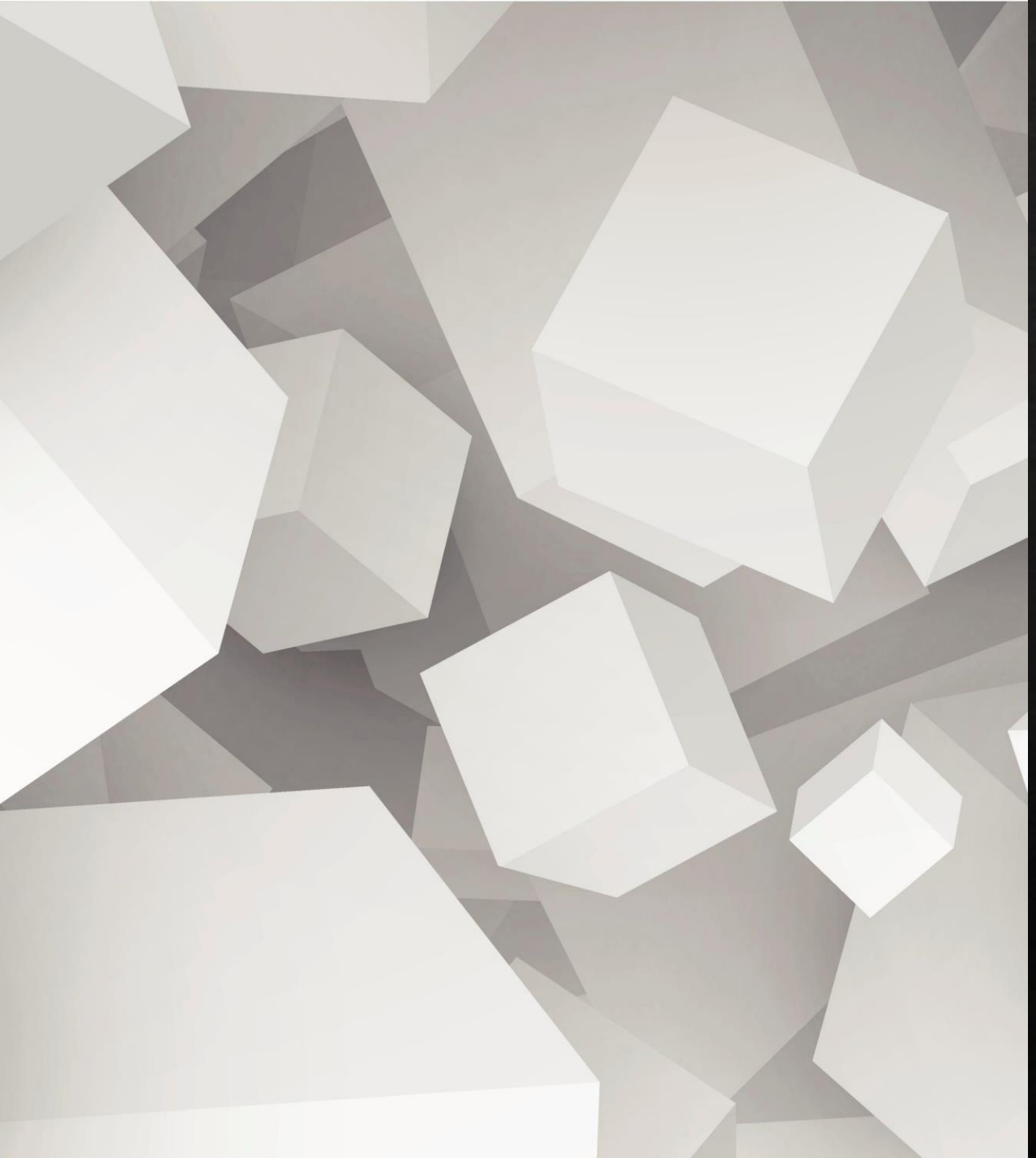
Building the Final Blocks of the Project: PCB Design

- Zain Rajani (c0752681)

Group # 8



Outline

- Until Now
- Task Objective
- Task Introduction
- Know PCB Design
- Steps of PCB Design
- View PCB Design: Zero PCB Attempt
- Conclusion
- References

Until Now.....

- Before we start the task, we will first review until now what all we have covered with respect to the project that we had proposed.
- Till the last week, we showed how we carried out the interfacing of the sensors like the IR (Infrared Transmitter and Receiver) sensor, LDR (Light Dependent Resistor) sensor, LED (Light Emitting Diode), GSM (Global System for Mobile Communication) Module and lastly how one can upload the data to cloud from the Beagle bone Black Wireless (BB-WI)
- In order to make the interfacing possible we used various functions and features of the C/C++ programming language along with some features of the python programming language for GSM Module and lastly, we made a final code in C++ language.
- To gain control over the GPIO pins of the BB-WI we used the user defined header file and used the POSIX API to get appropriate delay while suspending all the processes carried out by the BB-WI internally thus, eliminating any false or miss counts in the project.
- In all the previous meetings we demonstrated the interfacing live and proved the following:
  - On arrival of the letter the letter is sensed by the IR sensor and will make the LED high and alert the user about the arrival of the letter
  - When any letter is removed from the box the counter will be reset to zero using the LDR sensor and intimate user for the same.
  - Also the user can make a note about the number of letters in the box via the cloud service (ThingSpeak)

Task Objective

- To generate an appropriate PCB design for the letter box and make sure we get the required files generated to give to manufacturer to get the PCB ready for us.

Task Introduction

- In the previous meeting we saw how we broke the interfacing of sensors and peripherals into various tasks and thus we achieved their completion.
- In today's task we try to accomplish the final leg of making the project look like a product i.e. make sure we have minimum wires appearing on the project using PCB (Printed Circuit Boards)
- This task would be related to our task #3 of getting the schematic ready for this project. We will need the schematic prepared which were prepared in that task.
- In order to get the task completed we will use the same software that we used to generate the schematics for the project i.e. EasyEDA (Electronic Design Automation)
- As mentioned in the previous task #3 we don't need to install the software we can use it online using any web browser such Google Chrome or Firefox etc.
- At the end of the designed PCB we shall try to get the required files out so that we can hand over the files to any PCB manufacturer to get our files ready.

Knowing PCB Design

- In our previous meeting of task#3 we have seen how to use the software of EasyEDA and hence we shall not discuss them again here. But what shall discuss here is the various concepts and procedures that relate to PCB design
- The benefit of designing a circuit board for printing is that it is cleaner and more reliable than a circuit built on a breadboard or perf-board.
- The other main benefit is that it is repeatable. If you need to make ten boards, it will save you a lot of time to have a custom board printed.
- You can find PCB today in almost any consumer electronics like your telephone, TV remote, cell phones, alarm clocks and many more.

Knowing PCB Design

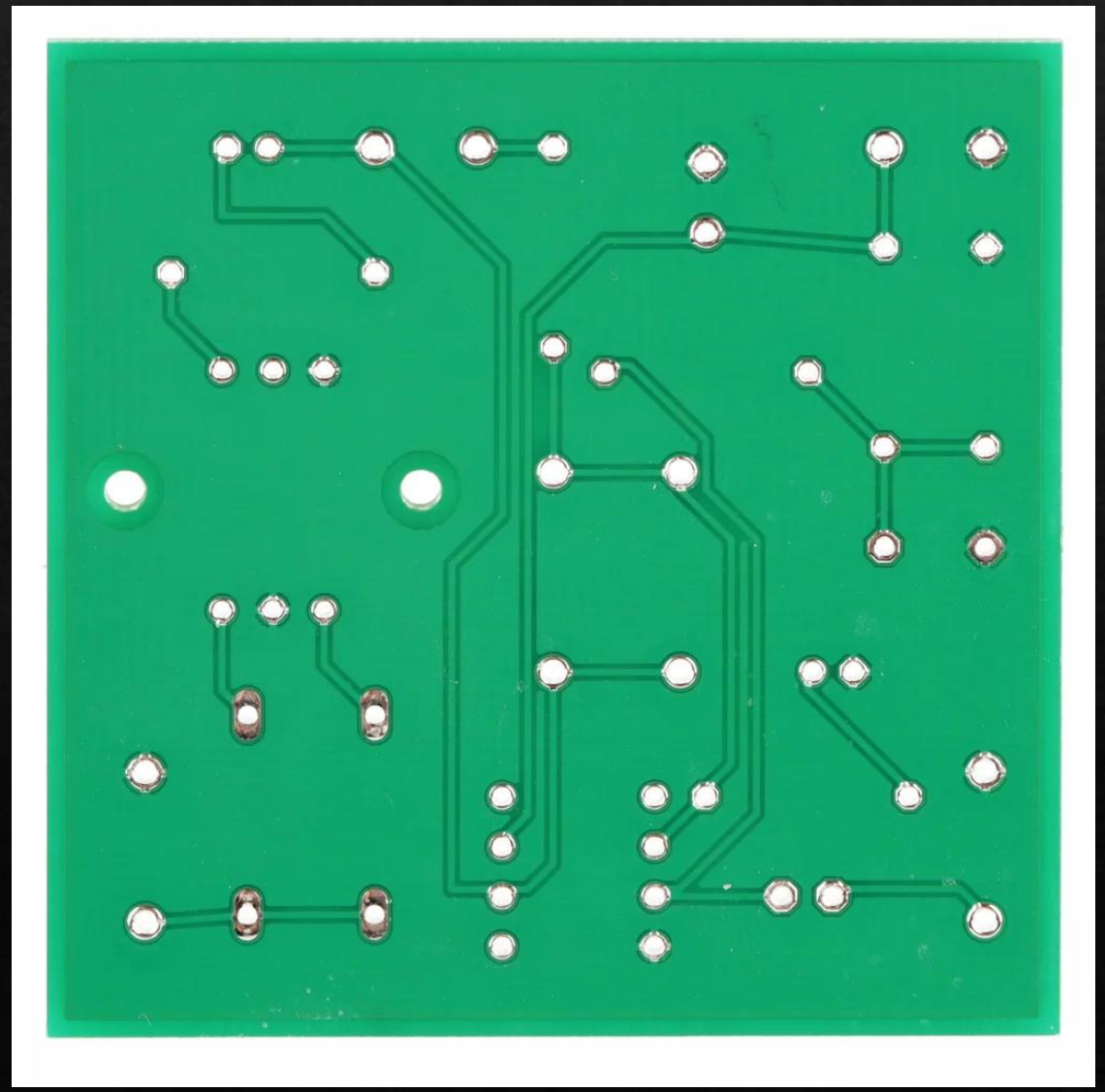
- If you notice a PCB one can see some lines on backside of the PCB which are known as traces and these traces helps one in establishing the connection between the components.
- The PCB is made of fiberglass material which is durable in nature and is made of reinforced plastic. These PCB Boards are usually green in color unless required or specified by the user.
- A PCB is made up using 4 layers which we need to be concerned and understand namely the base, copper layer, solder mask and silk green layer.

Knowing PCB Design

- The base layer of PCB is made of fiberglass material and is also known as FR4.This material provides rigidity and has good heat resistance properties. Thicknesses vary from board to board usually from 0.4 to 2mm , but 1.6mm is common with designers.
- Next is the copper layer this layer is laid on top of the base which is also known as Copper Clad. This is the layer that conducts electricity and the circuit's components are soldered to. All of the circuit's connections occur on the copper layer.

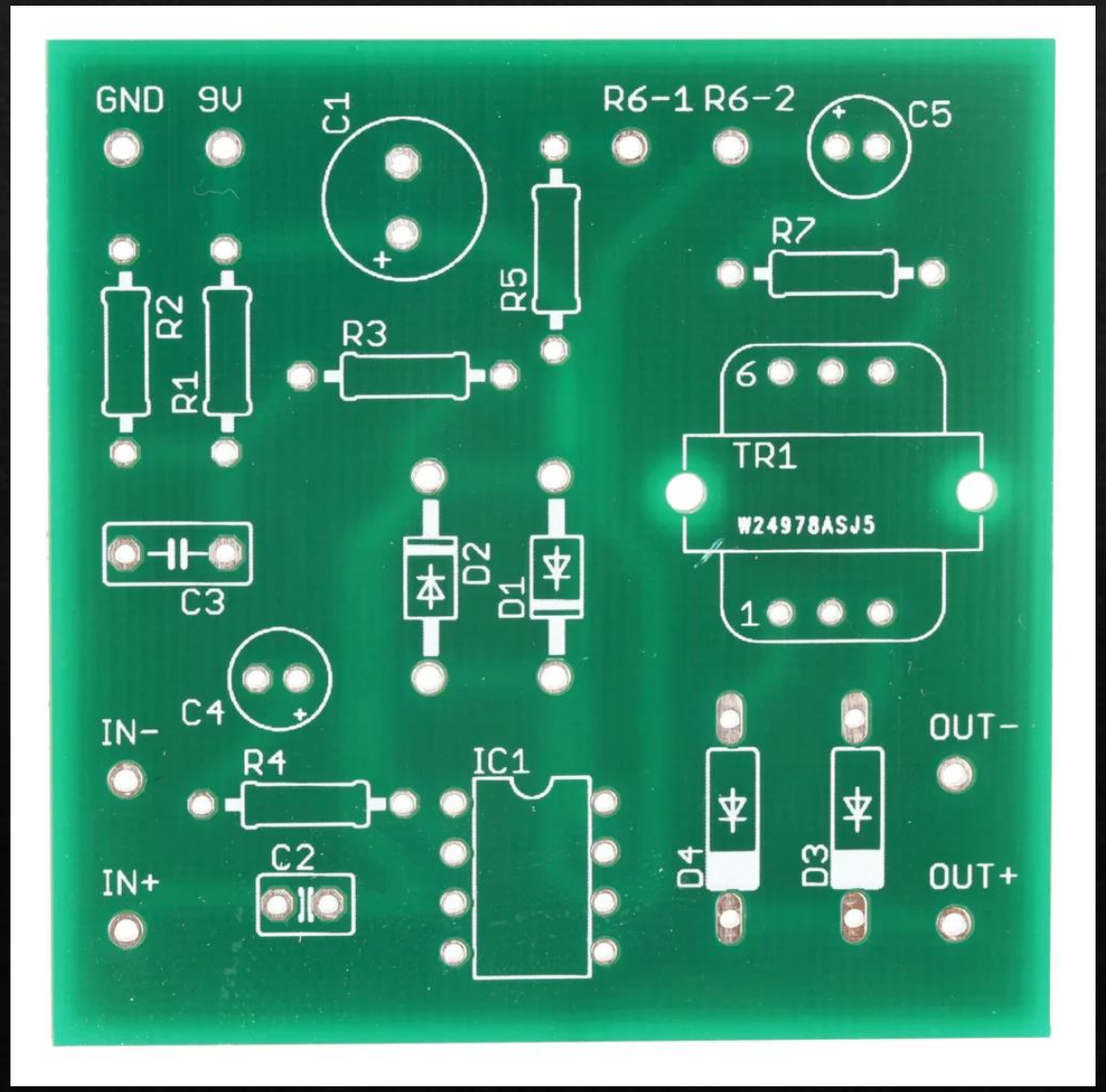
Knowing PCB

- The copper layer will help us laying out the desired tracks of the connection. That if we make a PCB ourselves, we will print the design and then iron it on the board then we will remove the excessive copper from the board thus, we will only be left with the desired tracks.
- The solder mask is a layer of colored resin strategically laid over the copper layer to insulate the parts of the circuit that do not get soldered to. Typically the solder mask is green, but it can be all kinds of different colors.
- One can also make the solder connection by oneself using the hand driller and get the holes on the board ready for the components that we need to connect.



Knowing PCB Design

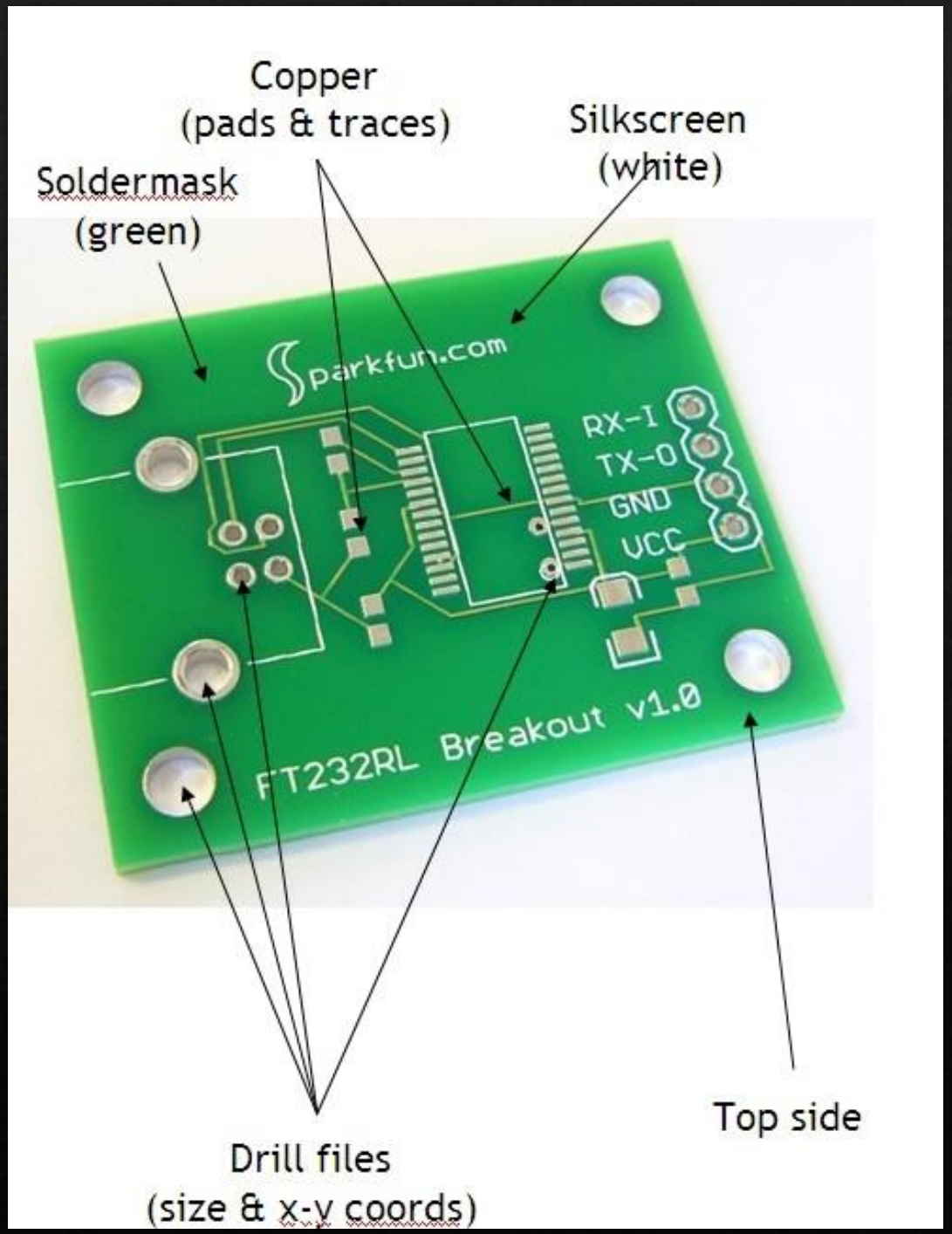
- After the solder mask we have the silk layer, and this layer is like a guide on the PCB which will help us understand as to which component needs to be connected where and how.
- Usually if we see a circuit board we will have two sides the bottom and the top side and all the 4 layers could be present on both the sides. But usually the last layer that is the silk layer is skipped at the bottom of the surface.
- More advanced circuit boards, such as you might find in a laptop or smartphone can have many conductive layers or "sides" sandwiched together.



Knowing PCB Design

- The component that usually get connected on the PCB are either of the type through-hole or surface mount.
- The through-hole component are round metal plated holes that serve to both connect the top and bottom copper layers together and give you a surface to solder the component to the board.
- Surface mount are small parts with metal tabs that get soldered directly to the surface of the board.
- During the task we will come across the terms like components, symbol, package and footprint. Thus we need to understand these terms before we start the to understand the designing procedure on the software

Summary of PCB Design



Terminology

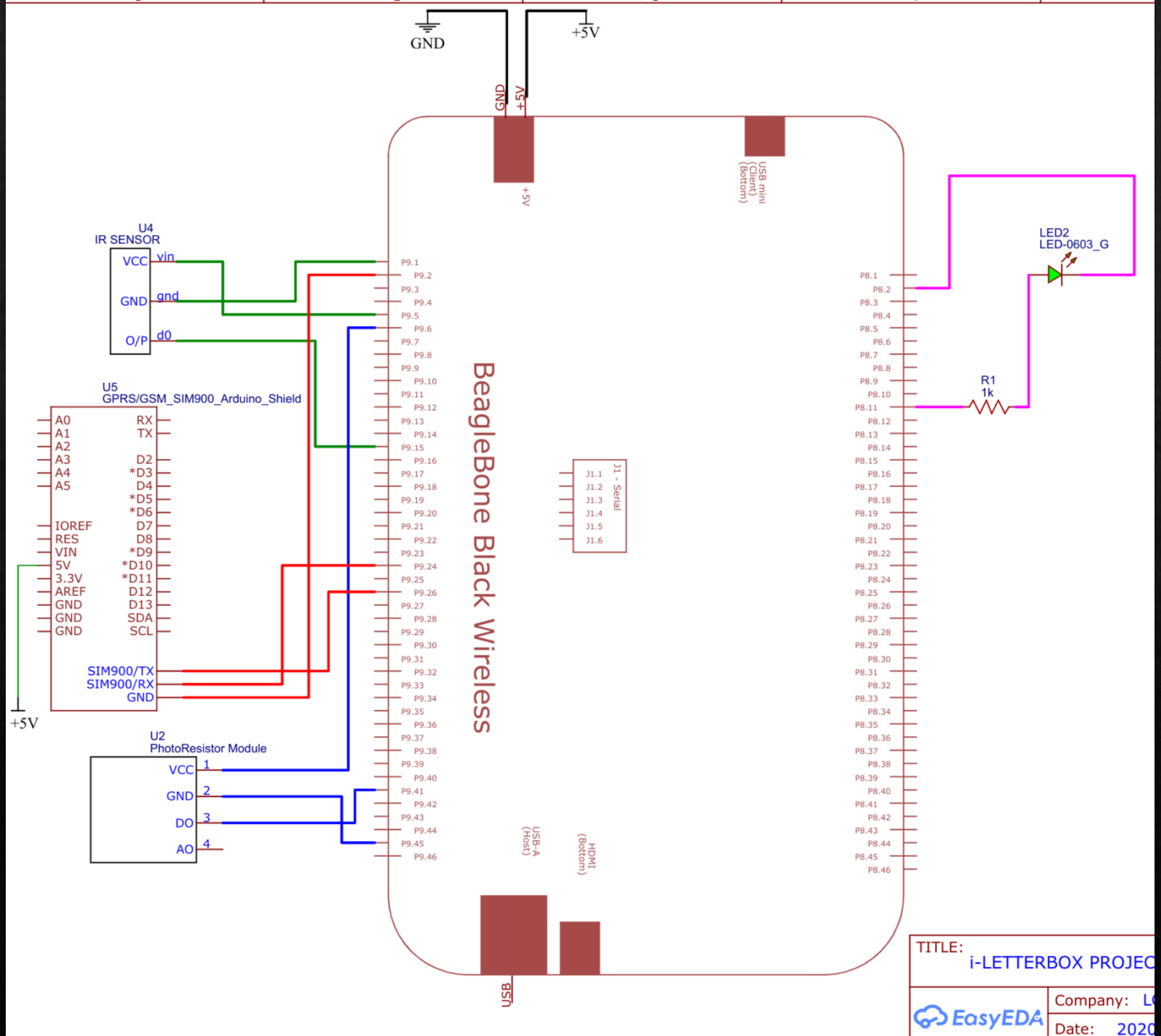
- A **component** refers to any part used in a circuit. This is always referring to the part itself, and always has the same symbol. It can and often does have a different package and footprint.
- A **symbol** is used when drawing a schematic as a stand-in representation of a component. It has all the necessary pin connections required by the component.
- A **package** is the 3-dimensional shape of the component, and indicates how it mounts to the circuit board (through hole, surface mount, or otherwise).
- A **footprint** is the 2-dimensional layout of the package atop the surface of the board. It consists of a silkscreen layout, name label, plated through holes or pads, and any other necessary layout information. This is one of the important term and is very important when making a PCB Design

PCB Design Flow

PCB Design Flow

- In order to design the PCB for our project we require first to research and look for the proper part of the project.
- Next, then we need to draw out the schematic of the project that is define the way in which the connection shall be made with the project.
- Once the schematic is ready, we can then convert that into the PCB and then lastly, we can verify the design and check for the connection paths if all is good then we shall manufacture the PCB else we might have to repeat steps of changing the schematics.
- After these stages, you can export your design to an industry standard format (Gerber) from which a physical board is fabricated.

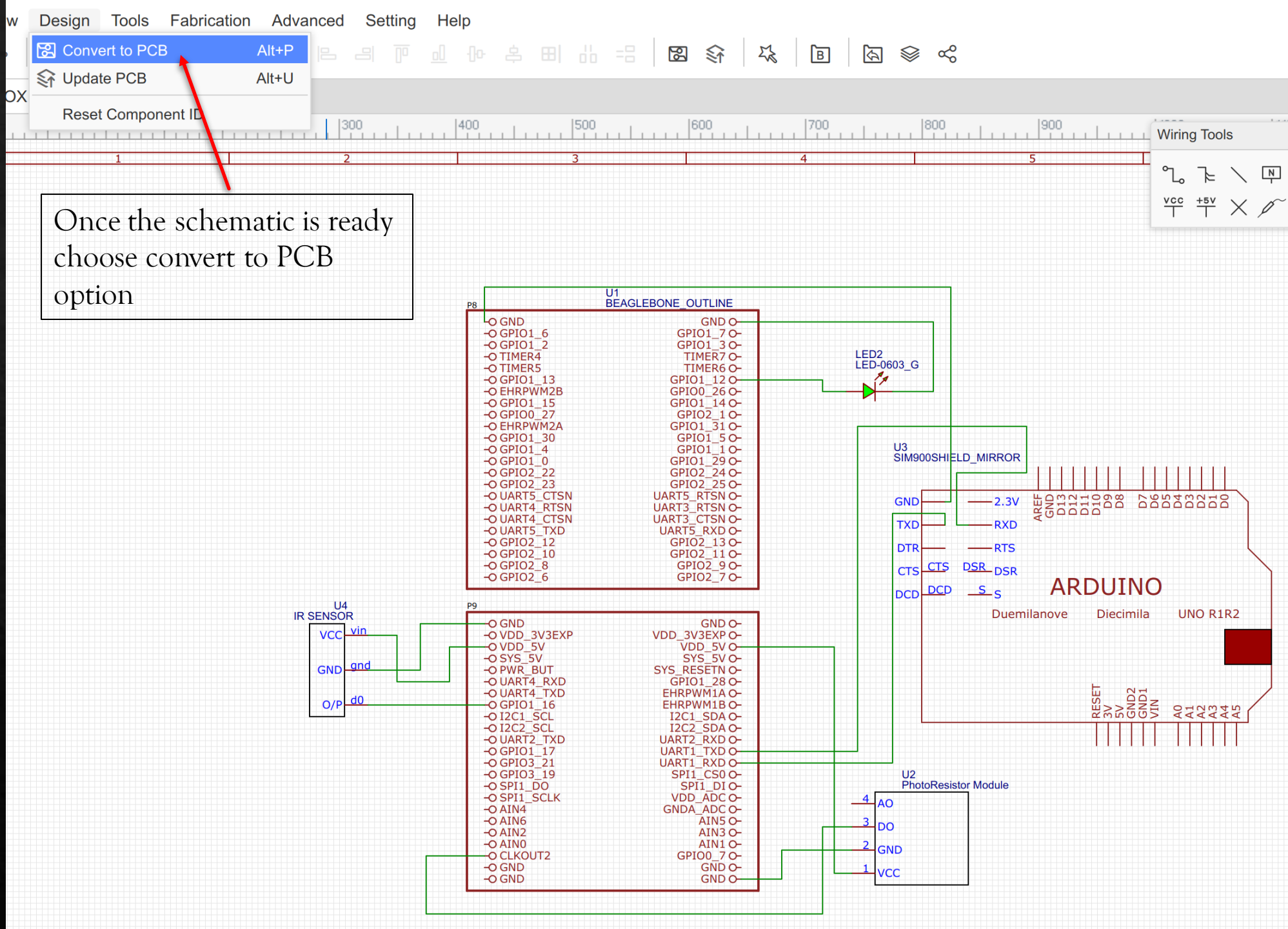
Project Schematic



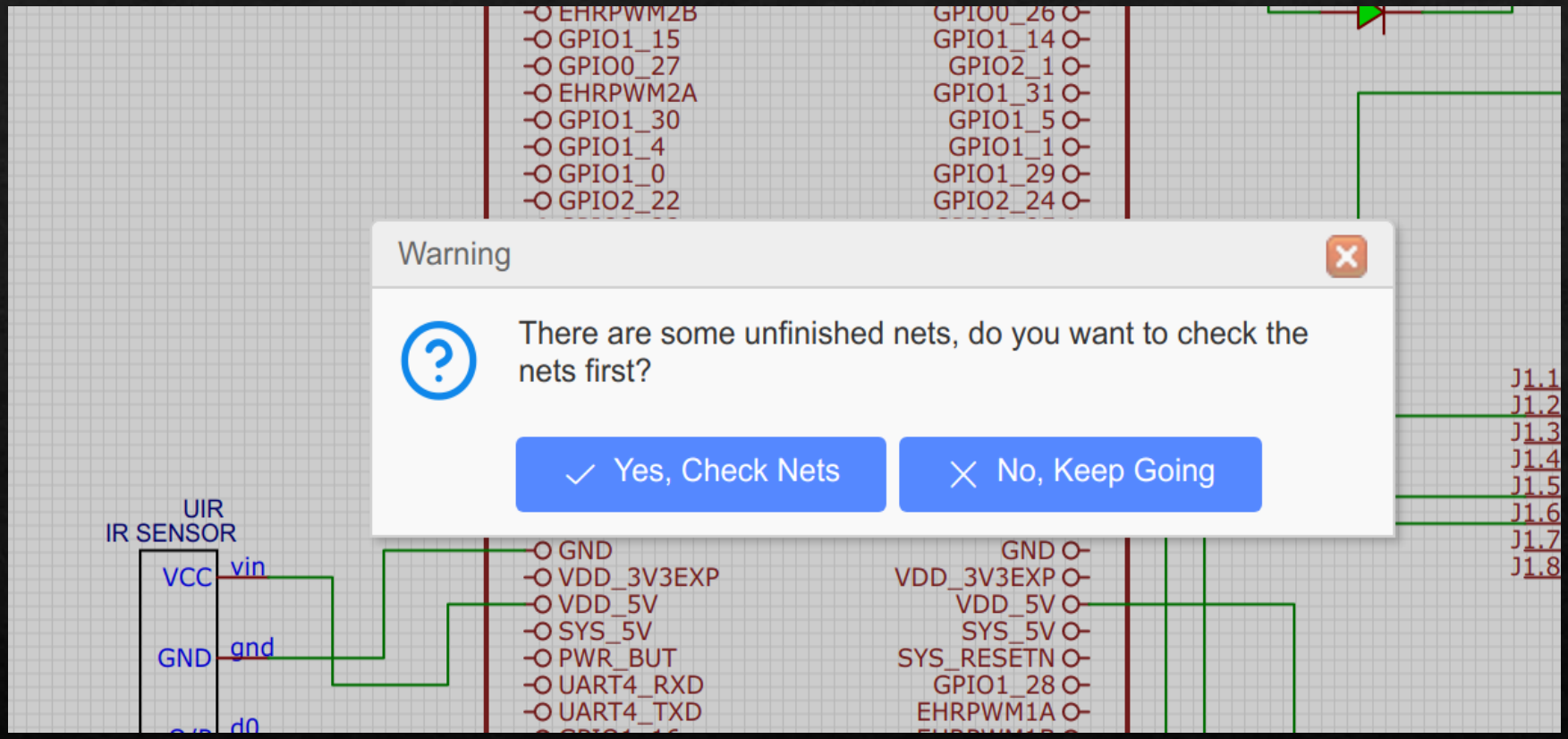
Steps for PCB Design

- Once our schematic was ready, we now were ready to generate the PCB for our desired project.
- When we tried to get the schematic converted to PCB, we faced some issues because some of the components that we used did not have a footprint thus we couldn't generate the PCB.
- Thus, after some changes in schematic we were successful in generating the desired PCB. The changed schematic can be seen on the following slides.

Steps for PCB Design

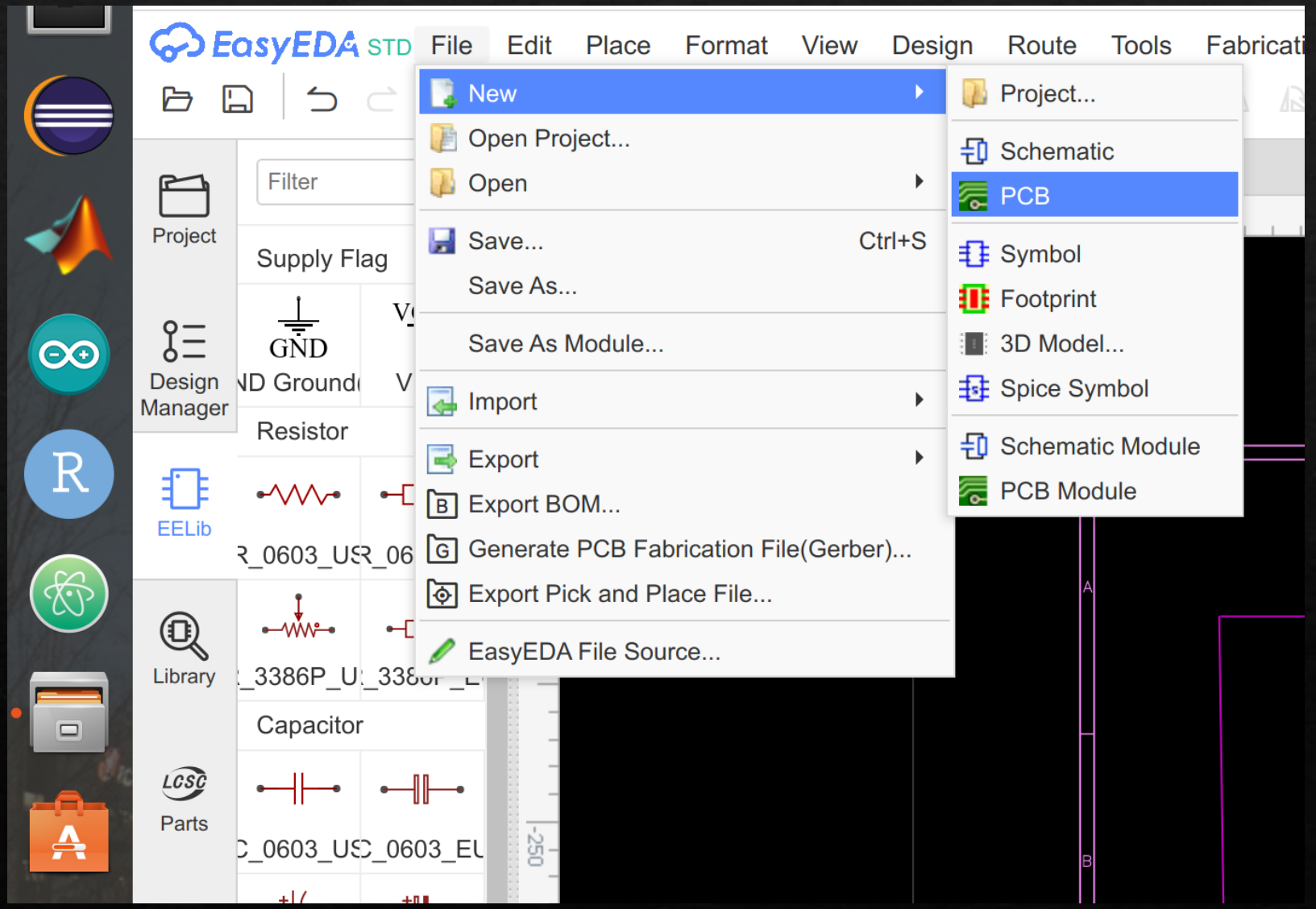


Steps for PCB Design



Steps for PCB Design

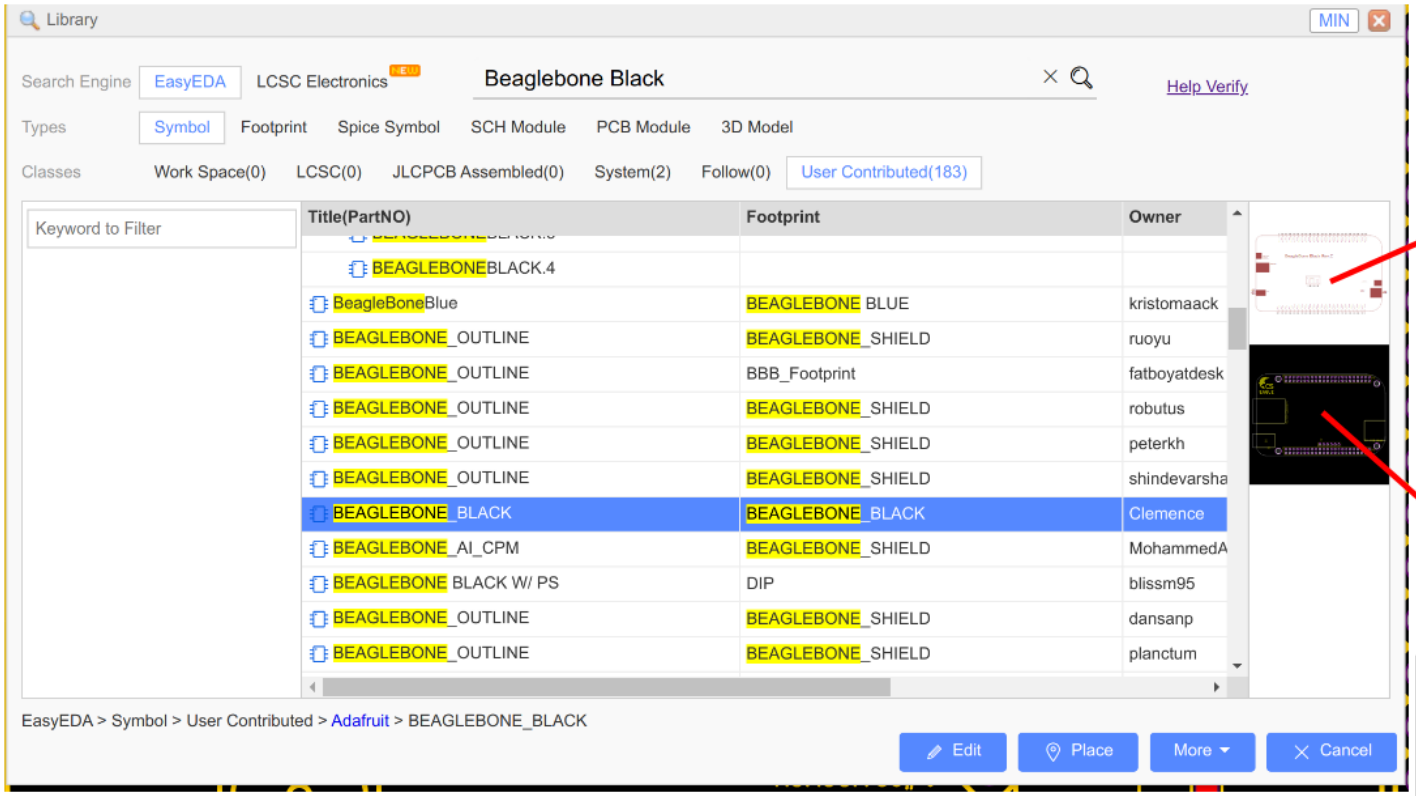
- An alternative to make the PCB design incase we don't have the schematic is not ready we can do as follows.



Steps for PCB Design

- Next, in case you want to directly make PCB without the schematic we must add the components the same way we added the component for the schematic capture.
- While selection we must make sure that the component has some footprint or PCB layout else, we won't be able to establish the connection. Once we have all the components ready, we can start the connection using the connect pad to pad (O) using the PCB tools.
- Once connection is done, we can use the autoroute option to layout the trace or we can manual do it using PCB tools. These steps are shown in the following slides.

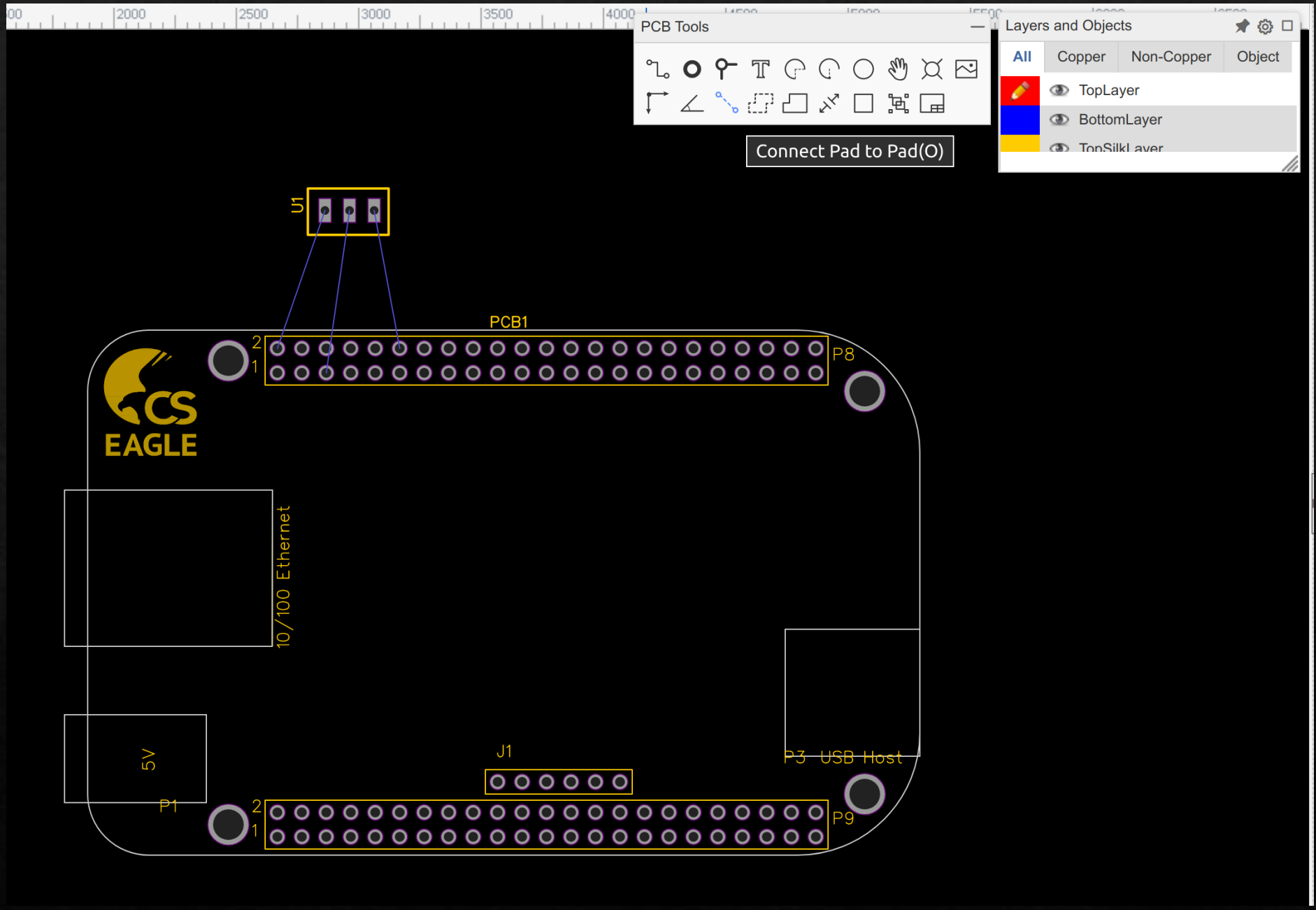
Steps for PCB Design



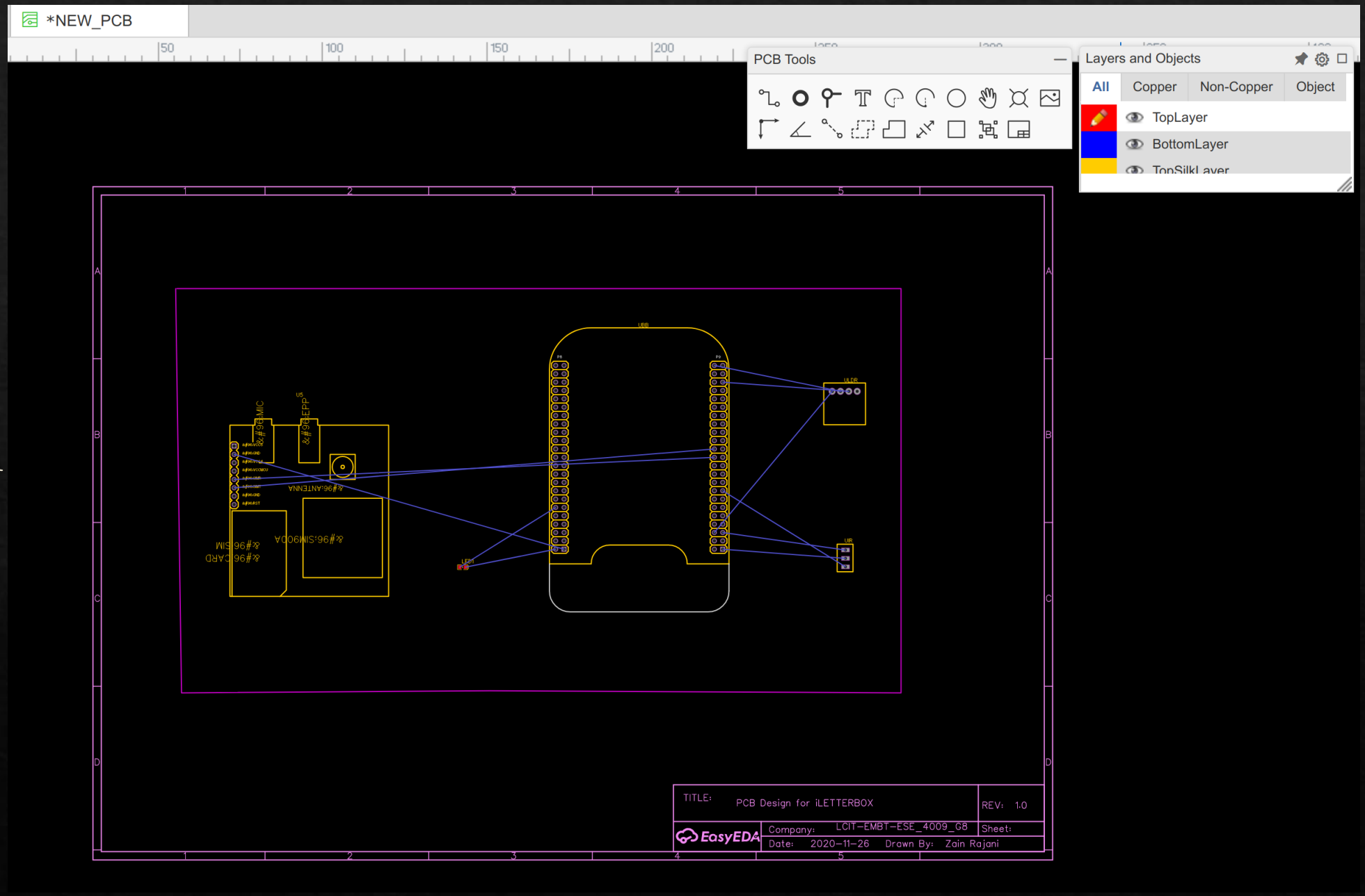
Schematic Symbol

Footprint of the component thus PCB should be possible for this component

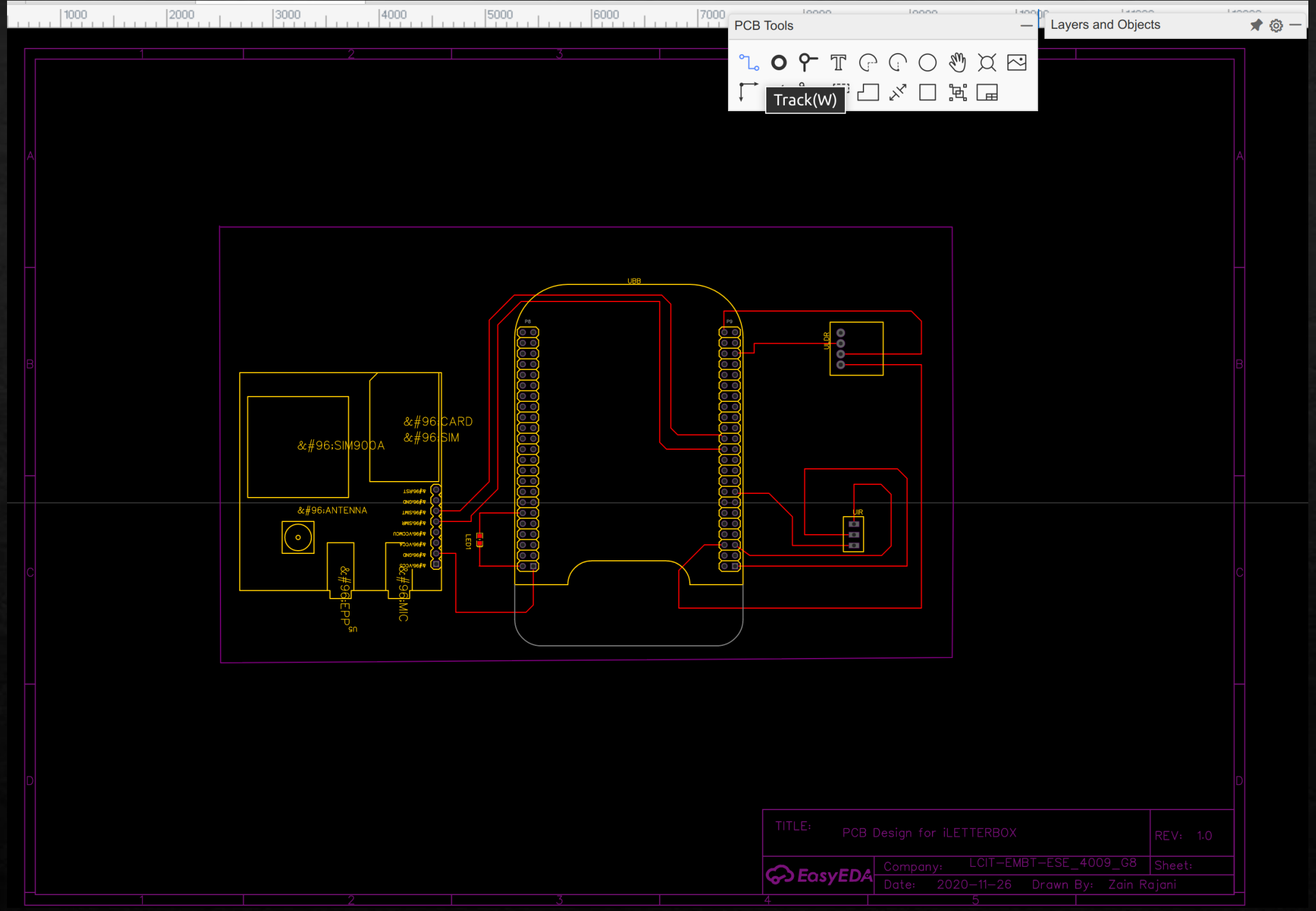
Steps for PCB Design



Steps for PCB Design

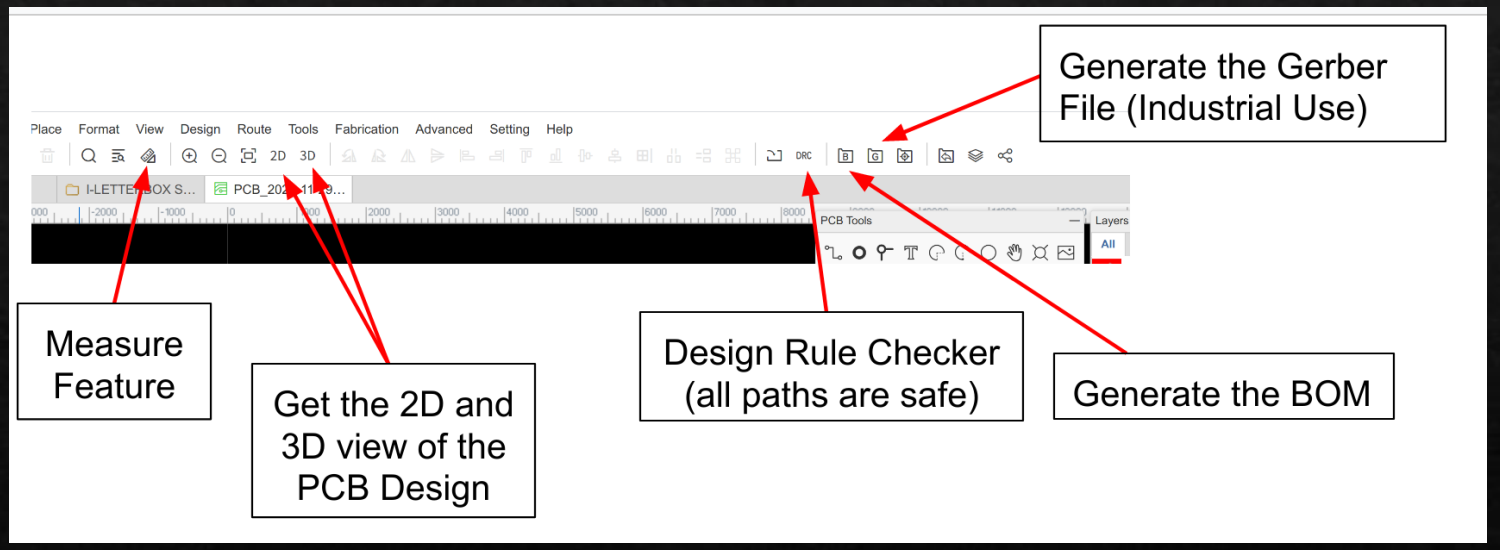


Steps for PCB Design

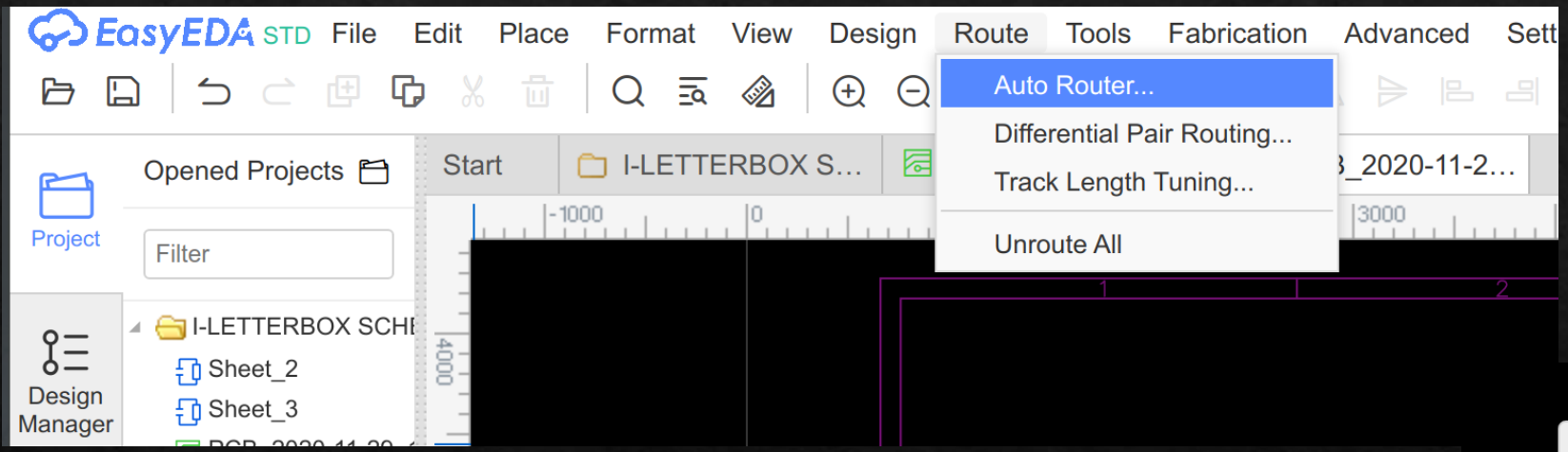


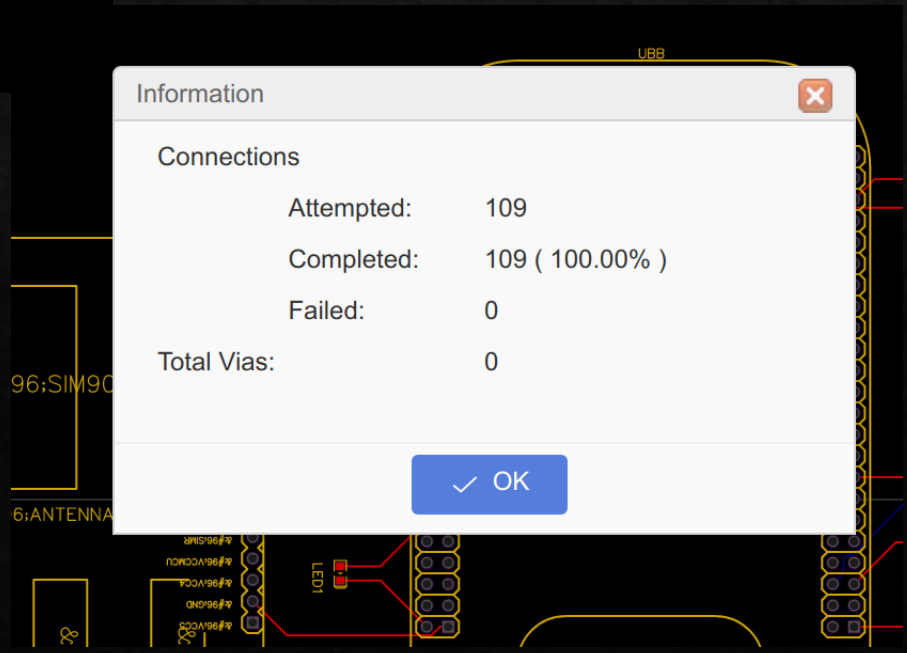
Steps for PCB Design

- Once we have made the proper routing, we can then generate the desired files which will be used for the manufacturing the PCB.
- Through this we could generate the BOM (Bill of materials and check if the tracks are proper using the DRC Design rule check



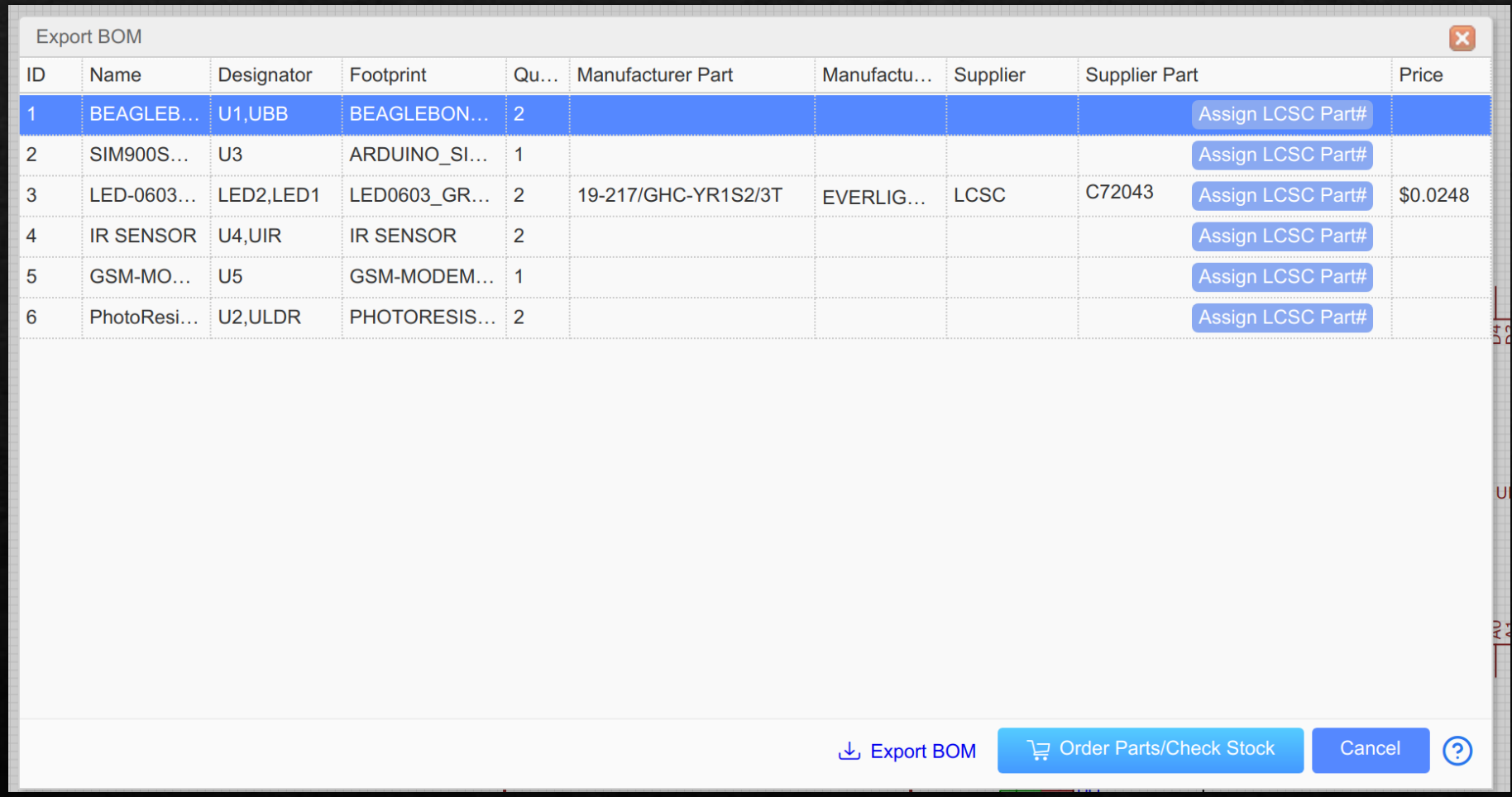
Steps for PCB Design: Auto Route





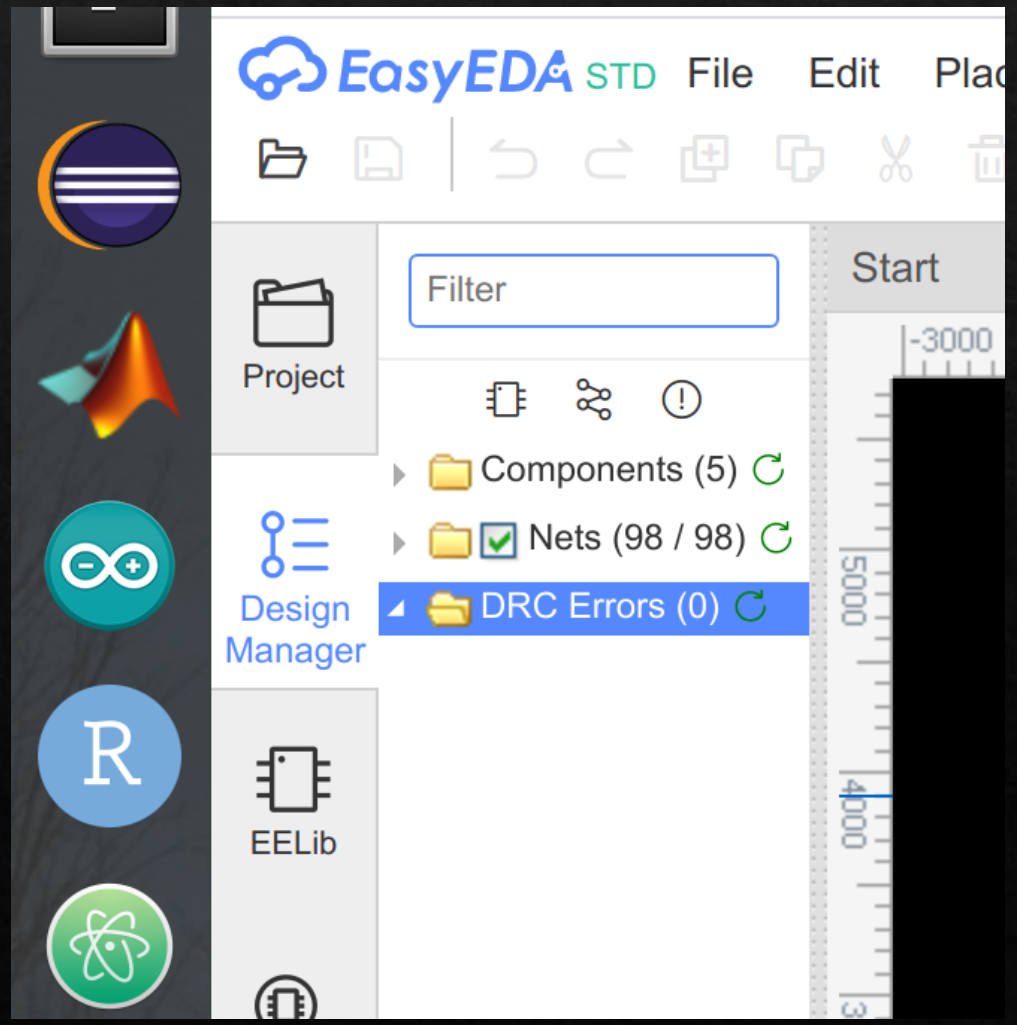
- In order to get the correct clearance between each trace, auto routing can be done after connecting each component.
- It provide effective routing of traces by minimizing vias.

Generating the BOM from PCB Design



Steps for PCB Design

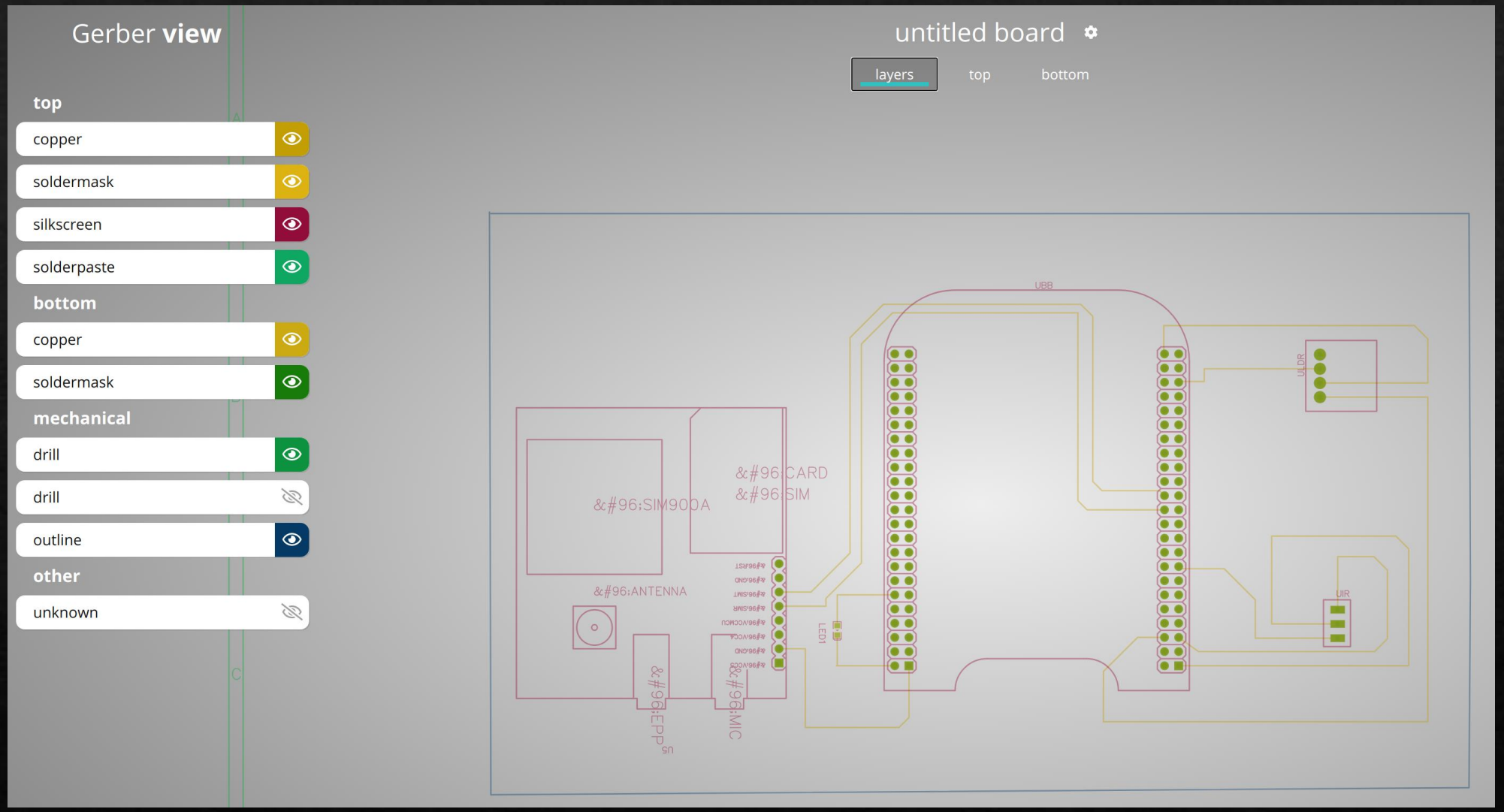
- If there is no problem with the DRC you can verify from the left pane of the software as shown in the figure on the right.



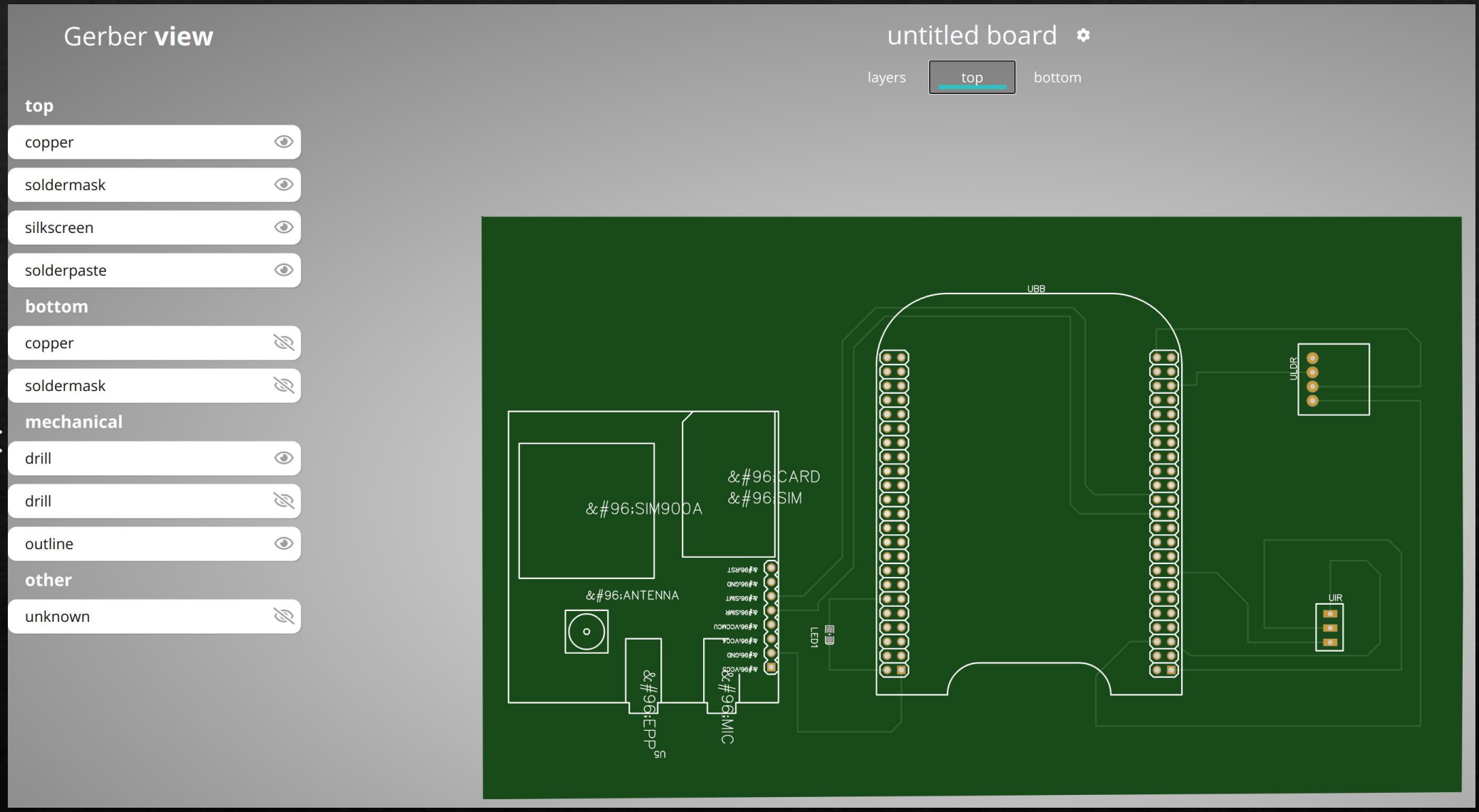
Steps for PCB Design: Gerber File

- A Gerber File is a file format used for PCB manufacturing. Fabrication machines can use these files to layout electrical connections such as trace and pads. The file also contains information for drilling and milling the completed circuit board.
- The standard file extension is *.GBR* or *.gbr* though other extensions are also used.
- Though the EasyEDA generates the Gerber files but one cannot view it using the same software as it is no longer supported. Thus we needed some third-party software to view the Gerber data. This file helps us to see how the final PCB would look like after manufacture.
- It is recommended that we use the gerbv tool, but we can also use the FlatCAM due to other alternatives avaliable we used a web-based tool known as Online Gerber Viewer by PCBway

Gerber File View: Layers



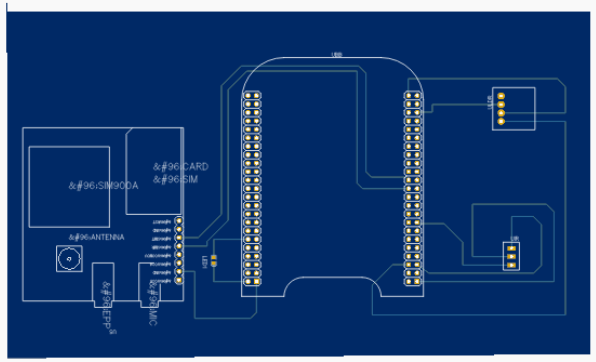
Gerber File View: Top



Gerber File View: Bottom

Ordering PCB Design

Generate PCB Fabrication File(Gerber)



Layers: 2

Dimensions(Estimated): 175.51mm x 104.39mm

PCB Qty: 5

PCB Thickness: 1.6

PCB Color: Green

Surface Finish: HASL(with lead)

Copper Weight: 1oz

Manufacturer: JLCPCB

PCB Price: **$9.3**

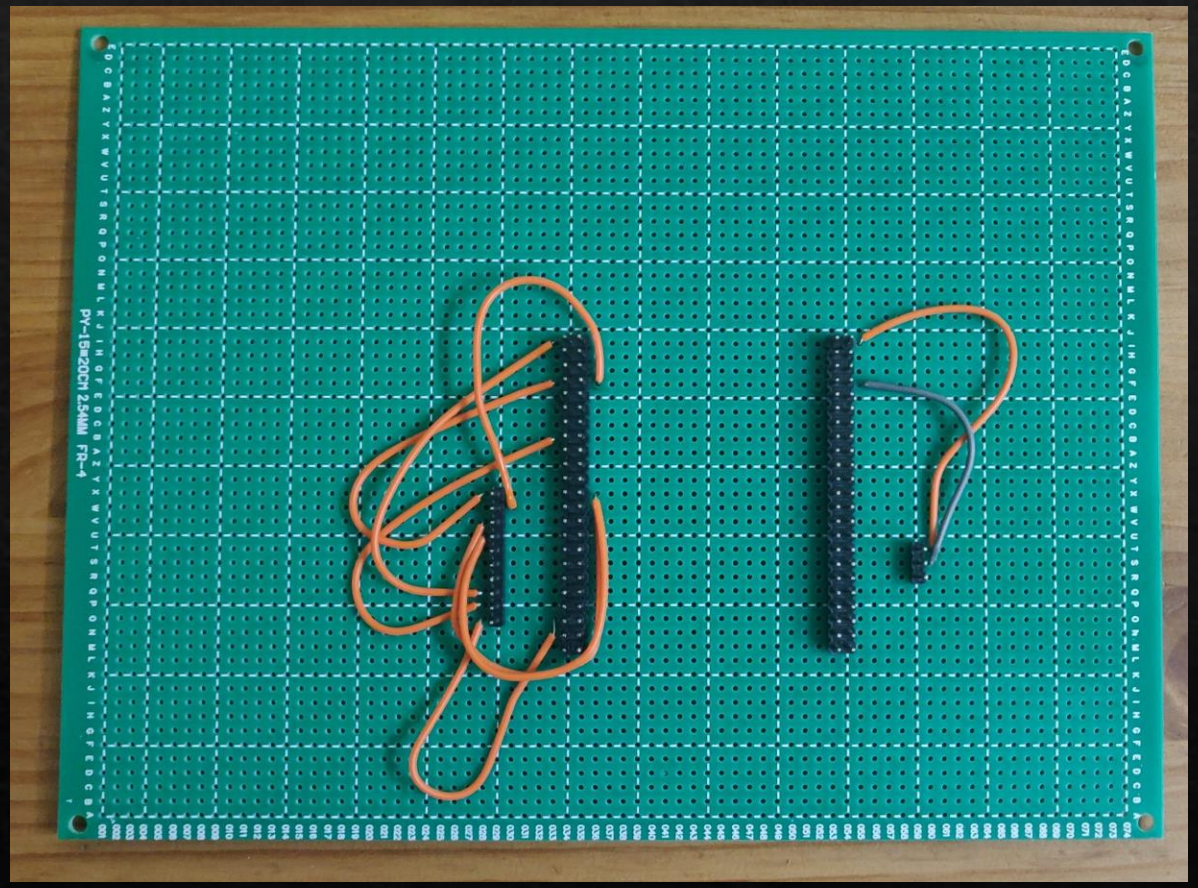
Estimated Delivery Time: **3-7 days**

Gerber View

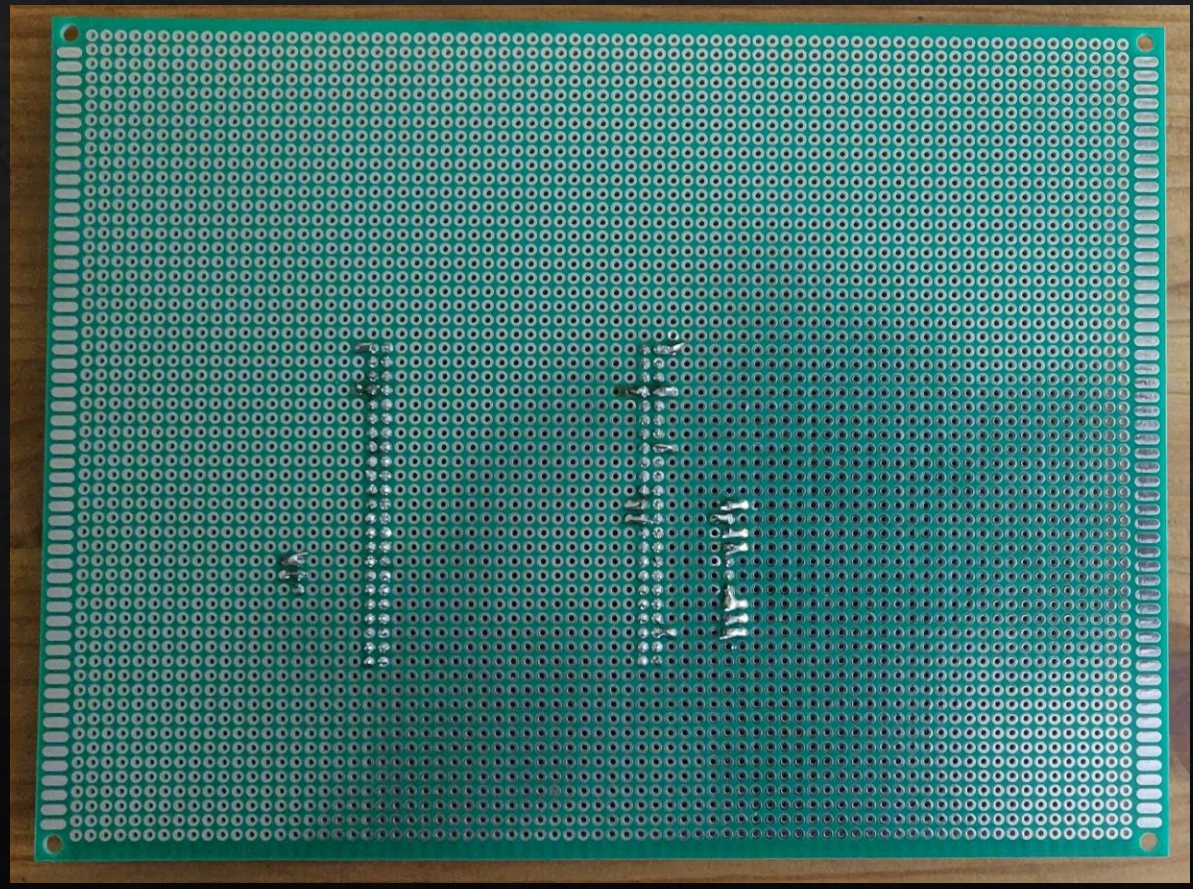
Generate Gerber

Order at JLCPCB

View PCB Design: Zero PCB Attempt



Top View of Zero PCB



Bottom View of Zero PCB

Conclusion

- This is one of the final task we were supposed to achieve successfully. During this task we faced some issues to get the PCB design but at the end completed and generated all the required files for the PCB manufacturing design.
- With completion of this task we are now ready to have the PCB designed manufactured from some manufacturer.
- Thus we tried to contact some of the manufacturer to get our design ready but due to cost constraints and due to quantity issues, we couldn't get the PCB manufactured and some of them had some delivery time issues due to the current circumstances.
- Due this issues we have made the zero PCB ready for the project but also placed the order for the same PCB with JLCPCB which is directly related to the EasyEDA software and we have ordered to get the same PCB of 5 numbers. If it is delivered on time, we shall use it for the final demonstration else, we can use the zero PCB that we have designed and soldered as shown on the previous slide.
- It is promised to us that the PCB will be delivered within 7-10 days (Estimated)

References

About EasyEDA. (n.d.). Retrieved November 29, 2020, from https://easyeda.com/page/about

An Easier and Powerful Online PCB Design Tool. (n.d.). Retrieved November 29, 2020, from https://docs.easyeda.com/en/PCB/Canvas-Settings/index.html

PCB Concepts and Materials. (2017, September 21). Retrieved November 29, 2020, from https://www.instructables.com/PCB-Concepts-and-Materials/

PCB Design Fundamentals: Overview. (n.d.). Retrieved November 29, 2020, from https://www.ni.com/tutorial/10580/en/

PCB Designing Using EasyEDA. (2017, September 18). Retrieved November 30, 2020, from https://www.instructables.com/id/PCB-Designing-Using-EasyEDA/

Printed Circuit Board Design Flow. (n.d.). Retrieved November 29, 2020, from https://www.pcbway.com/blog/Engineering_Technical/Printed_Circuit_Board_Design_Flow.html