

## To Do List: Week 5

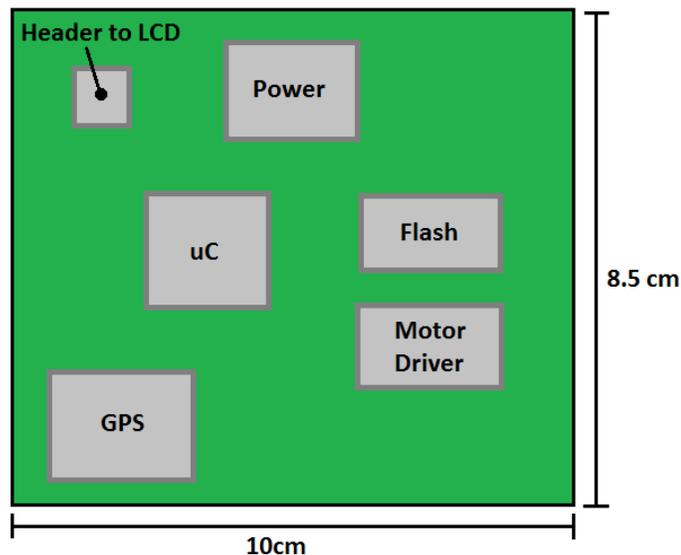
### Coursework:

#### 1. Assignment 7: Mechanical Overview

Due Friday at 11:59pm. Assignment upload portal: [link](#)

Notes:

1. Compare commercial packaging of 2 competing products
2. Generate a CAD model of your product packaging design. AutoCAD (free student version available), Google SketchUp, or other CAD packages are acceptable, provided design is (1) dimensionally accurate (drawn to scale) and (2) screenshots of CAD model include dimensions within the model.
3. Generate tables of packaging specifications. Include a list of materials (what your packaging will be made out of as well as connection hardware such as screws, nails, nuts, bolts, etc.) with estimated costs, tooling requirements (will your project require a 3D printer to make? Drills? Saws? Hammers? Tell us that sort of stuff here). Estimate the weight and provide us with any other packaging specifications you deem relevant here.
4. Provide us with a sketch of your PCB layout. This sketch should be fairly simple; treat your PCB and major components as boxes (draw all components to scale and list dimensions); its purpose is to allow you to plan ahead concerning the size and general layout of your printed circuit board. An example is included below:



### Manlab:

#### 1. **PCB Footprint Library and Electrical Schematics:**

1. The individuals responsible for the electrical schematic and PCB layout of their teams' designs should bring their own laptops to mandatory lab this week and work through the Eagle PCB Tutorial, posted here: <https://engineering.purdue.edu/ece477/Resources/T1-EaglePCBLayou.zip>
2. By Friday at midnight of week 6, students are expected to have a completed Eagle parts

library and a modestly-developed electrical schematic for their team's project. Students should be able to demonstrate substantial progress towards this objective by mandatory lab hours of week 6.

3. Upcoming deadlines for the PCB Design and Layout (Post all completed items to student project websites):
  - **Week 6:** Eagle Tutorial Completed, Eagle Parts Library Completed, Electrical Schematic preliminary but substantial effort has been made
  - **Week 7:** Electrical schematic completed, PCB Layout preliminary but substantial effort has been made
  - **Week 8:** Midterm Design Reviews. PCB Layout completed.
  - **Week 9:** Final tweaks to Parts Library, Electrical Schematic, and PCB Layout made. PCBs submitted at end of week.
  
2. **“Show me a thing”:** For individuals not directly involved with the hardware design process, prototyping progress is expected of students each week. Students should be able to demonstrate a prototyped system and describe progress made on the system to course staff. For any technical issues requiring support, students are expected to fill out a Tech Support Request, which can be found on the student web template in Files/support.
  
3. **Component Ordering:**
  1. Have one student from your team place a components order. ECE477 provides some money for reimbursement of parts ordered for the course which is described in detail here: <https://engineering.purdue.edu/ece477/Course/Process/ReimbursementProcess.pdf>
  2. For each major part used by your team (excluding passive components), obtain a copy of that part's datasheet and host it on your project website.
  3. For each part used by your team (including passive components), identify an acceptable component within your chosen PCB CAD tool for use in an electrical schematic. Any parts which cannot be found will have to be created. Develop a parts library and create all necessary schematic/layout footprints for your parts.