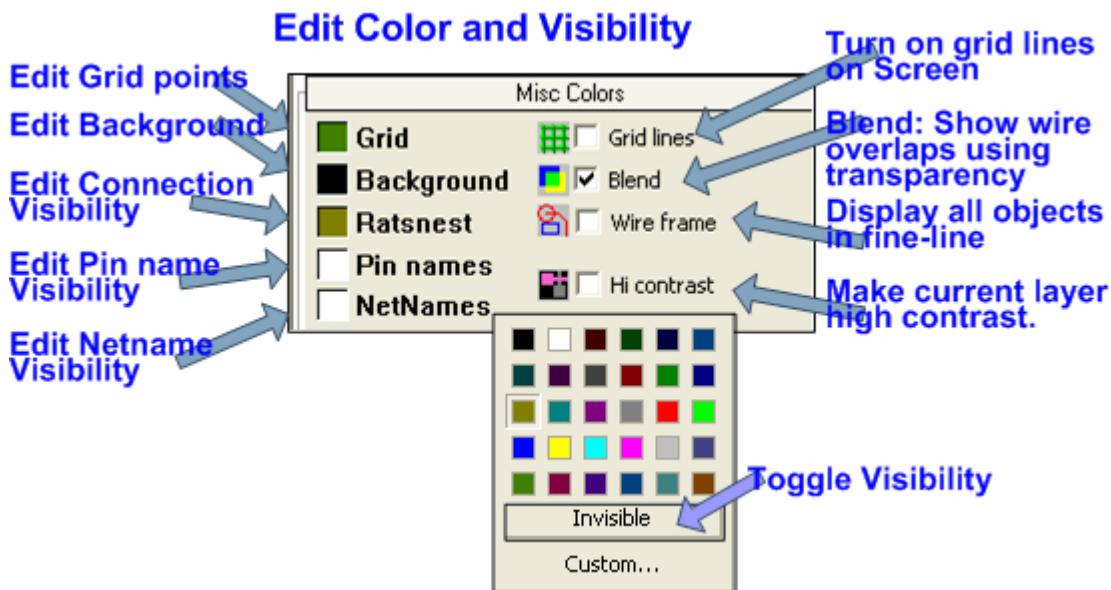


Figure 22: Misc. Colors Panel Explanation

Layers Panel

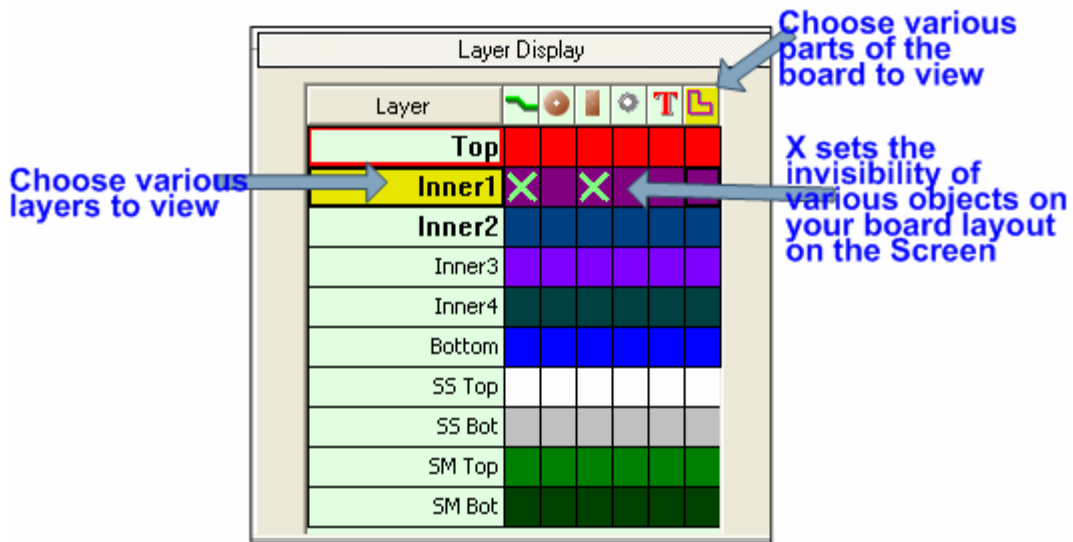


Figure 23: Layers Panel

Bottom Panel

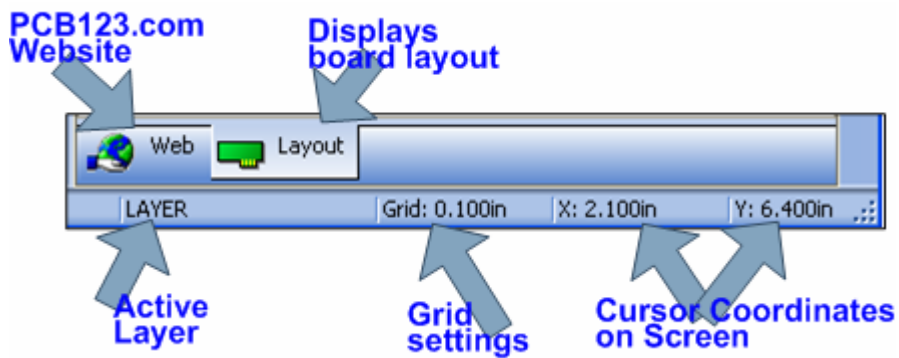
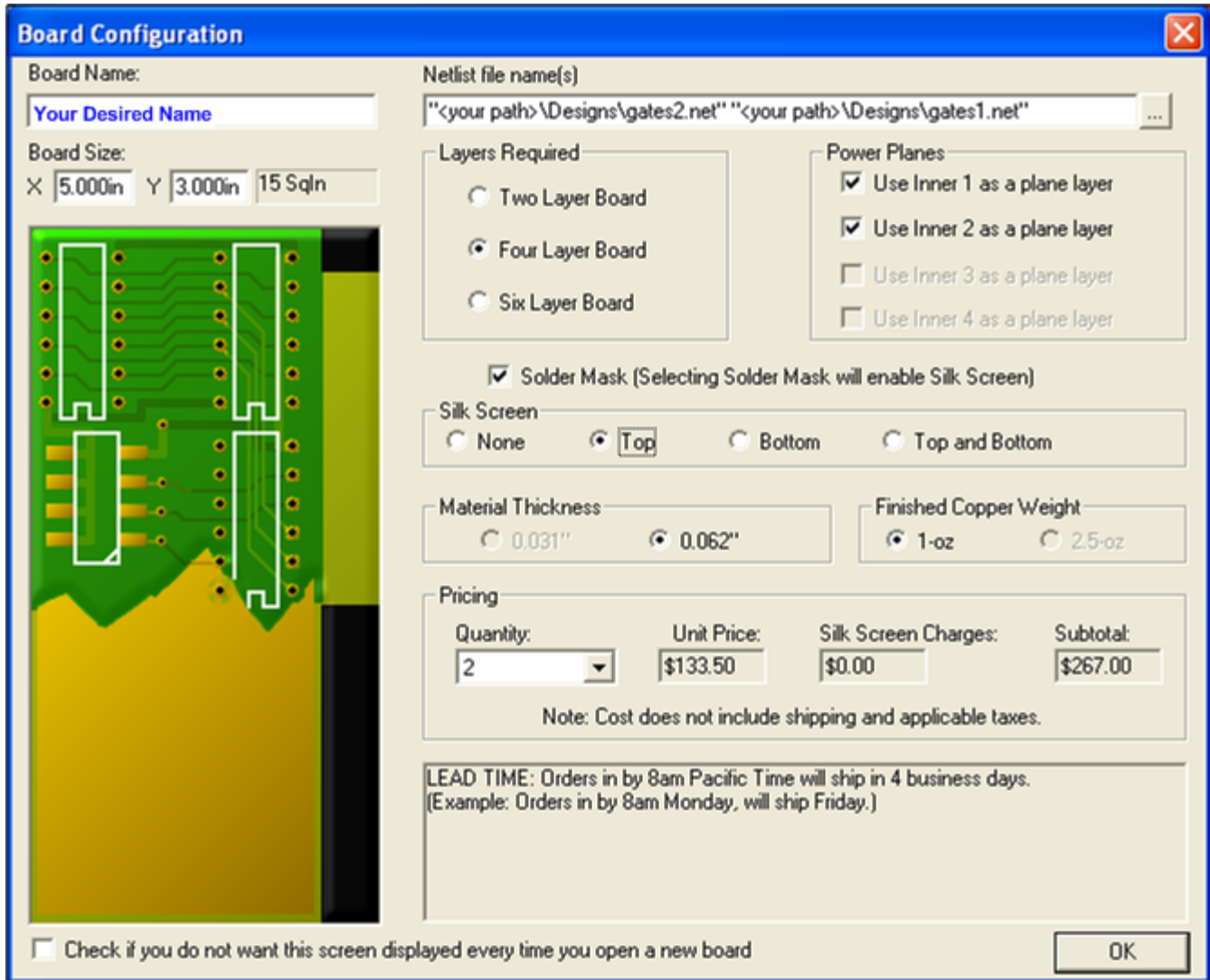


Figure 24: Bottom Panel

Board Configuration Dialogue



Board Configuration

Board Name:

Board Size: X Y 15 SqIn

Netlist file name(s)

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

Pricing

Quantity:	Unit Price:	Silk Screen Charges:	Subtotal:
<input type="text" value="2"/>	<input type="text" value="\$133.50"/>	<input type="text" value="\$0.00"/>	<input type="text" value="\$267.00"/>

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

OK

Figure 25: Board Configuration Dialogue Explanation

Board Name: Choose the name of your board.

Board Size: Any size up to:

- **2-Layer:** 12 x 18 inches
- **Multi-Layer:** 12 x 14 inches

Netlist Filenames: Browse to the necessary netlist files that you need.

Layers Required:

- **Two Layer:**
- **Four Layer:**
- **Six Layer:**

Power Planes: If no layers are checked, then the inner layers are plane layers. If they are checked, then they are routing layers (signal layers).

- **Inner 1:**
- **Inner 2:**
- **Inner 3:**



- Inner 4:

Sold Mask:

Silk Screen:

- Top:
- Bottom:
- Top and Bottom:

Material Thickness:

Finished Copper Weight:

Pricing:

- Quantity:
- Unit Price:
- Silk Screen Charges:
- Subtotal:
- Lead Time:
- Shipping and Taxes:

Appendix B: Cursor Modes

Cursor Selection Modes: Unselected Mode

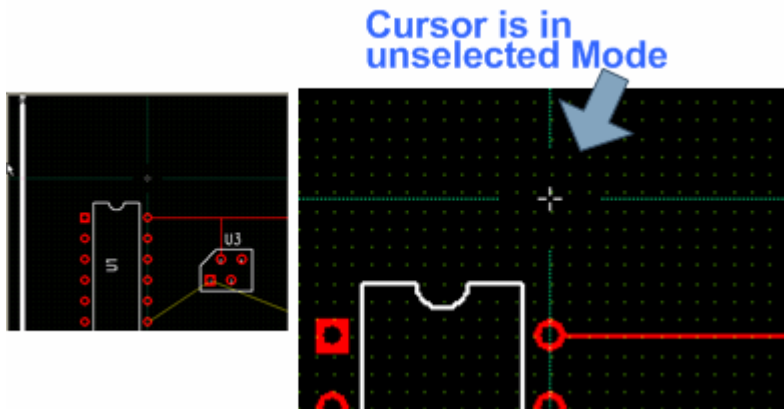


Figure 26: Cursor Modes: Unselected State

Cursor Selection Modes: Selection Mode

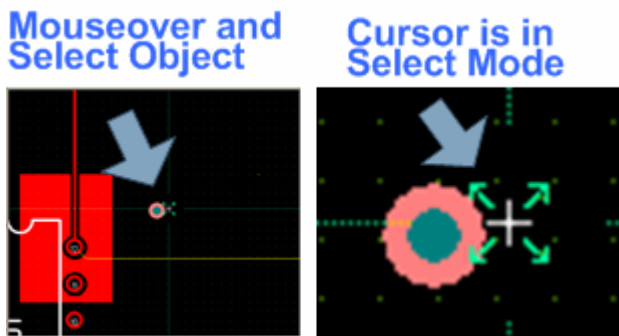


Figure 27: Cursor Modes: Select Mode

Cursor Selection Modes: Circle Edit

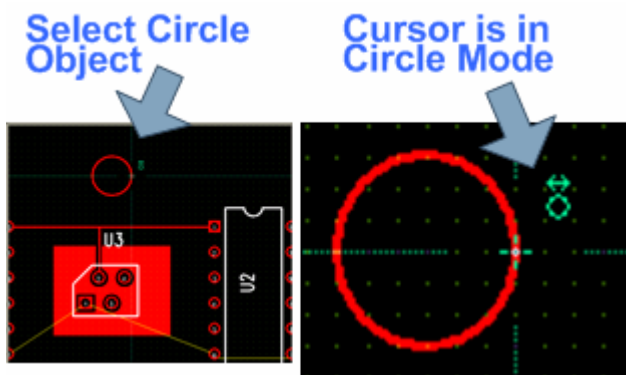


Figure 28: Cursor Modes: Circle Mode

Cursor Selection Modes: Pad Edit

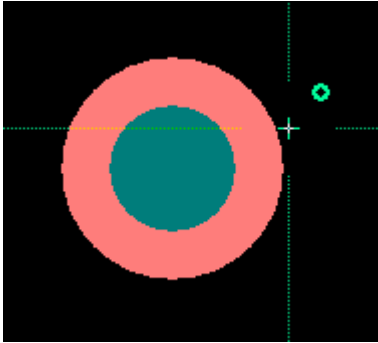


Figure 29: Cursor Modes: Pad Edit

Cursor Selection Modes: Arc Edit

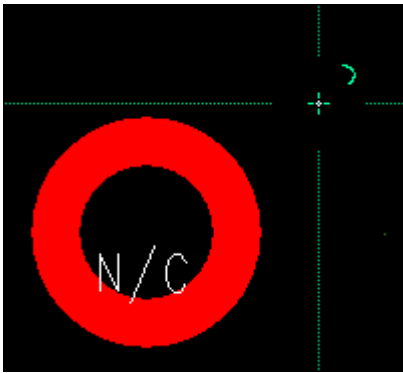


Figure 30: Cursor Modes: Arc Edit

Cursor Selection Modes: Text Edit

Initiate Text Mode by mousing over a text object.

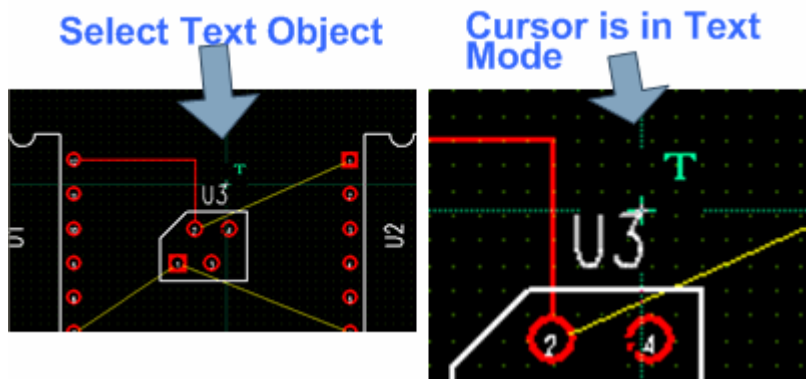


Figure 31: Cursor Modes: Text Mode

Cursor Selection Modes: Polygon Edit

Initiate Polygon mode by mousing over the board outline or a polygon.

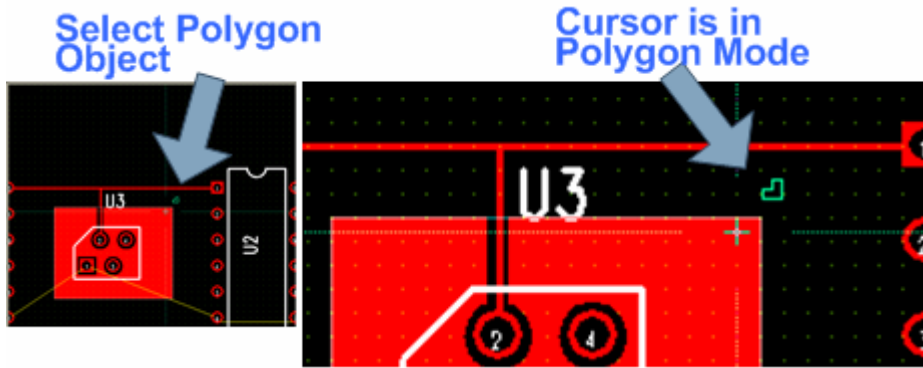


Figure 32: Cursor Modes: Polygon Mode

Cursor Selection Modes: Component Edit

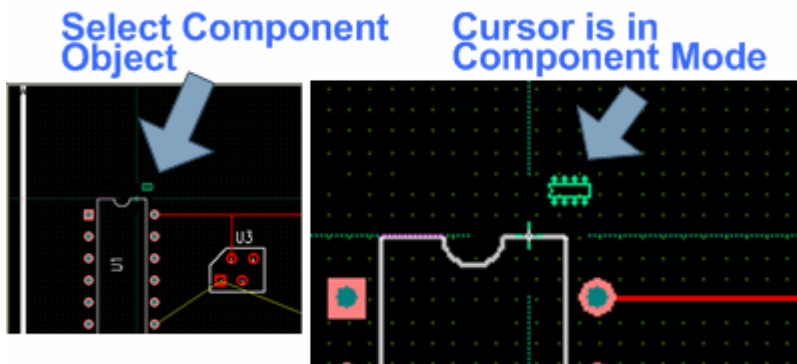


Figure 33: Cursor Modes: Component Mode

Cursor Routing Modes: Connecting

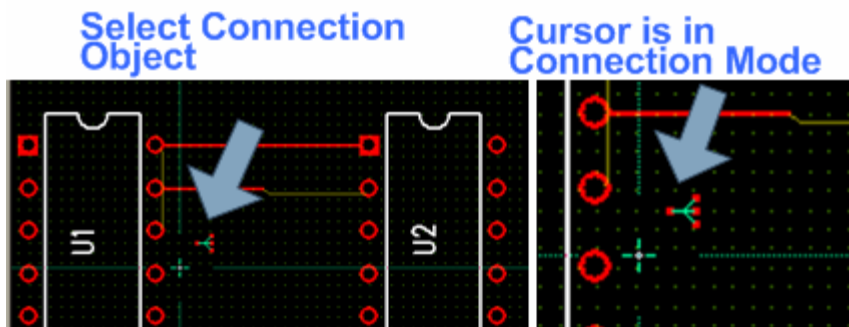


Figure 34: Cursor Modes: Connection Mode

Cursor Routing Modes: Highlighting

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

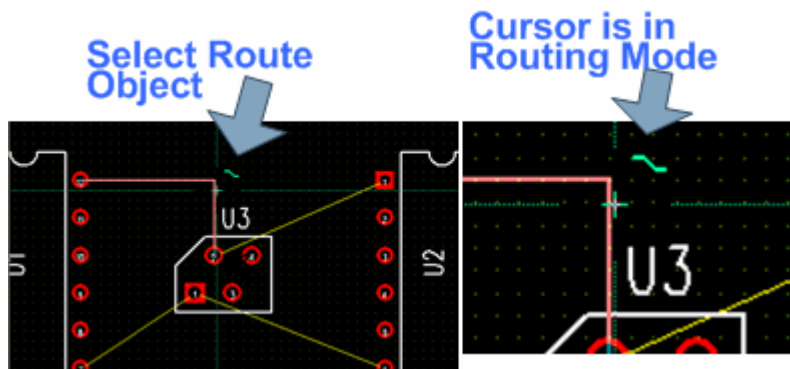


Figure 35: Cursor Modes: Routing Mode

Cursor Routing Modes: Re-Routing

Click on the route to re-arrange it.

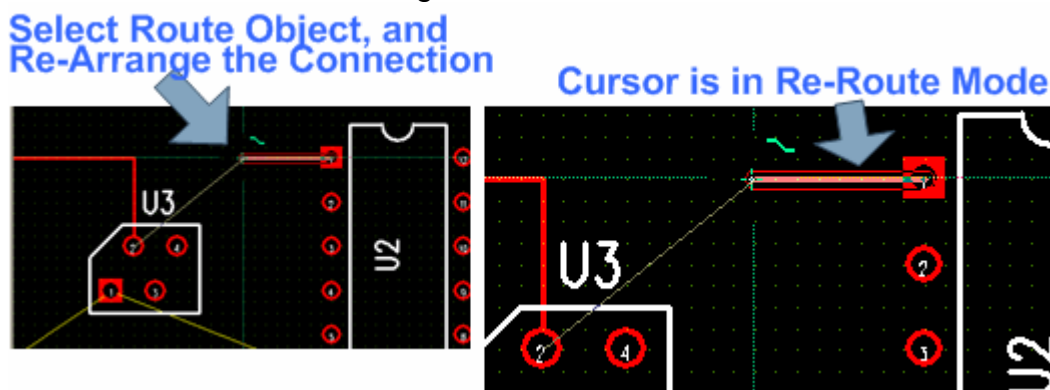


Figure 36: Cursor Modes: Re-Route Mode

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

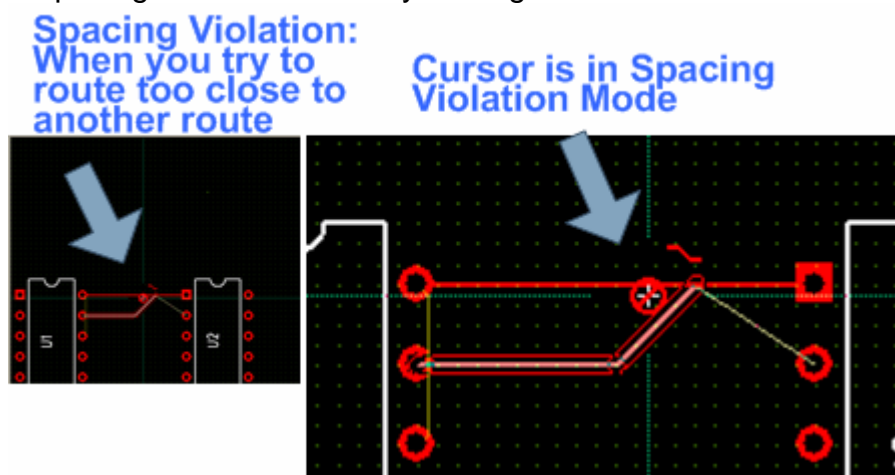


Figure 37: Cursor Modes: Spacing Violation Mode



Appendix C: Tips

Object Selection

“Modeless” Functionality

In general, all operations in this software are modeless. You can either click or mouseover an object to invoke the appropriate tool to interact with it, and it will change color (highlight) to let you know that it is selected. You can then move it, or perform other actions to it. The exception to this is the creation of new objects.

Mouseover Focusing

Mousing over an object brings it into focus, and it may change color. You can then click on it to perform various actions on it or press the **P-key** for properties.

Left-Mouse Click

If you left-mouse click and release, the object under the cursor is immediately selected and attached to the cursor where it can be interactively manipulated.

Crtl-Left-mouse Click

If you hold down the **control key** and click on an object, then that single object's selection state will toggle. The selected objects are not bound to the cursor in this mode. You can select one or a group of objects by **Ctrl-clicking** on them. You can then move them around by clicking or dragging over them and moving them around.

Area Selection

You can select an object, or group of objects, by **area selecting**: left-mouse clicking, and holding it down while dragging a rectangle over the desired objects. Any objects inside the rectangle will be selected but not bound to the cursor.

Object Creation

To create a new object, you have to select the type of object from the top toolbar. Otherwise, simply clicking on an object will invoke the appropriate interactive tool for that object type.

Escape Mode

The **Escape** key will always get you out of a mode or cancel an operation.

Undo/Redo Mode

The software has **Undo /Redo** functionality, so if you get out of sync with the tutorial you can always **Undo** back to your last saved point. Your Hot Keys for that are **Ctrl-Z** and **Ctrl-Y**, respectively.

Editing the Board Outline

You can delete the board outline and redraw it to as you need. You can do this by selecting or mouseovering it, right-mouse clicking and selecting **Delete**, then redrawing the board outline. To edit the shape of your board you can move one of the editing handles. To do this, click and hold down the left-mouse button and drag to a new position and release. You can also add additional lines or arc segments by right-clicking on one of the editing handles and select **Insert wire** or **Insert Arc** from the popup menu. From this same popup menu you can delete edges or reverse the direction of arcs.

Hot Keys

Control-Y: Redo the last function you performed.

Control-Z: Undo, limited to the last save.

Escape: Moves you out of whatever mode you are in, (ex. **Edit Mode**, **Insert Mode**).

G: Invokes the **Grid** dialog for the screen. Usually 0.025 is standard.

Home: Centers the work on the screen, and fits your work to the width of the screen (make sure your Num-Loc key is off).

M: Measure.

Mouse Wheel: Zoom in and out.

P: Opens the Properties box for the selected object.

R: Rotate by pressing the '**R**' key on your keyboard. Move the component to where you want and click again to drop it.

Z: Sets a via, or connection between layers.

Appendix D: Glossary

Autorouting: Using the **Design | Batch Route** function to generate connections, or **routes**, on your board.

Board: laminated layers of copper, fiberglass sheets, circuits and etched metal connections.

Circuit:

Corners: (....clicking corners of an object);

Gerber Files: PCB123 converts your files to standard Gerber 274X format files.

Gloss: (Cleans routes after using autorouter, but can create shorts.)

Modeless: In general, all operations in this software are **modeless**. You can either click or mouseover an object to invoke the appropriate tool to interact with it, it may change color to let you know that it is selected. You can then move it or perform other actions to it. The exception to this is the creation of new objects.

PCB: *Printed Circuit Board*: laminated layers of copper, fiberglass sheets, circuits and etched metal connections.

Plane: a layer or solid sheet of copper people attach connections to within the printed circuit board (PCB).

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

Route: Creating connections or **routes** between components on the board, later this is etched in metal on the fiberglass sheeting.

Run DRC: (Run Design Rule Check: looks for traces and spaces that are insufficient.)

Segment: a section of a connection or **route**.

Via: A connection between layers, it connects the top of the board to the bottom only, not between the inner layers. It is used when connections between components on one layer may overlap, but you can jump the connection to another layer using a **via**, so connections, or **routes**, don't overlap.