MRSD Project Course Assignment 10: Power Distribution System PCB

Due Dates:

Final PCB (Hard deadline):	Refer to Canvas

Milestones:

Conceptual Design:	Refer to Canvas
Draft schematic:	Refer to Canvas
Draft layout and BOM:	Refer to Canvas

Hand-in Instructions

Hand all files in through blackboard as zip files with the following naming conventions:

- Conceptual design: `yourTeamID_PDSconceptual.zip`
- Draft schematic (include sch and any libraries you made or altered): `yourTeamID_PDSschematic.zip`
- Draft layout (include sch, brd, lbr, and csv or xls for the BOM): `yourTeamID_PDSlayout.zip`
- Final files (include final sch, brd, lbr, manufacturing, and BOM files, and links to your FreeDFM results): `yourTeamID_PDSfinal.zip`

Purpose

Design a power distribution system printed circuit board (PCB) for your project using Eagle. If a power distribution system is not useful to your team's project, then select another function for your PCB and okay it with the instructors. This is a **group assignment** and should be worked on collaboratively with your team. This board should have a form factor appropriate for use in your system and should address power distribution and fault protection for your system.

Milestone 1: Conceptual Design

Begin by describing the power needs of your system.

List the source(s) of electrical power and the subsystems to be powered. **Be specific**: for sources include the type with as much specificity as possible (for example, include battery chemistry, number of cells), and also specify voltage range and continuous/peak current output.

For the various subsystems receiving power list the required voltage range, whether regulation is required, and how much current it will draw continuously and in peak situations (don't neglect peaks that you don't intend but which are likely, like a stalled motor).

Once you've analyzed your system's needs, describe the design requirements for power for each subsystem. These requirements should include:

1. Number of connectors and current capacity of the connector (for example, if you have three motors you will likely need three connectors).
2. For each source, detail if you plan to monitor input voltage, if you require a manual switch, and what the main overvoltage/reverse voltage protection will be.
3. If a subsystem requires a voltage regulating circuit, set a desired efficiency/cost, output voltage, and peak output current.
4. If a subsystem requires more advanced control, like FETs or H-bridges, provide the peak output current and minimum/maximum required operating voltage.
5. Detail any reverse voltage/overvoltage protection for the subsystem.

6. Detail if you need to be able to control power to the subsystem (does your controller need to be able to enable/disable power to a given subsystem)
7. Detail if you plan to monitor the input or output voltages with the system controller, and whether you intend to provide visual indication (i.e., an LED) that power to a given subsystem is on.

Note that it is not necessary to list individual parts at this stage in the process; these requirements should assist you in identifying parts for the draft schematic.

Milestone 2: Draft Schematic

You should draft a full schematic of the planned printed circuit board; make sure you allocate sufficient time to finding and capturing (in an Eagle library) parts that you need.

Include a brief analysis of how well the critical parts (voltage regulators, H-bridges, etc) on the circuit board meet your design requirements. Note that it is reasonable to change the design requirements if your original requirements were too stringent (i.e., the $0.10 99% efficient linear regulator you wanted just didn’t exist), but explain what changed and how.

Analyze the efficiency of each regulated subsystem and determine how much heat each regulator must dissipate at nominal and peak conditions. Discuss how you will dissipate this heat, e.g. through large ground planes, heat sinks, force air, or convection in ambient air. Frame this discussion around the maximum heat dissipation given in the regulator datasheet.

Milestone 3: Draft Layout and BOM

Once the schematic is complete, **prepare a layout of the power board**.

Additional requirements for the PCB layout:

a) Traces must be appropriately sized for the required currents (see PCB Trace Width Calculator for minimums). Non-critical traces should use a minimum of 10mil width unless a particular part requires smaller. Assume 1 ounce copper unless you discuss it with us.
b) Labeling should make it extremely clear what each connector and LED are connected to. Additional labels can be created using the ‘text’ command in Eagle. The ‘tnames’ or ‘tvalues’ layer are common top silkscreen layers.
c) Please include a short description or image showing how your PCB will mount into your larger system. In particular, show mounting holes and demonstrate cable harness routing. In general mounting holes secure the board as well as resist connector connection/disconnection stress. Thinking about wiring harnesses during layout generally yields simpler wiring and direct connections.

Please also prepare a BOM. The BOM should list:

- Quantity of a required part
- Value of the part (1k, 0.01uF, LM317, etc.)
- Which part designator(s) the part applies to
- Manufacturer or vendor part number
- Vendor you plan to purchase the part from
- Cost per part
- Parts that are not on the board but are necessary

Milestone 4: Final Hand-in

Your PCB assignment will be completed when you send us the manufacturing files required to make the PCB as well as a report from www.freedfm.com that shows no major errors[1]. Refer to the slides from the PCB Layout lecture for instructions on

[1] Please keep in mind that the report may not be immediately available.

creating the manufacturing files and for the list of critical files. To reduce cost, we will be placing all PCBs on a panel, so you need not worry about ordering these on your own. However, you will need to order electronic components independently, through the MRSD purchaser as usual. **Please do so such that the parts arrive in time for you to show your functioning PCB in the lab on the Wednesday between Progress Reviews 4 & 5 (Progress Review 5 is the Spring Validation Demonstration). This PCB parts order will not be charged against your budget.**

Things to check for the final submission:

1) Verify your CAD files online (www.gerber-viewer.com) before submitting the assignment.

2) Remember that the Advanced Circuits verification report may not be immediately available. Make sure you give yourself enough time. In other words, don't verify a few minutes before the assignment is due.

3) Please make sure that you **only** include the files described in the assignment in your **.zip** file. Do not send all the files in your Eagle directory! Also, please don't include additional directories: we want to see all your files right away when the .zip file is uncompressed, without the need for descending through other directories.

4) Along the same lines, please follow the file naming convention described at the beginning of the assignment. In particular, make sure that you do not use blank spaces in the file name. Use "_" instead of " ".

Tips

- Please use cmumrsdproject@gmail.com as a resource and do not delay work to the last minute.
- There is no limit to the number of times you can send us a schematic/board, so long as your question(s) are specific and coherent.

- You may create multiple PCBs unrelated to this assignment, and we are happy to answer questions about their design and do some basic review/critique of them.