ENGR 3430: Miniproject 1

due February 5, 2026

In this project, you will use KiCad9 to design a USB-powered LED flasher circuit on a small two-layer printed circuit board (PCB) using surface-mount components. This miniproject is an *individual* one. 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.

Requirements. Your design must meet the following requirements:

1. Your circuit must flash an LED with a period of within $\pm 10\%$ of 1 s.
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 and bulk capacitors for the integrated circuits, as specified in their datasheets.
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 24 mils with a 12-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 February 5, 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 period 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 or aquired that are not in the libraries that come with KiCad).