

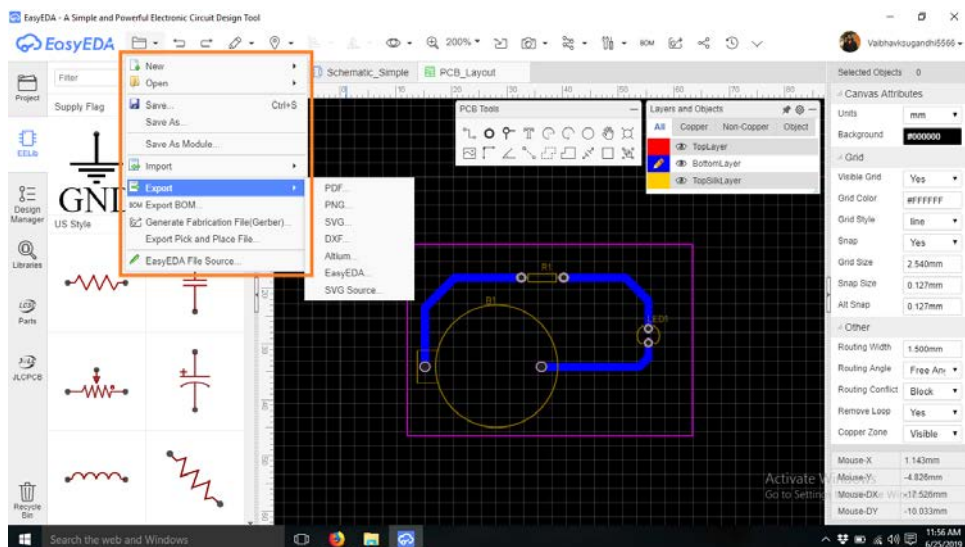
Generating Gerber files in EasyEDA

Created on 25 – July – 2019

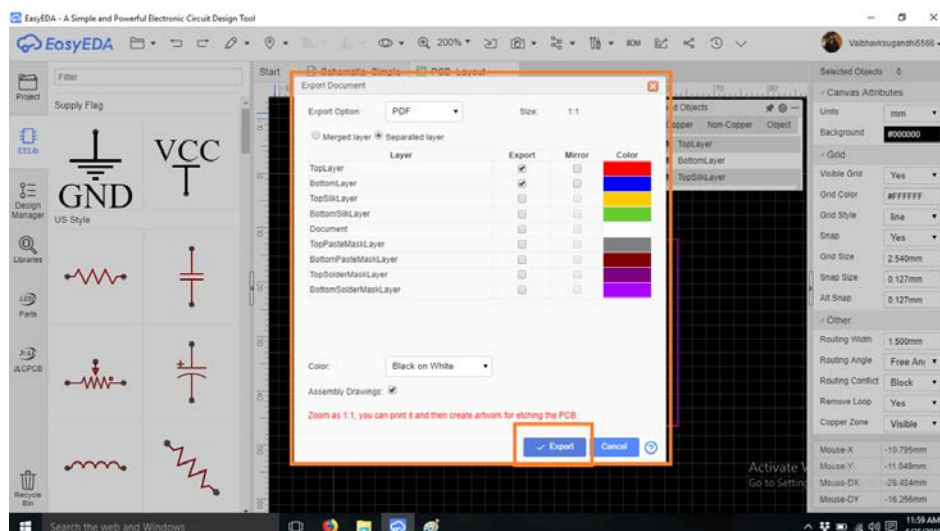


Steps to start from:

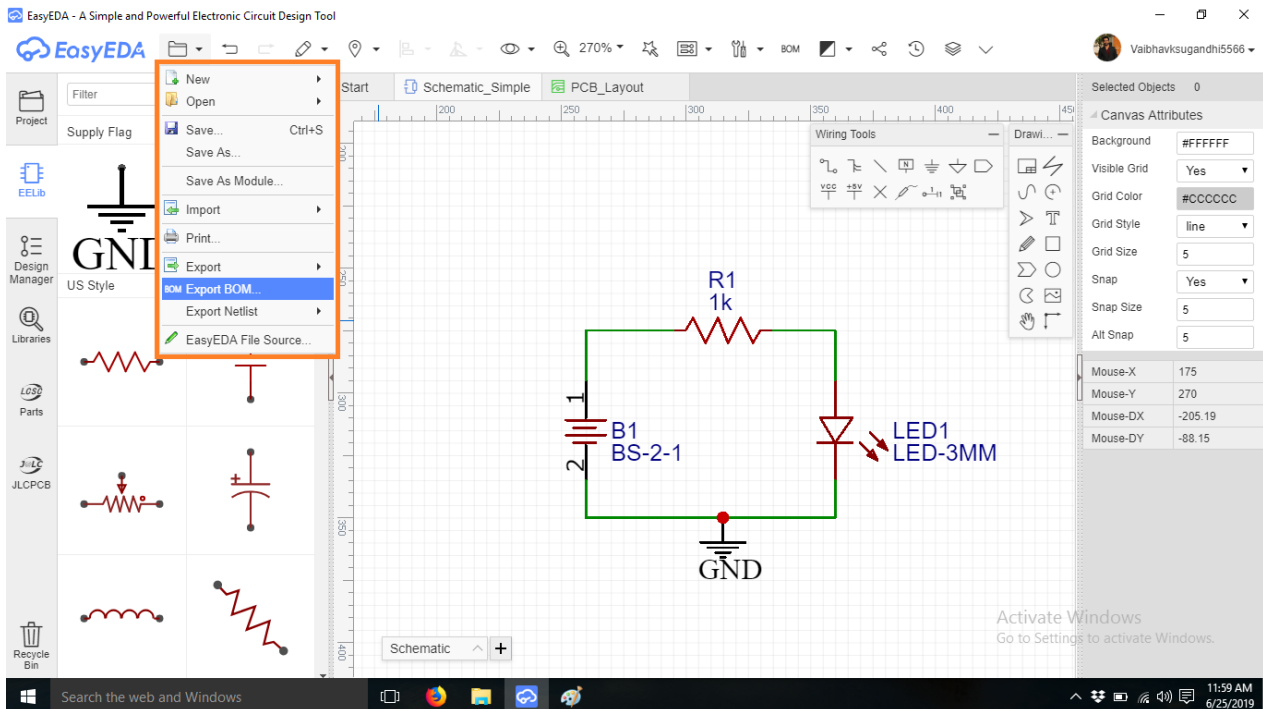
1. Once you are done with creating PCB Layout, then needs to export all necessary files. Export the PDF design by clicking on Folder icon → Export → PDF as shown in image below.



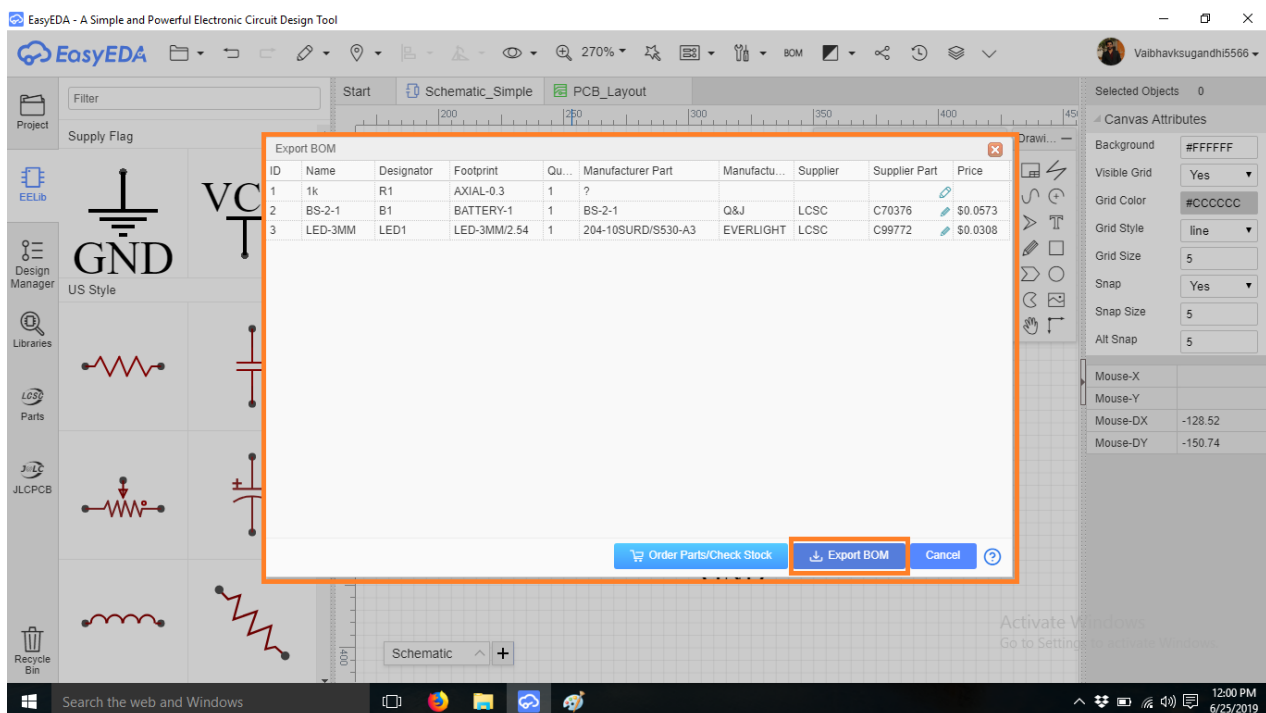
2. A new pop up window will appear for selecting layers and other options for PDF documentation. Then click on Export and save it to one proper location.



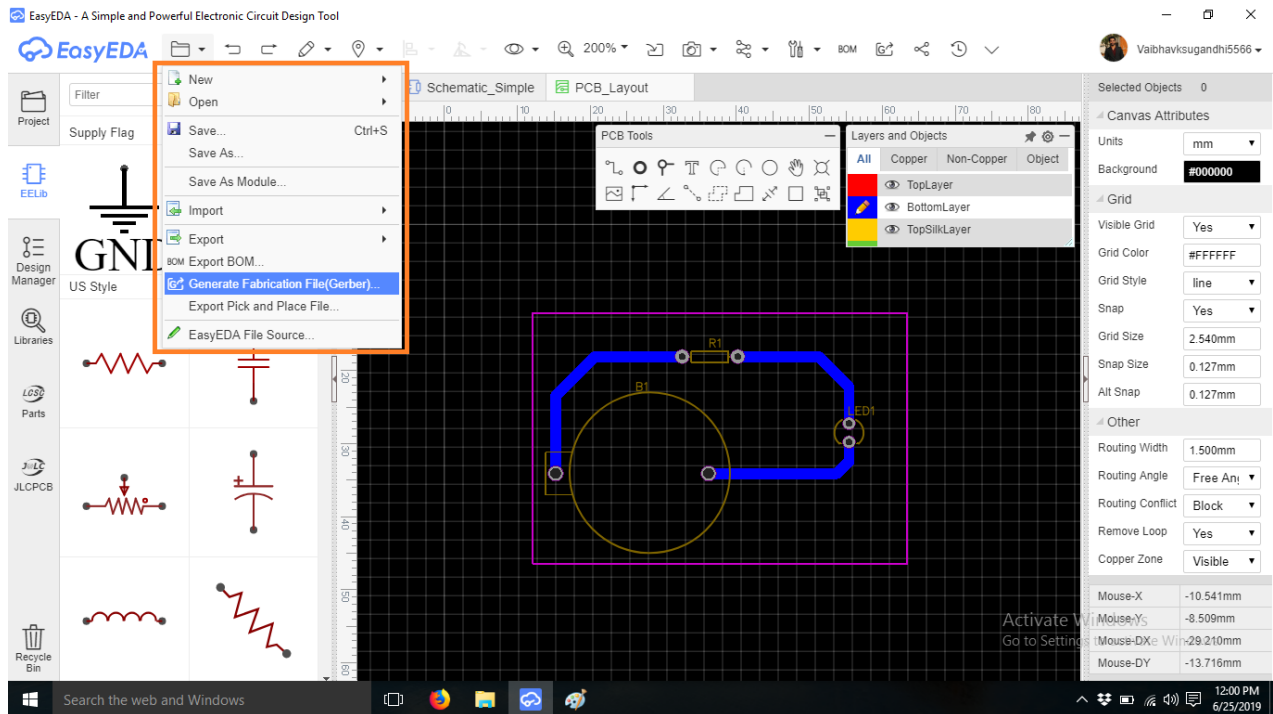
- Go to the same folder icon and click on Export BOM (Bill of Materials). This helps you in manufacturing and assembly process.



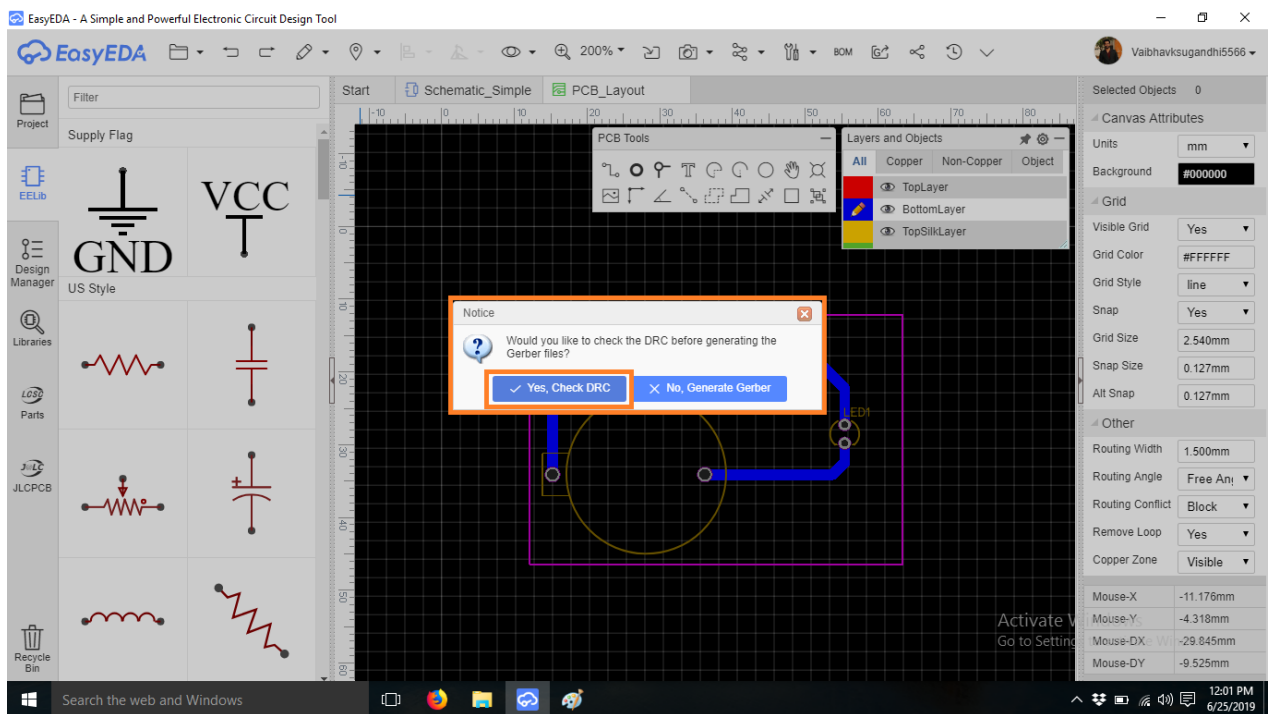
- The new window comes up showing materials list and their footprint associated. Click on Export BOM and save the file in proper location.



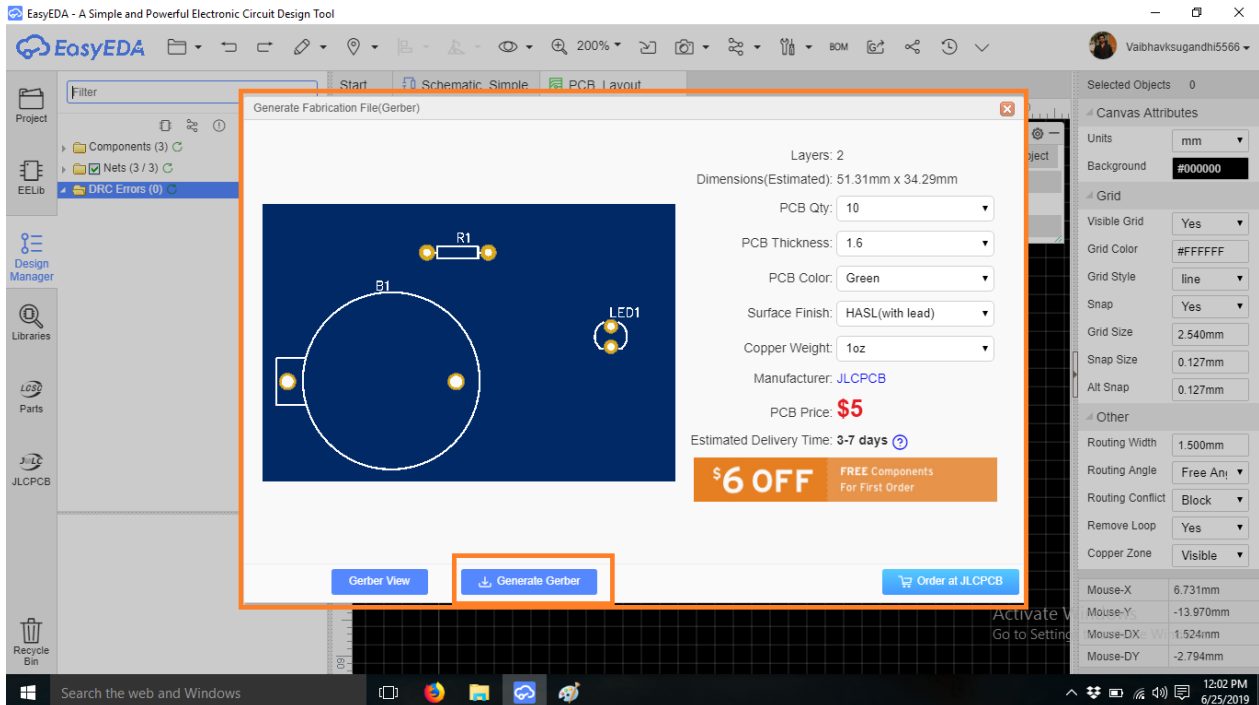
5. Now got to folder icon again → click on generate fabrication file (Gerber)... as shown in below image.



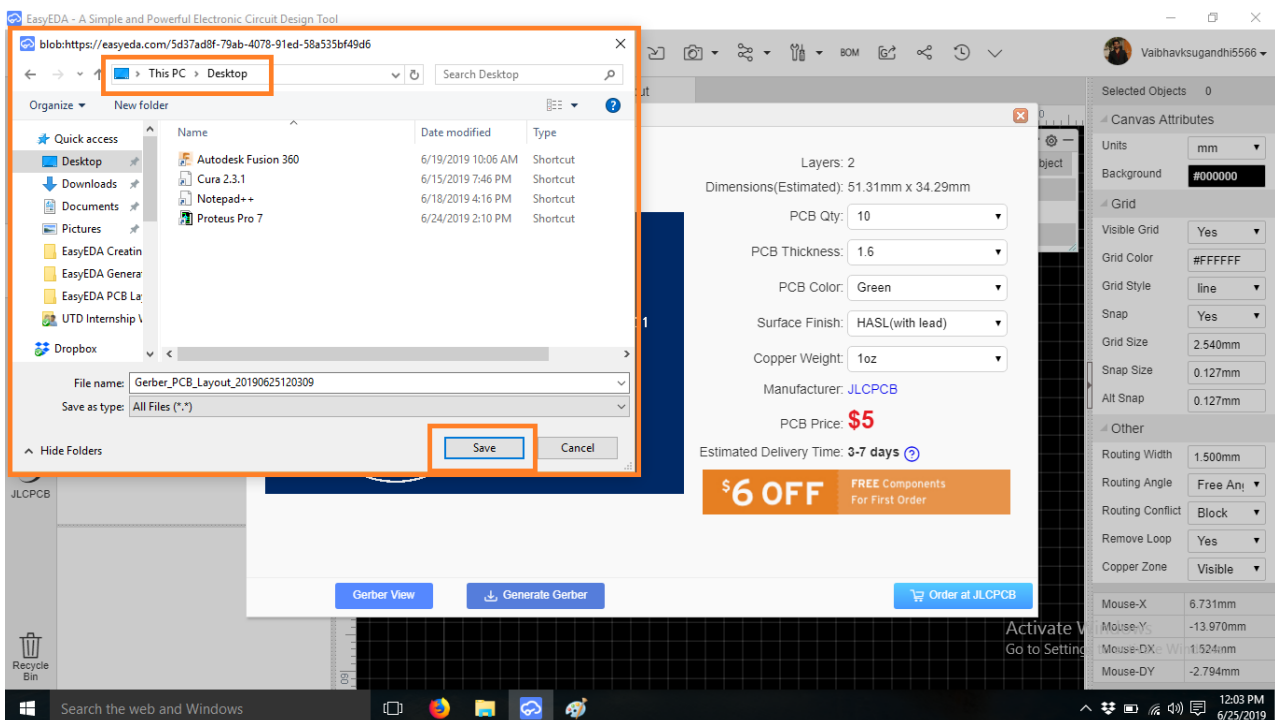
6. Now new pop up comes with asking for DRC check, Click on Yes, Check DRC so that any errors in the design will be notified by software.



- After DRC check, a new window will come up showing offer for PCB manufacturing, but do not order until you know the fabrication parameters. Now click on Generate Gerber.



- Save the final fabrication files in proper location and click on Save.



Congratulations, you are all done!