PADS-PowerPCB 4 Tutorial (with Blazeroute)

PADS-PowerPCB is the ultimate design environment for complex, high-speed printed circuit boards.

PROCEDURE FOR SIMULATION IN SCHEMATICS

1. Importing Design Data from PowerLogic or Using ECO mode to draw the circuit,
2. Edit component,
3. Setting the PCB,
4. Routing Connection,
5. Creating the Copper Pour,
6. Printing.

PowerPCB version 4.0 :

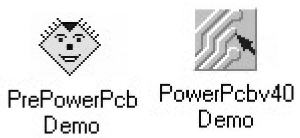
STARTING PADS

1. To open a PADS-PowerPCB :

Demo Version :

1. Click "PrePowerPcb Demo" icon,

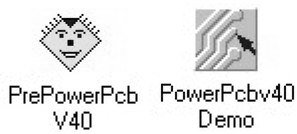
2. Click "PowerPcbv40 Demo" icon to open DEMO version.



Full Version :

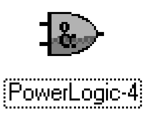
1. Click "PrePowerPcb V40" icon,

2. Click "PowerPcbv40 Demo" icon to open FULL version.



Importing Design Data

To avoid missing the connection line, Design data can be imported from PADS-PowerLogic.

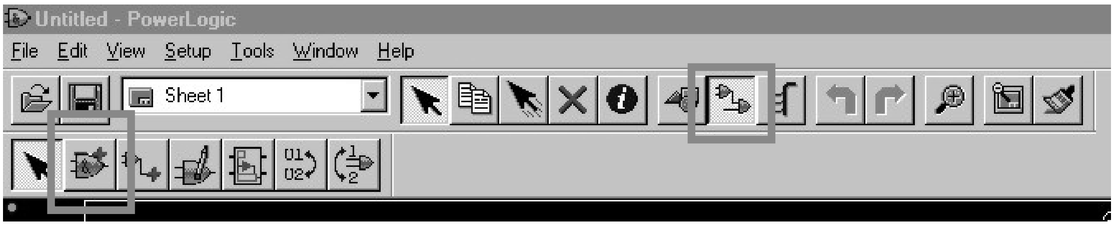


To open the PADS-PowerLogic,

Double-click on the **powerlogic-4** icon.

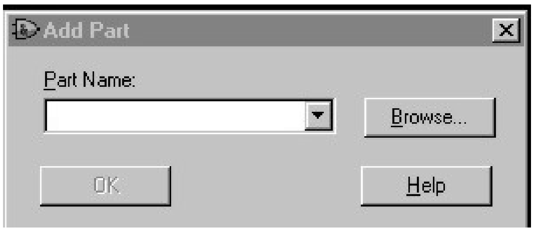
Start PowerLogic and draw the schematic file.

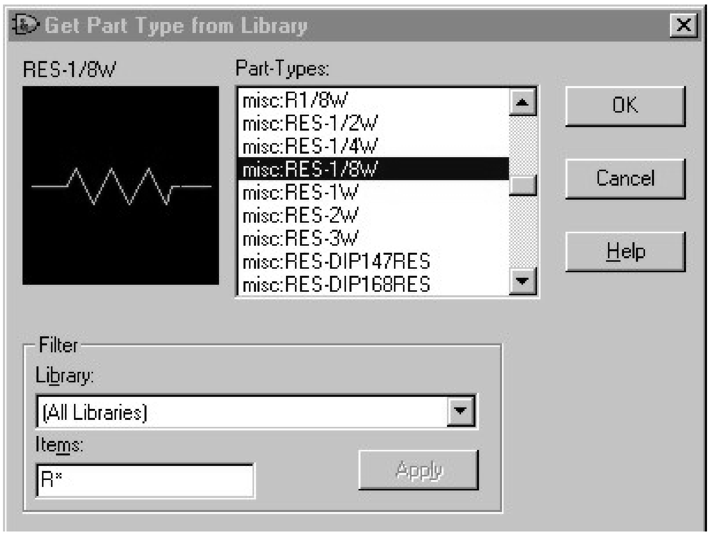
Click **Design** icon, and then click the **add part** icon.

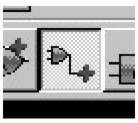


Select **Part Name** in Add Part windows.

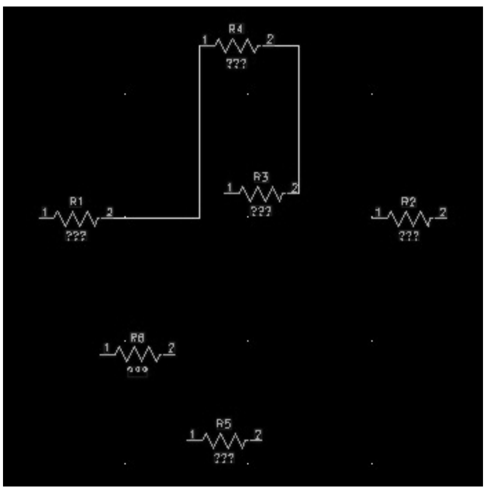
Or Click **Browse** to search New Part Name to select new part.







Click **Add Connection** icon to connect the components.



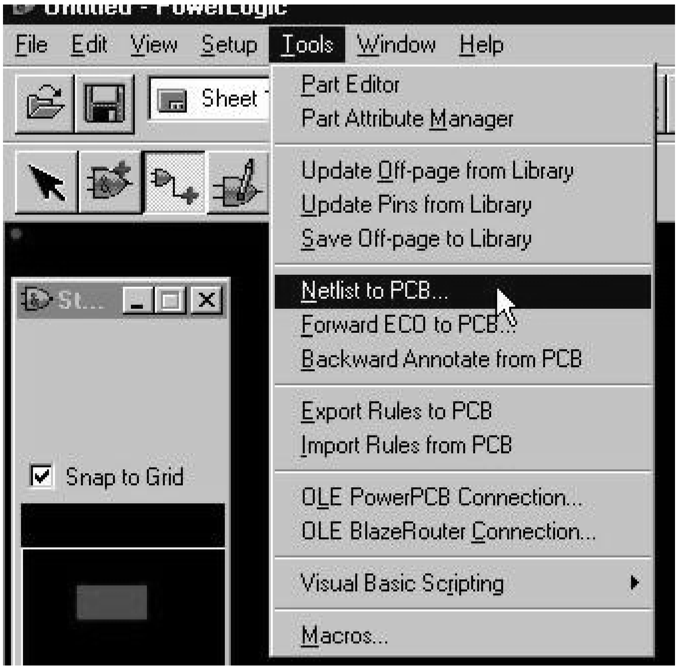
After connect all the components. Then you can use two methods to send the design data to PowerPCB.

1. Using Netlist to PCB method with .asc file.

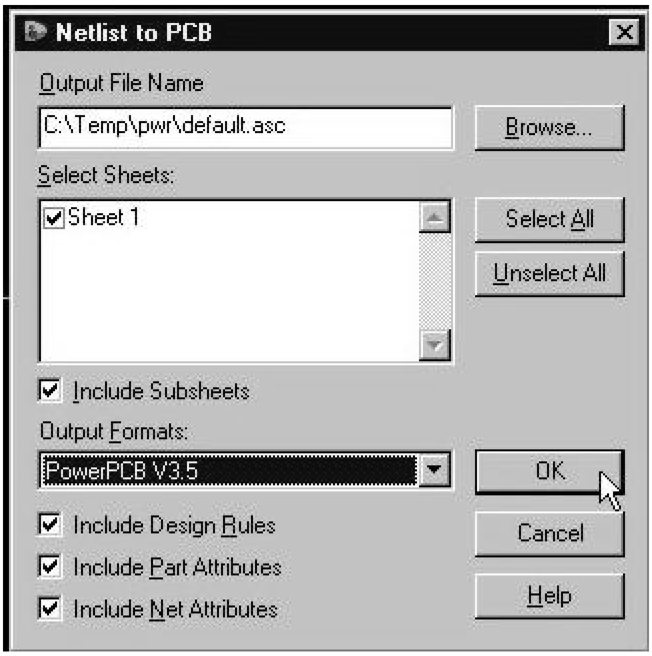
2. Using OLE PowerPCB Connection method. (Need open PADS-powerPCB first !)

1. Using Netlist to PCB method :

Select **Tools/Netlist to PCB** to send a Netlist to PADS-PowerPCB.



Save the output file name *default.asc*.



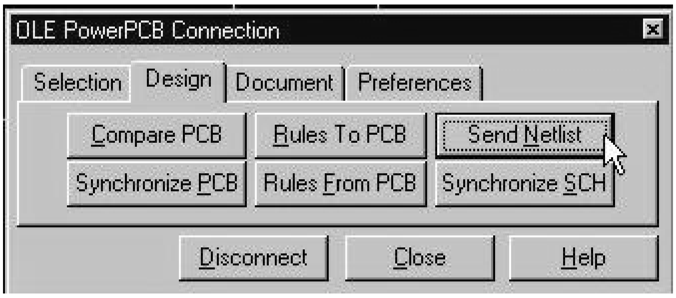
Open PADS-PowerPCB,
Select **File/Import** to import .asc file.

2. Using OLE PowerPCB Connection method.

Select **Tools/OLE PowerPCB Connection**.

Select **Design** tab from PowerLogic's OLE PowerPCB Connection dialog box.

Click **Send Netlist** icon to automatically export a netlist to PADS-PowerPCB as a netlist import.



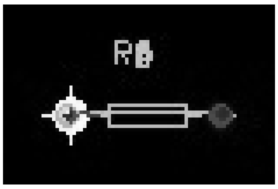
Disperse Components

To arranges all of the selected objects on grid sites without overlapping.

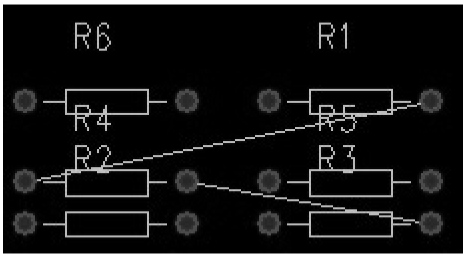
After import or send design to PADS-PowerPCB, all components are overlapping.

Select **Tools/Disperse components** to disperse the components.

Before dispersion :

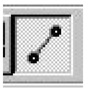


After dispersion :

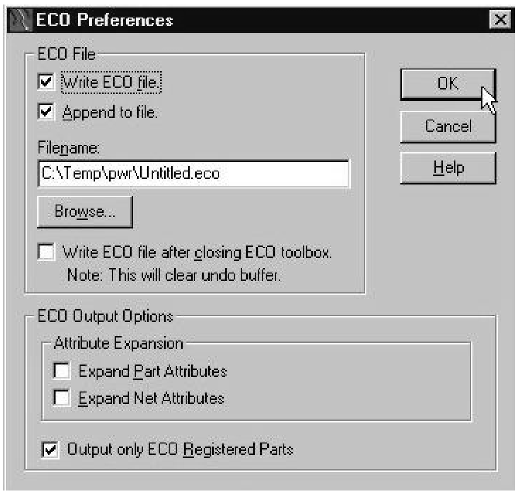


Using ECO

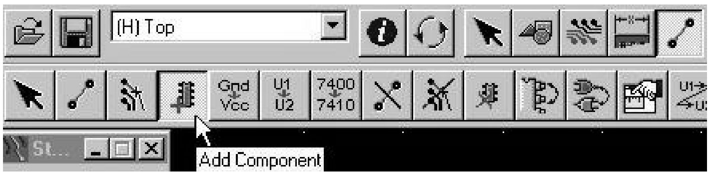
Use the **ECO** Toolbox to perform (Engineering Change Order) operations, which modify the net list, or to add a new component and connection.



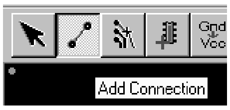
Click **ECO** icon, save the eco file. Click OK.



Click **Add Component** icon to add component.



Add Connection and Add Route



Decal Editor

PADS-PowerPCB use components from the parts libraries. Every part in the library has a decal associated with a part type in a parts library.

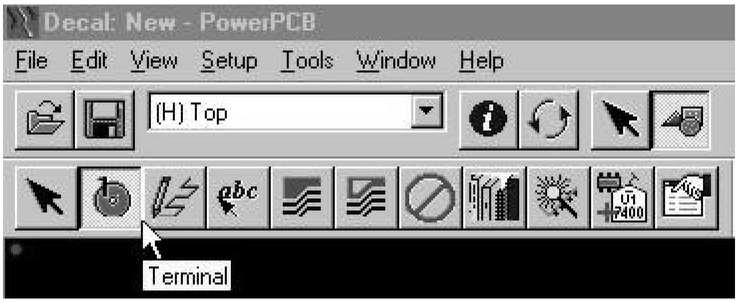
Use the Decal Editor to create and/or edit these decals.

1. Create new Decal

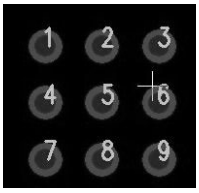
Select **Tools/Decal Editor.**

Click **Drafting** icon.

Click **Terminal** icon to draw terminal.

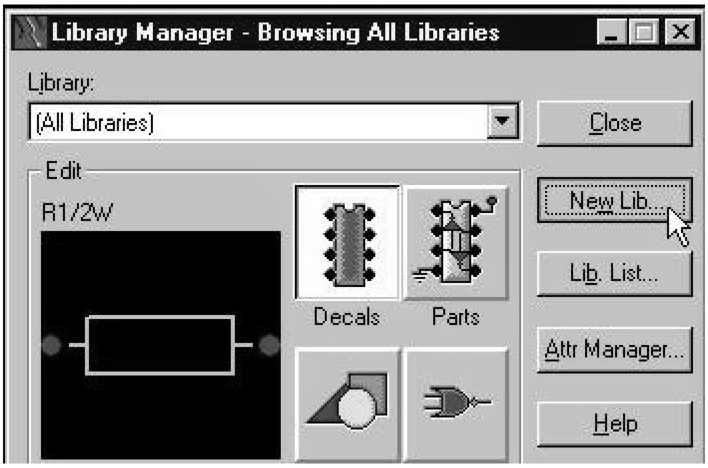


Example :



2. Create new library associate with new part.

Select **File/Library** to open the Library Manager dialog box.



Click **Decals** icon as the library type.

Click **New Lib…** icon to save a new library in ***TEMP*** folder.

3. Save the New Decal.

Select **File/Save Decal** to save decal in new library.

Select Library and type the name of decal.

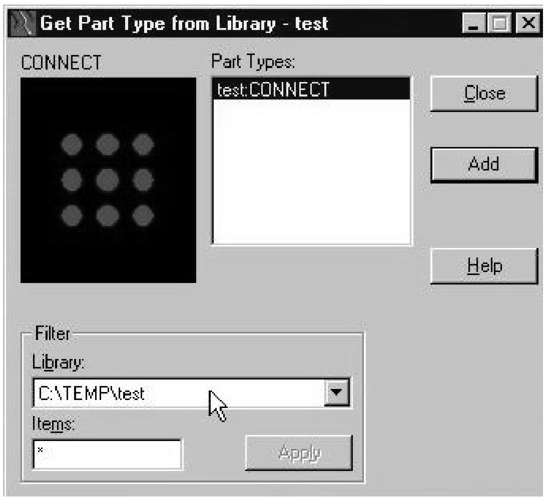
Select **File/Exit Decal Editor** to return to PADS-PowerPCB.

4. Add new Decal.

Click **ECO** icon and Click **Add Component** icon.

Select Library and Part Type.

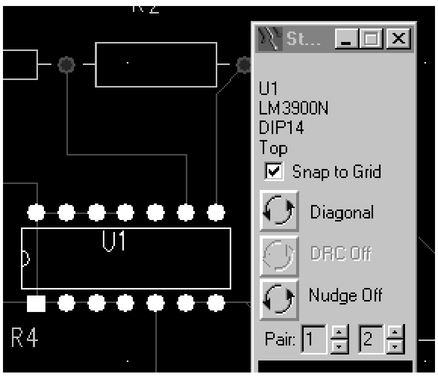
Click **Add** icon.



Work Area and Grid Settings

1. Status Window :

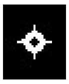
The Status Window is a floating window with useful summary information, preference controls, and a viewing control.



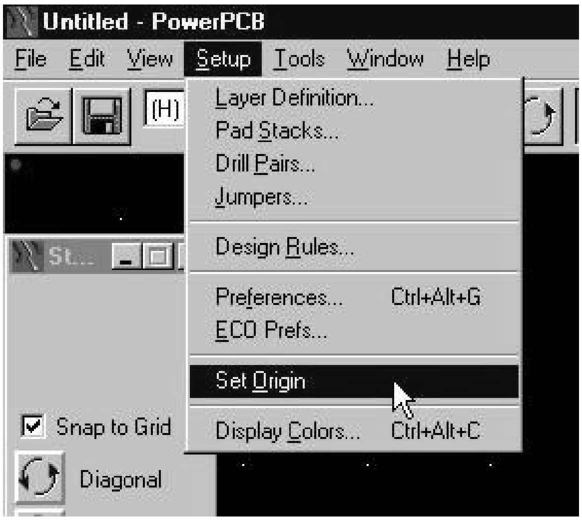
The Status Window is a floating window that contains:
·The net name and pin connects of of a selected part.
·A Snap to Grid check box. Snaps the cursor to the current design grid.
·The Postage Stamp viewer. Use it to pan and zoom.

2. Setting Origin :

The design positions are relative to the origin.



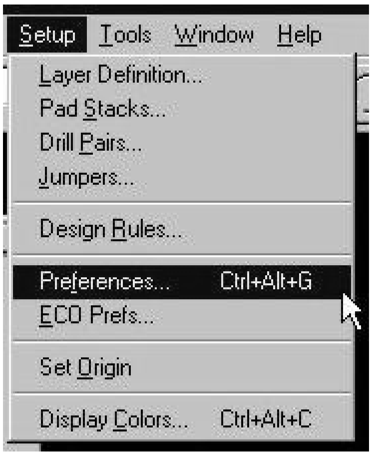
Select **Setup/Set Origin** then click on the workspace to indicate a new position for the origin.

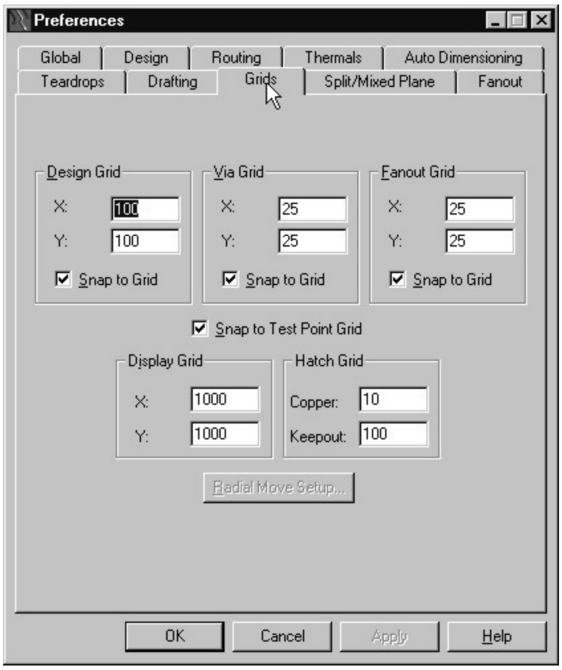


3. Setting Grids :

Sets the spacing of the design grid which controls the general placement of parts.

Select **Setup/Preferences** and select the **Grids** tab to view or set the current display grid setting.





Or typing **G500** and pressing Enter to set the Design Grid in one step.

Set Grids to 500, typing ***G500*** and pressing Enter.

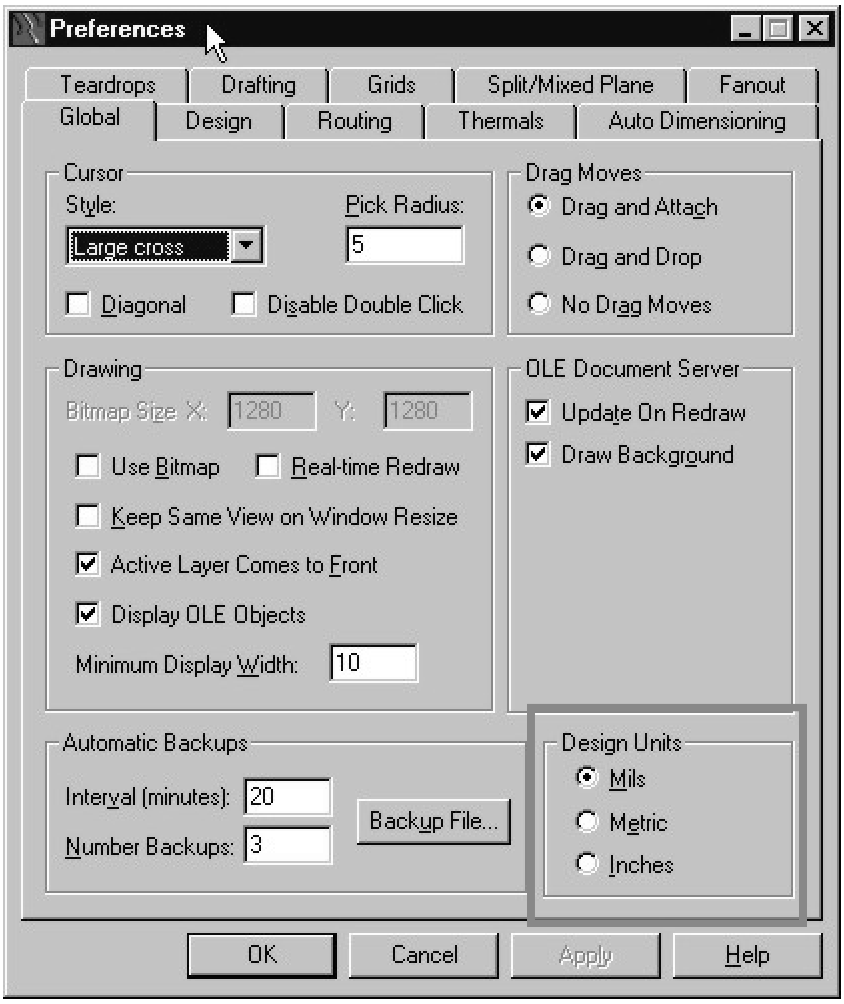
Set the Display Grid to 200, typing ***GD200*** and pressing Enter.

4. Setting Units of Measure :

Select **Setup/Preferences** to change the unit of measure to inches, mils (default setting 1000 = 1"), or metric units (1 mm).

The Design Units combo box is on the **Global** preferences tab.

Now, the current design units set to Mils.



5. Using the Selection Filter :

To focus your selections on specific objects, PowerPCB has a Selection Filter. The Selection Filter allows you to specify which design objects can be selected.

- *To access and review the Selection Filter:*

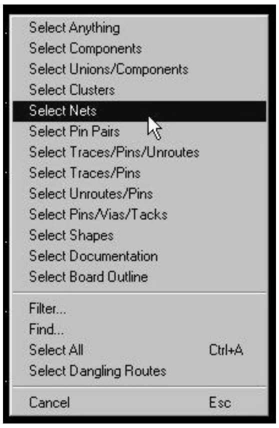
Select **Edit/Filter** to open the Selection Filter dialog box.
Objects are organized into three categories—***Design Items, Drafting Items, and Layers***.

- *Selection Filter Shortcuts :*

Click the right mouse button while no object is selected, a pop-up menu containing a list of **Selection Filter** shortcuts appears.

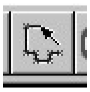
Select one of these shortcuts updates the Selection Filter to include only the items in the shortcut description.

Select **Nets** shortcut and note how the selection filter is updated to allow selection of nets only.

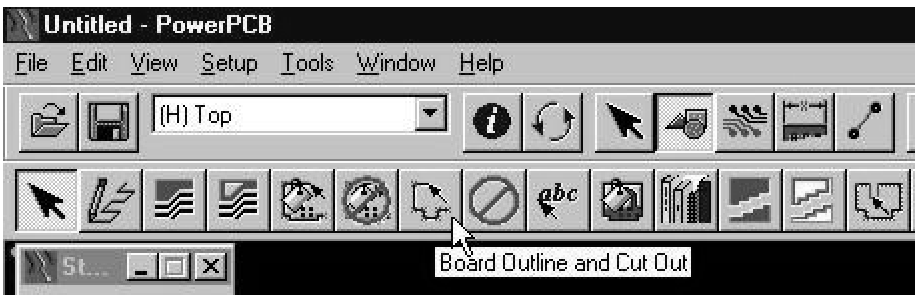


6. Creating a Board Outline :

It defines the board boundary.

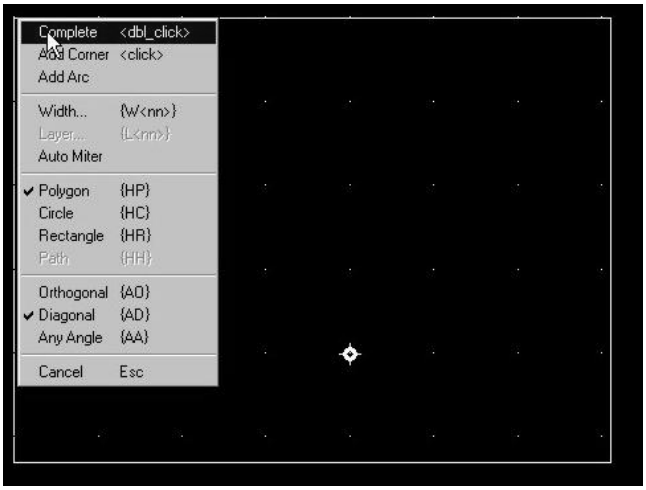


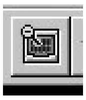
Click **Board Outline and Cut out icon** to Draw the shapes of board used for PCB design. (Default: Polygon [HP])



After draw the board outline to define the board boundary.

Click the right mouse button to open the pop-up menu, then choose ***Complete***,
or Double-click the left mouse button, to close and complete the polygon.



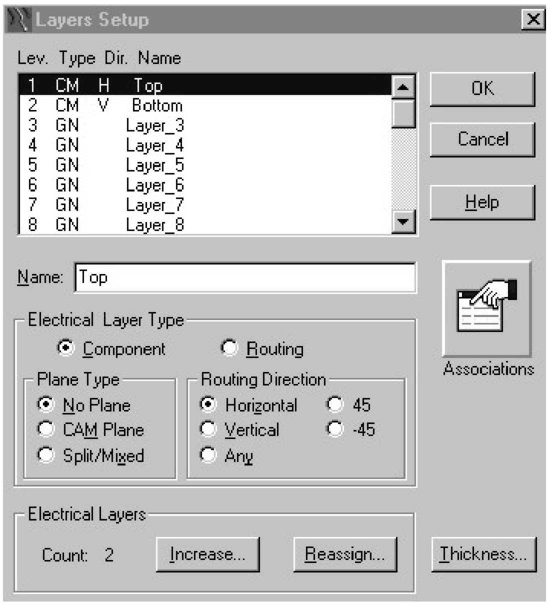


Then, Click the **Board icon [Ctrl-B]** from the toolbar to fit the board outline to the screen.

7. Setting the Layer of the PCB :

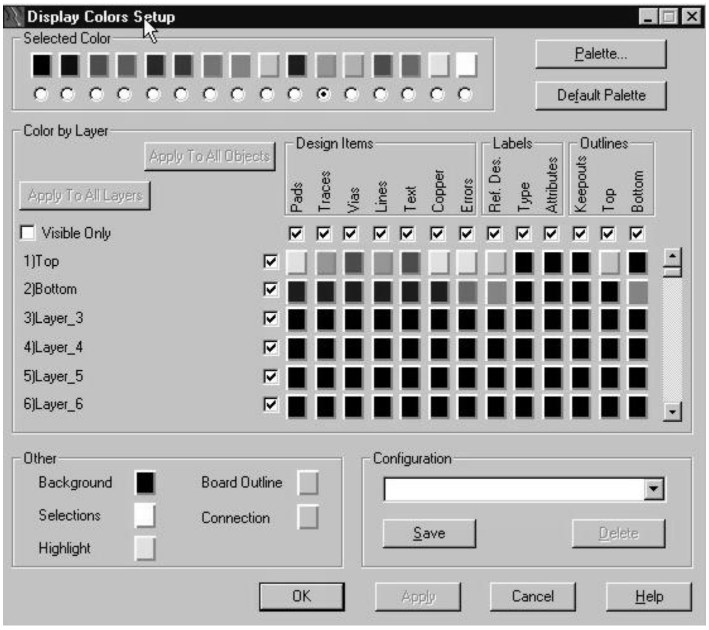
To assign the number of layers, the nets associated with embedded plane layers, layer stackup, and layer thickness.

Select **Setup/Layer Definition**. The Layers Setup dialog box appears.



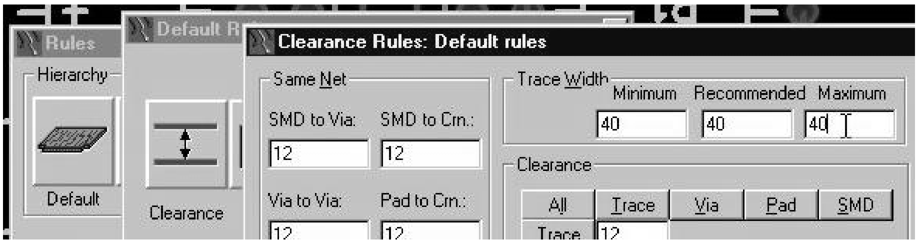
8. Setting Layer Colors :

Select **Setup/Display Colors**. The Display Colors Setup dialog box appears.



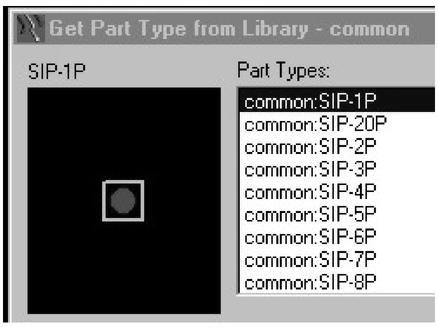
9. Setting Design Rules and Trace width :

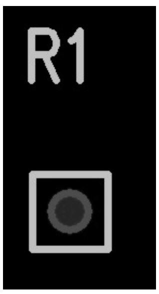
To assign and edit rules, or width, spacing and general routing.
Select **Setup/Design**, Click **Default** icon and **Clearance** icon to set Trace Width.



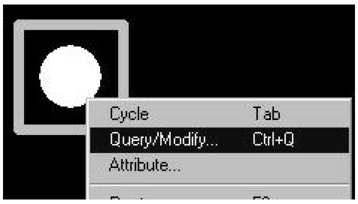
10. Setting Pad Stack width :

Example
From Get Part, Select SIP-1P.

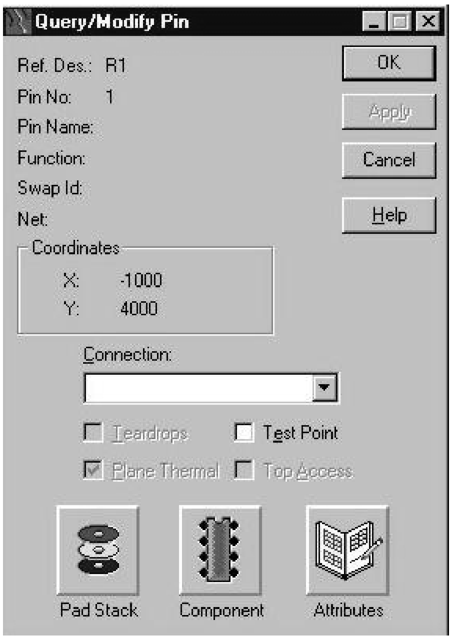




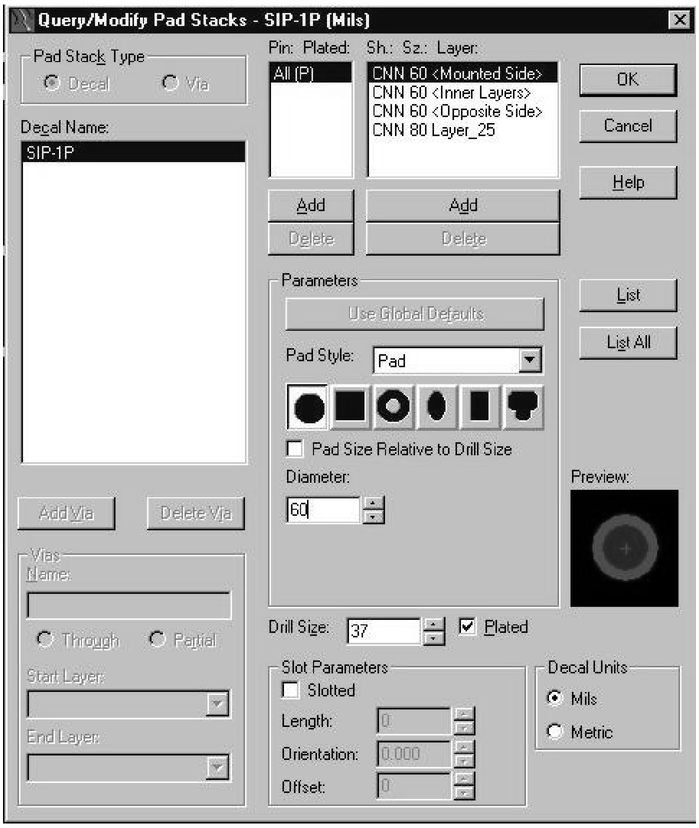
Select the Pads of components. and click right mouse to select Query/Modify to modify Pad Stack.



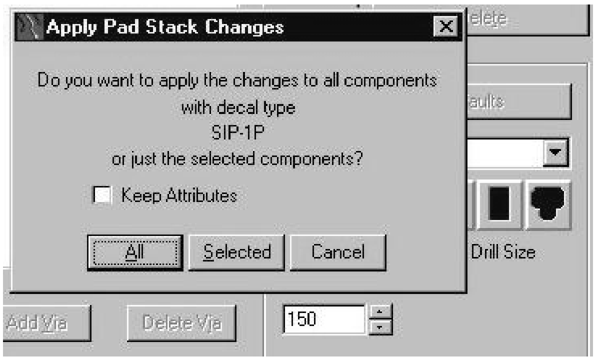
Click the Pad Stack icon.



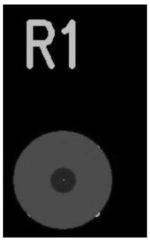
Input the Diameter of Pad. Click OK.



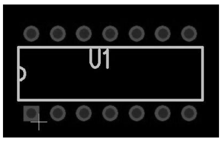
Click All or Selected components.



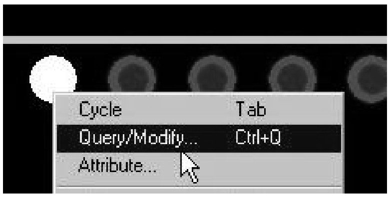
After increase the diameter to 150.



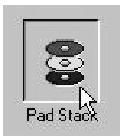
11. Change the pads shape:



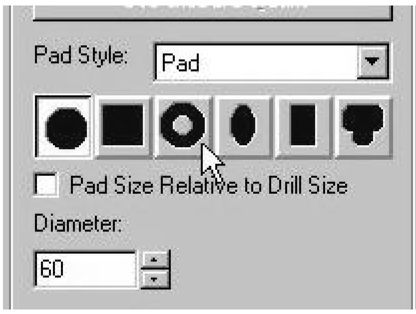
Click left-click mouse on the pad of component.

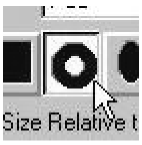


Click right-click mouse to select [Query/Modify]



Click Pad Stack icon

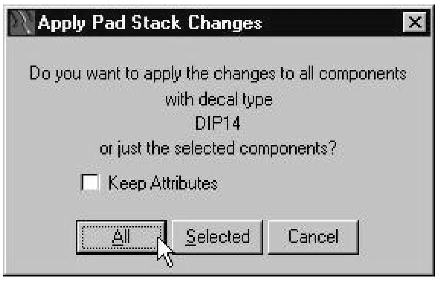




Select the icon pad shape.

Input the size of inner diameter and diameter of pad.

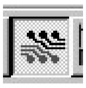
Then select other pin number of component.

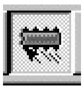


Click OK. Then click All if change all component's pads.

Controlling Selections

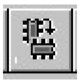
1. Parts Placement and Moving Components :

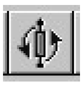
Click the **Design toolbox** icon. 

Click the **Move Component mode** icon. 

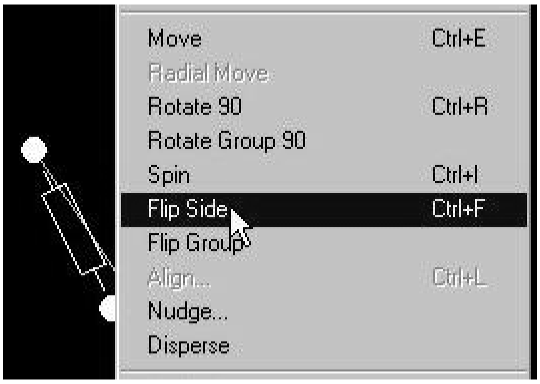
Select the component and click the left mouse button to move the component.

2. Rotating Components :

Click **Rotate** icon to rotate 90. 

Click **Spin** icon to rotate free angle. 

Select component and Click the Right mouse or Press Ctrl-F to flip Side.

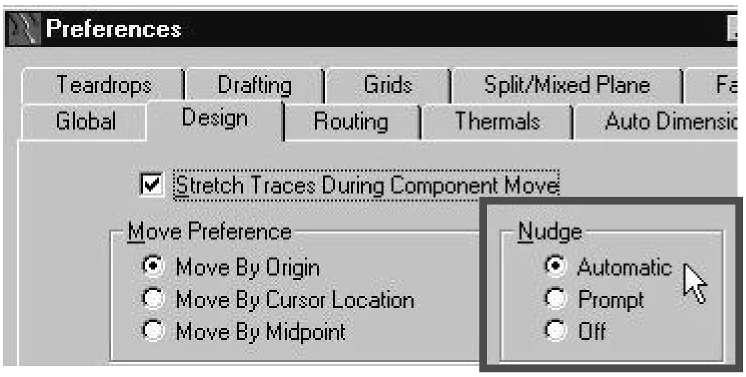


3. Enable Automatic Component Nudging :
PowerPCB's placement features let you automatically shove or nudge adjacent components whenever components are placed too close to each other or if they overlap.

To enable automatic component nudging:
Select **Setup/Preferences**. The Preferences dialog box appears.

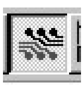
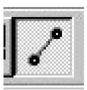
Select the **Design** tab.

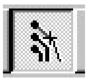
Select **Automatic** from the **Nudge** area of the dialog box to enable automatic nudging.
Choose OK to apply the changes and close the Preferences dialog box.



Adding Route

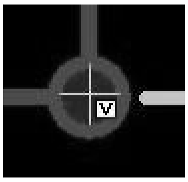
1. Using the Manual Route Editor :

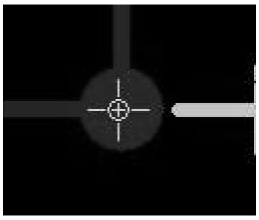
Click **Design** icon  or Click **ECO** icon 

Click **Add Route** icon to route 

Select the Pin.

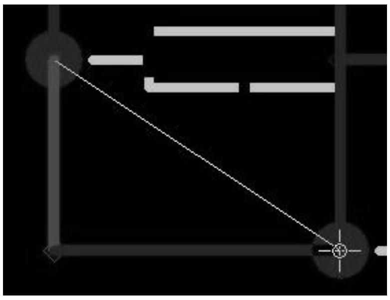
Before Selection: After Selection:

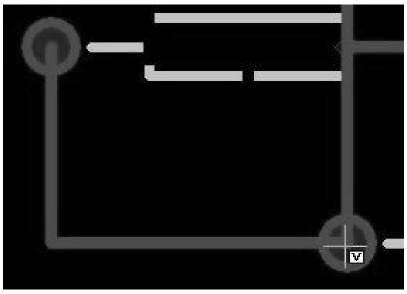




After select the correct pin, you can draw a line and path to connect other pin.

During connection: After connection:

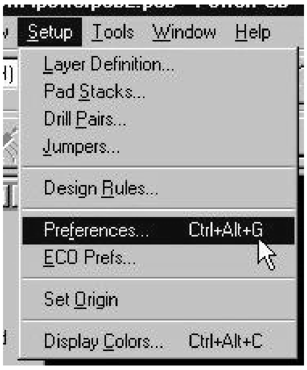


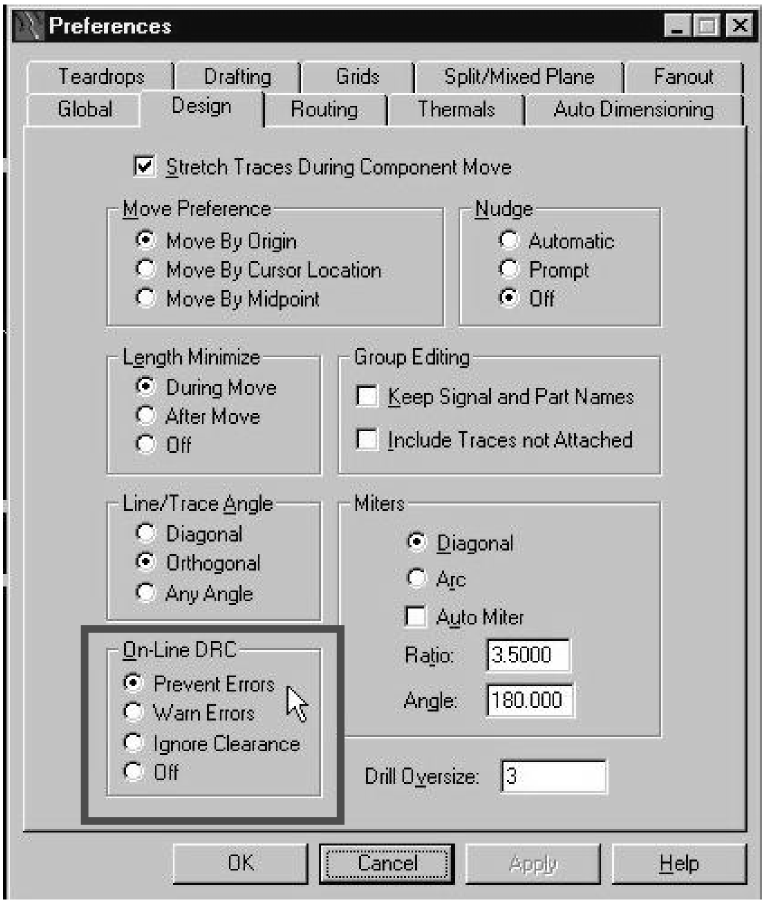


2. Using the Dynamic Route Editor (DRE) :

The dynamic route editor (DRE) is an interactive autorouter that follows the direction of your cursor as you move it, seeking optimal paths and installing corners as the route progresses.
*[Note: It is not reliable! **Save** file before execute Auto-Route]*

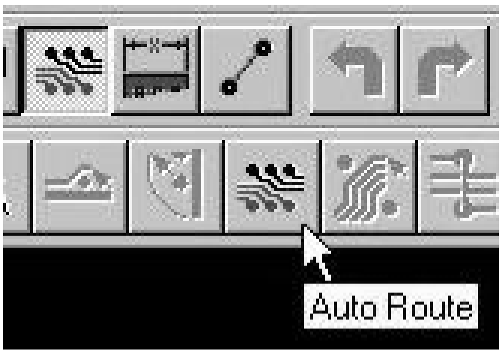
To enable Dynamic route:
In order to enable the On-line Design Rule Checking (DRC),
Select **Setup/Preference**, Select **Prevent Error** in **On-Line DRC** on Design Tab.
Then click OK.





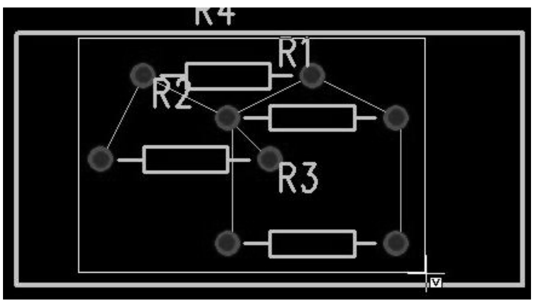
After setting the DRC,

Click **Design** icon and then click **Auto Route** icon to route the connection.

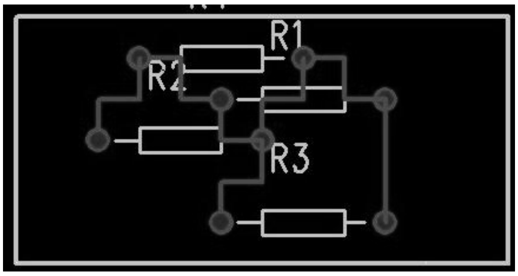


Select all components in box, then one connection will auto route and change colour.

Or Double-Click the pin to auto route connection.



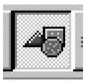
Select all connection again, then another connection will auto route and change colour.

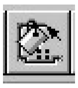


Creating Poured Copper

1. Creating the Copper Pour Outline :

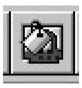
The pour outline defines the boundaries of the copper pour area.
Define the Pour Outline :

Select the **Drafting toolbox** icon from the toolbar. 

Click the **Copper Pour** icon from the Drafting toolbox. 

Draw the copper pour area.

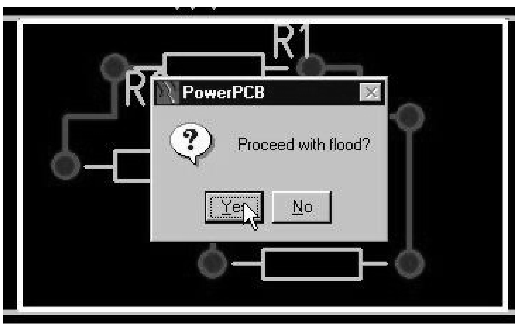
2. Flooding the Pour Outline :

Click the **Flood** icon. 

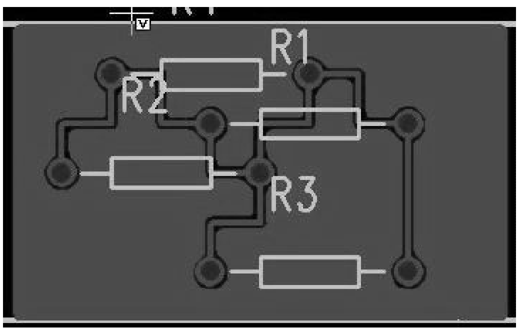
Select the flooding copper pour outline area. Then Choose Yes.

After flooding the copper pour outline, you can see the area of poured copper.

Before Flood:



After Flood:



Setting the Units for the Dimensions

1. Setting the Units :

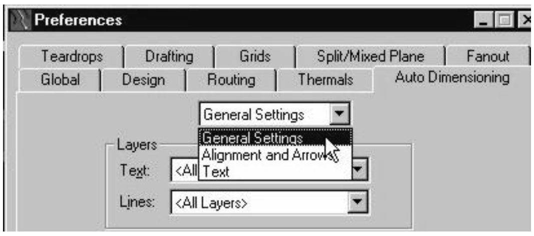
Select **Setup/Preference**,

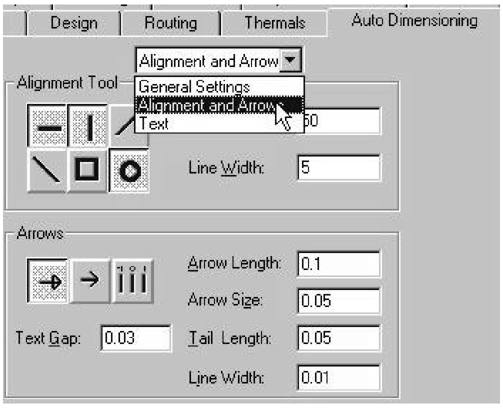
Choose the **Global** tab and set **design units** box.

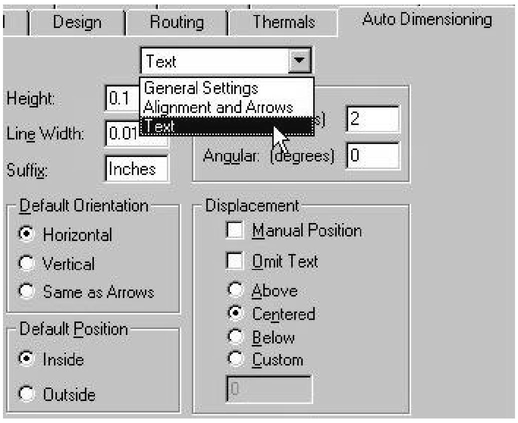
2. Assigning General Settings, Alignment and Arrows and Text Properties of Auto Dimension :

Select **Setup/Preference**,

Choose the **Auto Dimensioning label** tab.

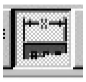




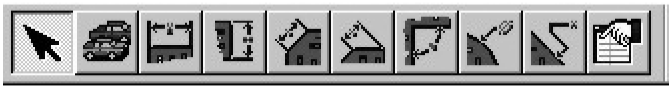


3. Adding the Dimension :

Auto-orient mode automatically establishes the orientation of newly added dimensions.

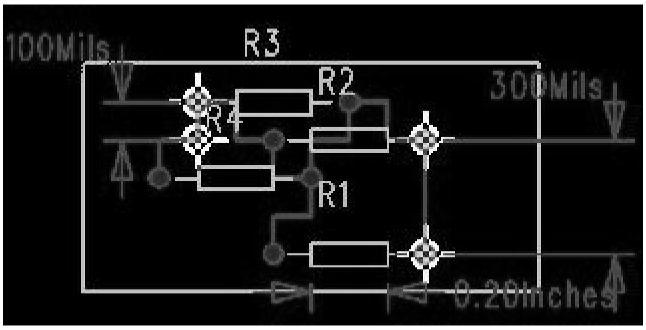


Click **Autodim** icon to add dimension



Click the **Auto** icon or other dimension icon.

After adding dimension:



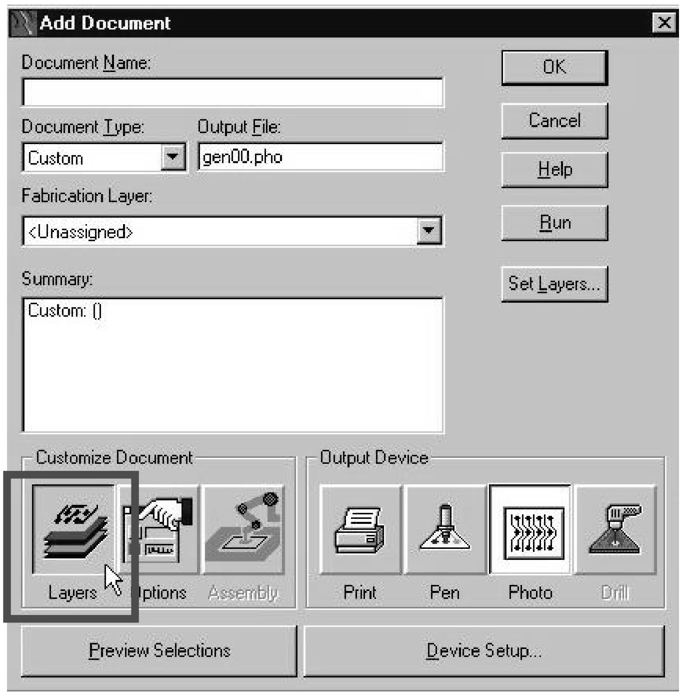
Printing the PCB

Use the CAM command on the File menu to produce laser printouts.

Select **File/CAM,** The Define CAM Documents dialog box appears.

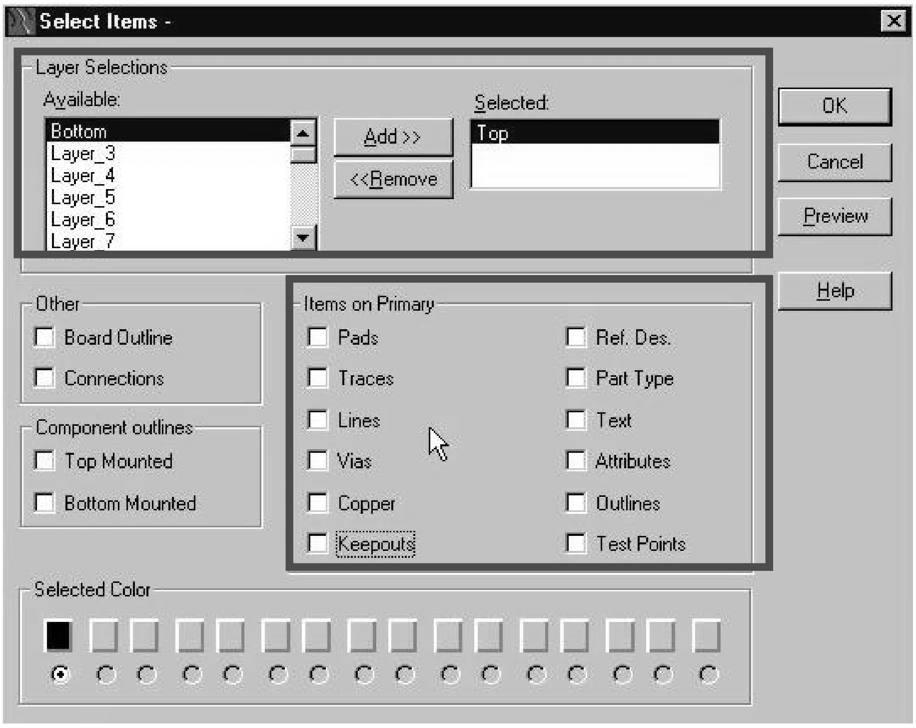
Click **Add** icon

Click **Layers** icon to define which layers and items should appear on printouts.



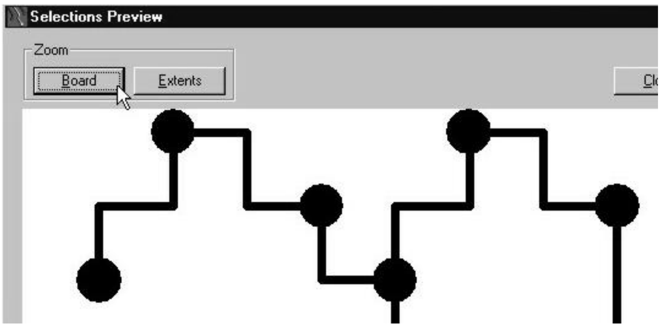
Select the **Layer** in Layer Selections, Click **Add** icon.

Check item in Items on Primary. E.g. Pads, Lines

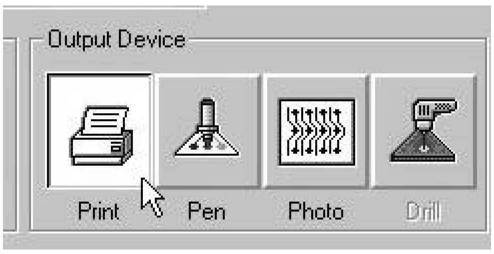


Click **Preview** icon to preview the output.

Click **Board** icon to zoom the output of PCB board. Then **close**.



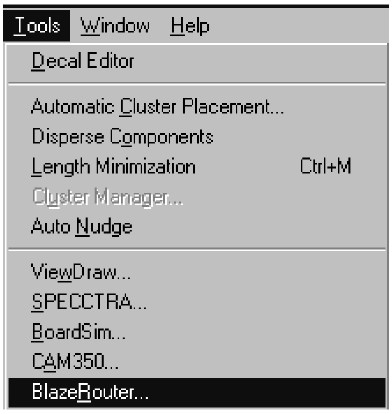
Click **Print** icon in Output Device box, and Input the Document Name.



Click **Run** to print the output to Printer.

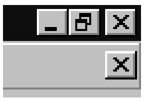
BlazeRouter

In PowerPCB,
Select Tools/BlazeRouter

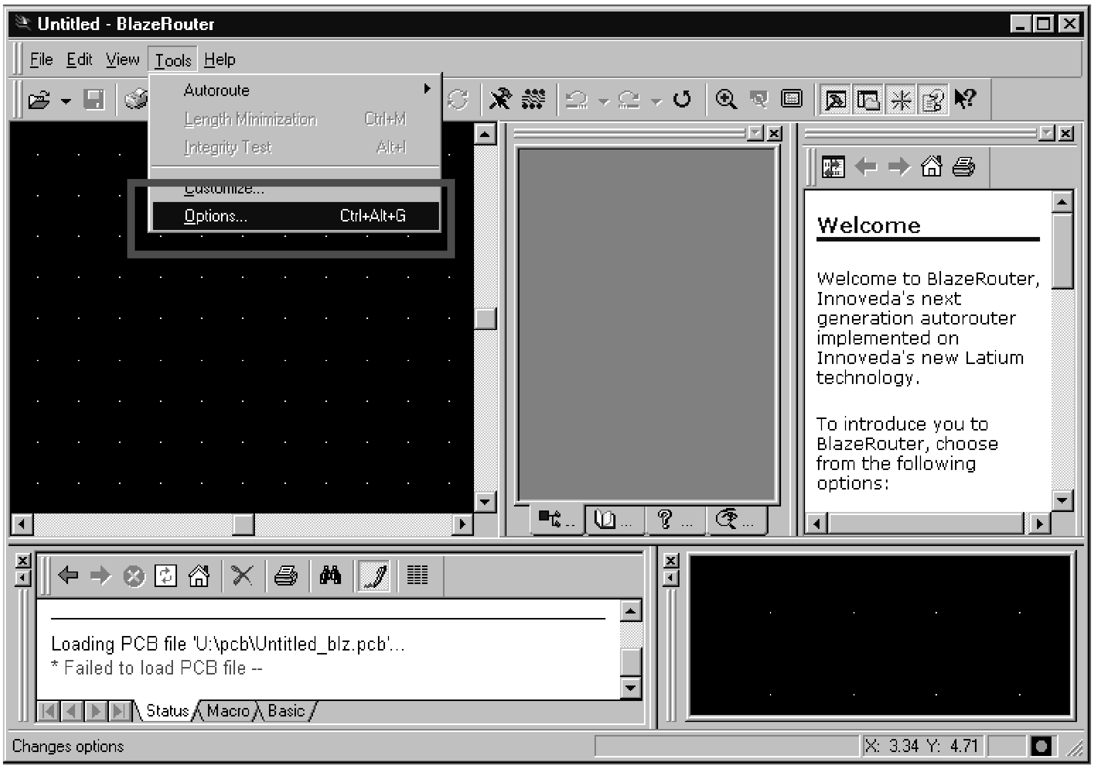


Select Launch BlazeRouter Only in Action box
Then press Proceed

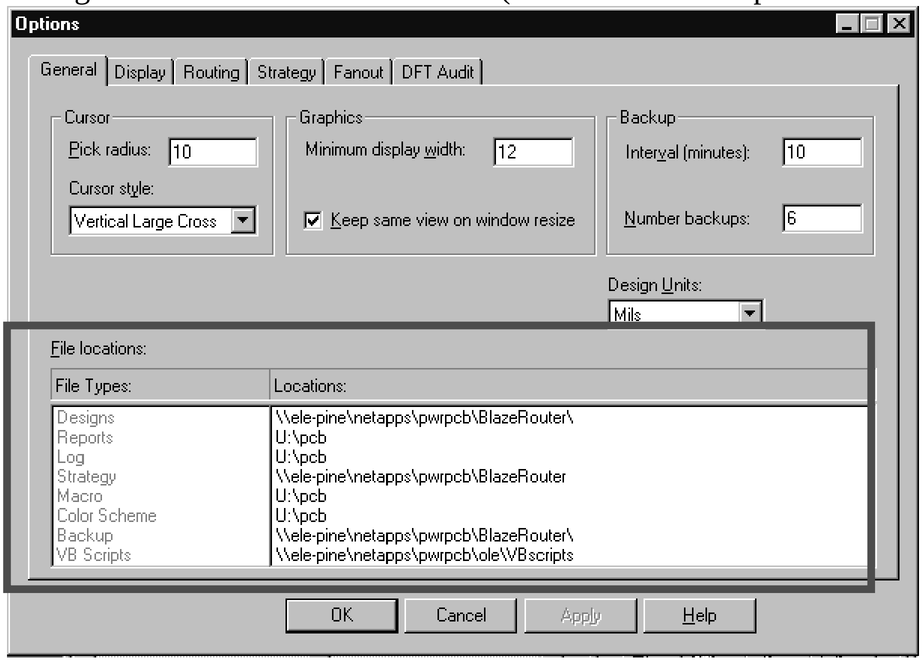
Click Ignore in BlazeRouter Window
(Hint: Wait about 1 min, it hide by other program, so minimum all program window first)



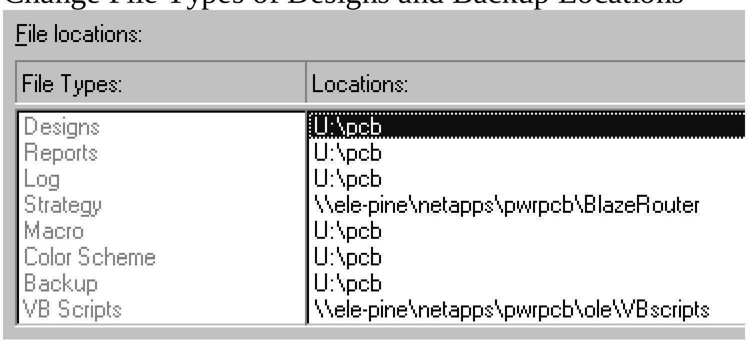
Select Tools/Options in BlazeRouter Program



Change File Location in General Tab. (otherwise cannot open file for auto route)

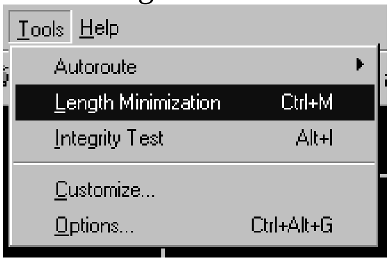


Change File Types of Designs and Backup Locations

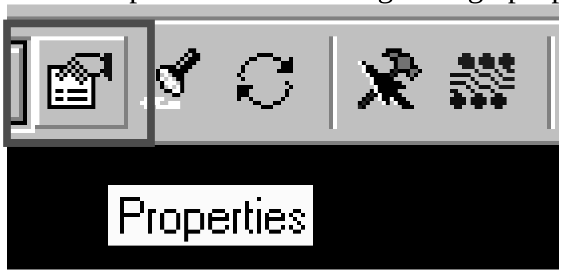


Now, can open file for auto-route

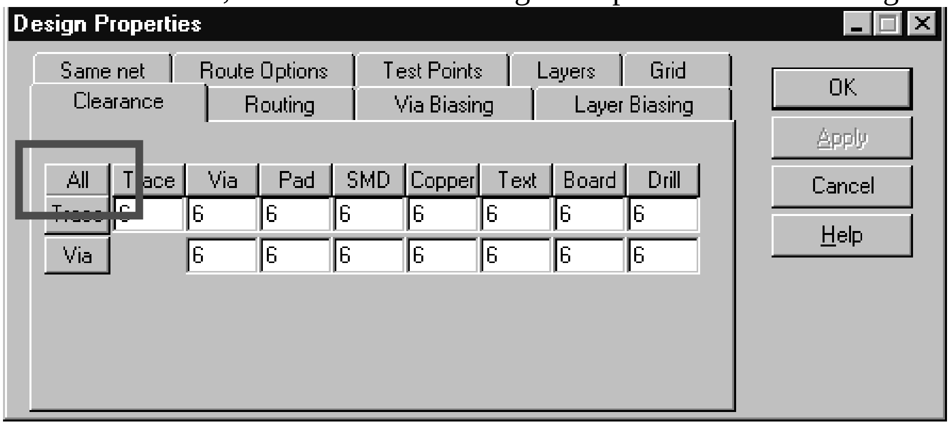
Select Length Minimization to short connection



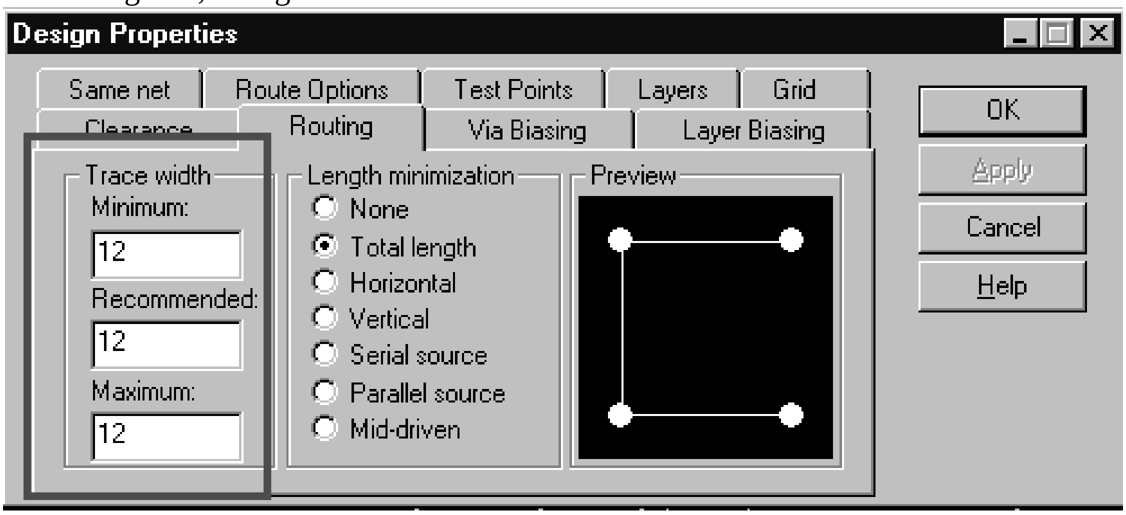
Click Properties icon to change design properties



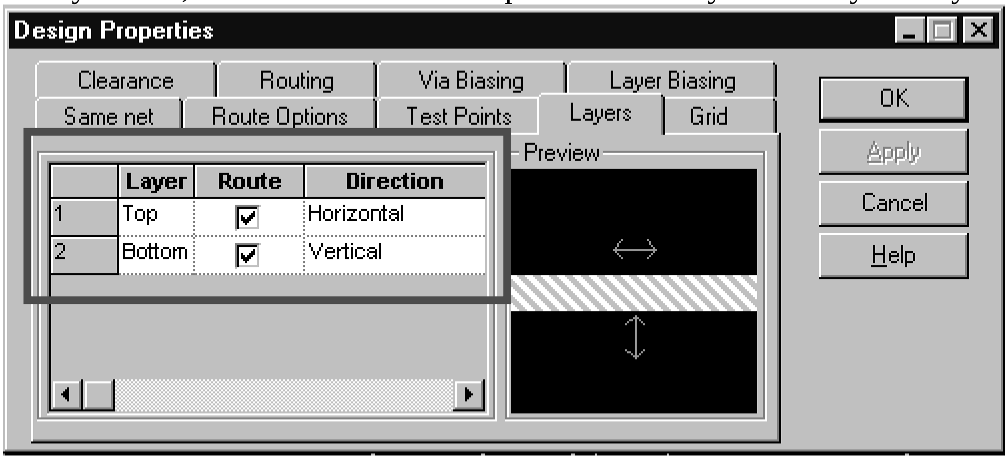
In Clearance Tab, click All icon to change the space between all thing.



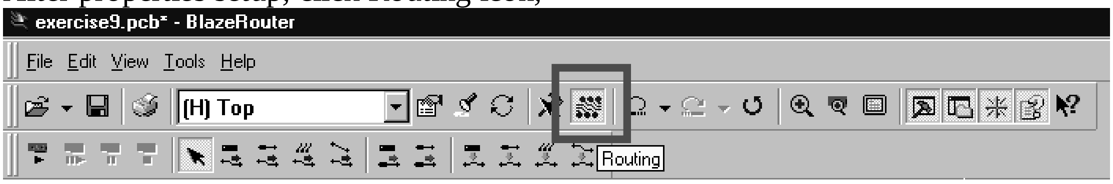
In Routing Tab, change the value in Trace width



In Layers Tab, check the Route box Top or Bottom only when only one layer design is used.

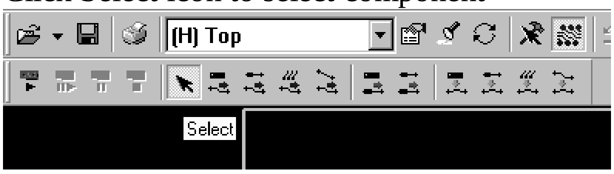


After properties setup, click Routing icon,

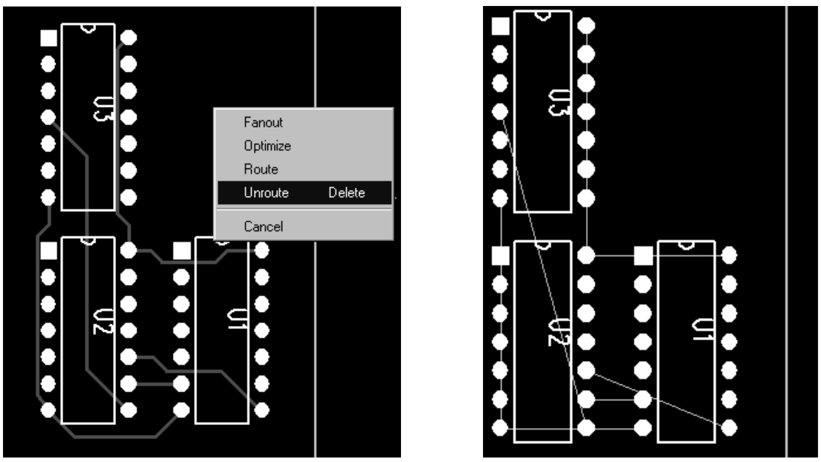


And Click Start Autorouting icon to start auto route.

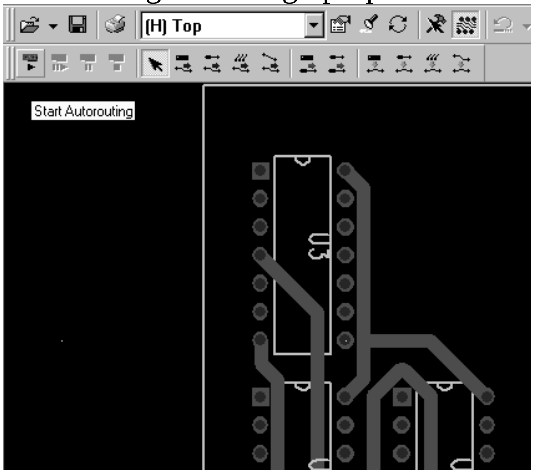
Click Select icon to select component



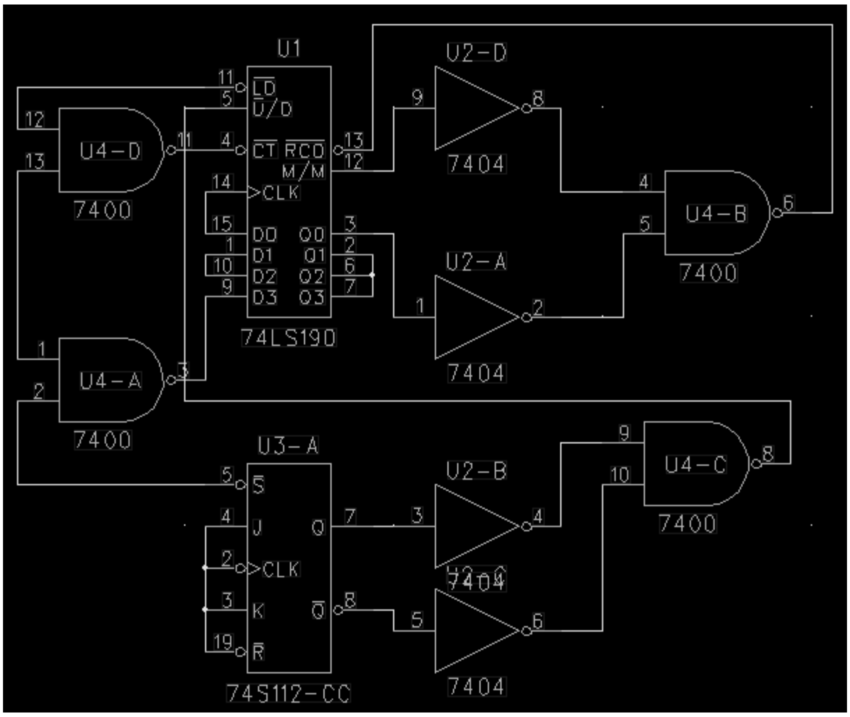
Select Component with cursor and click right-click mouse,
Select Unroute to delete the route.



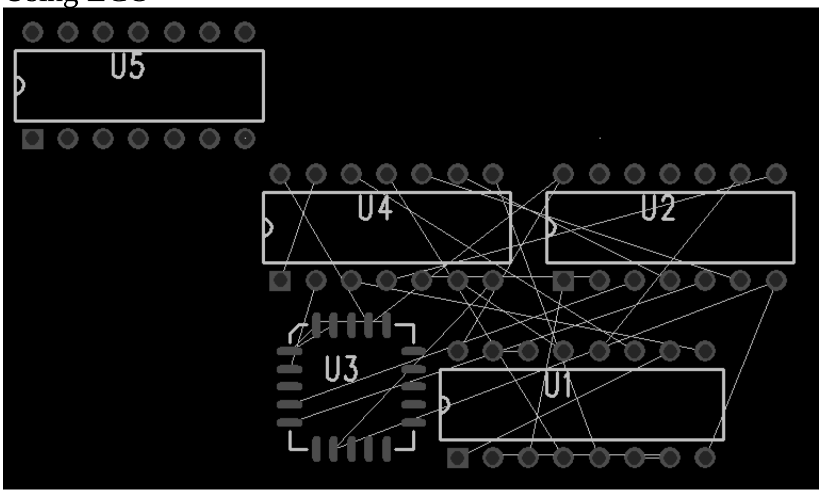
After Change the design properties and autorouting again.



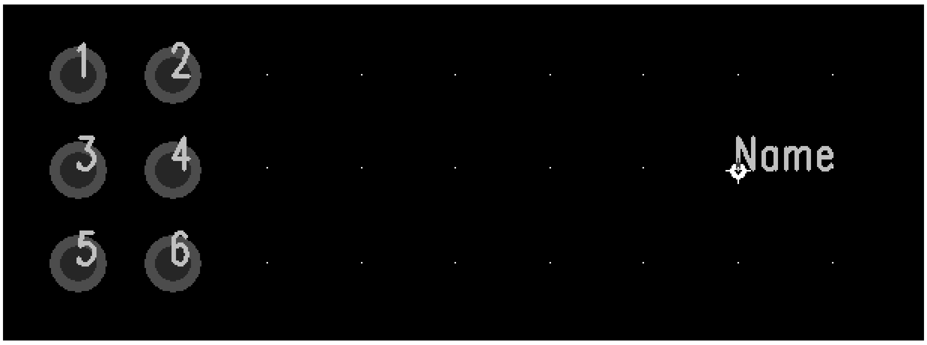
Exercise 1
Disperse Components



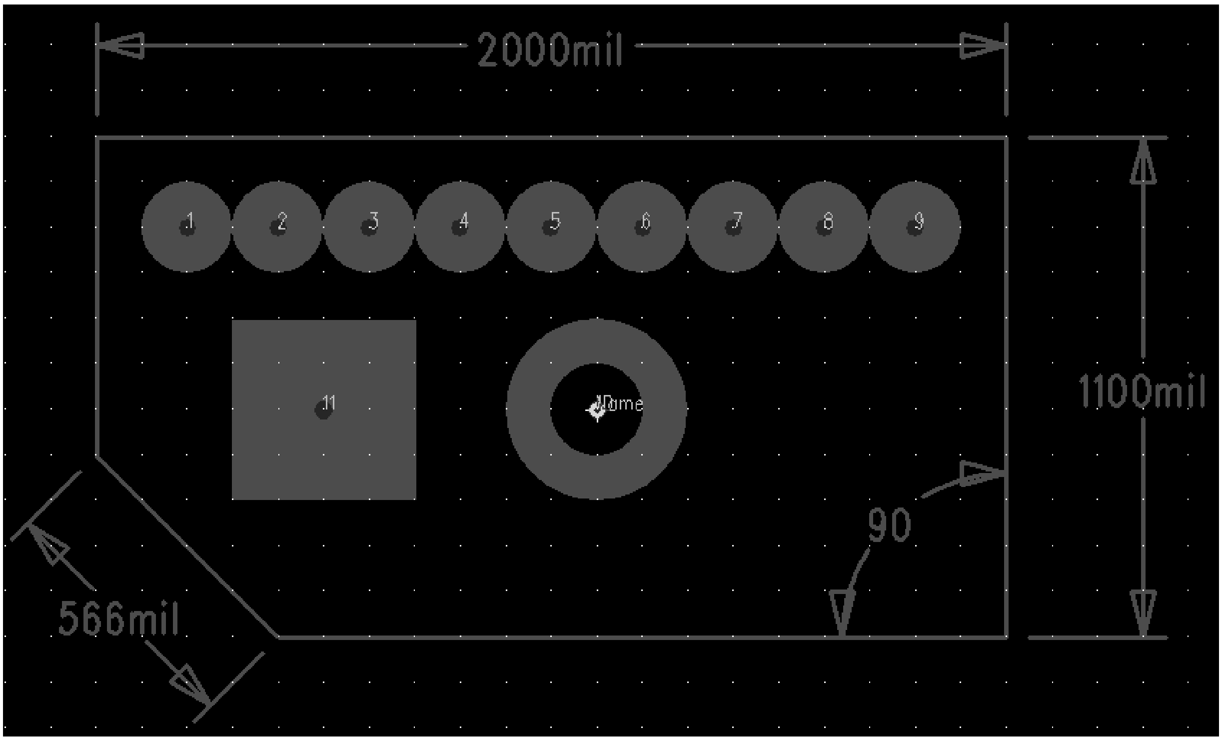
Exercise 2
Using ECO



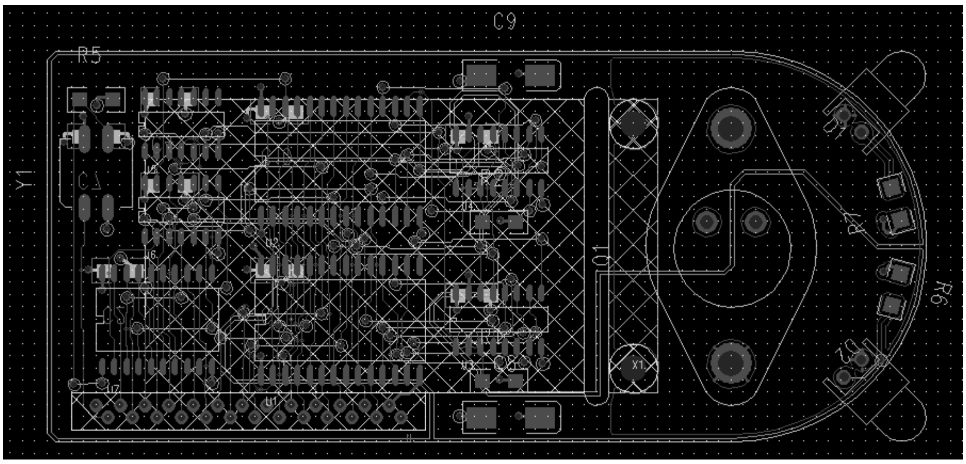
Exercise 3
Add new Decal



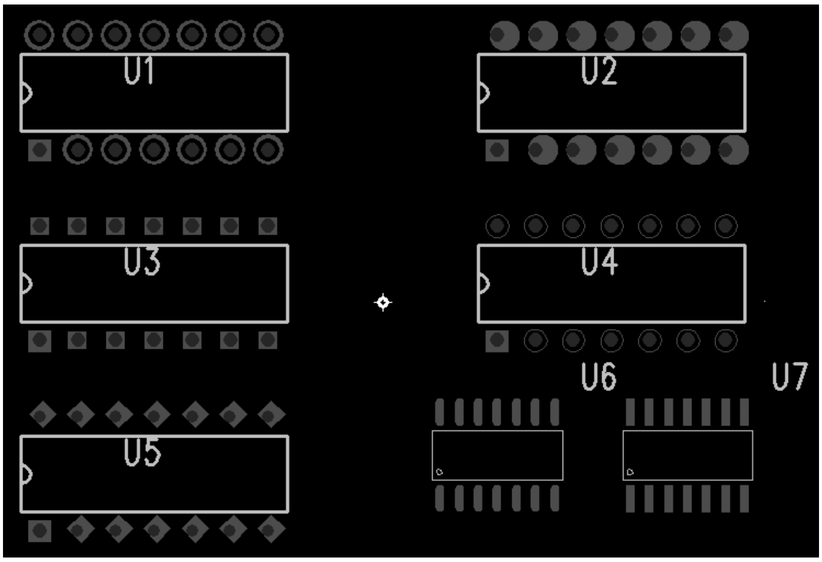
Exercise 4
Add new Decal with setting grids



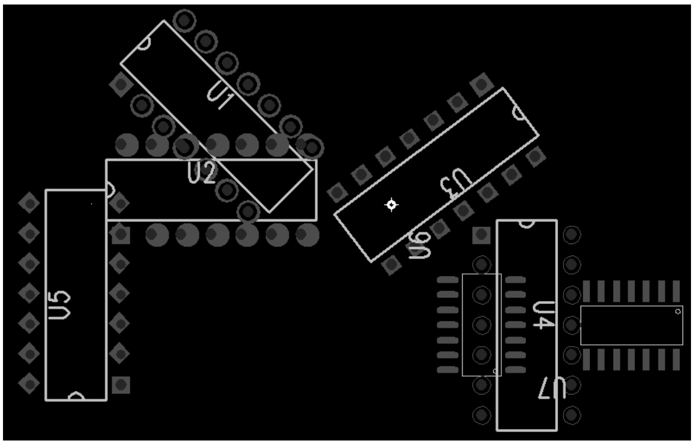
Exercise 5
Selection Filter



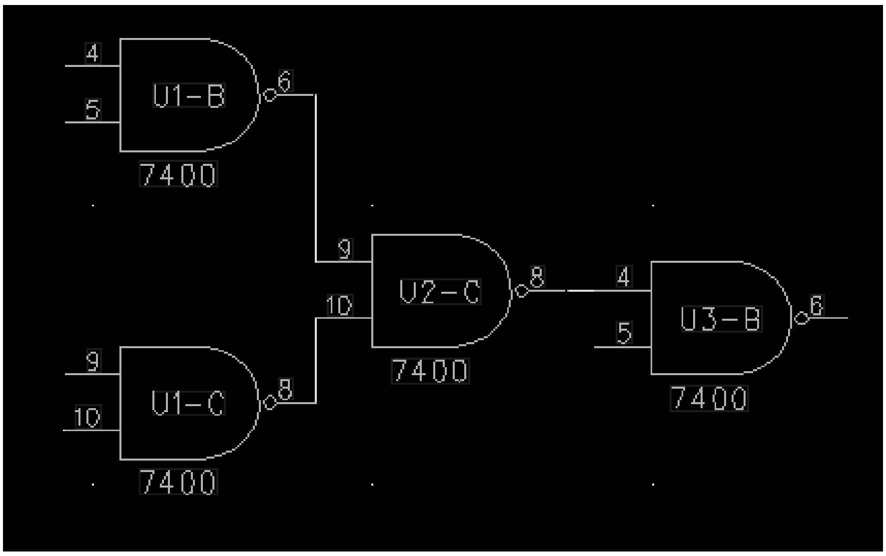
Exercise 6
Change the pads



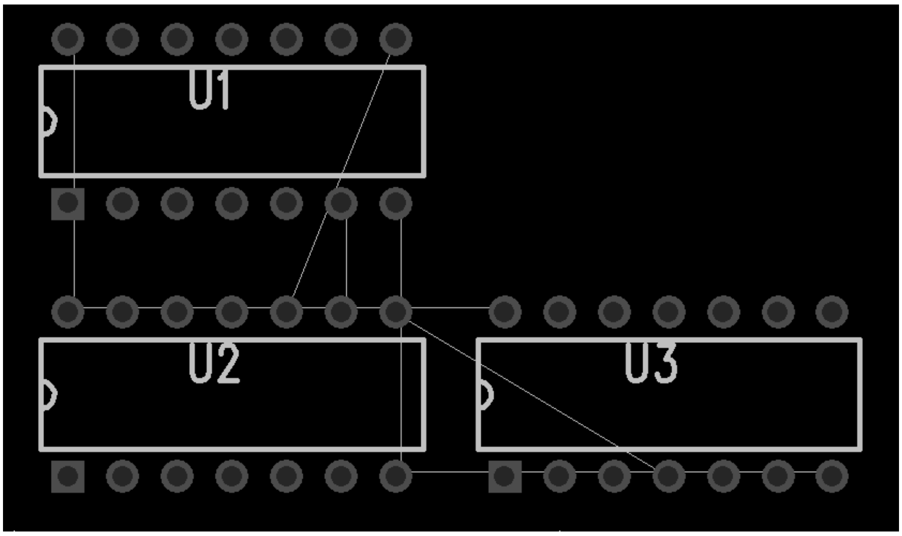
Exercise 7
Rotating Components



Exercise 8
Manual Route



Exercise 9
Dynamic Route



Exercise 10
Flooding the Pour Outline

Exercise 11
Adding the Dimension

Exercise 12
Printing the PCB