Task 7

Task Purpose: Design a Printed Circuit Board – Part 1

Due Date: November 29

The purpose of the last 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.

The board that you will be designing uses the 18LF4620 part and the FTDI FT232BM. The FTDI device is a USB to serial interface the can be used for designs where the designer wants to switch from an RS232 interface to a USB interface.

A parts library of some of the major parts for this design, and spec sheets for some of the parts, can be found in the course web space.

At a high level, this board will use the FTDI part to connect the EUSART on the 18LF4620 to software running on a PC using the FTDI FT232BM to provide the RS232 to USB conversion. (FTDI provides drivers for the PC side that make such a device look like a virtual com port.)

The board will be powered by the 5 volts present on the USB bus. The microcontroller is to run at 3.3 volts, derived from the 5 volts USB voltage using a voltage regulator. Figure 14 of the FT232BM spec sheet shows how this might be done.

The board requirements are:

1. The microcontroller used is the 18LF4620 using the TQFP package. It should run of 3.3 volts derived from the USB signal.

2. The microcontroller EUSART should be connected to the UART connections of the FT232BM. The data transmit and receive signals should be appropriately connected, and two handshaking signals (RTS# and CTS# on the FT232BM) should be connected to general I/O pins on the 18LF4620.

3. 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. (Eagle files for this board are available in the course web space.)

4. The microcontroller port D should be connected to LED’s as on the class board.

5. A 93 series EEPROM from Microchip should be connected to the FT232BM. Note that this is the recommended type of EEPROM, and spec sheets and library parts have been provided.

6. Discrete components should be surface mount, in sizes of 0603 or larger.

7. Decoupling capacitors should be used as appropriate.

8. You should implement the dual LED configuration shown in Figure 12 of the FT232BM document. Implementing includes finding surface mount LED (one red and one green) on Digi-Key, and appropriately sizing the resistor based on your LED choices. (The values given in the document may or may not be correct for your LED choice.)

9. Power and ground signals on your board should be a minimum of 15 mils wide with 10 mill spacing and 10 mil drill, and the rest of the signals can be 12 mils width with 10 mill spacing and 10 mill drill. See the Eagle manual about net classes. This might end up be adjusted down for routing reasons in the next task.

Notes:

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. Eagle schematic file which includes the team name in the file name.

2. 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.

3. A copy of your library if you created any parts.

4. The Digi-Key part numbers of the parts you used in part 8 of the task, along with an explanation of how you sized the resistors.