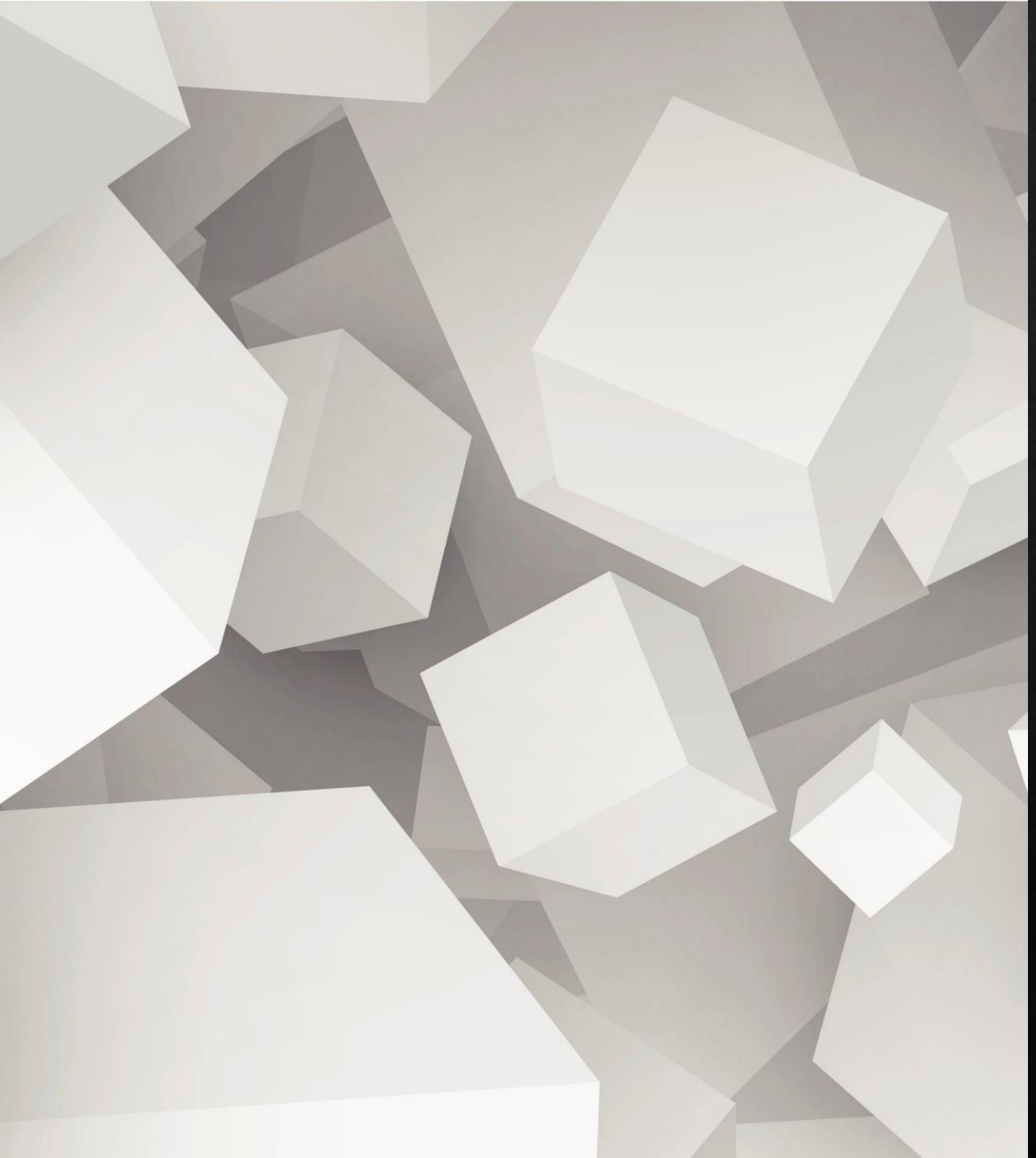
Schematic Capture for I-LETTERBOX

- Zain Rajani (c0752681)

Group # 8



Outline

- Task Introduction
- Tools Requirement
- Schematic Design: EASYEDA
- Project Schematic
- Conclusion
- References

Task Introduction

- Schematic Captures are typically associated with electrical circuits. They are also defined as the wiring diagram or circuit diagrams.
- These diagrams help one understand clearly about how the connections are made to know how may pins they will require from the main processing units etc.
- In these diagrams the lines represent the actual wiring that occur, and all other components shall be represented with their electrical symbols.
- In electronics, having a schematic diagram on hand may help a user design an entire circuit before building it, or troubleshoot an electronic that has stopped working.
- Schematic diagrams may also be used to explain the general way that an electronic functions without detailing the hardware or software used in the actual electronic.

Task Requirements

- Software Installed: EASY EDA
- Files Installed for the sensor/devices, etc.
- Physical Presence of Components like:
  - IR Sensor
  - LDR Module
  - Main Processing Unit (BB-WI)
  - GSM Module
  - LEDs

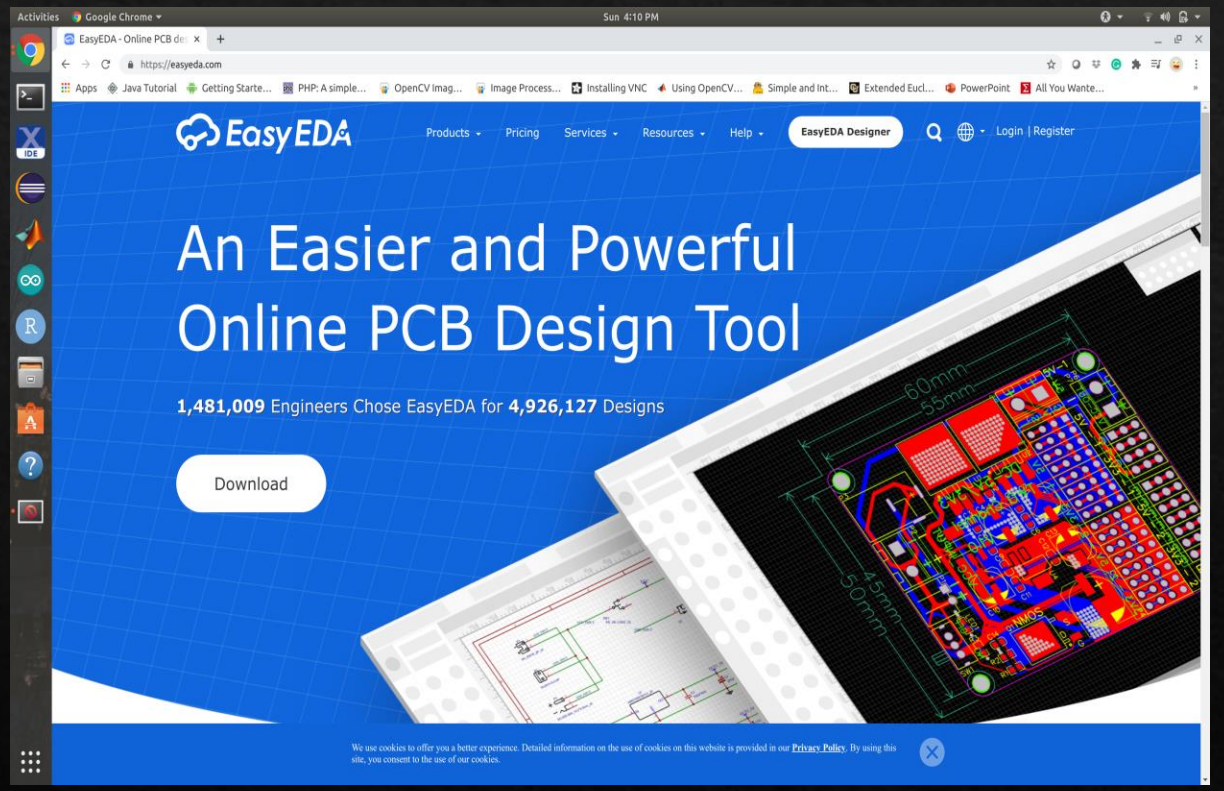
**Note: Physical Presence of components just required to know which pins are present on the device but even a datasheet of those devices will work*

Schematic Capture Software: EASYEDA

- For designing our circuit connections and as promised in our proposal we shall be using EasyEDA Software. Thus, before we get started with implementing the design, we must first understand its features and usage.
- EasyEDA is a web based Electronic Design Automation tool (EDA) thus, there is no requirement to download the software and use it. But if one wishes to, he/she can download (Desktop Client) and use it. The desktop client is compatible with Linux, Windows and Mac OS.
- The biggest advantage of EasyEDA is that it is an open source online tool which just requires a browser such as Google Chrome, Firefox, Safari, Internet Explorer and others.

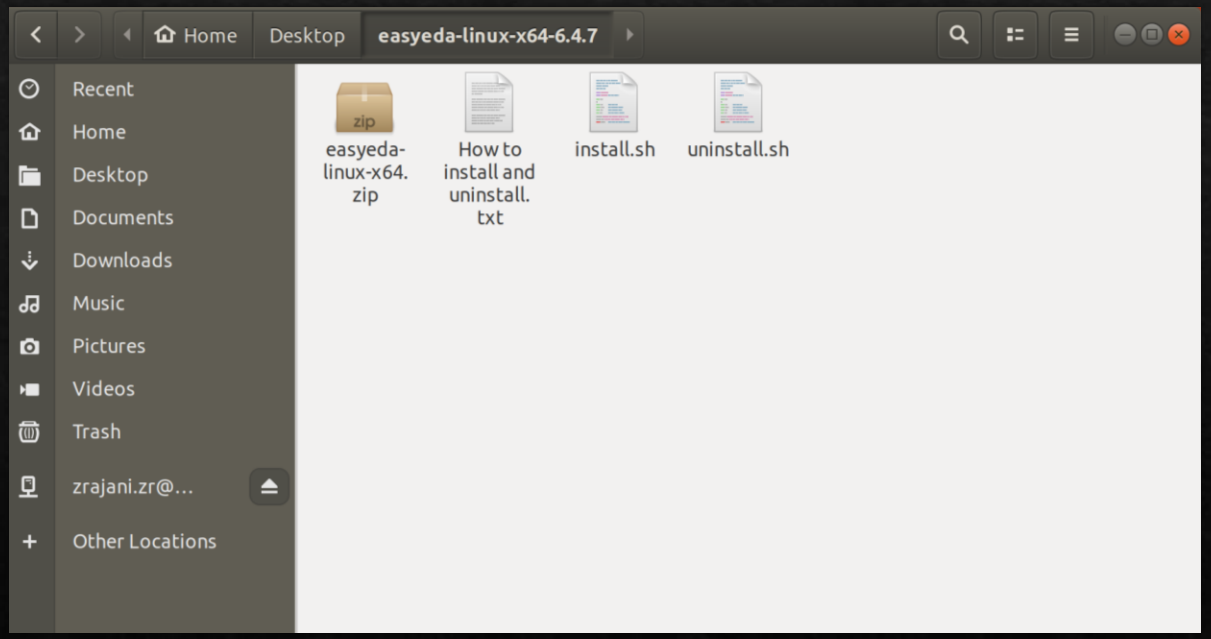
Know EasyEDA

- As a person who doesn't have much knowledge about PCB Design EasyEDA could be a good tool to use as it is user friendly and simple to use as the all the components needs to clicked and dropped (drag and drop).
- One advantage of this site is, it has large numbers of open source projects, which is easily available from the internet and very good tutorials are available in EasyEDA official website.
- It has more than 500000 libraries with symbols and footprints of components and you will ever need, there is also a feature to create your own component symbol and footprint if needed.
- One more advantage is, it can support other software libraries and schematic which includes Altium, EAGLE, LTspice, and DXF.

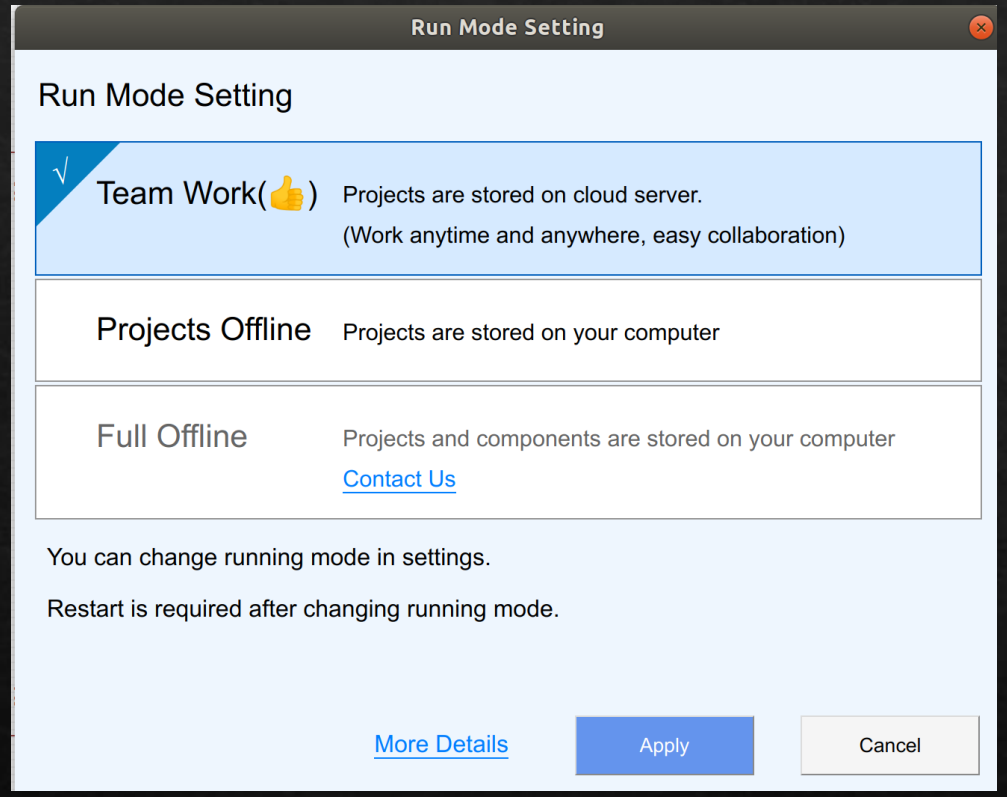


Know EasyEDA

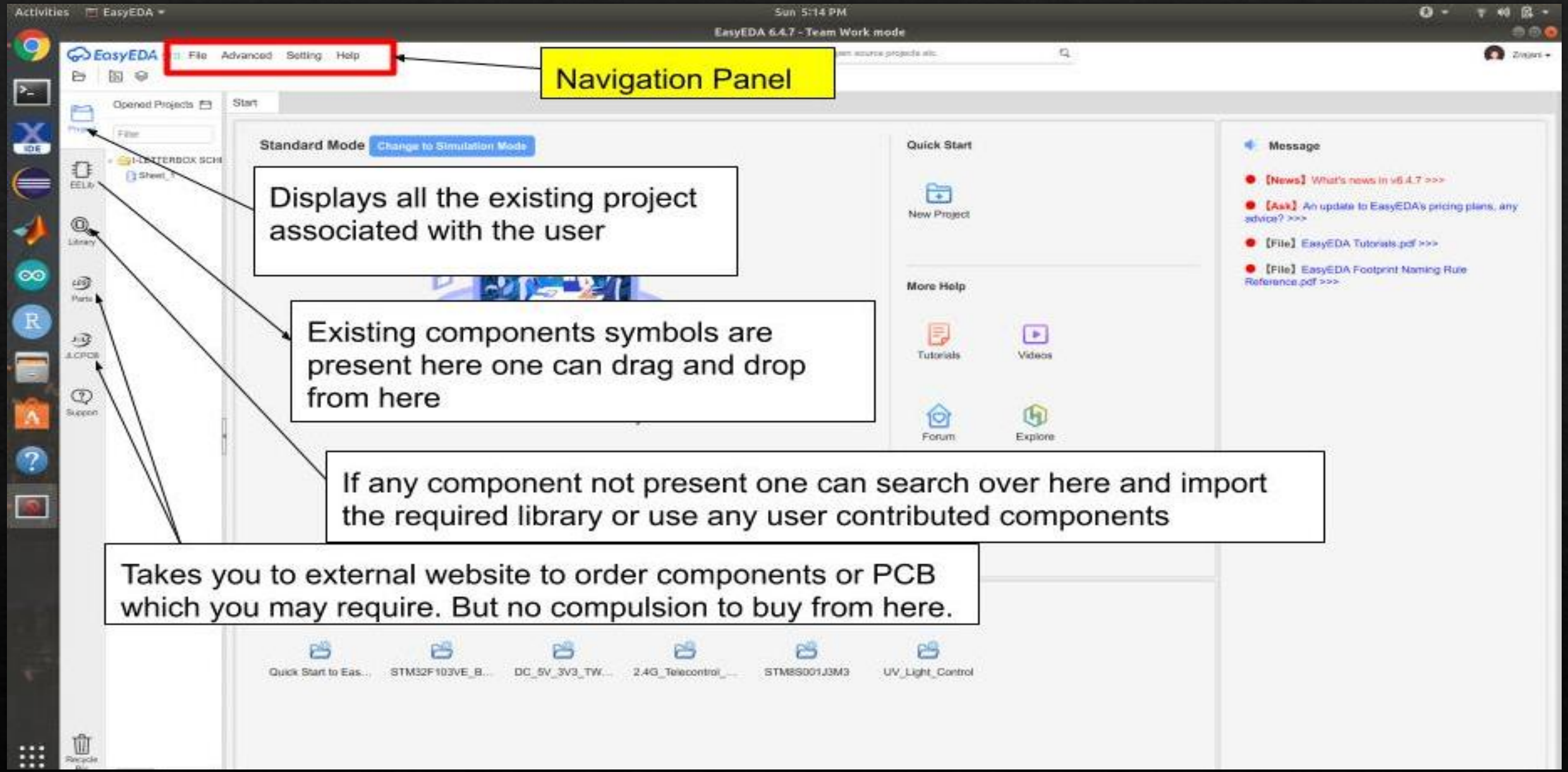
- If one wishes to install the desktop it is very simple. One needs to download the software file from the website and then using the command "sudo bash install.sh", this can also be found in the text file available along with the download
- The software may be priced if one wishes to get immediate support, cloud hosting, recycle retention, etc. Else for all other purpose it is free to use
- Alternatives of this software include KiCad, Altium, Proteus, OSCAD and some paid alternatives include Fritzing, OrCAD, Eagle, etc.



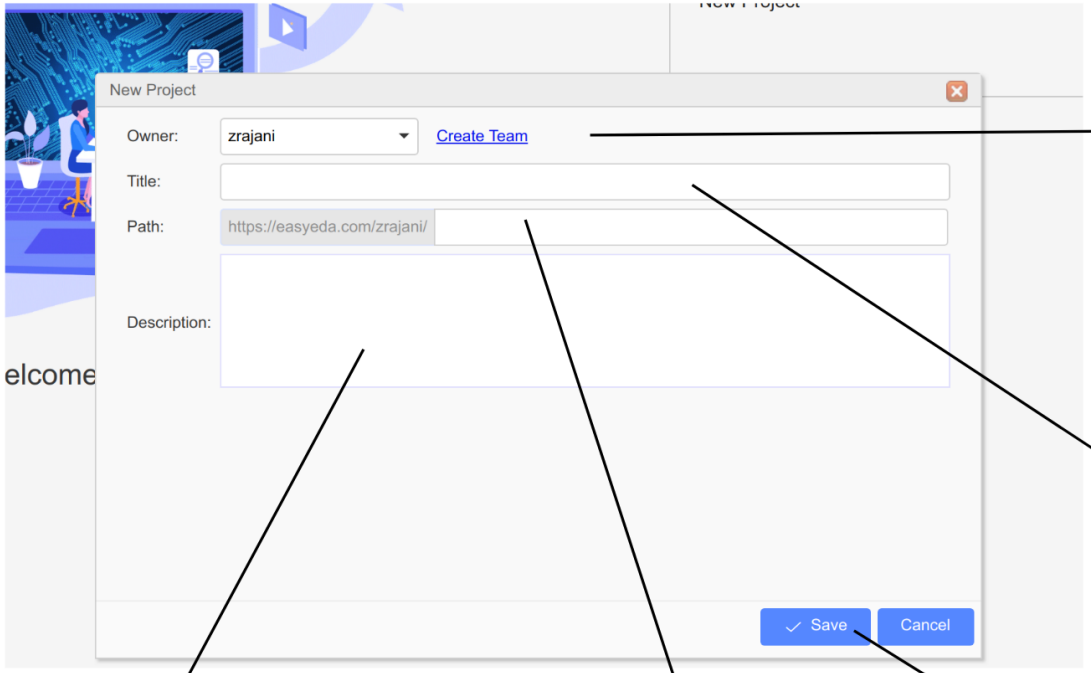
Getting Started with EasyEDA



- After the design one can download the schematic as Image, PDF Files or Altium. Also PCB files can be downloaded along with BOM
- In order to get started with designing your circuit one needs to register so that one can save the design always when an alteration is made. One can register using a form or via social media account using Google.
- There are two modes available for use Standard Mode or Simulation Mode. Since we have no simulation to show for the project thus, we focus on the Standard Mode.
- Also when using the desktop client one can choose the run mode (Team Mode, Project Mode or Offline Mode)



Initial Window



One can set who is the owner of the project. This could also be a team. You can assign it here or create one if not present

Give a suitable title to the project

Add a suitable description to the project

One can use this link to find the project online

Once all done press “Save”

Make a New Project

Create team - EasyEDA

Product Pricing Service Resource Help EasyEDA Designer

Projects
Teams
Modules
Libraries
Recycle Bin

Team

You can view and manage your team and team projects here

My Team
Participated

Team

Upload avatar

* Team Name

Team Name

* Team Path

https://easyeda.com/ Please Enter Team Path

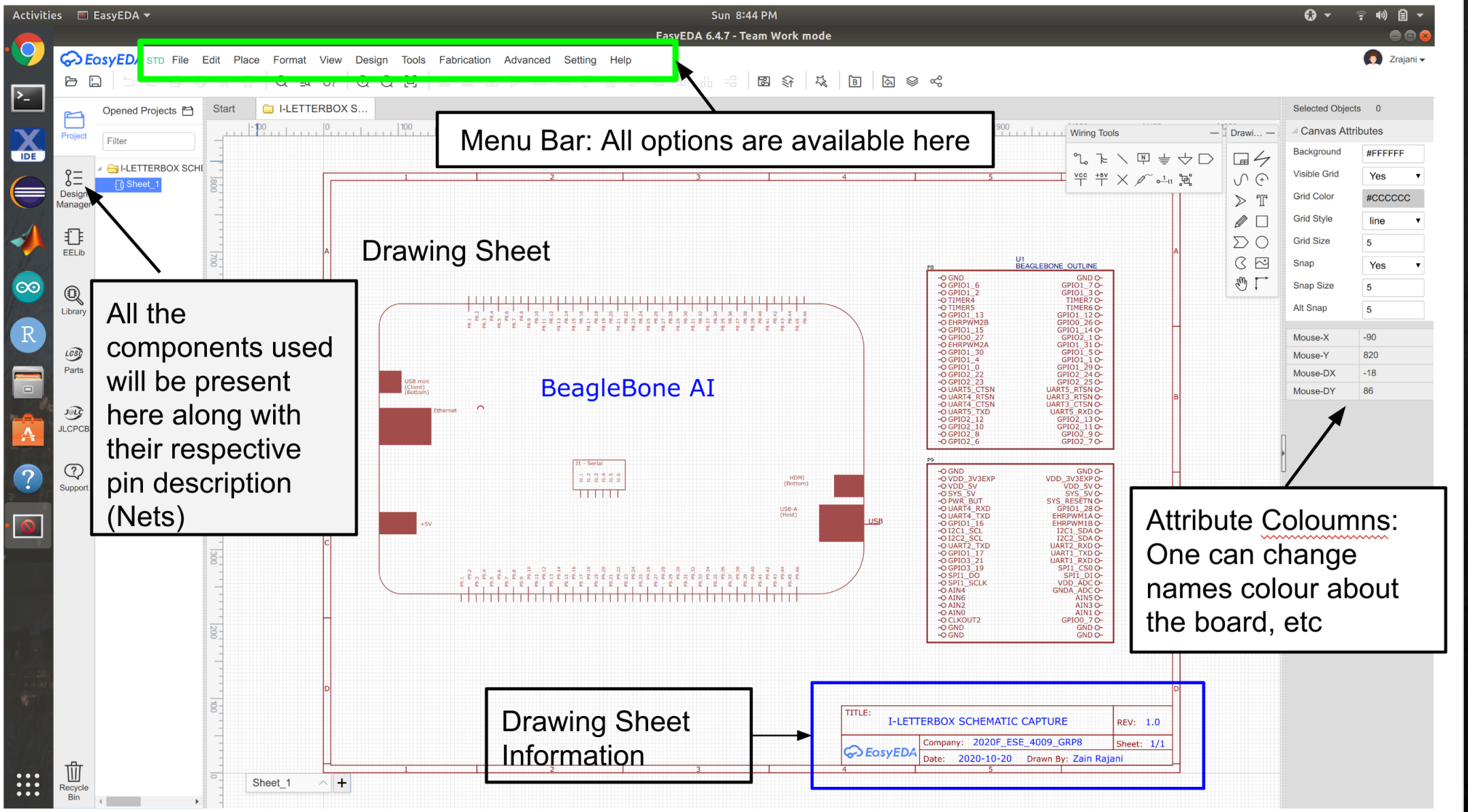
Team Introduction

Please Enter Team Introduction

Add members

Add members here

Creating New Teams



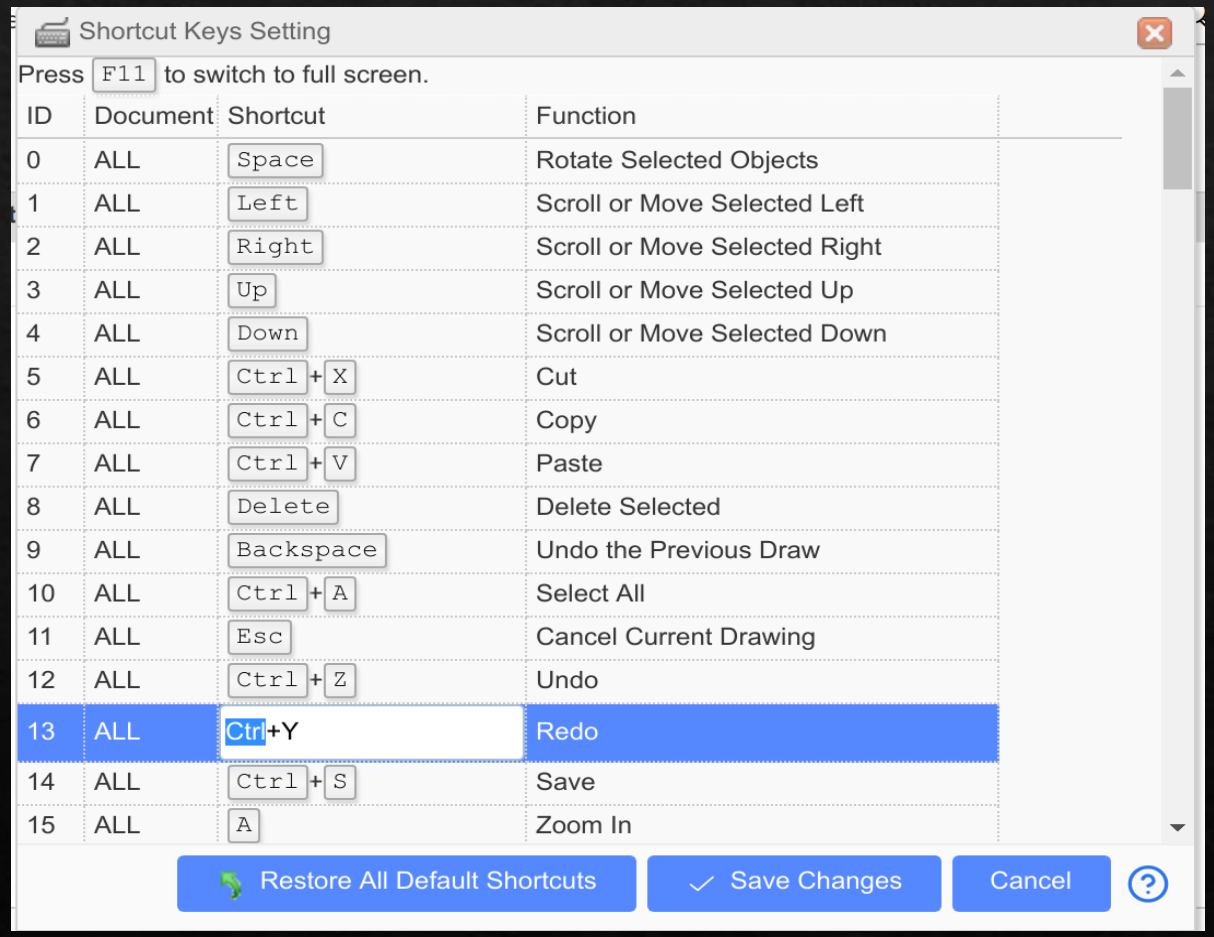
New Project Created (1 of 2)

New Project Created (2 of 2)

- **Format:** Various alignments of the components like rotating it, flipping it, etc.
- **Design:** Convert or Update into PCB
- **Fabrication:** Generates BOM and helps ordering parts and PCB
- **Settings:** Software and Document Settings. Like the Grid Size (measured in pixel) and Snap Size (Alignment of components usually in mm preferred any number between 20 and 100 but any number would do)
- **Drawing toolbar:** Sheet setting, line, image, Bezier, arc, text, freehand draw, arrowhead, rectangle, polygon, ellipse, pie, drag and canvas origin.
- **Wiring toolbar:** wire, bus, bus entry, netlabel, net flag VCC, net flag +5V, net port, net flag ground, voltage probe, pin, group/ungroup symbol.

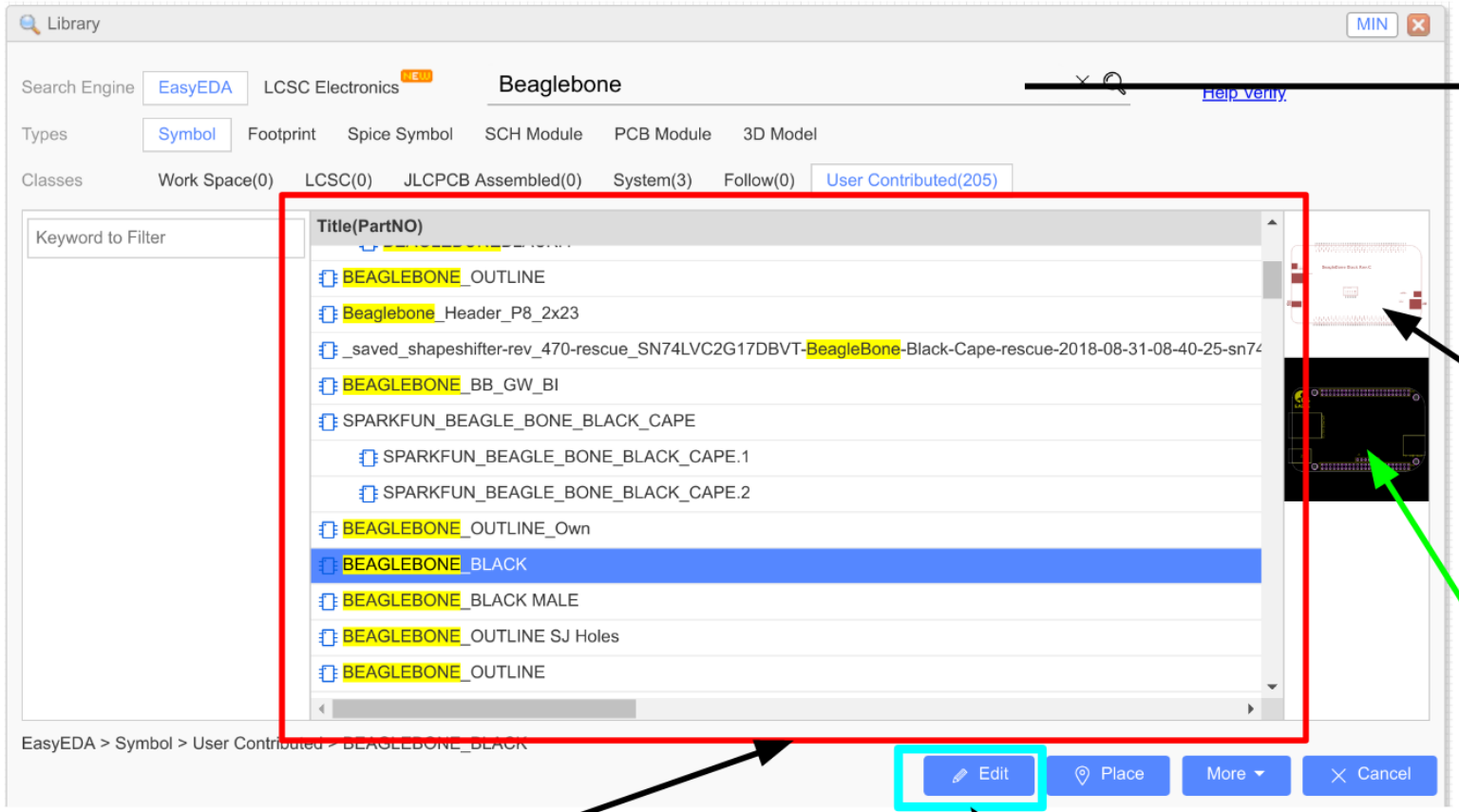
Shortcut Keys

- For any given software to make things simipler to use we have some shortcuts like to program the hardware, rotate some components or in programming to compile and run codes.
- Similarly in EasyEDA there are various sets of shortcut keys already present, but one can even modify these keys as per one's comfort level. To do this one must select the settings tab in the menu bar and click on "Shortcut Key Settings"
- When one wants to change the shortcut for something you must click on the shortcut keys displayed type the one and save it. This can be seen in the image on right



Creating Schematic Capture

- When tried to create the schematic capture our first step was to get the main processing unit i.e. Beaglebone Wireless (BB-WI). Unfortunately, this not directly present in the software.
- Thus since the software is flexible enough, we take the advantage of the library function where we can try to find if someone else from some corner of the world has contributed the component
- We try to find but as a result we couldn't find the BB-WI, but we could find the Beaglebone Black (BBB) and since the device has same pin functionality, we just had to modify it a bit as the BB-WI doesn't have the Ethernet port, and similarly by looking at what is there on the schematic we can change it.



Type the component name you want to search

Schematic Design

PCB View of the component

Search Results choose the one you desire

Use this if you want edit the desired component

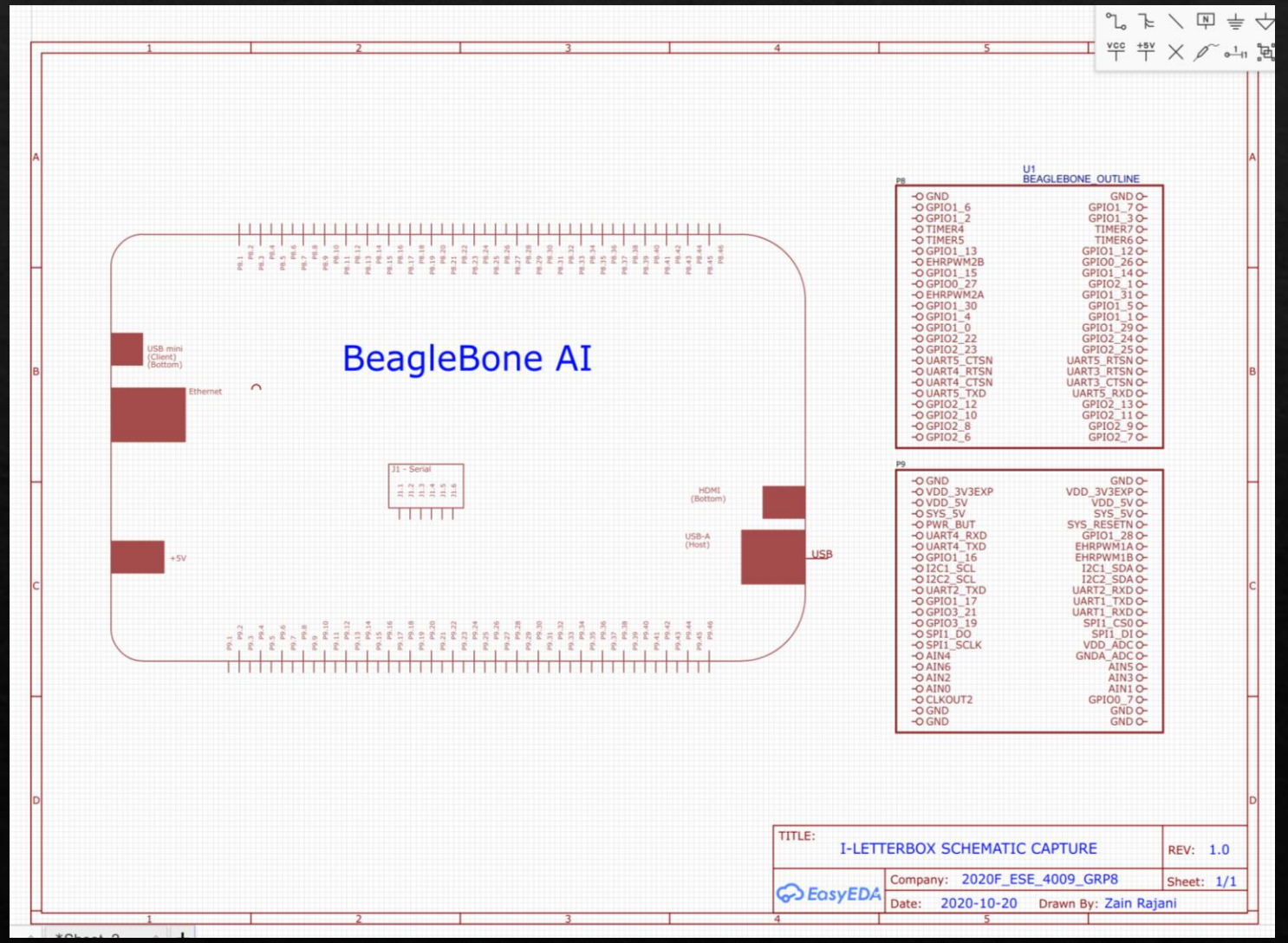
Searching Components

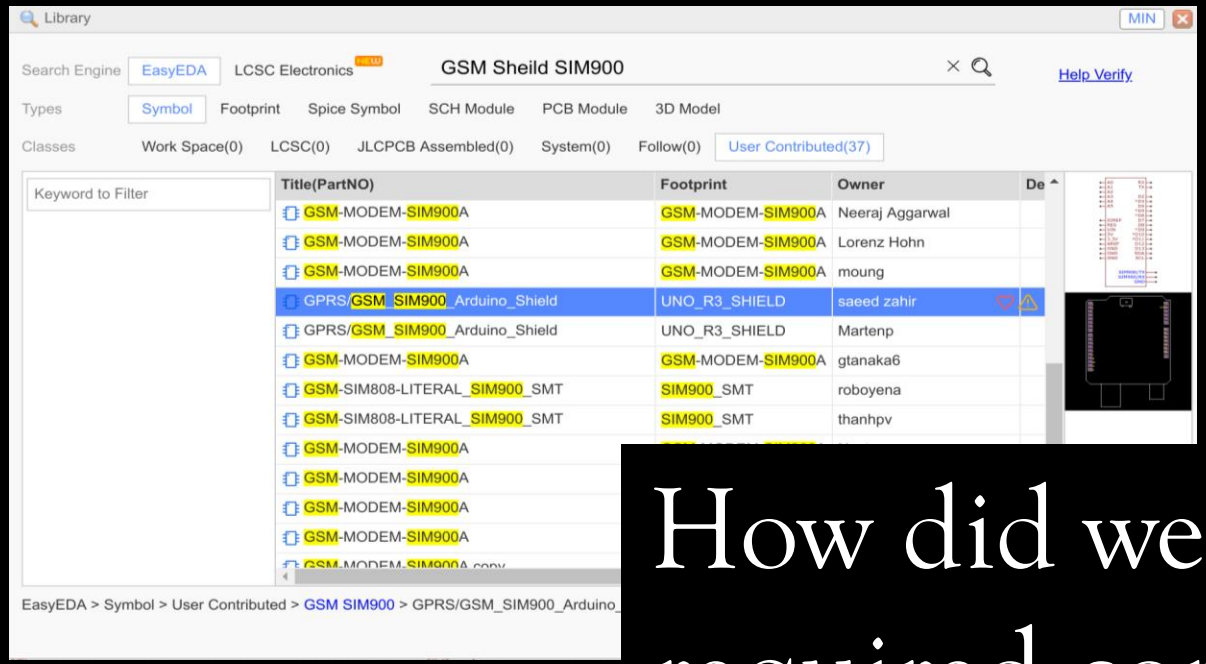
Creating Schematic Capture

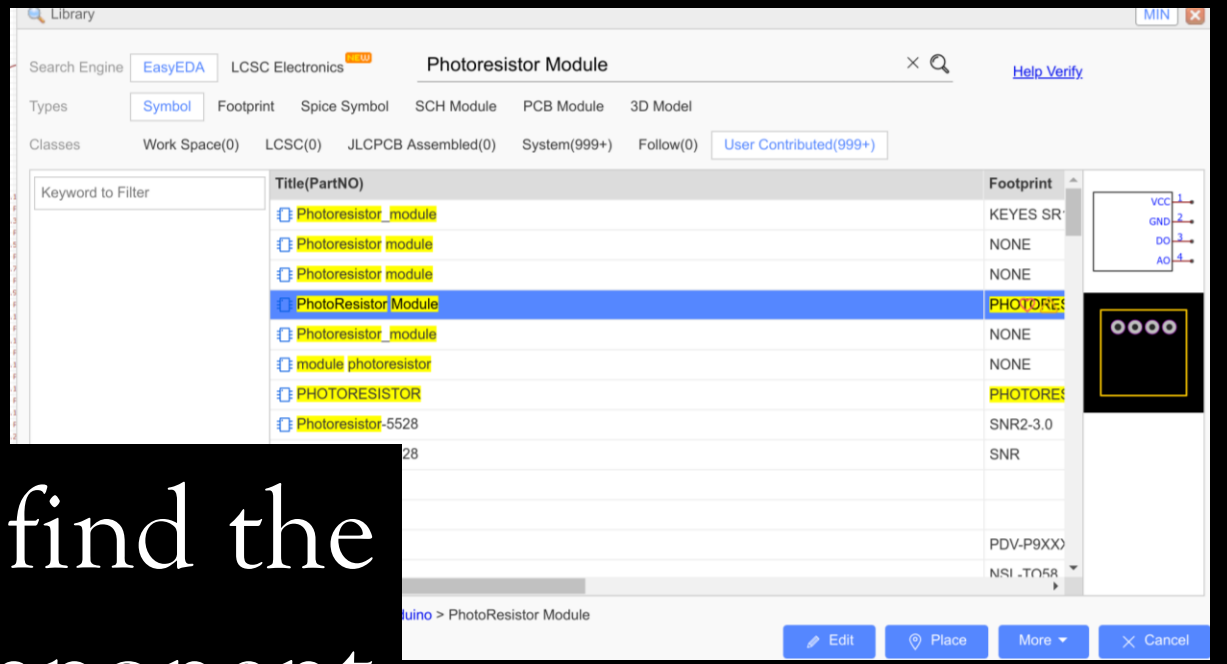
- Once selected the desired component just drag it and place it on the desired sheet with required orientation and position in the project.
- Now we further find the rest of the components that we want for the project which include IR Sensor, GSM Module, LDR Module, Light Emitting Diode (LED) and a few resistors which may be required.
- From the software we see that LED and Resistors are already avaliable in EElib feature of the software. The remaining we find using the same method as we find the Beaglebone Wireless (BB-WI)

Choice of Component

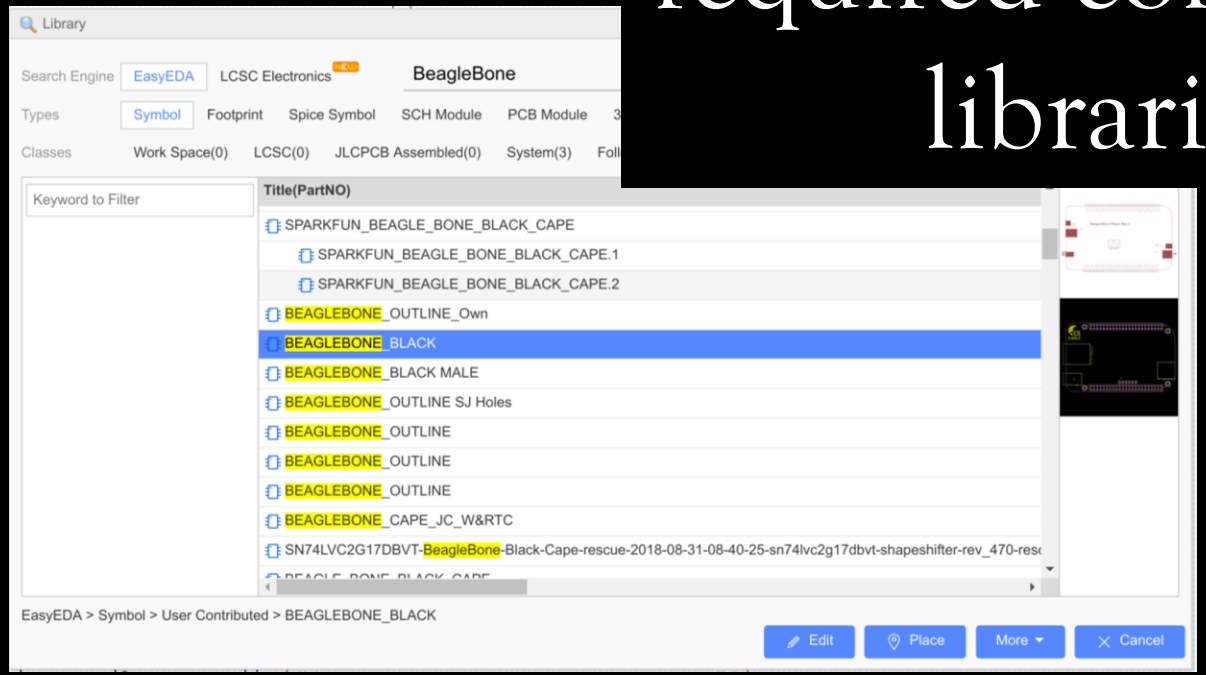
- When we search, we get many options to choose from, but one needs to check though the schematic may be avalaible for that but may be the PCB Schematic may not be avaliable thus, one needs to choose properly if he/she wishes to convert the schematic into PCB directly.
- For example, the in the figure shown the right side Beaglebone doesn't have a PCB Schematic whereas the left side has it. Thus, this may be suited for our project.

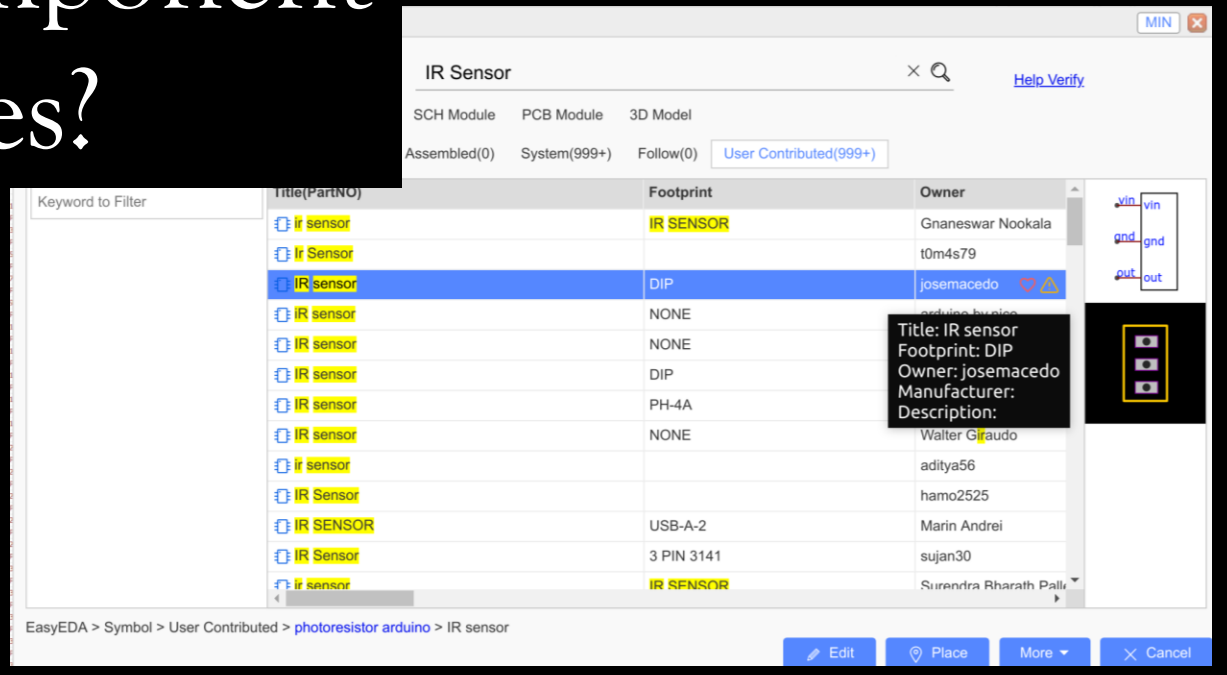




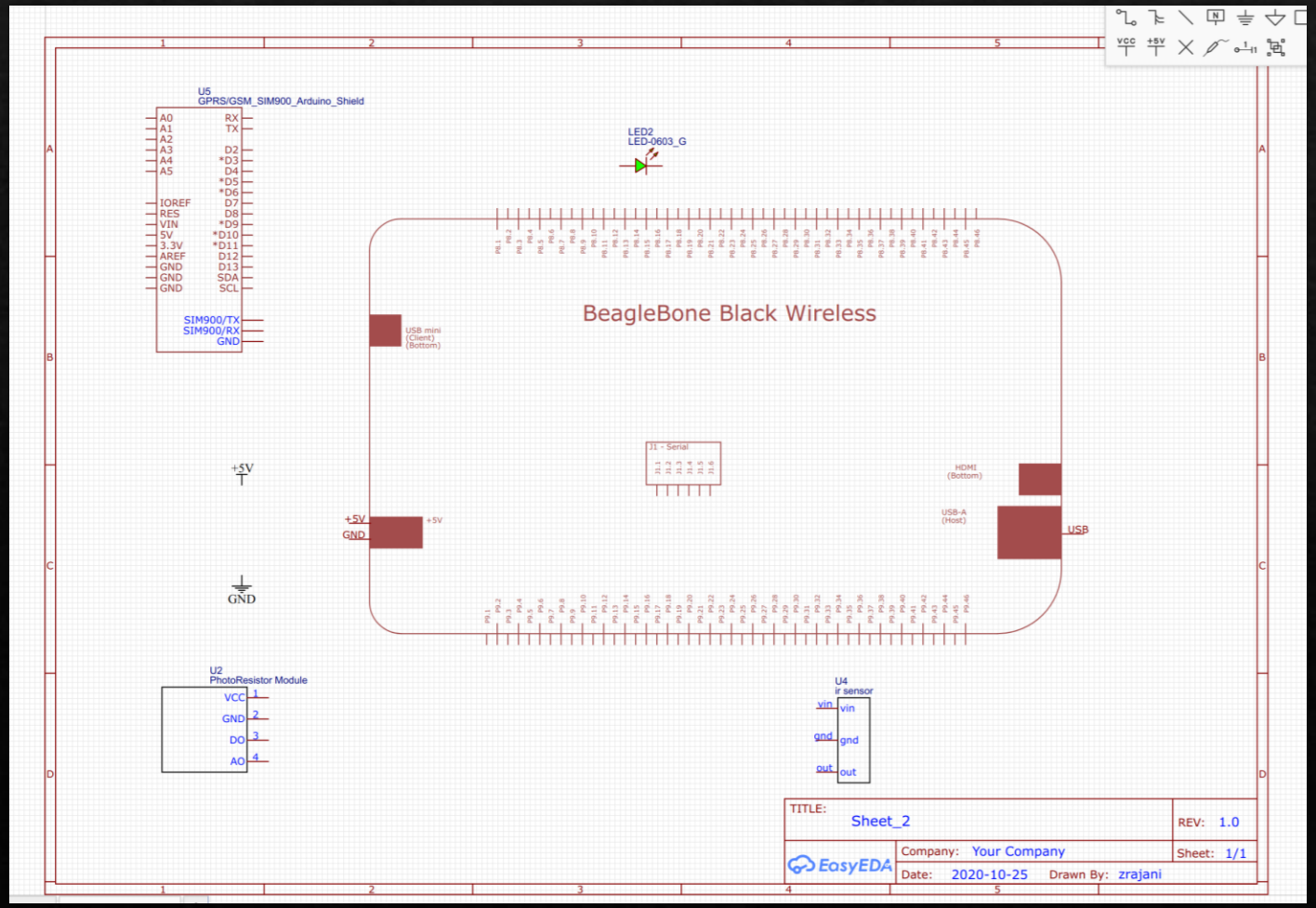


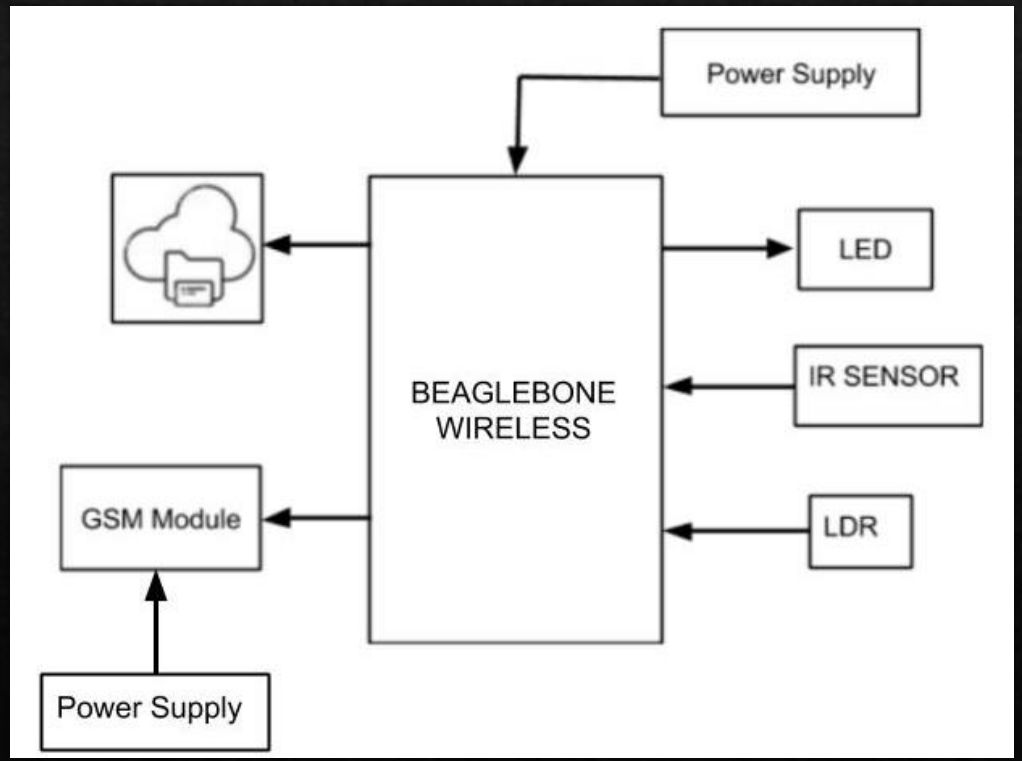
How did we find the required component libraries?





Components Ready for Connection





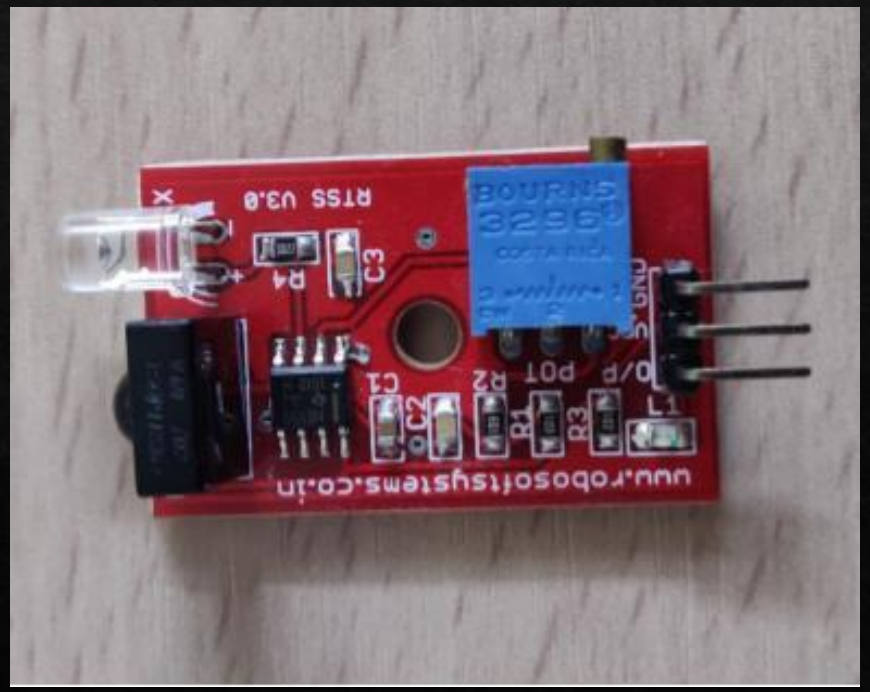
Deciding Connection Methods/ Protocols

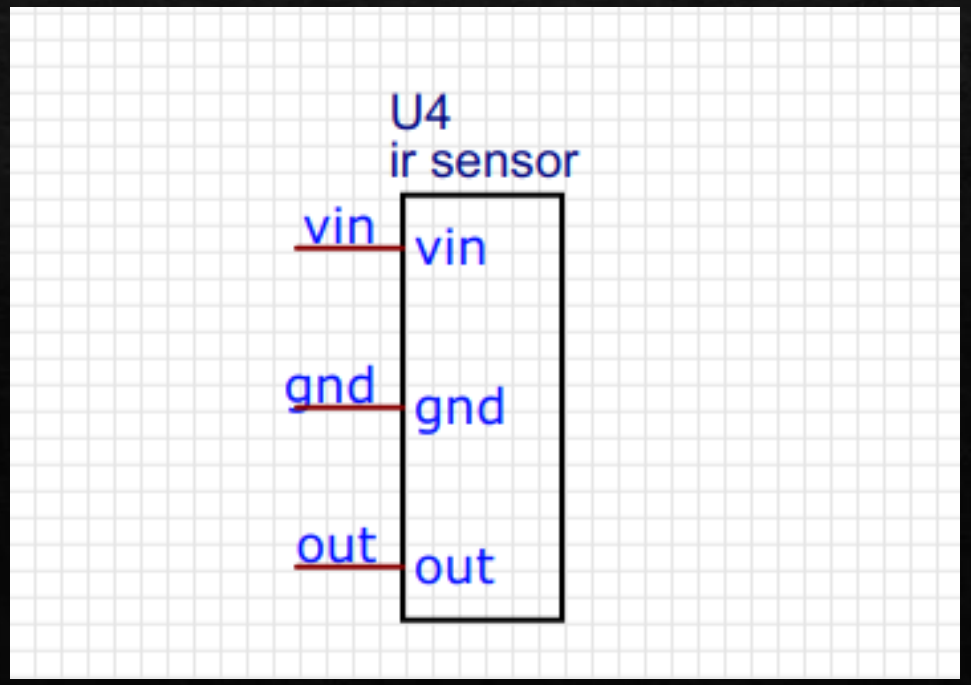
- Before we actual start to connect we must first decide upon the way we must connect all the sensors and other peripherals. The table below depicts the same.

Sensor/ Peripheral Device	Communication Method Intended to Use
IR Sensor	GPIO Mode
LDR Module	GPIO Mode
GSM Module	UART Mode
LED	GPIO Mode

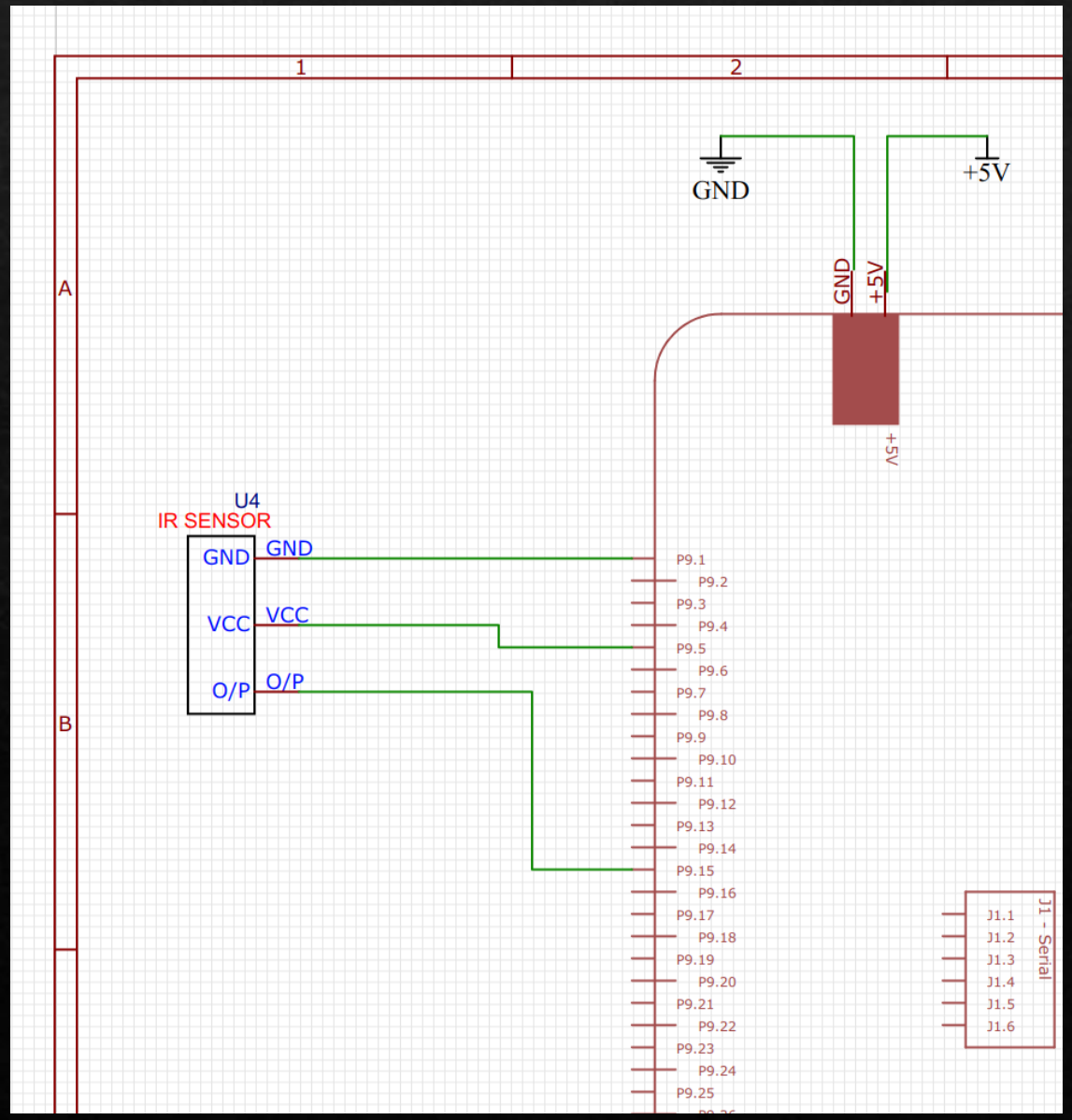
Connection 1: IR Sensor

- Our first step for connection we have the IR Sensor the connection will be from the GPIO pins directly. But before that we need to verify that the component pins and the schematic pins are the same. This can be verified as below.
- This module will be used to detect the arrival of the letters in the letter box.





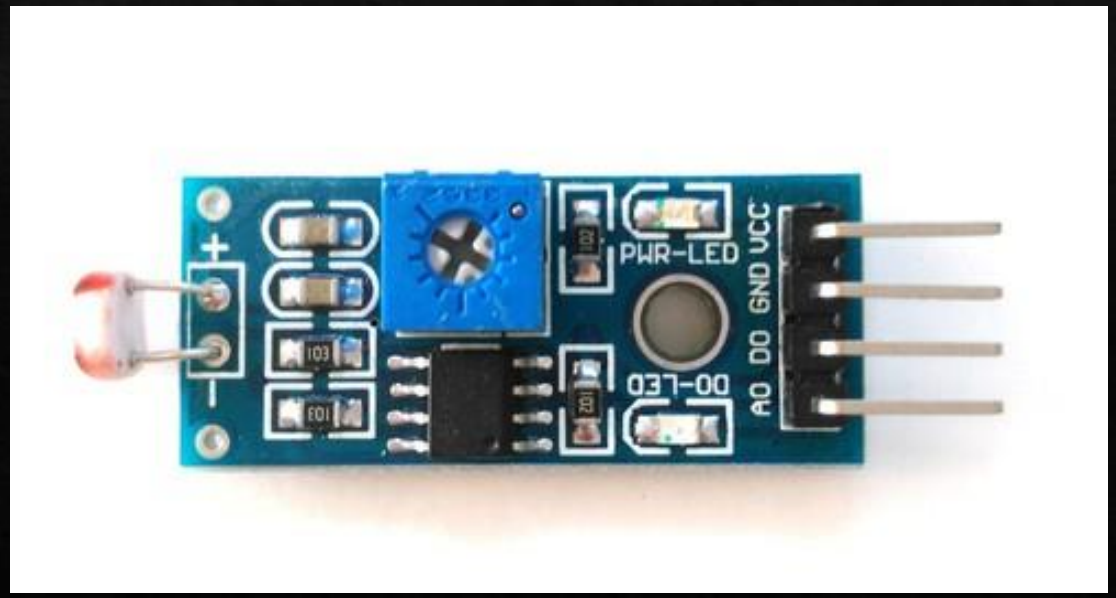
Connection 1: IR Sensor

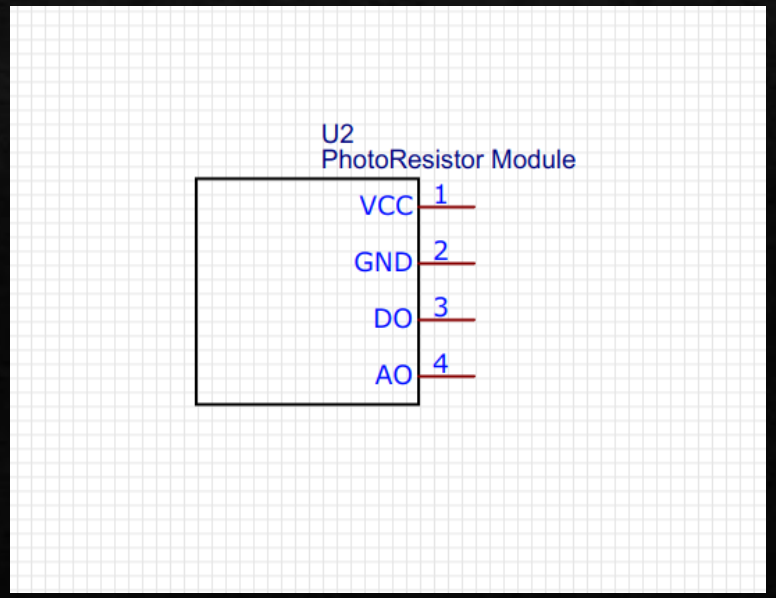


IR Sensor Pin	BB-WI Pin
Vcc (5V)	P9.5
GND	P9.1
O/P	P9.15

Connection 2: LDR Module

- LDR (Light Dependent Resistor) which varies the resistor depending upon the intensity of light falling on the photosensitive material.
- This module helps us to reset the count of the letters. The voltage required to operate this sensor is 5V and Output is of 3.3 V. Since the sensor has both digital and analog voltage, we are using the Digital pin in our project. But first let us verify the component and the schematic pins.

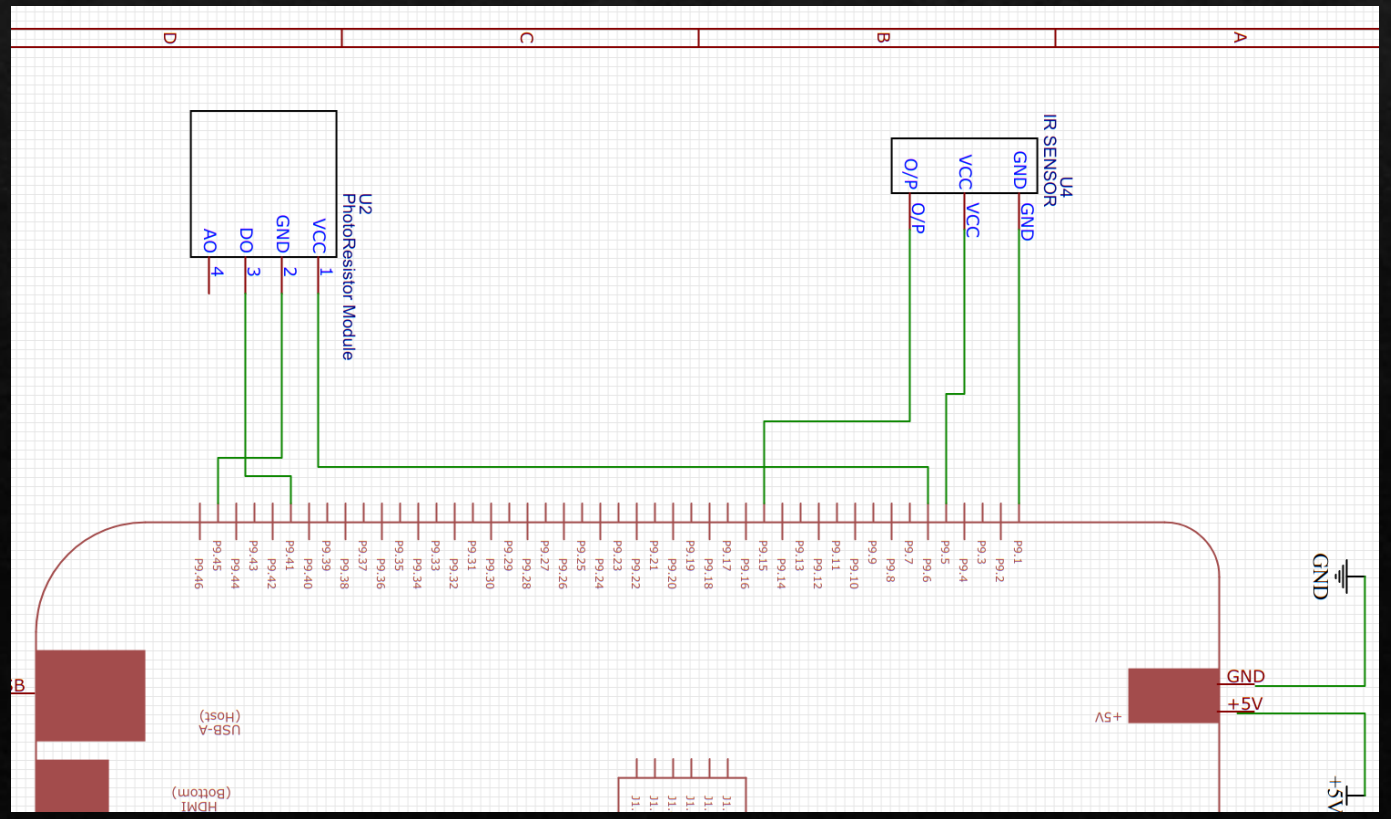




Connection 2: LDR Sensor

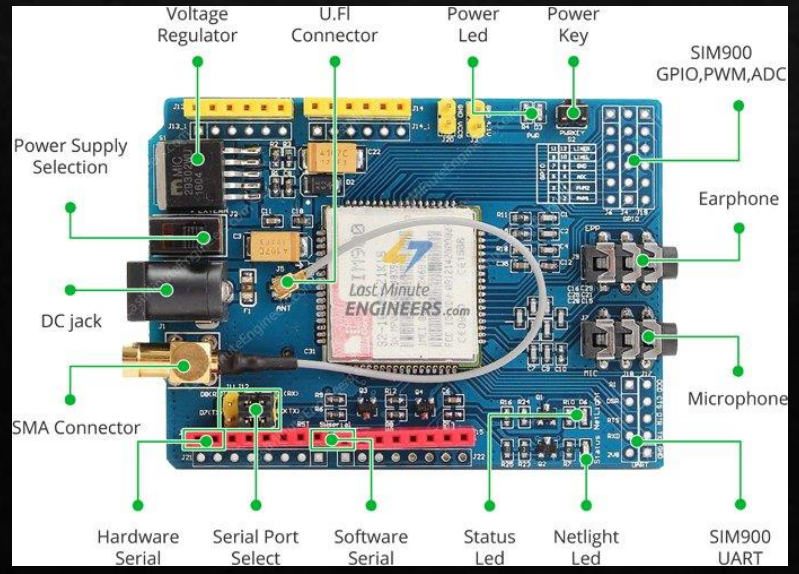
- As decided, we will use the Digital pin and the GPIO connection for the same. The pin connection and the schematic capture of the same is shown.

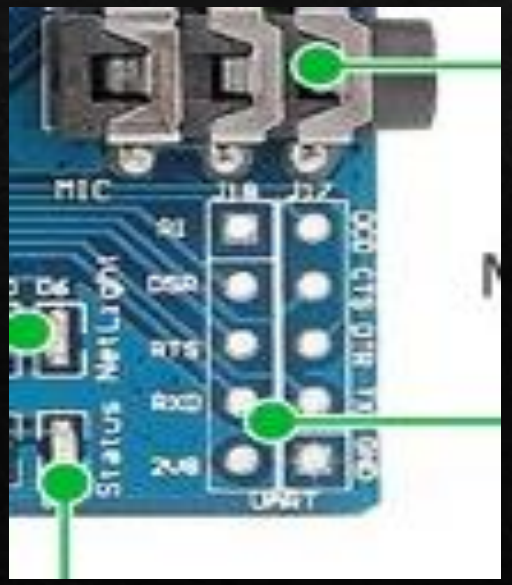
LDR Module Pin	BB-WI Pin
VCC	P9.6
GND	P9.45
A0	N/A
D0	P9.41

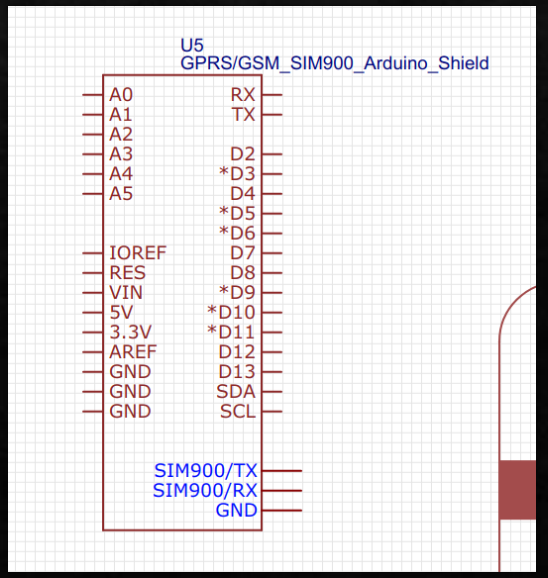


Connection 3: GSM Module

- GSM (Global System for Mobile Communication) for this device we use the UART side of the module. Thus we must use the UART communication protocol for the same.
- This module helps us to send messages to the user when any letter arrives in the letter box. Before getting the schematic capture for this done, we will first verify if the physical module and the schematic we have choosen have the same configuration





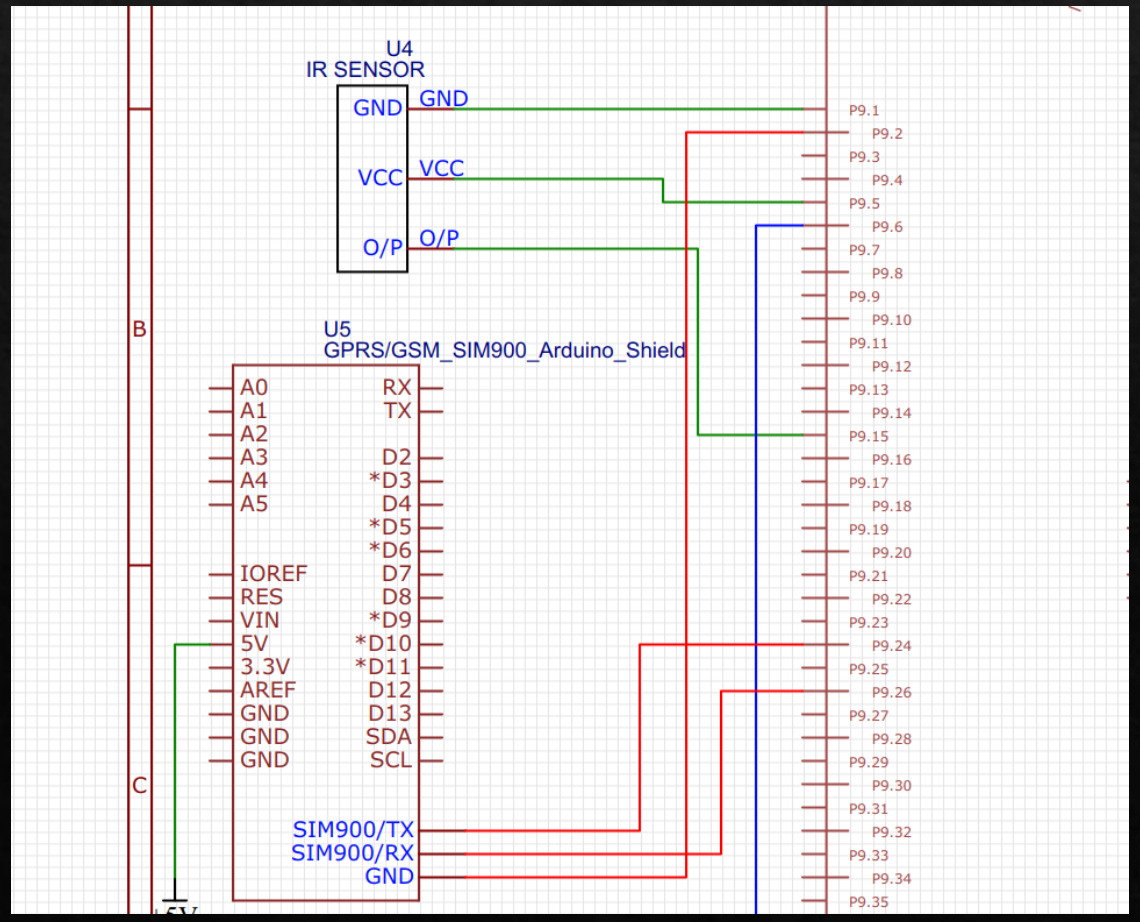


Connection 3: GSM Module

- Since the module we selected for the schematic has the pins that we require thus we can now decide the connection to BB-WI, and this is shown in the table below. Also this module works on a 5V Supply (Barrel Jack).

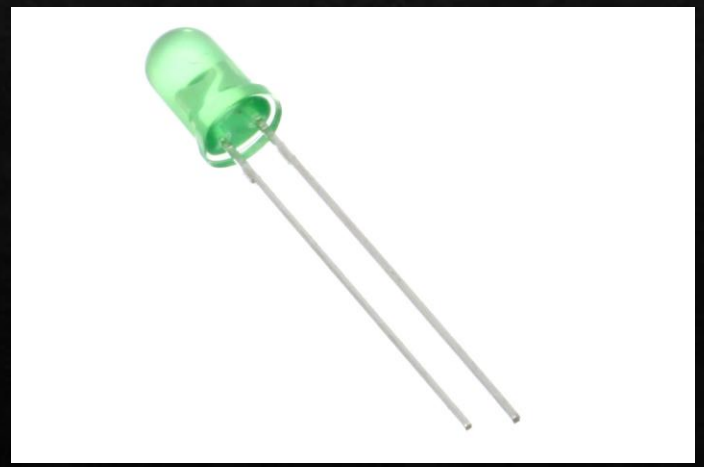
GSM Arduino Sheild Pins (UART Side)	BB-WI Pins
TX	P9.24
RX	P9.26
GND	P9.2
5V	Adapter

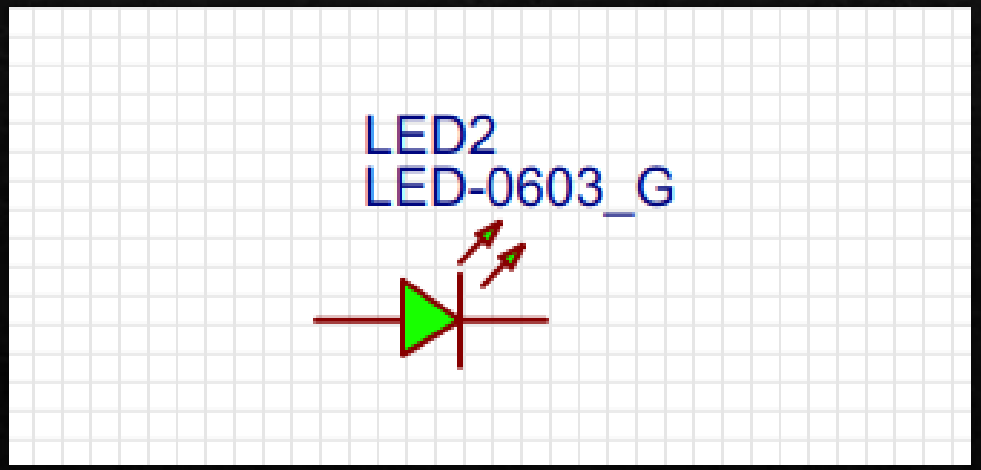
**The red lines indicates the connection of GSM Module to BB-WI*



Connection 4: LED Connection

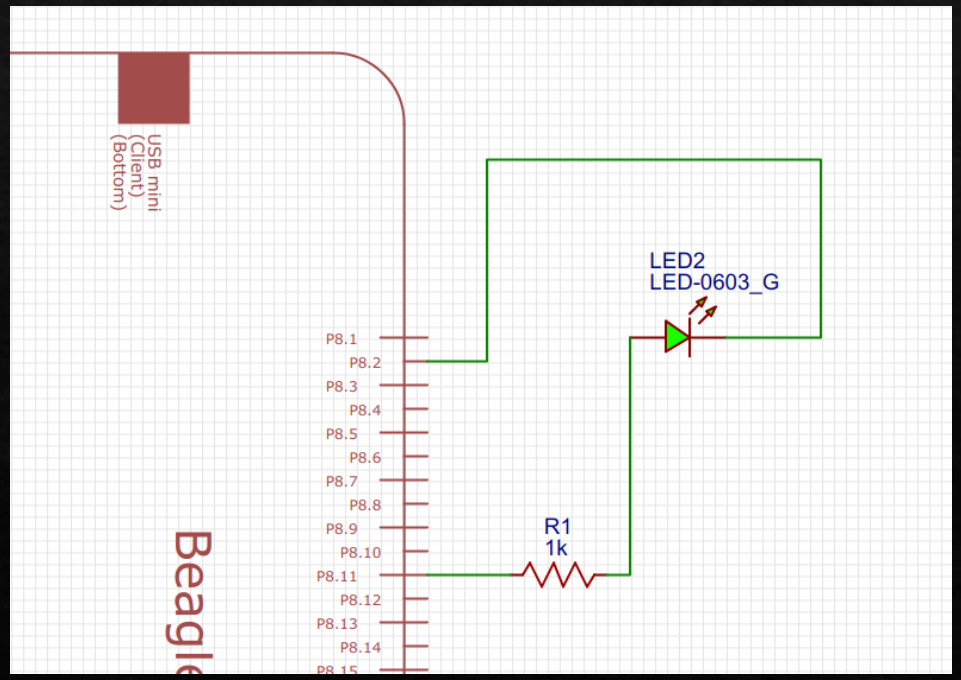
- This simple LED (Light Emitting Diode) can be connected to any GPIO pin with a resistor if required to control the brightness. The LED colcour may vary we have shown the GREEN LED, but we may use any colour LED that we have ready we us.
- The colour of the LED can simply be choosen even in the software by opening the drop-down list and select the colour as required (RED/GREEN/BLUE). We verify the schematic with actual LED as follows.





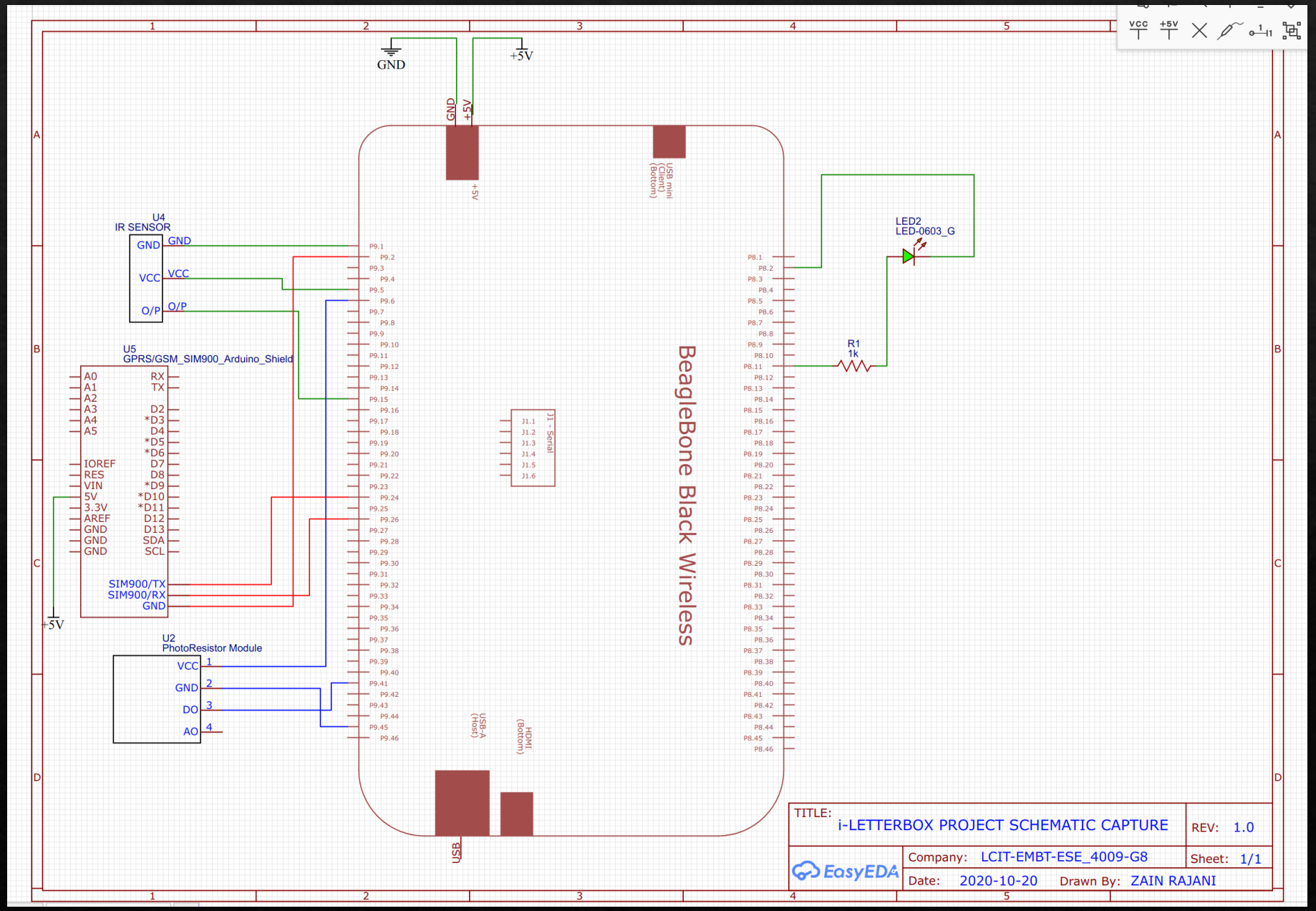
Connection 4: LED Connection

- The LED present is to indicate if there is any letter in the box if the no letter is present the LED status would be OFF. The pin connection along with the schematic capture is shown

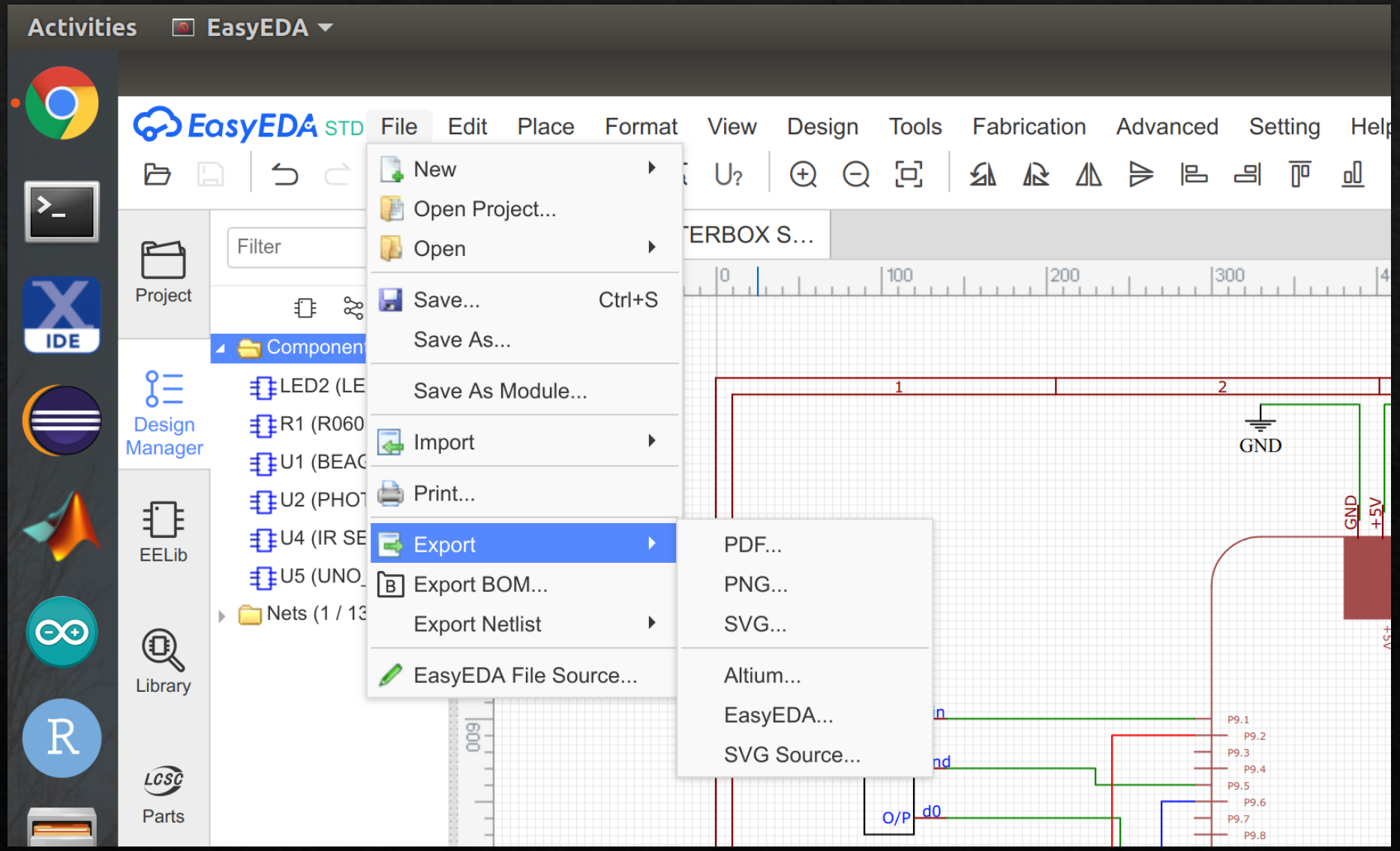


LED Leg	BB-WI Pins
Postive LEG	P8.11
Negative LEG (GND)	P8.2

Final Schematic Capture of Project

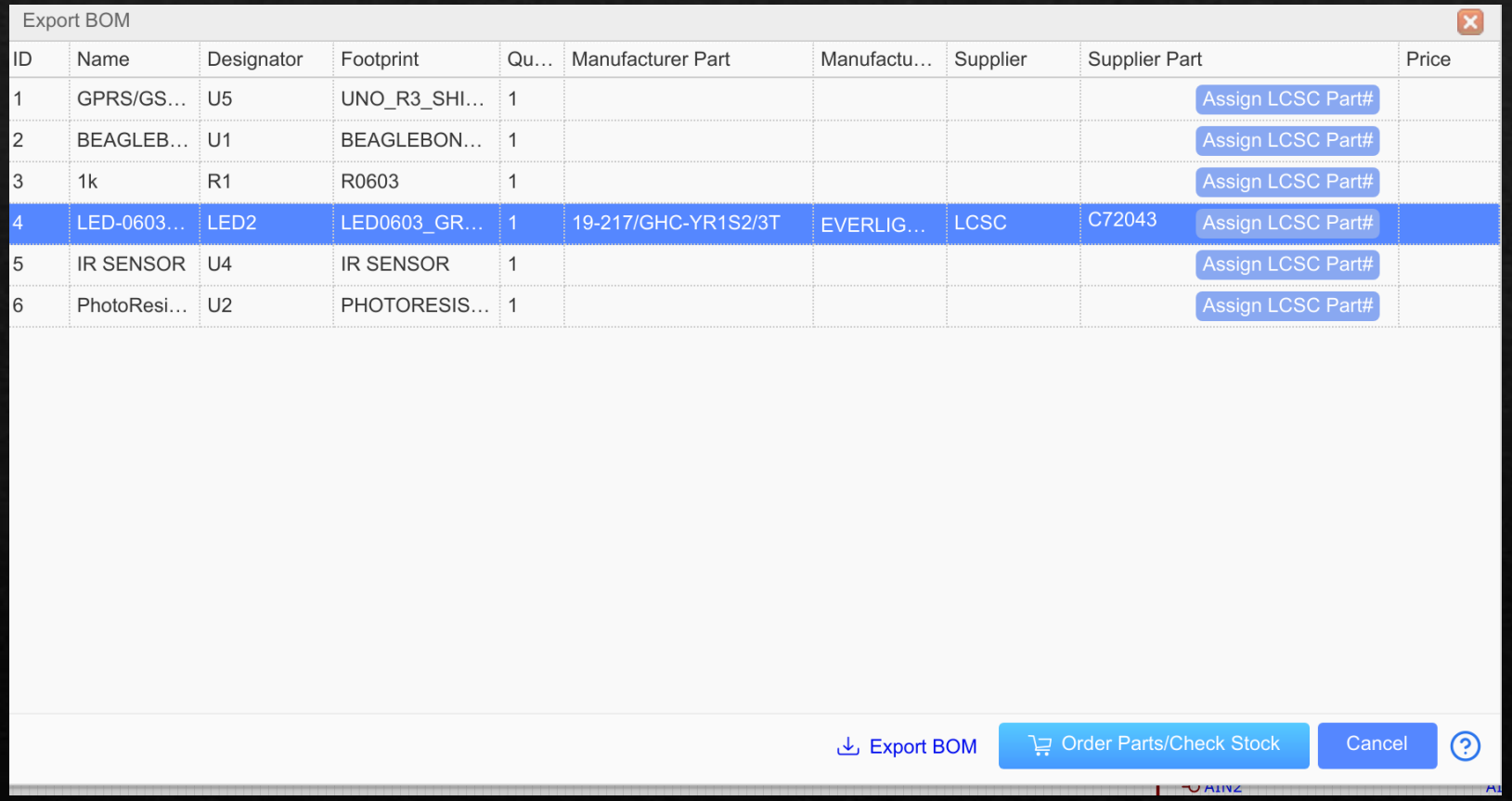


Save the Schematic Capture



Generate the BOM

- To Generate the BOM one can go to Fabrication Option (Avaliable in Menu Bar) and select BOM this will generate it for the given project for our project it is shown as below.



References

Demo: PIR Motion Sensor. (n.d.). Retrieved October 26, 2020, from http://partybot.com.au/Support/BoneScript/PIRMotionSensor/

Dies, M. (2019, November 10). LDR or PhotoResistor and the BeagleBone Black Wireless. Retrieved October 22, 2020, from https://beagleboard.org/p/functt/ldr-or-photoresistor-and-the-beaglebone-black-wireless-8ba096

An Easier and Powerful Online PCB Design Tool. (n.d.). Retrieved October 26, 2020, from https://docs.easyeda.com/en/Introduction/Introduction-to-EasyEDA/index.html

How to Choose The Best PCB Design Software. (2019, November 19). Retrieved October 24, 2020, from https://circuitdigest.com/tutorial/best-pcb-design-software

Iosub, D., & Itbrainpower.net team. (2017, April 12). Who's barking? BBB meets h-nanoGSM. [BeagleBone Black gsm how to]. Retrieved October 20, 2020, from https://itbrainpower.net/a-gsm/BBB-gsm-how-to

Kamat, H. (2017, June 02). How do I connect an IR Sensor with Beaglebone Black? Retrieved October 26, 2020, from https://www.quora.com/How-do-I-connect-an-IR-Sensor-with-Beaglebone-Black

Mathworks. (n.d.). BeagleBone Black Pin Map. Retrieved October 26, 2020, from https://in.mathworks.com/help/supportpkg/beagleboneio/ug/beaglebone-black-pin-map.html

Monk, S. (2013, June 17). Blinking an LED with BeagleBone Black. Retrieved October 20, 2020, from https://learn.adafruit.com/blinking-an-led-with-beaglebone-black/writing-a-program

Shetty, R. (Producer). (2013, November 27). *Beagle bone black GSM modem interface part 2* [Video file]. Retrieved October 20, 2020, from https://www.youtube.com/watch?v=814CbzFG6JE