

INTRODUCTION

PCB stands for Printed Circuit Board is a laminated sandwich layers of insulating layers. A basic PCB consist of a flat sheet of insulating material and a layer of copper foil, laminated to the substrate. PCB can be single sided (one copper layer), double-sided (two copper layers on both sides of one substrate layer), or multi-layer (outer and inner layers of copper, alternating with layers of substrate). Chemical etching divides the copper into separate conducting lines called tracks or *circuit traces*, pads for connections, vias to pass connections between layers of copper, and features such as solid conductive areas for electromagnetic shielding or other purposes. The tracks function as wires fixed in place, and are insulated from each other by air and the board substrate material.

SOFTWARE

We will use PCB designing software to design our required schematic. But before jumping to schematic design, first we must be able to draw our circuit and we much check whether it is electrically correct or not. Thus, assuming we can draw circuit on paper, we will learn to simulate the circuit in computer.

Proteus is one of the popular circuit simulation software. The software is used mainly by electronic design engineers and technicians to create schematics and electronic prints for manufacturing printed circuit boards. Having large community of this application it is easy to learn and also possible to get library for such components which are not by default at the time of installation.

The main downside of this software is that, it is not free and open source. One must buy it before using. But as the internet is full of pirated products, we can easily find its crack products as well. Also, all cracks are not stable and might not work for everyone.

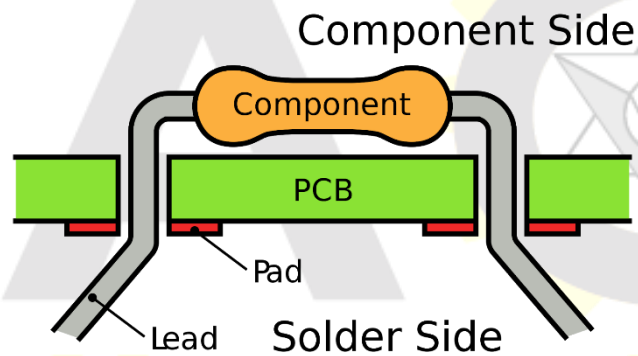
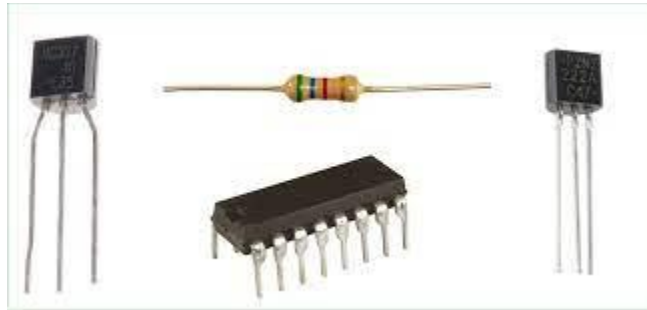
Similar, to proteus there are also many other applications like MultiSim, Ltspice, KiCad and many more. Although we are not promoting to use pirated software, firstly we are going to learn how to simulate the hand drawn circuits on proteus. Once we are able to simulate circuits then without any delay, we will jump to design PCB using KiCad.

KiCad is one of the best applications of making schematic design. Unlike proteus, Kicad is totally free and open source making this software widely used around the world by students and professionals.

Types of Components

1. Through Hole (THC)

Through hole components are mounted by their wire leads passing through the board and soldered to traces on the other side.



2. Surface Mount (SMD)

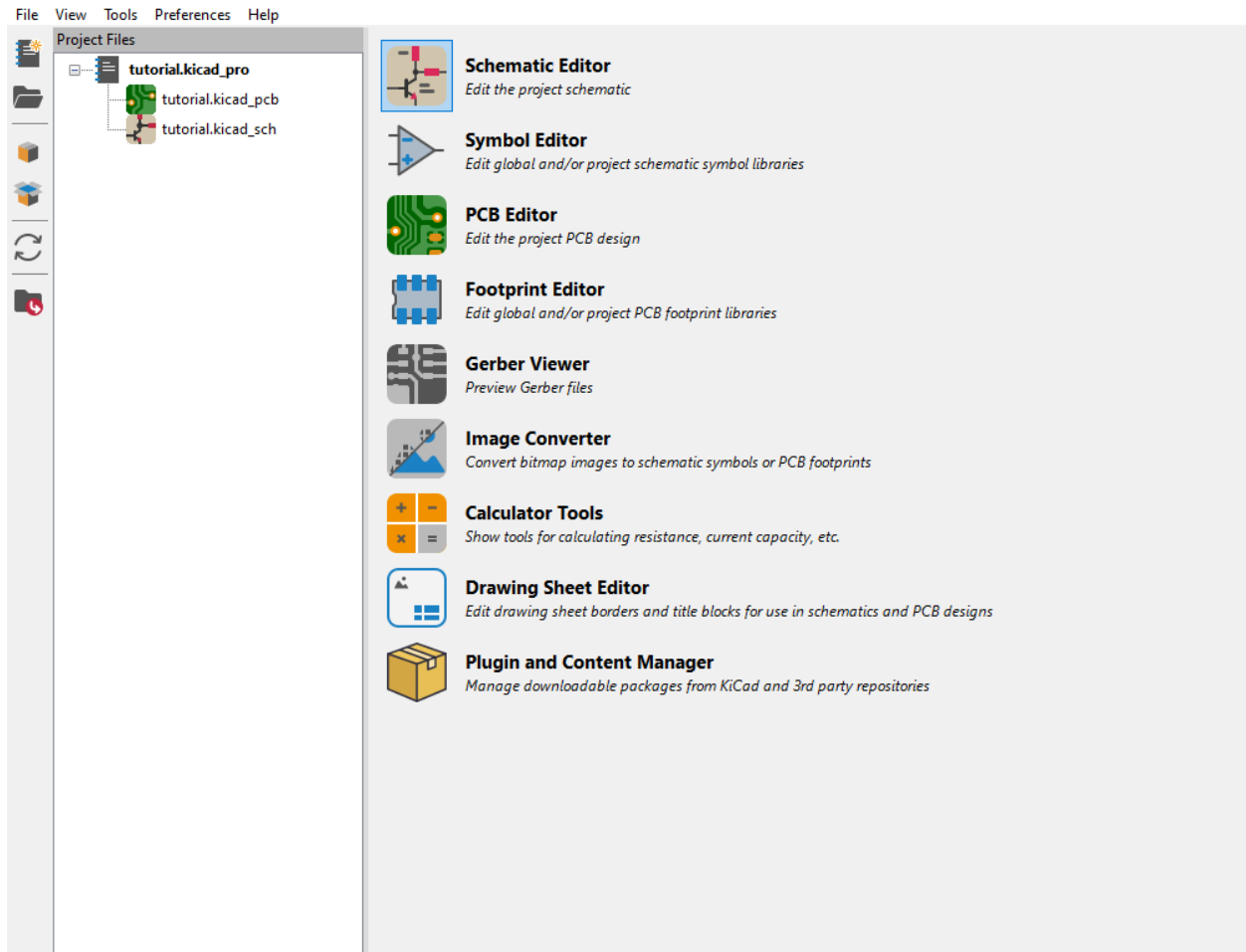
Surface mount components are attached by their leads to copper traces on the same side of the board.



Designing Schematic

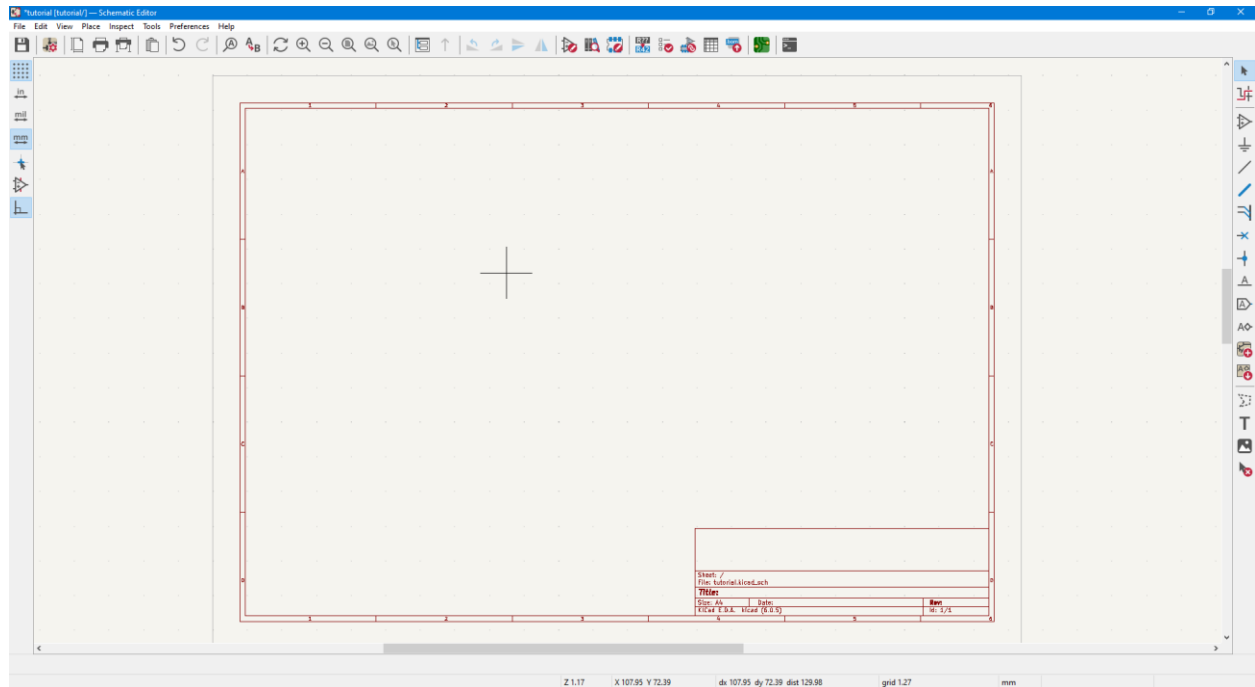
Now we will learn how design schematic using KiCad. Here KiCad version 6 is used and some of the upcoming steps might not be exactly same.

First open KiCad and create a project by giving a relevant name for your project.



Here, I have created project with name tutorial. Now open the file with _sch to start schematic design.

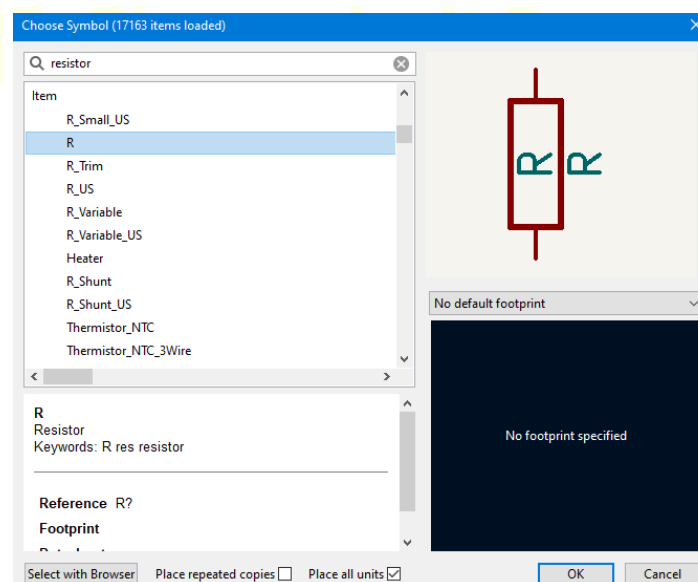
After that you might get interface similar to this.



This is Schematic Editor where you can draw your required circuit. You can draw circuit in any layout as its is not final trace which is going to be in PCB.

3. Insert Components

To insert component, click on Add a Symbol button in right or simply press 'A' key. It will then open a dialog box where you can search and select components.

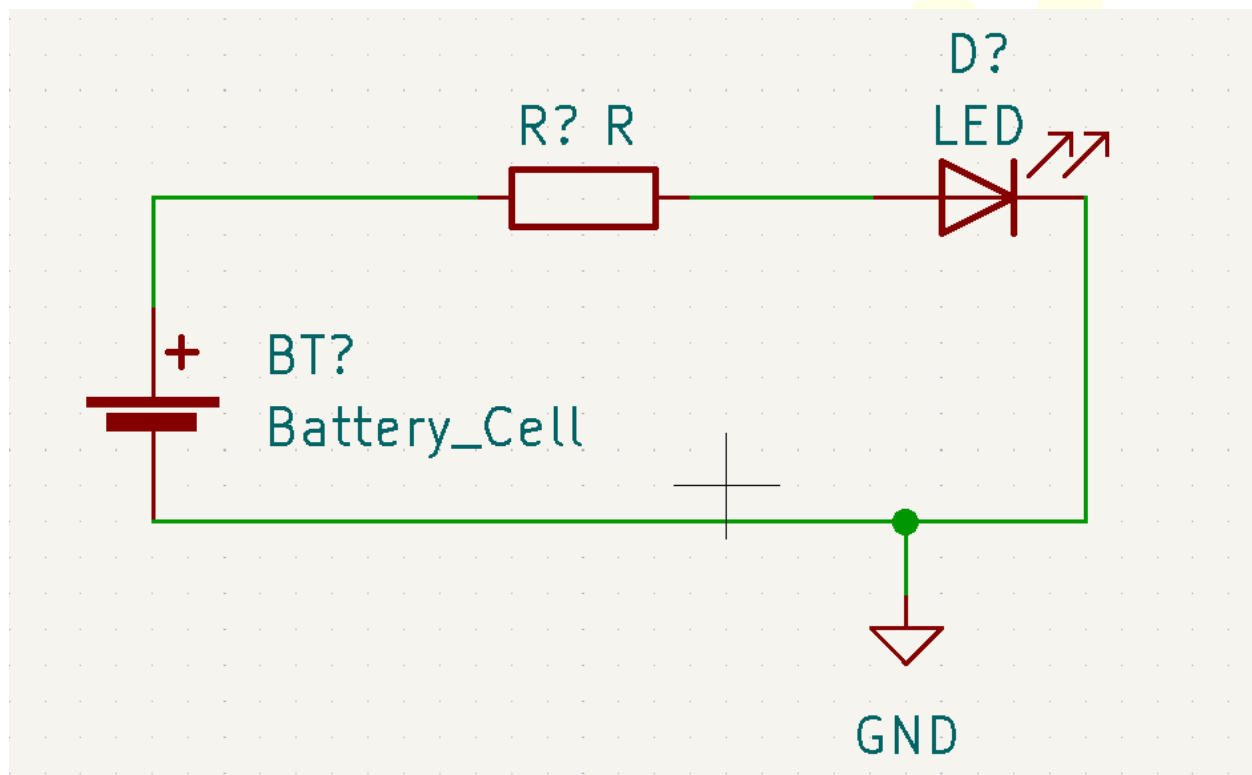


After selecting component, it will attach to your cursor and place it in desired location by left click.

- To move component, hover over it and hit 'M' key.
- To rotate component, hover over it and hit 'R' key.
- To duplicate component, hover over it and hit 'Ctrl + D' key or use "Ctrl + C" and "Ctrl + V" to copy and paste.
- To delete component, hover over it and hit 'Delete' key.

Connecting Components

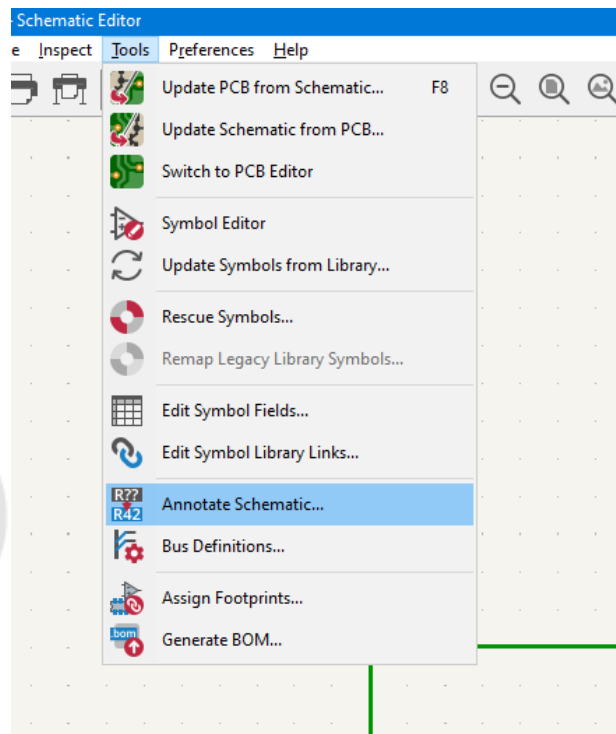
After placing all components, we need to connect them by wire. So to get wire, select Add Wire option from right or simply hit 'W' key.



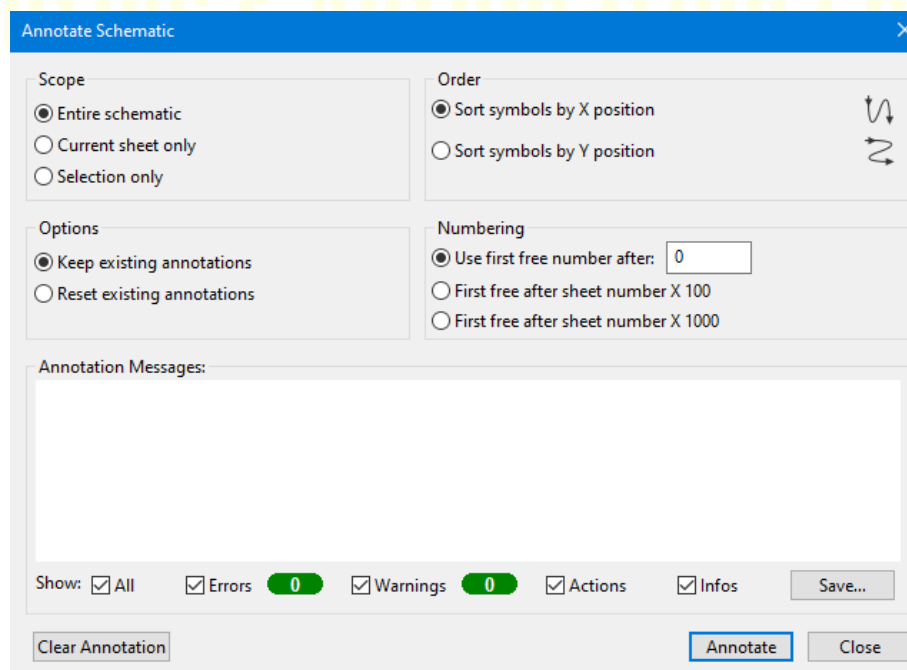
After this your circuit might look like this. Adding a ground to your circuit might be useful later but it is not compulsory.

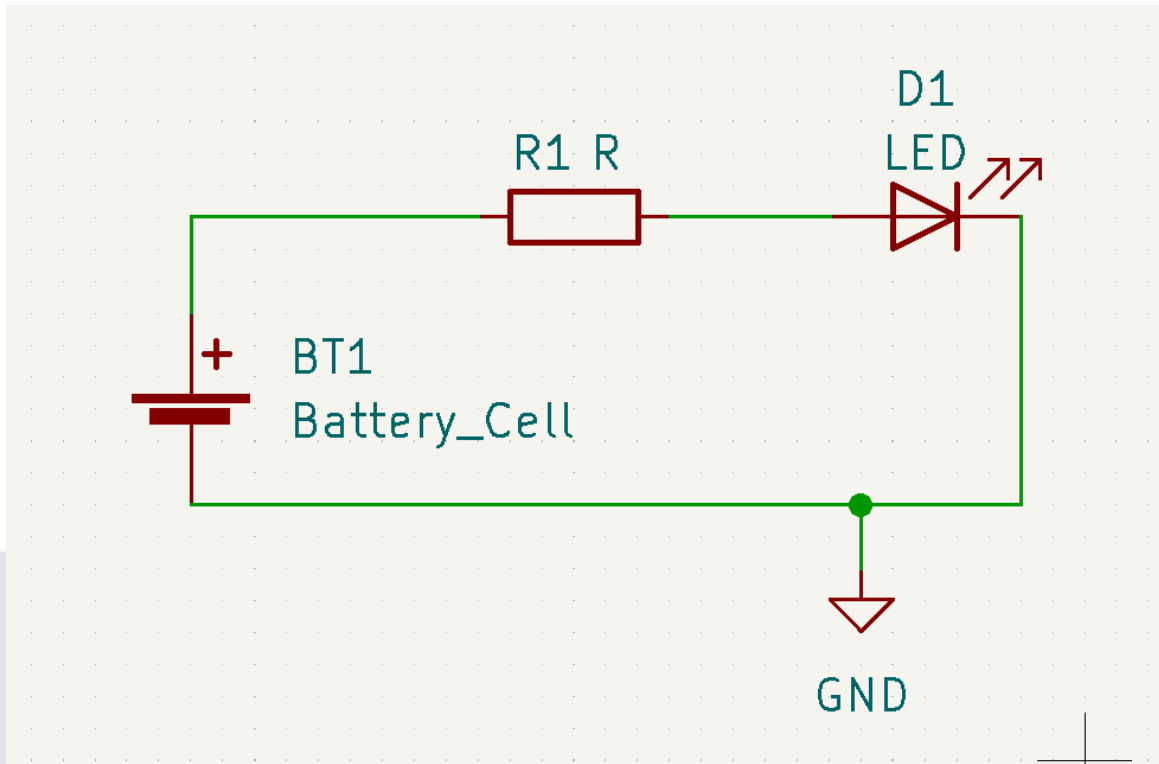
4. Component Naming

Components in this circuit are missing their proper label so, we need to annotate them. To annotate, goto tools and select “Annotate Schematic”.



A window will appear keep all settings default unless you want to make any changes then press Ok.

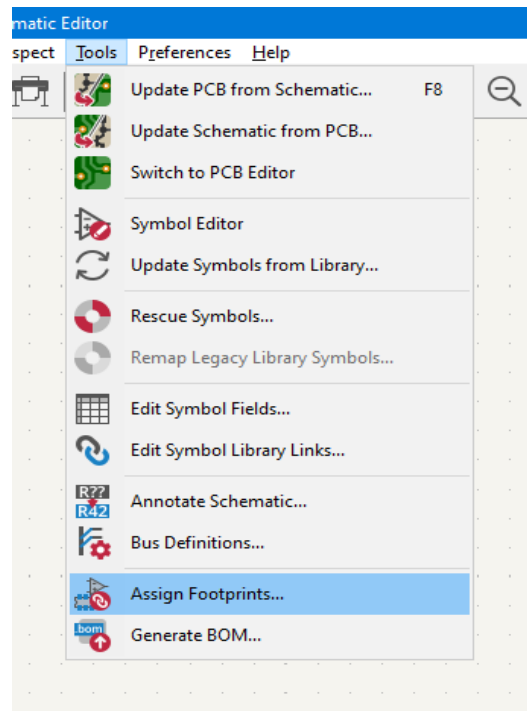




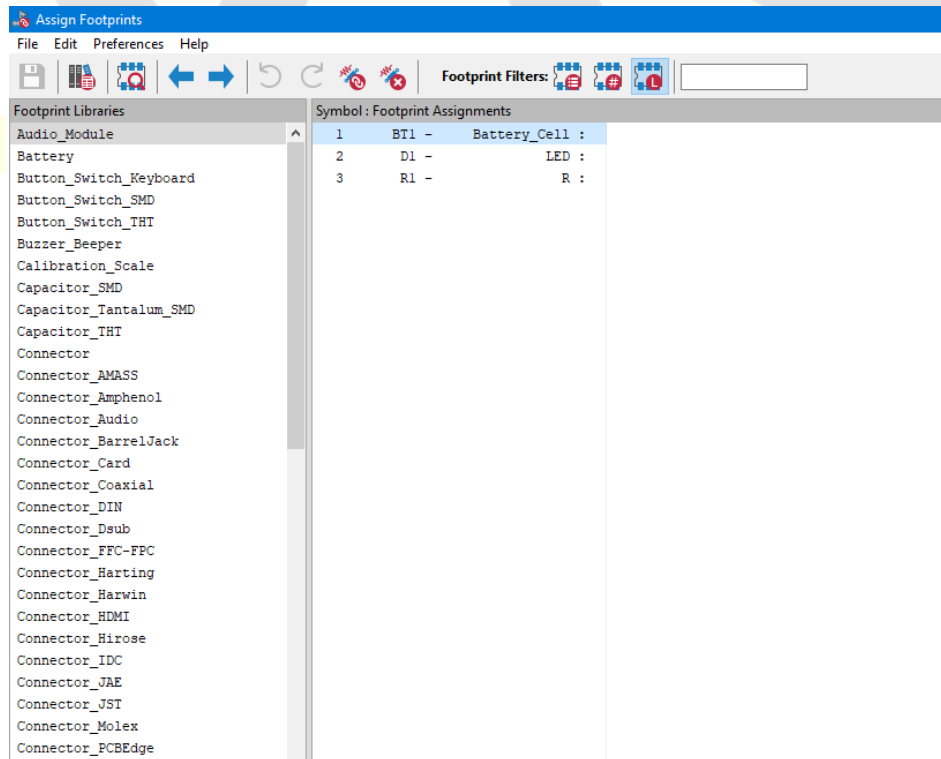
Now all components have proper label like R1 for resistor and D1 for LED. If there are multiple same components like three resistors then their label will be R1, R2 and R3.

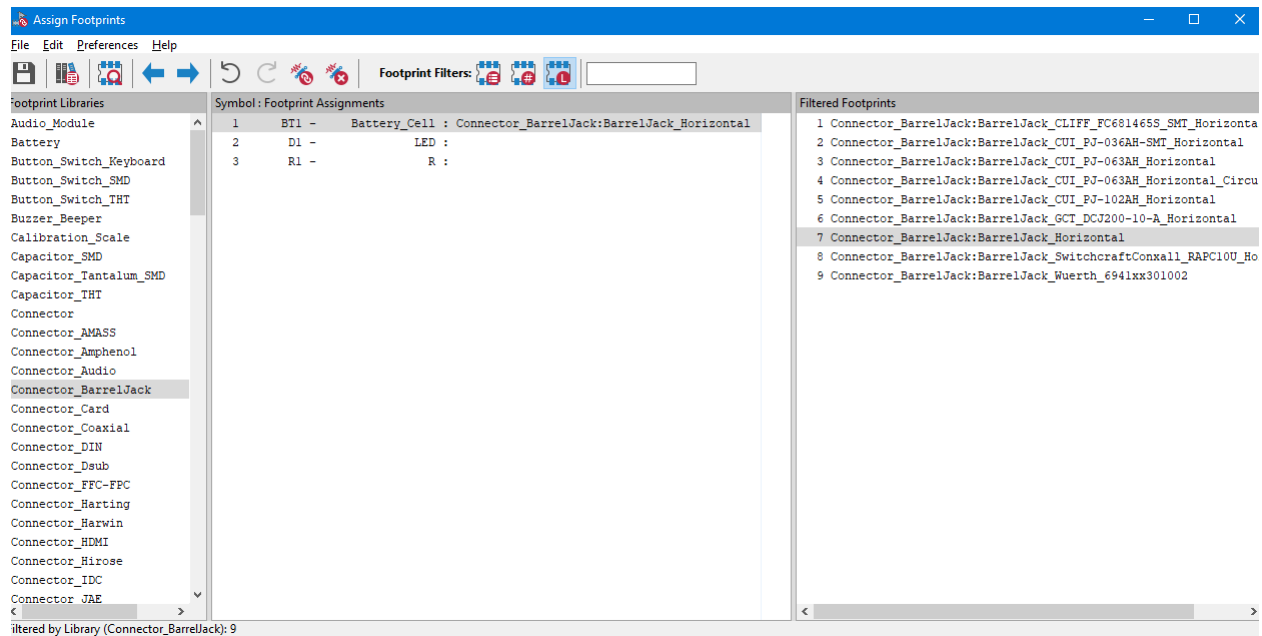
Robotics & Automation Center
IDE. Thapathali Campus

5. Assigning Footprint

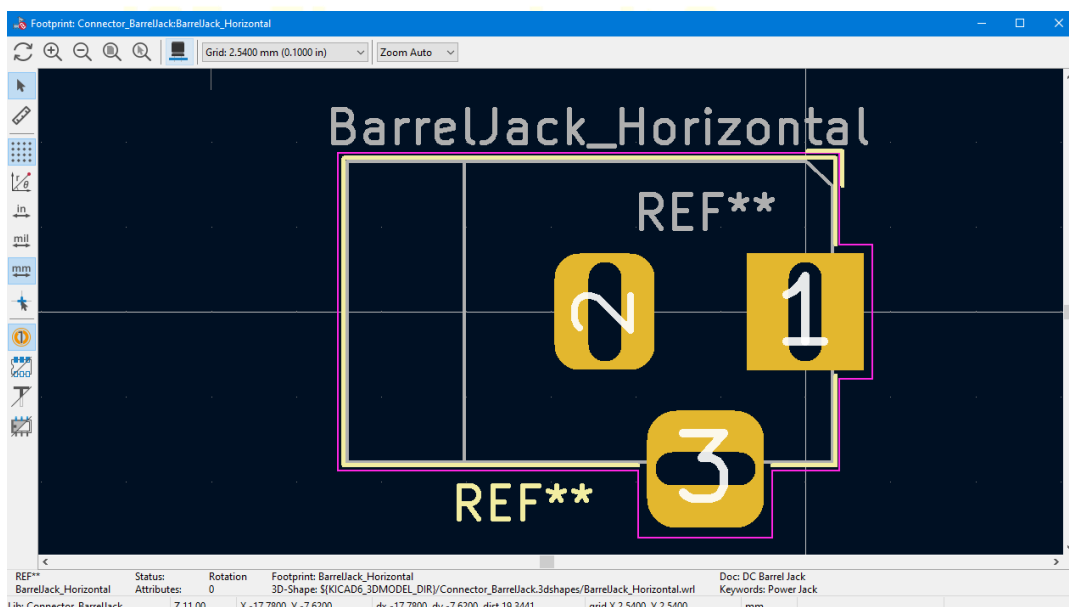
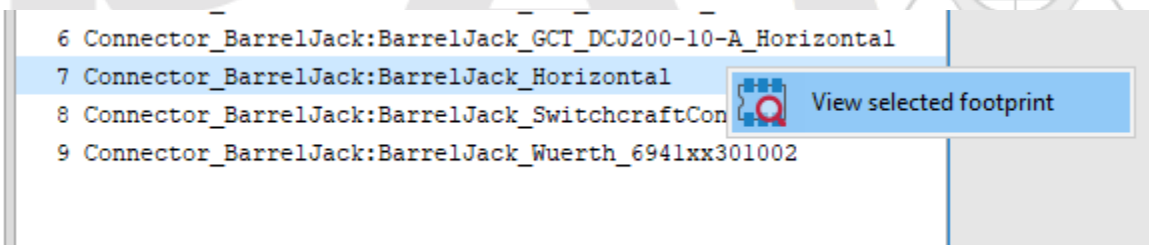


Now we need to assign footprint for all components. From Tools select “Assign Footprints” and wait until all components are loaded.

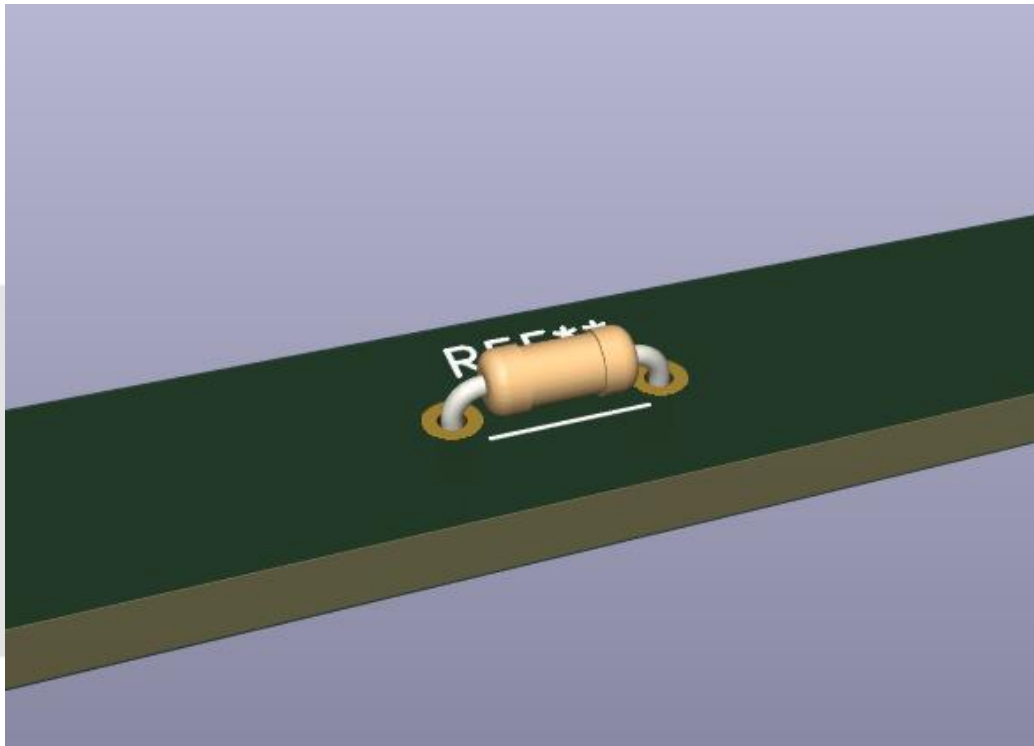
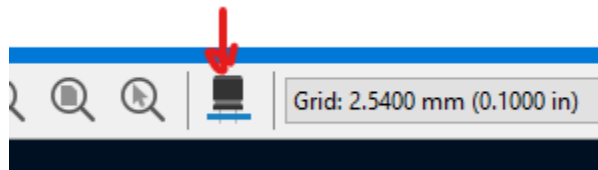




For battery I have selected Horizontal Barrel Jack. You can also see footprint by right clicking the selected footprint and click on “View Selected Footprint”.



You can also see its 3D view by clicking on “Show 3D Viewer” option.

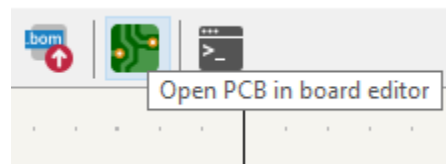


This is 3D view of axial through hole resistor.

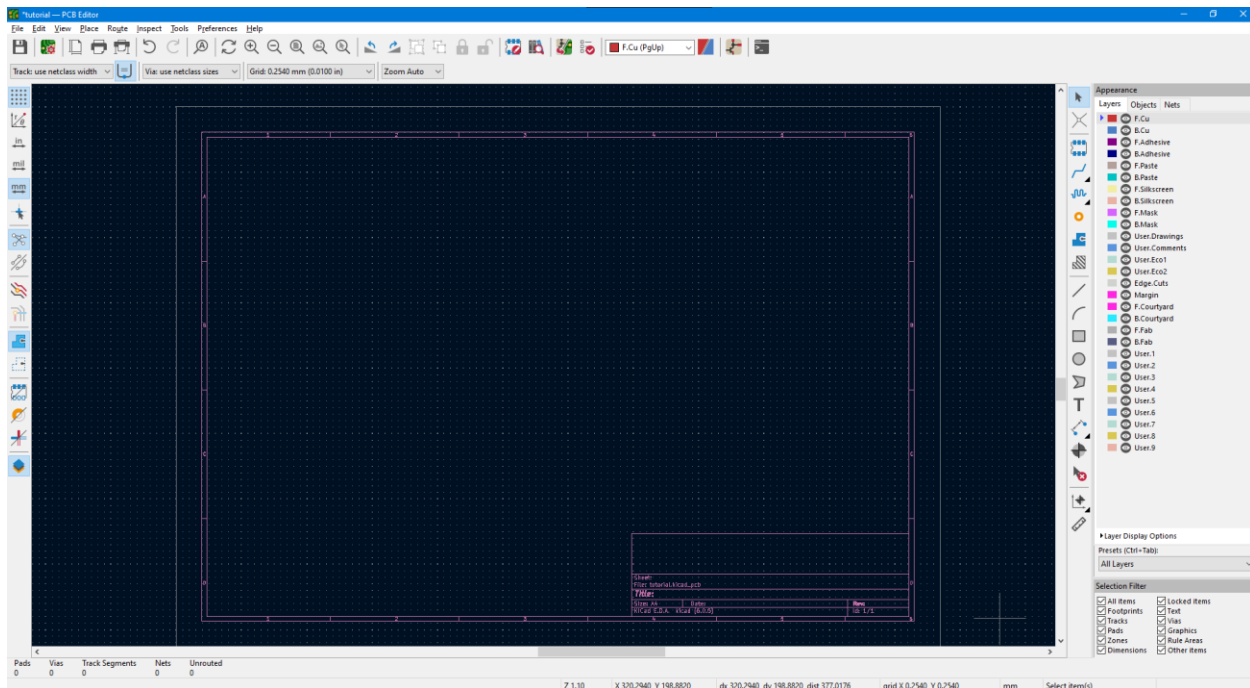
After assigning footprint for all components, click on “Apply” and click on “Ok”.

6. Inserting Components in PCB Editor

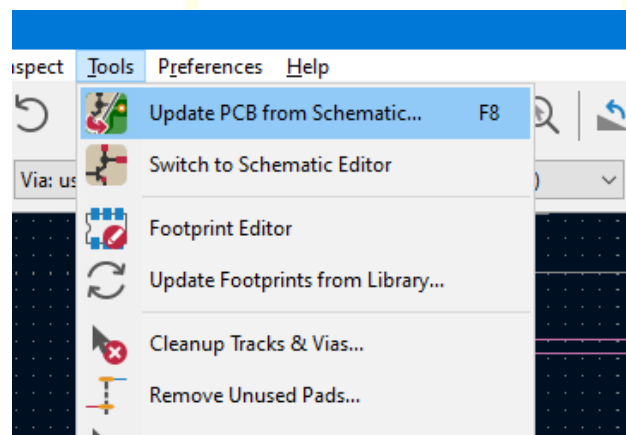
Now we can start actual PCB building. Open PCB editor from top menu.




A new window will open with empty workspace.



First, we need to import all of our component to PCB Editor. To import components, select “Update PCB from Schematic” from Tools.

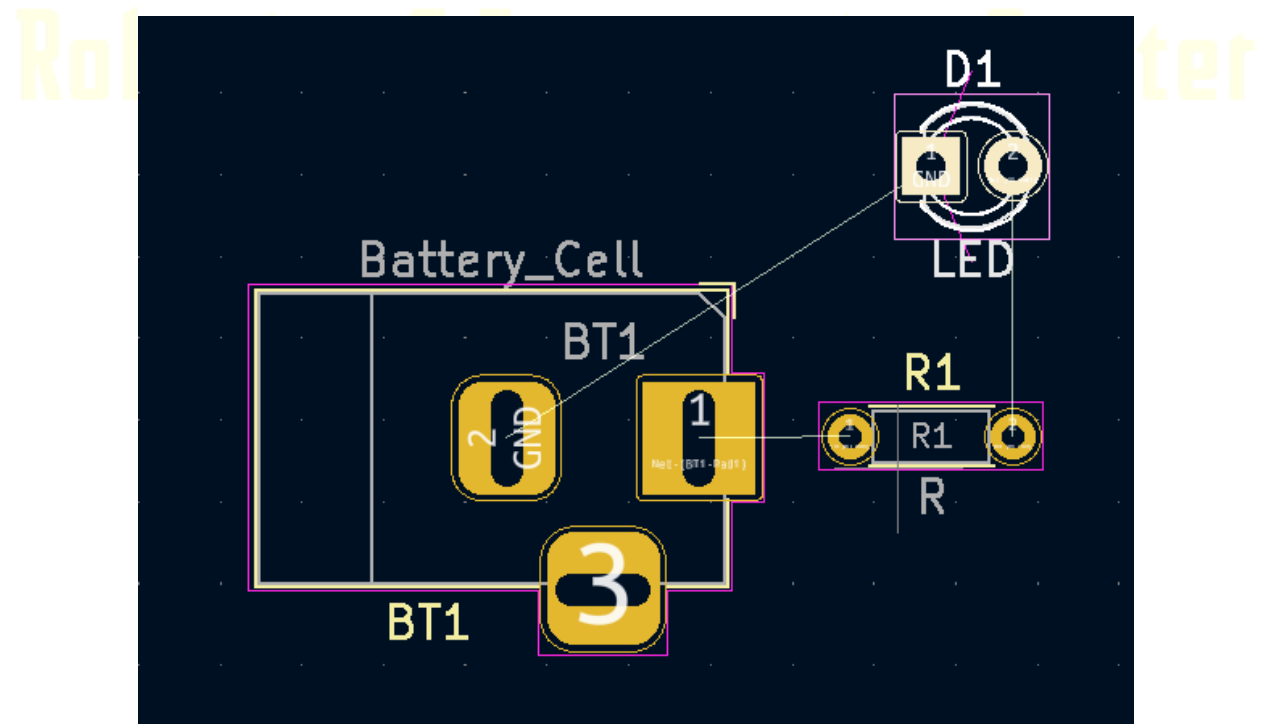




Here, we can see footprints we selected earlier is being used instead of components symbol.

7. Component Placement

Arrange all footprints in a layout as you want to see in your PCB board. To move, select footprint by left click and hit 'M' key to move and drop it in required position. You can also rotate by 'R' key.

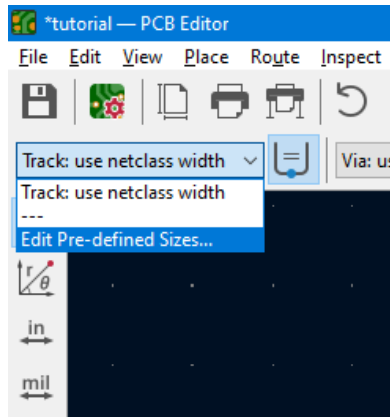


Prepared By: Ashmin Bhattarai
Robotics Week 2022, RAC Thapathali

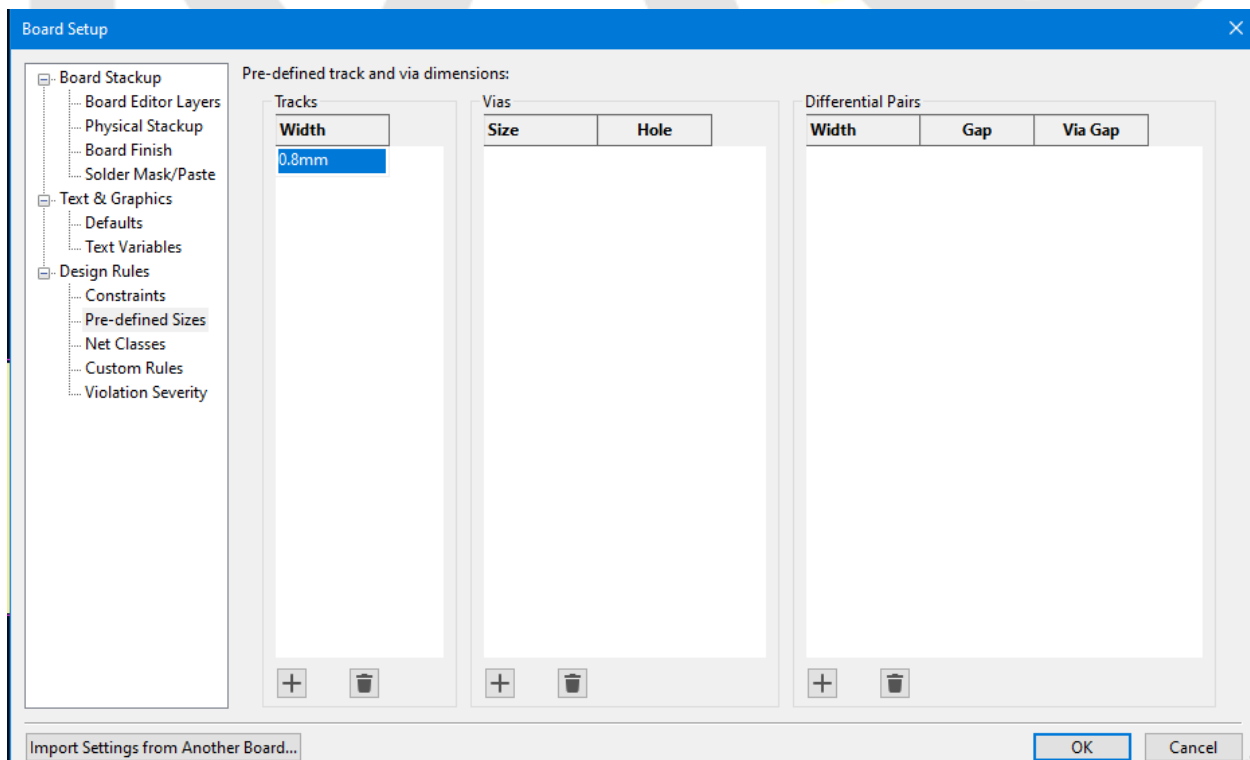
8. Traces

If you are using single layer PCB, then select Back Copper (B. Cu) from options in right side of window. Before drawing traces, we need to set trace width.

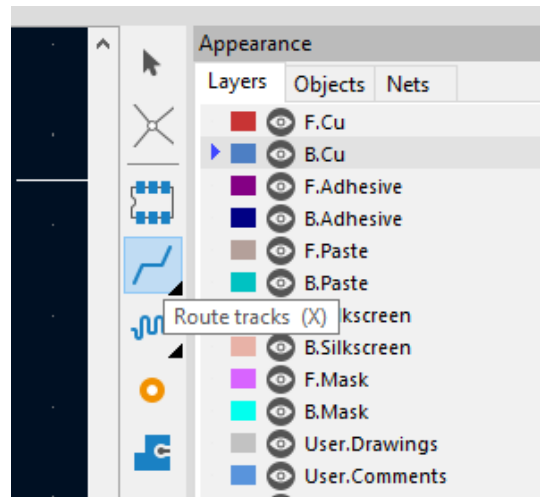
From drop down option called Track in top left select trace size or add new by selecting “Edit Pre-defined Sizes”.



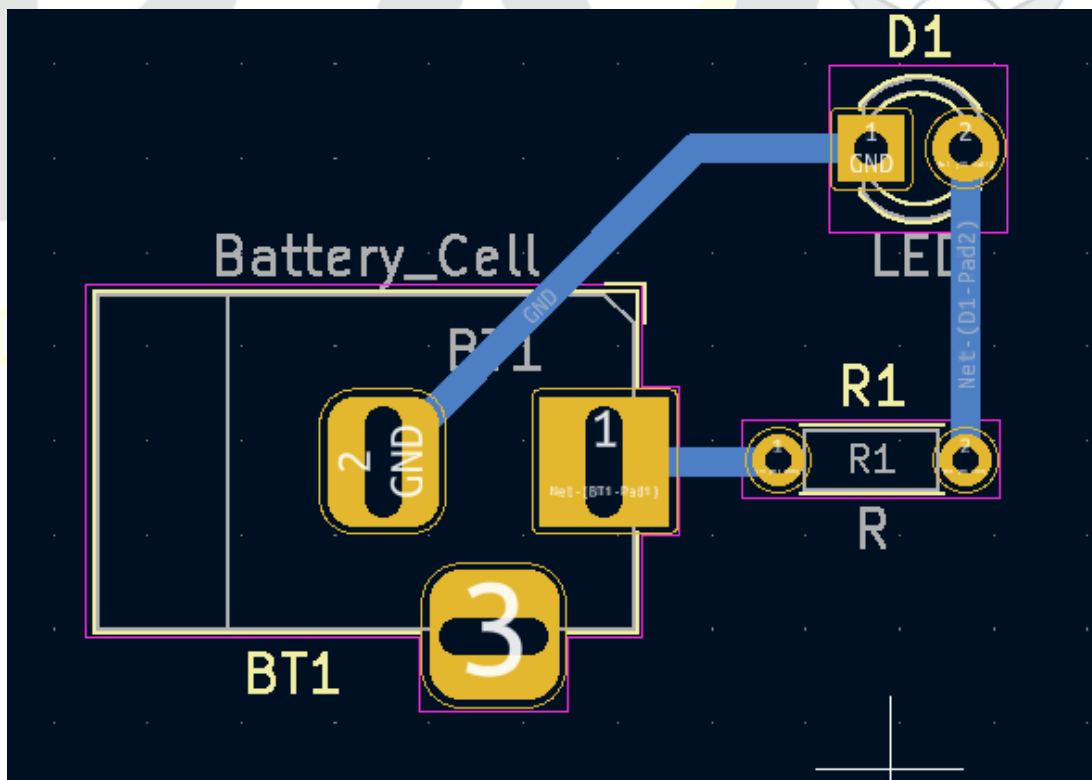
This will open new window. Click on + button in Tracks column and add trace size. Usually, we will use trace size of **0.8mm** but it might differ with different standards. Then click on Ok and select new trace size you have just created.



To draw traces, select Route tracks option or use 'X' key. Remember to check layer whether it is F Cu or B Cu.



Now start drawing traces.

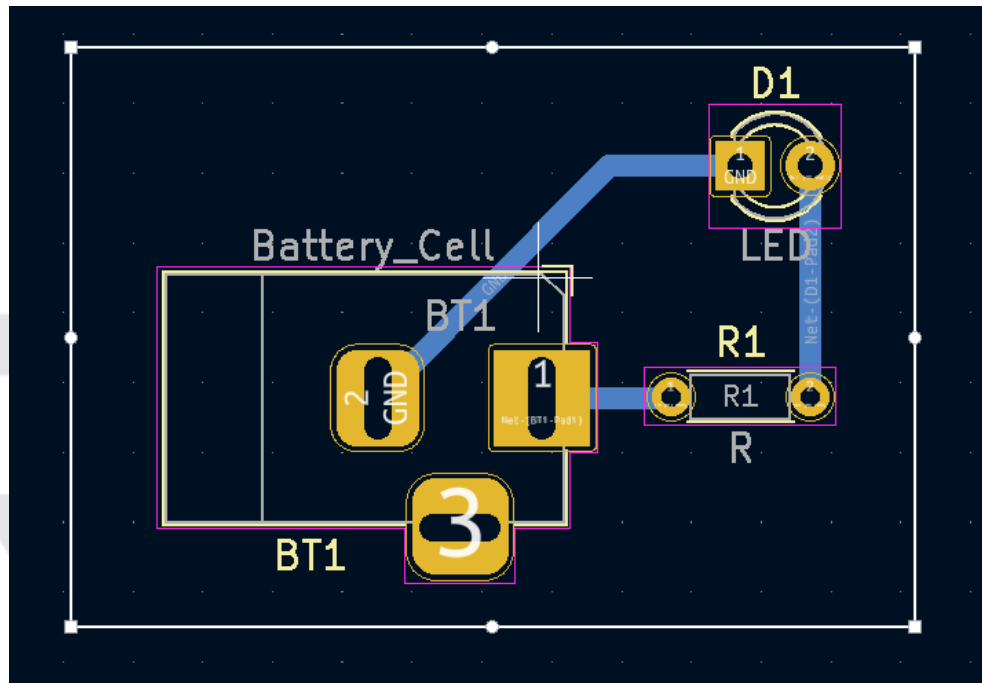


Caution

- Avoid 90-degree angle in your trace
- Don't overlap any traces.

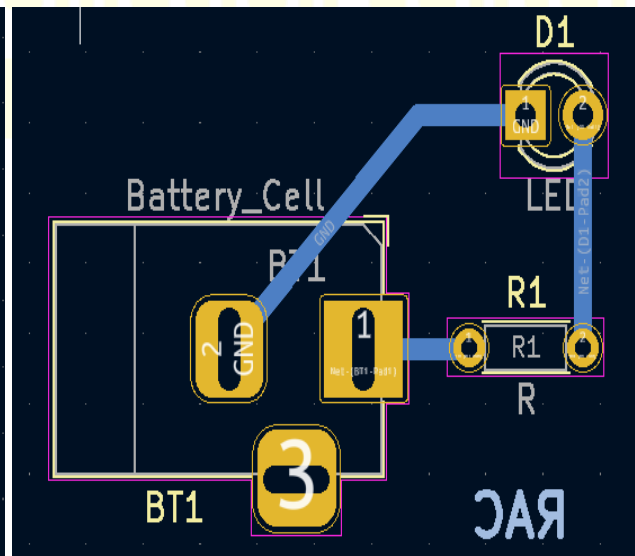
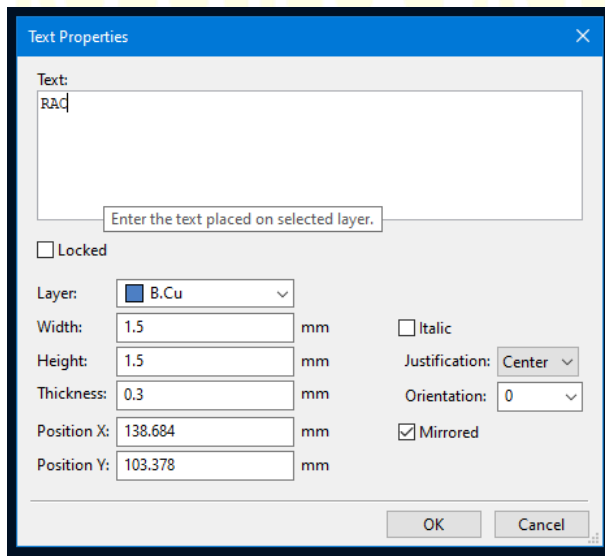
9. Edge Cuts

We can add edge cuts to make boundary for our PCB. To add edge cuts, select Edge Cuts from layers tab in right side and use draw rectangle tool or polygon tool. Now draw boarder for your board.



10. Adding Text

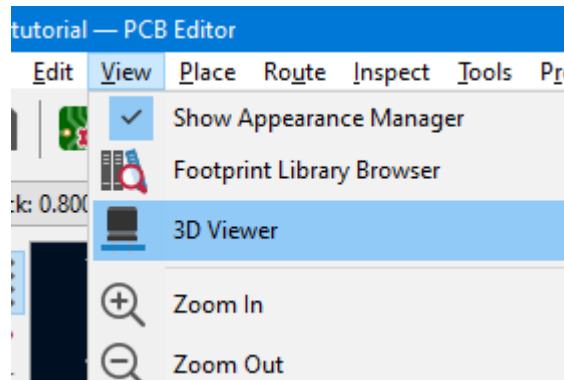
You can use text tool to add text in your board. Remember to select proper layer.

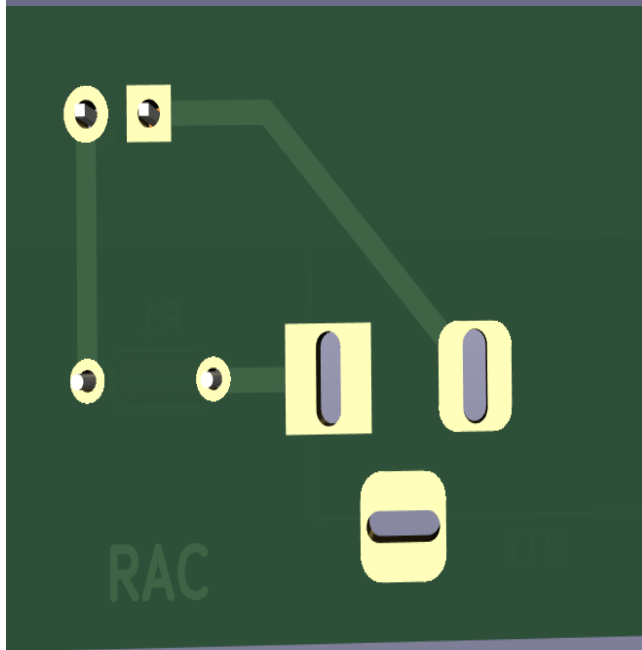
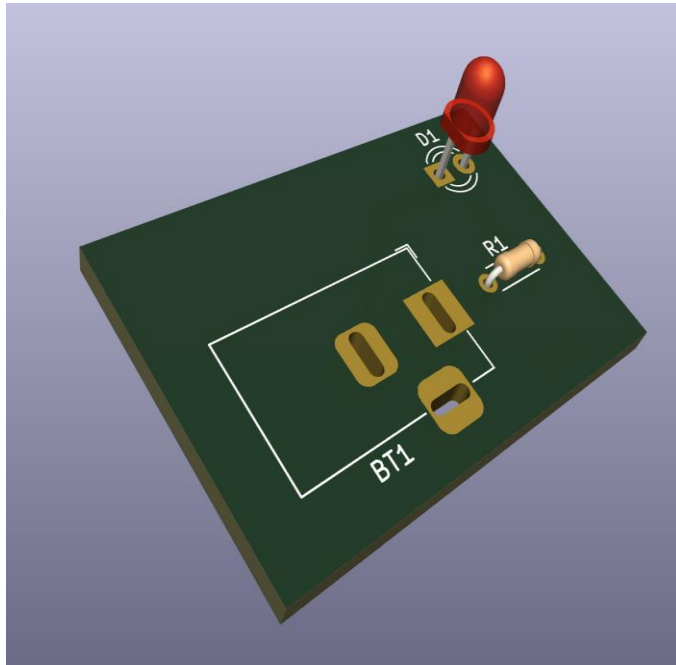


Although text might be seen flipped, it will be correct in actual PCB.

11. 3D View of PCB

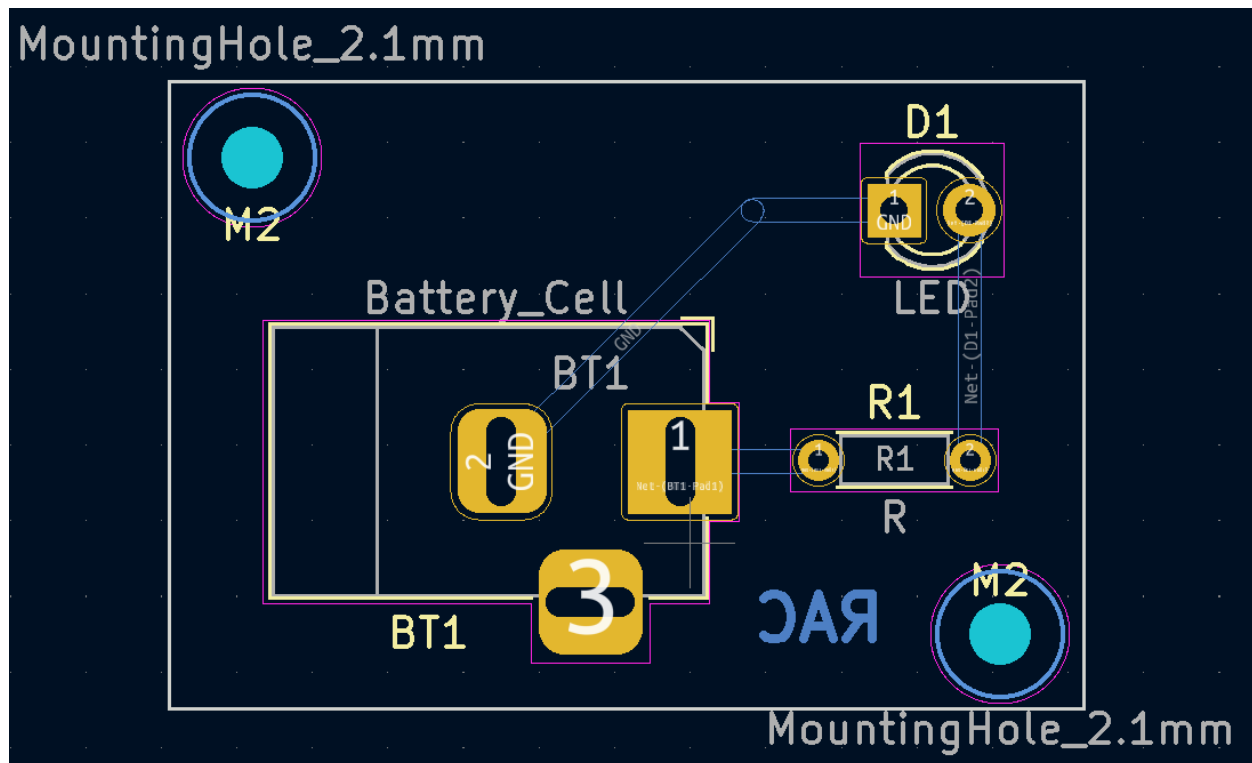
You can see how actually the board will look like. From View option select 3D Viewer to see actual board.





12. Mounting Holes

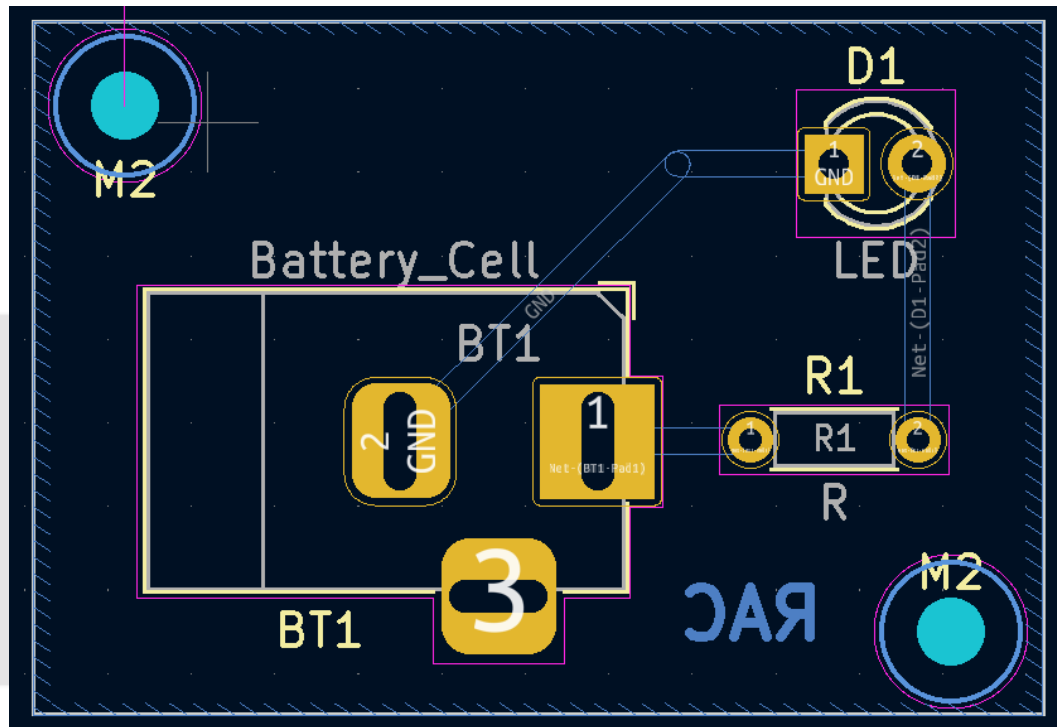
PCB must be attached with other body of device, so we can use mounting holes to mount it. To add mounting hole, select “Add footprint” from the list select the desire size mounting hole. Left click to drop required hole is place. By default, “Ref**” label is given so to change label hover over it and press ‘E’ key and give it a proper name.



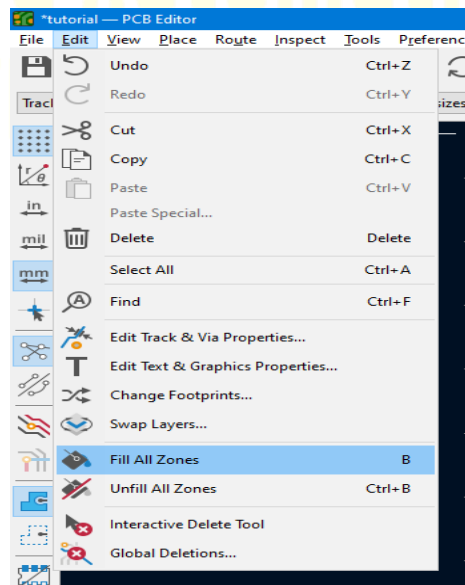
Now this design is ready to be export and print. But before exporting let's see to make common ground island i.e., instead of making separate copper line for ground we can use the unused copper as a ground.

13. Ground Island

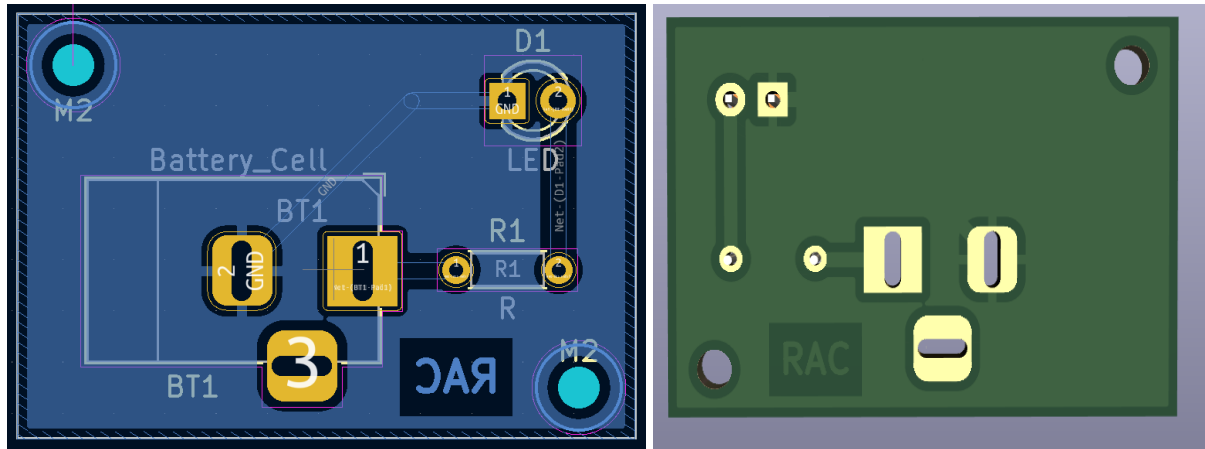
To make ground island you must have a ground while designing schematic. If you didn't add ground at that time, you can now add ground and update PCB. Now select "Add a Fill Zone" from right side with selecting proper layer, click on any corner of boundary then a new window will pop up. Select GND and press Ok. Now complete a boundary. It might look like this.



Now goto edit and select "Fill All Zones".

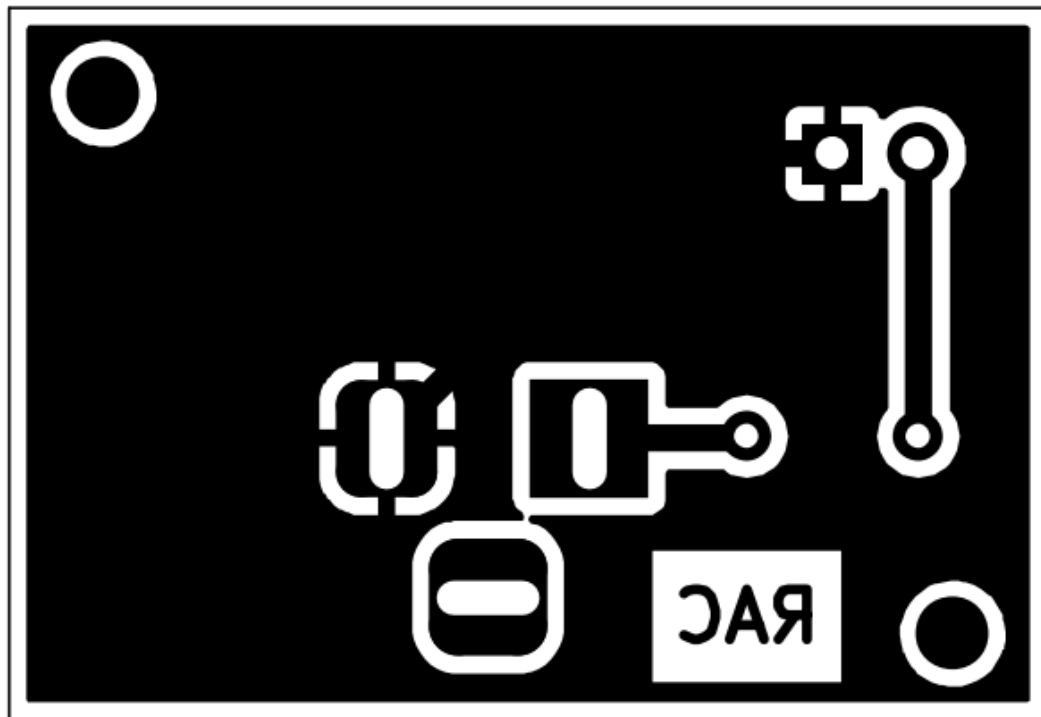


Now the circuit will like this.



14. Exporting Layout

To export, press Ctrl + P to print and select proper layers, adjust page size if necessary and press on print. Then while selecting printer, select Microsoft print to pdf the save it on desired location.



This is all you have to do in KiCad. Now you need to work on fabrication part.

15. Printing

First of all, you need to print exported pdf. But unlike printing on regular papers, it must be printed on *lossy paper* by *laser printer*. These two things should always keep in mind while printing, otherwise we won't be able to make traces on copper layer.

16. Ironing

Once printing is done, the traces made by toner must be transfer to copper layer of PCB. Thus wrap PCB by the paper you have just printed where PCB traces must be facing with copper layer. Now using heated iron, press gently up to 10 to 15 minutes.

17. Etching

After the toner has transfer to copper layer, we need to remove expose copper. Thus, removal of excess copper is done chemically. We need to prepare solution of Hydrogen Peroxide (H_2O_2) and conc. Hydrochloric acid (HCl) in the ratio of 3:1 in a plastic container. Now deep the PCB in the solution and gently stir until expose copper dissolve.

Now remove PCB from solution using tong and wash the board by clean water. After this only copper tracks will remain.

18. Drilling

Using proper size drill bits make holes for through hole components and mounting holes. Now you can start soldering the actual components on board then your PCB is ready to be used.

