

Project Kratos

Electronics

QSTP

Week 6 : Board Layout and Routing

This week we will learn about the more interesting part of PCB designing which is the board layout and about routing. Most of the topics to be covered this week are given in the following link:

<https://learn.sparkfun.com/tutorials/using-eagle-board-layout/all>

You can also watch this video for Board layout design: <https://youtu.be/CCTs0mNXY24>

Routing of PCB boards can either be done manually or by using the autorouter tool. However, I would suggest all of you to do manual routing as it allows much more control over the design. For understanding more on routing and autorouter tool and to get some tips on how to do routing do have a look at these links:

<https://www.autodesk.com/products/eagle/blog/routing-autorouting-pcb-layout-basics-2/>

<https://www.autodesk.com/products/eagle/blog/top-10-pcb-routing-tips-beginners/>

One of the important decisions that a PCB designer has to make is to decide the trace width of the copper track. Therefore it is important to understand the factors which affect the trace width in a PCB:

<https://circuitmaker.com/blog/deciding-trace-width-part-1>

For calculating trace width for this week's assignment use the trace width calculator given in the following link:

<https://www.digikey.com/en/resources/conversion-calculators/conversion-calculator-pcb-trace-width>

Make sure you apply Design Rule Check (DRC) at the end in order to ensure that your board is free from errors and that it is ready for manufacturing. You can have a look at the following link for some of the most commonly occurring errors in board design shown in DRC:

<https://www.element14.com/community/thread/17622/!description-of-errors-in-erc-drc?displayFullThread=true>

In order to send our PCB designs to the manufacturers, we need to send them gerber files which can be understood as a universal language for PCB designs used across all PCB designing softwares:

<https://www.instructables.com/id/How-to-Generate-Gerber-Files-From-Eagle-9/>

Assignment

For all the questions:

- consider depth of track to be 10mil, ambient temperature to be 25°C and temperature rise to be 5°C
- submit only the board layout file (.brd)
- calculate the trace width of tracks using the link shared on the previous page
- Take the readings for 'external layers in air' section in the trace width calculator link

1. A 15 V battery supplies current to a component whose resistance is 7 ohms. Calculate the trace width and design the board. Assume input and output is through XT60s.
2. Design PCB board for Q.1 of the last assignment (Assignment 5). Assume that the current rating for all the components is 0.5 Amps.
3. Design PCB board for Q.3 of last assignment (Assignment 5). Assume that the current rating for sensor is 200mA and for component is 0.5Amps.
4. Design PCB board for the schematic file given along with this pdf . For this particular question use track width to be 80mil.

The deadline for this assignment is **Sunday, 21 June, 11:59pm**