

ENGI301: Project #2

Due: Friday 12/03/2021 11:59pm (Will not be considered late until Tuesday 12/14/2021 11:59pm)

Goal: To create a Printed Circuit Board (PCB) that can be shared with the open source community.

This project is about creating a PCB that is based on your prototype from Project #1, if possible. Depending on your Project #1 prototype, you may need to add or remove components for Project #2. Projects should be scoped to take about 2 weeks of development time.

Project will consist of 3 parts:

1) Proposal (20% - Due: Tuesday 11/16/2021 11:59pm):

Using the Project 2 PPT template, create a proposal that contains the following:

- a. A description of the board functionality
- b. A high-level functional block diagram
- c. A mechanical block diagram that gives approximate dimension of the PCB

The file should be checked in to your github account under “project_02/docs” before the assignment due date. Email welsh@rice.edu or erikwelsh@gmail.com with a link to the document. You will have a meeting to discuss the proposal on 11/16 or 11/17.

2) Implementation (40%):

- a. Create github repository for your PCB
- b. Create an EAGLE sub-folder that contains your Library (lbr), Schematics (sch), and Layout (brd) (all files should have the same root name)
- c. Library
 - i. You must create **at least two** devices
 1. Your own PocketBeagle device. It can be similar to the one we used in class but should contain at least two symbols and have a logical organization for your project. The footprint should be labelled appropriately for your project.
 2. The other component can be of your choosing
 - ii. All components used in your schematics / on your board should be in your library
- d. Schematics
 - i. Make sure schematics are printable
 - ii. Add Frames and documentation
 - iii. Add and connect all components
 - iv. Add any supporting documentation as text
- e. Layout
 - i. Create board outline based on mechanical spec
 - ii. Place components
 - iii. Route the board (please come discuss if you want smaller design rules):
 1. 10 mil trace / 10 mil space (signal)
 2. 15 mil trace / 15 mil space (power / gnd)
 3. 12 mil drill / 24 mil vias
 - iv. Clean up silk screen & Add documentation (board name, revision, initials, etc.)
- f. Create an MFG sub-folder for your manufacturing files
 - i. Generate and check in gerbers using the default CAM settings
 - ii. Check gerbers with gerbv (<http://gerbv.geda-project.org/>) (or other gerber viewer)
 - iii. Check pictures (screen shots) of your top and bottom layers in to “project_02/docs”

You are free to import any symbols, footprints, devices from the ENGI301 library or any other Eagle Library you find. You must create **a least two** library components for your design (see section 2c). These components can be created from a datasheet or an open hardware reference design.

3) Documentation (40% - Due: Friday 12/03/2021 11:59pm; Not late until 12/14/2021 11:59pm):

Your project needs to be documented with the following information:

- a) All implementation files should be checked in to github under the "project_02" directory
- b) Project has appropriate README.md describing the project
- c) Create a PDF of your schematics and check it in to the "project_02" directory
- d) Generate a Bill-of-Materials (csv) that has:
 - Designator, Manufacturer/MPN, Quantity, and Description for all components
- e) Create an account at <https://macrofab.com/> with a dummy password and upload your board
 - a. In your MacroFab project, please make sure that you have selected appropriate components. When possible use house parts. Through-hole components can be DNP since you can solder those to reduce cost.

Building your board is up to you, but you will be responsible for any costs associated with the build.

Turn In: Email welsh@rice.edu or erikwelsh@gmail.com with:

- 1) Link to your github repository
- 2) Names of the library components you created
- 3) MacroFab login information (you can change your password after 12/23/2021)
- 4) Price quotes from MacroFab for: 1, 10, 100, 1000 boards

Extra Credit (5pts): Add your own logo to the board layout by importing a bitmap.