University of Colorado Boulder
ECEE Department

ECEN 2270 - Electronics Design Lab - Spring 2024
Location: Engineering Center, ECEE 281, T,TH, 3:30 - 5:20 PM
Instructor: Steven Dunbar
Lab Title: Lab 1: Circuit Diagrams, Simulation, Prototyping and Circuit Testing
Date of Experiment: February 18th, 2024
Name: Connor Sorrell

Introduction and Objectives

Introduction:

In this lab, we explore the realm of electronic circuits with a focus on generating and analyzing sine, square, and pulse waves, alongside measurement of voltage, current, and resistance using tools like the oscilloscope. Through the integration of Arduino PWM signals, we dive into digital-analog interaction, while also constructing prototype circuits on breadboards, becoming proficient in circuit-building techniques which we will use all semester long. Using SIMetrix, we simulate OpAmp filter circuits, connecting our mathematical analysis with hands-on practice in order to build a strong understanding of circuit behavior, not only using software but on real breadboards as well. With the goal of mastering simulation software, scope probes, and hardware implementation, this lab is key in helping to lay the groundwork for the future.

Objectives:

Throughout the lab, there are some key objectives that are accomplished.

- A sine, square, and pulse wave can be generated and captured.
- Voltage, current, and resistance are measured using the oscilloscope and mathematical analysis.
- An Arduino PWM signal is set up and analyzed.
- A prototype circuit is built on the breadboard and displays proper circuit-building techniques.
- A SIMetrix schematic for an OpAmp filter circuit is built and operational.
- The schematic for all circuits can be simulated using waveforms, as well as probed to achieve two waveforms.
- The circuit prototype built on the breadboard can be powered from an 8V source and is fully functional both using scopes and simulation.
- A strong understanding of SIMetrix and the hardware of the circuits is built.

Experiment 1A

Experiment 1.A.2: Sine Wave Generation and Capture

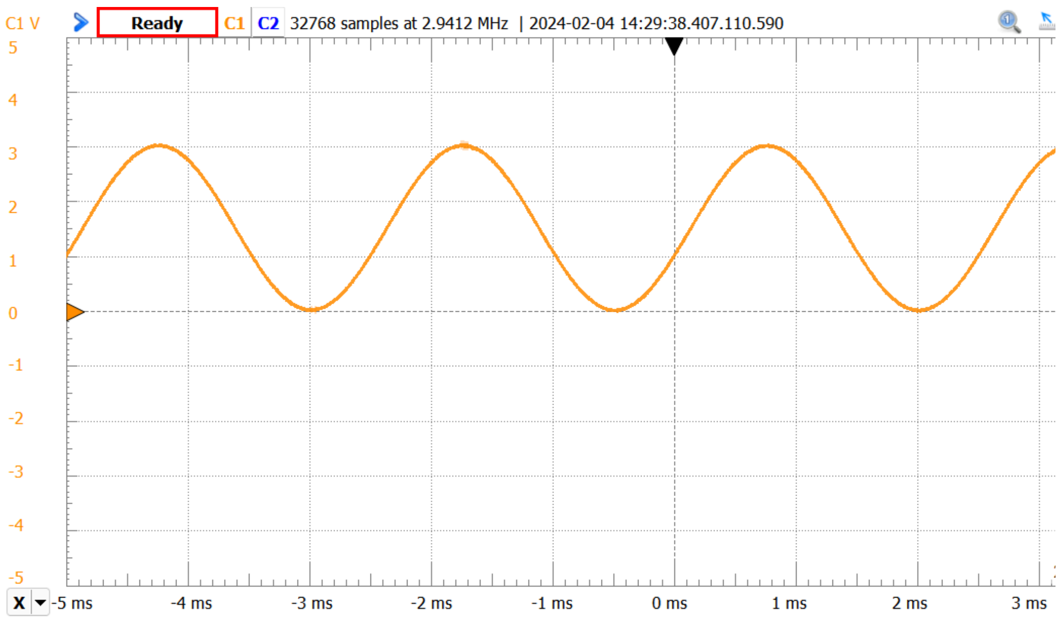


Figure: Sine wave

Maximum voltage: 3V

Minimum voltage: 0V

Amplitude: 1.5V

Peak-to-peak voltage: 3V

AC RMS voltage: 1.06

DC RMS voltage: 1.84
Computed with the formula:

$$V_{RMS} = \sqrt{\frac{1}{T}\int_0^T f(t)^2 dt}$$

$$DC\ V_{RMS} = \sqrt{\frac{1}{.0025}\int_{0}^{.0025}(1.5sin(2\pi * 400t) + 1.5)^2 dt}$$

$$V_{RMS} = 1.84\ V$$

This value is greater than the AC RMS because the AC RMS accounts for negative voltages, the equation has no offset:

$$AC\ V_{RMS} = \sqrt{\frac{1}{.0025}\int_{0}^{.0025}(1.5sin(2\pi * 400t))^2 dt}$$

Average voltage: 1.51 V

Period of the waveform: 2.5 ms

Experiment 1.A.3: Square Wave Generation and Capture

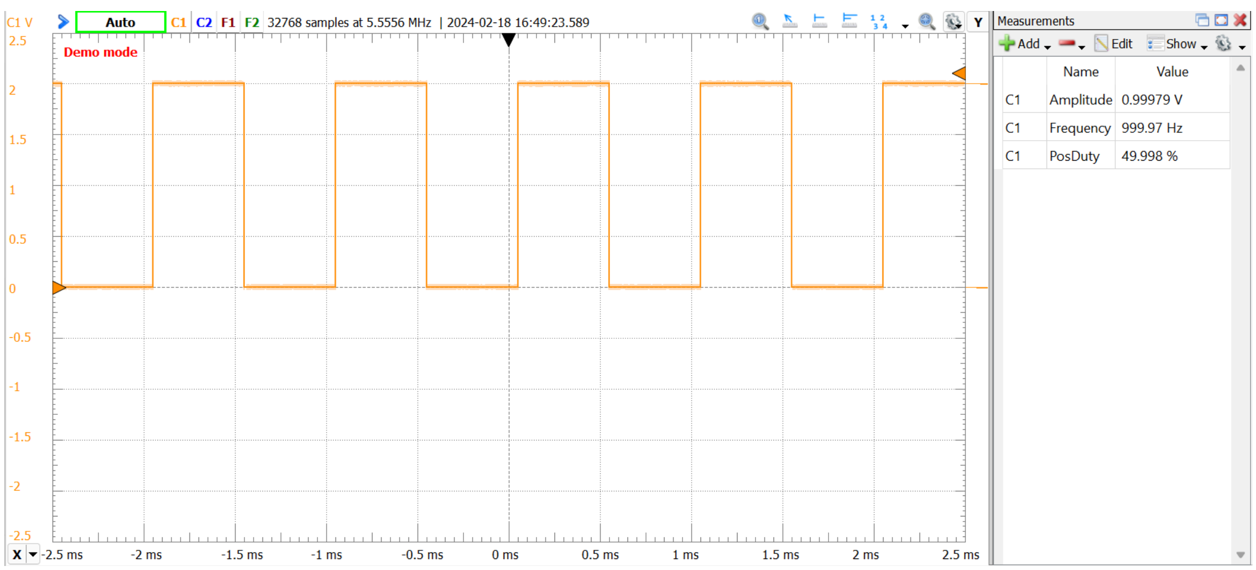


Figure: Square Wave

Amplitude: 0.99 V

Frequency: 1 kHz

PosDuty: 50%

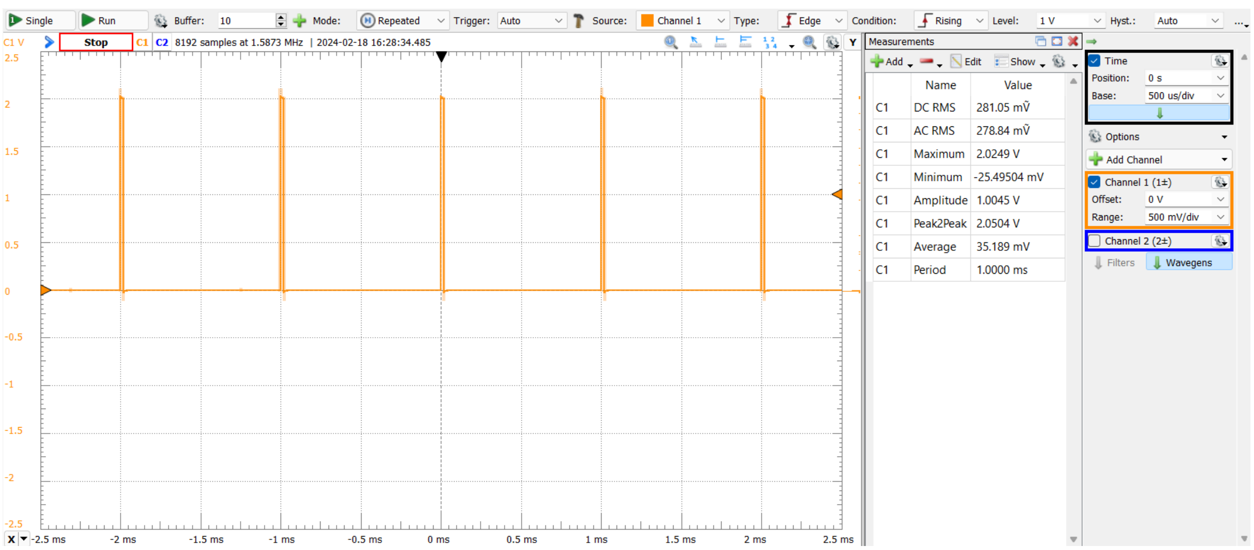
- Describe the waveform in words:
    - The waveform is a square wave that has a 50% duty cycle, meaning it is at its peak voltage (1V) for 50% of the period, and at zero voltage for the other 50%.

Experiment 1.A.4: Periodic Pulsing Waveform

To achieve this waveform we use the following parameters:

Type:	Pulse
Frequency:	10 kHz
Period:	100 us
Amplitude:	2 V
Offset:	0 V
Symmetry:	20 %
Phase:	0 °

<u>Scope:</u>



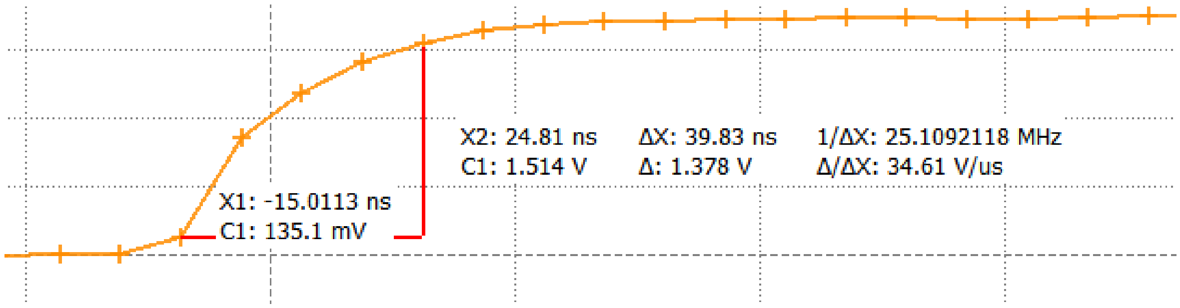


Figure: Zoomed in picture showing the Rising Edge. Rise time = 40 ns.

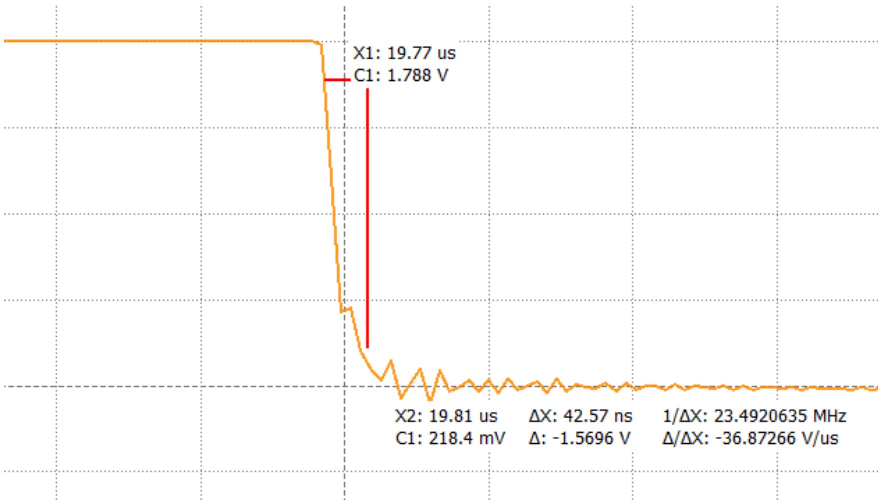
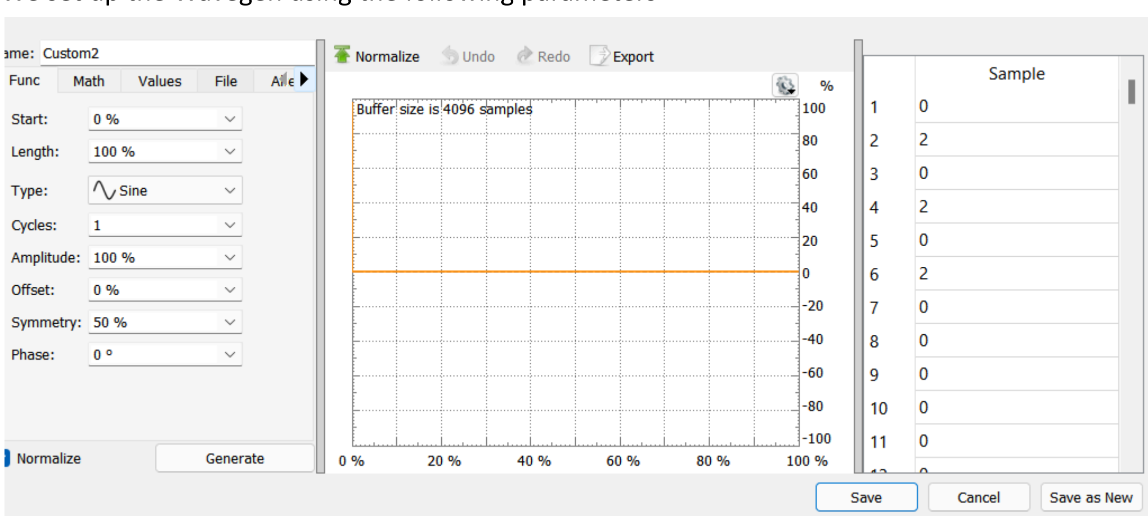


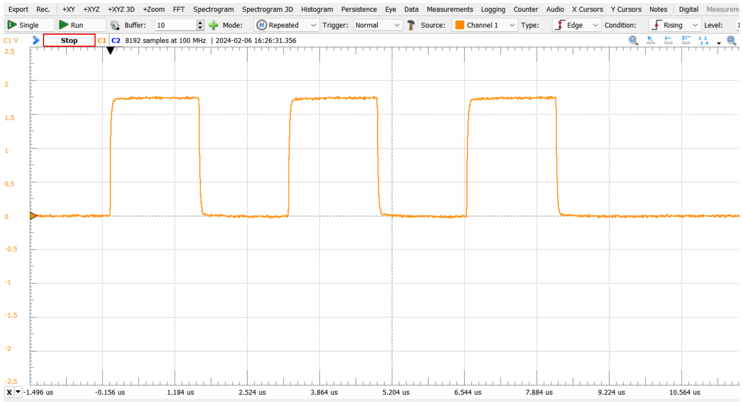
Figure: Zoom in on the Falling Edge. Fall time = 42ns.

Experiment 1.A.5: Burst of Pulses (ms)

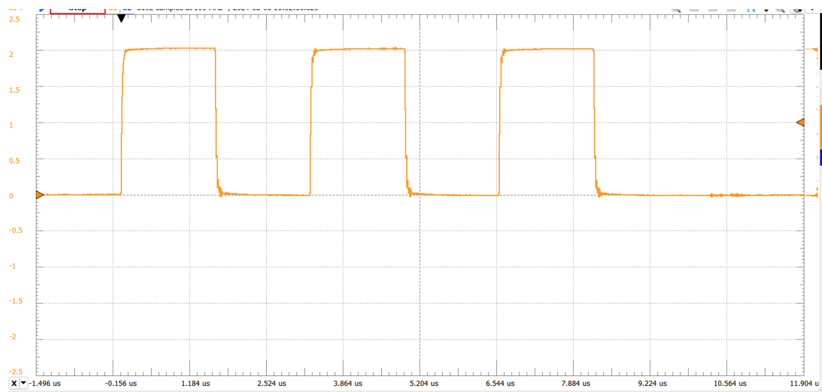
We set up the Wavegen using the following parameters



Flywire:



Scope Probe:

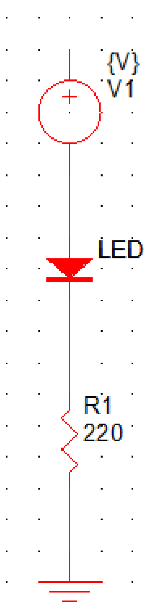


The Flywire has an attenuation on the signal while the scope probe has no attenuation.

Rise time and fall time in the scope are the same using either probe.

The resulting waveform is 3 pulses with magnitude of 2V, a period 2.5 us, and a duty cycle of 50%.

Experiment 1.A.6: Measure Voltage and Current



1) Step the positive supply voltage (V+) in increments of 0.5 V from 0.5 to 5.0 V.

2) Measure the voltages on Channel 1 (VR1) and Channel 2 (Vcc=V+) for each step of V+.

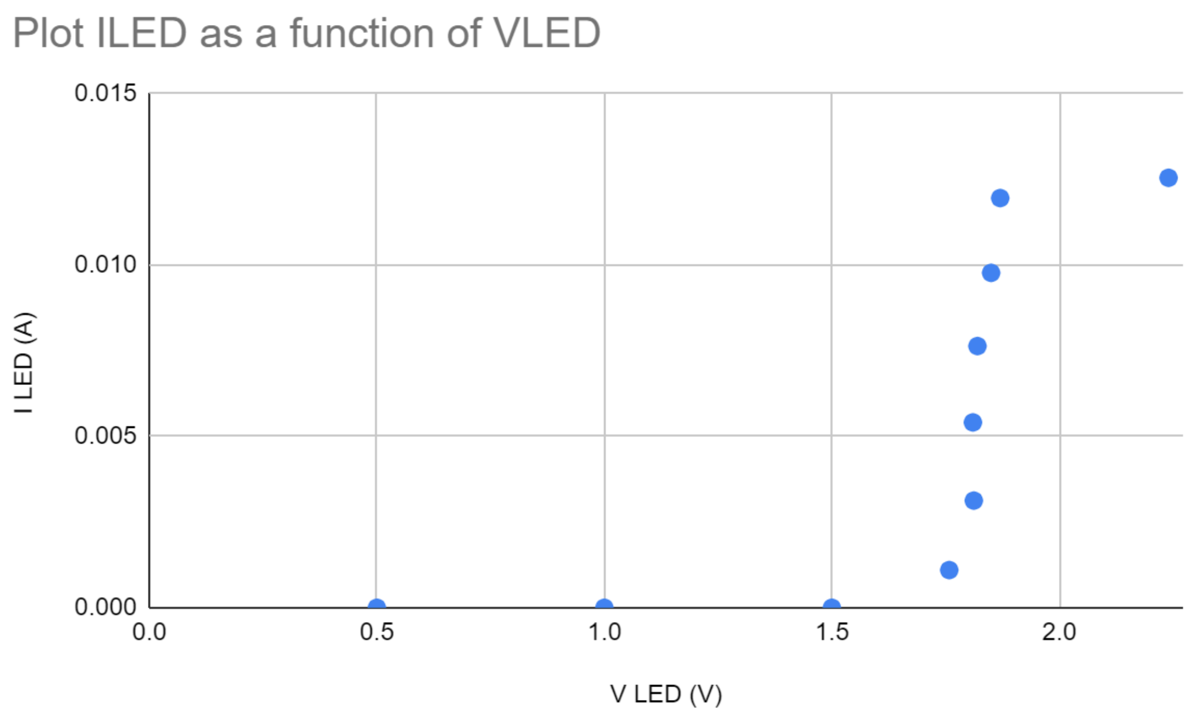
Step Voltage	0.5	1	1.5	2	2.5	3	3.5	4	4.5	5
Voltage of R1	0	0	0	0.242	0.688	1.19	1.68	2.15	2.63	2.76
Total voltage (V+)	0.5	1	1.5	2	2.5	3	3.5	4	4.5	5

3) From V+ and VR1, determine the voltage (VLED) across and the current (ILED) through the LED

Vled (V)	0.5	1	1.5	1.758	1.812	1.81	1.82	1.85	1.87	2.24

We are using a 220 ohm resistor therefore current of the LED = Vled / 220

Iled (A)	0	0	0	1.1 m	3.13m	5.41m	7.64m	9.77m	12.0m	12.5m



- If you wanted to measure the LED IV characteristics with the AD2 up to Iled = 30mA, what would you have to change? Discuss in your lab report
    - The Current through the LED is a function of the V+ supplied to the element and the Resistance the element has. In our case we will keep the same V+ and change our resistance
    - We know V=IR ; We know max Vled = 2.76 , we want I = 0.03 A
      R = 2.76/0.03 = 90 Ohms
      To measure the IV characteristics up to 30 mA. We should replace our 220 ohm resistor with a 90 ohm resistor.

Experiment 1.A.7: Measure Resistance

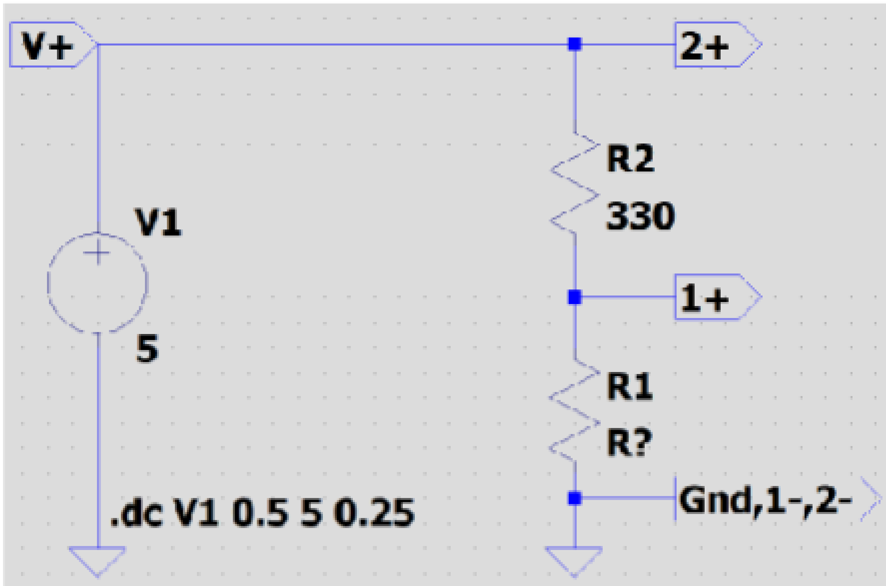


Figure: 1.A.7 Circuit Schematic

Determine the value of R1 such that the voltage across it is close to 5mV when V1=0.5V:

$$.005 = .5\left(\frac{R_1}{R_1+330}\right)$$

$$R_1 = 3.33\ \Omega$$

Computational Equations:

$$V_{R2} = V_{CC} - V_{R1}$$

$$I_{R2} = \frac{V_{R2}}{330\ \Omega}$$

	0.5V	1.0V	1.5V	2.0V	2.5V	3.0V	3.5V	4.0V	4.5V	5.0V
V_{R1} [V]	0.005	0.010	0.015	0.019	0.024	0.029	0.034	0.039	0.044	0.049
V_{R2}[V]	0.495	0.990	1.485	1.981	2.476	2.971	3.466	3.961	4.456	4.951
I_{R2} [mA]	1.5	3.0	4.5	6.0	7.5	9.0	10.5	12.0	13.5	15.0

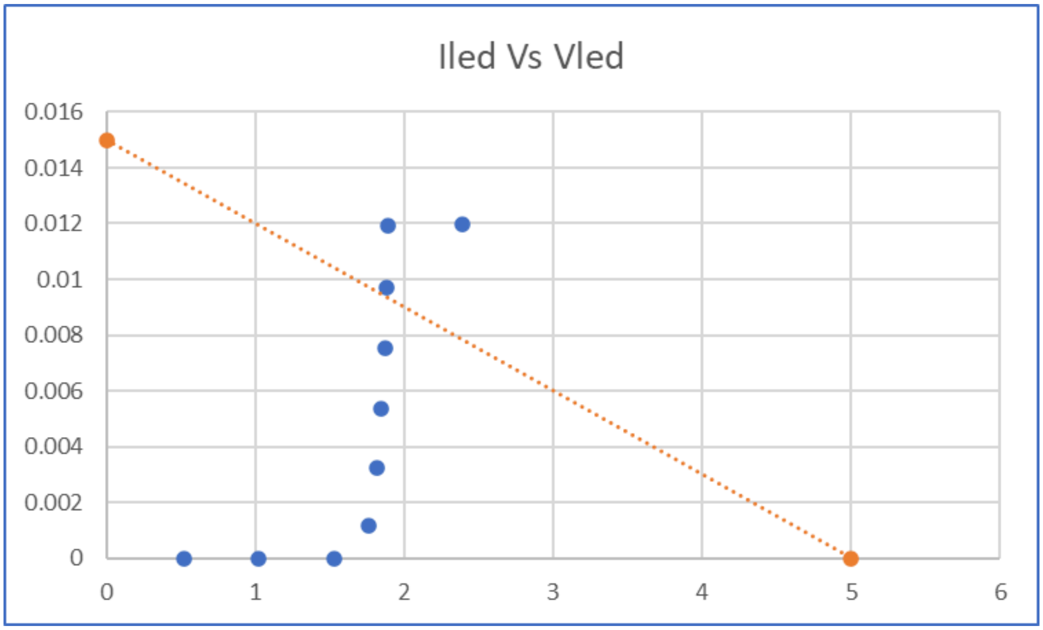


Figure: Current through LED (A) Vs. Voltage over LED (V)

The operating point we found was ~1.8 V at ~10mA. This operating point was verified by connecting an LED in series with the R2 resistor and placing the supply voltage to 5V. With this setup, the LED turned on, which confirms that 5V is higher than the operating point voltage. This verifies our operating point voltage as being lower than the five-volt input, so the 1.8V is reasonable.

A small discrepancy between the calculated operating point and the actual LED is that the LED began to light up at voltages slightly under the operating point voltage. During our tests, sometimes the bulb lit up at voltages from 1.6V-1.7V, which is under our operating point voltage, yet close. This observation reveals that the load line method is effective because it is simple, yet it includes some errors.

Experiment 1.A.8: RC Circuit Characterization

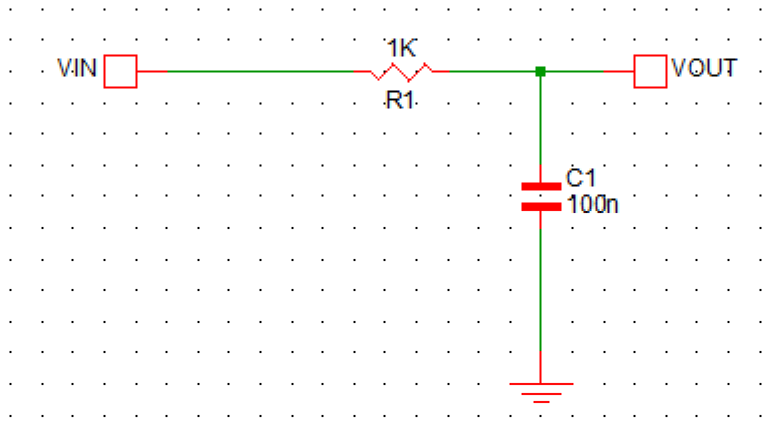


Figure: RC Circuit, with values of R = 1k Ohms and C = 100nF.

We then built the circuit and used the BNC adapter and two scope probes to measure Vin and Vout. Using the Wavegen, we then generated a sine wave with amplitude 5V and offset 0V. We then measured the Vin rms and Vout rms voltages for 20 data points in the range from 100 Hz to 100 kHz on a log scale.

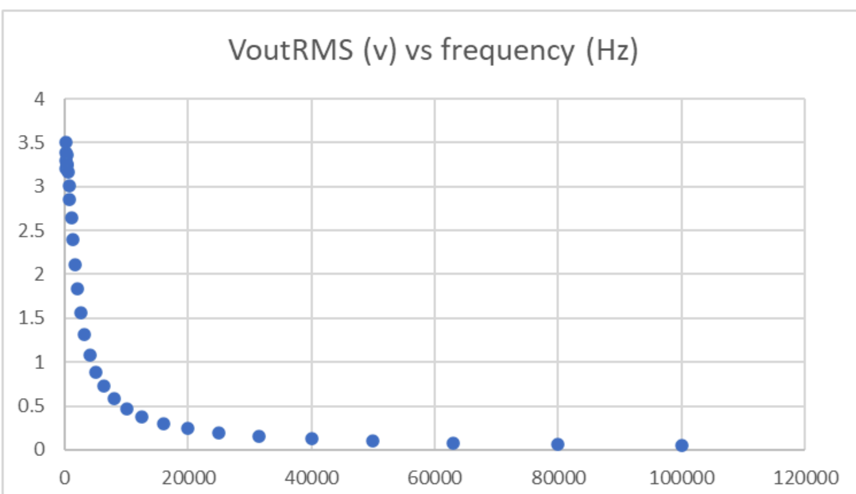


Figure: Voltage vs frequency plot for the RC Circuit

- Does the value of Vin(f) change? If so, why?
    - The value of Vin rms changes very slightly as the testing frequency increases. At 100 Hz, we had a Vin of 3.443 Volts, whereas at 100kHz, we had a Vin rms of 3.313 V. This is a very slight change in voltage, but it most probably could be attributed due to the changing impedance of circuit at different frequencies.

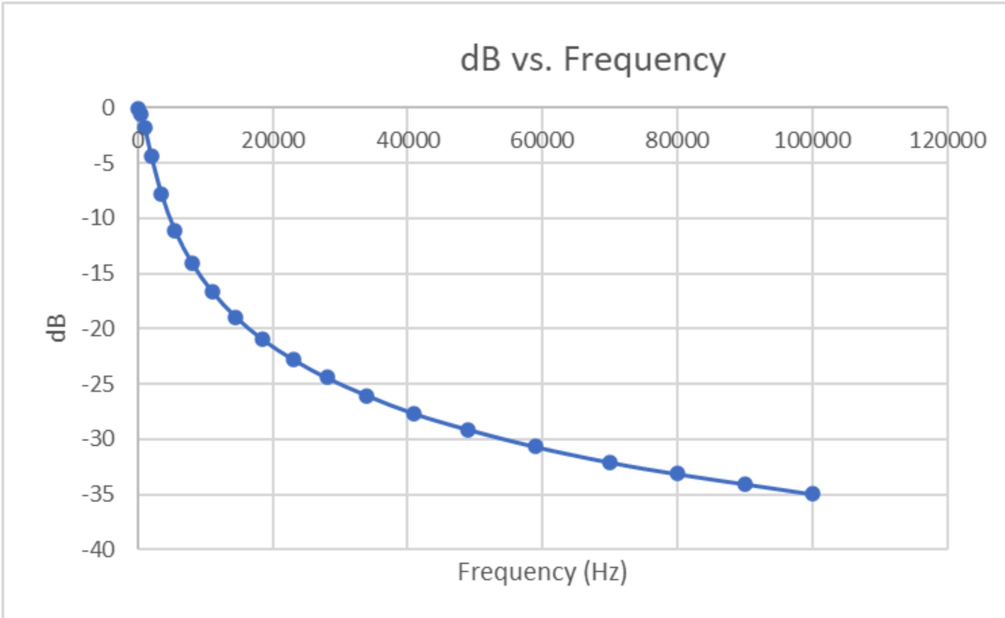


Figure: 20*Gain (Captured with 20 points of data) in dB vs frequency (Hz) plotted on a logarithmic scale

Now, using the same values for R and C, we can use the network analyzer of the AD2 to display the frequency response (using loglog scales) of the same single RC Circuit.

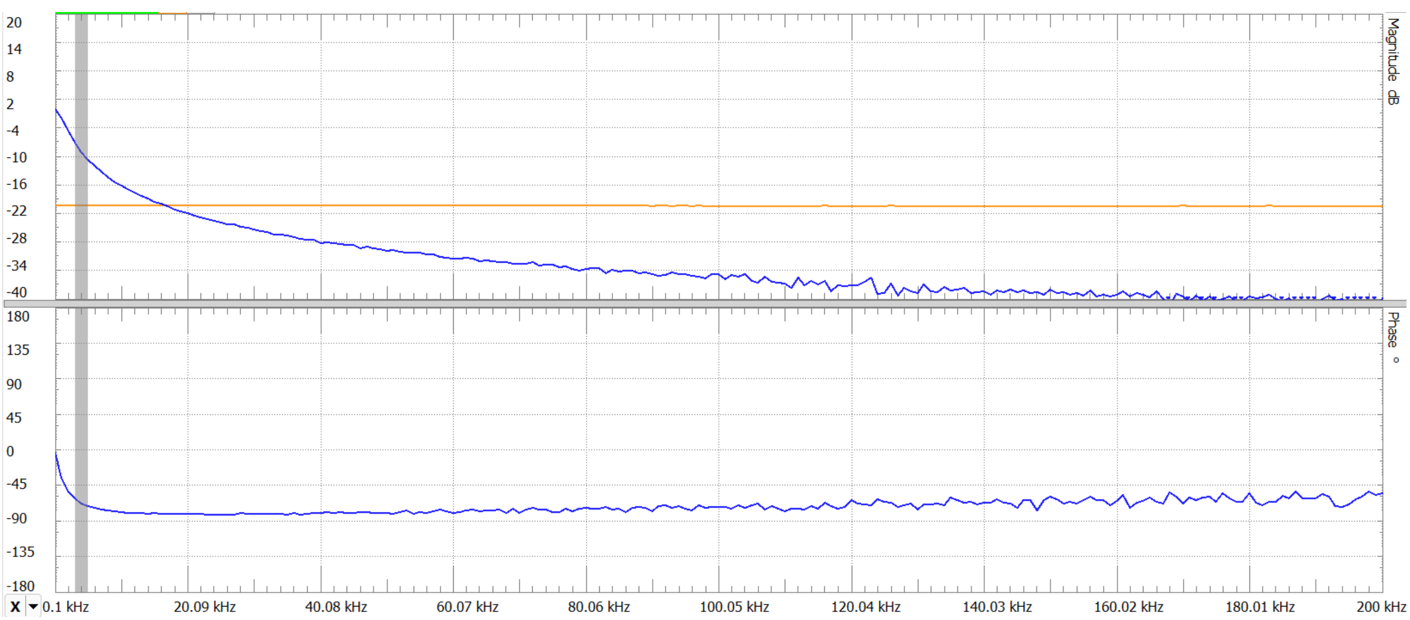


Figure: Network analyzer displaying the frequency response of the RC Circuit

As can be seen, our plots closely match each other. They are not exactly the same, but we are in the realm of what we expect, with a max dc Gain of ~0 dB and both match the characteristics of an RC lowpass filter.

If two copies of the single RC circuit are cascaded as in the schematic shown below, we would expect the performance of the LPF to be twice as good.

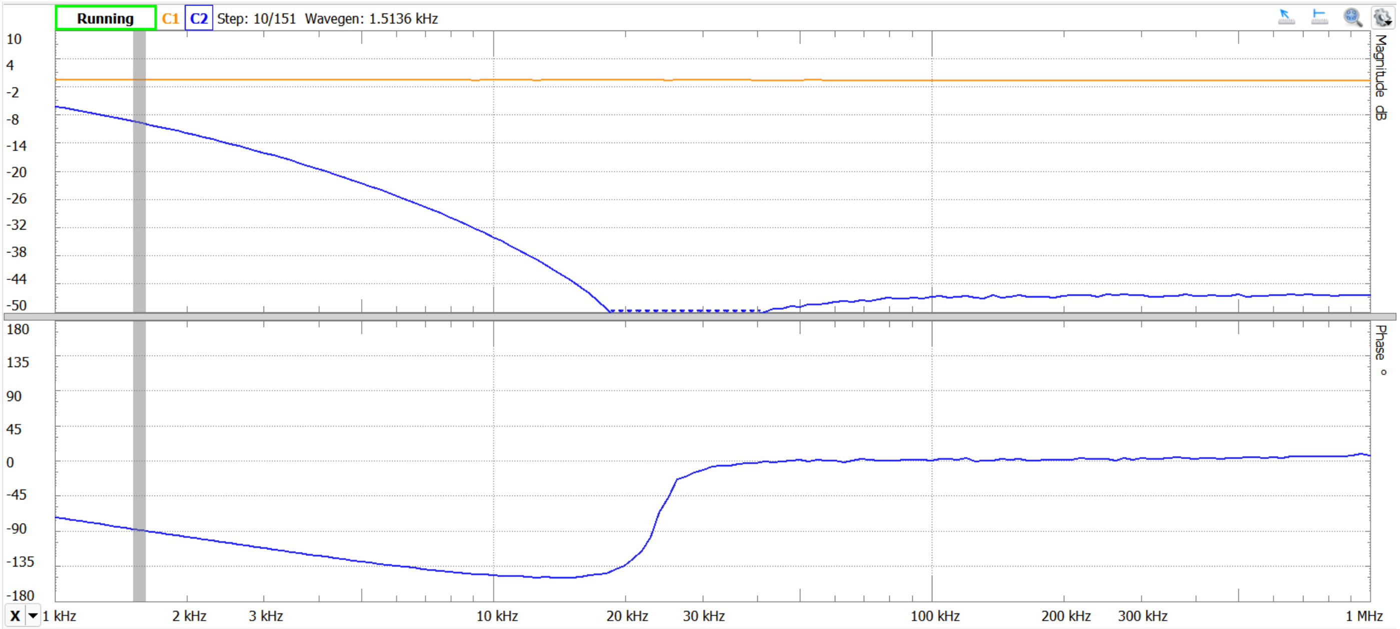


Figure: Network analyzer measurement of the circuit with the same values of R and C.

The circuit is "twice as good", meaning that the high frequencies are filtered out at a much lower frequency, in this case around 20 kHz.
The circuit performs as expected, two copies of the RC circuit simply double the attenuation of the low pass filter, and makes it even more effective in serving its purpose within specific applications.

- Why is this circuit called a low-pass filter?
  - This circuit is a low-pass filter because it allows signals with low frequencies to pass through unaffected, while it simultaneously attenuates (blocks) signals with higher frequencies. As seen from the frequency response curve, as the frequency increases, the output voltage decreases.

Experiment 1.A.9: Arduino PWM Signal

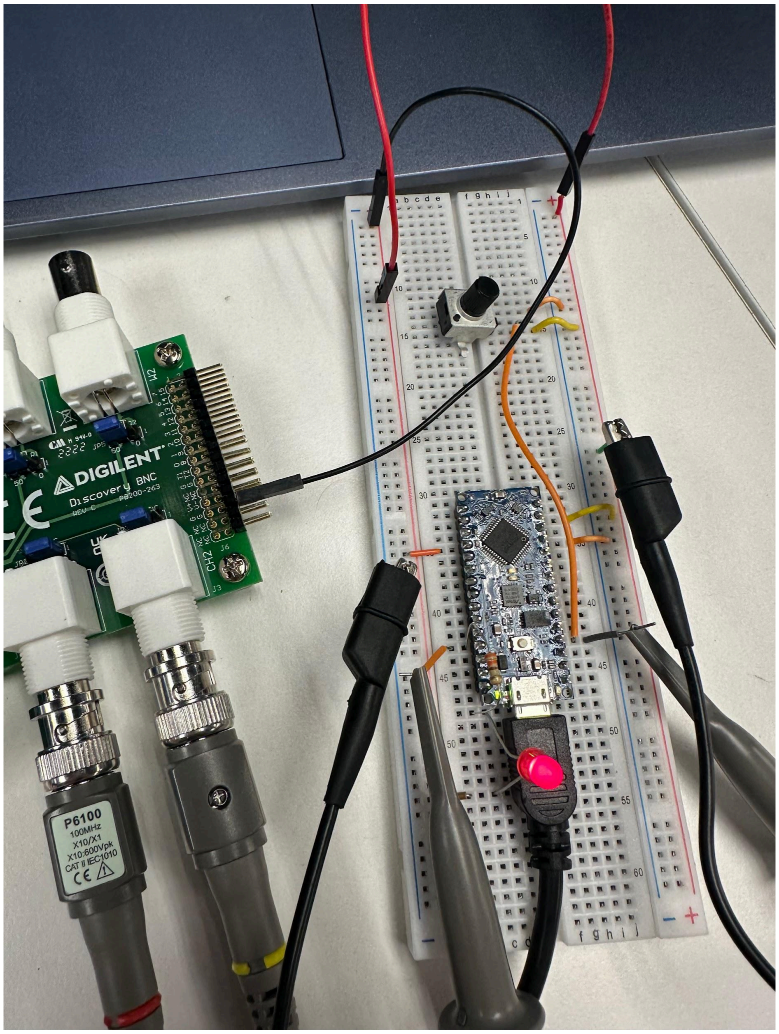


Figure: Shows the breadboard connected to the waveform generator, also in connection with the Arduino Nano Every, which is powering a red LED based on the state of the potentiometer.

We can then measure the characteristics of the PWM signal (at pin D9) in response to the analog input signal (pin A0).

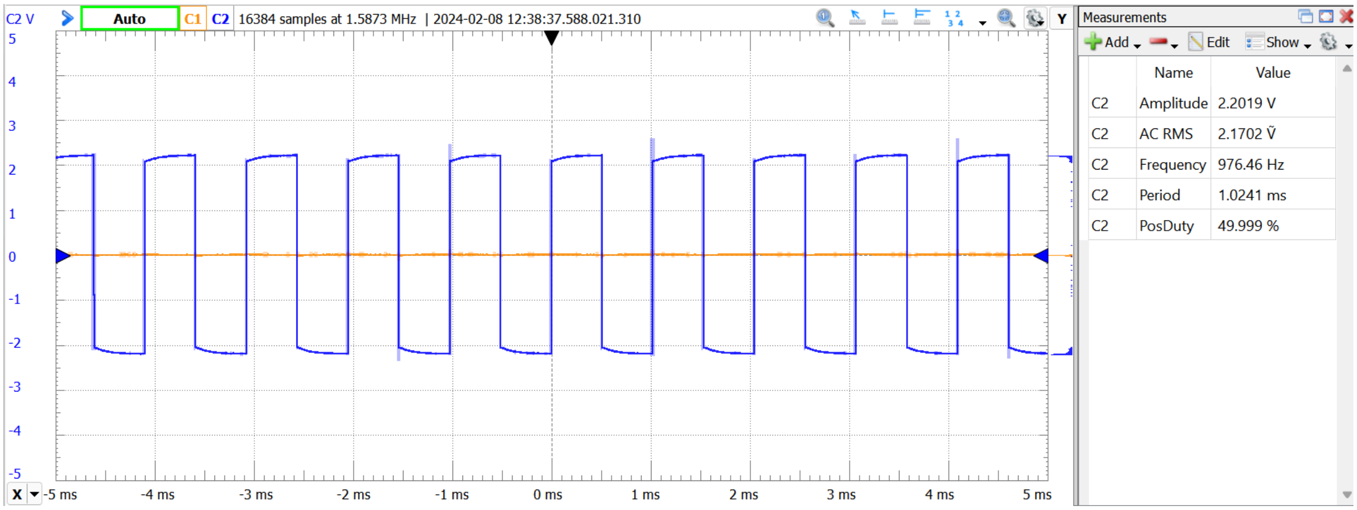


Figure: Shows the input analog signal (orange line) and the PWM signal (blue wave)

With a very small amplitude input signal (close to zero), the resulting PWM signal has a ~2V amplitude, ~1000 Hz frequency, ~1ms Period, and a 50% duty cycle.
This is as expected because essentially the PWM signal just involves producing a series of pulses whose widths are proportional to the original signal's magnitude.

- How could the PWM signal be converted back to an analog signal?
  - To convert the PWM signal back to analog voltage, we can use a RC lowpass filter. This filter would average out the PWM signal in order to produce an analog voltage that will be proportional to the duty cycle of the PWM signal.
  - We could do this by connecting the voltage source in series with a resistor, which is then connected to the PWM signal, and the analog output, which is then in series with a capacitor leading to ground. By implementing this, we can convert PWM to an analog signal which would be more for powering components such as a motor and LED. Adjustments to the components could be made to meet specific requirements and performances.

Experiment 1B

Experiment 1.B.2: Testing SIMetrix Installation

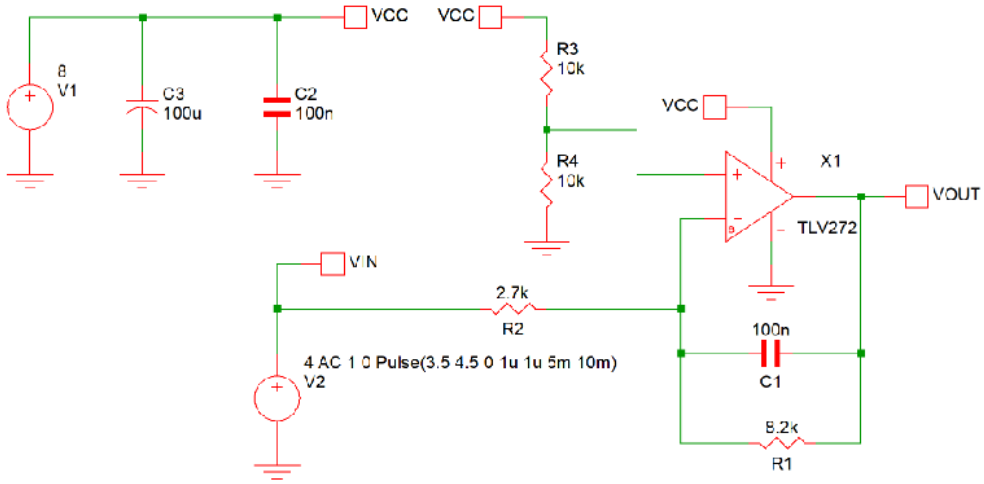


Figure: SIMetrix implementation of the assigned circuit

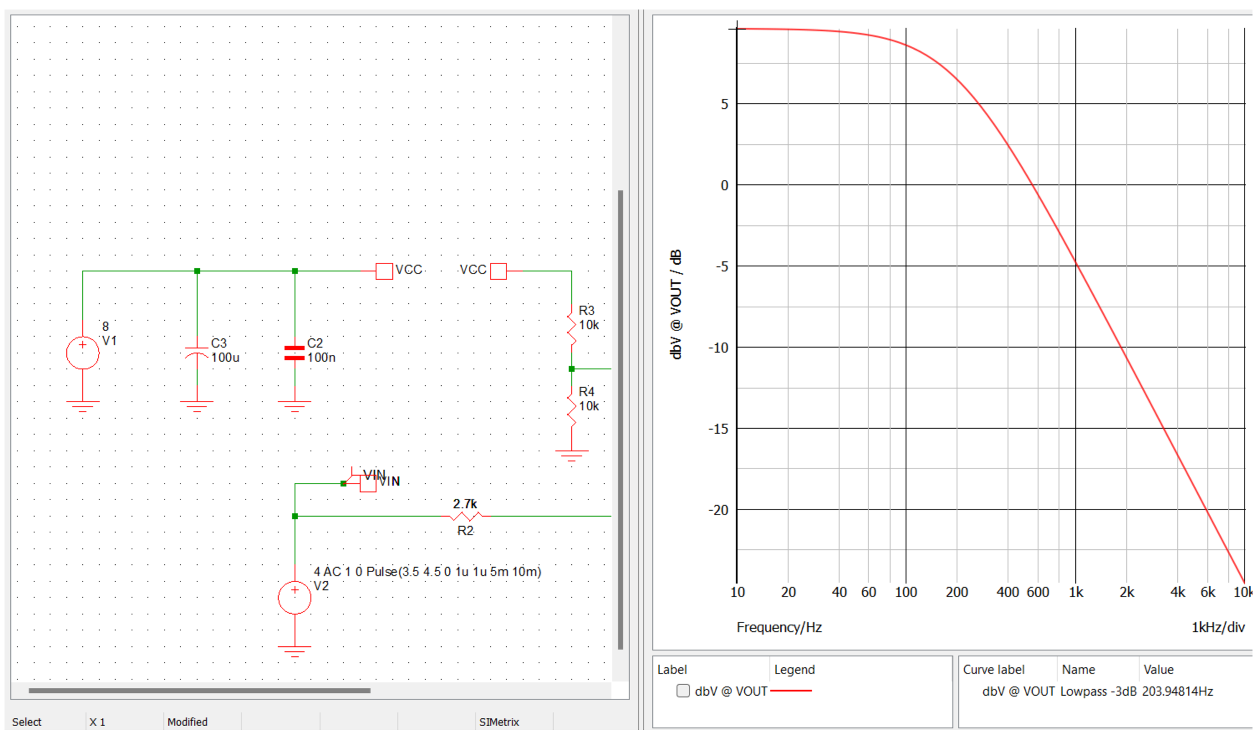


Figure: AC Analysis simulation from 10 Hz to 10 kHz

- The 4 and AC 1 for source V2 are crucial for the correct AC analysis, why?
    - The 4 and AC 1 for source V2 are crucial because the quantity ‘4’ gives us our average voltage (being 4 volts), whereas the ‘1’ is our amplitude, showing that the curve ranges from 3.5 V to 4.5 V.
- Determine what the function of this circuit is. Measure quantities such as dB gain, -3dB frequency, magnitude, change per decade, etc.
    - The function of the circuit is a low pass filter. The -3dB frequency: ~204 Hz, the change per decade: -5dB, magnitude of the dc Gain is 10 dB.
- How do you know your results are correct?
    - We know our results are correct because they are exactly what we expected after solving for our circuit analytically within the prelab. The bode plot looks as it should, the values are reasonable, and the results match up with other experiments we have done.
- What is the function of R3 and R4? Explain.
    - The function of the two resistors between VCC and the positive terminal of the opAmp are there to minimize offset due to the input bias current. We do this in order to assure that the voltage drop due to bias current is the same for both inputs.

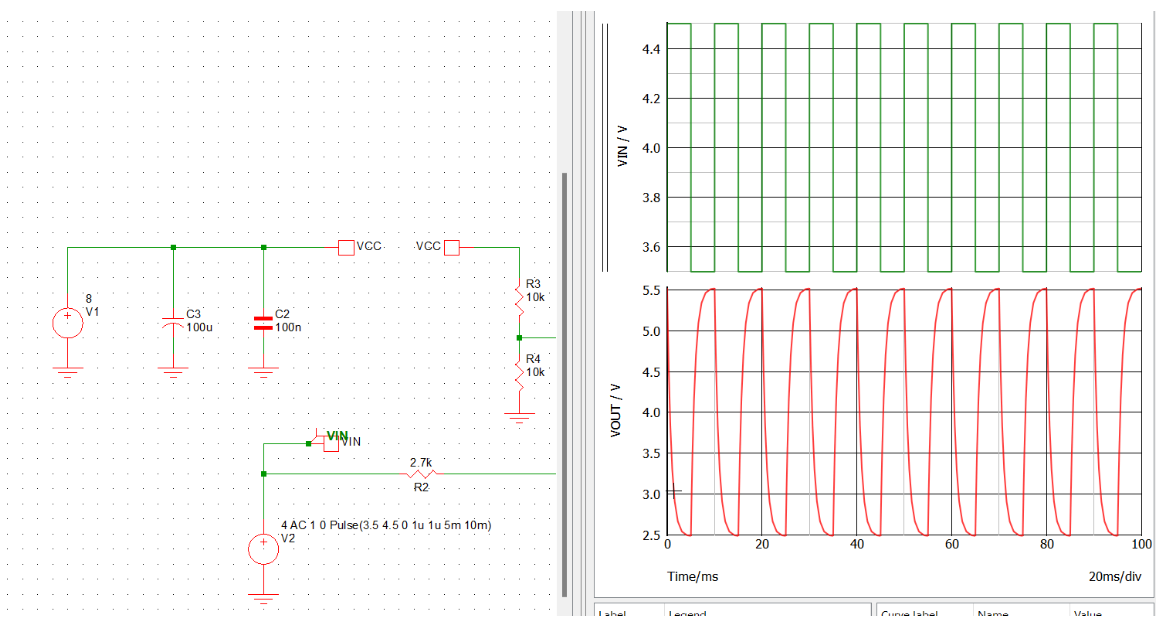


Figure: 100ms transient analysis of the LPF showing both Vin and Vout waveforms

- Describe output waveform in words and measurements
    - The output waveform (red line), is a pulsing wave with magnitude 5.5V, frequency of 100Hz, a rise time of 1.8ms and a fall time of 1.8ms, equating to a 50% duty cycle.

Experiment 1.B.3: Building a SIMetrix Subcircuit

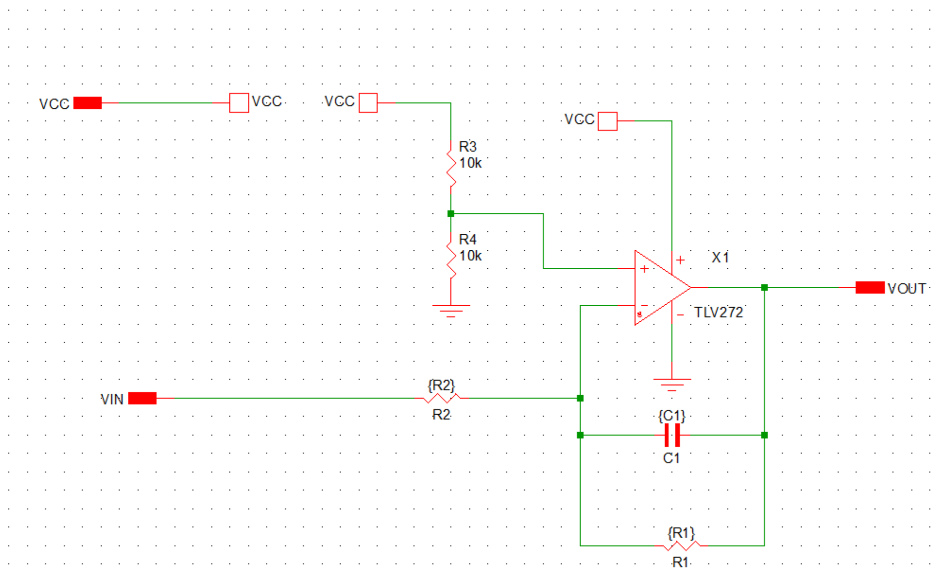


Figure: OpAmp filter subcircuit SIMetrix implementation

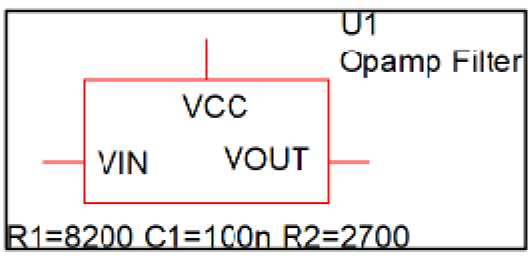


Figure: Black Box Diagram of the OpAmp Subcircuit

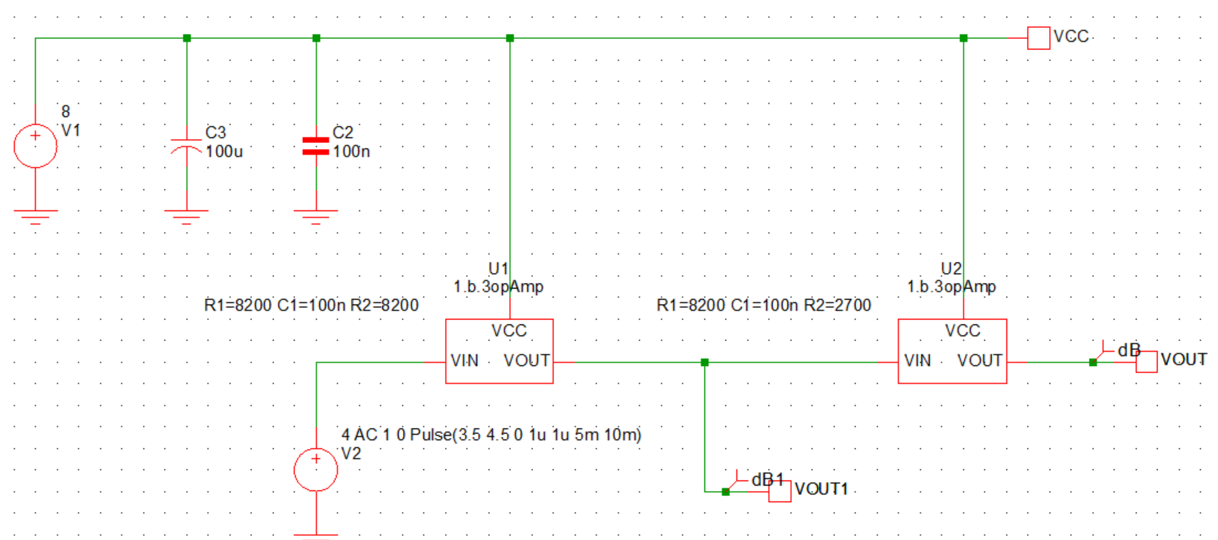


Figure: New circuit in SIMetrix after implementing the subcircuit OpAmp

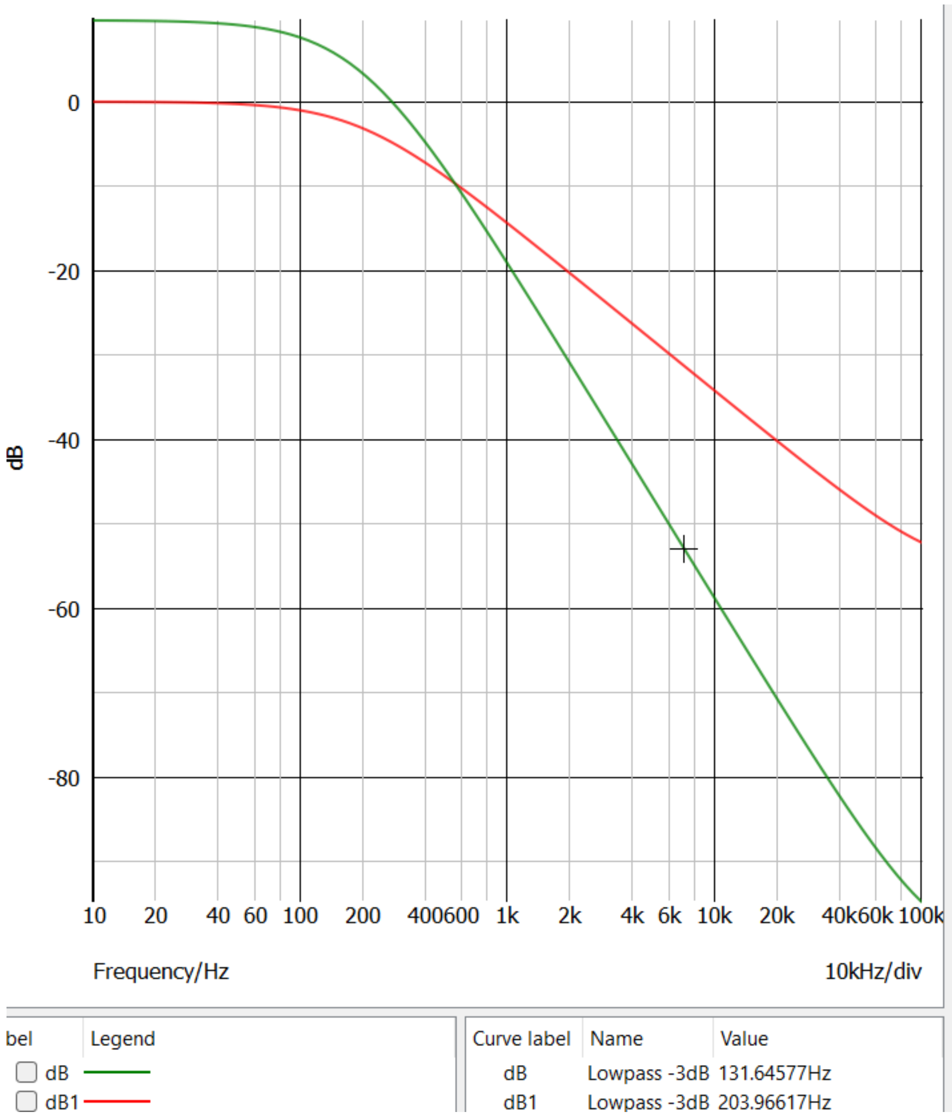


Figure: AC analysis simulation of the new circuit measuring dB in a log scale

- Determine the DC Gains, the -3dB frequencies, and the change in magnitude for VOUT1, VOUT2.
    - For VOUT1, the DC Gain is 0dB, the -3db frequency is 194 Hz, and the change in magnitude is -20dB per decade. For VOUT2, the DC Gain is 10dB, the -3db frequency is 115 Hz, and the change in magnitude is -40dB per decade.
- Why was R2 = 8.2k used for the first block, rather than the default value?
    - This value was used instead of the default value in order to change the gain of the opamp filter to 0dB.
- Describe how you checked that your results are correct
    - We knew our results were correct due to the cascading OpAmp properties of this schematic. First, we verified the VOUT1 plot, verifying its -20dB/dec slope due to it being the first OpAmp in series. Then the gain was verified by solving for the magnitude of this inverting opamp circuit, and finding its expected dc gain to be 0dB. The corner frequency

was verified by calculating the phase equation of the inverting OpAmp, then finding the expected frequency for -3dB. Next, the VOUT2 plot was verified, as the expected slope was -40dB/dec. Then, the gain of the second OpAmp was expected to be 10dB, which our plot verified. Lastly, the corner frequency was expected to shift due to the new cascading OpAmp characteristics, which was shown in our plot. This verified our values as accurate and confirmed our schematic was correct.

- To make the -3dB frequency 10 times higher, what would you change and how?
    - To make the corner frequency ten times higher, we could either lower the capacitance or resistance of the inverting OpAmp (R1, C1 above) by ten times.

Experiment 1.B.4: Enter Circuit from Pre-lab into SIMetrix and Simulate

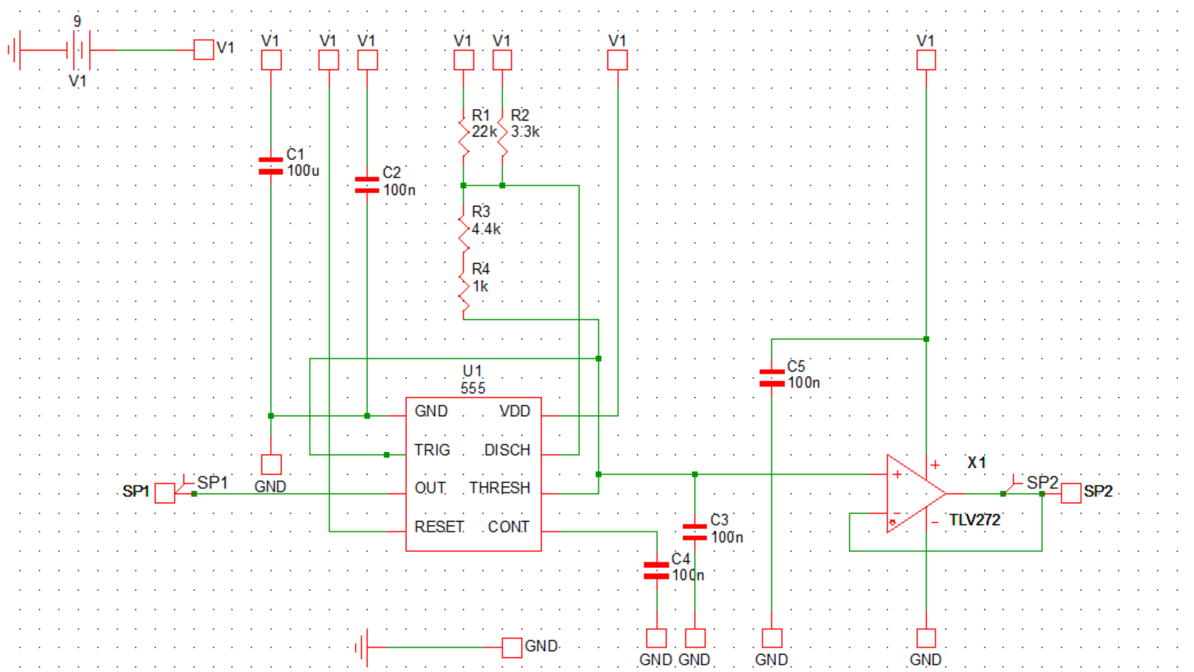


Figure: Pre-lab schematic implementation in SIMetrix

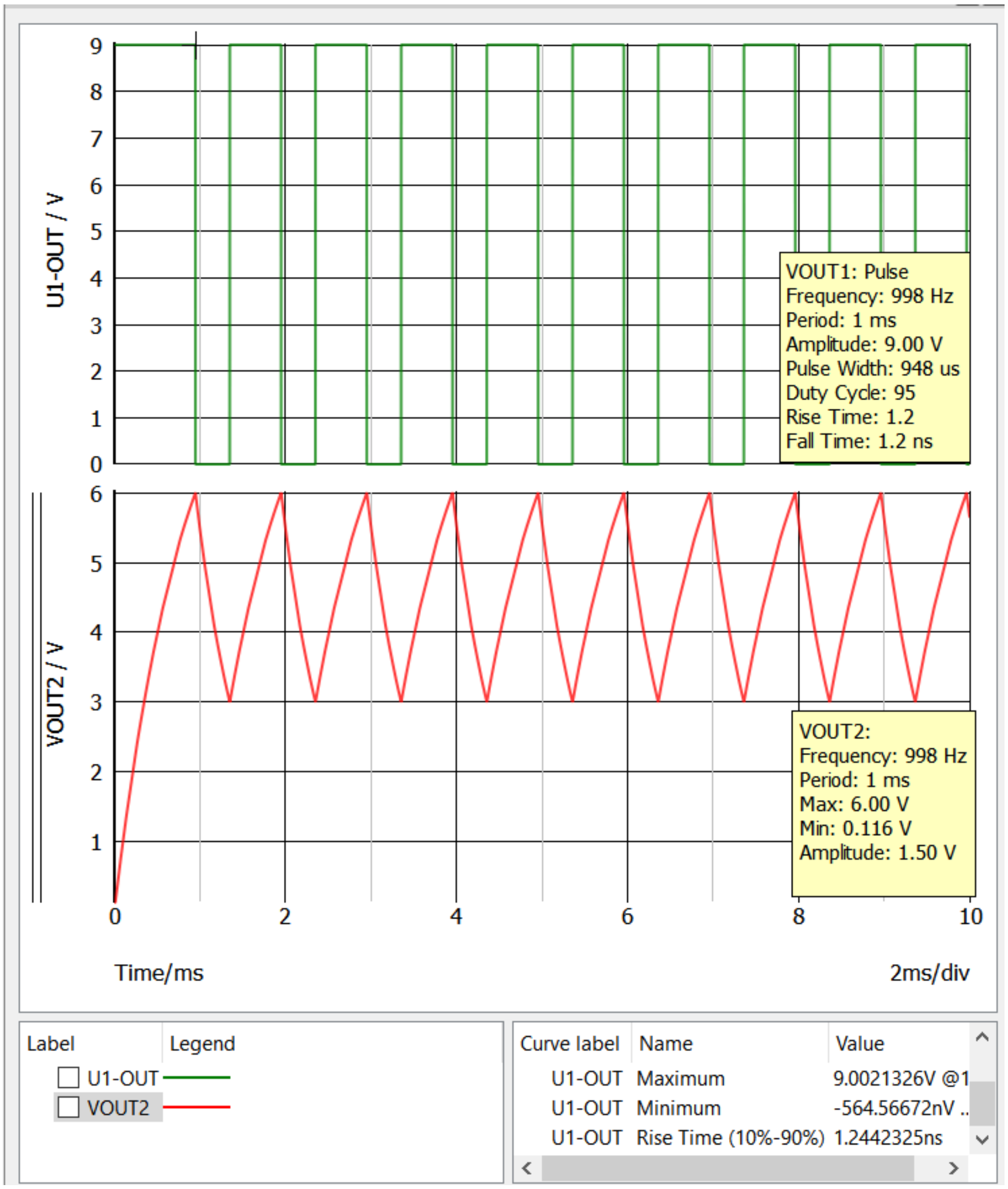


Figure: Shows the transient simulation of the circuit in steady-state operation

As seen from the transient analysis from 0-10ms, the waveform outputted by the timer is a pulsing waveform, whereas the output waveform of the opAmp is a unique waveform. The pulse wave from the timer had a frequency of 998 Hz, a duty cycle of 95%, and a rise/fall time of 1.2 ns.

The OpAmp output waveform is unique, similar to a sawtooth waveform. This waveform has a frequency of 998 Hz, an amplitude of 1.5V, and an offset of 4.5V.

Experiment 1.B.5: Construct Circuit on the Breadboard

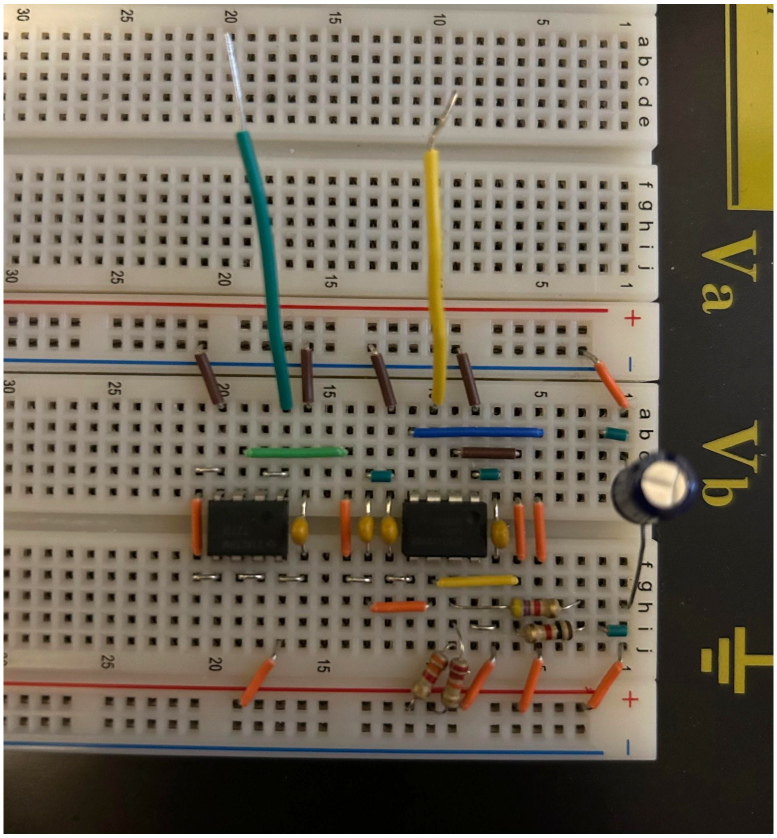


Figure: Shows the circuit from Prelab 1.B built on the breadboard.

Experiment 1.B.6: Test the Circuit using Oscilloscope

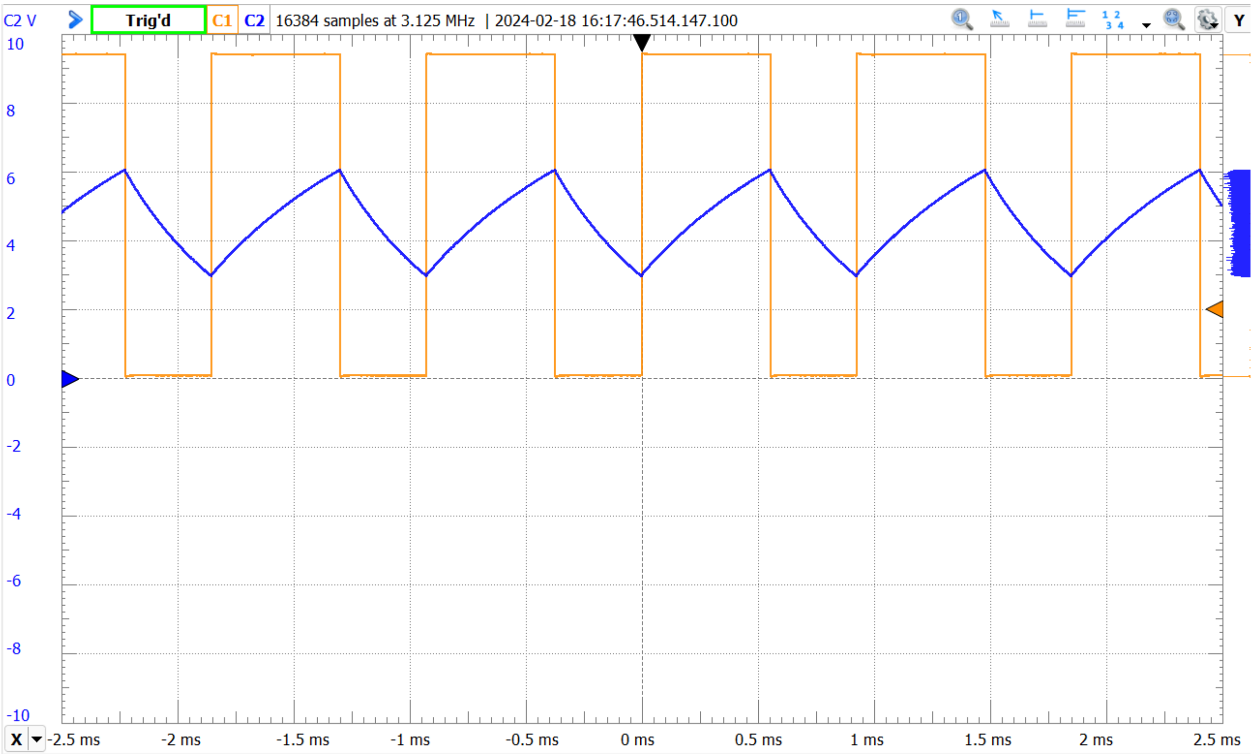


Figure: Shows the outputted waves captured by the AD2 after powering the circuit with a 9V source.

- Find the following features of the waveforms: Max/min value, frequency, amplitude, period, pulse width, duty cycle, rise/fall times.
    - Channel 1 (Orange, Output of 555 Timer):
        - Maximum Voltage: 9.43 V
        - Minimum Voltage: ~0 V
        - Amplitude: 4.66 V
        - Frequency: 1.08 kHz
        - Period: 0.96 ms
        - Pulse Width: 0.55 ms
        - Duty Cycle: 59.8%
        - Rise Time: 430 ns
        - Fall Time: 430 ns
    - Channel 2 (Blue, Output of OpAmp):
        - Maximum Voltage: 6.05 V
        - Minimum Voltage: 2.95 V
        - Amplitude: 1.55 V
        - Frequency: 1.09 kHz
        - Period: 0.92 ms

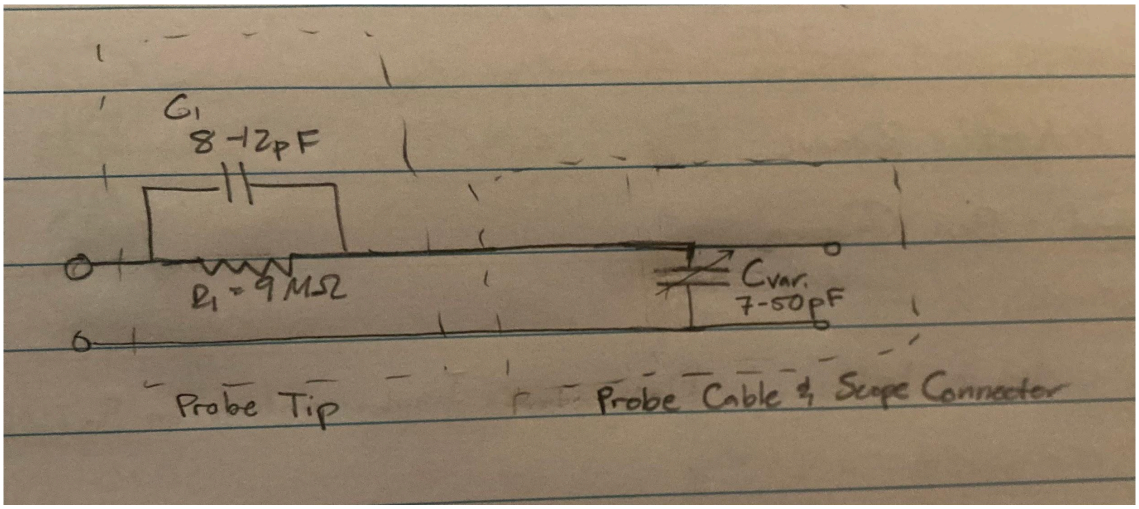
- Compare experimental and simulated results and comment on the differences.
    - The results are very similar to the simulated results and therefore we can confirm that this is correct. A difference is that the real results aren't perfect, the maximums are higher/lower than the simulated results. Other than this, the real results closely mirror the simulated results.
- Describe the process of debugging and verifying the results are correct.
    - The debugging process was quick, it involved: Making sure 10x attenuation was used, adjusting WaveForms to allow the scales and measurements to best depict the results.

Discussion and Conclusions

This lab has been instrumental in strengthening and broadening our understanding of electronic circuits, while also teaching us countless techniques and methods for waveform generation and analysis. Through hands-on experimentation, we improved our skills in making precise measurements using tools such as the physical oscilloscope and the AD2 scope. Then, we dived into practical tasks such as integrating Arduino PWM signals, soldering headers, and constructing prototype circuits on breadboards. We also showed our strong understanding of digital-analog interaction and various circuit-building techniques, including the utilization of OpAmps and timers like the 555 timer. Using SIMetrix constantly provided us a better understanding of things, as we were able to seamlessly transition between theoretical knowledge and practical implementation, which over the course of the lab was particularly seen in the simulation and construction of OpAmp filter circuits. Accomplishing the outlined objectives, we successfully generated and captured sine, square, and pulse waves, while also meticulously measuring voltage, current, and resistance. Tasks like analyzing Arduino PWM signals and transferring circuits from SIMetrix onto real breadboards highlighted our increasing skills and understanding of the topics. As we strengthen our grasp of circuit techniques, analysis, software, and equipment, we feel comfortable saying that this lab sets a strong base for our future explorations.

Lab Exploration Topics

Scope Probes: (Part A)

1. 1X setting on the scope probe means no attenuation occurs within the scope. This means the oscilloscope will read the same signal at the input of the scope probe. 10X setting on the probe means the signal is attenuated to 1/10th its original signal within the probe. This means the oscilloscope will read 1/10th of the actual signal.
2. 

3. For the 10X setting of a scope probe, the probe's compensation can usually be adjusted. The compensation is controlled by a variable capacitor within the probe, which can be adjusted to find the right compensation. The "right" compensation will make pulse signals look rigid.
4. The advantage of the 1X probe is that it maintains its signal to noise ratio (SNR) for all size signals. Its disadvantage is it cannot be used for large signals, as the oscilloscope cannot read large amplitude signals.
5. The advantage of the 10X probe is that it can read a large range of signals, as it shrinks large signals down to an amplitude the oscilloscope can read. One disadvantage of the 10X probe is that for low amplitude signals, the SNR is decreased, meaning the signal will be more crowded with noise.
6. The advantage of using wires is that they're simple, so they are less likely to break, and they are simple to use. The disadvantage of wires is that they can contain a lot of noise, which will distort the signal.

555 Timers: (Part B)

1. How does the 555 timer work in astable mode? What makes the timing capacitor charge and what makes it discharge?

   In astable mode, the oscillation frequency and duty cycle are set by two external resistors and one capacitor. The timing capacitor charges through R_{ext1} and R_{ext2} and discharges through R_A at ⅔ of the supply voltage. It charges at ⅓ of V_s.

2. How is the formula $f = \frac{1.44}{(R_A + 2R_B)C}$ derived?

   $\tau = (R_A + R_B)C$

   General form of first order response: $v(t) = V_{fin} - (V_{fin} - V_{iin})e^{\frac{T}{\tau}}$

   Half of the way to fully charged: $e^{\frac{T}{\tau}} = .5t$

   $T_{upward} = ln(2) * \tau$

   $T_{upward} = ln(2) * (R_A + R_B)C$

   After $\frac{2V_s}{3}$, the capacitor is discharging and R_B is the only resistor that affects the time constant.

   $T_{downward} = ln(2) * (R_B)C$

   $T_{total} = ln(2) * (R_A + R_B)C + ln(2) * (R_B)C = ln(2)(R_A + 2R_B)C$

   $f = \frac{1}{T_{total}} = \frac{1}{ln2} * \frac{1}{(R_A + 2R_B)C} = \frac{1.44}{(R_A + 2R_B)C}$

3. What is the duty cycle of the rectangular output at pin 3 in astable mode, i.e., what is the percentage of time the output is high?

   The duty cycle may be precisely set by the ratio of R_A and R_B.

   50% is the percentage of time the output is high.

4. Is the frequency f dependent on the supply voltage? Why or why not?

   No, it depends on $\frac{1.44}{(R_A + 2R_B)C}$, which does not involve V_s. The voltage has nothing to do with the time constant τ.