

EAGLE PCB TUTORIAL

2013

Schematic Layout and PCB Design



Author
Ali Hadi Al-Hakim

e.quinox
Department of Electrical and Electronic Engineering
Imperial College London

1 INTRODUCTION

This guide will attempt to show you some of the basics of using the [CadSoft EAGLE](#) (Easily Applicable Graphical Layout Editor) software. The function of EAGLE is to provide the easy design of PCBs. The benefit is using EAGLE is you can get a relatively functional free version very easily.

This guide is broken into parts but in total will teach you the following using graphical step-by-step instruction.

- Creating component libraries based off component data sheets
- **Producing a circuit schematic**
- **Designing a PCB suitable for manufacture**

You can get your hands on a free version of EAGLE from <http://www.cadsoftusa.com/download-eagle/freeware/?language=en> and provided you have an internet connection you can start designing your own PCBs straight away.

NOTE

Be very careful when using this PCB version. If you create or modify EAGLE project files using the EAGLE version (6.x) you install on your computer there is a high chance that it will not be compatible with the EAGLE versions on the university computers. EAGLE compatibility issues are highly frustrating, so be careful.

2 IMPORTANT BOOKMARKS

(Click to jump to section)

Circuit Design	Page 3	Steps 1 to 14
Naming and Labelling Nets	Page 6	Steps 9 to 10
Net Classes	Page 6	Steps 11 to 13
PCB Design	Page 8	
Generating Gerber Files	Page 10	

3 CIRCUIT SCHEMATIC DESIGN

The EAGLE Circuit Schematic design workspace is where you will connect components together to create an overall complete circuit. Here you will learn how to find and add components, how to link them together using nets, the advantages of using labels, and how to use net classes.

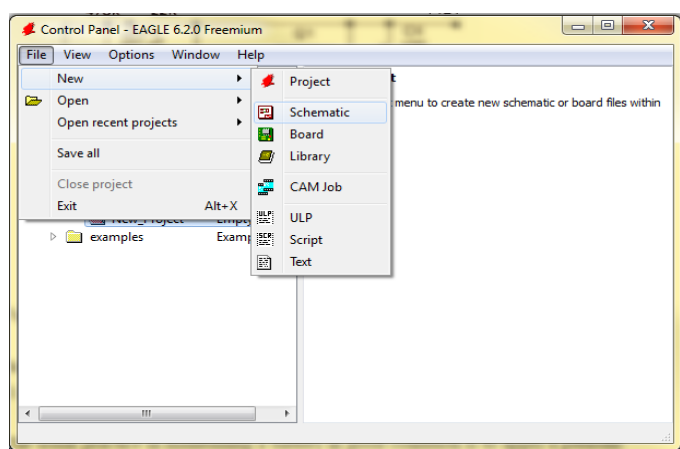
You are going to design a simple astable circuit where you will get to use the 555 timer you just created in the previous tutorial.

By now you should have worked out you don't have to click many of the toolbar buttons but just type commands instead. The following tutorial will use text commands only, you can learn the toolbar buttons yourself.

Step 1

Open a new schematic through the EAGLE control Panel and select:

File → New → Schematic

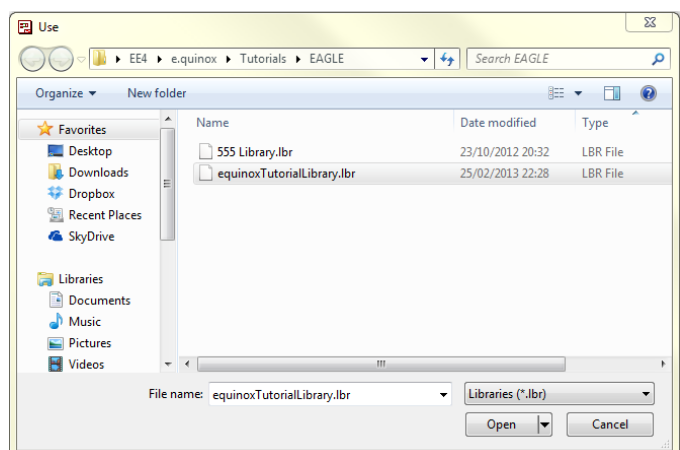


Step 2

Let's first add the library we just made. At the top menu bar go to:

Library → Use

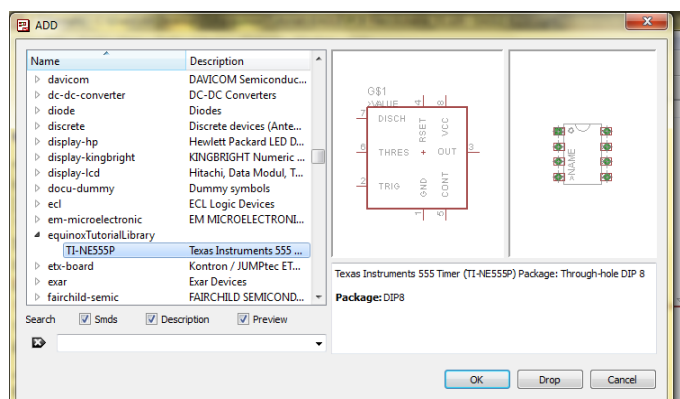
Navigate to where you saved your library and open it.



Step 3

To add the TI – NE555P type "add" and press return. Find and expand your library in the list shown; select your device and click OK.

You should now be able to place the 555 Timer anywhere on the workspace.



Step 4

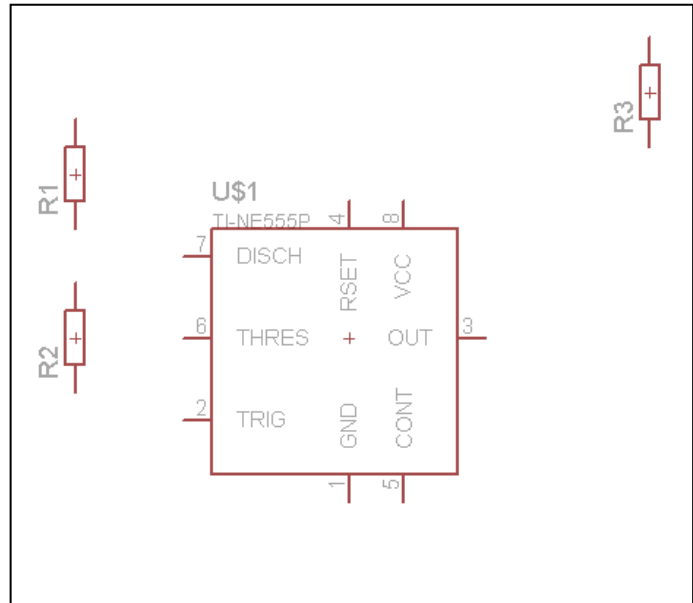
You will need three resistors. Add these by typing "**add**" and press return.

Search for "**Resistor**" and expand the library:

rcl → R-EU

Find the component **R-EU_0207/10**. This a resistor of diameter/width 2mm, and length 7mm. The pads are 10mm apart. This should match the standard through-hole resistor package you're probably used to.

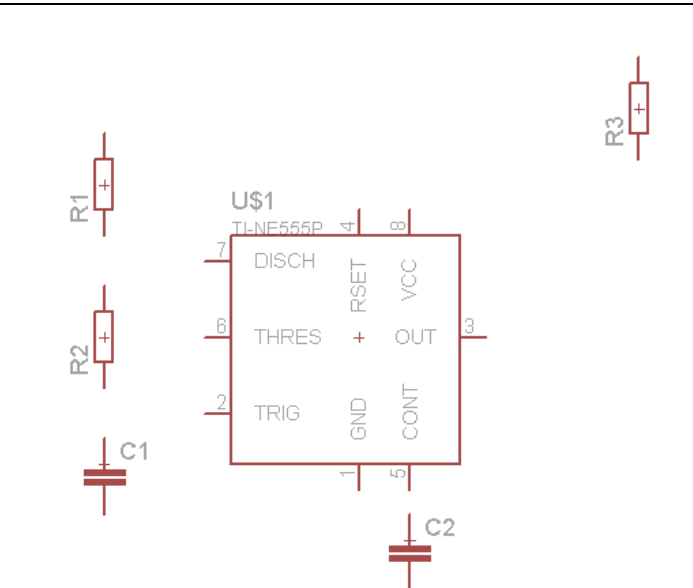
Place three on the workspace. Press mouse right-click to rotate a component symbol. If you add too many use "**del**" to delete some.



Step 5

Next add two capacitors by doing the same thing. Capacitors come in many shapes and sizes so if you know which components you're using (which you normally should at this stage) you may want to make your own package.

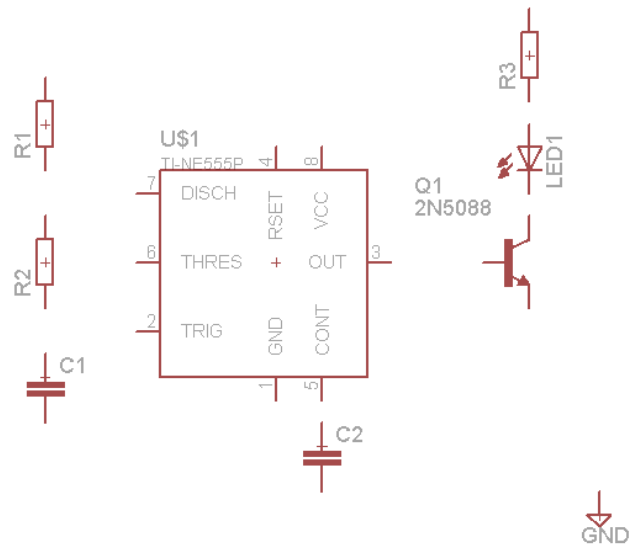
For now add "**C5/5**" and place two on the workspace.



Step 6

Finally add:

- an LED (search for "LED5MM"),
- a TO92 packaged transistor (search for "2N5088"), and
- a ground symbol (search for "GND").

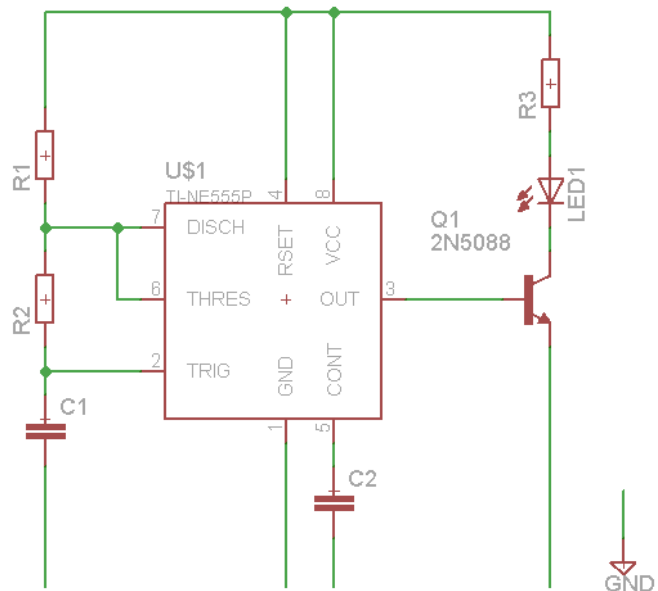


Step 7

You connect the components by using the "net" command. Type this and then connect each component as shown by clicking where you want a net to start and end.

If a pop-up appears asking to "Connect Net Segments?" this is simply informing you that you are about to connect two different wires into the same node and they will be now named the same. In general, you probably just want to click OK.

Always check that when two nets are connected a green dot appears. If not (which does happen more often than you think) delete and re-wire!



Step 8

You should name and value components next.

You do this using the "name" and "value" commands. Name all of the symbols first but value only those that need them (resistors and capacitors in this case).

Step 9

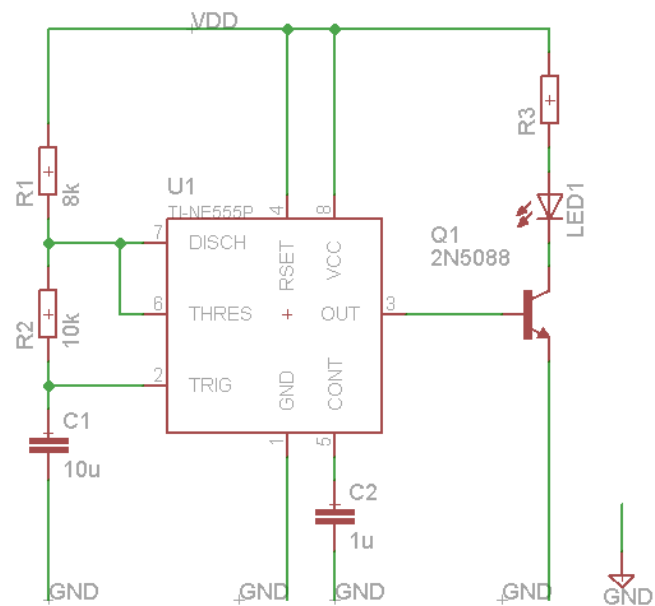
Following, name the nodes.

Call the very top node "VDD" using the "**name**" command. Then, name all of the bottom nodes "GND".

NOTE: By naming the nets (green lines) with the same name they will be connected together, even if there appears to be no clear connection visualised. This is useful for large and more complex circuits.

Step 10

Use the "**label**" command on nodes to reveal their name.

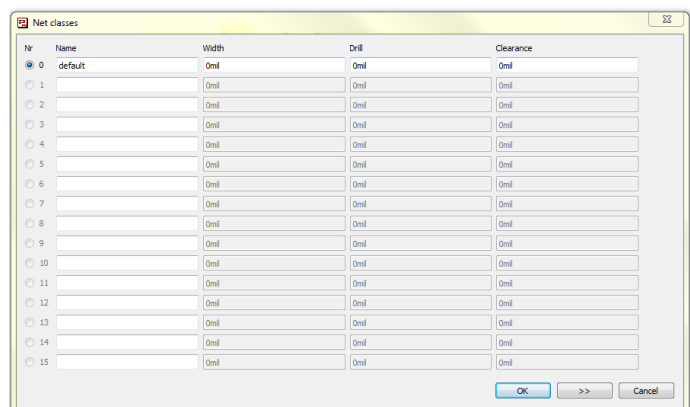


Step 11

For many circuits it becomes very important to provide power tracks. These will be wide tracks capable of carrying higher current without breaking. To do this you should create some net classes.

Go to:

Edit → Net Classes



Step 12

In the **Name** field of row 1, type "**Power**".

To calculate the correct track width for a pre-determined current you should use an [ANSI PCB Track Width Calculator](#). Here you insert some basic input values and are provided with a recommended track width.


For now use:

Width: "**30mil**" Drill: "**0.9mm**" Clearance: "**0.7mm**"

This is rated for about 3A with an allowable track temperature rise of 10°C and a peak voltage of about 20V.

Nr	Name	Width	Drill	Clearance
<input checked="" type="radio"/> 0	default	0mil	0mil	0mil
<input type="radio"/> 1	Power	30mil	0.9mm	0.7mm

Step 13

To assign this class to your power nets (VDD and GND) you need to use the general change settings tool again ().

This time use:

Class → 1 Power and click on the VDD and GND nodes. You can click on any of the individual nets and the whole node will be assigned to the Power net class.

Step 14

Save and name your schematic somewhere you'll know where to find it. You are now ready to start the actual PCB design process.

Your schematic is now complete and you can begin to layout your PCB.

4 PCB DESIGN

The PCB design follows directly from the circuit schematic design. It is very important to save your schematic under a given name before switching to the PCB workspace. When you do switch to the PCB workspace and save it to file it should be saved using the same name as you have just given your schematic file. This is important to note because it shows the linking of the schematic and PCB.

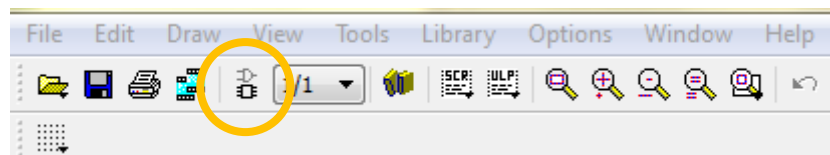
This linking means that any new components or changes to wiring you make in the schematic will automatically show on the PCB. This can be very helpful for last minute modifications to the circuit design.

HOWEVER, if you wish to redesign a PCB from scratch (to try a new layout) this linking property means you should go back to your schematic and save it under a new name before switching to the PCB design again.

If you **ever** modify any of the files whilst the other is not open, they will no longer be linked together and this can potentially ruin a large amount of work and cause you to start over again. Whenever you modify a board (.brd) file or schematic (.sch) file which has been linked, you should also ensure both linked files are open together.

Step 1

To switch to PCB design you must click the icon shown on the upper toolbar.



Due to current time issues this section will be left incomplete for now. However, you have learnt by now how to interact and use EAGLE in a relatively efficient manner. Many of the commands you have used before will still work in PCB design.

PCB design is very simple. Your main task is to just move components into a position within the white box (which can be resized) and then route the connections to see do you get a successful design. I will list now the commands of most use to you.

move Use this to move components around in the workspace.

You can also modify the size of the PCB board (white box). To do this, once the move command is active simply click on the edges of the white box and move them.

group Once active you can drag a box or build a polygon around the items you want to group. Once grouped this will be saved until another group is formed. Thus, type another command such as move or copy, right-click and you will be given the option to affect the group in the drop down menu.

auto; This will route your wires for you into copper tracks. In the bottom left hand corner you will see how complete it is (i.e. if all wires have been routed). The use of the semi-colon skips an options

window that will pop-up otherwise. For now you do not need to concern yourself with these options.

ripup; This will return the circuit to what is called 'ratsnest' mode. Use this mode to modify the PCB design. Use the semicolon to ripup all track automatically.

NOTE: *you should avoid modifying the PCB design when you have routed the circuit as this can cause a lot of problems with regards to PCB design rules. Instead always use the ripup; command and modify the design when in ratsnest mode.*

right-click: Use mouse-button right-click to rotate an object when you have it selected.



Layers button: Use the layers button to hide or show different layers of the PCB schematic, or change colours. Try changing colour of the 'vias' layer to pink or yellow and maybe experiment with turning on or off different layers.



Zoom buttons: Make use of your zoom tools as scroll zooming can be awkward. The best once to use are the far left

(fit to screen) and far right (zoom to window) buttons.



Switch-to-Schematic: Use this button to return to your schematic where you can either modify it or save it under a new name to begin a new PCB design.

It is important to note the fabrication of a PCB is completely dependent on the facilities available to you. i.e. machine accuracy and precision, material properties etc. To accommodate for the variation in available facilities design rules can be applied to a PCB to ensure that it is possible to fabricate your PCB in-house. You should have been provided with these rules with this guide. To use them please do the following:


1. From the menu-bar go to: **Tools → DRC**.
2. On the new window, select "**Load**".
3. Now open the rules that should have been provided to you, title: "**ee_rules.dru**".
4. Press "**Apply**" and then "**Select**".

Take these rules with a pinch of salt. In some cases they will inform you of a very serious error. For example any form of "overlap" error is likely a serious one. However, in some cases it may just be complaining of your text font size is too small or a clearance on a component you have no control over is breaking the rules. You can ignore these if you wish. However, wherever possible, try and ensure there are as few rules being broken as possible. Once happy, you can move onto manufacture.

5 PCB MANUFACTURE

Once you're confident that everything in your PCB is correct and you're satisfied that all ERC and DRC issues can be ignored or have been resolved you can go on to fabricate your PCB. To get a PCB fabricated, you need to provide the manufacturer with a series of files called gerber files.

These are simply a series of files which describe the details of your PCB so that the machines that use the files know how to manufacture the board to your design. You should have been provided with a file with the extension .cam. This is a *job* which you can load up in EAGLE to generate for you the necessary EAGLE files to get your board made. To process a job you need to follow the steps below:

1. Load up your .brd PCB file.
2. Do a DRC check on it and make sure you're happy with the results
3. If happy, click on the light blue icon on your upper tool bar ().
4. A new window will pop up. Select: **File → Open → Job...** and open the .cam file provided to you. It's named in this case **eee_gerbers.cam**.
5. When it's open just click "Process Job"!
6. When the job is complete you should find your gerber files in the file location that your .brd file is saved in. This file path is also shown at the bottom of the "Process Job" window.
7. Zip up the gerber files together ready to be sent off to your manufacturer.

When you process a job using the provided .cam file, you should end up with a series of files with the following extensions:

.cmp	This file describes the top copper layer	(extension stands for comp onent side)
.sol	This file describes the bottom copper layer	(extension stands for sold er side)
.stc	This file describes the top layer solder mask	(extension stands for stop comp side)
.sts	This file describes the bottom layer solder mask	(extension stands for stop solder side)
.drd	This file describes the drill positions	(extension stands for drill description)
.dri	I'm not entirely sure what this actually describes	(extension stands for drill information)

.got This outline file is important if you weight and shape the dimension layer (layer 20).

If you make use of the dimension layer you can round the PCB corners or create shapes other than a four-sided polygon which is sometimes desirable. To add weight to the dimension layer just change its width properties to at least 0.1mm. The default dimension layer is the o width white square which you place your component packages in when beginning to design the PCB.

There are some layers that are not generated for you by the provided .cam job file. These are the silk screen top and bottom layers. The reason for this is that the in-house manufacturing in the EEE Department at Imperial is not capable of providing silk screen layers.

Also, note that sometimes you will come across gerber extensions that differ from these. There is not a common extension used by everyone unfortunately, but normally there is a clear indication of what each gerber means in its name (e.g. sometimes .top and .bot are used to top and bottom copper layers, respectively).

A short description of all the mentioned layers is given below:

Copper layers are the top and bottom copper paths that form the electrical connections between through-hole and surface mount pads. The files describing these layers will simply define where tracks go and where copper should be removed or left to form tracks/pads/planes.

The **stop mask** is the green stuff that you'll see on most PCBs. This basically just acts as a protective layer on your copper tracks and protecting the copper from oxidising over time. The stop mask is not always necessary and in some cases may be undesirable. Such a case would be if you wanted to 'tin' your tracks which is where you solder the tracks so they can handle more current. Nonetheless, it is possible to strictly define where you will allow the green mask to be applied and where you don't want it to be applied using tStop or bStop layers.

The **silk screen** is the white writing and markings that you see on most PCBs. It's a very effective way of marking your PCB with text or component names/values and component positions. Dimension, tPlace, bPlace, tName, bName, tValue and bValue are all silkscreen layers.

In-House (Imperial EEE) Manufacturing

If you want to manufacture your PCBs in house using Mike Harbour's equipment then there are some extra things you want to be aware of and perhaps change on your board:

1. The in-house manufacturing equipment does not have the ability to provide you with a silk screen. This can be an issue as the silk screen is a very powerful method of labelling your board with component values, supply polarities etc. If you do need to write on your board to help you during component population and testing you should do it using the copper layers. This is literally etching characters into any space on the PCB that isn't used up by tracks. You can do this by selecting the Text icon from your toolbar (or typing "text") and then just placing the text where you want it. To draw it in copper you must use the layers 1 or 16 for top or bottom copper layers, respectively. It's important to note that you should change the text properties to be vector format and of size at least 1.8 inches. If you don't, you'll likely get your board back with the text unreadable.
2. A stop mask is not always necessary to have and in some cases you may want to avoid using it, especially for initial prototyped boards. The addition of a stop mask means more work has to be done during PCB fabrication so if you get your boards made in-house you may want to not bother with one as it just means it will take longer before you get your board back. However, for tricky soldering sections you may wish to include a stop mask as it means solder bridges are less likely to occur and cause a short circuit.

To get your PCB made, email the zip folder containing your gerber files to Mike Harbour with information about what you want. i.e. number of boards, with or without a stop mask, any other special requests. End with a thanks and in a few days you should receive an email informing you your PCB is ready to be picked up.