eagle tutorial

Department of Electrical and Electronics Engineering

SSN College of Engineering

ERF I-Cell/EEE

(Electrical Research Fraternity)

TABLE OF CONTENTS

EAGLE CADSOFT PROFESSIONAL

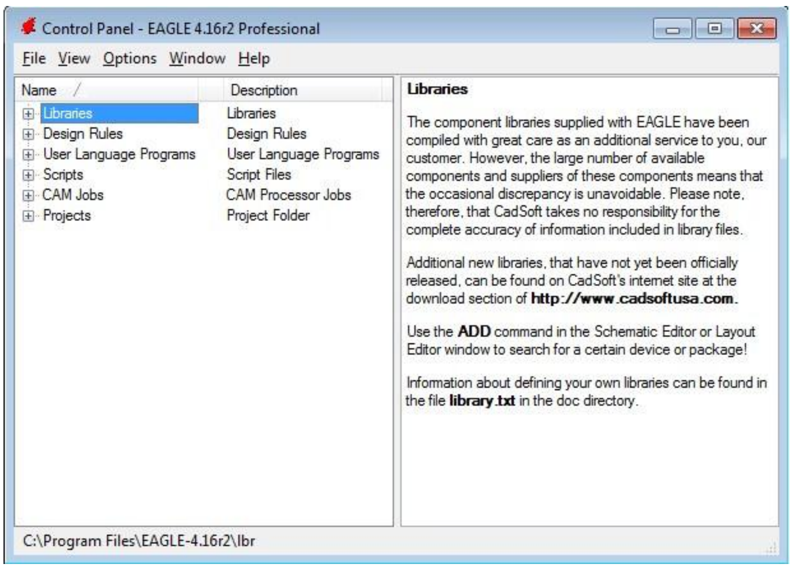
EAGLE (**E**asily **A**pplicable **G**raphical **L**ayout **E**ditor) is a schematic capture and PCB layout tool for hobbyists and DIY enthusiasts. EAGLE is a popular choice as it is an easy to use, powerful and affordable schematic capture and printed circuit board design package. Here you will learn how to get started, how to use the interface, and how to design a board file from a schematic and export it for hardware use.

GETTING STARTED

OPENING

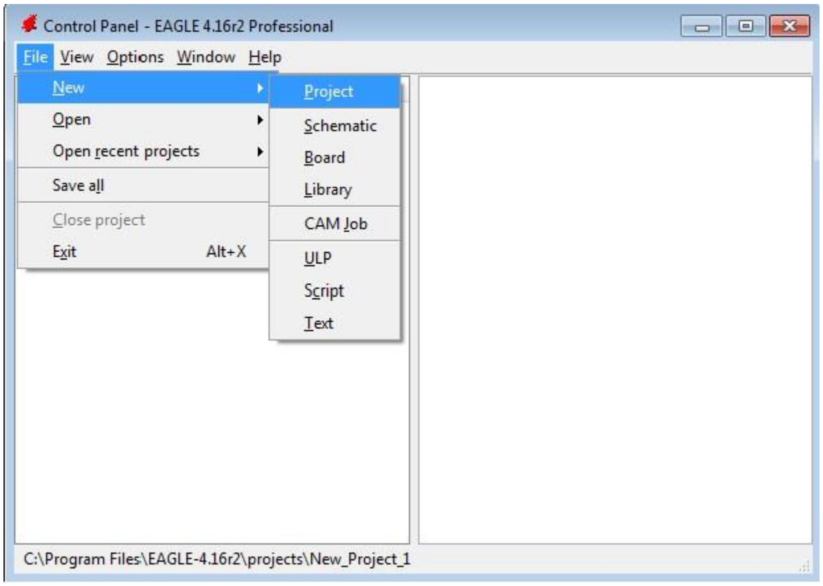
Begin by opening EAGLE CADSOFT PRO. Click Start>Programs>EAGLE Layout Editor 4.16r2>EAGLE 4.16r2, or click on the shortcut on the desktop.

You will see the EAGLE control panel.

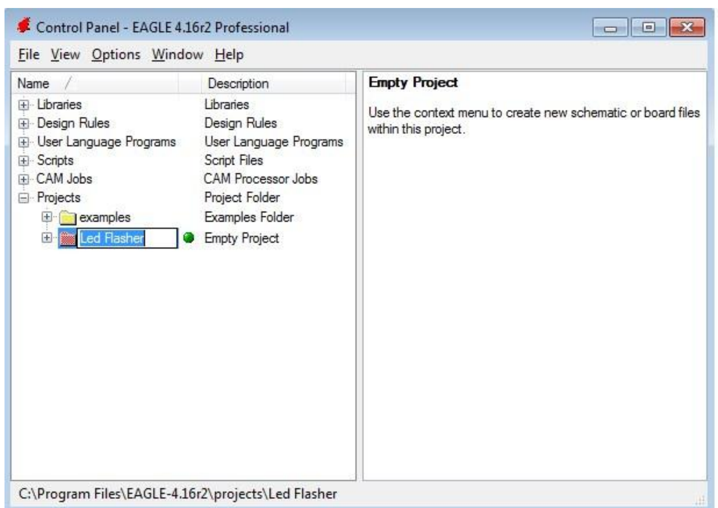


Now we need to create a Project which will contain the Schematic and the Board File. For

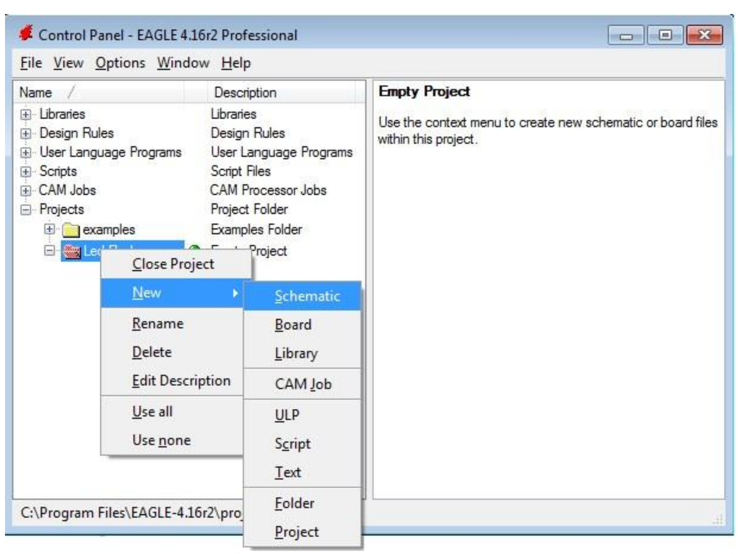
this click on File>New>Project



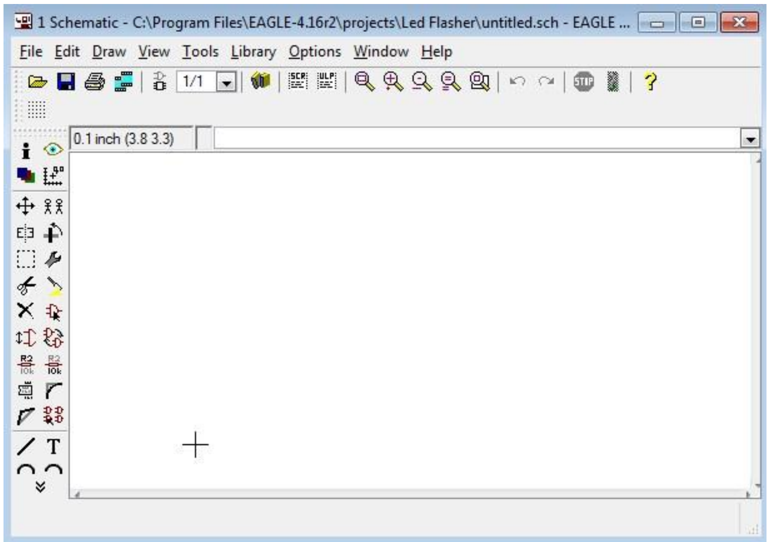
Rename the Project to the desired name.



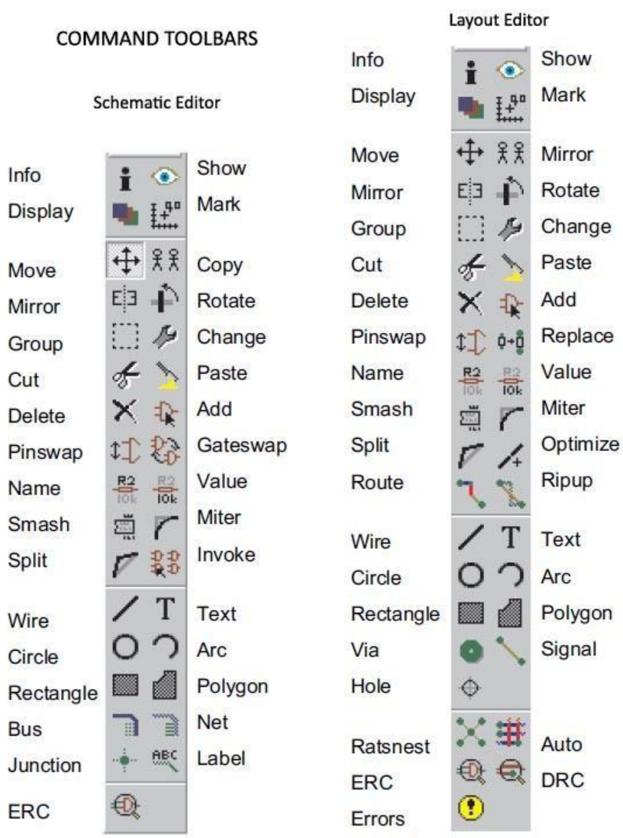
Now right click on the Project, select New>Schematic.



You will be able to see the Schematic window which is used to draw the desired circuit.



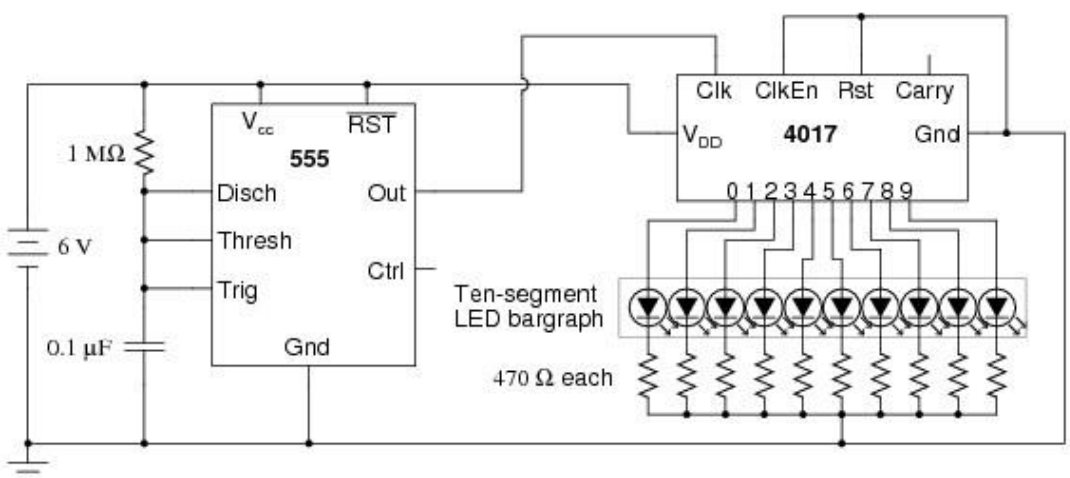
Let us also look at the various Toolbar Buttons available in the Schematic editor as well as the Layout editor.



Tool	Usage	Tool	Usage
Info	Provides information when clicked on something	Net	Used to draw wire connections between component pins
Display	Used to show/hide various details in schematic and Layout	Wire	Same as above
Add	Displays the component library	Junction	Used to place junctions in wire connections
Gateswap	Used to change pin sequence in ICs	Route	Used for manual routing of tracks
Name	Used to change name of the component		
Value	Used to mark the value of the component	Ripup	Used to break established track connections
Miter	Used to change perpendicular lines to smooth edges	Via	Used to place Via points
Split	Used to change net shapes	Auto	Autorouter feature
Invoke	Reveals Power Ports for select ICs	DRC	Used to Load and check design rules
Text	Used to insert text	Errors	Displays errors if any

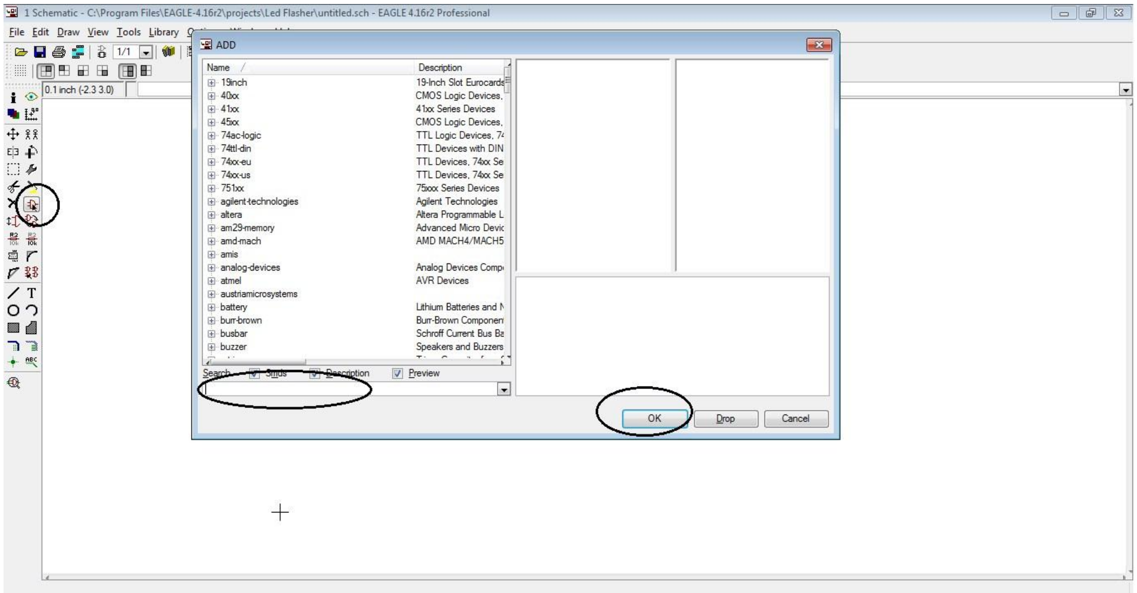
CREATING THE SCHEMATIC

In this discussion, we will be creating the following circuit. It is a common LED sequencer circuit which lights up a series of 10 LEDs sequentially.



Source: http://www.allaboutcircuits.com/vol_6/chpt_7/6.html

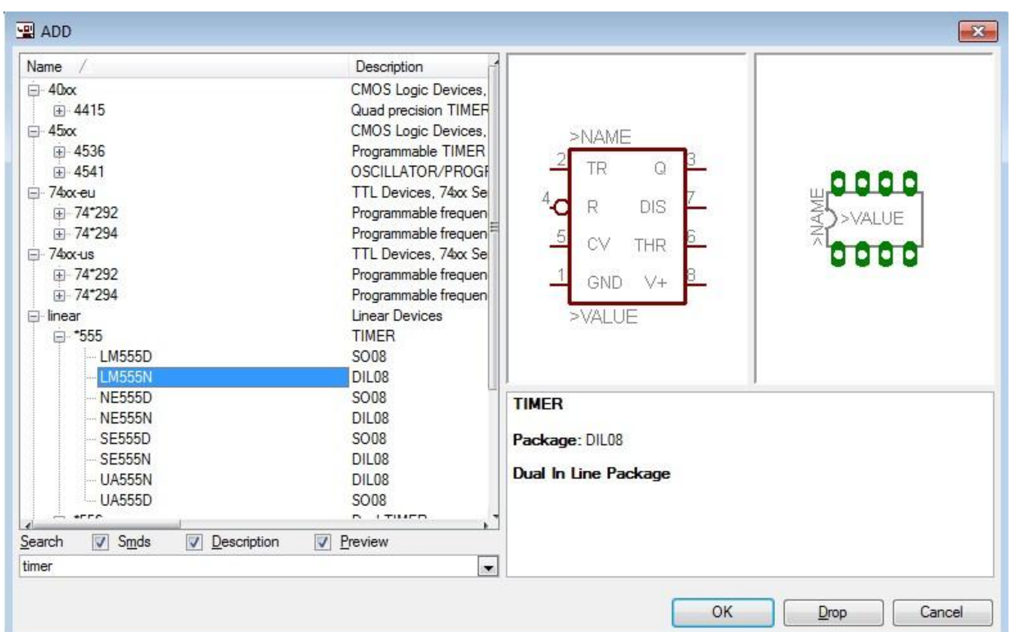
Now in the schematic window click on the ADD button in the toolbar, a window pops up. In that use the search bar to choose the component that is required. Almost all the components in the market are available in this library but the names of devices have to be specific or else a result may not appear. Click on the OK button once the component has been chosen.



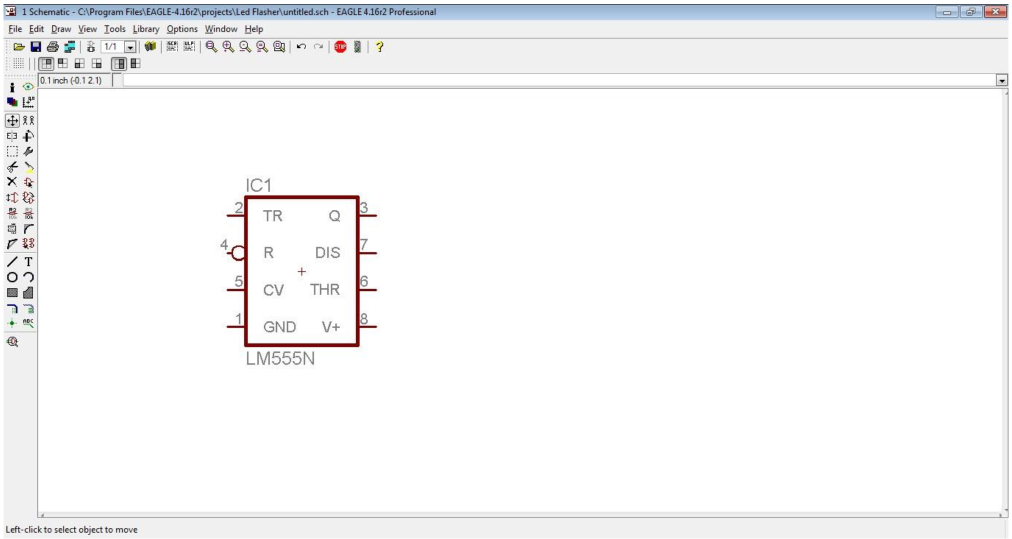
CHOOSING A 555 TIMER

For example to get a 555 timer, type “timer” in the search bar and you will get multiple results for the IC. Choose LM555N if your IC is a dual in-line package (the most common ones) or if it’s a surface mount device, choose LM555D.

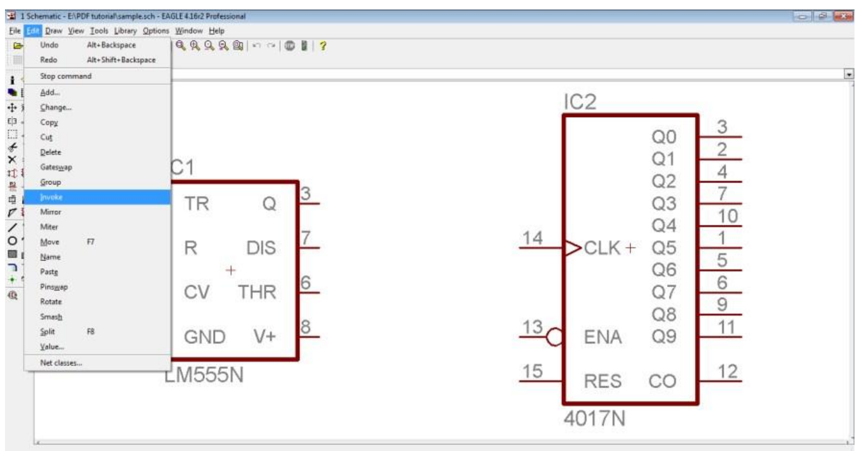
DIL08 stands for Dual In-Line 8 pin IC, likewise SO08 for Small Outline 8 pin IC (SOIC : Small Outline ICs)



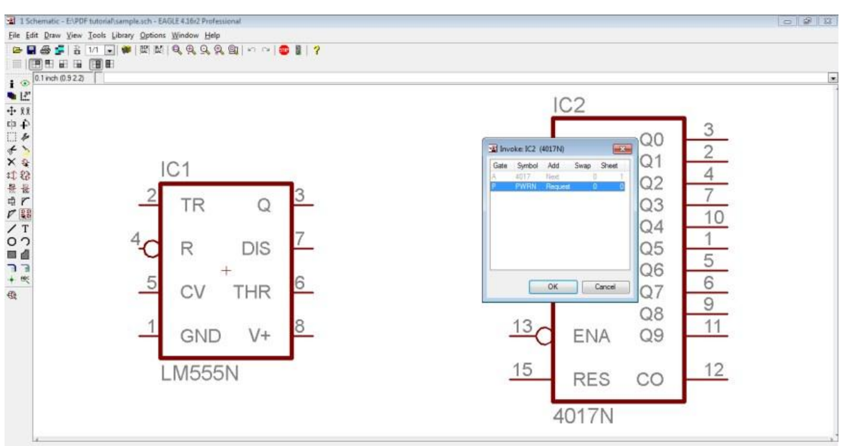
Now drag the component onto the workspace and place it in the white area.



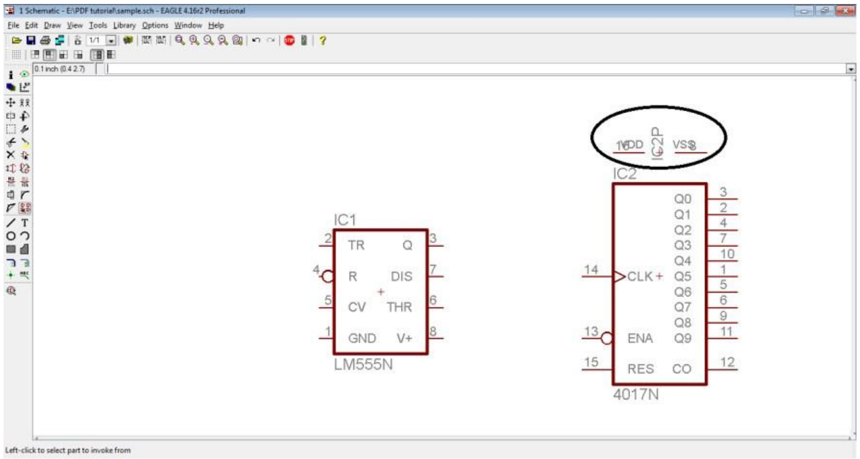
A few components like the 4017N decade counter may not show its Power pins (VCC and GND). For such ICs we have to reveal those connections. To do this click on Edit>Invoke.



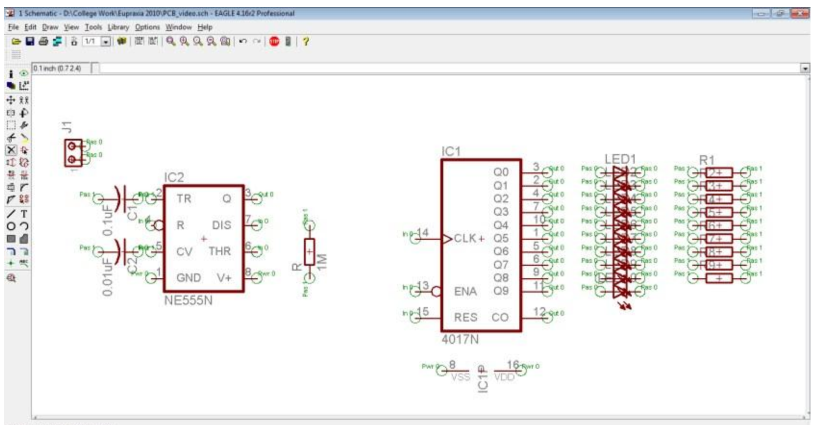
Click on the space inside the IC, the Invoke window will pop up. Select the field inside and click OK.



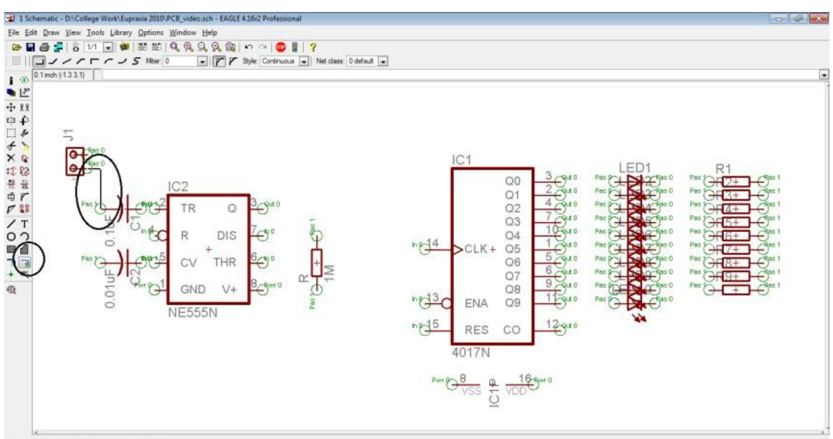
Now the Power pins will be displayed, place it in any convenient location near the IC. All connections to VCC and GND of this IC should be given to these two pins.



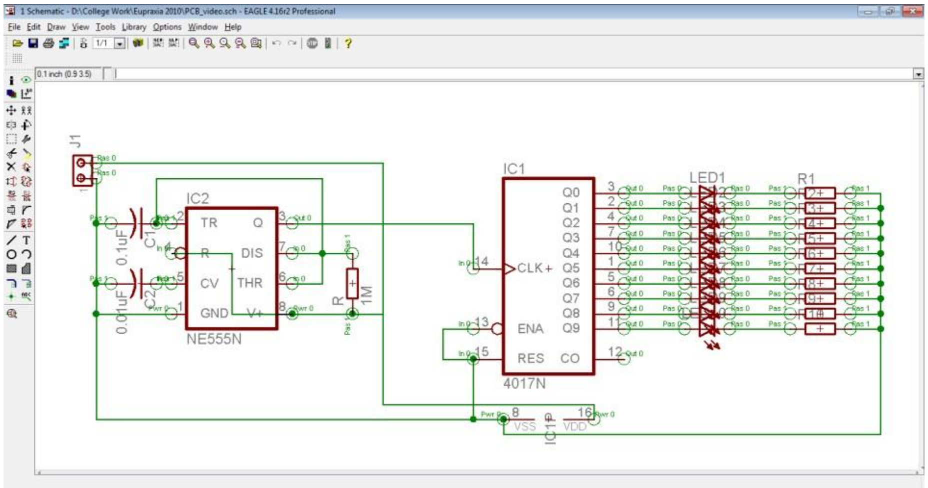
Now in this way add the rest of the components required in the LED Sequencer circuit using the ADD Component Library.



Now we need to connect the components using wires or "nets". For this select the Net tool from the Toolbar. Now click on one component pin and stretch the wire to the other component pin and click again to fix the wire between the two pins.

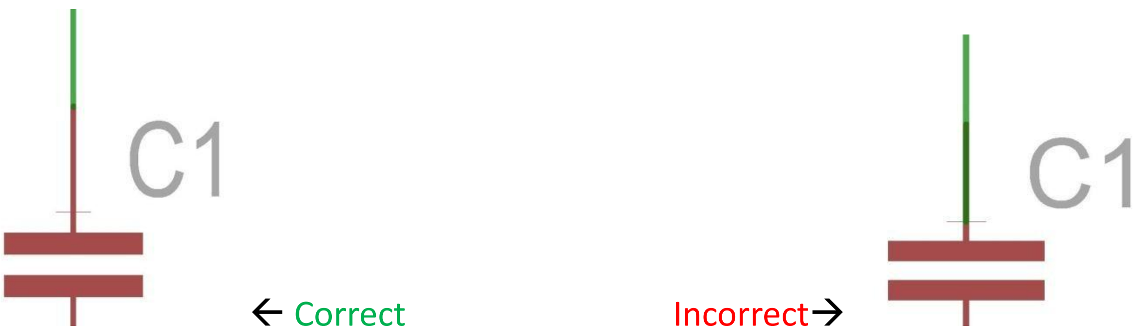


After all the wires connections are established, the schematic should look like this. Neat perpendicular connects always makes sure that there is no confusion in the wiring or any accidental connections.



NOTE:

1. Care should be taken while connecting the wires to the components. You should connect the wire exactly to the edge of the terminal as shown below:



2. If you come across any junction you need to indicate the same using Junction  button.

3. Make orthogonal connections that de-clutter the schematic and make it look organized.

4. After giving connections ensure that all required wiring is established. This can be checked by selecting the Move tool and by clicking and dragging the symbol to see if the pins/leads are connected to the wires.

With the above precautions connect the components.

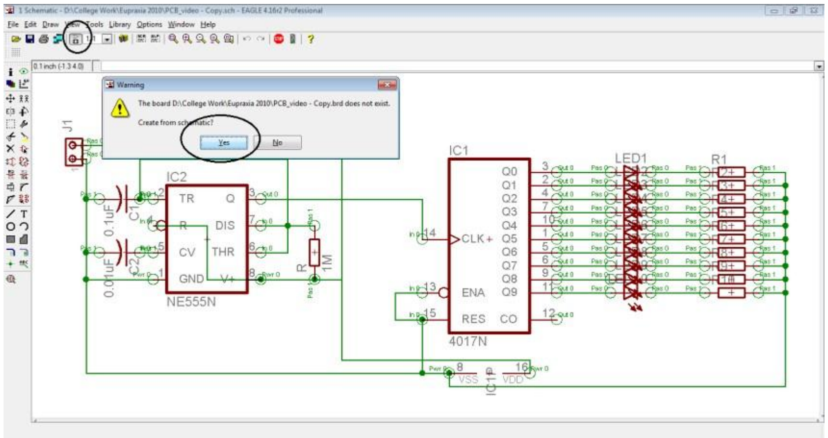
This table will provide you the guideline to add some common components:

#	COMPONENT	KEYWORD	GUIDELINE	DESCRIPTION
1.	Berg strip/Connector/Push or Pull Switch	connector	Scroll down to MTA02-100	'MTA' signifies type of connector, '02' signifies number of pins in the connector and '100' signifies the size of the connector. If you want 3 pin select MTA03-100.
2.	Resistor	resistor	Expand R-EU_ and select R-EU_0207/10.	'EU' indicates European style resistors '0207' signifies resistor type 10 signifies grid length. If you want Vertical mounting style select REU_0207/5V or R-EU_0207/2V. 'V' indicates vertical mounting type resistor
3.	Capacitor	capacitor	Expand C , select C2.5/5	'2.5' signifies the area and '5' signifies the grid length
4.	Diode	diode	Select 1N4004	You can use the same for 1N4007 also
5.	Bridge rectifier	rectifier	Scroll down to RB1A and press OK	
6.	LED	led	Scroll down and expand LED. Select LED5MM	5MM LED is the usual LED available in market.
7.	Potential Divider/Trimm Pot	trimm	Expand R-TRIMM and select R-TRIMM64W	
8.	OP-AMP	op amp	Expand *741 and select LM741P	
9.	PIC Microcontroller 18F4550 (40 pin)	microchip	Scroll down and expand PIC18F4*_20 and select PIC18F4550_40	'40' signifies 40 Pin IC
10.	PIC Microcontroller 18F2550 (28 pin)	microchip	Scroll down and expand PIC18F2*_28 and select PIC18F2550_28DIP	'28' signifies 28 Pin IC and 'DIP' is abbreviation for Dual In Package

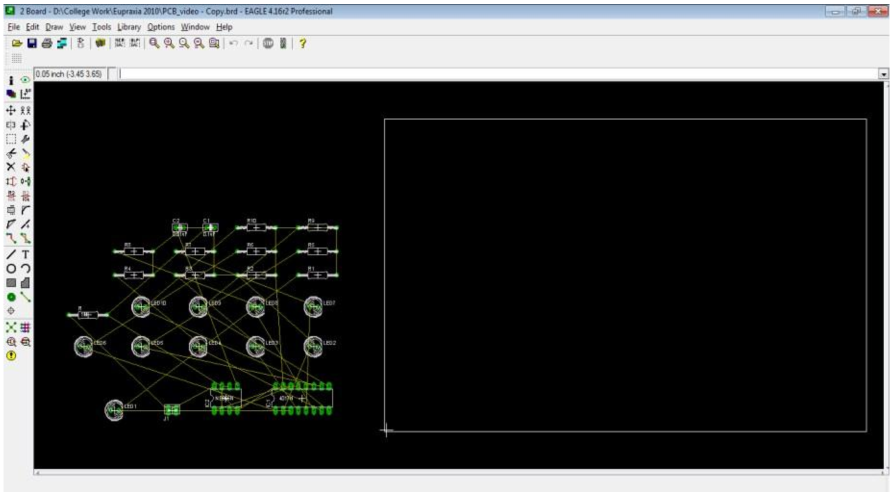
NOTE: If you cannot find the required component you can create your own library file of the component which will be discussed later.

CREATING THE LAYOUT

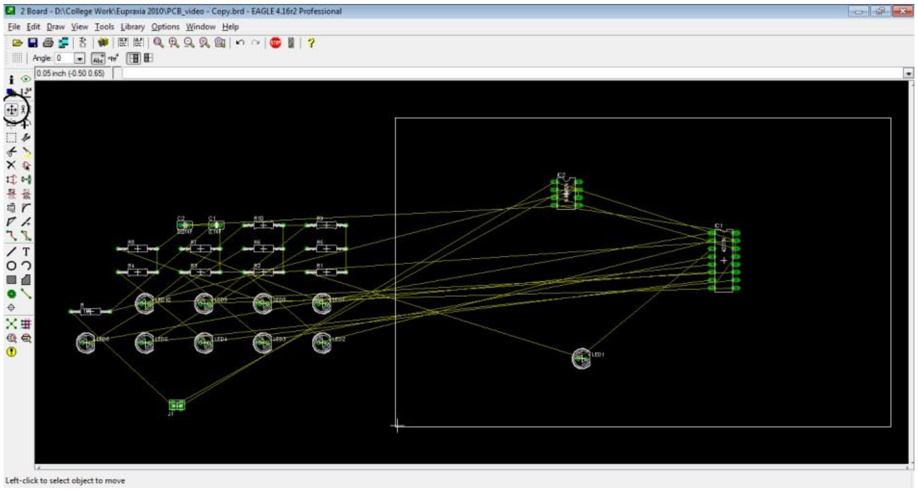
After all the connections are given, we create the “board” file. This is done by selecting the “create board file” tool in the upper toolbar. An info box appears, Click YES to proceed.



This is the Layout window. The components and the workspace frame are shown. These components have to be optimally arranged inside the workspace frame which will be the actual placement of the devices in hardware.



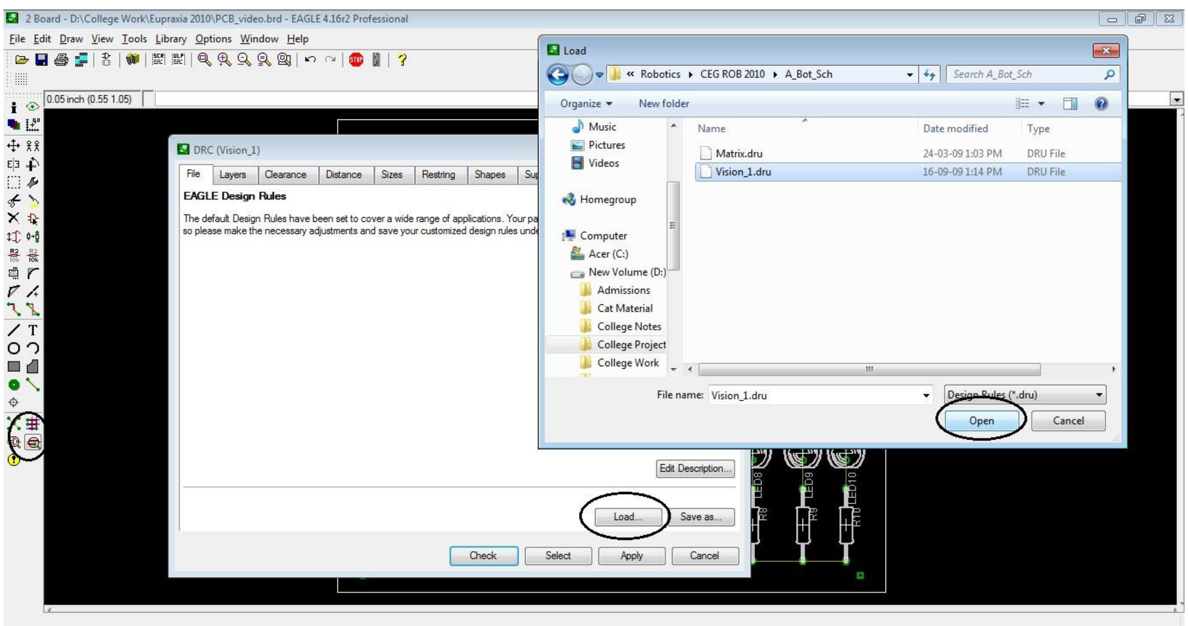
Drag and drop the components onto the workspace frame as shown below.



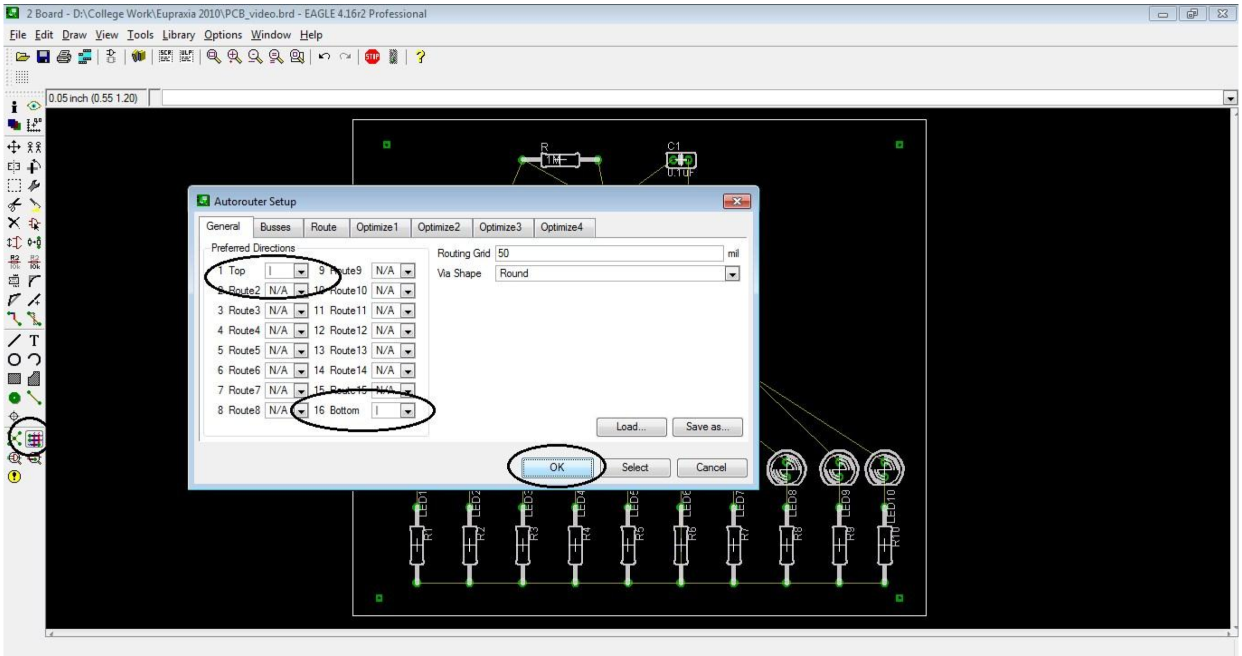
Now we need to load the standard DRC settings file which will determine various factors of the wires such as

- Track thickness
- Track spacing
- Track routing

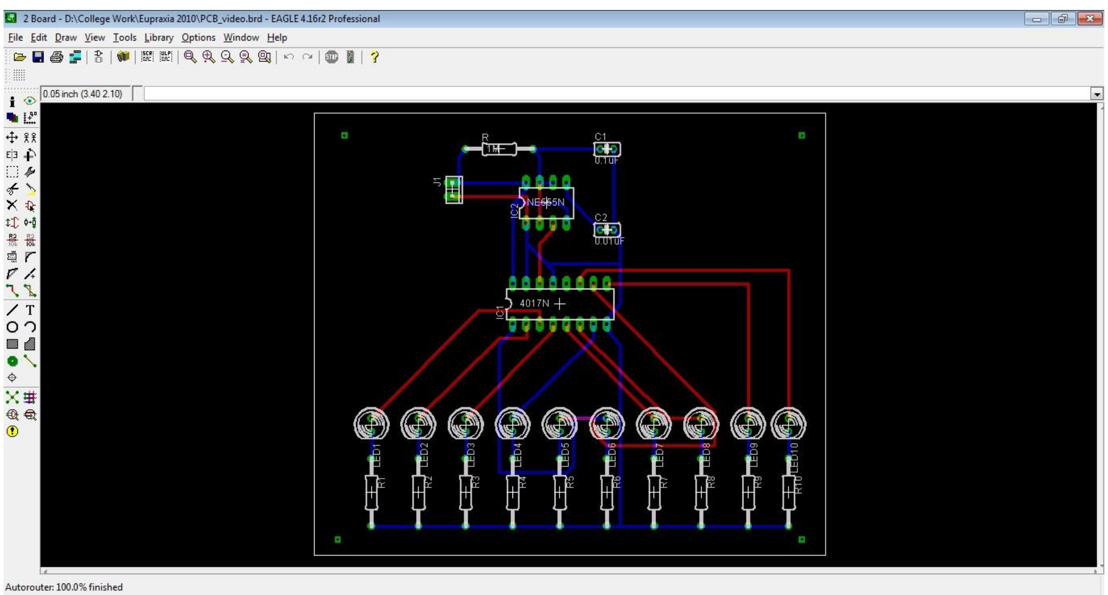
Click on the DRC tool and the Load button in the DRC window. Load the DRC file (provided in the CD) and click open. Click on the Check button and ignore errors if any.



In the next step, we need to lay the tracks or otherwise “route” it. To do this, select the Autoroute tool from the toolbar. Select the “|” option in both top and bottom fields since this is a double sided PCB. (For a single sided PCB select N/A in the top field and set only the bottom field). Press OK when done.



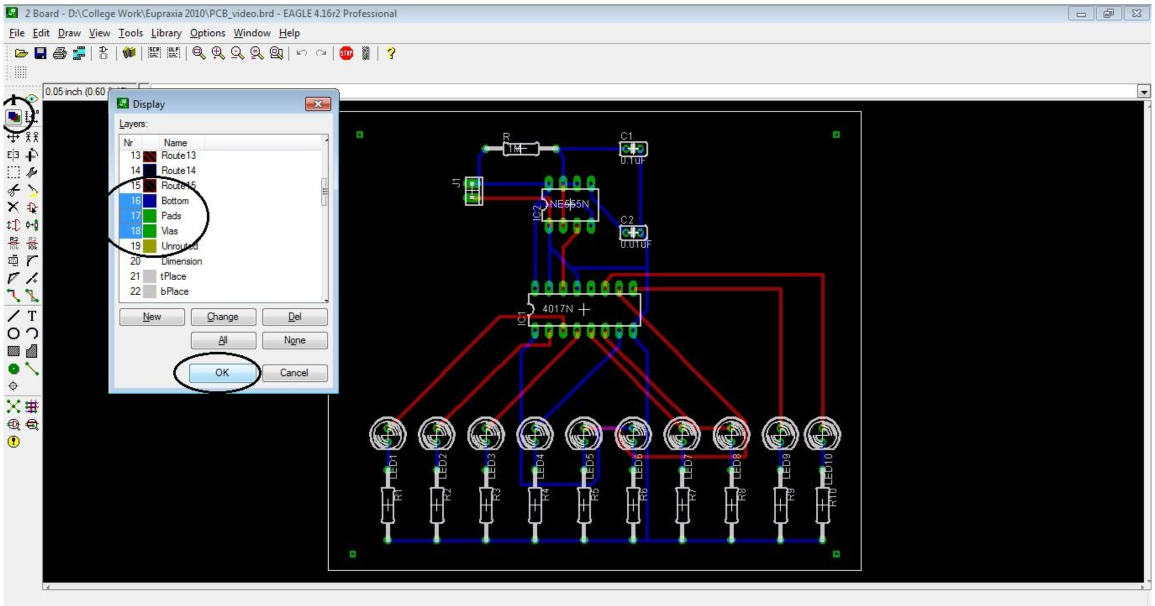
After autoroute completes, the board file will look like this. The blue lines represent the bottom side tracks and the red lines are the top side tracks. The autorouting is a programmed feature in EAGLE which optimally selects the best path for a track (though sometimes the tracks may unnecessarily be elongated) using search optimization algorithms.



EXPORTING THE IMAGE

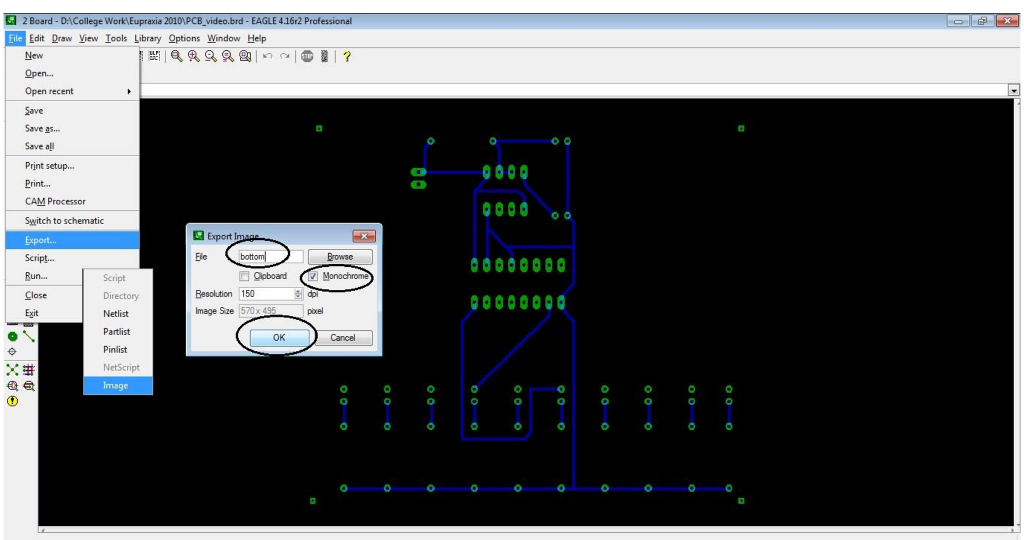
From this board file we can export images of the top side and bottom side separately for taking print. This can be done as follows.

Select the display tool and in the display window select the labels called “bottom”, “pads” and “vias” and click OK.

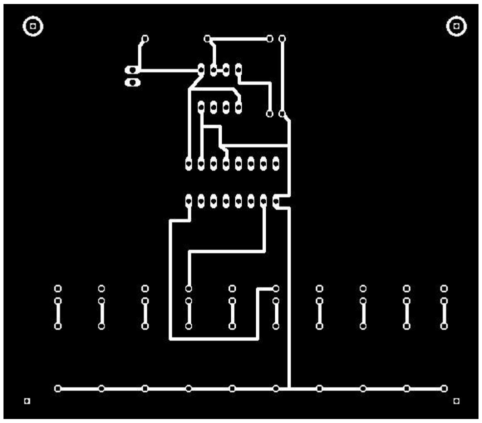


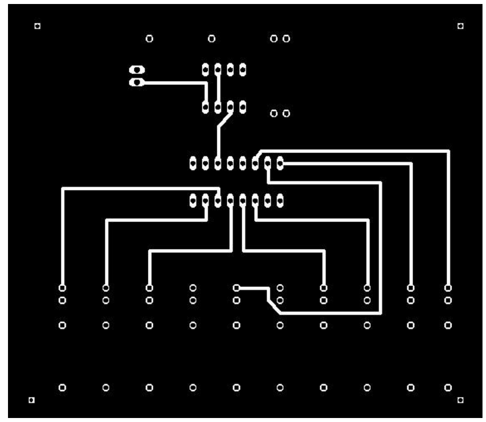
The board file now looks like this. To create an image of this bottom side track pattern click on File>Export>Image. In the Export Image dialog box

- set resolution to 150 dpi
- check monochrome
- Provide a file name (“bottom”) and press OK.

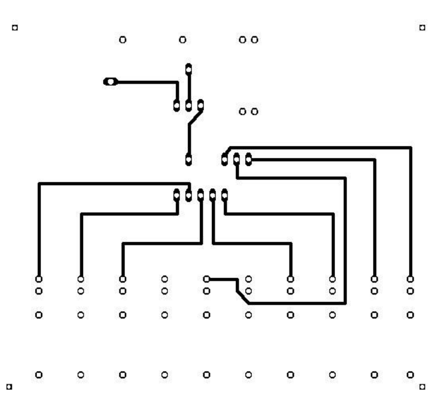


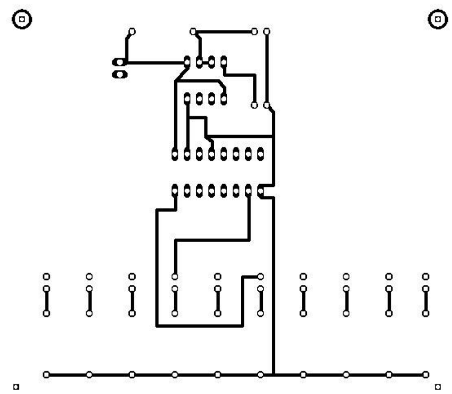
Repeat the same for the Top side by deselecting "bottom" in the Display tool and selecting Top in its place. The exported top and bottom side images should look like this.





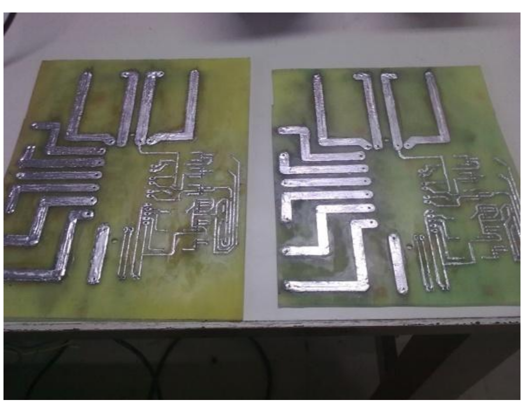
Use an image editor such as MS Paint to invert the colors and to flip the top side image laterally.



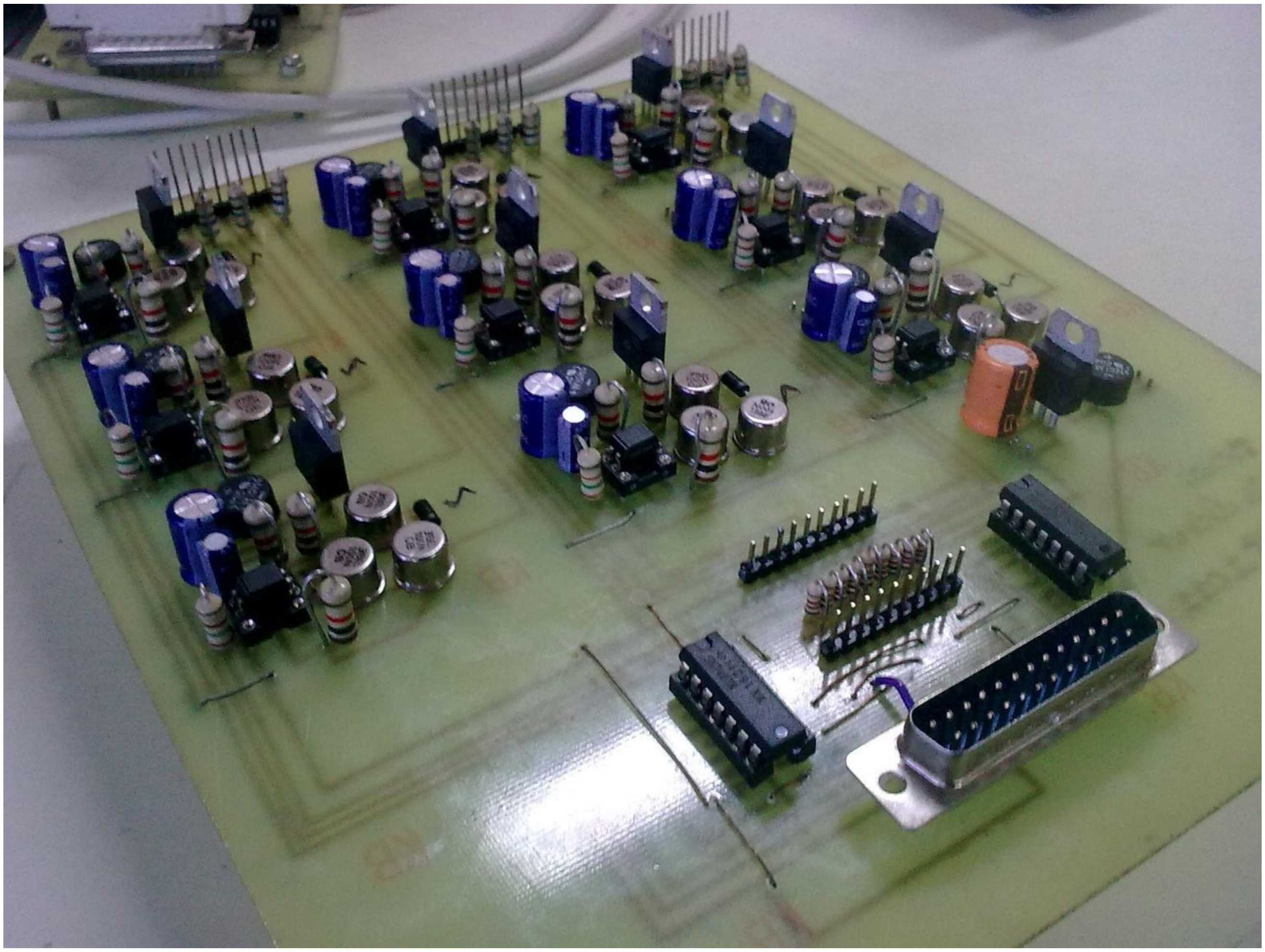


Take a printout of these two images in separate papers to get the final board file images for the two sides of the PCB. With the images we can proceed to the actual process of PCB fabrication.

Thus we have demonstrated the design of circuits using the EAGLE Schematic editor and the creation of board file images using the Layout editor which is used in the fabrication of printed circuit boards.



A finished single sided PCB that is ready to be used!



CREATING A CUSTOM LIBRARY FILE

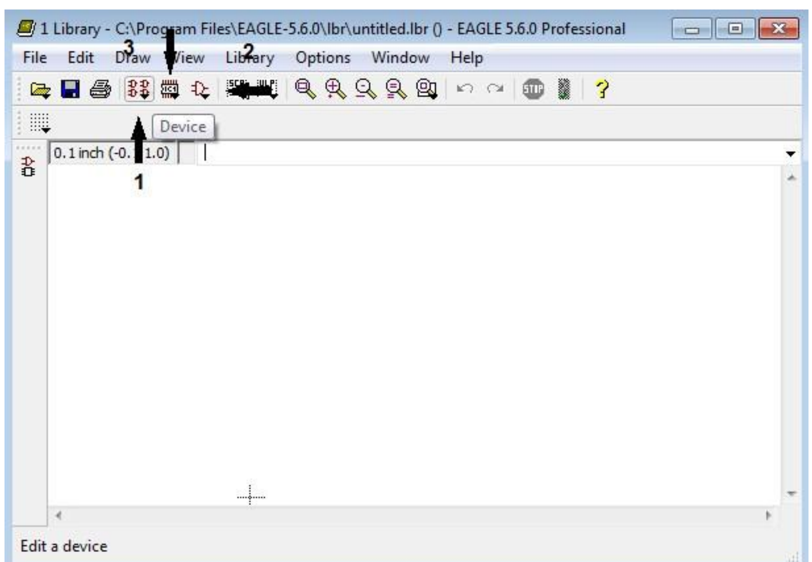
In this section, we will show you how to create a user defined component that can be used in the EAGLE schematic and layout editor. The need to create a user defined component is due to absence of certain rare components in the otherwise extensive parts library of EAGLE CAD.

Begin by selecting File>New>Library.

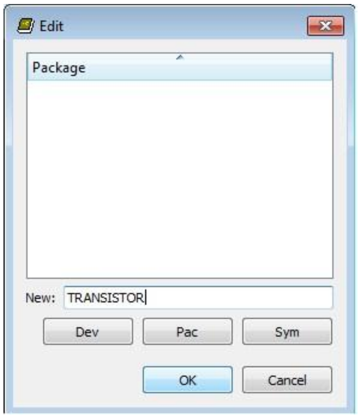
You will notice 3 important buttons:

1. Device
2. Package
3. Symbol

These define the library file; the Package is the diagram that appears in the Board layout editor and the Symbol is the diagram that appears in the schematic editor. These two combined together forms the “Device”.

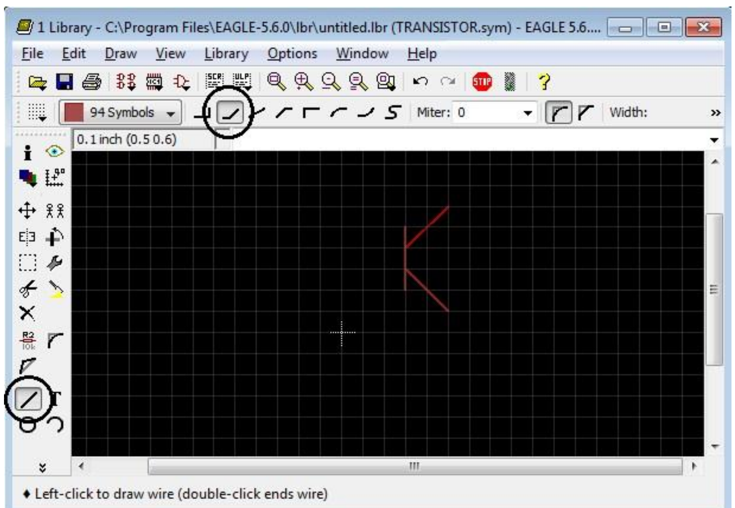


Begin by clicking on Symbol. A pop up window will appear where the name of the device has to be provided. In this tutorial we will be creating a 4 pin Transistor IC. Hence name the device as “TRANSISTOR”.

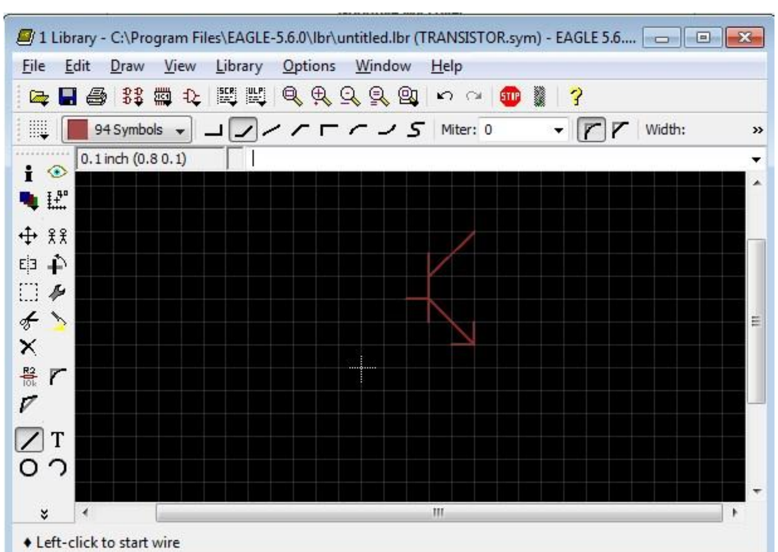


Click Yes in the Warning dialog box. The Symbol editor window now appears. Here we will be drawing the image of a standard NPN transistor.

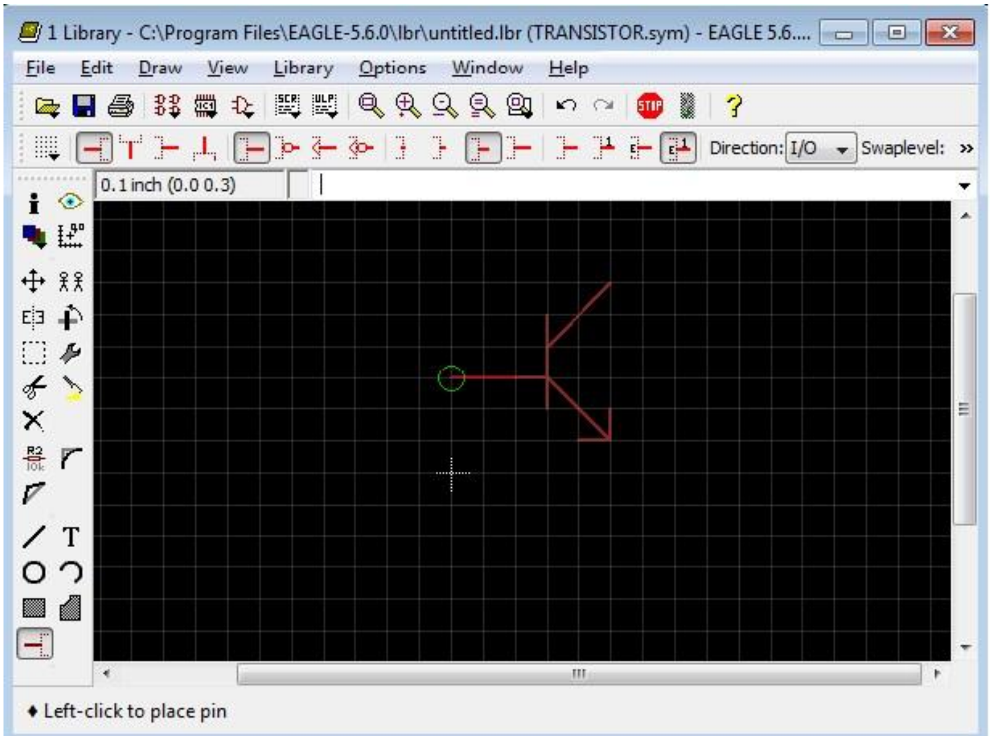
Use the Wire tool to draw the diagram as shown. For 45 degree lines select the angle button in the upper toolbar.



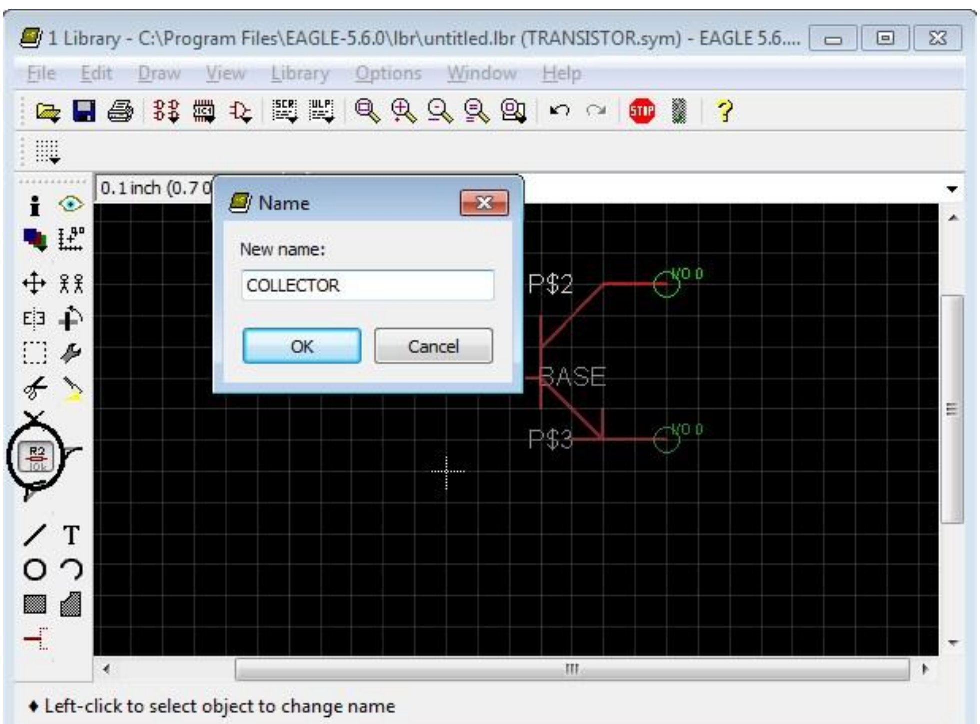
The completed image should look like this:



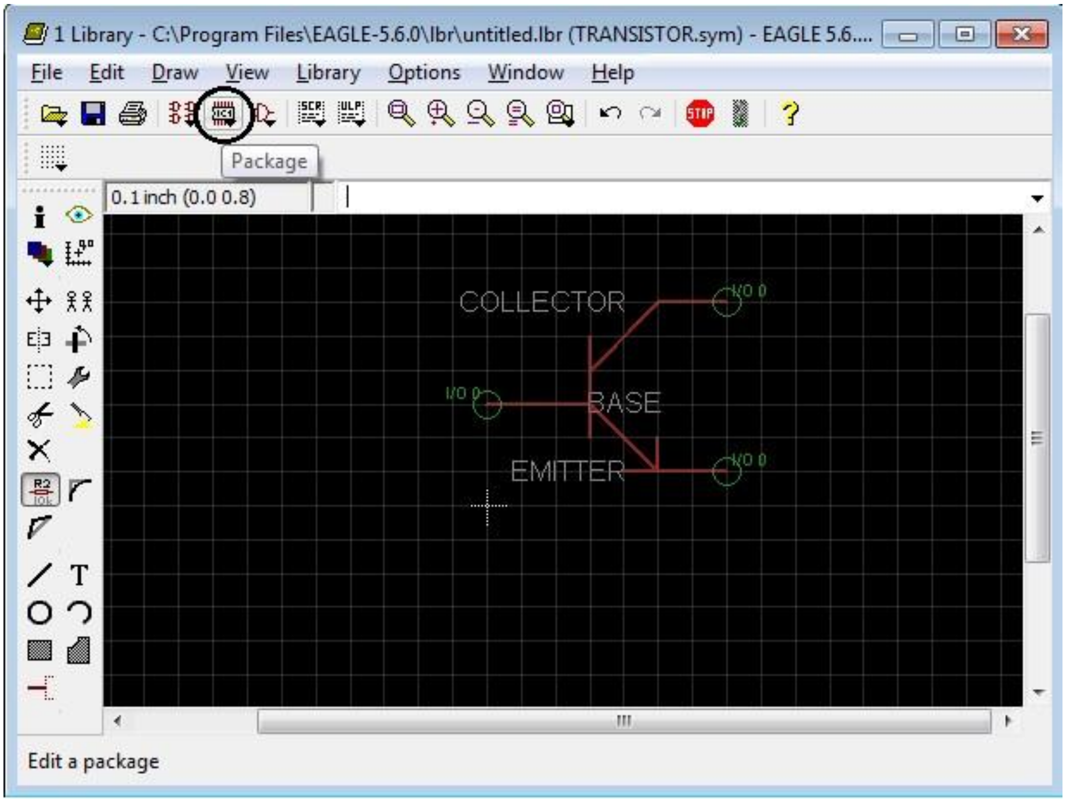
Now we need to add “Pins” to the diagram and for this select the Pin tool and add like this:



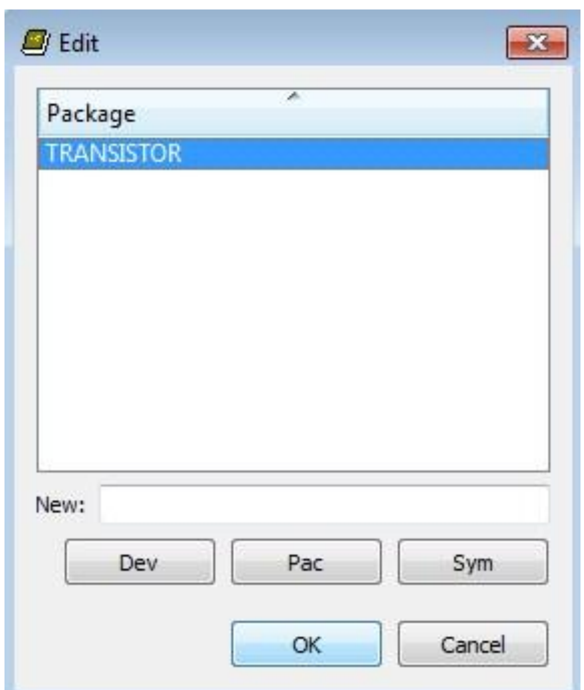
Additionally we can name the pins by selecting the “Names” tool and naming each pin separately.



Once this has been completed select the Package button to create the package for the Transistor.

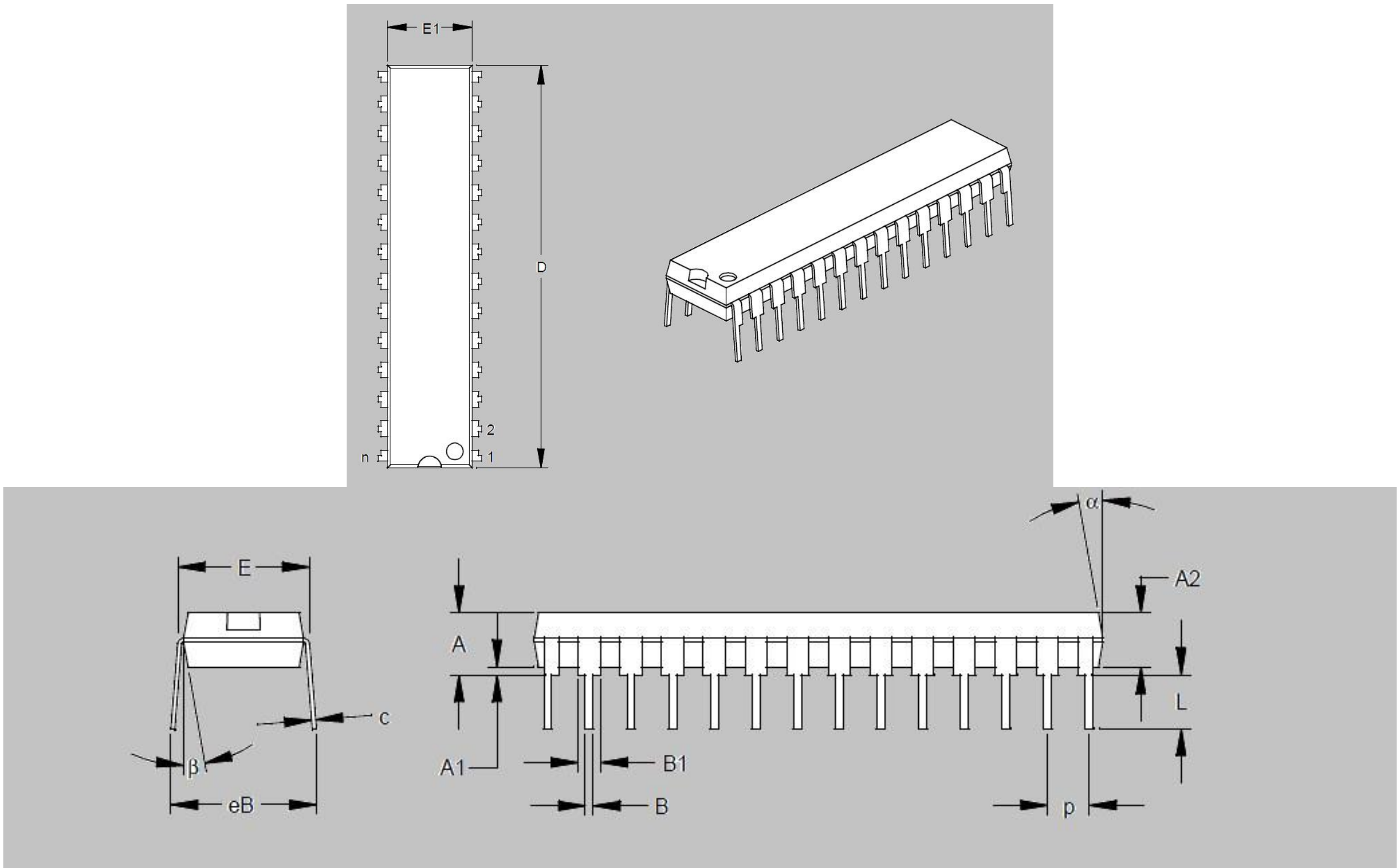


Click OK when this dialog appears:



We begin by fixing the pads for the component. This is the most important part of creating a Library file as the dimensions of the custom drawing should match exactly with the specifications provided in the component's datasheet. For example let's consider a 28 pin PIC18F2550 Plastic Dual In-Line Package (PDIP) IC.

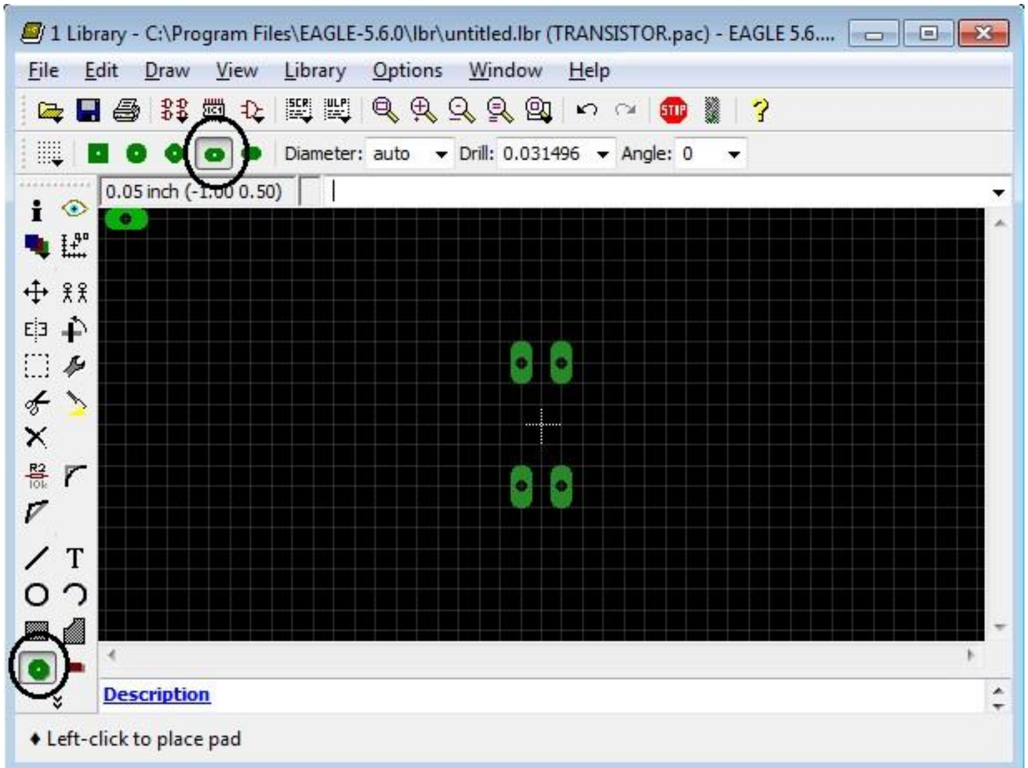
Note the values of Pitch ‘p’ and the IC width ‘E1’. These values would be the same for our 4 Pin Transistor IC, hence we can borrow them.



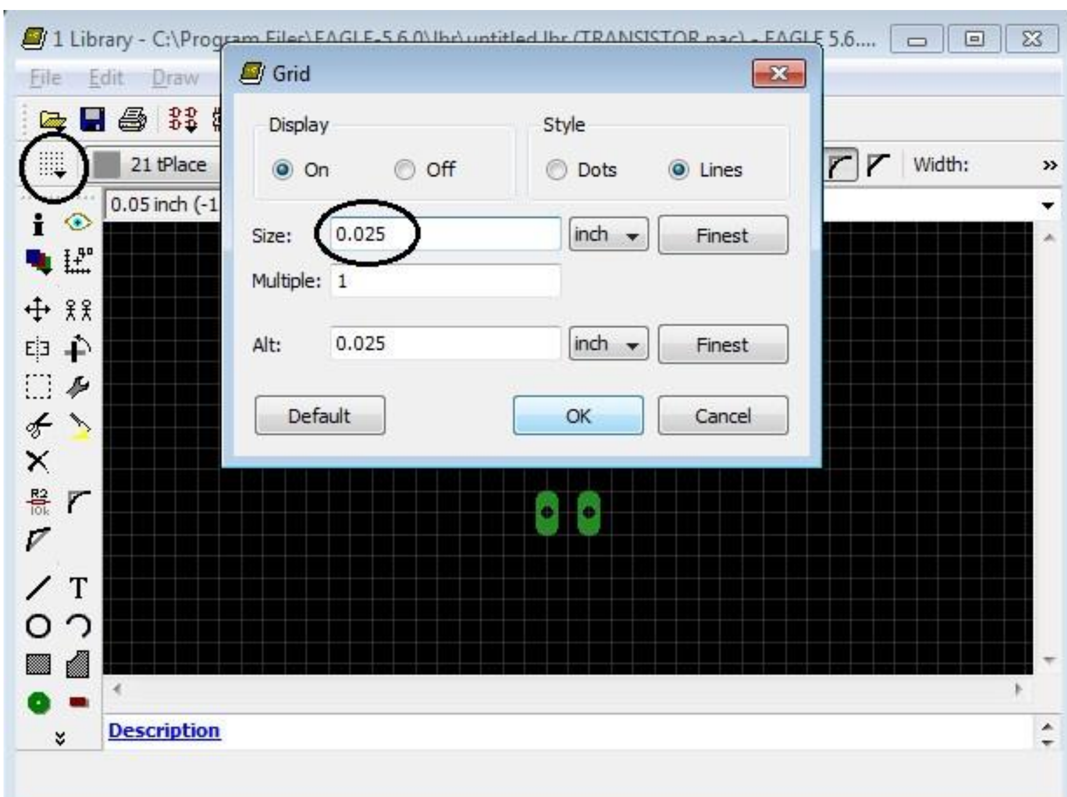
Units		INCHES*			MILLIMETERS		
Dimension Limits		MIN	NOM	MAX	MIN	NOM	MAX
Number of Pins	n		28			28	
Pitch	p		.100			2.54	
Top to Seating Plane	A	.140	.150	.160	3.56	3.81	4.06
Molded Package Thickness	A2	.125	.130	.135	3.18	3.30	3.43
Base to Seating Plane	A1	.015			0.38		
Shoulder to Shoulder Width	E	.300	.310	.325	7.62	7.87	8.26
Molded Package Width	E1	.275	.285	.295	6.99	7.24	7.49
Overall Length	D	1.345	1.365	1.385	34.16	34.67	35.18

Use the grid to create these dimensions so that the pads can be placed precisely. By default the grid is at 0.1 inches or 2.54mm. We can change this to any value that is convenient.

Now use the Pad tool to place 4 pads as shown.

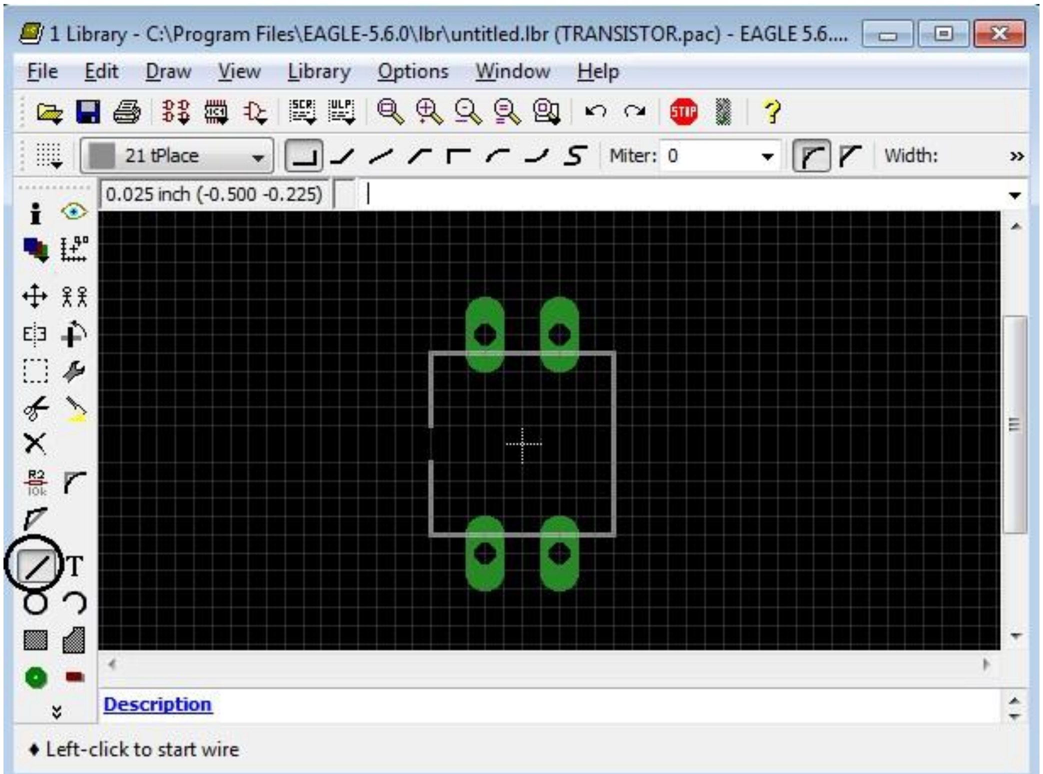


Next would be to draw the package outline. Set grid to 0.025 inch as shown.

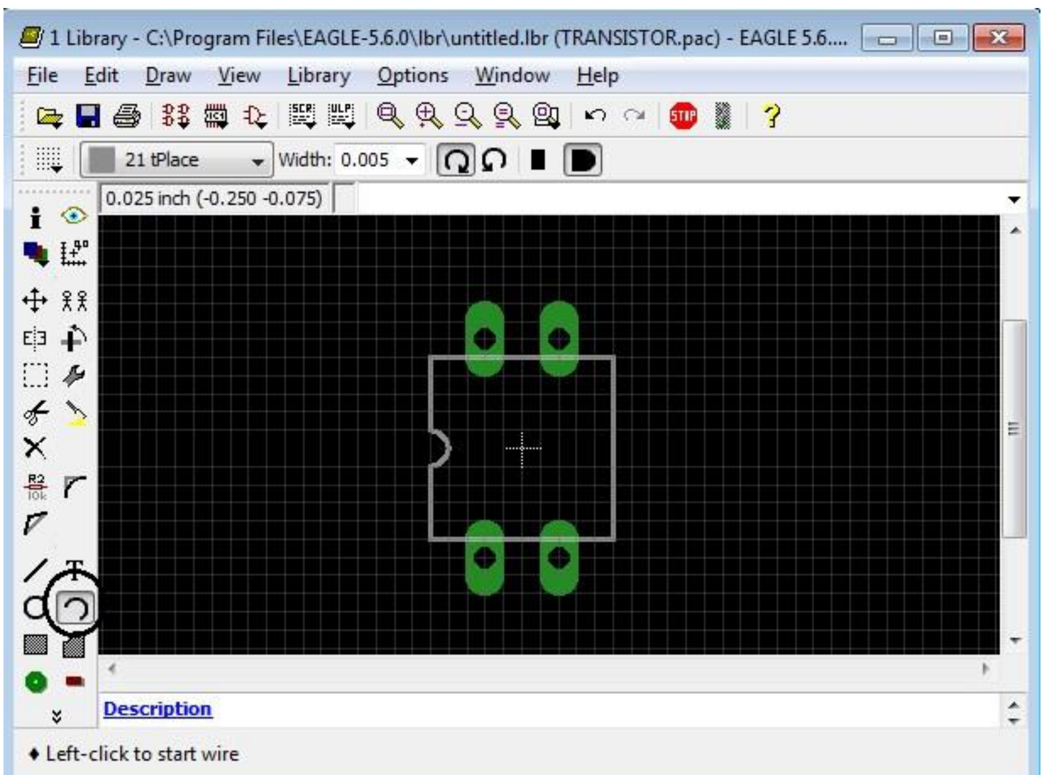


While placing the pads care should be taken to place it in the proper sequence. In this case the pads should be placed in anti-clockwise direction starting from the left bottom.

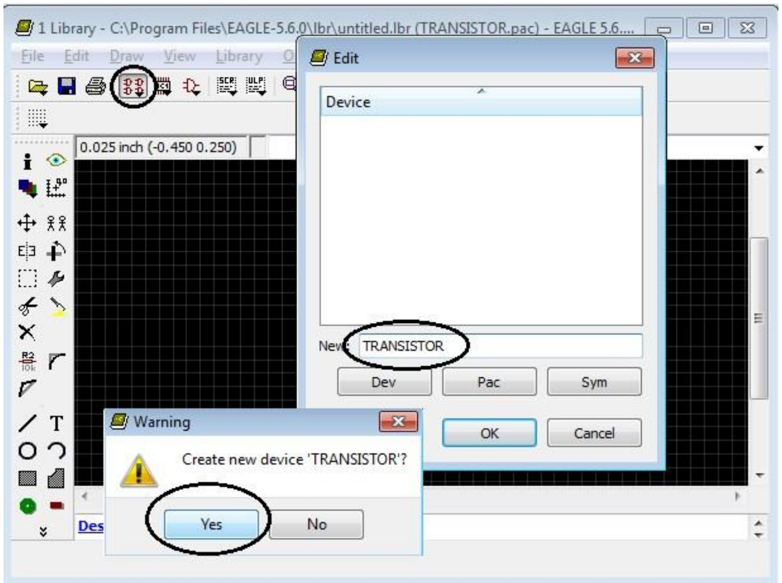
Use the Wire tool to draw the package outline. Leave a gap in the left which will be filled up with an arc that represents the "Notch" of the IC.



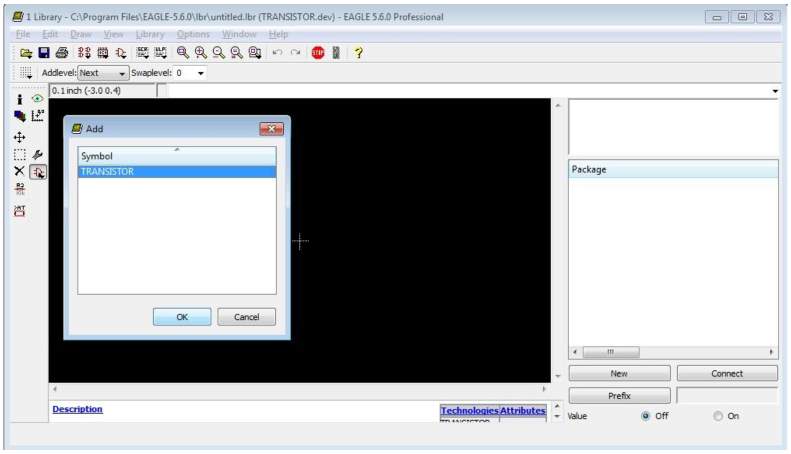
Drawing the Notch using the Arc tool:



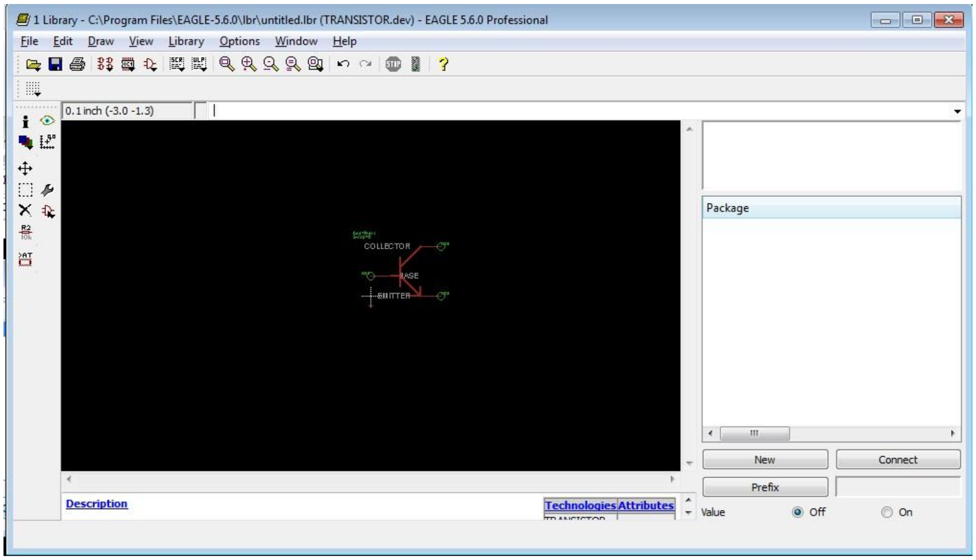
At this stage we have created both the symbol and the package diagrams using the respective editors. Now the two files have to be merged in what is known as the Device editor. Click on the button and name the Device as "TRANSISTOR".



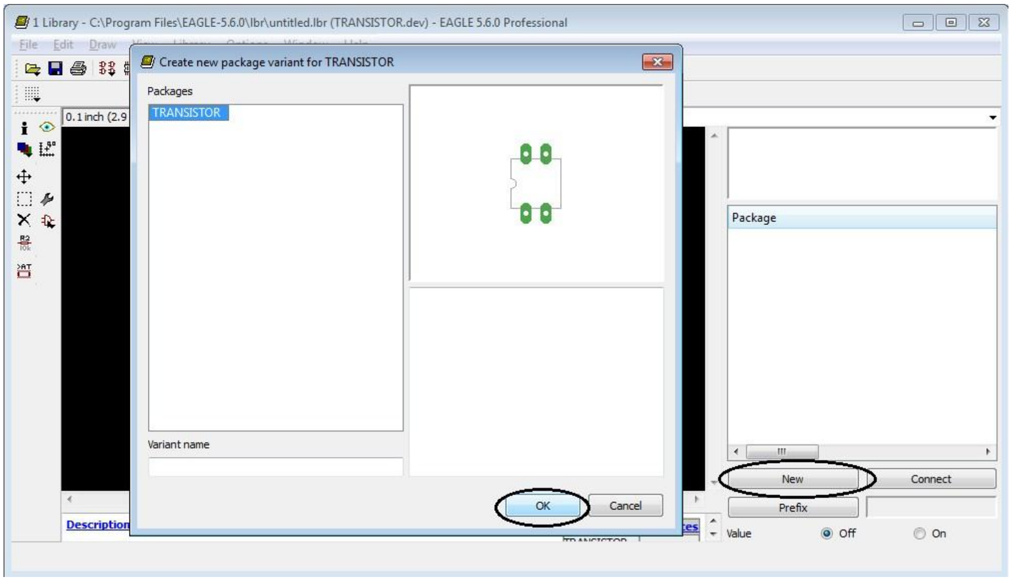
Add the transistor symbol using the Add tool as shown.



Place Symbol in the given space.



Next click on New and add the transistor package:

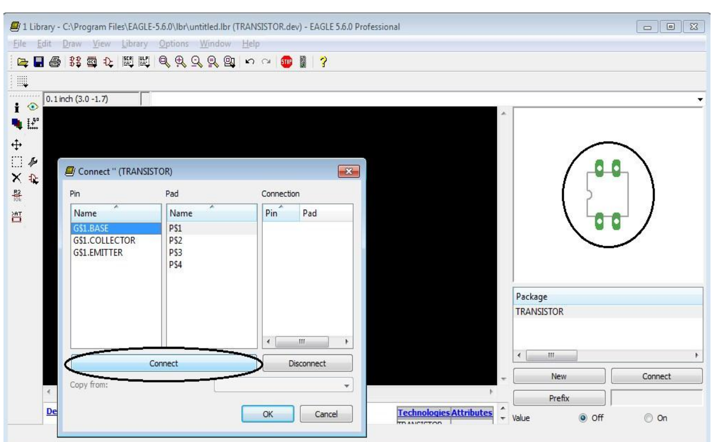


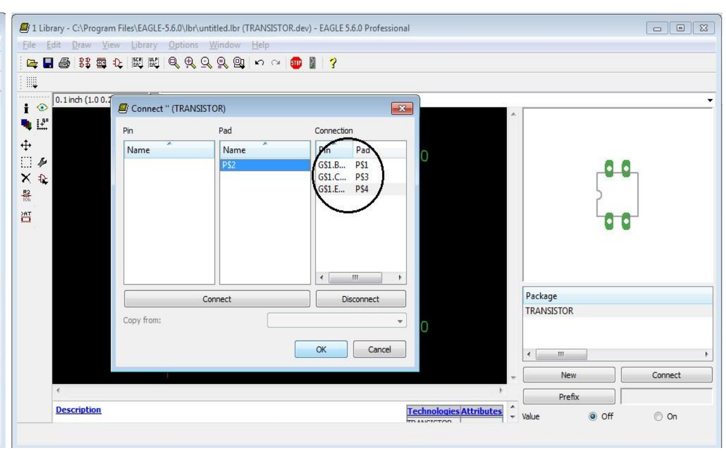
The IC package diagram appears as shown. Click on Connect button to link the pins of the symbol with the pads of the package. This step completes the Device creation.

Now the package has 4 pins whereas the transistor has only 3 leads viz. Base, Collector and Emitter. So 3 pins can be connected and the last pin can be left unconnected.

We connect the pins in the following way:

PAD 1 ←→ BASE
PAD 2 ←→ N/C
PAD 3 ←→ EMITTER
PAD 4 ←→ COLLECTOR

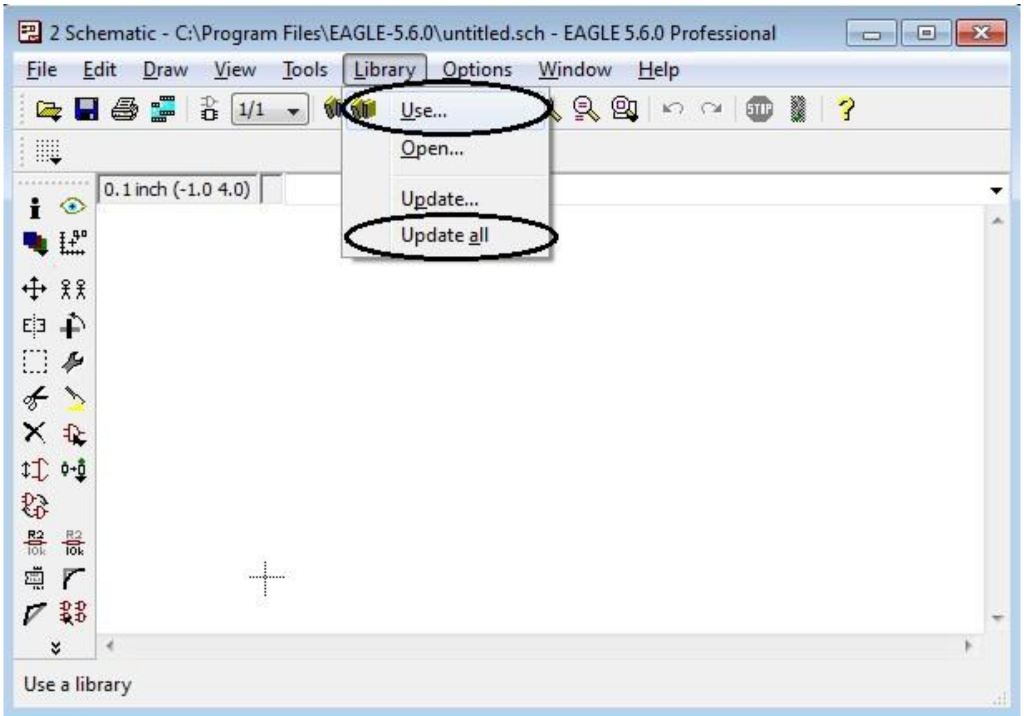




Now save the file at any preferred location in your hard drive. Before using this new library file in the schematic editor, the following actions are to be performed:

In the schematic editor,

1. Click on Library
2. Select Use: A window appears where the library file has to be located from hard drive and selected.
3. Select "Update All" which will add the new library file to Eagle's library database.



The new user defined component is now ready to be used in the schematic editor!

This completes the basic introductory tutorial of Eagle CAD and the rest of the finer details are left for the users' exploration!

HAPPY PCB MAKING ☺