ENGI301: Project #2

Due: Tuesday 12/13/2022 11:59pm (Last Day of Finals)

Goal: To create a Printed Circuit Board (PCB) and practice skills learned in Eagle

This project is about creating a PCB that is based on the block diagram in ENGI301_project_02.pptx. Project will consist of 3 parts:

1) Mechanical Block Diagram (20%):

   Using the Project 2 PPT template, create a mechanical block diagram based on the System Block Diagram

   a. Board should be no larger than 4" x 5"

   The file should be checked in to your github account under "project_02/docs"

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 your own PocketBeagle device.
      ii. You must import the SPI screen from Adafruit: https://www.adafruit.com/product/1770
         1. Use method described in class to import from layout.
      iii. All components used in your schematics / on your board should be in your library
   d. Schematics (Hint: Use ENGI301_Breadboard_Wiring.pdf to help with SPI and USB hookup)
      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 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.

Documentation (40%):

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 12/21/2022)
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 (Hint: Use "import-bmp" ULP)