

SIT210: Embedded Systems Development

Task 10.1DHD Design a PCB

Hardware Required

None

Software Required

Any Printed Circuit Board (PCB) design software. Our preference is with EasyEDA (<https://easyeda.com>). However, you can use any other tools as long as we can use it for free to test your PCB.

Pre-requisites: You must do the following before this task

Your project should be at a state that its wiring, sensors and devices are finalised. Alternatively, if your project design is not complete, you need to have completed task 7.3D.

Task Objective

Objective of this task is to understand, trial, and design a Printed Circuit Board (PCB). PCBs are important components in prototyping and delivery of any electrical system. For this task, you will need to design a PCB for your project (please note that you do not need to print the designs for this task, just the design). Your design should consider the following:

1. Your PCB does not need to include the SoC you are using (although some PCB design tools allow you to include chips like ATmega on your design). You can design your PC as a board to be used on the side, or as a shield/hat for your main device depending on which mark you are targeting (see below requirement for D/HD).
2. Your PCB should include all wiring, resistors, LEDs, sensors (either on the PCB itself, or with dedicated location/pins to solder sensors on your PCB). If you are not designing a shield/hat, have interfaces for wiring from your main device to the PCB and clearly mark them on the PCB.

D: If you are targeting Distinction, your PCB could be in any form you wish. You can either make the board fit an enclosure/box of your liking.

HD: If you are targeting High Distinction, your PCB needs to take the form of a Shield for either a 40 pin Raspberry Pi (if using in your project, or designing PCB for 7.3D), or follow Feather standard (to be used with particle Argon).

SIT210: Embedded Systems Development

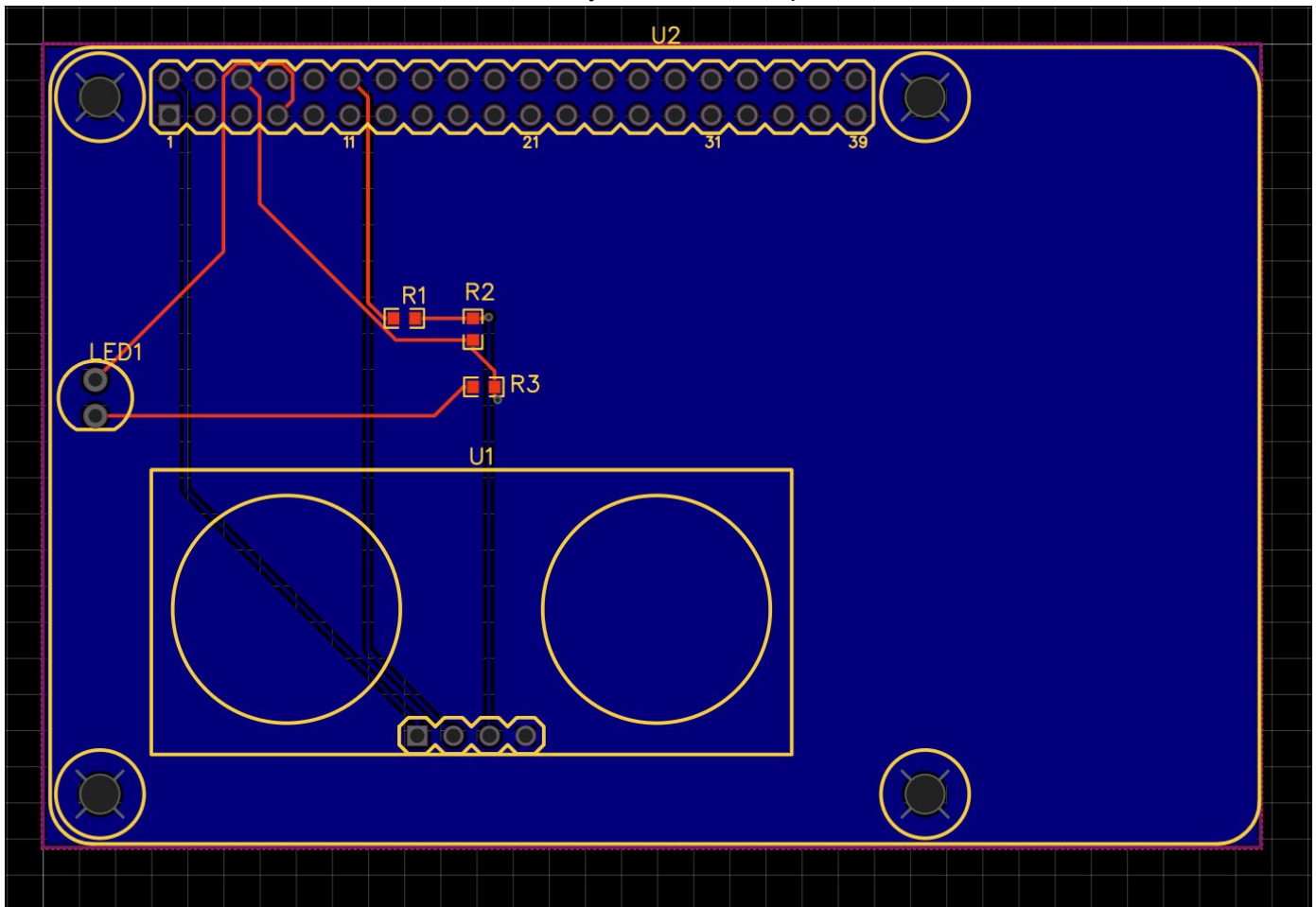
Task Submission Details

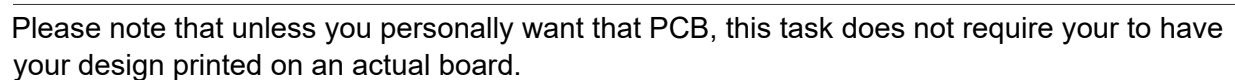
You are required to submit two files for this task:

1. Create a Repository on Github and name it SIT210_PCB_Files and Put your design files/project on Github and a share the link to that repository on a document and submit on OnTrack

https://github.com/coskun-kilinc/SIT210_PCB_Files

2. Provide a screenshot of the PCB you have developed.





Remember, anytime you submit a task to OnTrack, it is a good practice to check the status of any existing tasks, and the future tasks you are expected to complete. If you have got feedback on previous tasks, you may need to fix and resubmit some of your work. You want to check out why, so that you can learn from this and make it faster and easier to accomplish later work to the required standard.