

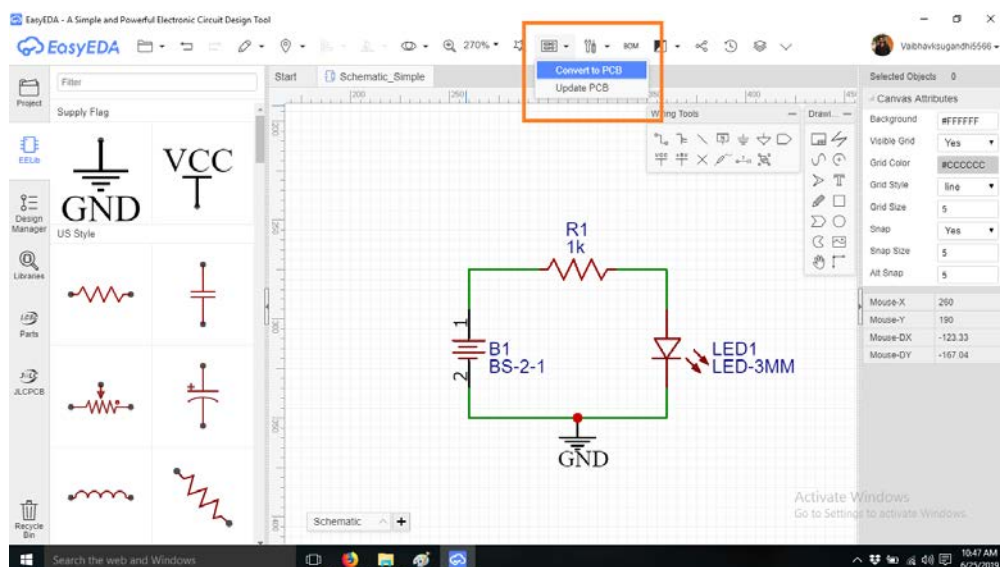
PCB Layout designing in EasyEDA

Created on 25 – July – 2019

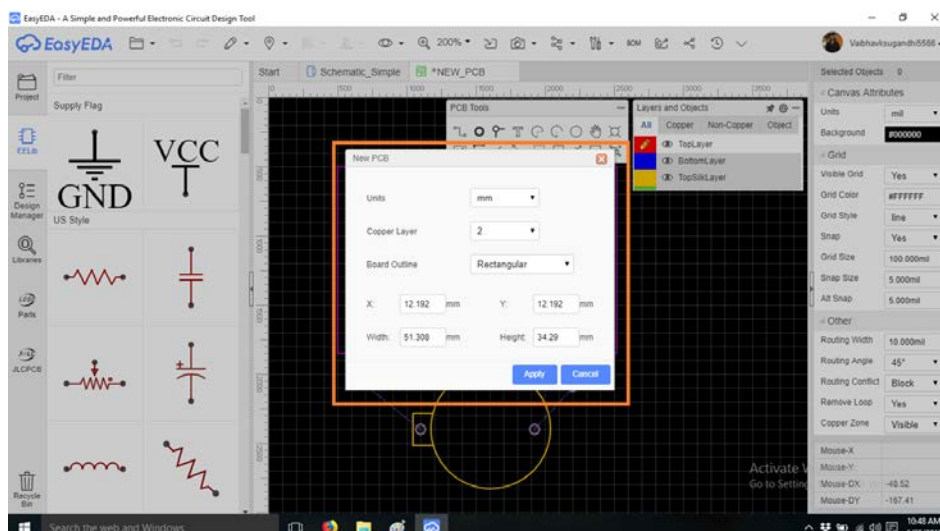


Steps to start from:

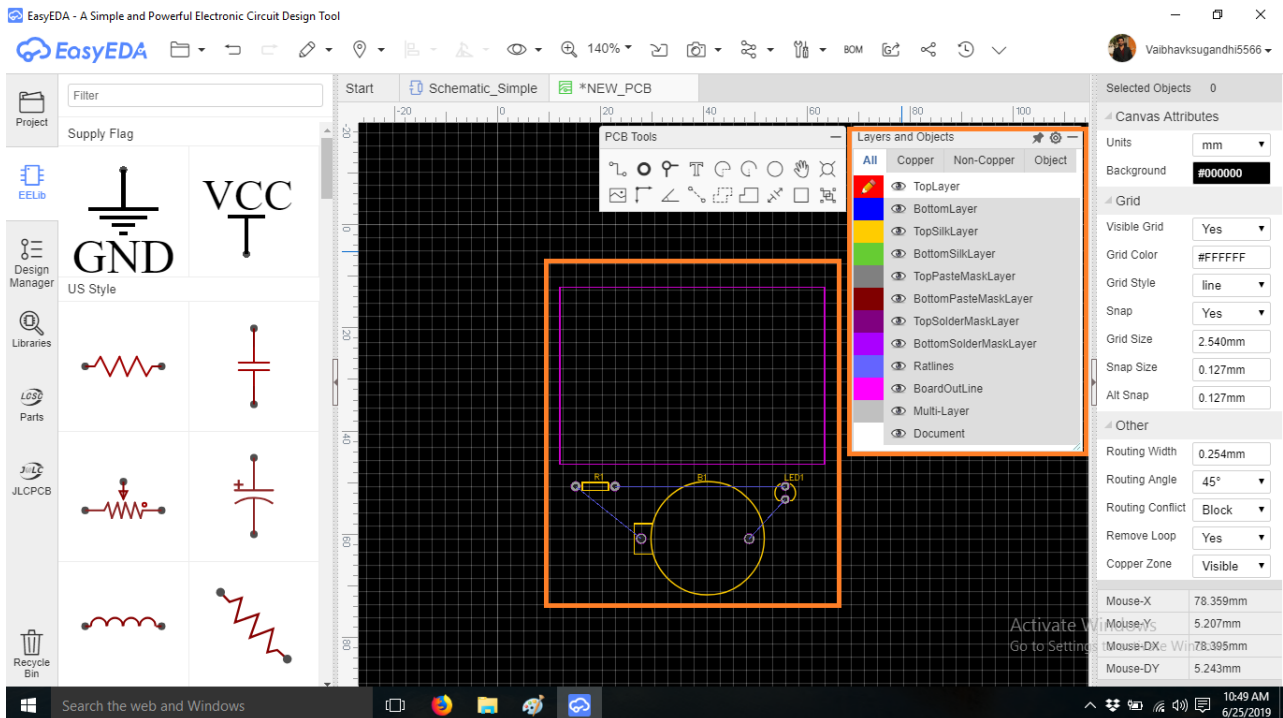
1. Once schematic is ready, then go to Convert icon shown in the image below then click on Convert to PCB.



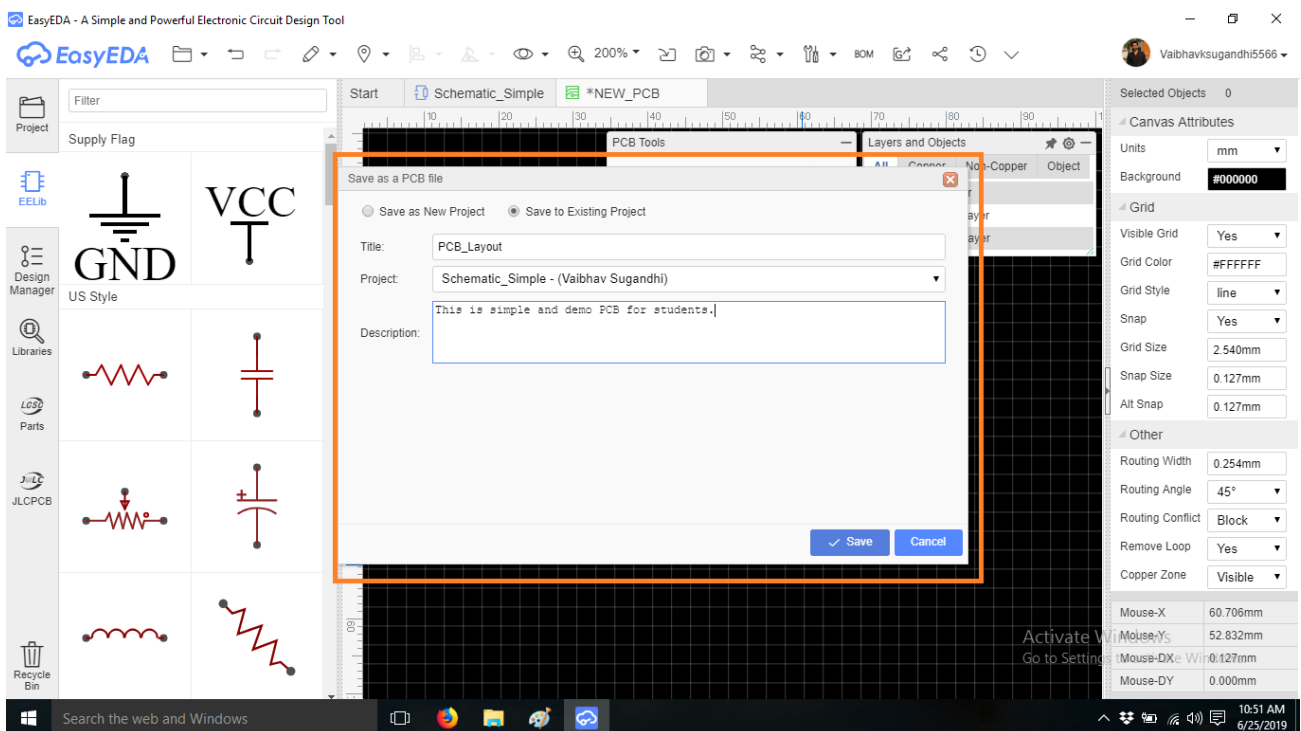
2. A new pop up window will appear for setting board dimensions and units. Choose according to your design and manufacturing. The click on OK.



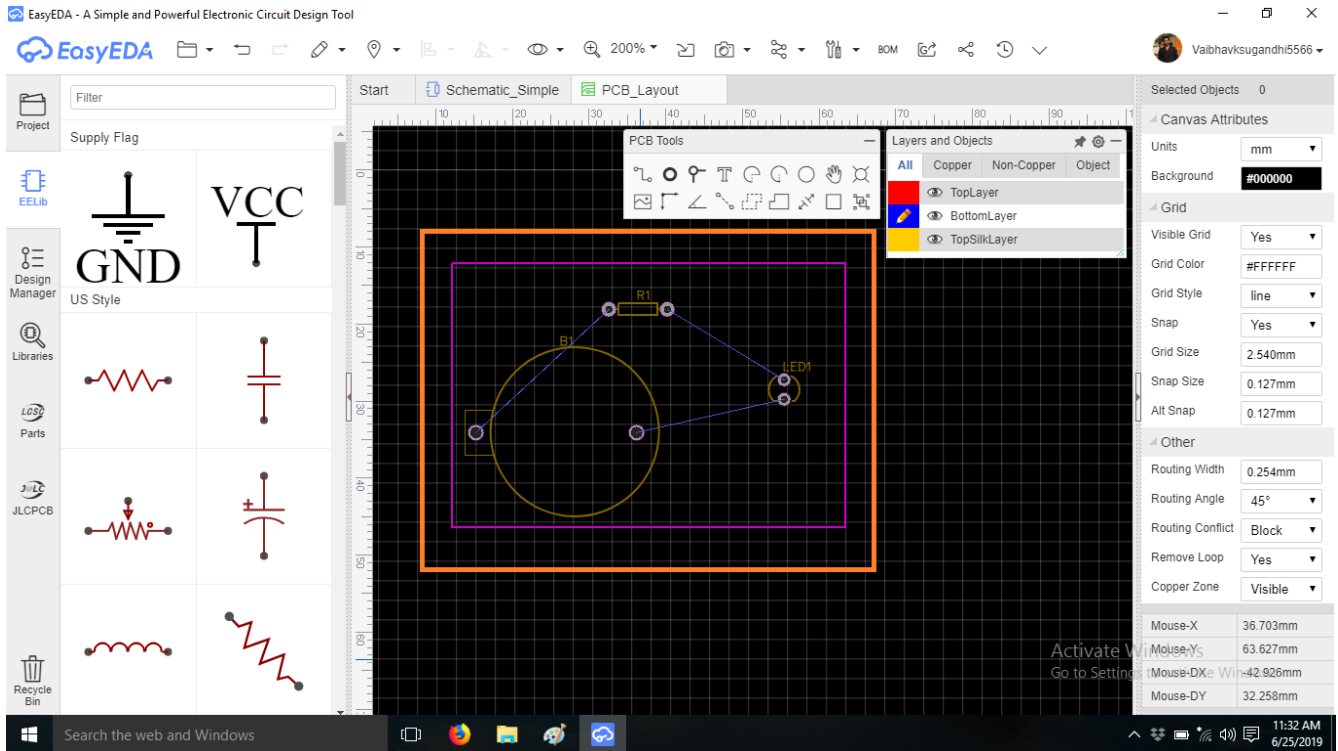
3. New black screen appears with components footprint and empty board. Also you can see the list of layers you can use while designing in the top right.



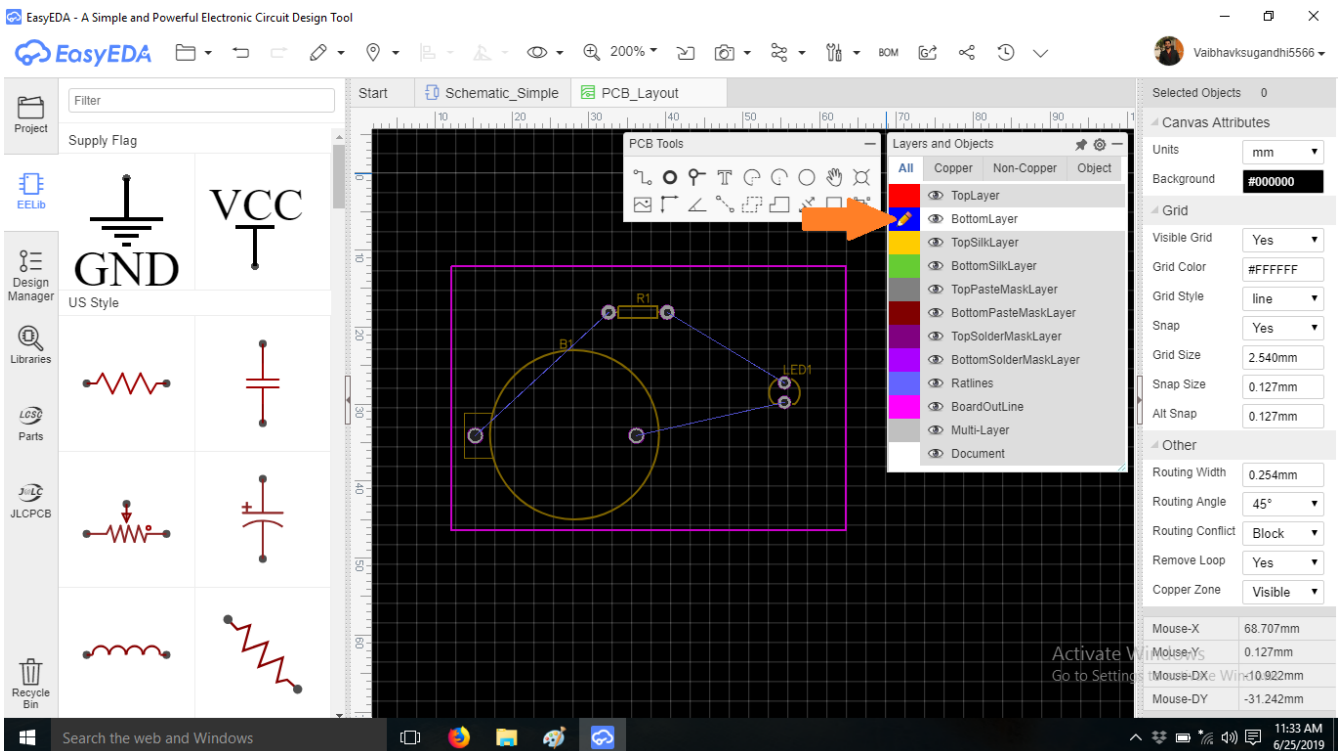
4. Before starting anything use CTRL + S for saving layout file, a window appears and input all details then click Save.



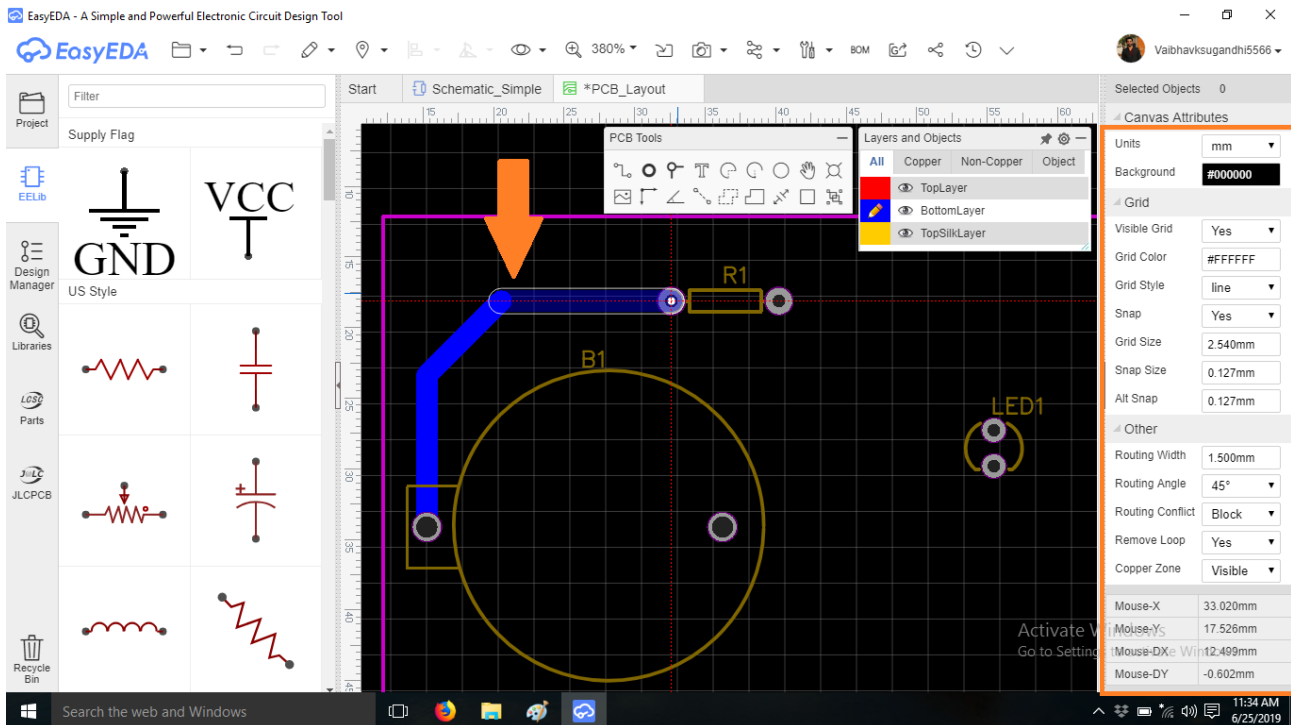
- Now place the component footprints in board as per the requirement and change the orientation if needed by using “R” key on keyboard.



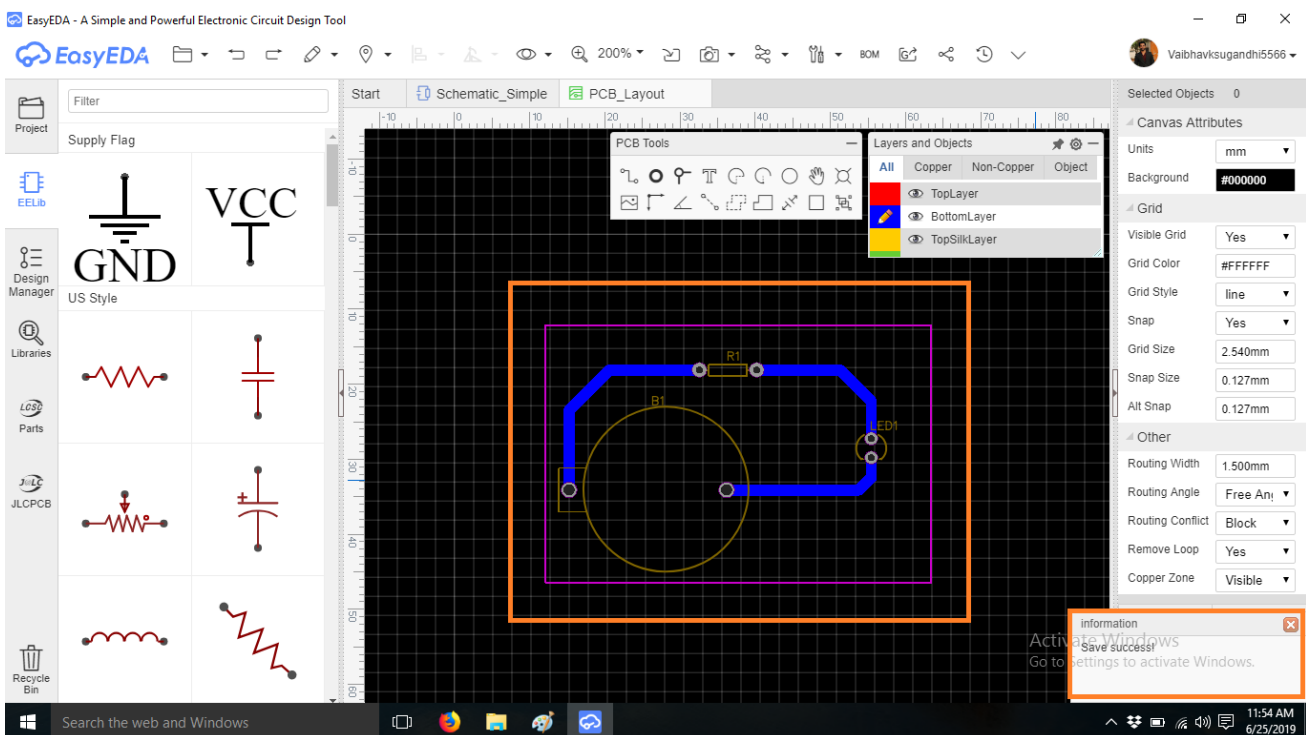
- Before beginning with routing part select the layer as Bottom from the list at right side.



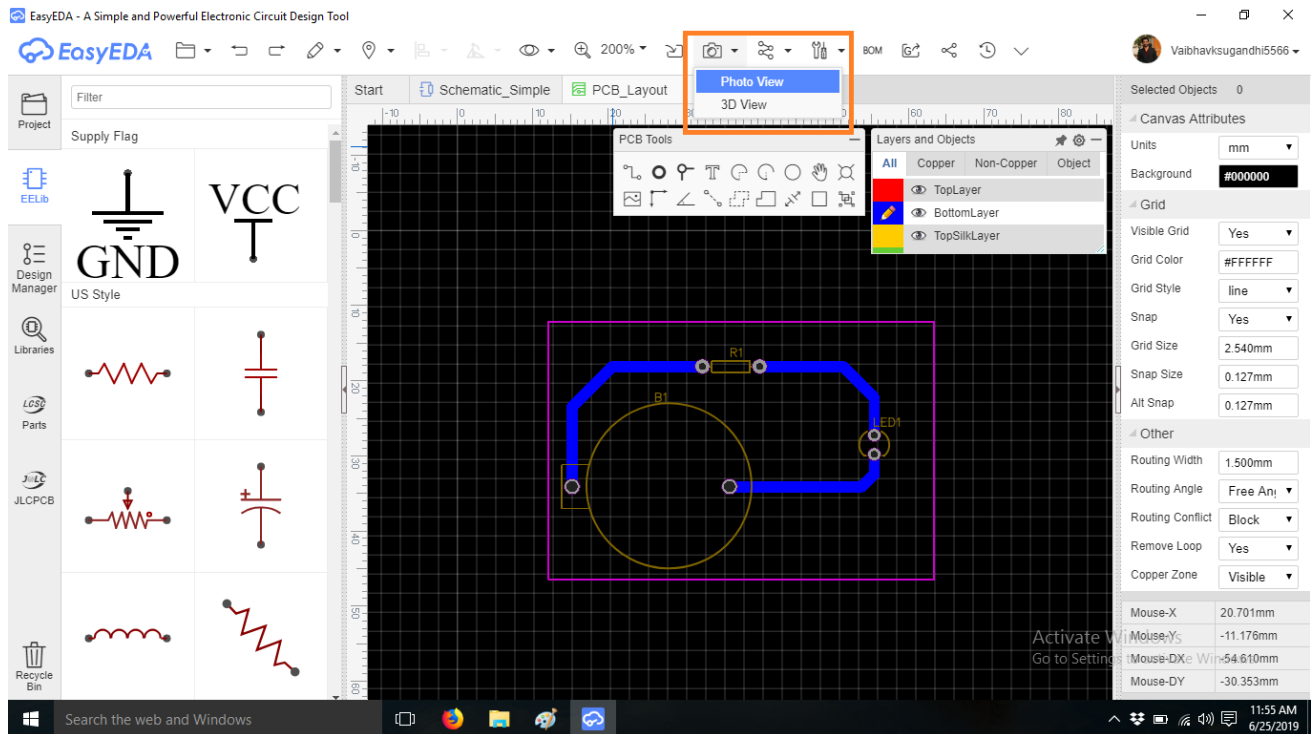
- Use “W” Key from keyboard to quick start wiring/track design on PCB, but if any need to change track width, style or any modification, it can be done from right short window.



- Once you join the one end of component footprint to another end of other component footprint as per the guiding vectors, you are creating PCB. Make sure that all vectors are done.



- Once every track is made, please do save the file by using CTRL + S in keyboard. And small pop up comes every time you save the design.



Congratulations, you are qualified for next step!