Eclectronics MP1 Report

Carter Harris

Project Overview

Mini-Project 1 for Olin's Eclectronics course involved building a hysteretic oscillator using an operational amplifier (op-amp) to flash an LED with a period of 1 second, $\pm 10\%$.

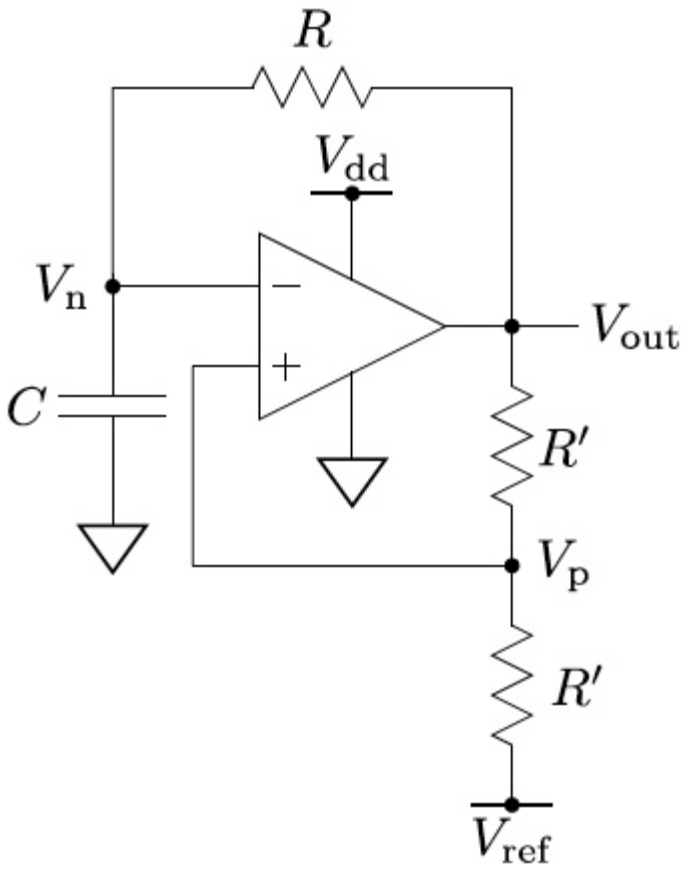
Specifications dictated that this circuit be powered from a USB-A connector's 5 volt bus, which is subsequently regulated down to 3.3 volts using a MCP1702 linear regulator. The op-amp specified is a MCP6022 with two independent op amps available; one used for the actual oscillator, and one as a buffer to isolate the reference voltage divider. The remaining passive components were selected from a provided list of components based on calculated values and tolerances.

The circuit schematic was first designed in KiCAD. Subsequently, the design was validated in LTspice with a worst-case tolerance simulation to ensure that the specified period tolerance would be met with all selected components. Finally, a PCB layout was completed in KiCAD with the goal of minimizing the physical footprint of the PCB. Other constraints on the PCB design included 2 layers, all components on the top, a minimum of 6 mil traces, and minimum 24 mil vias.

All supplementary materials and design files are avalible in this repo: https://github.com/CarterKoopa/eclectronics

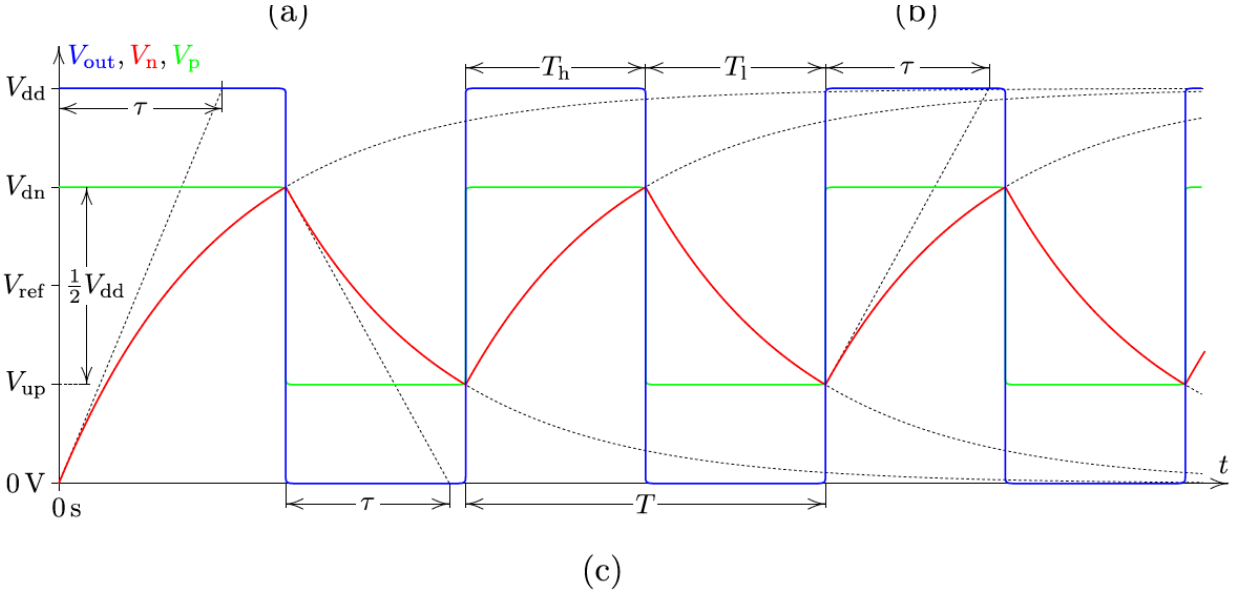
Circuit Overview & Component Selection

An overview of the hysteretic oscillator is shown below:



The hysteretic oscillator uses the op-amps theoretical infinite gain to generate a square wave. The period of this square wave is dictated by the time constant of the RC filter attached to the op-amp's inverting input. A Schmitt trigger configuration is connected to the op-amp's non-inverting input.

The result is the output shown by the blue V_{out} curve in the figure below:



(all figures from *Another Book on Circuits* by Brad Minch)

A typical configuration of this oscillator involves:

- a voltage V_{ref} which is half that of V_{dd}.
- equal resistors for R'

These design choices result in a configuration where $V_{up} = \frac{1}{4}V_{dd}$ and $V_{dn} = \frac{3}{4}V_{dd}$, which simplifies the selection of other components.

Based upon the derivation provided in the above-referenced text, we can determine a relationship to describe the time required for voltage V_n to charge from V_{up} to V_{dn}, the points at which the op-amp switches from infinite gain to off. This relationship, described by the time constant τ, is equal to the resistance multiplied by the capacitance of the RC filter. This relationship is defined as

$$V_{up} = V_{dn}e^{\frac{-T_l}{\tau}}$$

where T_l is the low time of the oscillator's period.

Substituting in our values of V_{up} and V_{dn} relative to V_{dd} as described above, this equation can be rearranged to solve for the time constant τ, such that

$$\tau = \frac{-T_l}{\log \frac{1}{3}}$$

Given our period of 1 second, we want a low time of $T_l = 0.5$. Substituing this value results in $\tau = 0.455$. In the context of our circuit, this means that $RC = 0.455$.

To make a first component selection, I chose the $0.1\mu F$ capacitor, as this was the largest capacitor available in the X7R standard with a tolerance of $\pm 5\%$, rather than the $\pm 10\%$ tolerance provided by all larger capacitors. Later simulation would prove that a 10% tolerance would not meet the required 10% tolerance on the duty cycle in a worst-case scenario.

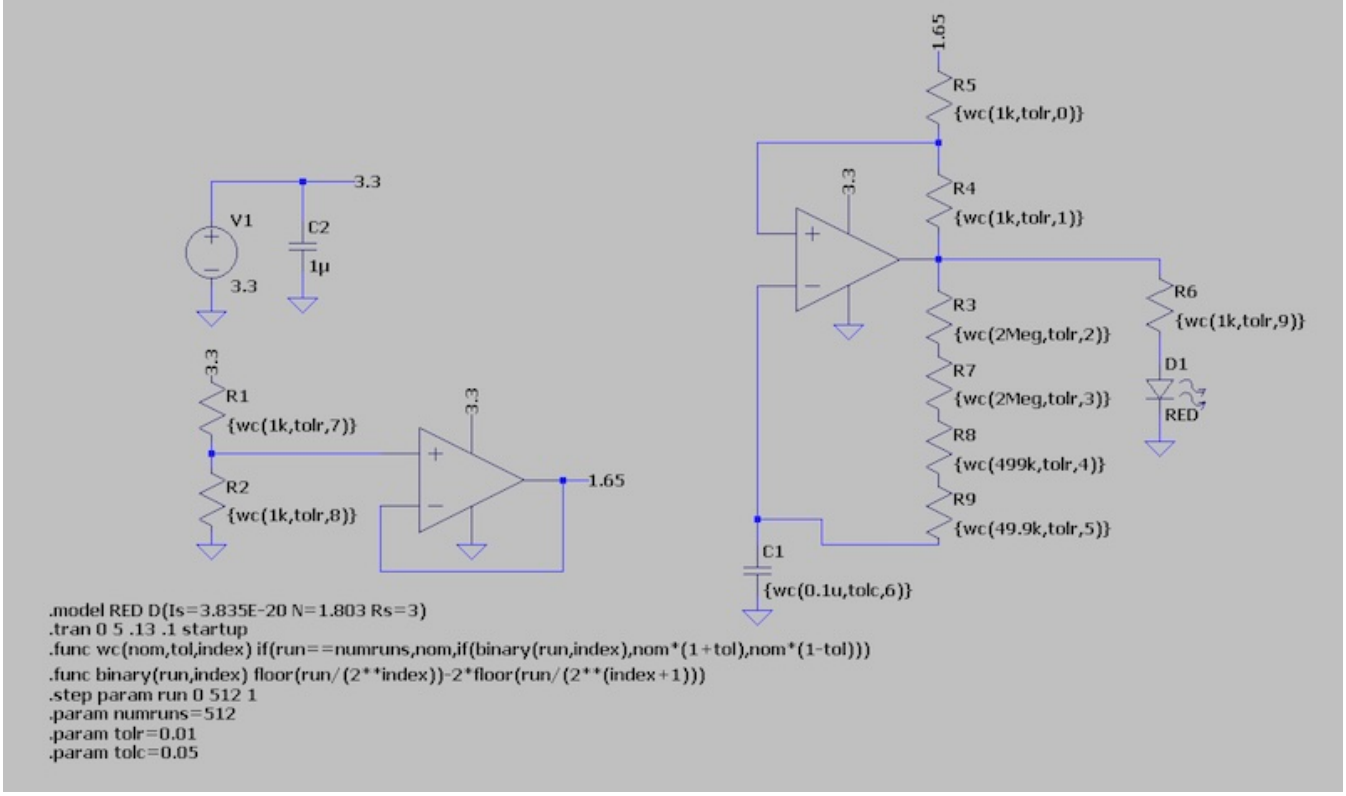
Based upon this capactior value and time constant, we can solve for a resistance of $4.55M\Omega$. While this is not a value available within our provided component selection, we can approximate this value quite well by combining in series two $2M\Omega$ resistors, one $499k\Omega$, resistor, and one $49.9k\Omega$ resistor to get a series resistance of $4.549M\Omega$.

All other resistors in the circuit, including R', the voltage divider to generate V_{ref} from V_{dd}, and the current-limiting resistor for the LED were selected to be $1k\Omega$ resistors. This satisfies all requirements while reducing the number of unique resistors needed.

Other components in the circuit are the MCP1702 linear regulator, who's datasheet recommends $1\mu F$ bypass capacitors at the input and output. While this acts as a bulk capacitor for the MCP6022's supply, the datasheet for this op-amp additionally recommends a small $0.1\mu F$ bypass capacitor within 2mm of the supply pin, so this additional capacitor is included.

LTspice Validation

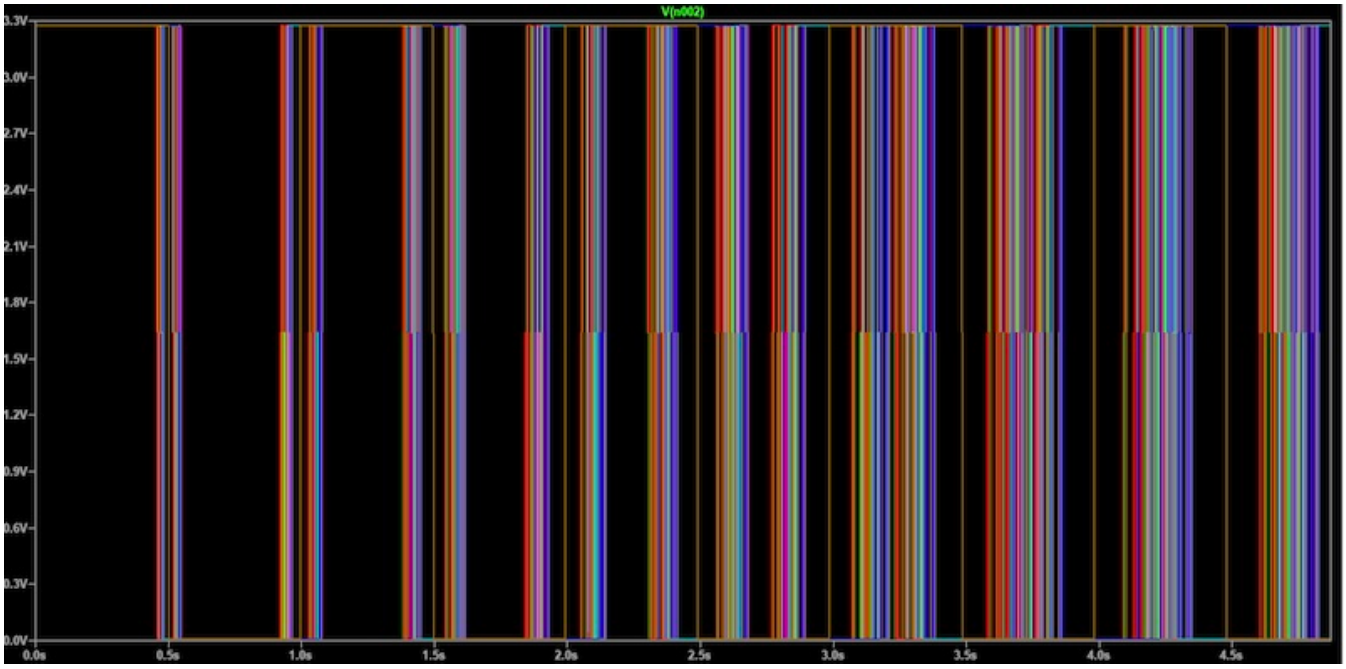
This circuit was then validated to LTspice to ensure the design constraints related to duty cycle would be met. The LTspice schematic used is shown below:



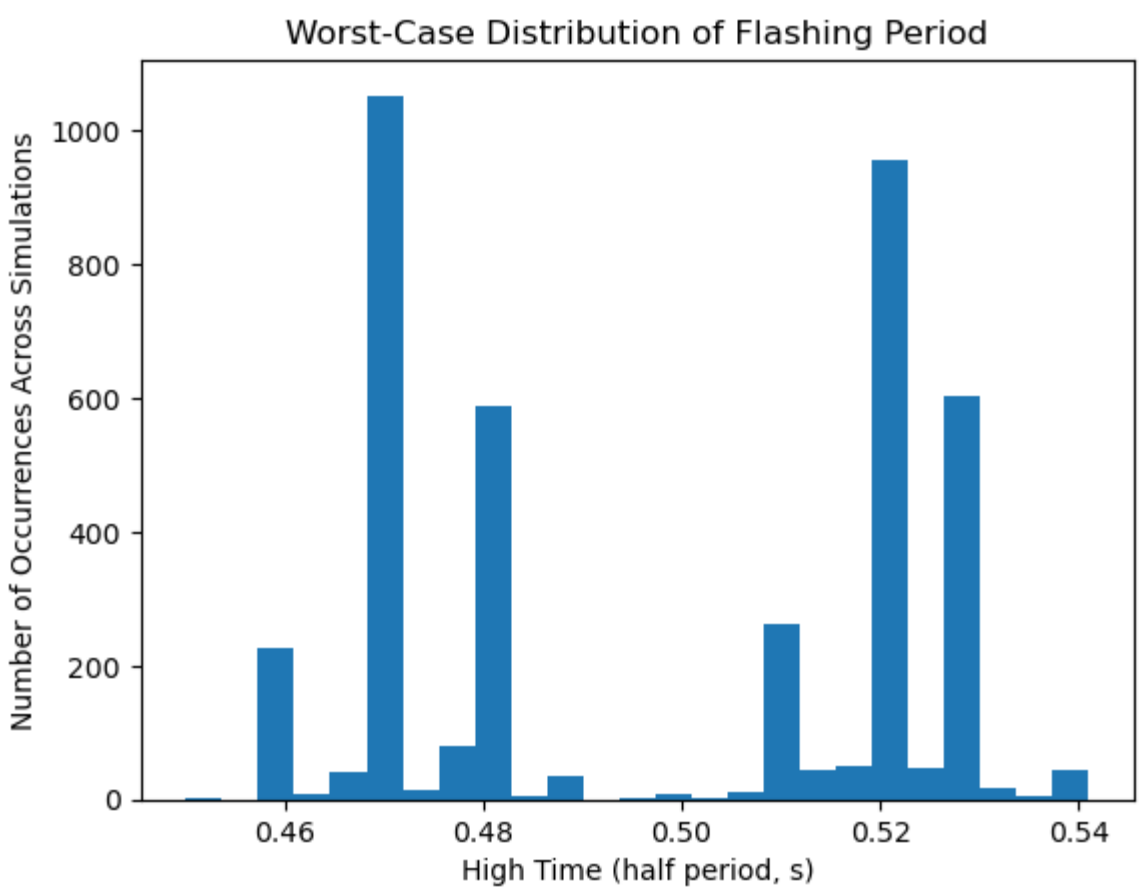
While I originally tried using a SPICE model of the MCP6022 from Microchip, this model was designed for pSPICE and did not play nicely with LTspice. As such, I used ideal op-amps from the ENGR2420 LTspice library. All other components were standard from the LTspice built-in library.

This simulation uses the `wc` and `binary` functions provided in the article LTspice: Worst-Case Circuit Analysis with Minimal Simulations Runs by Analog Devices. This code runs $2^N + 1$ simulation runs to test all possible combinations of worse-case tolerances (positive and negative) of all components. A 5-second transient response is recorded for each run. The first ~1.25 seconds of each response is clipped to remove the slightly longer charge-up time of the initial charge cycle.

The simulation output, although not entirely useful as a graph, is shown below:



A histogram of the resulting simulation outputs is shown below:



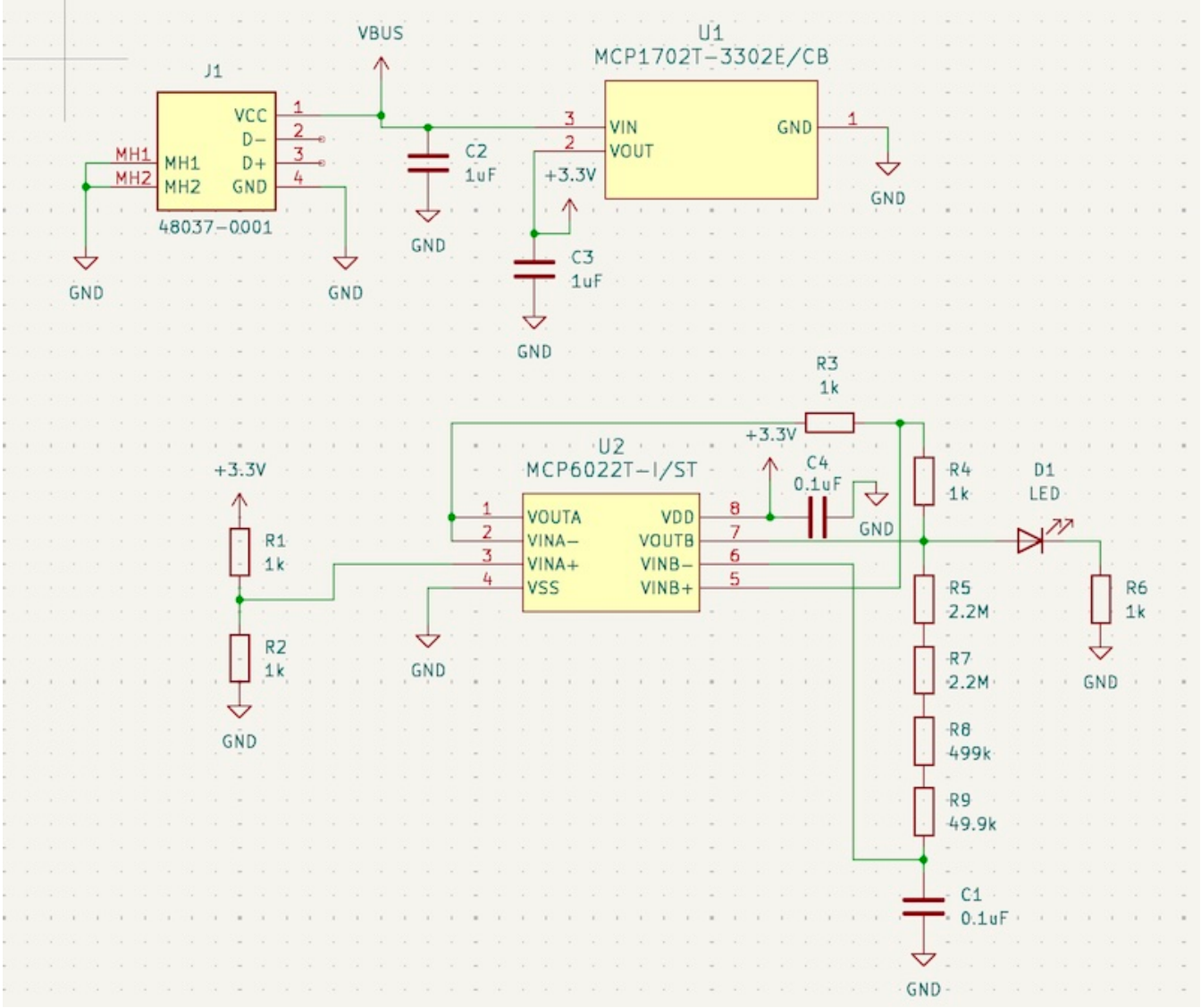
Looking at the raw data (included as mp1_out_1.csv in this repo), the minimum high time (ie, half the period) is 0.450 seconds and maximum is 0.541 seconds. Although the minimum is bordering on the edge, this shows that the $\pm 10\%$ duty cycle tolerance is met.

Potential limitations of this test include omitting the linear regulator and bypass capacitors, although this shouldn't have a large impact on the final duty cycle, and are therefore fairly safe to abstract.

KiCAD Schematic

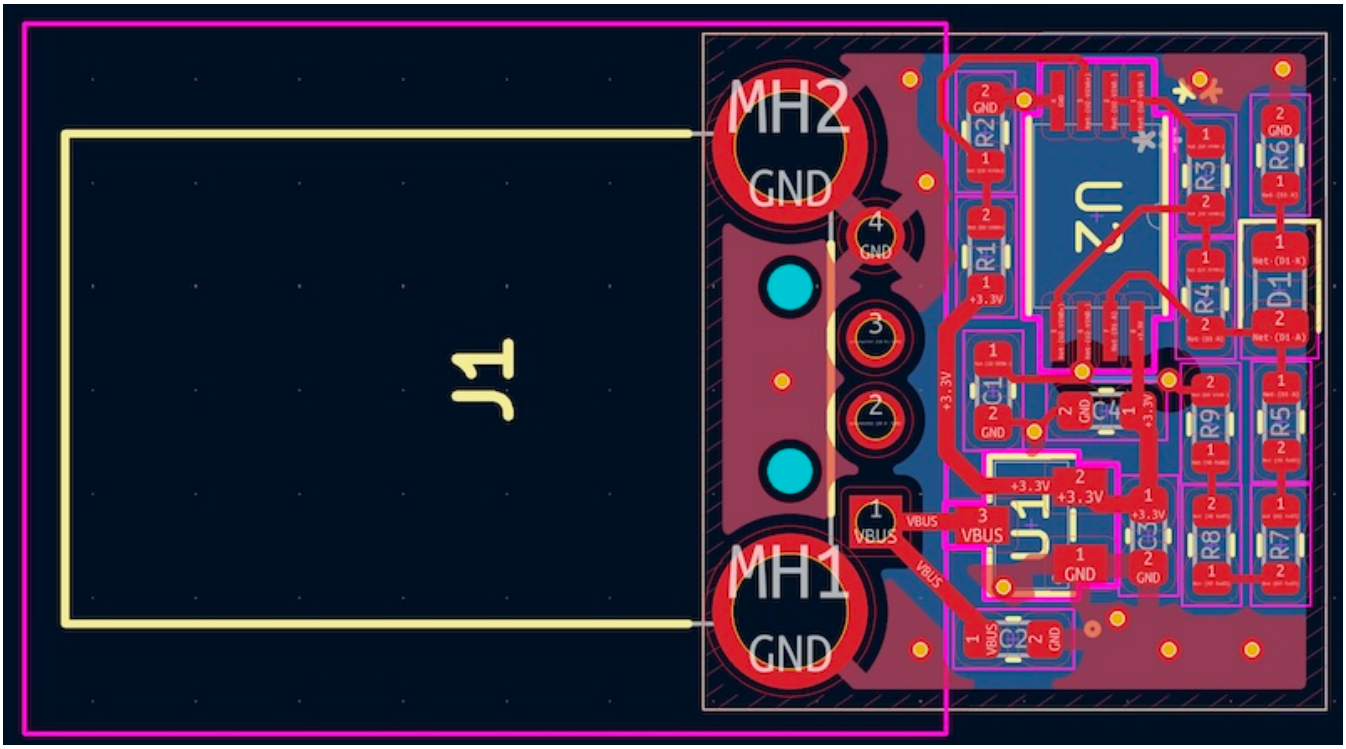
The two ICs' circuit symbols were both downloaded from EE Concierge. The USB connector was downloaded from Ultra Librarian. Both are included in the repo in the `symbols` and `footprints` folder. All other components are from the KiCAD standard library with custom manufacture and Digikey part numbers added.

The KiCAD schematic is included below, as described in the various sections above:



KiCAD PCB Layout

The KiCAD layout is included below:



This layout was chosen to minimize the physical footprint, although this was partially limited by the size of the USB-A port. All components use a 0603 package as specified in the allowable components. Both top and bottom layers are filled ground planes around the signal traces with stitching vias spread around the board. A single 3.3V supply is routed on the bottom layer.

The signal traces are slightly wider than the typical 6 mils at 7.9 mills, or 0.2mm, as this is the KiCAD default. Power traces are 17 mils to approach the 20 mil typical width but still fit within the required locations.

To minimize the footprint, some routing compromises were made. These compromises mainly include routing reference and feedback traces under the op-amp IC footprint. There is also one feedback signal connection routed under a power trace. However, other signal lines are separated as much as possible to given the small board size.

Bill of Materials

The bill of materials is included in the repo as `blinker_bom.csv`