UNIVERSITY OF COLORADO - BOULDER

ECEN 2270

ELECTRONICS LAB | SPRING 2024

ECEN 2270 Electronics Lab: Lab 1

Team Papa:
Gabriel AGOSTINE
Sam WALKER
Julian WERDER
Jonah YUNES

Lab Instructor:
Steven DUNBAR

Lab: Section 12

Sunday, February 4, 2024

College of Engineering & Applied Science
UNIVERSITY OF COLORADO **BOULDER**

I. Experiment A

A. Exploration Topics

What is the difference between the 1X and the 10X settings of the Scope Probe?

A probe's attenuation factor (i.e. 1X, 10X, 100X) is the amount by which the probe reduces the amplitude of the oscilloscope's input signal. A 1X probe doesn't reduce or attenuate the input signal while a 10X probe reduces the input signal to 1/10th of the signal's amplitude at the scope input.*

What is inside the Scope Probe? Draw and label a schematic.

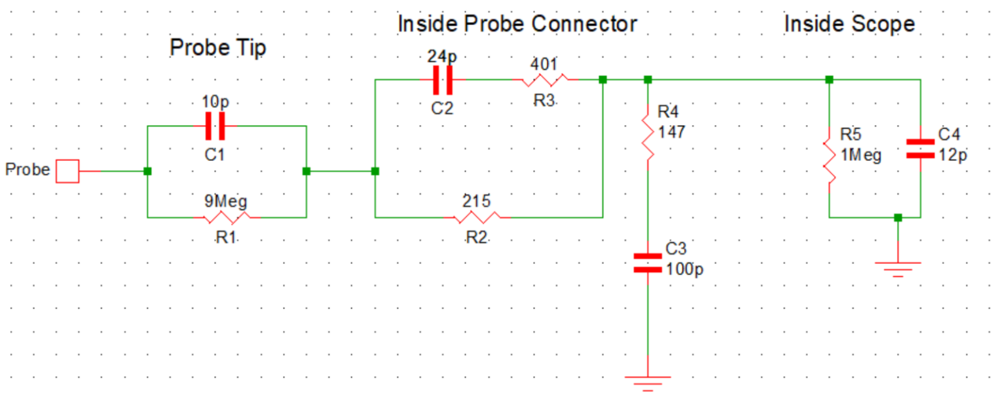

Fig. 1 Oscilloscope Probe Internals

For the 10X setting there is usually an adjustment that can be made. What does it change and why?

The 10X scope probe achieves a better high-frequency response and is used for high-voltage measurements.

What are advantages and disadvantages of using 1X probes?

Can be used for low-frequency responses and can be very hard to use for very high frequency responses.

What are advantages and disadvantages of using 10X probes?

Can be used for high-frequency responses and can be very hard to use for very low frequency responses.

What are advantages and disadvantages of just using wires?

Overall, it is easier and more convenient to use but has many fallbacks such as limited bandwidth, no signal amplification, and higher input capacitance.

B. 1.A.2

To get this graph we used the wave generator tool to make a sine wave with a 1.5 amplitude and an offset of 1.5. These values were chosen to make sure that the maximum is 3V and the minimum is 0V. And to make the frequency 400Hz we typed "400" in the frequency menu.

*Tektronics, https://download.tek.com/document/51W_27668_0_MR_Letter.pdf

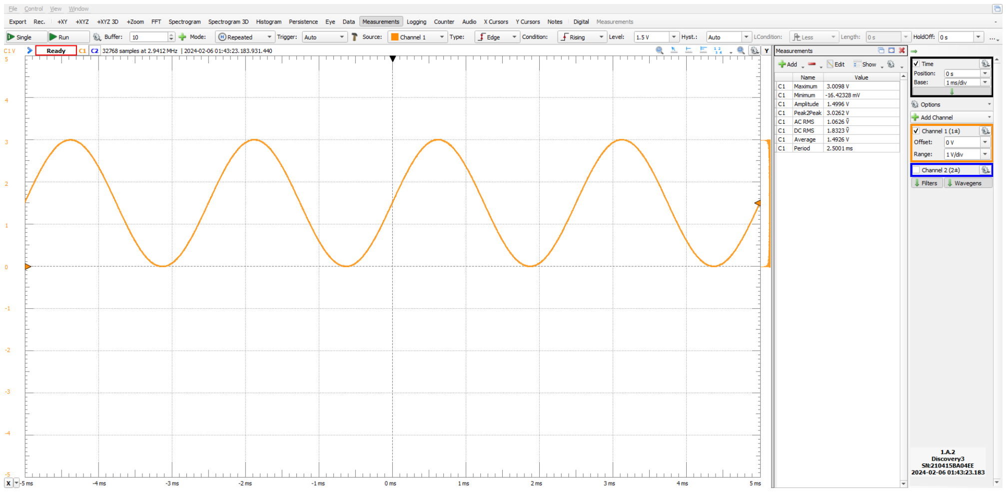

Fig. 2 Sinusoidal Wave

V_{rms} can be calculated for most waves using the equation:

$$V_{rms} = \sqrt{\frac{A^2}{2} + A_{DC}^2}$$

Where A is the amplitude of the waveform (the value from the average/middle point of the wave to the crest or trough of the waveform) and A_{DC} is the DC offset of the waveform. A derivation that results in this equation is in the **Appendix & Derivations** section.

C. 1.A.3

To get the specified square wave we chose the square wave option on the waves menu and put an amplitude of 1 and an offset of 1V. To get the specified frequency of 1kHz we typed this value in the frequency menu on the square wave.

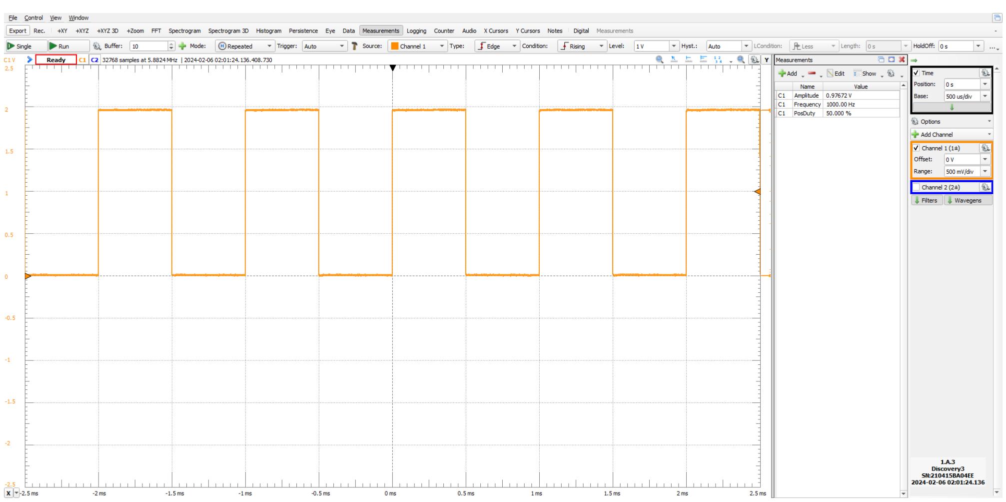

Fig. 3 Square wave

This is a square wave function with a peak-to-peak voltage of 4V and a 1ms period. Its duty cycle is 50%. There was not much attenuation at a rising or falling edges once we calibrated the scopes.

D. 1.A.4

We achieved the desired pulse width by knowing the fundamental frequency is 1kHz, which translates to 0.001s or 1ms. If we want a duty cycle value, this is defined as $\frac{time_{on}}{time_{off}} * 100$. Thus, if our on time is 20μs or 0.02ms, our duty cycle ends up being 2%...

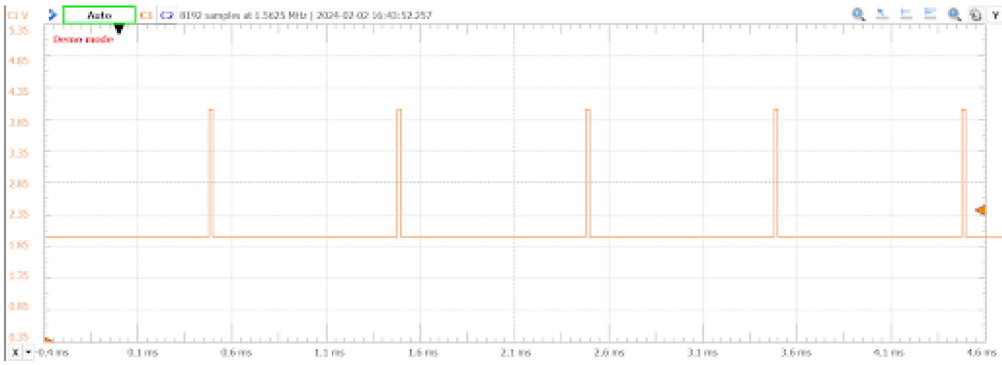

Fig. 4 2% duty cycle Square wave

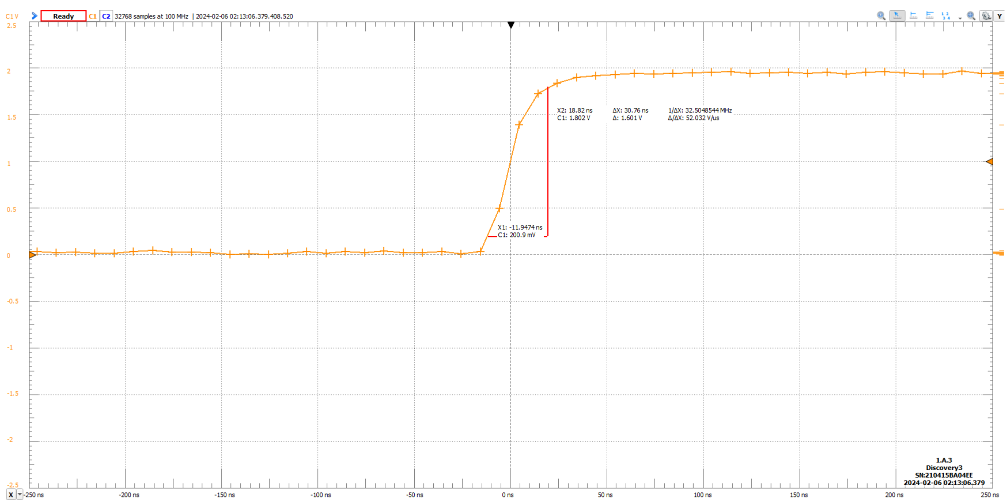

Fig. 5 Square wave Rising edge

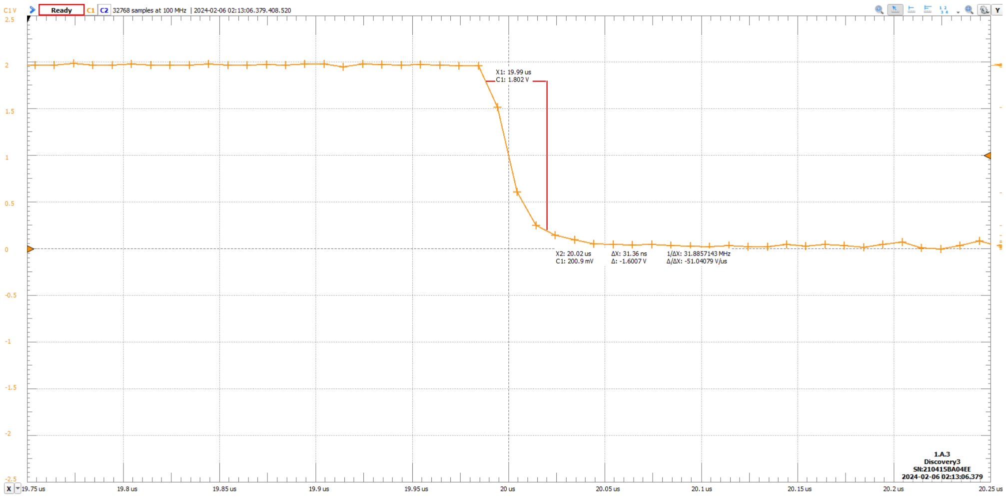

Fig. 6 Square wave falling edge

For both the rising and falling edge of the square waves shown above, they both have a defined rise and fall time as well as (in the case of the rising edge) move from a minimum to maximum (or set amplitude) or vice versa.

E. 1.A.5
Flywire Setup

Fig. 7 Pulse wavegen using flywire assembly

Probe

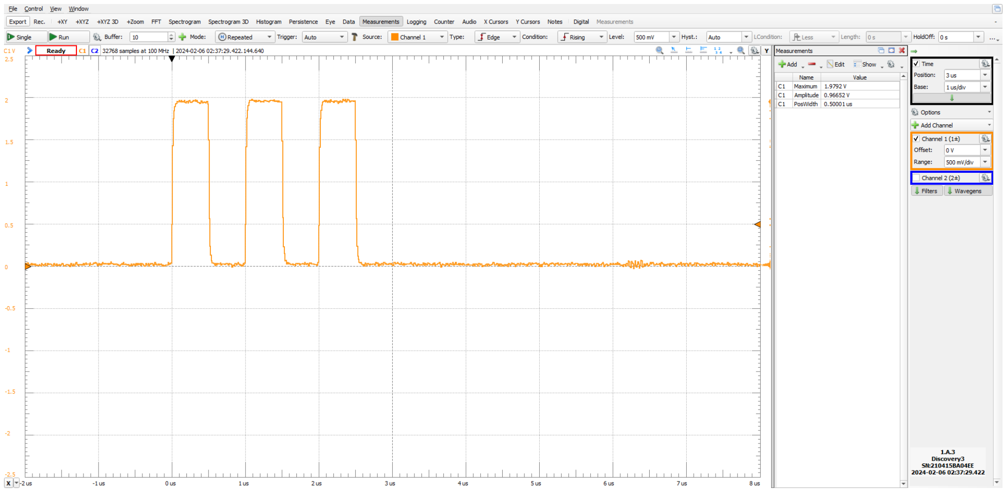

Fig. 8 Pulse wavegen using 10X oscilloscope probe

We can see in **Figures 7-8** that the 10X probe setting was much more accurate than the flywire assembly.

In Waveforms, one can generate a custom waveform by selecting this option on the waveform generation screen. By defining the base frequency (which specifies the time length of each value) and the value that each timestep is associated with, we can create a custom waveform. In this case, a fundamental frequency of 195kHz was chosen and a three pulse (2 V amplitude, [2 0 2 0 2 0 ... 0] pattern) signal was created and then verified in the scope to ensure it was indeed correct, as seen in **Figure 8**.

F. 1.A.6

For this section, we aimed to measure the current through a 220Ω resistor in series with a LED and plot this relationship to get the IV curve of the LED (which is a Light Emitting Diode).

By using the voltmeter tool on the Analog Discovery device, we were able to determine the following values and plot them as seen below...

V+	Channel 2 (V_CC)	Channel 1 (V_R1)	V_LED	I_LED
0.5	0.528	0.018	0.51	8E-05
1	1.028	0.02	1.008	9E-05
1.5	1.528	0.016	1.512	7E-05
2	2.028	0.264	1.764	0.001
2.5	2.528	0.712	1.816	0.003
3	3.028	1.178	1.85	0.005
3.5	3.528	1.652	1.876	0.008
4	4.028	2.132	1.896	0.01
4.5	4.528	2.612	1.916	0.012
5	5.028	3.098	1.93	0.014

Fig. 9 LED Voltage and Current values

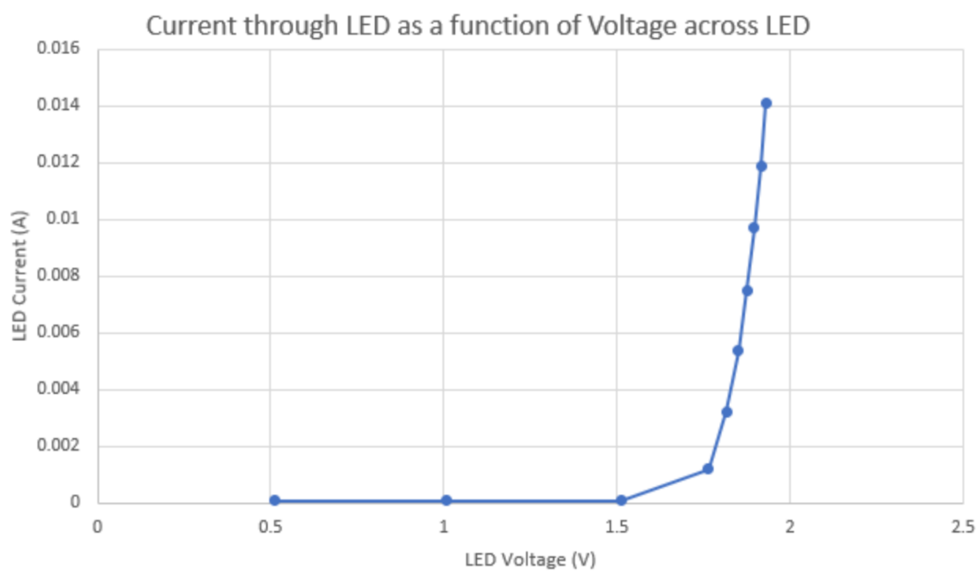

Fig. 10 LED (Light Emitting Diode) IV curve

If we wanted to measure the LED's IV curve characteristics with the AD2 up to I_{LED} = 30 mA, we would change the resistor for a higher value for safety and change the voltage to step up to 10 volts.

G. 1.A.7

$$R_1 = \frac{\frac{V_{R1}}{V_{in}} R_2}{1 - \frac{V_{R1}}{V_{in}}} = \frac{V_{R_1} R_2}{V_{in} - V_{R_1}}$$

Where: V_{R1} = 5 mV, V_{in} = 0.5 V and R_2 = 330Ω.

Thus, R_1 = 3.33Ω, which can be represented by three 10Ω resistors in parallel.

V_{in} [V]	V_{R1} [V]	V_{R2} [V]
0	0	0
0.5	0.494	0.008
1	0.99	0.014
1.5	1.484	0.018
2	1.978	0.024
2.5	2.476	0.03
3	2.968	0.034
3.5	3.462	0.038
4	3.956	0.044
4.5	4.45	0.05
4.66	4.612	0.05

Table 1 Dual Resistor Voltage values

Using the values in **Table 1** we can derive the current through R_2 (whose values is specified above alongside R_1) in series with R_1 and plot that against the IV curve from the previous section.

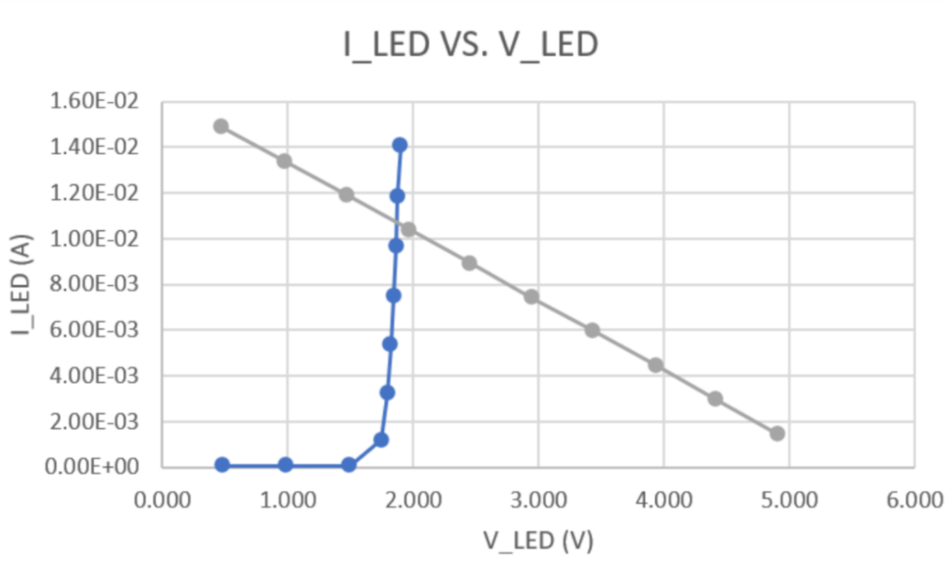

Fig. 11 Operating Point IV curve

It is important to remember when calculating the load line that we must flip the current values so that we have a negative slope.

By connecting our LED once more in series with the 330Ω resistor and setting the voltage source to 5 V, the led lit up. The channel one output jumped down to 48 mV.

H. 1.A.8a

For this section of the lab, we built a Low Pass Filter (LPF) and measured the output RMS voltage as we increased frequency past the corner frequency (a point 3 dB below the passband gain) of the filter.

Freq	Vin_RMS	Vout_RMS	Gain, A	log(f)	dB
100	3.427	3.408	0.994456	2	-0.04829
125	3.443	3.417	0.992448	2.09691	-0.06584
160	3.432	3.396	0.98951	2.20412	-0.09159
200	3.436	3.385	0.985157	2.30103	-0.12989
250	3.478	3.419	0.983036	2.39794	-0.14861
315	3.448	3.346	0.970418	2.498311	-0.26083
400	3.462	3.31	0.956095	2.60206	-0.38998
500	3.467	3.251	0.937698	2.69897	-0.55874
630	3.452	3.136	0.908459	2.799341	-0.83389
800	3.459	3.001	0.867592	2.90309	-1.23369
1000	3.454	2.824	0.817603	3	-1.74915
1250	3.455	2.611	0.755716	3.09691	-2.43282
1600	3.454	2.334	0.675738	3.20412	-3.40443
2000	3.451	2.063	0.597798	3.30103	-4.46891
3150	3.444	1.515	0.439895	3.498311	-7.13301
4000	3.442	1.256	0.364904	3.60206	-8.75642
6300	3.438	0.858	0.249564	3.799341	-12.0564
10000	3.436	0.565	0.164435	4	-15.6801
31500	3.382	0.189	0.055884	4.498311	-25.0542
100000	2.891	0.055	0.019025	5	-34.4137

Fig. 12 RC circuit V_{rms} and Gain values

$V_{in_{rms}}$ does change, decreasing exponentially with input voltage frequency increase. This is expected as since the impedance of a capacitor is $\frac{1}{j\omega C}$, the capacitors impedance (or complex resistance) approaches 0 as frequency increases. Thus for a configuration of a capacitor in series with a resistor, as frequency increases, less voltage will drop across the capacitor and the output voltage will decrease with the capacitor voltage.

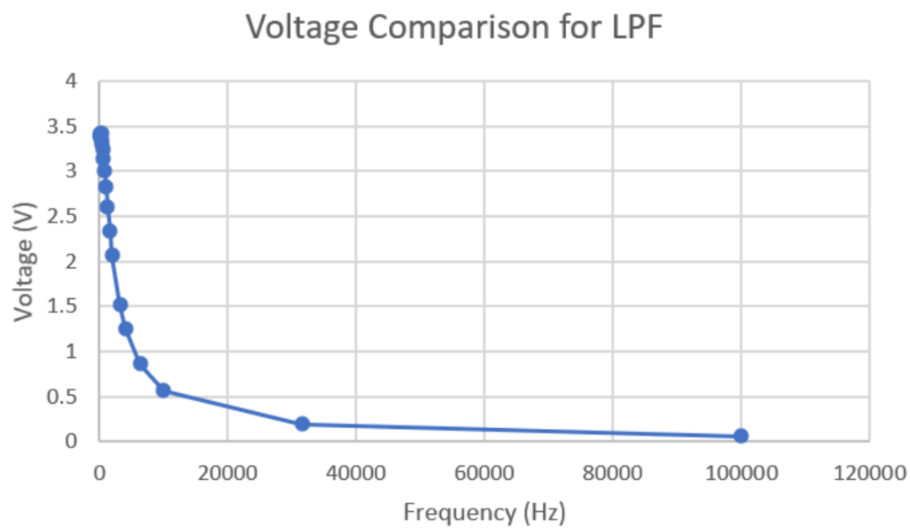

Fig. 13 Linear RC circuit Voltage vs. Frequency

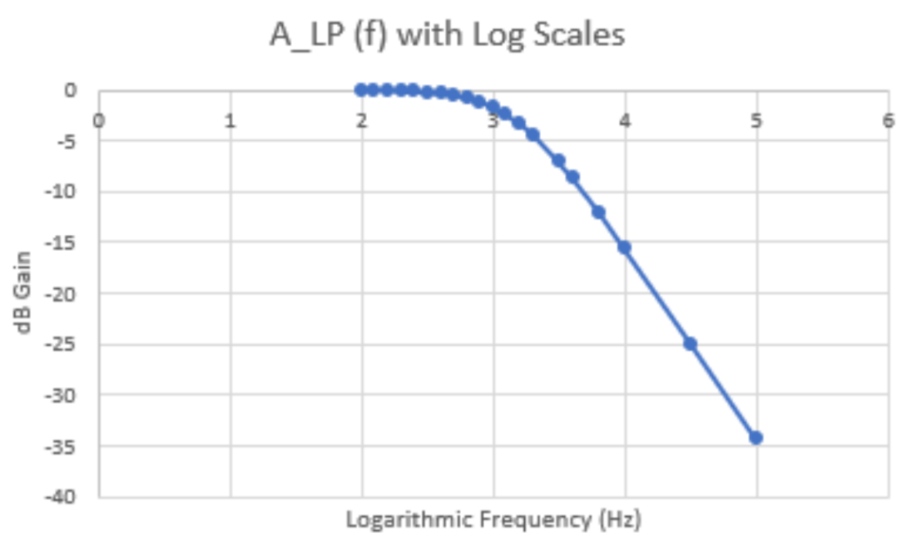

Fig. 14 RC circuit Bode Plot

This circuit is called a Low Pass Filter (LPF) as any input frequencies below the corner frequency (a point 3 dB below the passband gain) are unattenuated, while frequencies past said frequency are (in the case of a simple first order LPF) attenuated by 20 dB per decade.

I. 1.A.8b

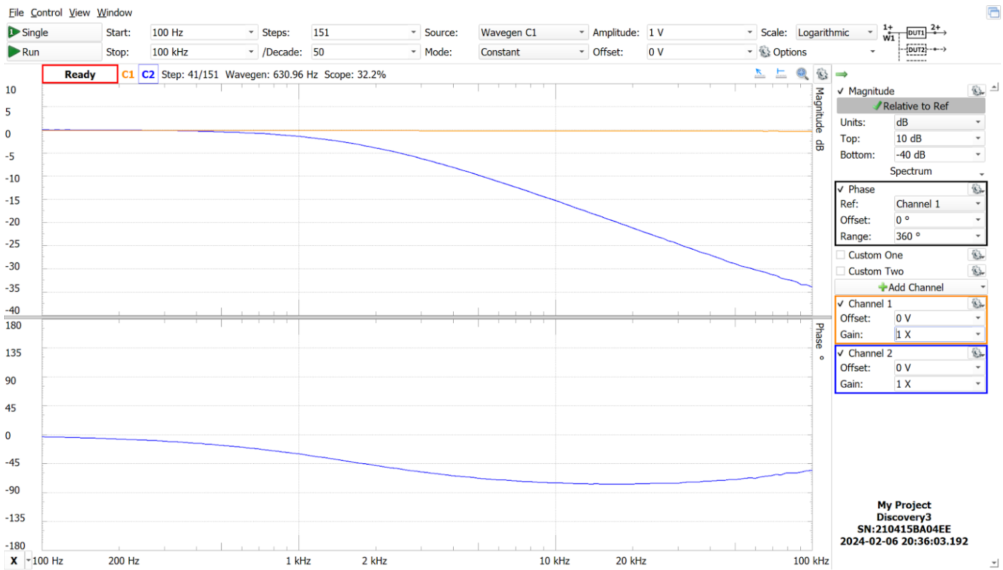

Fig. 15 Waveforms Frequency Analysis for a Simple First Order LPF

Figures 14 and 15 have similar shapes and values which is expected, with their being slight errors between the two due to realistic system errors. Using Excel to get a graph proved more time-consuming and the “network” tool on Waveforms is faster and more real-time accurate with the measurement graphs.

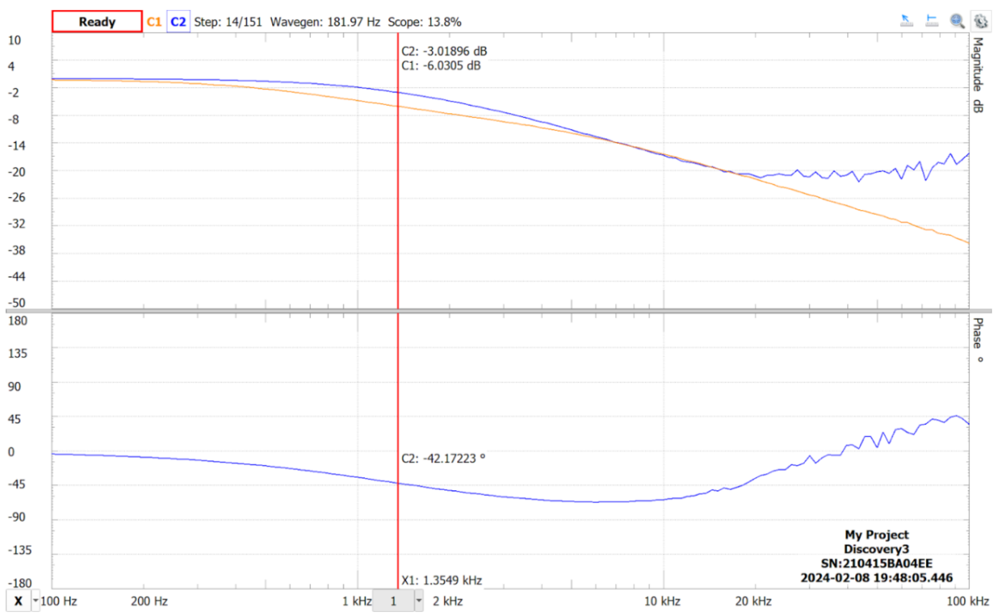

Fig. 16 Waveforms Frequency Analysis for a Simple Second Order LPF

A second order LPF is "twice as good" as a first order LPF filter as the attenuation per decade (normally -20 dB/dec for a first order filter) is "doubled" to become -40 dB/dec in a second order LPF since they outputs are cascaded together.

J. 1.A.9
For this section, we used the Arduino Nano to generate a PWM signal which was used to light up an LED. The on time of the PWM signal can (to the human eye due to its flicker speed) give the appearance of a dimmer or brighter light.

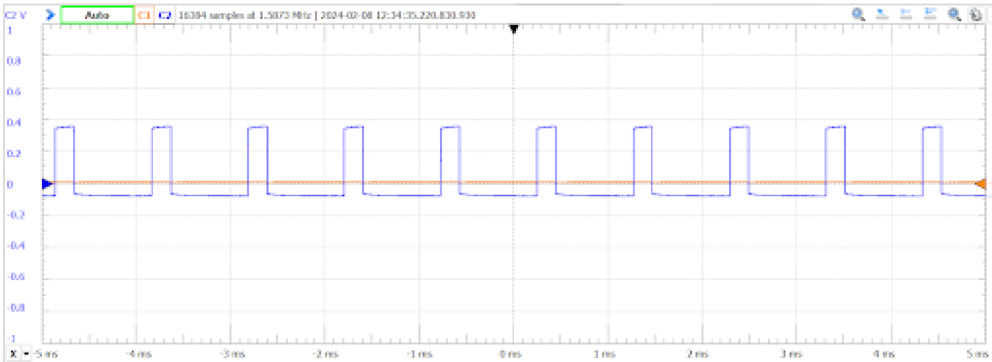

Fig. 17   25% Duty Cycle PWM wave

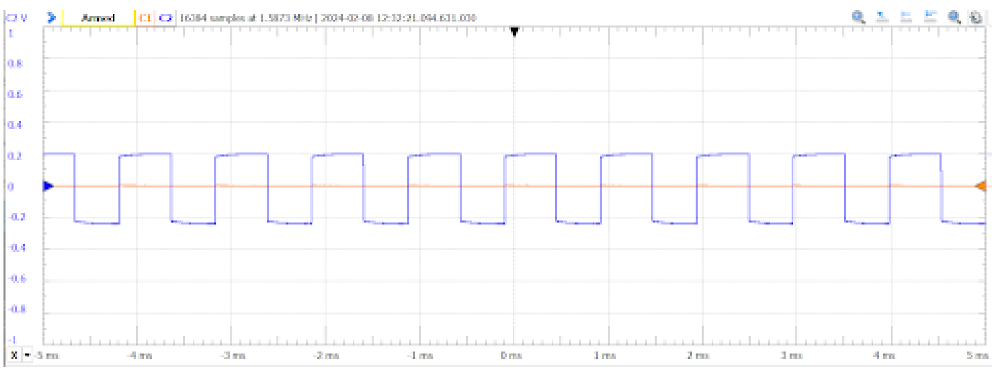

Fig. 18   50% Duty Cycle PWM wave

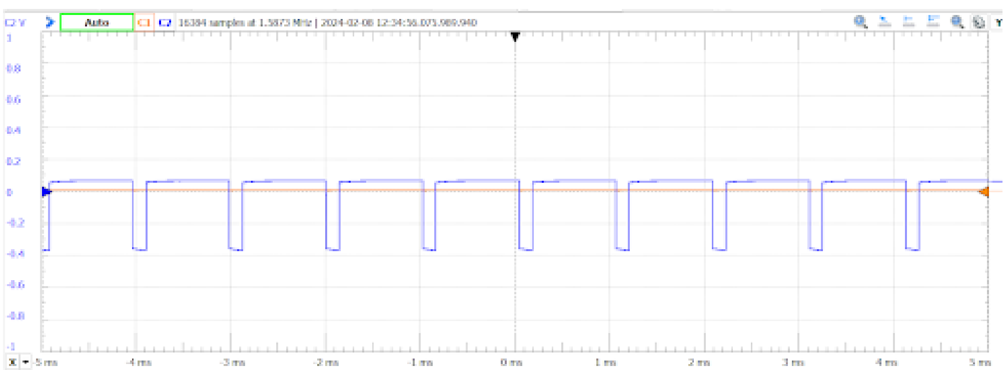

Fig. 19   99% Duty Cycle PWM wave

In order to convert a PWM signal into a analog signal (using a Digital to Analog Converter or DAC), we must remove all harmonics of the input signal except the first harmonic (often called the fundamental frequency). This can be achieved by adding a LPF on the end with specific R and C values such that its corner frequency is just past the first harmonic (as can be seen in a FFT plot). Adding subsequent filters or designing a higher order filter are also possible options if the harmonics are not generously spread.

II. Experiment B

A. Exploration Topics

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

The oscillation frequency and duty cycle are determined by 2 external resistors and 1 capacitor. When the capacitor reaches $\frac{2}{3}$ V's the flip-flop flips to discharging the capacitor until $\frac{1}{3}$ V's is reached; then it flips back into charging mode.

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

Derived by adding the charge and discharge time to find the period then dividing 1 by the resulting equation, to convert period to frequency.

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?

$\frac{100(R_A+R_B)}{(R_A+2R_B)}$

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

No, the supply voltage just changes the voltage range the waveform takes place in.

B. 1.B.2

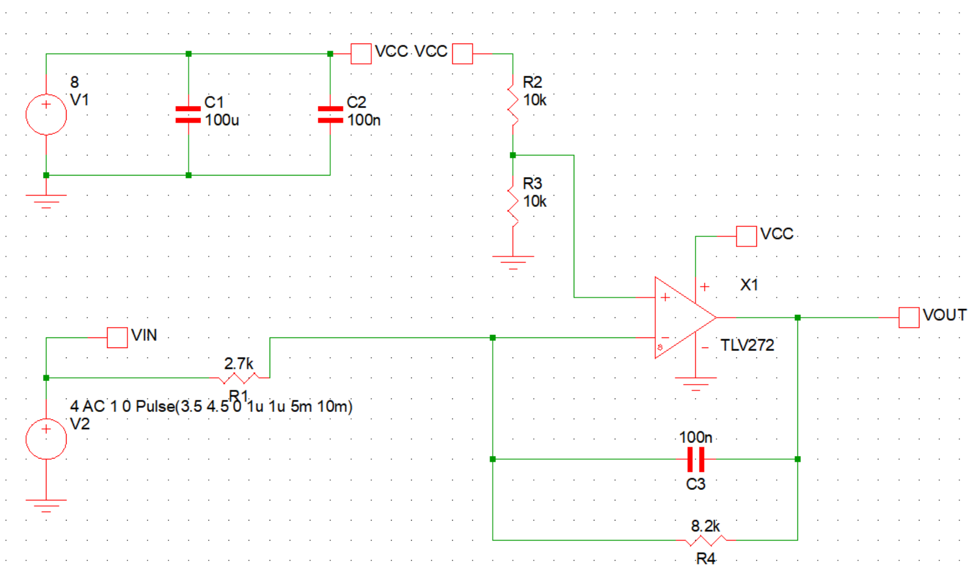

Fig. 20 Simple Active First Order Op-Amp LPF

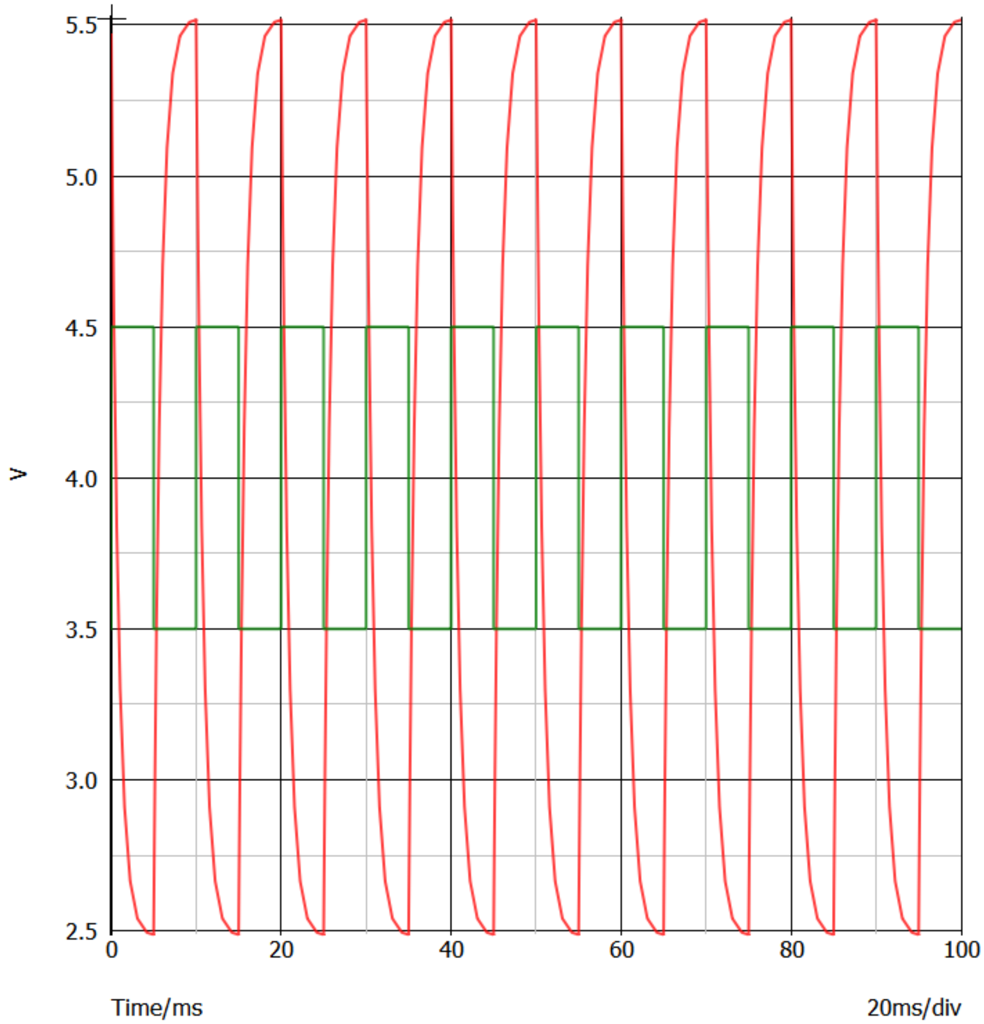

Fig. 21 Op-Amp LPF V_{out} (red) compared to V_{in} (green)

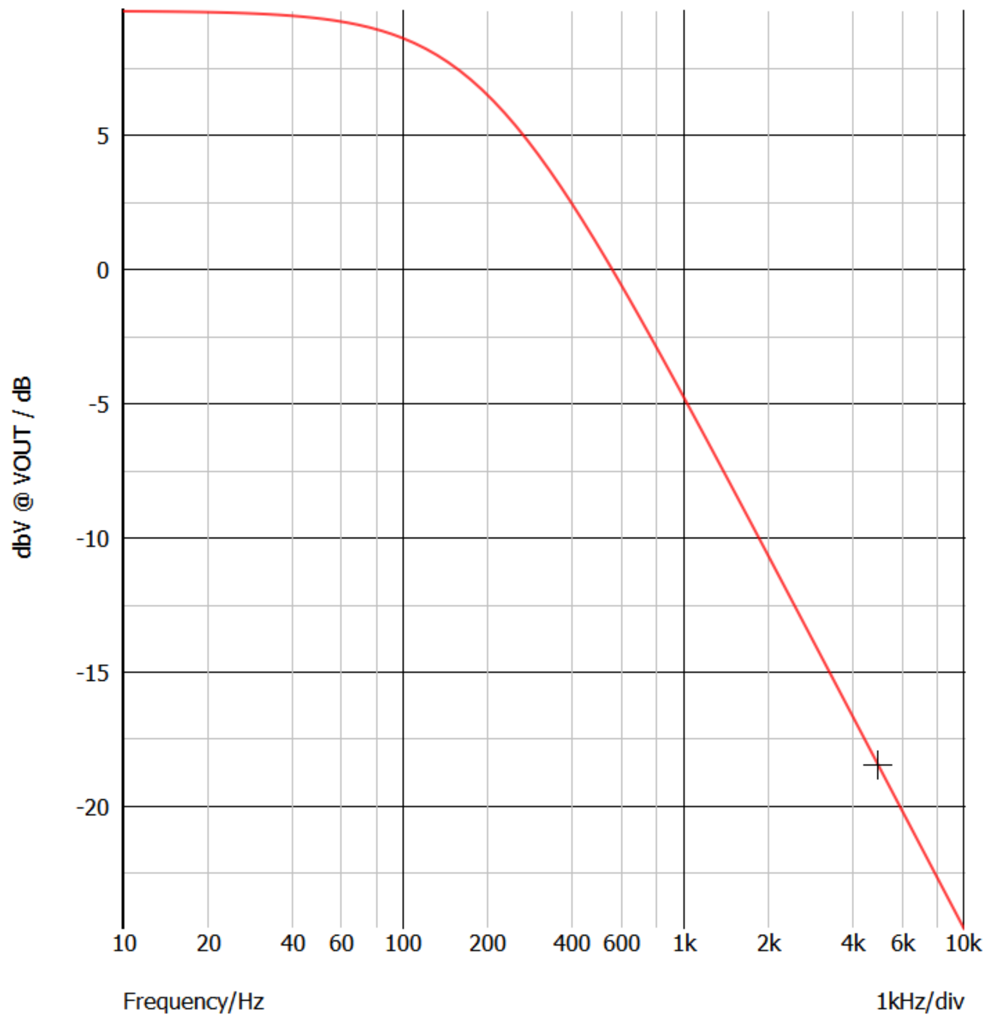

Fig. 22 Op-Amp LPF Bode Plot

Looking at the Logarithmic graph of the Inverting Op-Amp Filter, we can determine some important values:

- DC Gain: 9.64 dB = 3.03
- Corner frequency: 194.3 Hz
- Magnitude change per decade: -20.1 dB/dec

These results are what we expect as this is a Op-Amp First Order Low Pass Filter.

R_3 sets what a voltage divider that splits off into the positive terminal of the Op-Amp. Thus, if at any time the voltage at the positive terminal is greater than that at the negative terminal, the output voltage swings up to the positive rail V_{CC}. R_4 is a integrated part of the circuit and defines the value of the DC gain. Since for a Op-Amp filter the DC gain (or passband gain) can be increased or decreased by increasing or decreasing the value of R_4, as defined by its transfer function.

The output waveform (red) is similar to a sawtooth function with a peak-to-peak voltage of 3V and a 20ms period. Notice it is expectedly inverted from the input waveform (green).

C. 1.B.3

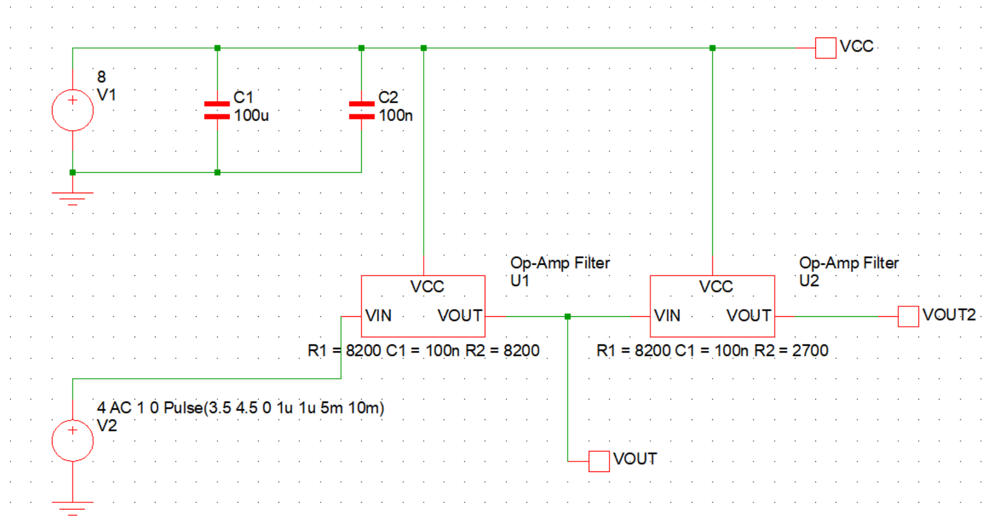

Fig. 23 Second Order Op-Amp LPF using Integrated Circuit (IC) Components

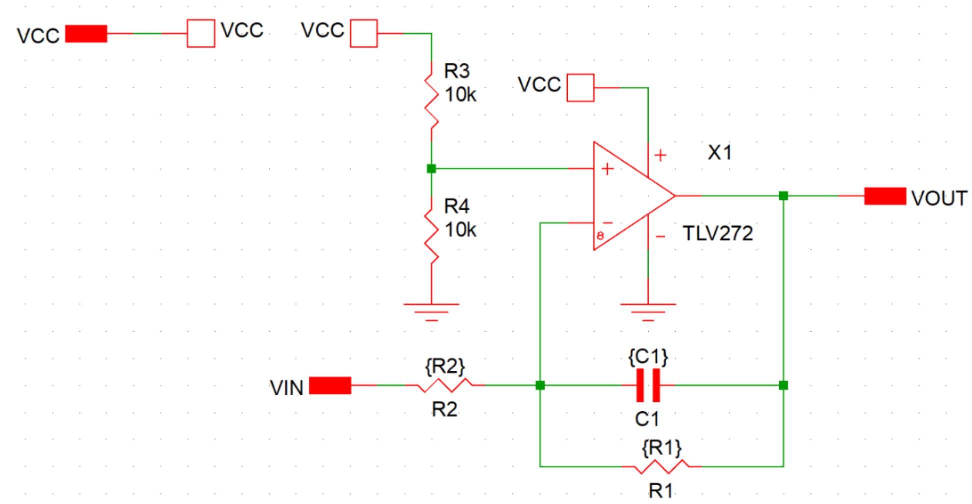

Fig. 24 Op-Amp LPF Internal Schematic

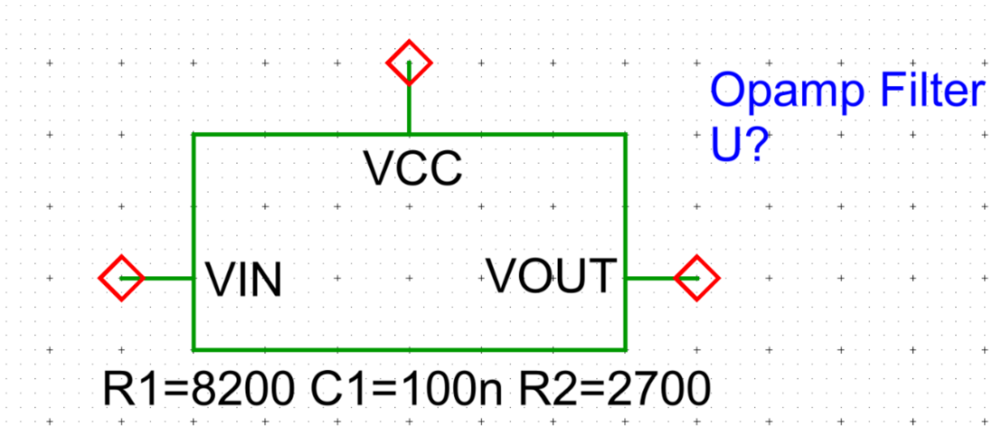

Fig. 25 Op-Amp LPF Symbol

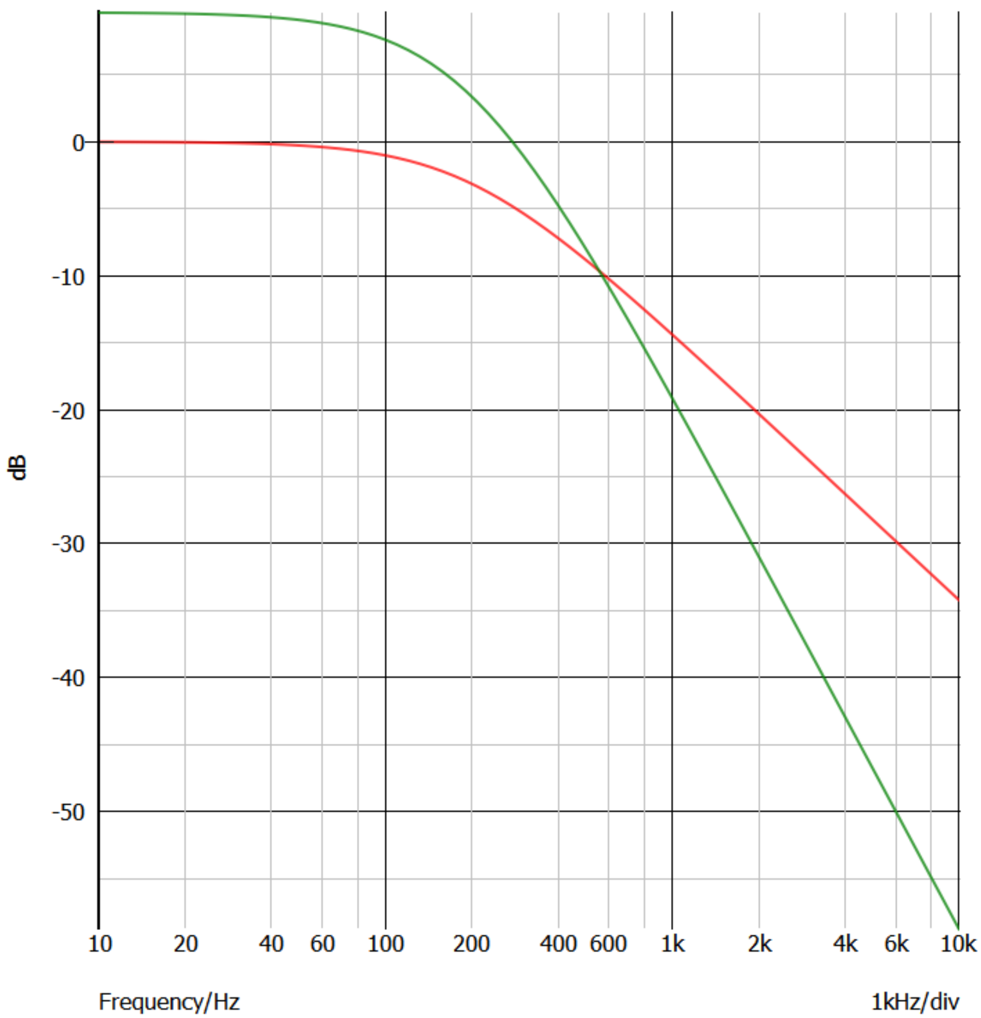

Fig. 26 Second Order Op-Amp LPF V_{out_1} (red) and V_{out_2} (green)

Looking at the above graph, we can determine important parameters:

- V_{out_1} DC gain: 1
- V_{out_2} DC gain: 3
- V_{out_1} corner frequency: 194.2 Hz
- V_{out_2} corner frequency: 125.5 Hz

Using a R_2 value of 8.2kΩ for the first filter allows us to give it a DC gain of 1 (or 0 dB).

We checked that our filters were correct by following ensuring they followed the expected behavior of the Simple Active LPF Inverting Op-Amp transfer function.

$$H(j\omega) = -\frac{R_1}{R_2}\left(\frac{1}{1 + j\omega R_1 C_1}\right)$$

$$|H(\omega)|_{dB} = 20log_{10}\left(\frac{R_1}{R_2}\right) + 20log_{10}\left(\frac{1}{\sqrt{1 + (\omega R_1 C_1)^2}}\right)$$

Our circuit does indeed follow the expected transfer functions. We also know they are correct because they represent a two-stage LPF. When measuring the first LPF we get the cut-off frequency at -3dB then measure the second filter with that same cut-off frequency which gave a -6dB voltage.

To make the -3dB frequency 10 times higher, we simply need to move the passband gain up 10 times its normal gain. This can be achieved by making R_1 10 times higher.

D. 1.B.4

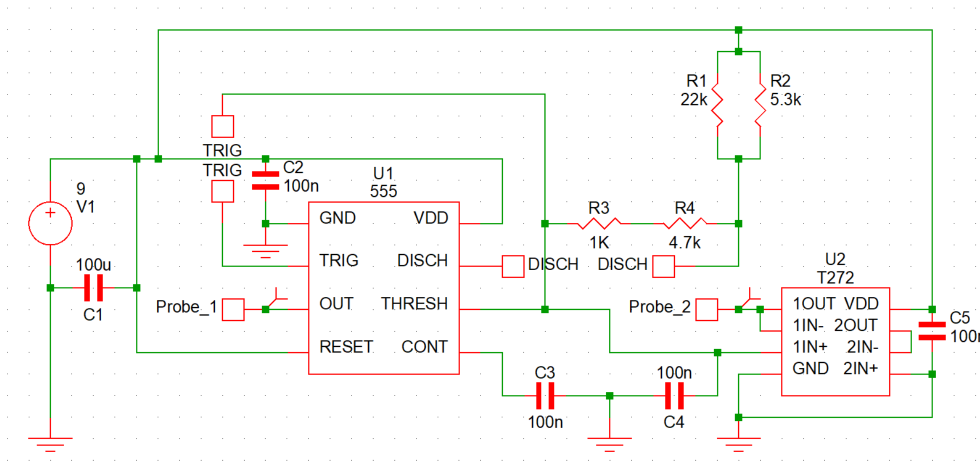

Fig. 27 SIMetrix Prelab Circuit

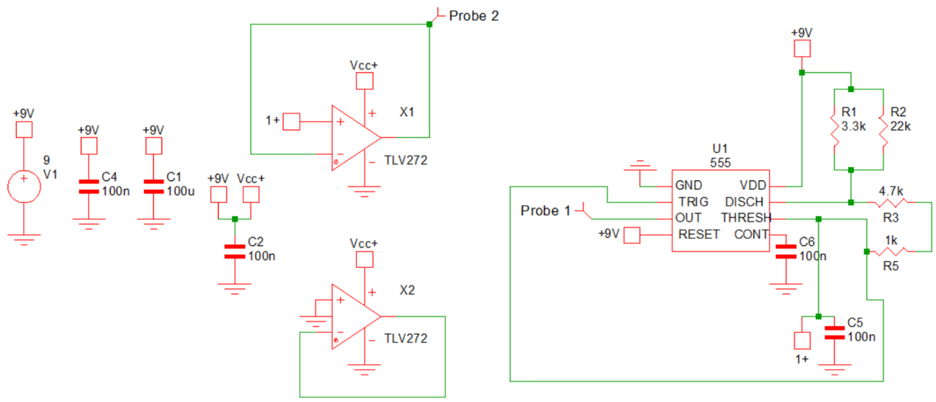

Fig. 28 SIMetrix Prelab Circuit (Alternate Form)

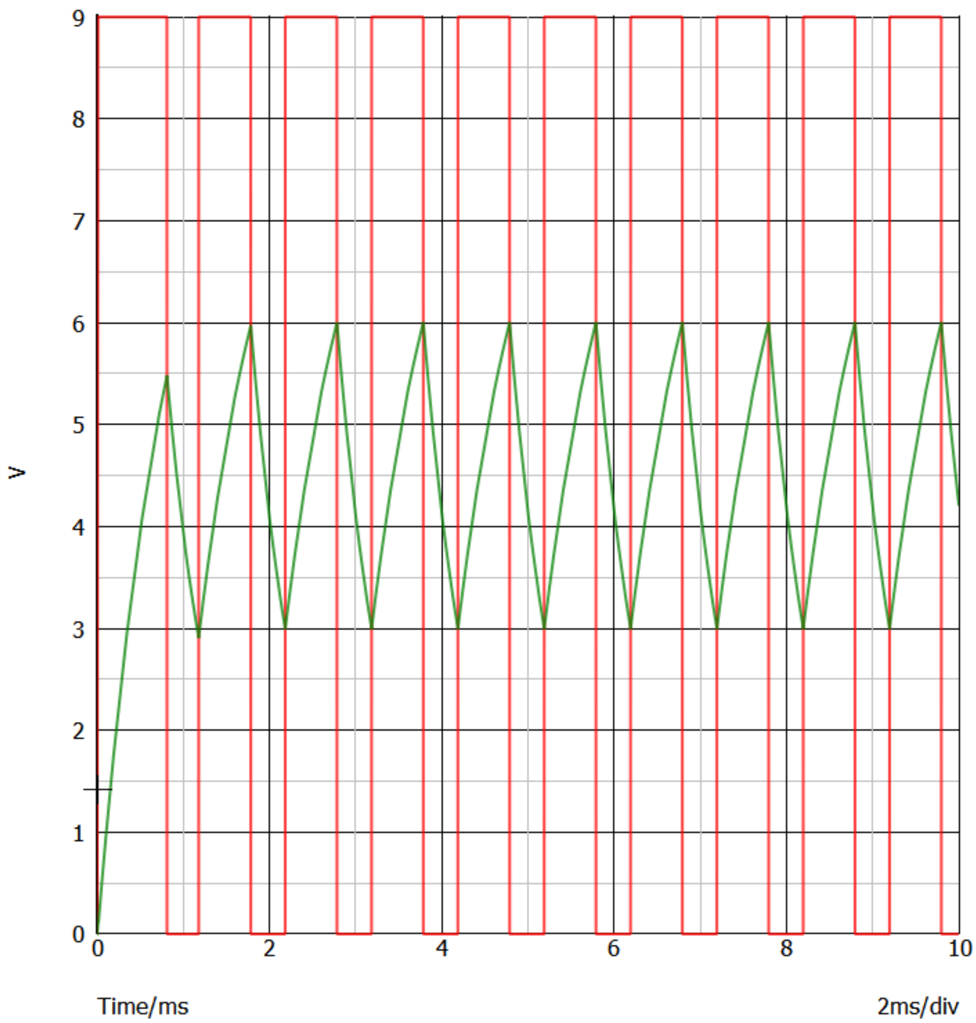

Fig. 29 Prelab Circuit Output Channels 1 (red) and 2 (green)

E. 1.B.5

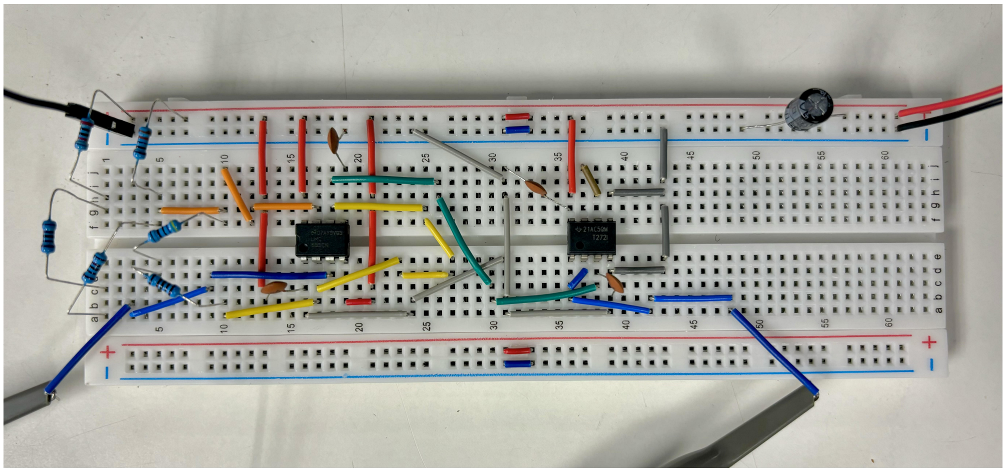

Fig. 30 Physical Prelab Circuit on Breadboard

F. 1.B.6

When making the 1.B.4 circuit we ran into some issues with calibrating the probes to make the Graph from probe 2 measuring the out terminal on the Op-Amp look as closely as an integrated square function as possible.

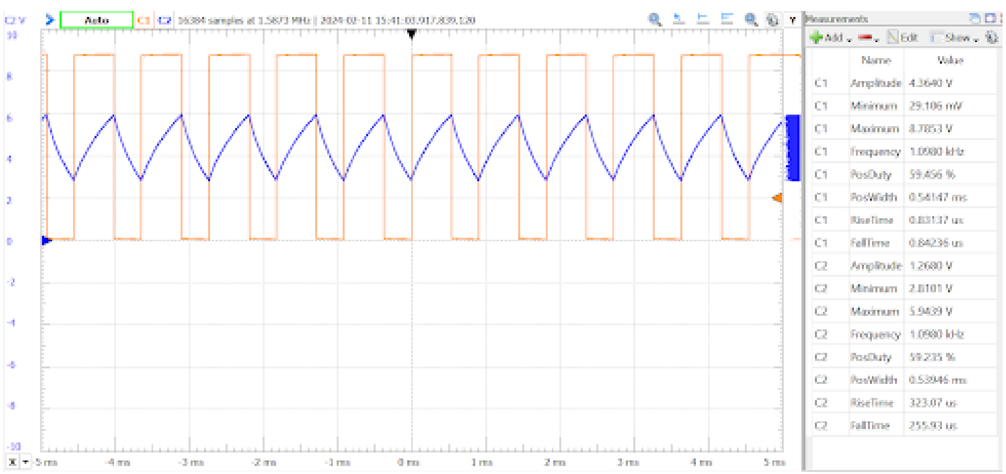

Fig. 31 Prelab Circuit Waveform Oscilloscope Output Channels 1 (yellow) and 2 (blue) with Measurements

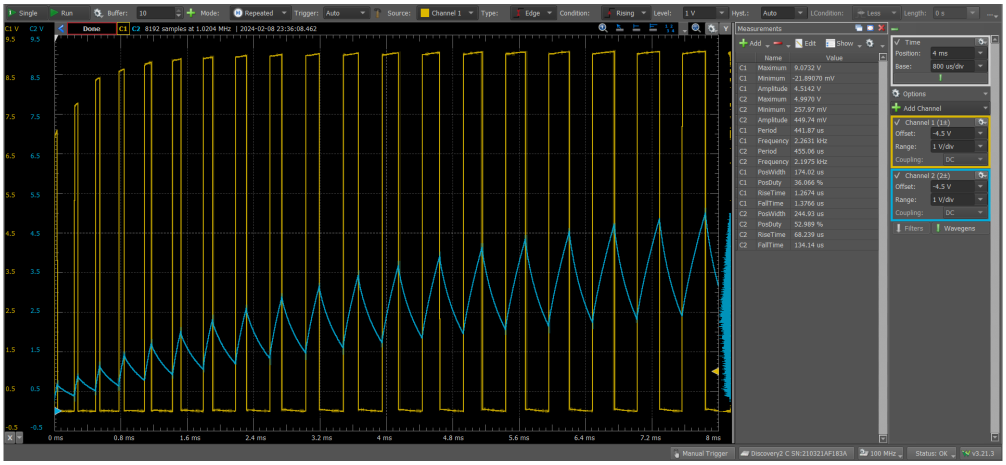

Fig. 32 Prelab Startup Waveform

The experimental and simulated results match up very well. Their steady state properties align as expected and their rising shapes are similar. There is a small difference in the maximums and minimums of the simulated and measured but that could be from the loss due to the resistance and capacitance in the wires.

Debugging was a mater of making sure that no wires were missing in the circuit and making sure that it is as easily "readable" as possible. It should be noted that in realistic systems a more compact circuit is naturally desirable, however for a breadboard design this isn't necessary beyond developing good practices. In future labs a more compact designs/layouts (dissimilar to that seen in **Figure 30**) will be adhered to.

III. Conclusion

Overall, this Lab helped our group learn how both Passive and Active Filters operated, how to derive their Transfer Functions (both Linear and Logarithmic for Normal, Magnitude and Phase), how to use them (for example plotting Bode and Phase plots) and how to interpret them. These skills are very useful for circuit design and can be applied widely in order to meet a number of various tasks.

Our group also learned what a PWM waveform can be used form, how to generate our own custom waveforms in the Waveforms application using our Analog Discovery devices. These PWM waveforms can be used to vary the intensity of a LED or the average value of a source input.

Lastly, our team learned good circuit design strategies as well as how to read and take advantage of various resources in order to learn about the internals of IC chips (such as the 555 Timer) and use them to our advantage. We also learn how to back-propagate a circuit and ensure or determine its functionality using software such as SIMetrix.

IV. Appendix & Derivations

A. Appendix A

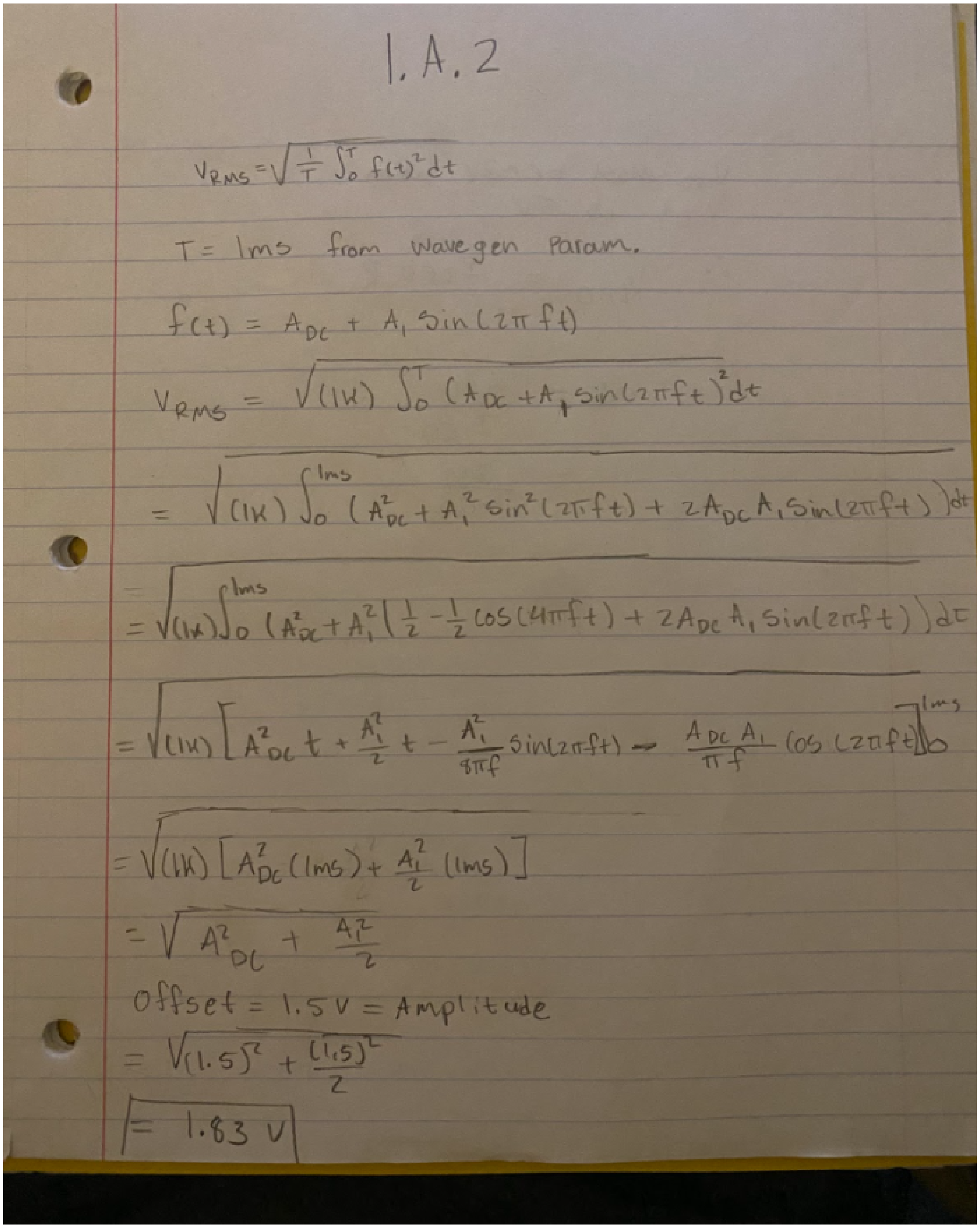

Fig. 33 V_{rms} **Derivation**

B. Appendix B

Tools used:

- Waveforms Software
- Analog Discovery 2/3 (AD2/3)
- Excel
- MatLab
- Oscilloscope & Probes
- Arduino Nano
- Breadboard and various circuit components