ENGR 3430: Project 1

due September 14, 2017

In this project, you will use KiCad to design a USB-powered LED flasher circuit on a small two-layer printed circuit board (PCB) using surface-mount components. This project is an *individual* project. You can discuss design approaches and help each other with learning KiCad, but each of you must complete all aspects of this assignment in order to learn how to use the tools. In the process, you should learn many aspects of the PCB design process and software tools that you will be using later in the semester on your team projects.

Requirements. Your design must meet the following requirements:

1. Your circuit must flash an LED at a frequency of within 15% of 1 Hz.
2. Your circuit must run on a single-ended 3.3-V supply that is derived from the 5-V VBUS supply in a standard type A USB port.
3. Your circuit design may only use components from the (unmodified) spreadsheet provided with this assignment.
4. Your circuit must include adequate bypass capacitors for the integrated circuits.
5. Your PCB must be a two-layer design with all components on the top side of the board. The minimum allowable trace width and spacing is 6 mils. The minimum allowable via size is 23 mils with a 15-mil drill hole. A small part (i.e., less than 10%) of your grade for this project will be related to the area of your final PCB layout.

Deliverables. By the start of class on September 14, you must turn in the following items:

1. An explanation of your circuit design choices including an analysis demonstrating that your design will meet the frequency specification given the tolerances of the components that you have selected.
2. A copy of your circuit schematic.
3. A bill of materials (BoM) for your design.
4. Copies of all of your KiCad design files (e.g., schematic, PCB layout, and any libraries with schematic symbols/part footprint that you have created).