# MRSD Assignment 1.2

## Due Date:  10/4/2013

## Hand-in Instructions

Hand the files in via blackboard as a single .zip file labeled yourgroupID.zip

## Purpose

Design a simple power system using Eagle.  This is a **group assignment** and should be worked on collaboratively with your team.

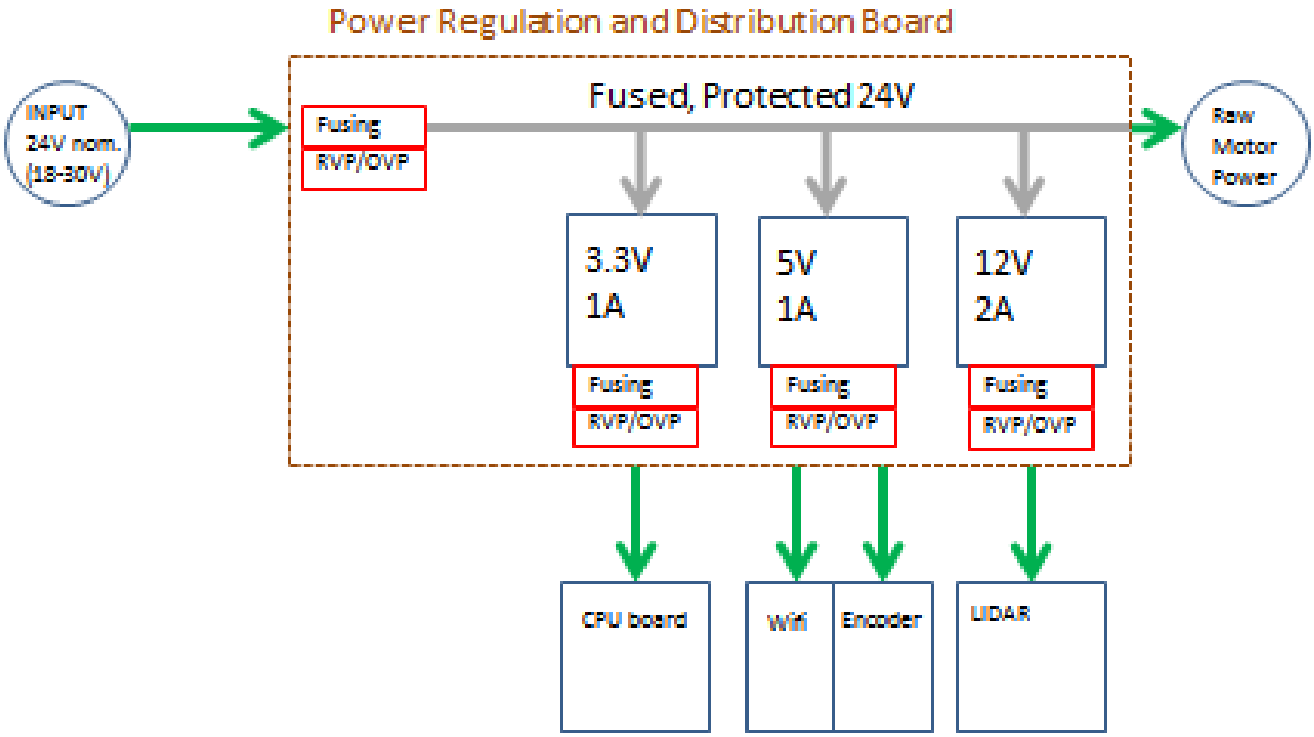
## Background

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.

# Section 1: Layout

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




a) Four (4) voltage ranges (24V, 12V, 5V, 3.3V) at varying currents
b) Four (4) protection circuits to protect against short circuit, overvoltage, and reversed input (as needed). Define overvoltage as 10% above nominal voltage.
c) Six (6) connectors for the various inputs and outputs
d) Four (4) status indicator LEDs to show that each protected supply is operating.
e) Assume the motor requires 10A at 24V. Use fast blow fuses rated for 150% of the maximum output current.

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:


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.

b) Four (4) mounting holes should be included for 6-32 screws (approx. 0.125" diameter holes).

c) 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.

# Section 3:  Analysis & 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:

a) 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.
b) State the input power used for each subsystem at maximum rated output.
c) 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:

a) Overall dimensions of PCB
b) X,Y offsets of mounting holes from a common point (usually a board corner)
c) X,Y offsets of the connectors from a common point.  Connectors are usually referenced from its **centroid** or from **PIN1**.  Be clear which you use.

# Deliverable(s)

One zip file containing the following:

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

# Supplementary Documentation / Links

The latest non-commercial version of Eagle can be downloaded here:

http://www.cadsoftusa.com/download-eagle/

Eagle command references for schematic and layout are available here:

http://gadgetry.ri.cmu.edu/content/eagle-command-reference

Eagle command references for creating library parts are available here:

http://gadgetry.ri.cmu.edu/content/creating-eagle-libraries

Additionally, lectures 3.1, 3.2, 6.2, 7.1, and 9.1 on the Gadgetry website (http://gadgetry.ri.cmu.edu) are of use in figuring out Eagle.