16-681: MRSD Project Course

Task 6: PCB Power Distribution Board

Due Date:

Refer to the assignment on Canvas.

Submission:

Submit single .zip file per team labeled Team[yourteamletter]_Task5.zip, e.g., TeamB _Task5.zip.

Purpose:

To learn the basics of power PCB design and to design a simple power system using Eagle. The skills that you learn in here will help you develop the power distribution and management board for your project.

Teamwork:

This assignment is a **project group assignment** so work collaboratively with your team.

Software:

Download and install CadSoft EAGLE 7.4. This is a free software with certain limitations but it is more than enough for our purposes.

In case the link above doesn't work, go to: http://www.cadsoftusa.com/download-eagle/

Background

Power distribution and regulation are essential to all electronic devices. Integrated circuits generally require extremely narrow voltage ranges to function properly. Devices like motors generally require a minimum voltage to operate in the correct speed range and minimum current to deliver their rated torques. These different requirements mean that even fairly trivial systems will have need for several different voltage sources.

Rather than use separate power supplies or batteries for each subsystem, it is common to derive all needed voltages and currents from a single DC source (such as a large battery) and use DC-DC converters to create supplies for subsystems. Most systems also employ varying levels of protection to protect against common problems, such as short circuit, reversed input, and overvoltage.

STEP #1: Design

We would like you to design an Eagle Schematic that implements the system shown below.

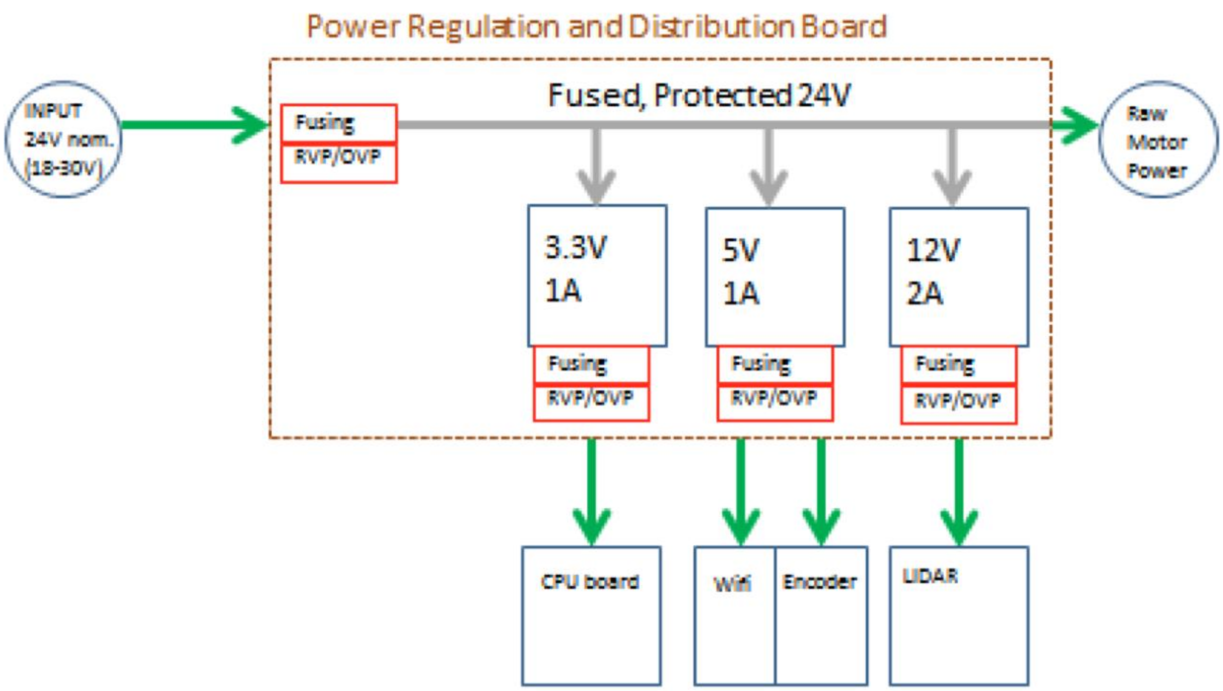

Figure 1. Overview of a basic power distribution board architecture.

The power board architecture consists of the following elements:

A. Four (4) voltage ranges (24V, 12V, 5V, 3.3V) at varying currents;

B. Four (4) protection circuits to protect against short circuits, overvoltage, and reversed input as needed. Define overvoltage as 10% above the nominal voltage;

C. Six (6) connectors for the various inputs and outputs;

D. Four (4) status LEDs to show that each protected supply is operating;

E. Assume the motor requires 10A at 24V. Use a fast blow fuse rated for 150% of the maximum current output required by the motor.

To simplify the design, we suggest you limit the parts you use to the following:

Part	Digikey P/N	Notes
Micrel 29300	MIC29300-3.3WU	3.3V/3A max
Micrel 29300	MIC29300-5.0WU	5V/3A max
Micrel 29300	MIC29300-12WU	12V/3A max
Murata UWS	UWS-3.3/15-Q48NM-C	3.3V/15A max
Murata UWS	UWS-5/10-Q48NM-C	5V/10A max
Murata UWS	UWS-12/4.5-Q48N-C	12V/4.5A max
3.3V TVS Diode	SMBJ3V3-E3/52	SMB package
5V TVS Diode	SMAJ5.0	SMA package
16V TVS Diode	P4SMA16A	SMA package
26V TVS Diode	SMAJ26A-E3/61	SMA package
Mini-Fit JR	High Current Connector	Located in con-molex.lbr
C-GRID	Normal Current Connector	Located in con-molex.lbr
Fusing	TE5 or TR5	Located in fuse.lbr
LEDs	Choose a color in VALUE	led.lbr, use LED with package 1206

Other basic parts can be found in the RCL library in Eagle, such as 1206 resistors and capacitors and 2210 capacitors.

Be sure to implement the suggested reference schematic for each regulator (generally just a few capacitors)!

SECTION #2: Layout

Once the schematic is complete, prepare a layout of the power board. Dimensions, connectors, and layout are up to your team but the connectors and indicator LEDs should be placed in the following way:

Figure 2. PCB general layout of the inputs, outputs, and indicator LEDs.

Additional requirements for the PCB layout:

A. Traces must be appropriately sized for the required currents (see any online "PCB Trace Width Calculator" for minimum width constraints). Non-critical traces should use a minimum of 10mil width unless a particular part requires smaller;
B. Four (4) mounting holes should be included for 6-32 screws (approx. 0.125" diameter holes);
C. Labeling should make it extremely clear to what each connector and LED is connected. Additional labels can be created using the 'text' command in Eagle. The 'tnames' and 'tvalues' layers are common top silkscreen layers.

SECTION #3: Analysis and Documentation

Normally this would be done during schematic design, but in this case we would like you to **analyze the efficiency** of your system now. We would like the following information:

1. State the efficiency of each of your regulators. For linear regulators, it is simply [1 – ((Vin - Vout)/(Vin))]. Switching regulators will state a nominal efficiency in their datasheets.
2. State the input power used for each subsystem at maximum rated output.
3. State the total system efficiency at maximum rated output.

Finally, a simple CAD model of the board must be created for automatic or manual import into a mechanical CAD software package (such as Solidworks). A basic, dimensioned drawing showing the following will suffice for submission:

1. Overall dimensions of PCB
2. X,Y offsets of mounting holes from a common point (usually a board corner)
3. X,Y offsets of the connectors from a common point. A connector is usually referenced from its **centroid** or from **PIN1**. Be clear which you use.

Deliverables

Submit single .zip file per team labeled Team[yourteamletter]_Task5.zip, e.g., TeamB _Task5.zip. The file shall contain:

1. Your eagle .sch, .brd, and any custom .lbr files created;
2. A simple text file, in .PDF format, containing your analysis;
3. A .PDF of your dimensioned drawing. No .DXF or .SLDDRW files.

Submit the zip file on Canvas.

References:

Eagle command references for schematic and layout (EagleCommands.pdf) and for creating library parts (Creating Eagle Parts Libraries.pdf) are attached to the assignment on Canvas.

Additionally, lectures 3.1, 3.2, and 9.1 from the former RI Gadgetry course are attached to the Tiny Gadget PCB assignment on Canvas and are of use in figuring out Eagle.