Creating schematic in EasyEDA
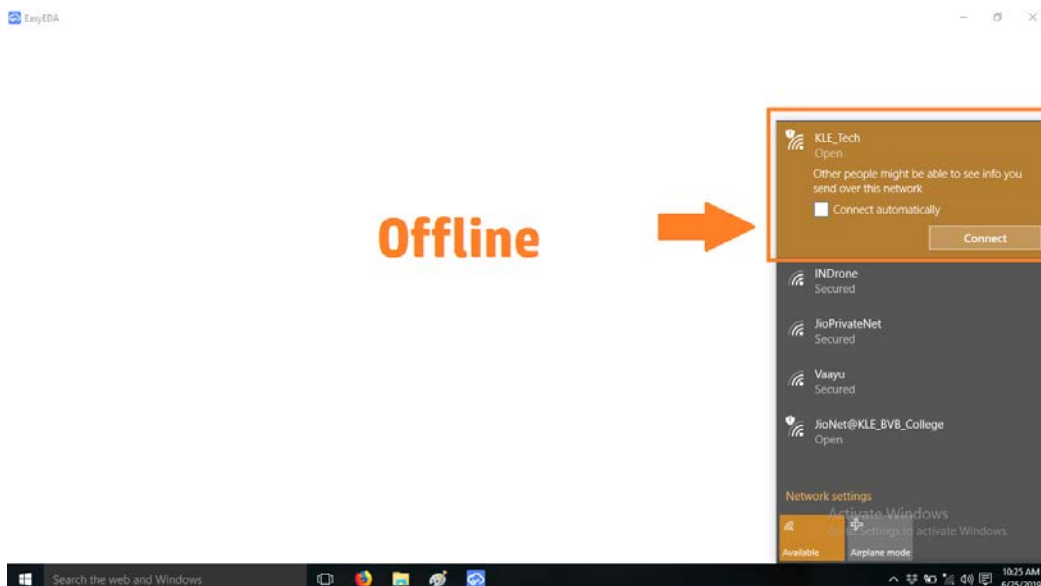
Created on 25 – July – 2019
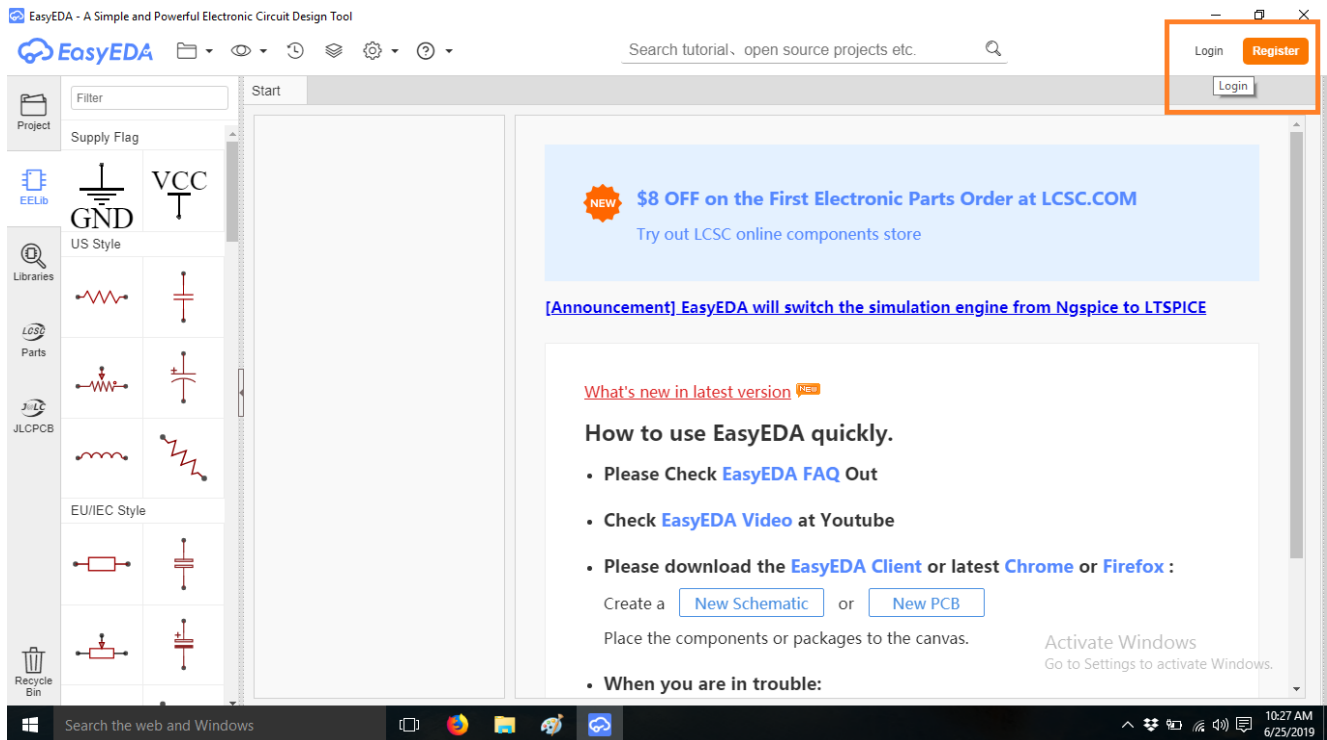
EasyEDA

Steps to start from:

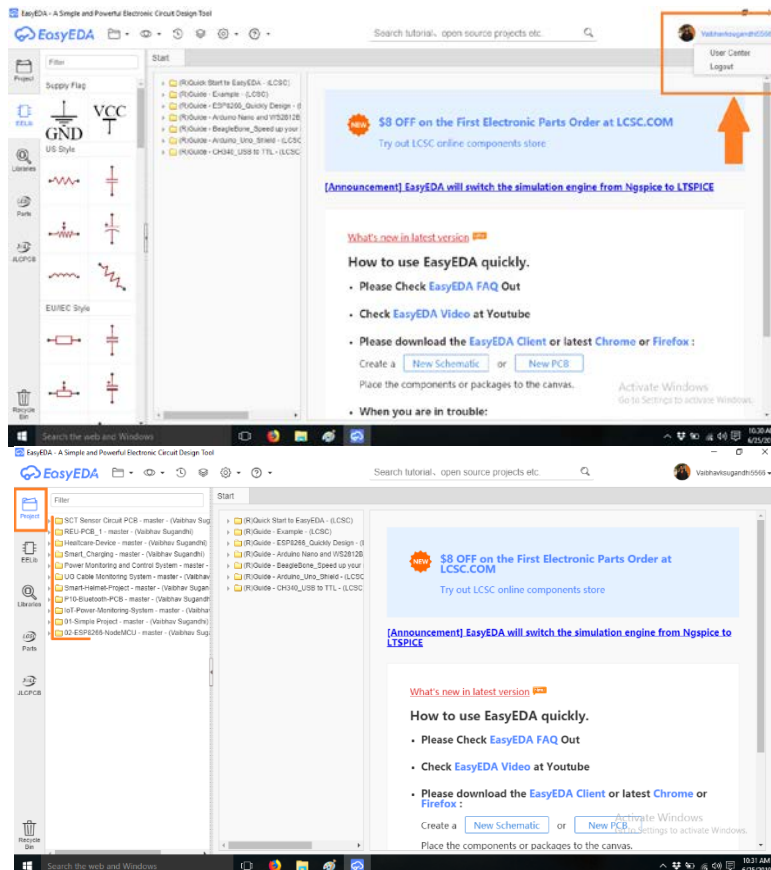1. Open the software from desktop icon and wait a while it opens up.

2. Make sure that your computer is connected to active internet service, else software will not open up.
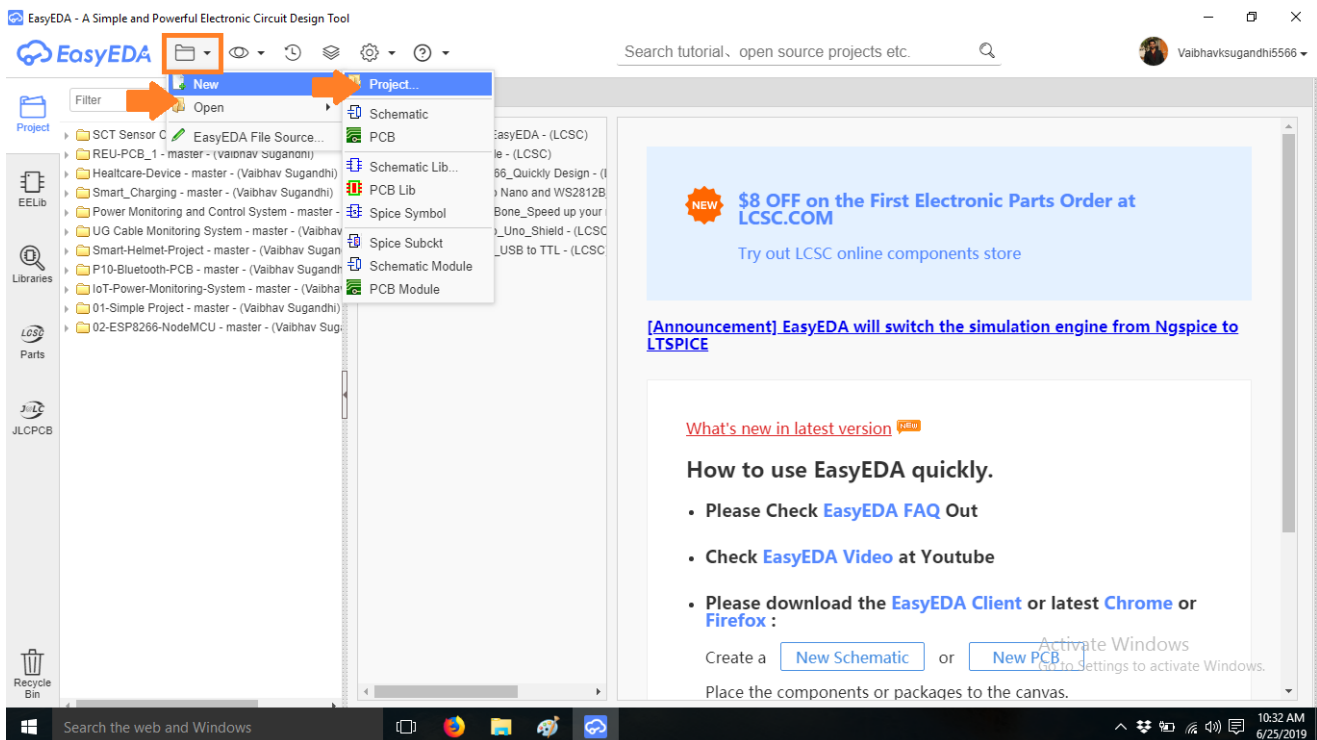
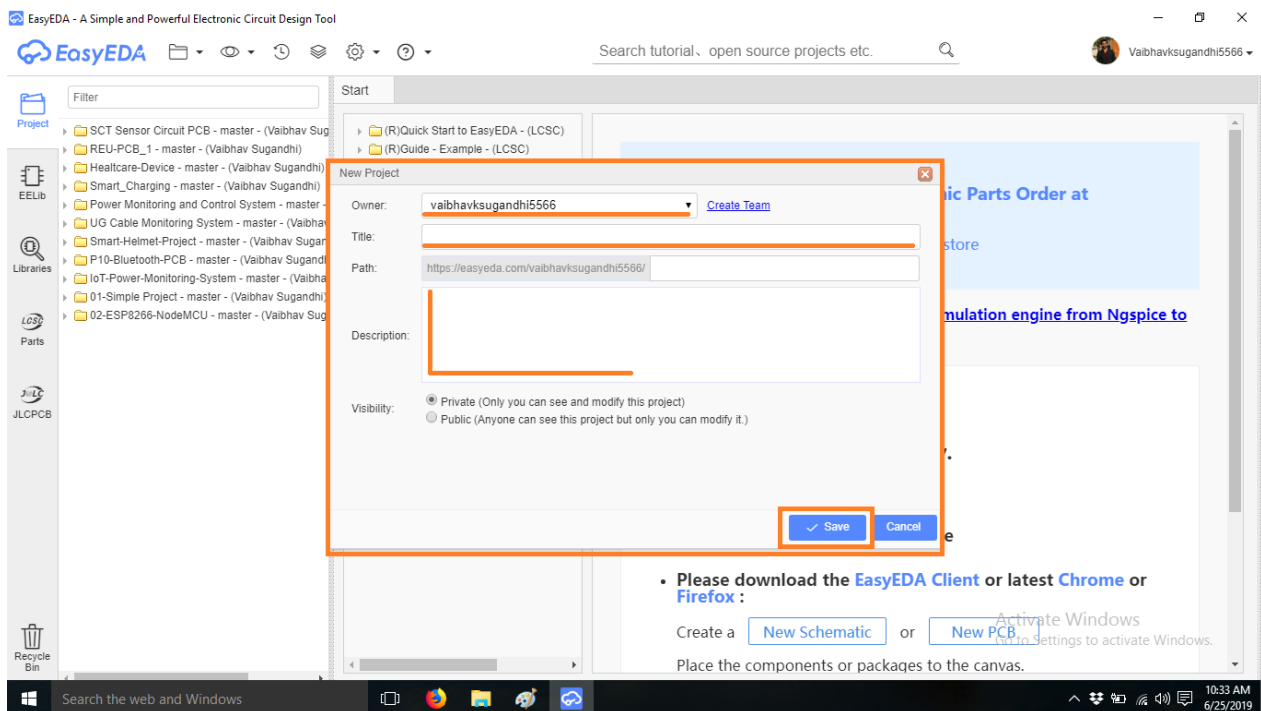3. Once it opens, now click on Login to start accessing your account.

4. Once it is logged in, it shows your user ID and projects list in the screen as shown below.
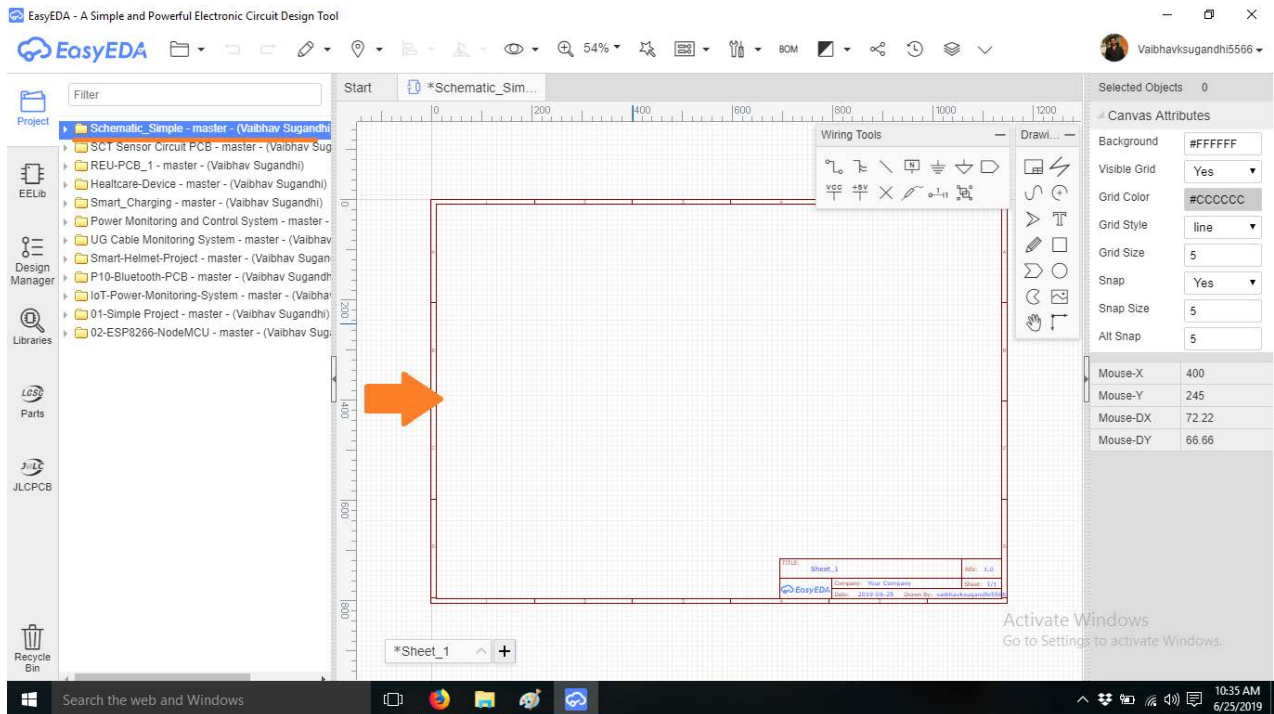
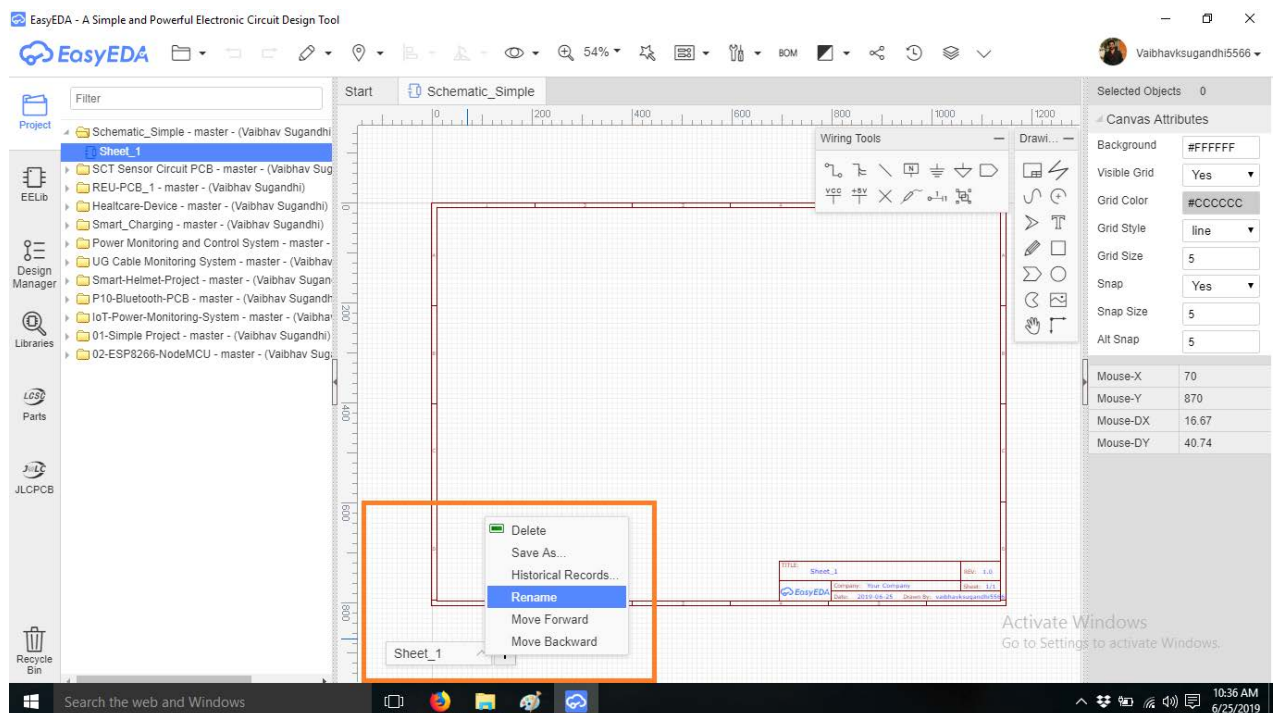5. Now click on Folder icon shown in image below → New → Project.

6. New window will pop's up, in which you have to input the details of your new project and click on save.

7. Once new schematic windows pops up, you are ready to go. The name of your project is shown in list of projects at left part window.
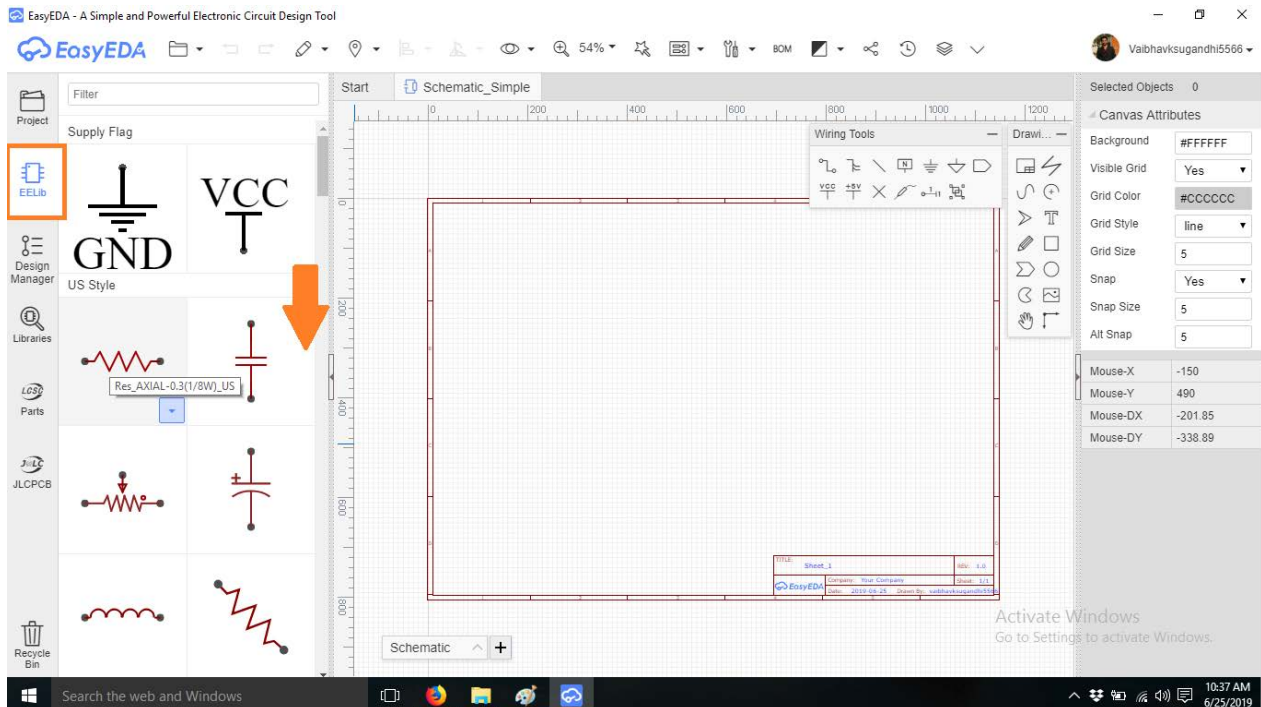

8. Click on the upside arrow mark shown in *Sheet_1* to rename the sheet as Schematic.
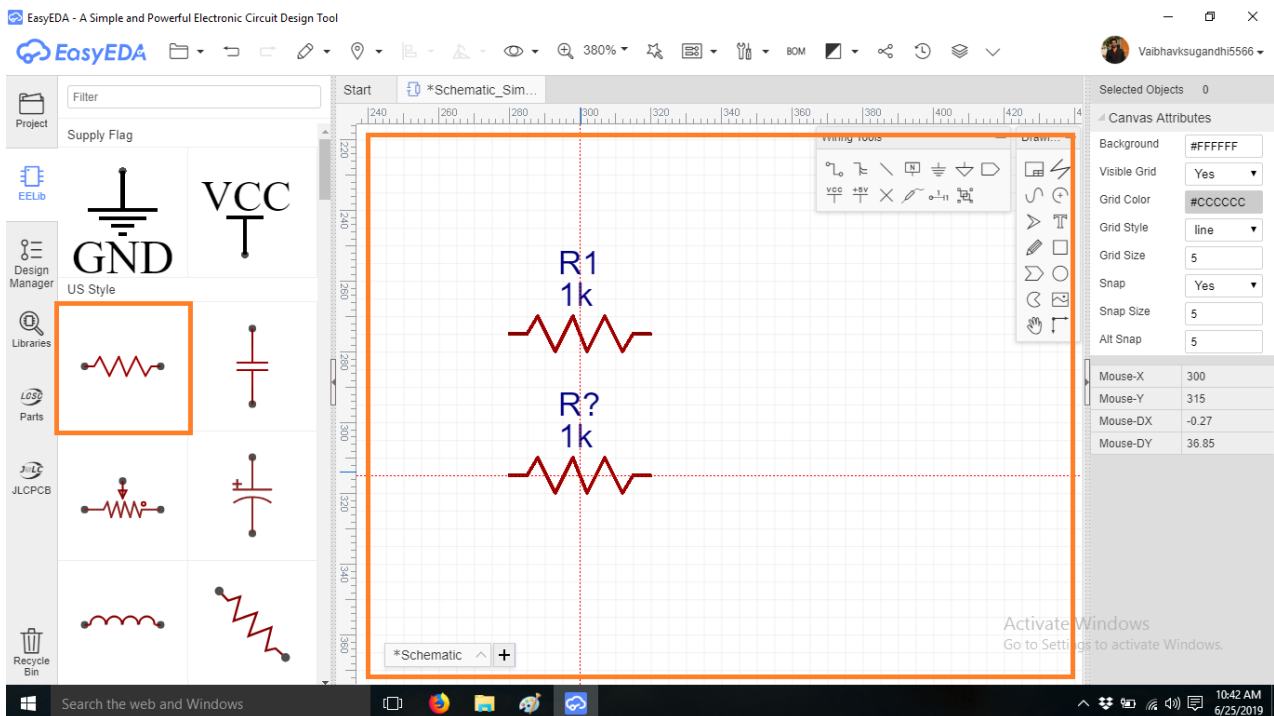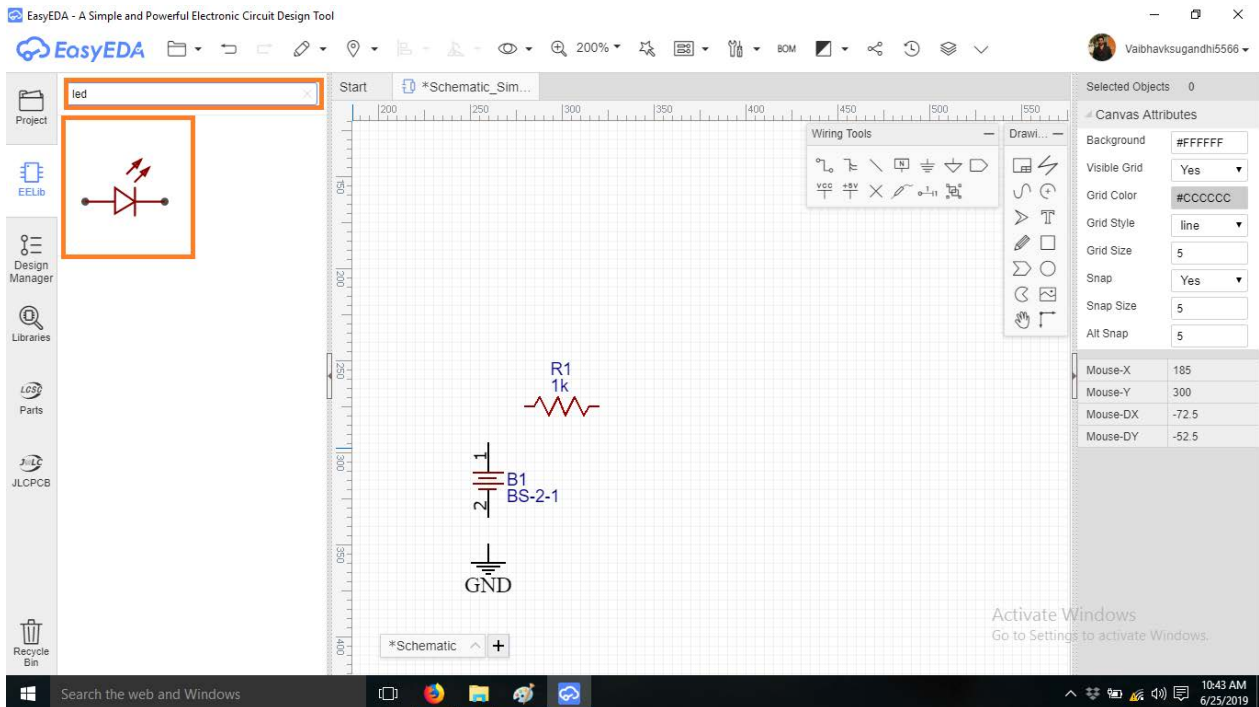
9. Now click on *EELib* at the left side and you can see the instruments, signals and components for creating schematic.
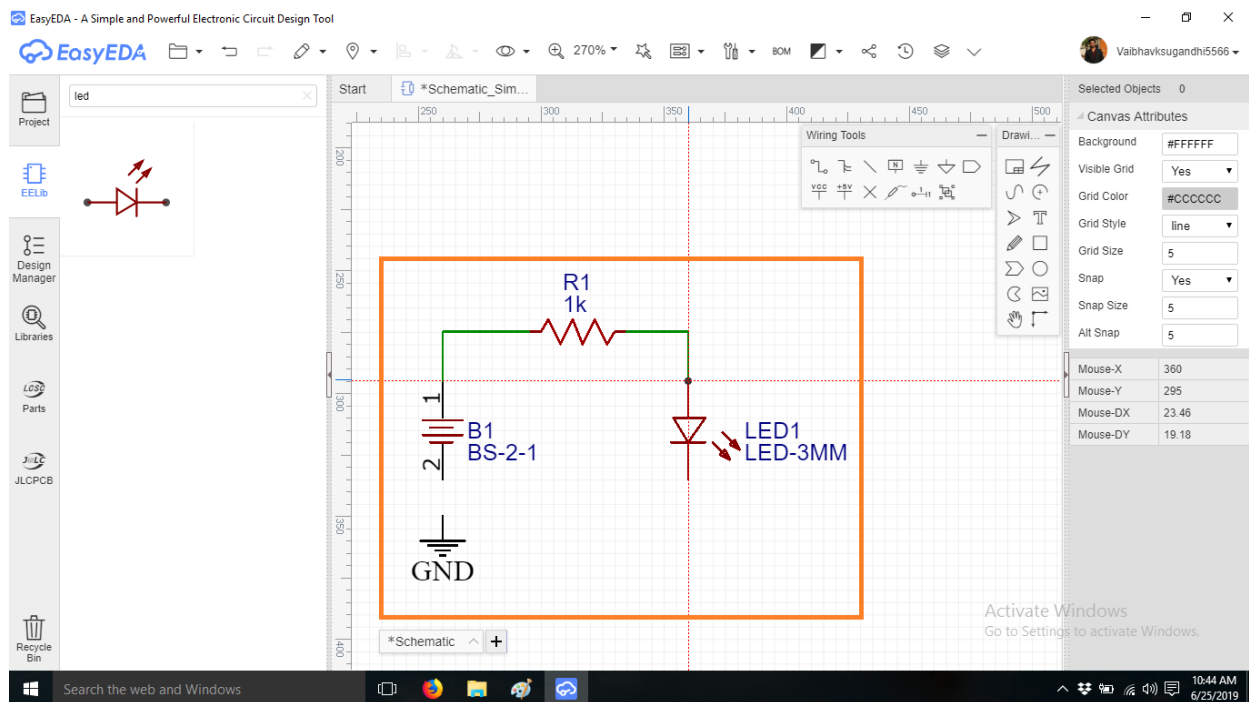

10. Now choose the simple resistor as first component and place it on schematic as shown in the image below.
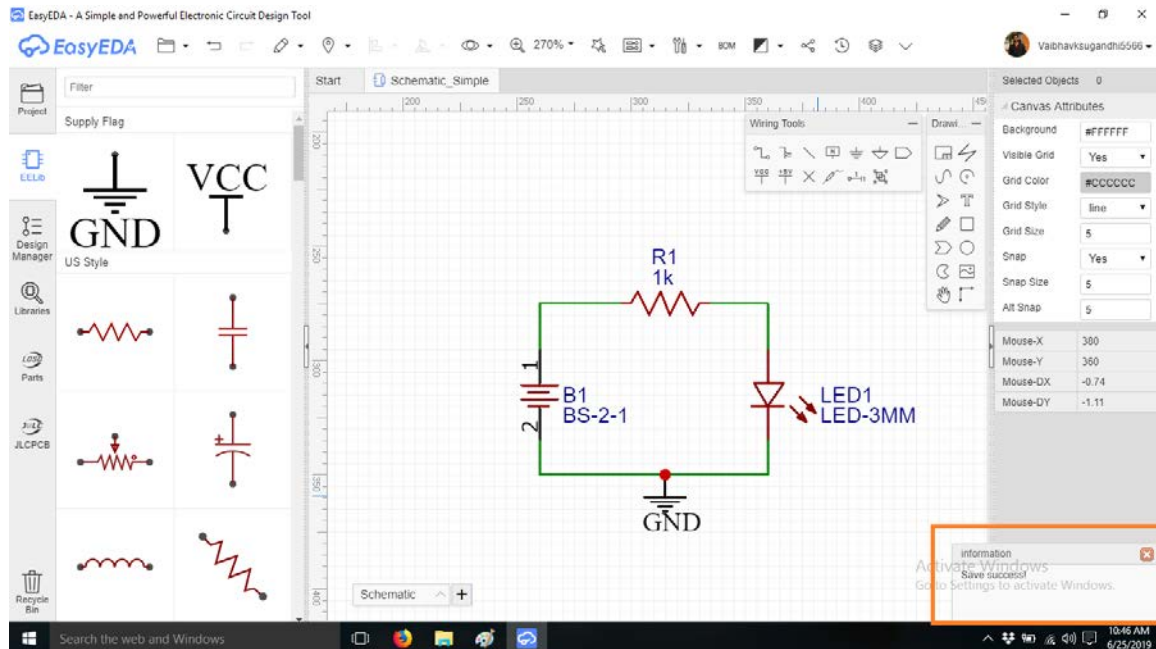
11. Similarly choose the components you want and place them, if any component is difficult to find, then search them in search bar as shown in the figure below.

12. To make connection, we need to use *Wire tool* by pressing "W" key on the keyboard. Join the ends of components with wire as per the circuit design.

13. Once every connection is made, just save the file to avoid unexpected voids in the process. Use CTRL + S for saving. It also shows a small pop up for every time you save.

Congratulations, you are qualified for next step!