

**Task 7a****Task Purpose:      Design a Printed Circuit Board – Part 1****Due Date:            See Web Site**

The purpose of the next two tasks is to have the group go through all the steps in designing a circuit board. In this step, you will do the schematic portion of the design. I expect that we will iterate several times on this task.

We are in the process of upgrading our Eagle licenses to version 5. You should download the free version from the Cadsoft web site and use it for this task. Note that this version limits you to a single page in the schematic, and a fairly small board. When we have completed the license upgrades, some of the computers in the Electronics lab will be equipped with the full licensed version.

You should base your design on the project that you are doing. This includes the choice of microcontroller, voltage regulator, etc. There should be some common inclusions on your board. They are:

1. All major parts should be surface mount.
2. An FTDI FT232RL serial to USB converter, and associated circuitry. While not all of these projects require this serial interface, debugging software does, and this allows for a simple interface to a terminal, while only requiring 2 pins from the microcontroller. The design should include LED (0805 surface mount) to indicate transmit and receive as shown in figure 15 of the FR232RL document. (Note that the choice of resistor depends on your choice of LED and how your board is powered. The value given in the figure may not be correct for your design.)
3. An SPI serial EEPROM such as the Microchip 25LC1024.
4. A voltage regulator appropriate to your design with the associated capacitors chosen based on information found in the voltage regulator spec sheet.
5. A crystal or ceramic resonator to provide a clock to your microcontroller
6. The microcontroller should use the same reset circuitry and programming circuitry that are used on the class board you were given as part of the kit. The schematic and board files are available in both raw (Eagle files) and PDF's on the wiki.
7. Assume for now that we are powering from an external power supply such as the one you are using in the kit. There is a device in my library called DC-IN that is the appropriate part.
8. Discrete components should be surface mount, in sizes of 0603 or larger.
9. Decoupling capacitors should be used as appropriate.

Notes:

Please create the following net classes in your schematic:

Nr	Name	Width	Clearance	Drill
<input checked="" type="radio"/> 0	Default	20mil	10mil	10mil
<input type="radio"/> 1	Power	20mil	10mil	10mil
<input type="radio"/> 2	Signal	12mil	10mil	10mil
<input type="radio"/> 3	LVSignal	10mil	8mil	10mil
<input type="radio"/> 4	powerfeed	20mil	12mil	10mil
<input type="radio"/> 5		0mil	0mil	0mil
<input type="radio"/> 6		0mil	0mil	0mil
<input type="radio"/> 7		0mil	0mil	0mil

We may end up changing the parameters, but I would like every board to have the names above in the order above. As you are entering nets, please remember to set the class of the signal you are using. In your designs, you should use power and signal for now.

1. Power supply symbols are found in one of two standard libraries named supply1 and supply2. Eagle is a little counter intuitive when it comes to power supply connections. Read the manual.
2. For places to put header pins, such as where the programmer connects on the kit board, look in the pinhead library. (Short for pin headers.)
3. For discrete components are found the in the rlc library. Resistor networks can be found in resistor-sil for single inline resistor networks.

### Task Report:

Your task report should include:

1. A description of the choices you made based on your project. (Microcontroller, powering, etc.)
2. Eagle schematic file which includes the team name in the file name.

3. A copy of the output of the Erc (Electrical Rules Check.) Note that there should be no errors in this listing, but warnings are fine.
4. A copy of your library if you created any parts.
5. The Digi-Key part numbers of **all** the parts you for the USB to serial interface section, along with an explanation of how you sized the resistors.