Lab 1

ECEN 2270
University of Colorado Boulder

February 9, 2022

Luke Hanley
Nicholas Haratsaris
Ginn Sato

Introduction

In this lab we explored the many different functions of our Waveforms software as well as the applications of these functions in circuit analysis. We took time learning how to effectively use the software to both measure and generate different waveforms. LTSpice helped us simulate different waveforms that we could then test in the real world using the AD2 scope.

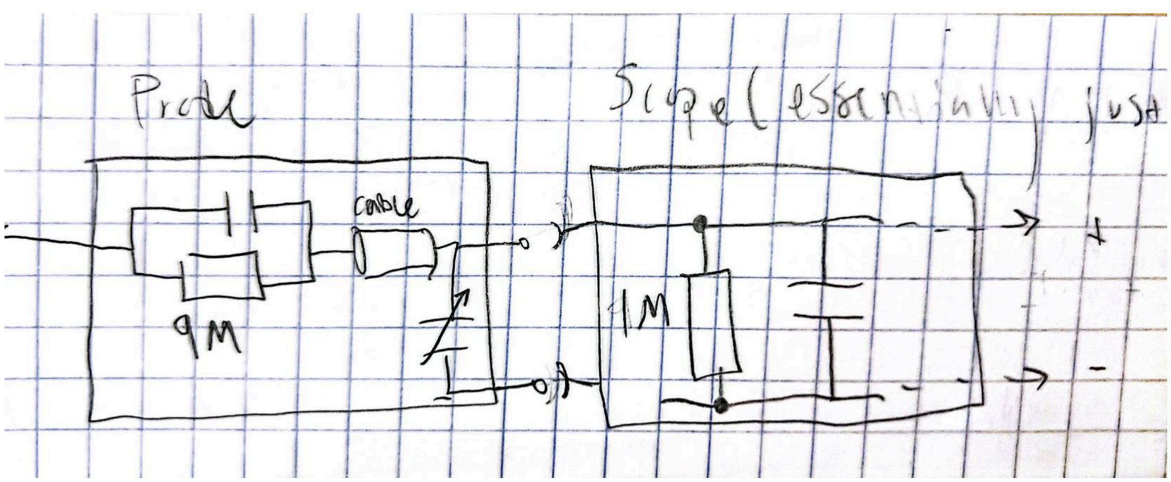
The goals of this lab included gaining proficiency with the waveforms and LTSpice softwares. Specifically, we want to learn how to use the wave generator, take accurate measurements, create a frequency vs voltage plot, trigger our waveform, use the BNC probes, and more. We also wanted to understand how nonidealities can play into our measurements when, for example, we calculate the resistance of a resistor. Our objectives include generating and measuring real world waveforms, then comparing these to an ideal world in LTSpice.

Equipment List:

- AD2 Scope w/ BNC Probes and Flywire Assembly
- LTSpice
- Waveforms
- LMC555 Timer
- TLV272 OpAmp
- Jumper Wires and Breadboard
- Arduino Nano Every
- Resistors
- Capacitors
- Red LED

1.A.1 Prelab

1) It refers to the impedance of the scope, in that the probes can be used to multiply the impedance of the scope by a factor of 10. Otherwise, it remains the same with 1x. Thus the measured signal is attenuated by a factor of 10

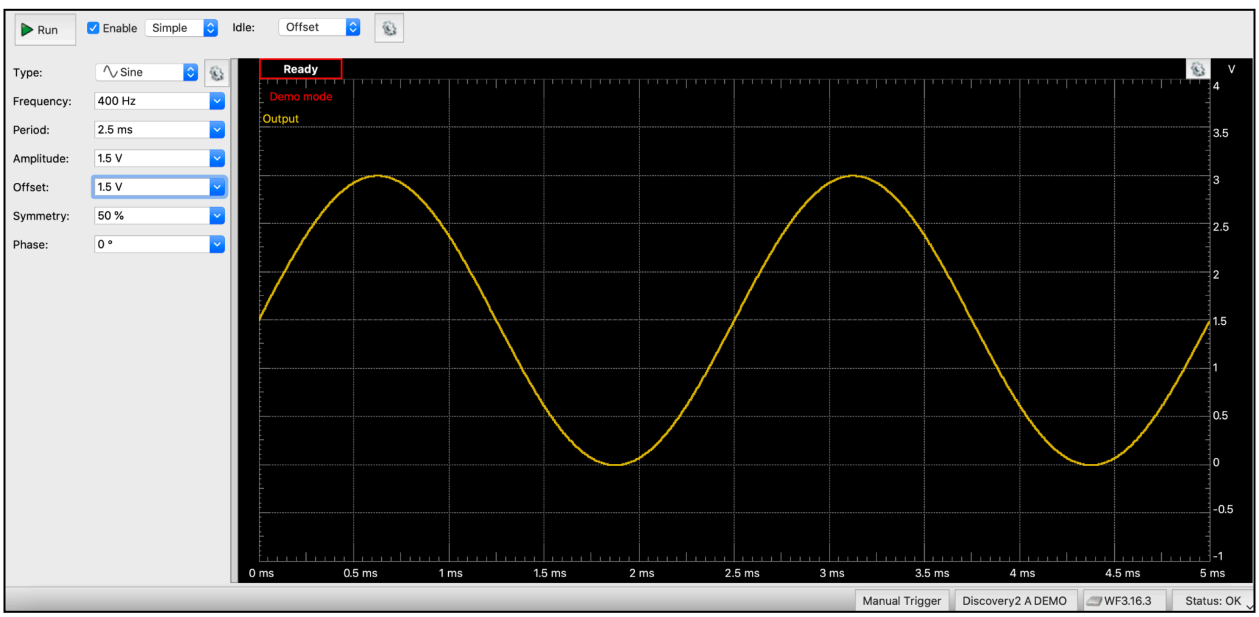
2) 

At 1x, we essentially just have the right half schematic, with 1MΩ impedance. When we switch to 10x, we essentially add another 9MΩ of impedance in series, creating 10MΩ of impedance, and thus attenuating the signal by a factor of 10.

3) We can adjust one of the capacitors in the probe to compensate for low frequencies. We screw in a part of the probe, which adjusts capacitance, so we can better capture different frequencies of a signal.
4) The advantages of using a 1x probe is that it has a low input impedance thus the measurements will be more accurate for lower amplitude signals in low impedance circuits. The disadvantage is that its not good for higher amplitudes.
5) The advantages of a 10x probe is that its good for higher amplitude signals since it reduces the voltage by 10x so the oscilloscope can read 10x higher voltages. Disadvantages mostly come from the amount of noise it will measure and create at lower frequencies.
6) Wires are easier and more straightforward. However, they are more susceptible to noise and the inductance can skew our measurement.

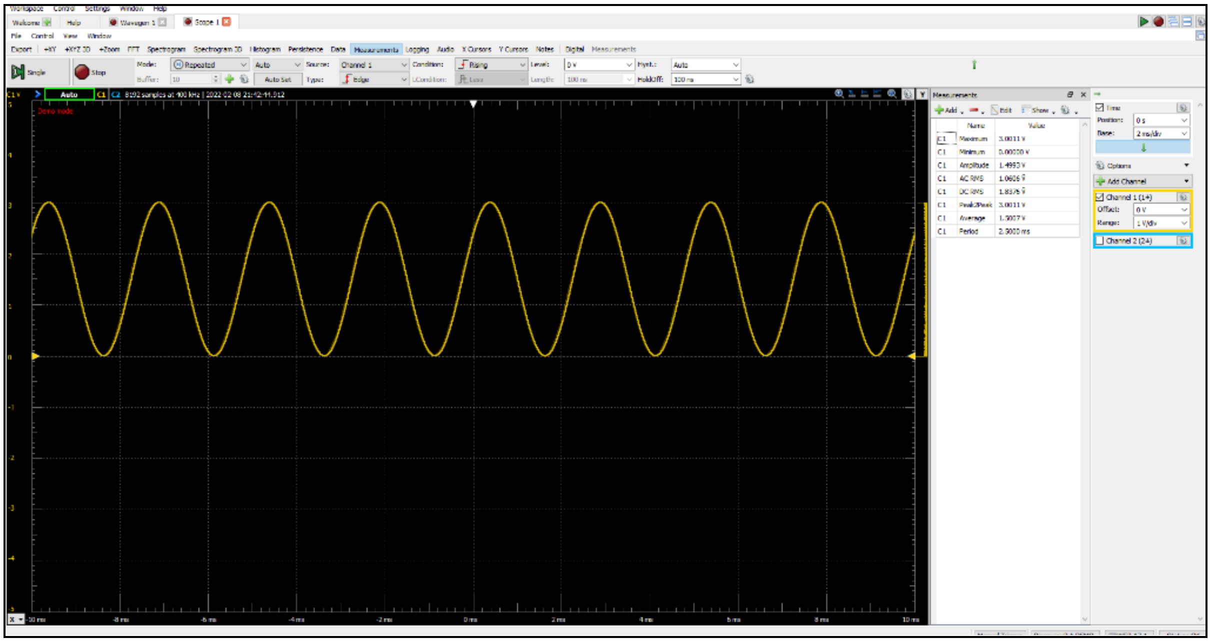
1.A.2 Sine Wave Generation and Capture

First we set up our waveform generator using the WaveGen with the following values.



Wavegenerator of 400 Hz, V_{min} = 0 V, and V_{Max} = 3V

After setting up our circuit so that the wavegen reads into channel 1 of our scope we navigated to the scope window. We then calculated the following values using the measurement feature shown in the right panel. It may be hard to see these values, so a second figure is shown after with a zoomed in view.

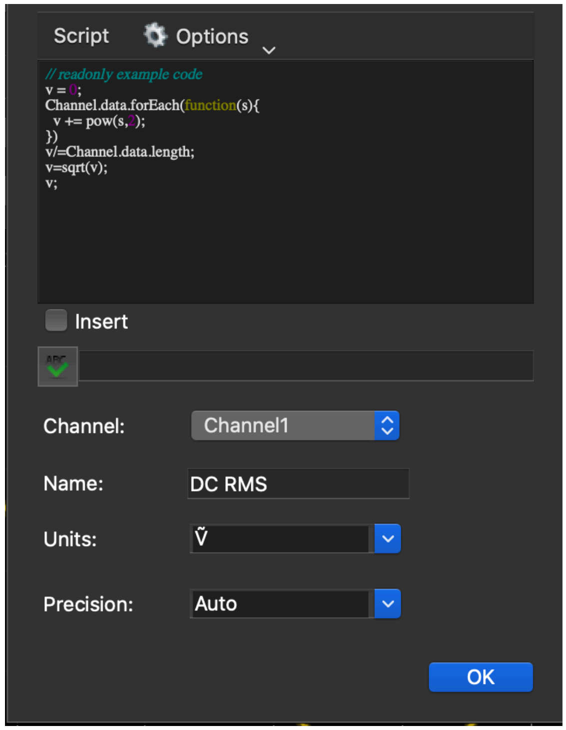


Scope Results with Several Periods Shown

	Name	Value
C1	Maximum	3.0011 V
C1	Minimum	0.00000 V
C1	Amplitude	1.4993 V
C1	AC RMS	1.0606 Ṽ
C1	DC RMS	1.8376 Ṽ
C1	Peak2Peak	3.0011 V
C1	Average	1.5007 V
C1	Period	2.5000 ms

Zoomed in View of Measurements

Below is an example of how one of the measurements, DC RMS, was calculated using properties of the waveform and its parameters. This is essentially the code behind the calculations that can help us realize where these measurement values are coming from. This code should look reasonably familiar, given that we know how to find the RMS voltage of a signal given the peak to peak voltage.

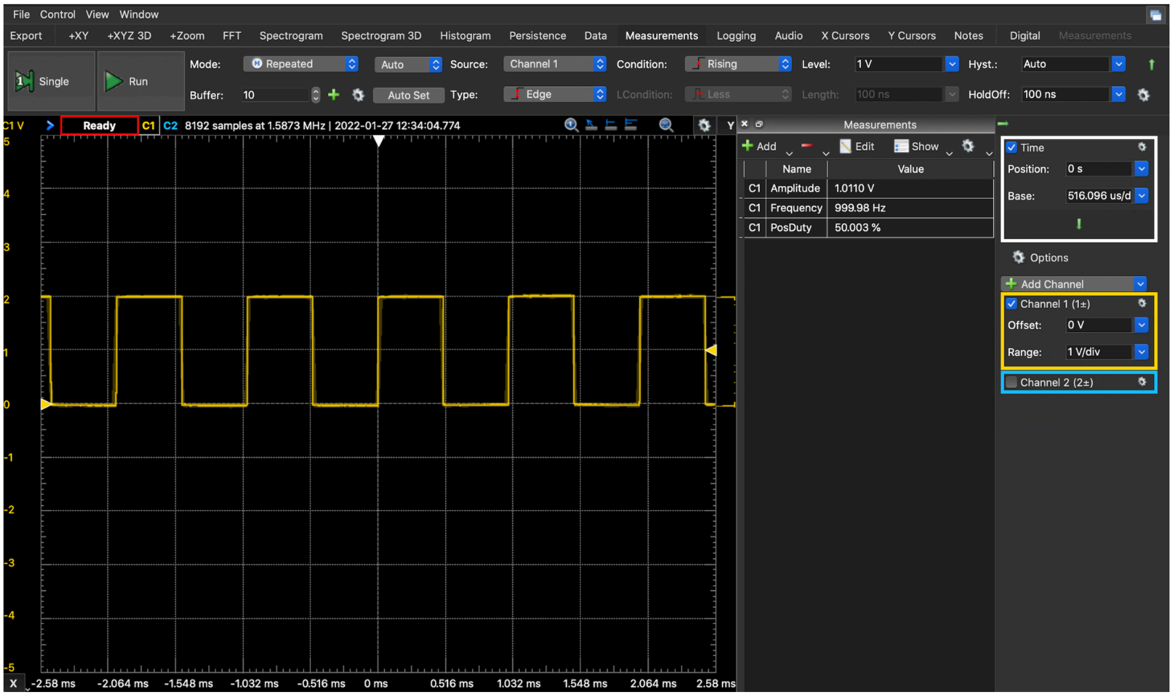


DC RMS Value Computation Code

1.A.3 Square Wave Generation and Capture

First we set up a square wave with our wavegen software, with minimum voltage 0V, maximum voltage 2V, frequency 1kHz, and duty cycle 50%. Using the BNC adaptor, we run the signal through our breadboard and into our oscilloscope capture software. The captured waveform is shown below.

Given our signal and use of oscilloscope probes, we had to calibrate our probe. We adjusted the oscilloscope screw compensation so that our square wave was captured correctly, which is something we expect to be a common procedure in the future. Initially it was undercompensated, and finally it was correctly compensated.



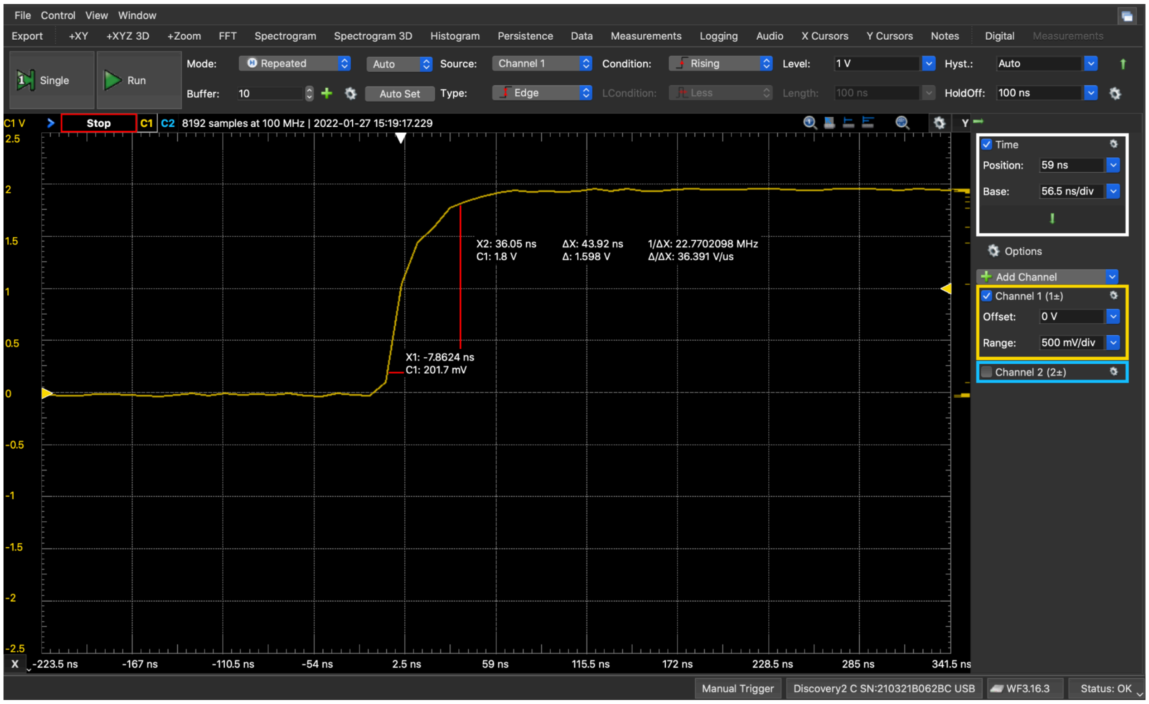
Oscilloscope Capture with Amplitude, Frequency, and Duty Cycle Measurements

The wave is on for half the time, off the other half, which makes sense given our duty cycle measured value of 50%. The rise and fall time are extremely short, and essentially are just vertical lines. In layman's terms, the signal is alternating between 2V and 0V at equal intervals. These descriptions are what make the signal a square wave.

1.A.4 Periodic Pulsating Waveform

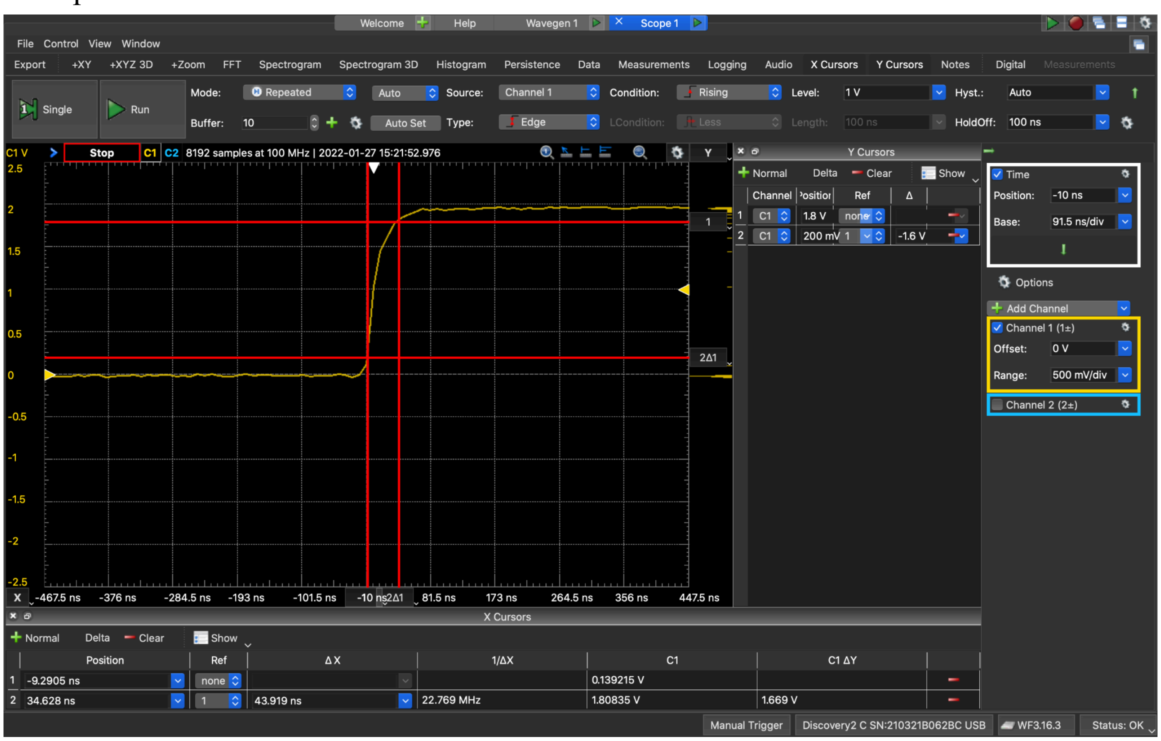
In order to create a periodic pulsating waveform with a frequency of 1kHz and a pulse width of 20 us. We know that the period is 1/1000 = 1 ms. We can then take the time on, t_{on}, and divide it by the total time, t. So t_{on}/t = 2e-5/1e^-3 = .02. We then take this as a percent so 0.02 * 100 = 2 % duty cycle. This is the duty cycle we apply to our wavegen.

We then go to our oscilloscope to capture this waveform. We want a precise measurement for the rise time of a given pulse, so we increase our horizontal division and sample rate significantly.



Rise time measured with Quick Measure

Quick measure is a useful tool for double checking values quickly. However, if you are using the oscilloscope for the sole purpose of measuring some value in a circuit, then you may want a more precise measurement.

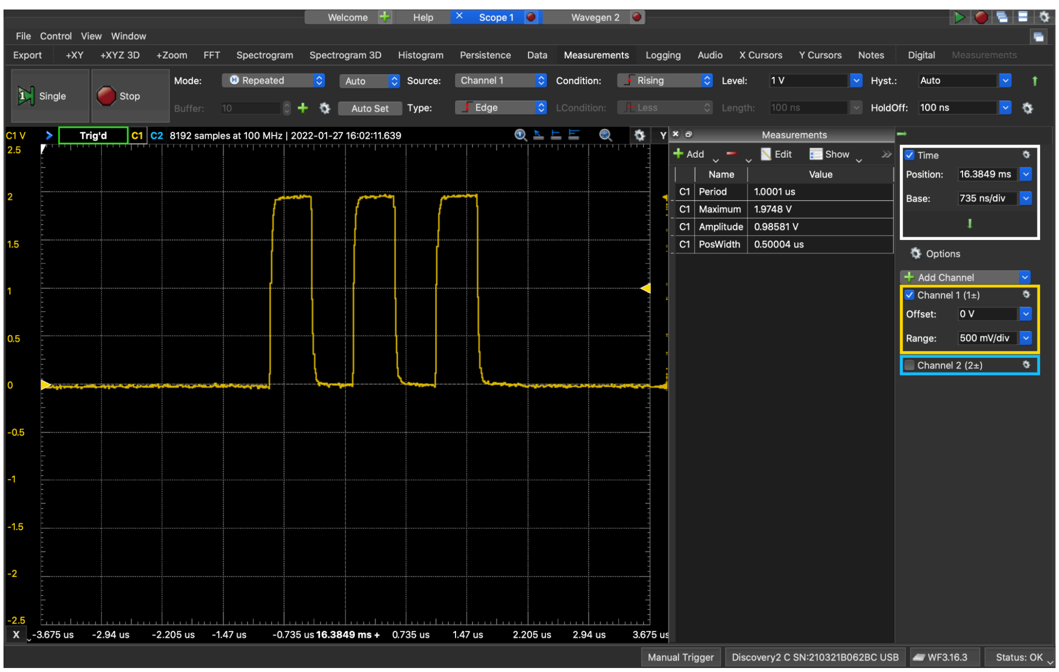


Rise time measured with X and Y cursors

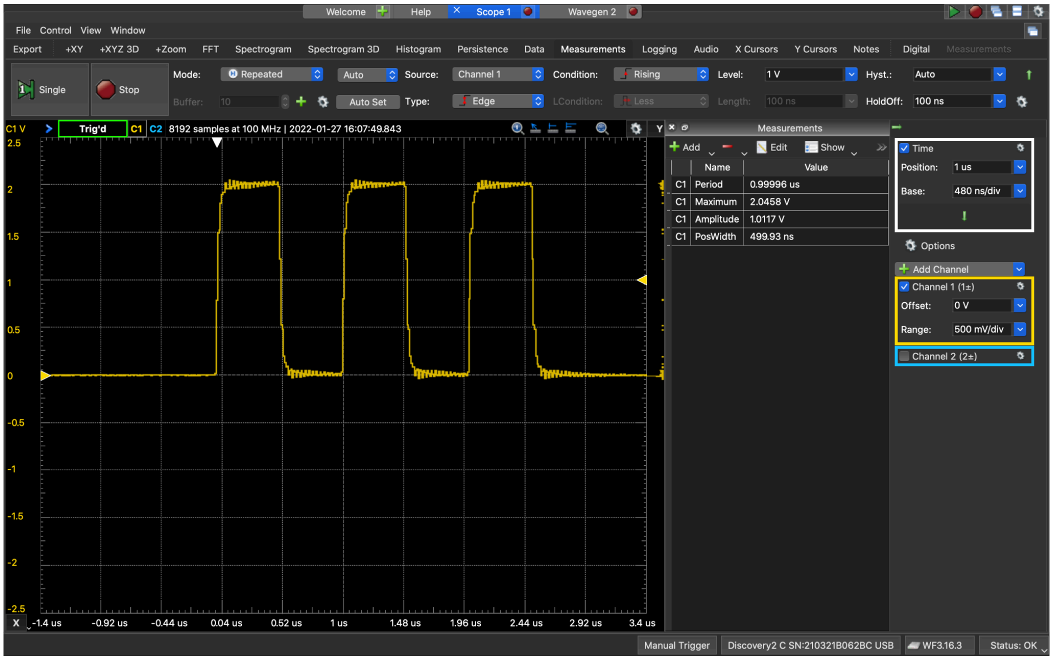
A more precise measurement can be achieved using X and Y cursors. These allow you to set edges in the oscilloscope, in which the exact coordinates are recorded so that you can measure changes in the x and y axes, which, in this case, correspond to changes in time and voltage. Being able to take accurate voltage and time measurements allow one to calculate nearly all of the characteristics of a given signal.

1.A.5 Burst of Pulses

Below we can see the difference of measuring the same pulse wave using BNC Probes and the AD2 flywire assembly. There is much less noise while using the BNC Probes with x10 attenuation.



Burst of pulses measured with BNC probes

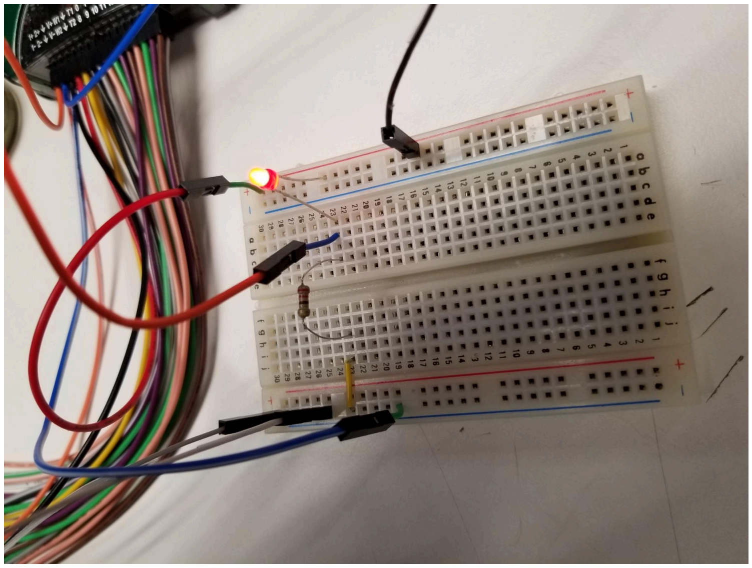


Burst of pulses measured with flywire assembly

In order to set up our waveform in the desired manner we first have to create a custom waveform. Firstly, we know that our amplitude is 2, so we set that in accordingly. We then choose our sampling frequency. We know that each pulse width needs to be 0.5 us, thus 1/0.5us = 2 MHz is our sampling rate. This means that every 0.5us we are going to take the sample from our sample values and multiply our amplitude by this value. It is important to note that the value 1 is the maximum value for a sample value and anything greater will round down to 1. For example with an amplitude of 2 and a sample value of 2, the maximum voltage will only be 2, and not 2*2 = 4. Our first 5 sample values were 2, 0, 2, 0, 2, and then every sample value after that is defaulted to 0. This creates three pulses with a 50% duty cycle with a frequency of 1MHz. These pulses can be described as extremely fast growing voltages that have an extremely minimal rise time and falling time. Ideally these rise times and fall times are infinitesimally small. They are on for as long as they are off.

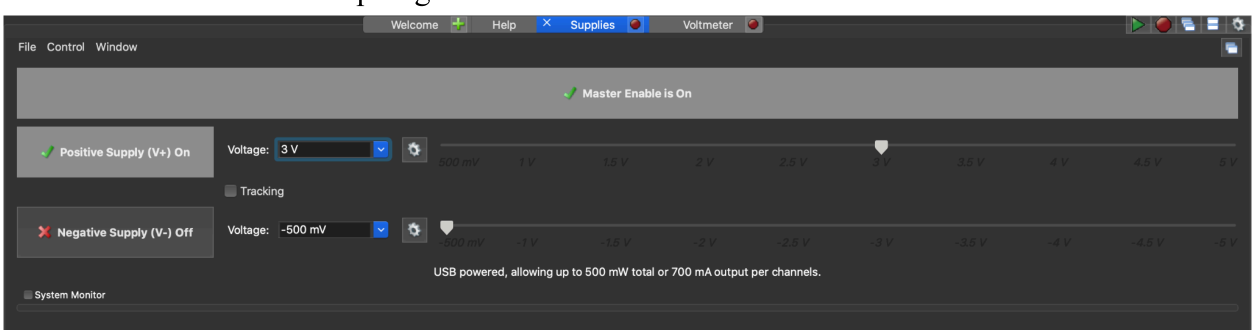
1.A.6 Measure Voltage and Current

For this section of the lab we set up a relatively simple circuit, with our V+ supply connected in series to our red LED diode and 220Ω current-limiting resistor. This circuit is shown below in the figure below.

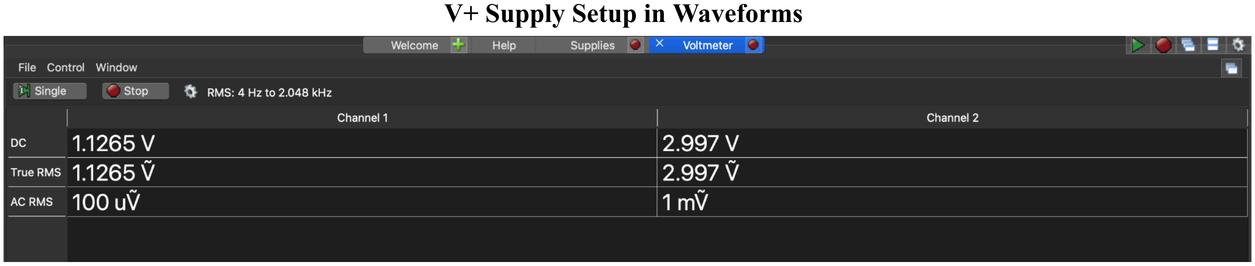


Circuit Setup to Measure VI Characteristic of Red LED

We stepped the V+ supply from 0.5V to 5V in increments of 0.5V, measuring the supply to ground voltage (channel 2) and the voltage in between the two components (channel 1) at each increment. This allowed us to solve for our LED voltage, by subtracting the resistor voltage (V_{R1}) from our real (not ideal) supply voltage. We put our data into matlab and plotted the current through the LED vs the voltage across the LED. Our measurement techniques, code, and data are shown below as well as the plot generated.



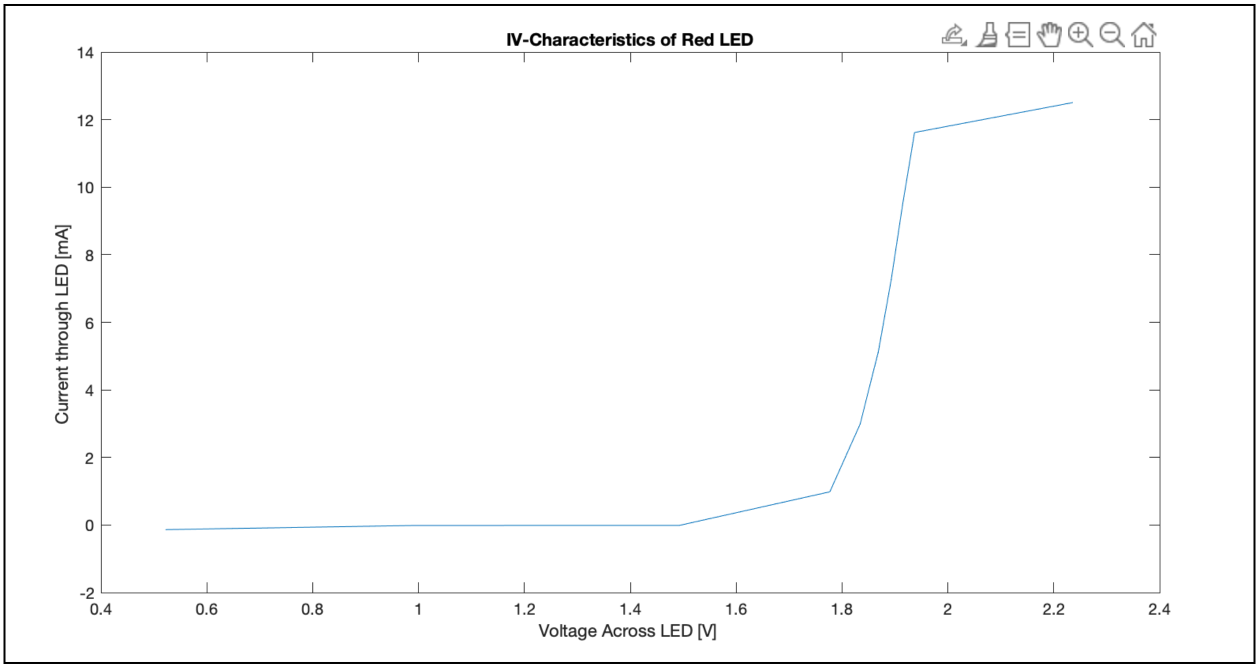
V+ Supply Setup in Waveforms



Voltmeter Data

V_1	-0.030	-0.003	-0.003	0.216	0.659	1.127	1.598	2.076	2.556	2.751
V_2	0.0492	0.992	1.492	1.993	2.493	2.995	3.491	3.991	4.493	4.987

Measurements of voltage across LED where V_1 is channel 1 and V_2 is channel 2

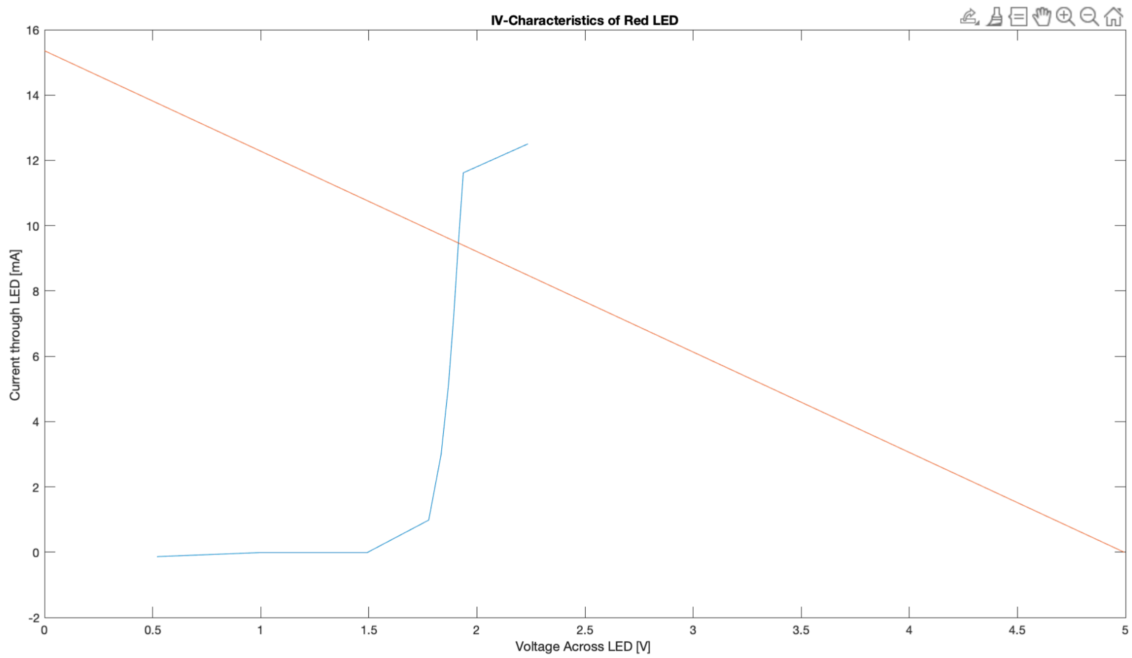


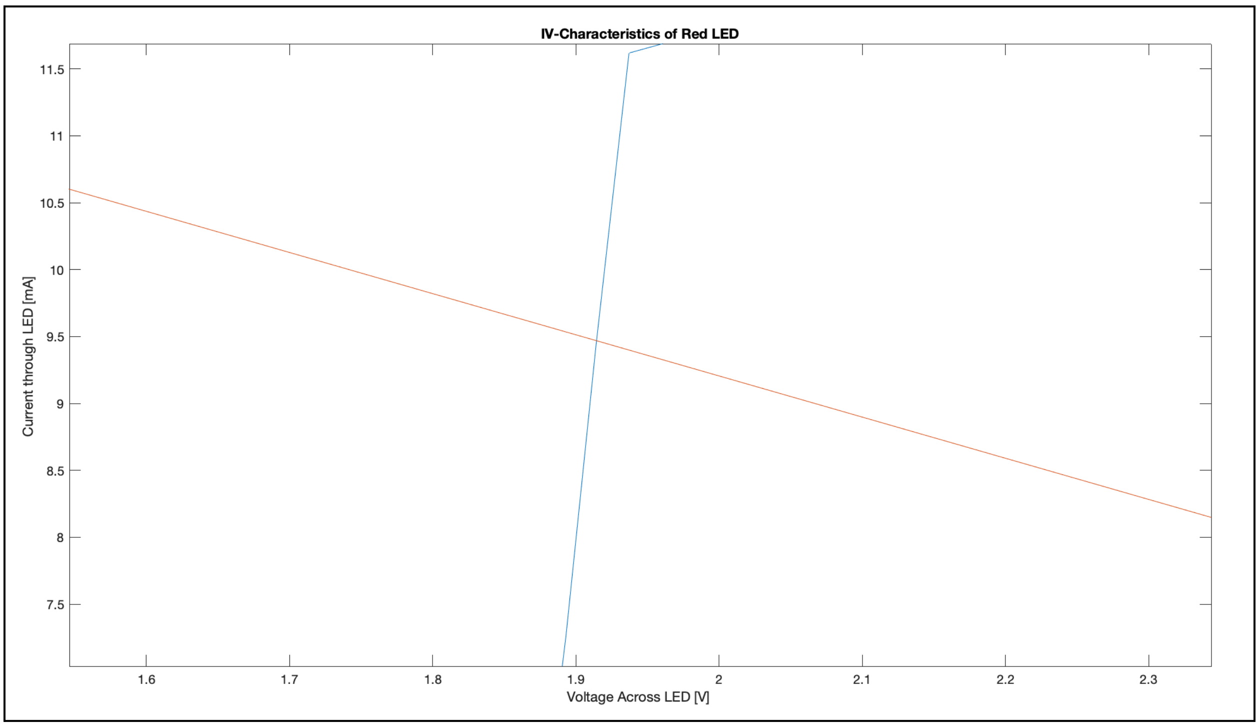
Plotting IV Characteristics of the RED LED

To measure the LED I-V characteristic with the AD2 up to 30mA we would have to use a smaller resistor in series with the LED, which would allow more current to pass from our V+ supply. However, in doing so, we would run the risk of putting too much current through the LED and damaging it.

1.A.7 Measure Resistance

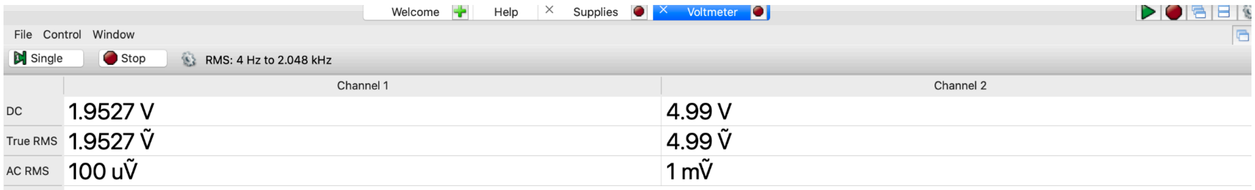
In order to measure the actual resistance of a 330 ohm resistor, R_2, we put it in series with another smaller valued resistor, R_1. We want the voltage across R_1 to be around 5mV with a source of 0.5V, which was achieved using a 4.7 Ohm resistor. It is important to note that since the accuracy of our scope is only 14-bit resolution, the voltages we measure at around 5mV isn't completely accurate. We then increment our source from 0.5V to 5V in steps of 0.5V, measuring the voltage across R_2 and current through R_2 at each voltage increment.





Zoomed In so intersection at ~1.9V and ~9.5 mA

To verify the operating point, we put our LED in series with a 300 ohm resistor and measure the voltage across it. We find that indeed the voltage across it is right around 1.9 volts. When we measured the current we got something slightly less than what we expected at around 9mA. Instead, using a DMM, we get a current of around 7mA, however this is most likely due to the inaccuracy of measurement device at such low current.

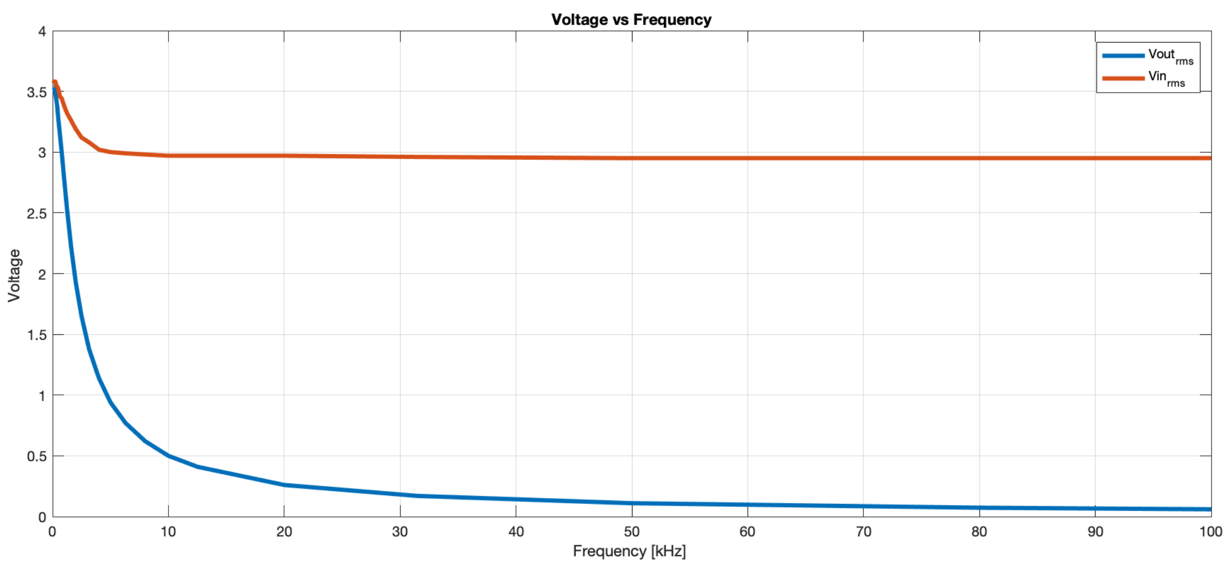


Measuring Voltage Across LED in Channel 1 and V+ in channel 2

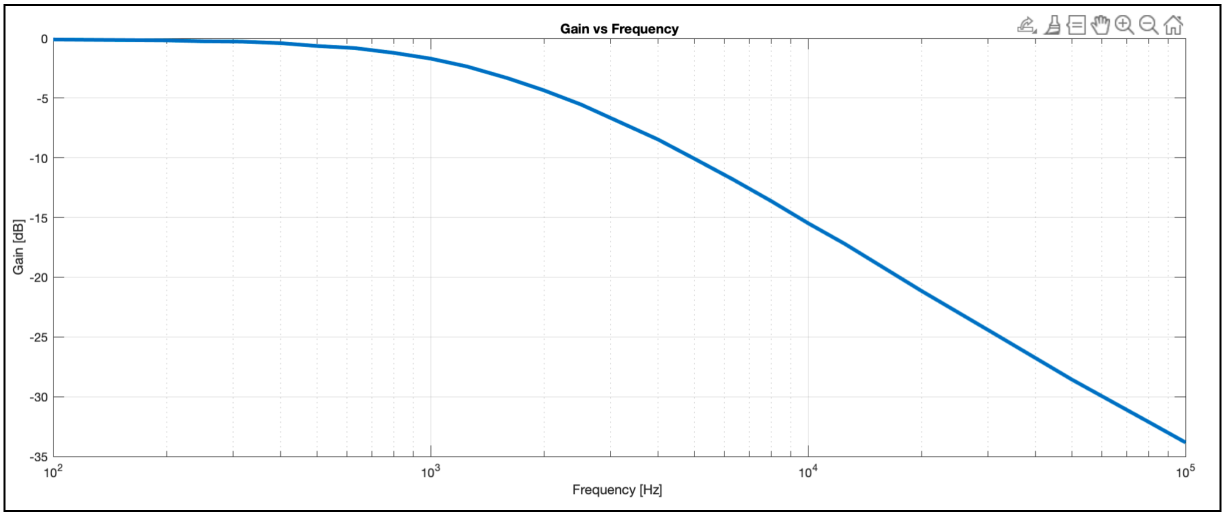
1.A.8a RC Circuit Characterization

We built a simple RC LPF with a 1kΩ resistor and a 100nF ceramic capacitor. We used waveforms to generate a 5V amplitude sine wave, and two BNC probes (10x) to measure $V_{IN,RMS}$ and $V_{OUT,RMS}$ of the filter. We measured the two voltages at frequencies from 100 Hz to 100kHz, with 27 data points shown in the figure below. We used DC RMS analysis, which should've been the same as if we used AC RMS values. Entering these measurements into matlab, we plotted voltage vs. frequency (linear), and gain vs. frequency (logarithmic).

```
F_Hz = [100, 125, 160, 200, 250, 315, 400, 500, 630, 800, 1000, 1250, 1600, 2000, 2500, 3150, 4000, 5000, 6300, 8000, 10000, 12500, 20000, 31500, 50000, 80000,
Vi=[3.57, 3.58, 3.58, 3.58, 3.58, 3.55, 3.54, 3.52, 3.46, 3.44, 3.38, 3.32, 3.26, 3.19, 3.12, 3.08, 3.02, 3.00, 2.99, 2.98, 2.97, 2.97, 2.97, 2.96, 2.95, 2.95,
Vo=[3.53, 3.53, 3.52, 3.51, 3.48, 3.44, 3.38, 3.27, 3.15, 2.99, 2.78, 2.53, 2.22, 1.93, 1.65, 1.38, 1.14, 0.94, 0.77, 0.62, 0.50, 0.41, 0.26, 0.17, 0.11, 0.073,
```



Vout and Vin vs Frequency of RC LPF on Linear Scale



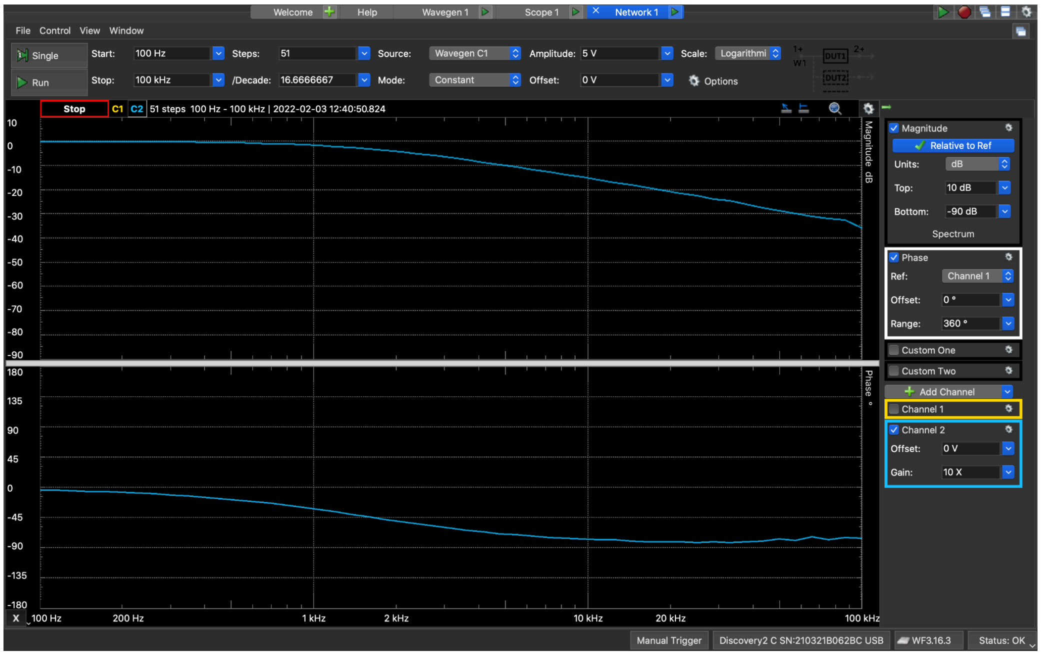
Gain vs Frequency of RC LPF on Log-Log Scale

We notice that our V_{IN} decreases in magnitude from around 3.5 V to 3 V from 0 to 5 kHz. This decrease in voltage is likely due to a decrease in the overall impedance of the RC circuit. As frequency increases the impedance of the circuit will decrease and demand more current from our voltage source. Since we are at such high frequencies with non ideal components, a high frequency AC flowing through a wire will likely create a bigger voltage drop across that wire, which is why when we measure V_{IN} at higher frequencies we get a smaller value. Ideally the interconnects of a breadboard are 0 ohms, however they are not in real life. This nonideality of our circuit alternates what we would expect in an ideal world.

This circuit is called a low pass filter because we can clearly see that as we increase frequency, our $V_{OUT,RMS}$ value decreases. Thus, the only signals that get passed through the *low pass filter*, are the ones that have a low frequency. Looking at the linear scale it is quite obvious that the magnitude of the gain significantly decreases from 100 Hz to 10 kHz, and remains less than 0.2 from then on.

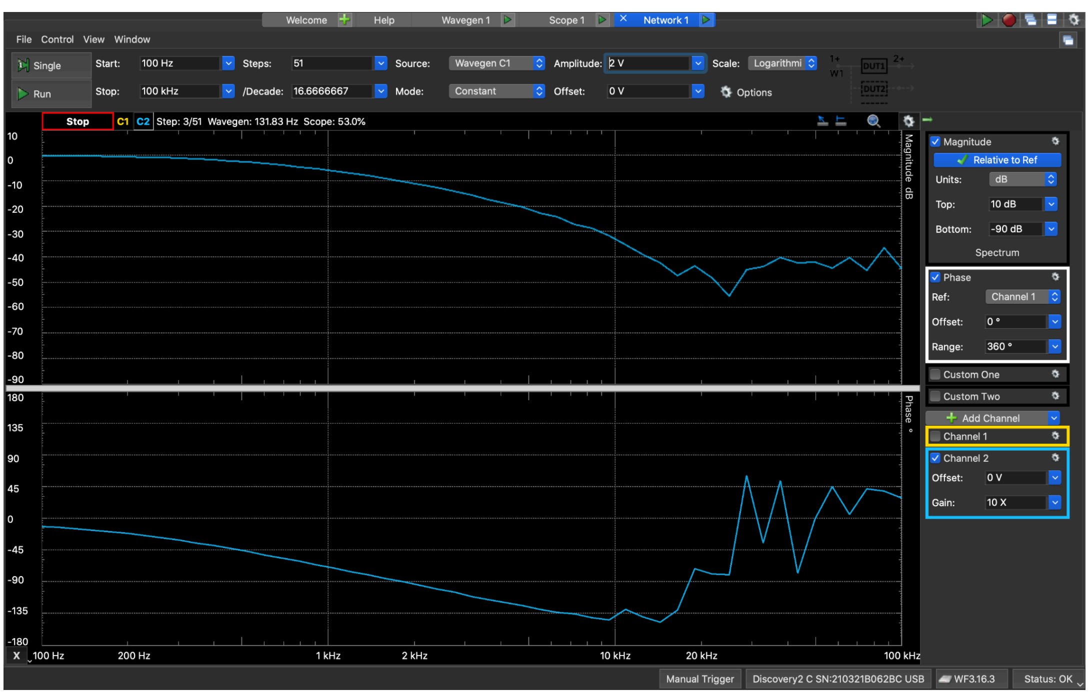
1.A.8b RC Circuit Characterization

Using our network analyzer on Waveforms, we affirm our calculations and matlab bode plot, as seen in the figure below.



Network Analyzer plot of gain (dB, top) and phase shift (°, bottom), single-stage

We also built a circuit consisting of two LPF's in series, with the same RC values. We used our network analyzer to get a plot seen in the figure below.



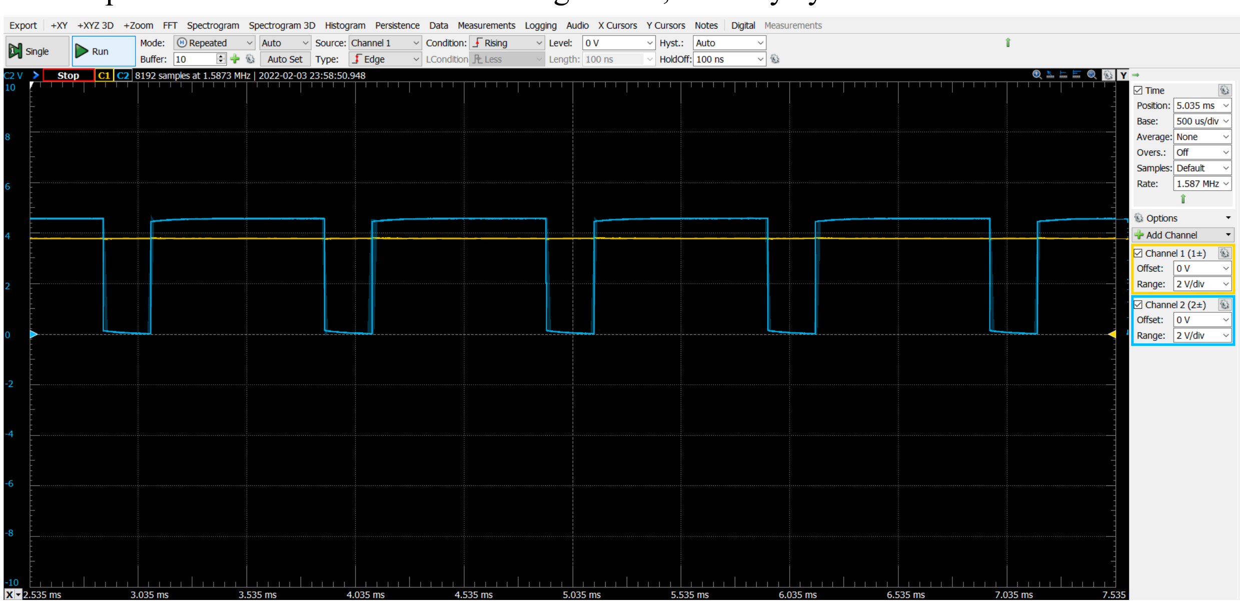
Network Analyzer plot of gain (dB, top) and phase shift (°, bottom), two-stage

We can see here that the signal $V_{OUT,RMS}$ gets attenuated at a faster rate, ideally twice as fast per decade, which is what makes the two-stage LPF twice as “good”. We ran both circuits in LTSpice, and the single LPF attenuates at -20dB/dec and the two-stage LPF attenuates at -40dB/dec.

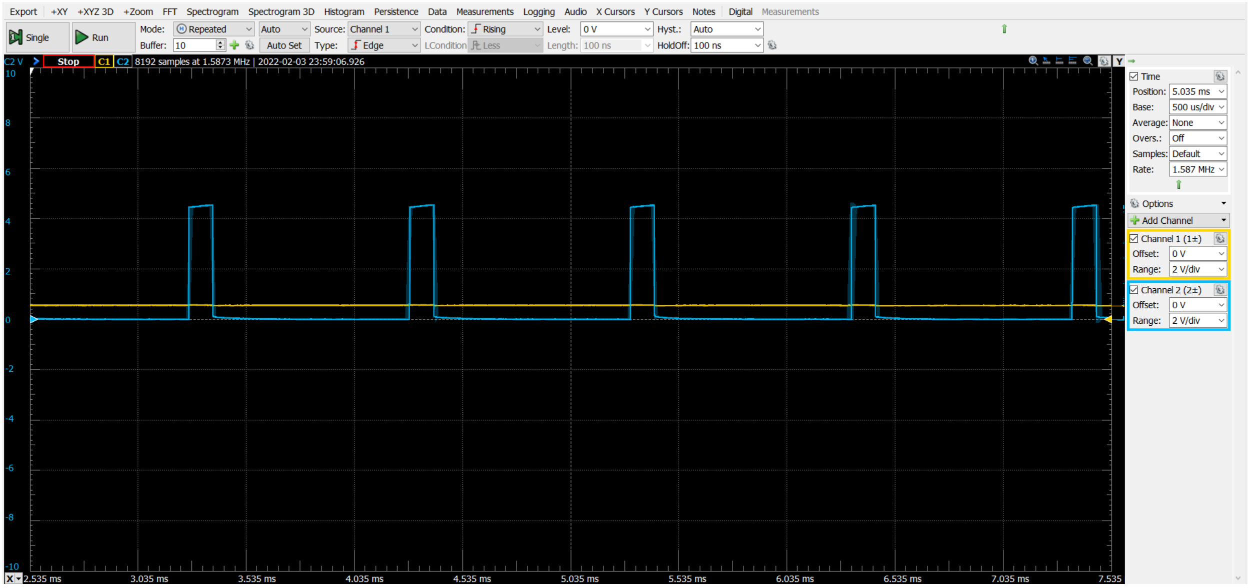
However, we can see in the graph that this isn’t what we get exactly. At about -60dB we see a spike in both the gain and phase shift. This is due to some imperfections in the breadboard, circuit layout, and AD2. It could be resolution problems, capacitance that is induced in the breadboard and in the air, and also loops of wires that could be picking up waves and inductance. Either way, we know what we expect to see, and we also know that the real world isn’t ideal, it’s messy. Nonetheless, it is evident that the second graph has a steeper slope after the cutoff frequency.

1.A.9 Arduino PWM Signal

After setting up the circuit given using the Arduino Nano Every, we use the AD2 scope and BNC Probes to measure the voltage at pin A0 and the PWM at pin D9. We notice that as we adjust the potentiometer, the duty cycle of our PWM signal changes. What is really happening is that we use the potentiometer to change the voltage at A0. Our code reads the voltage at A0 and uses this to adjust the duty cycle of our signal at D9. We can read the voltage at A0 from the serial monitor of our arduino IDE, or by looking at channel 1 of the waveforms program. It is clear that an increase in voltage at A0 also increases the duty cycle of our PWM signal. Plotting them on the same scale we see that when Channel 1 is high, the duty cycle of channel 2 is larger. As we turn the potentiometer to decrease the voltage at A0, our duty cycle decreases.

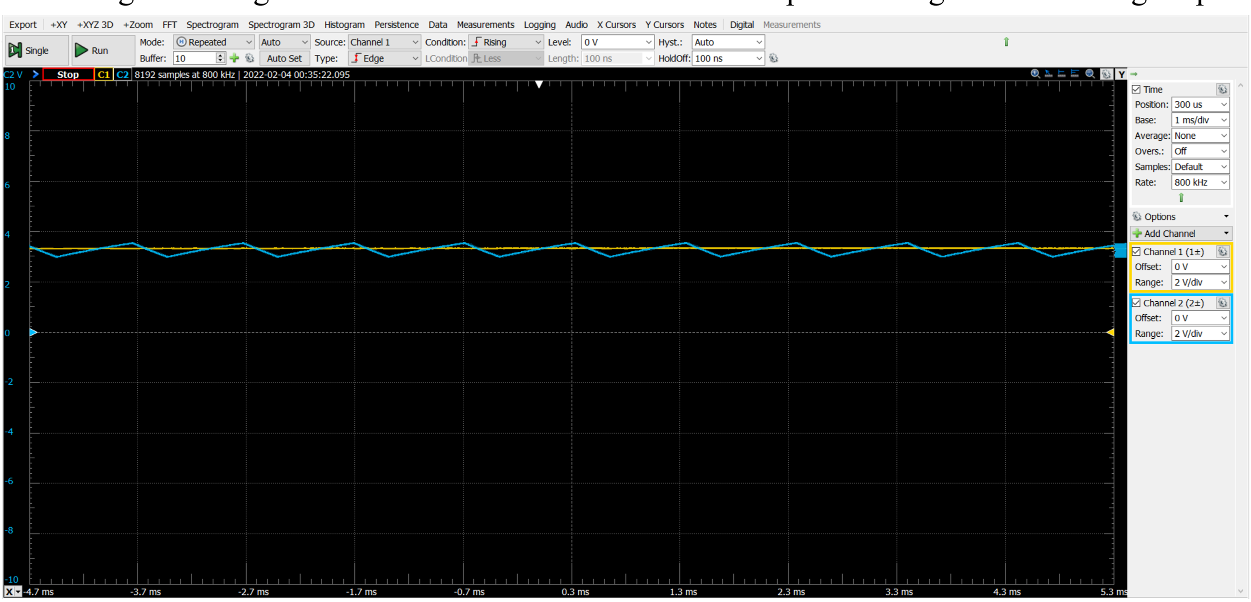


Higher Voltage at A0 with a bigger duty cycle PWM at D9



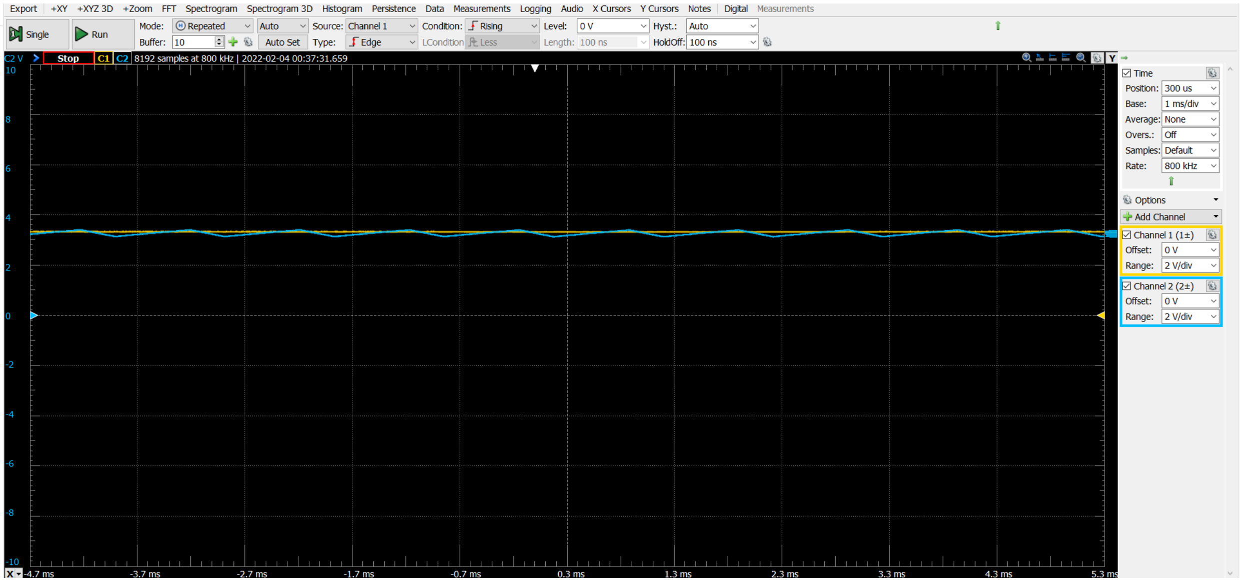
Smaller Voltage at A0 with a smaller duty cycle PWM at D9

If we wanted to convert this PWM signal back to an analog signal we could use an RC LPF. First we notice that the period of each pulse is about 1ms, so we know the frequency of our PWM signal is about 1kHz. To be safe we want our -3dB frequency to be about 100 Hz. We can get this by using the equation $f = \frac{1}{2RC\pi}$ and choosing the correct RC values. If we want to use a 100 nF capacitor we can solve for $R = \frac{1}{2\pi f C} = \frac{1}{200\pi 100e{-9}} \approx 16000\ \Omega$. We can round resistor value up because it will only produce a lower cut off frequency, so we round up to a $22k\Omega$ resistor. Measuring the voltage at the node between the resistor and capacitor we get the following output.



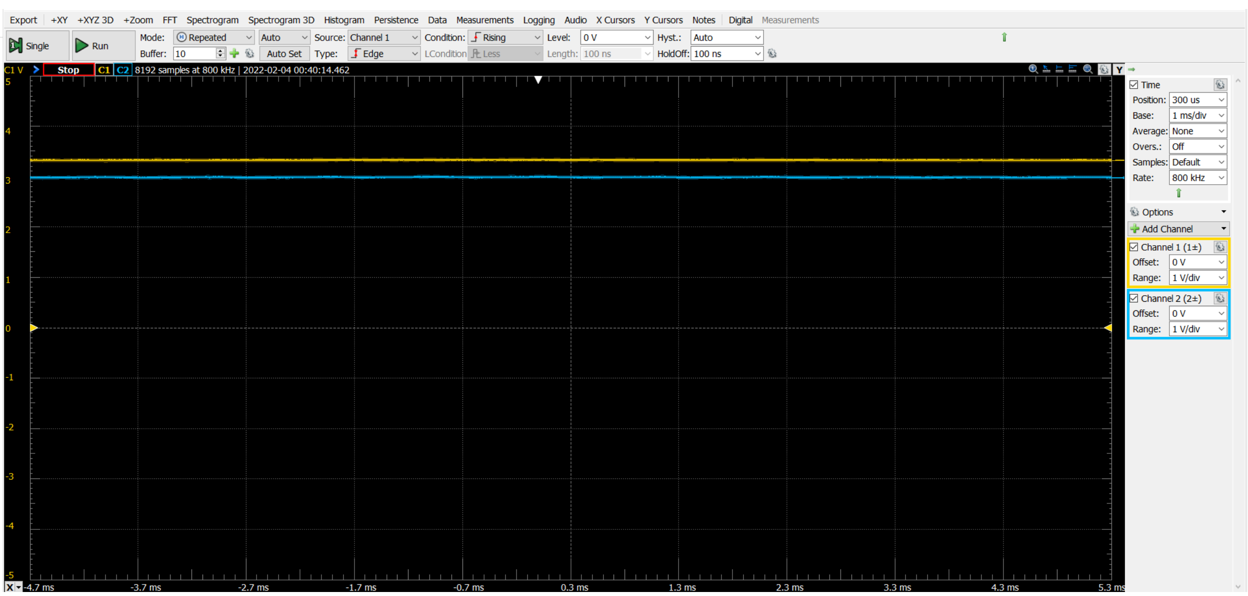
Using RC LPF to convert PWM to DC with R = $22k\Omega$ and C =100nF

As we adjust the potentiometer, the voltage from channel 2 follows the voltage from channel 1, which is what we expect. If we instead use a resistance of $47k\Omega$, we can flatten out the waveform generated at channel 2. This is shown below at the same voltage level.



Using RC LPF to convert PWM to DC with R = $47k\Omega$ and C =100nF

We could even use a 1MΩ to get a seemingly flat waveform, however this will increase the voltage drop to a much more noticeable level at our output voltage.



Using RC LPF to convert PWM to DC with R = $1M\Omega$ and C =100nF

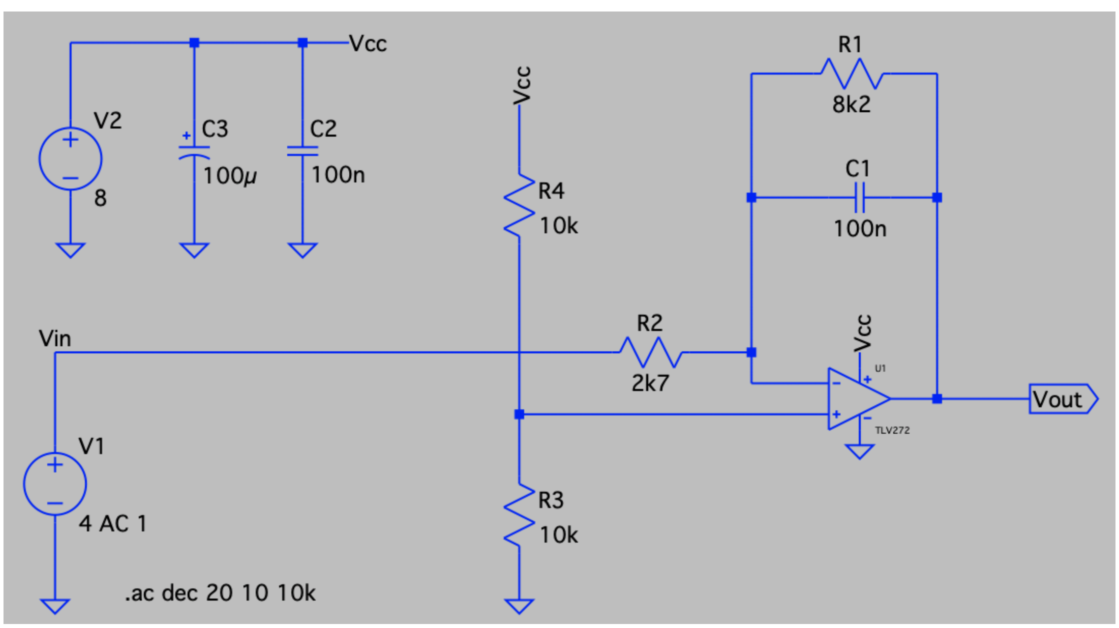
1.B.1 Prelab

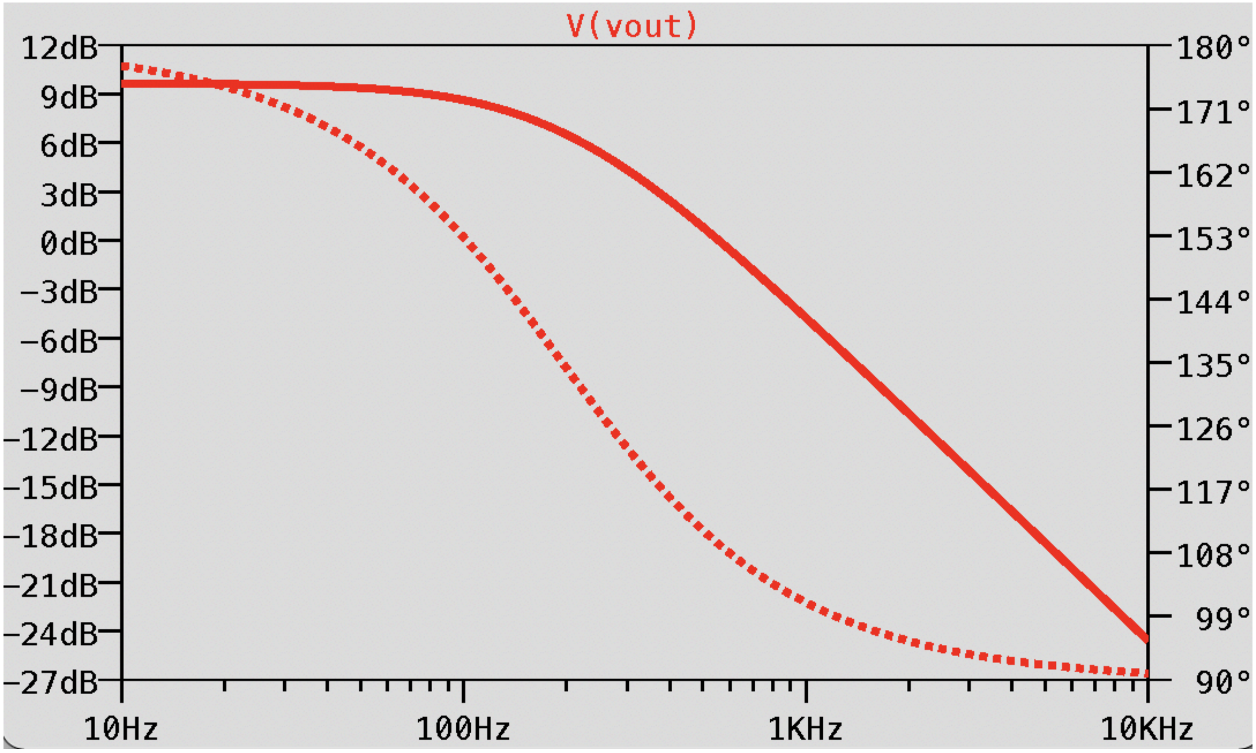
Lab exploration topics:

1. When the LMC555 is in astable mode the oscillation and duty cycle are set by two external resistors and one capacitor. The astable mode produces a series of rectangular pulses vs the monostable mode which generates a single pulse. The pulse begins when the trigger input is ⅓ of the supply voltage and stops when it reaches ⅔ the voltage supply.
2. The formula given is derived from the time to charge the capacitor as well as discharge the capacitor. To find the period we sum the charge time and fall times, which gives $0.693(R_a + 2R_b)C$. To find the frequency we take the inverse of the period, which then yields $1.44/(R_a + 2R_b)C$.
3. According to our waveforms analysis of the circuit, our duty cycle is 60%, meaning that it is high for 60% of the duty cycle and off for 40% of the cycle. Our timer is operating in astable mode so we know this measured cycle percentage is for astable mode.
4. No, because the capacitor charges and discharges between ⅓ and ⅔ of the supply voltage. Thus the actual supply voltage value doesn't affect frequency, it's merely decided on the ratios of different terminals in relation to the supply voltage.

1.B.2 Test your LTSpice Installation

After building the circuit given for 1.B.2 we ran an AC analysis simulation from 10 Hz to 10kHz. The output is shown below in the figure below.



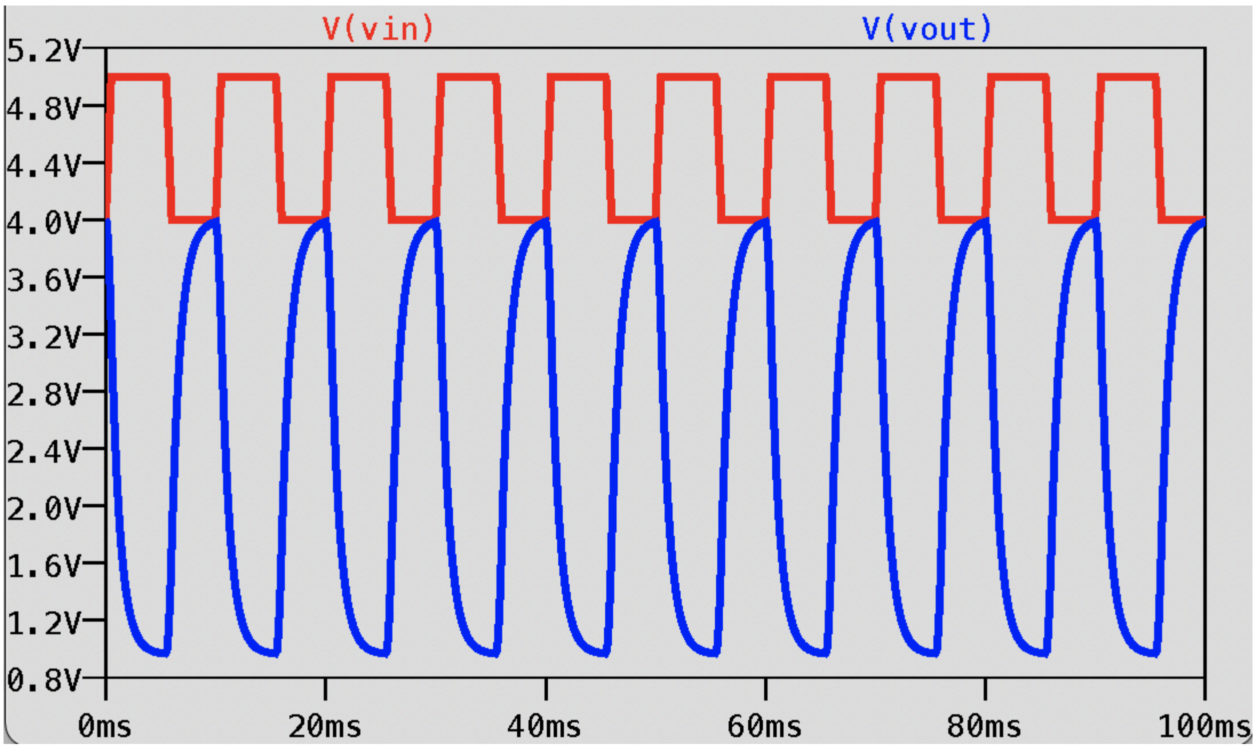


Bode Plot for AC Analysis simulation from 10 Hz to 10 kHz using a Decade sweep.

After running our LTSpice simulation we notice that our circuit acts as a low pass filter. Our DC gain value is essentially our starting value in dB. At DC, our op amp integrator circuit with DC gain control essentially acts as an inverting op amp. We take the ratio of our resistor to get our DC gain. So our gain is $-\frac{8200\,\Omega}{2700\,\Omega} \approx 3.037$. We confirm this by taking the magnitude of this and converting it to dB. We find that $20log(3.037) \approx 9.65dB$, which is exactly where our plot starts. The -3dB frequency would be our starting gain value which is 9.65 dB - 3 dB = 6.65 dB, which has a frequency of about 194 Hz. We can confirm this by using the equation $f = 1/(2\pi RC) \approx 194$. We can also measure the magnitude change per decade after the cutoff point, testing the decibels at 1 kHz and 10 kHz. The respective decibels were -4.76dB, and -24.6dB. These are almost exactly 20 apart, indicating that after the cutoff point there is a change of -20dB/decade. This is exactly what we expect for a circuit of this configuration.

In the circuit shown in the figure below, R_3 and R_4 are a voltage divider that drops the voltage source V_{CC} down to 4V at the node that goes into the non-inverting side of the opamp. This is useful because with an ideal op amp, the voltage on the non-inverting and inverting inputs are 4V, and with an AC voltage source with an offset of 4V, the voltage drop across R2 would be the same as if we had the non-inverting and inverting inputs grounded and a 0V offset AC voltage source.

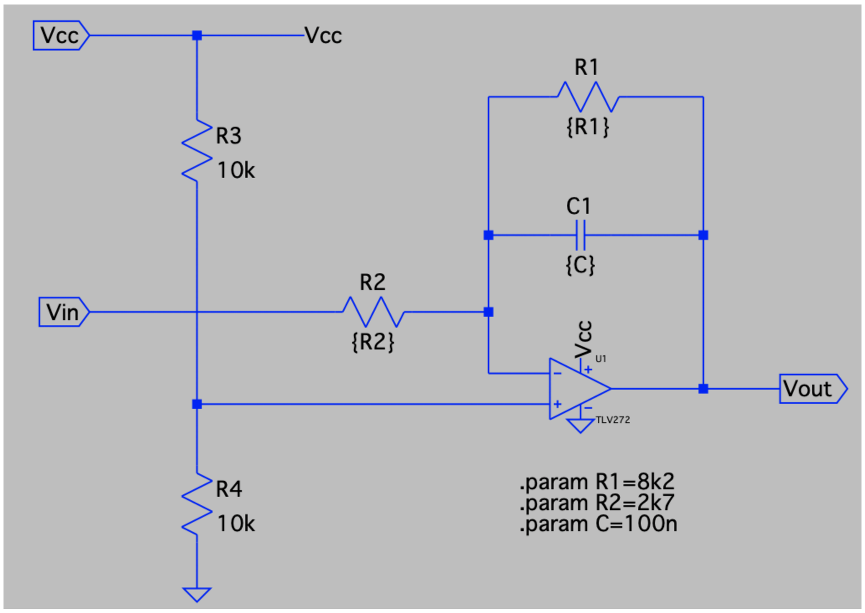
After running a transient analysis using a 100 Hz 50% duty cycle waveform with an amplitude of 1V and an offset of 4 V, we attain the graph shown in the figure below. The output waveform at V_{OUT} is an exponentially decreasing voltage when V_{IN} is high and then an exponentially increasing waveform when V_{IN} is low.



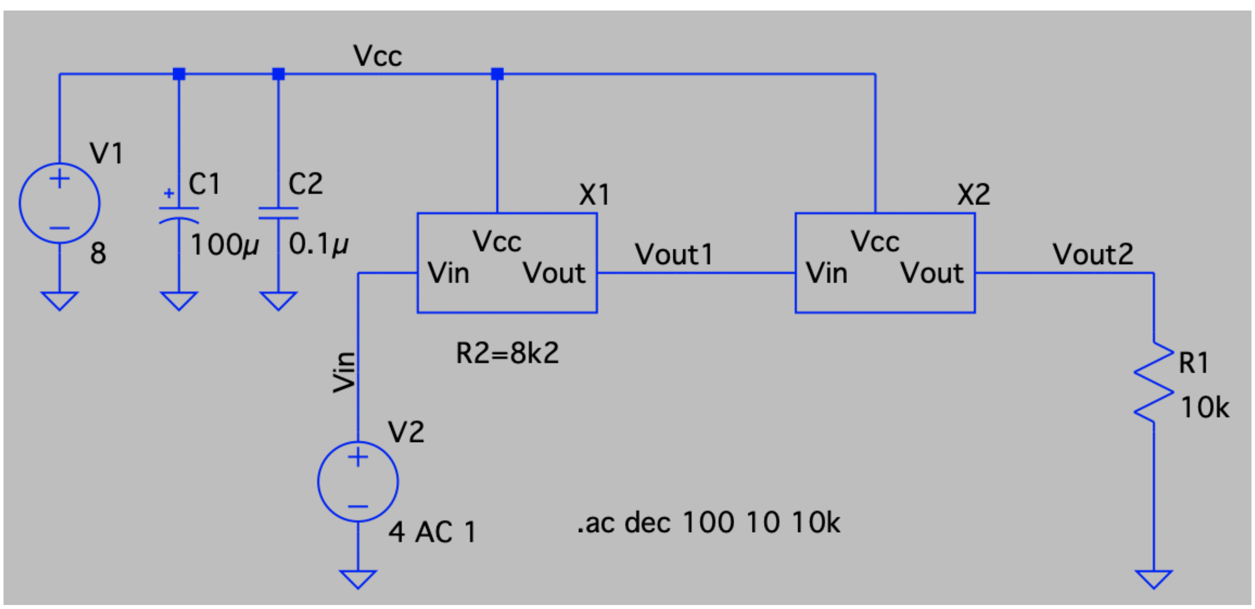
Transient Analysis of 100 Hz Rectangular Waveform with 50% Duty Cycle, 4V Offset, 1V Amplitude

1.B.3 Op Amp Filter LTSpice Circuit Model

The Op Amp filter circuit from part 1.B.3 can be saved as an .asy file to allow for it to be represented by a more manageable block in future LTSpice circuit models. For example, a circuit can be constructed with two Op Amp filters connected in series with the same Vcc power supply. This is the general setup of the circuit shown below.

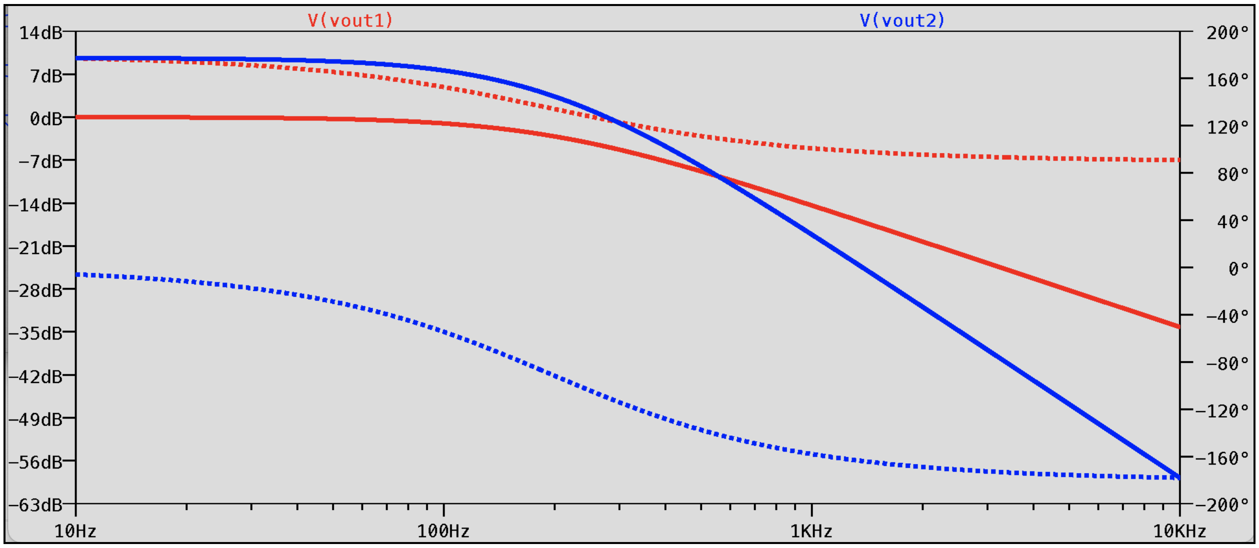


Subcircuit Diagram of Op Amp Filter with R1, R2, and C as Parameters



Circuit Diagram of Two Op Amp Filters in Series, X1 block with 8.2 Value for R2 Overriding Default Value

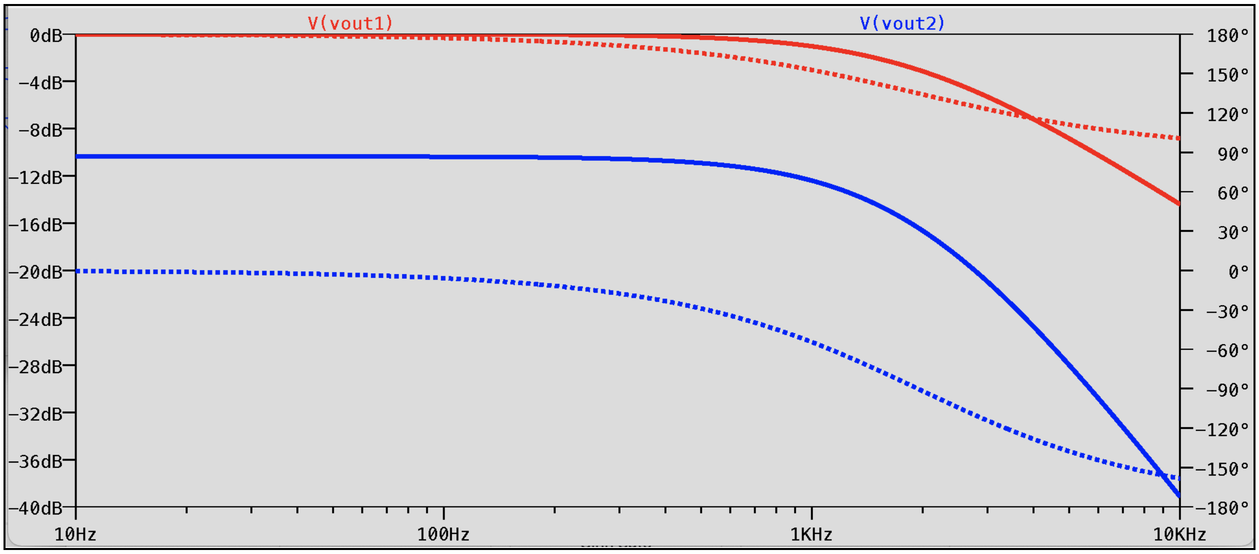
Now that the circuit has been constructed with an adjustment on the R2 value for Op Amp Filter X1, and default values for X2, the characteristics of the circuit at $V_{OUT,1}$ and $V_{OUT,2}$ can be done.



Primary Output for Two OpAmp Filters in Series

Looking at the basic aspects revealed by this Bode plot, it can be seen that the op amp X1 initially filters to 0 dB, with a cutoff frequency of 196 Hz, then trailing off at -20dB/dec. This makes sense as X1 has a -20 dB frequency of about 1.9 kHz. X2 on the other hand has an initial magnitude of 9.6 dB and a cutoff frequency of 126 Hz, before trailing off at a larger rate of -40 dB/dec. Again, this makes sense by looking at the -20 dB frequency, which falls at about 1.2 kHz.

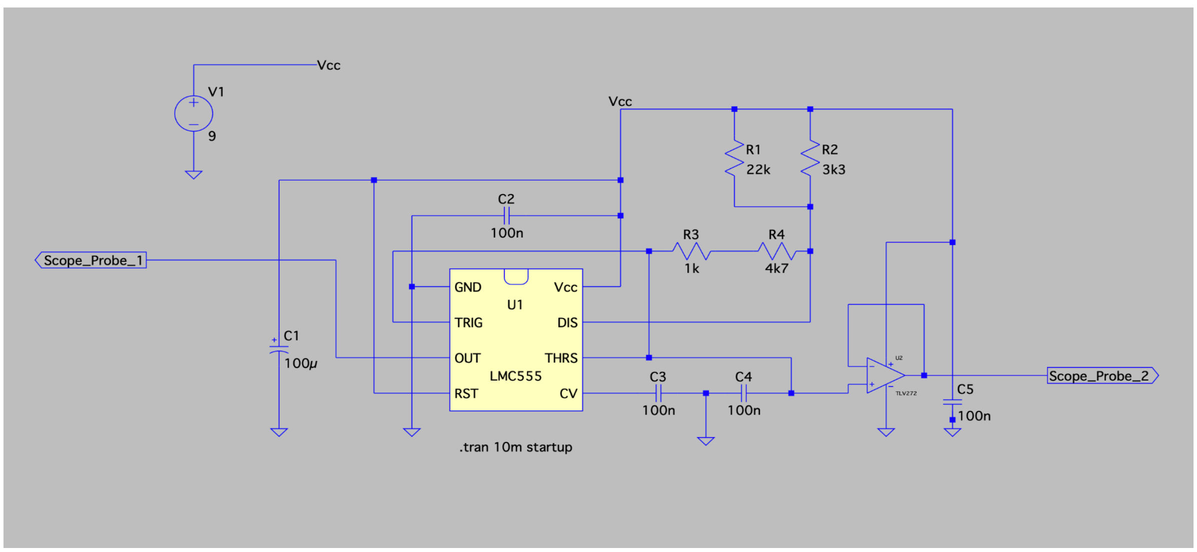
The value for R_2 of the first op amp filter block is $8.2k\Omega$ so that our DC gain at $V_{OUT,1}$ is just 1. Our DC gain is equal to the ratio of the resistors, so with same valued resistors we get a gain of 1, which is 0 dB. If we kept our original value of 2.7 $k\Omega$ the DC gain would be approximately 3.04. This gain would then have a gain of its own, which would amplify the original gain of 3.04 by 3.04 which would be a total gain of about 9. The dB value of this would be 20log(9) = 19.3 dB. This is a huge gain especially when we are using an input voltage of 4V. Instead, we use the R_2 value of $8.2k\Omega$ so that the gain at our 2nd output is still 9.6dB (as it was in section 1.B.2). We use the DC gain equation of an inverting opamp to verify our results. With $R_1 = R_2 = 8.2\ k\Omega$, we know that their ratio is 1 and the dB gain should be equal to 0. This is confirmed when we look at our output waveform. To make our -3dB frequency for $V_{OUT,1}$ 10 times larger, we can divide either our resistance or capacitance by 10, since they are inversely related. We choose to divide our capacitance by 10 and now use a 10nF capacitor and 8.2 $k\Omega$ resistor in parallel across the opamp. Similarly, to increase the -3dB frequency for $V_{OUT,2}$ by a decade, we can decrease the resistance or capacitance by a factor of 10. Output for this model is shown below where we use a resistance of 820Ω for R_1 and R_2 on the first filter and 820Ω for R_1 on the second filter.



Second output of Two OpAmp Filters in Series with -3 dB Frequency 10 Times Higher

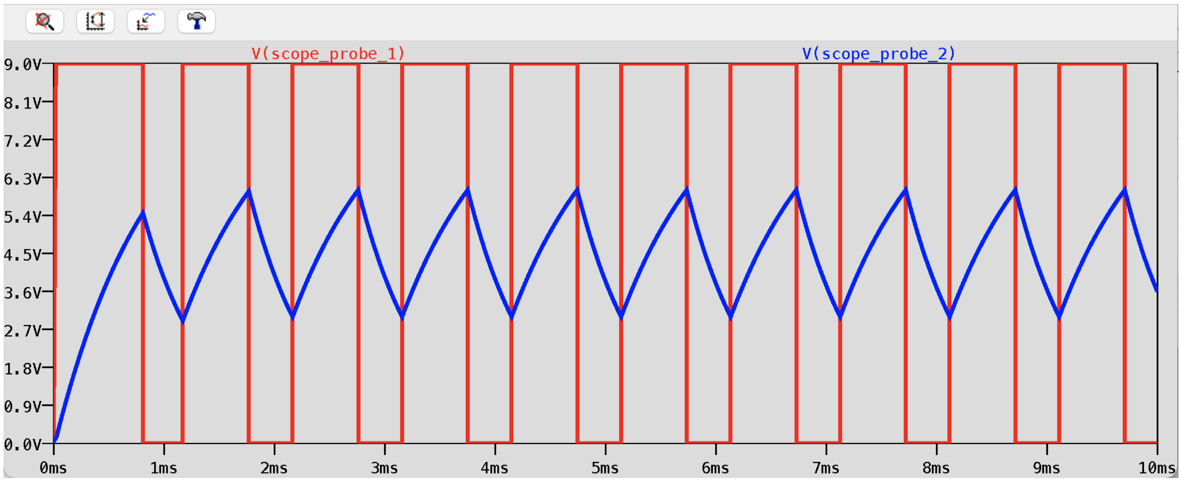
1.B.4 Enter Circuit from Pre-Lab in LTSpice, Simulate

Shown below is our LTSpice schematic modeled after the breadboard circuit given.



Pre-Lab Circuit in LTSpice

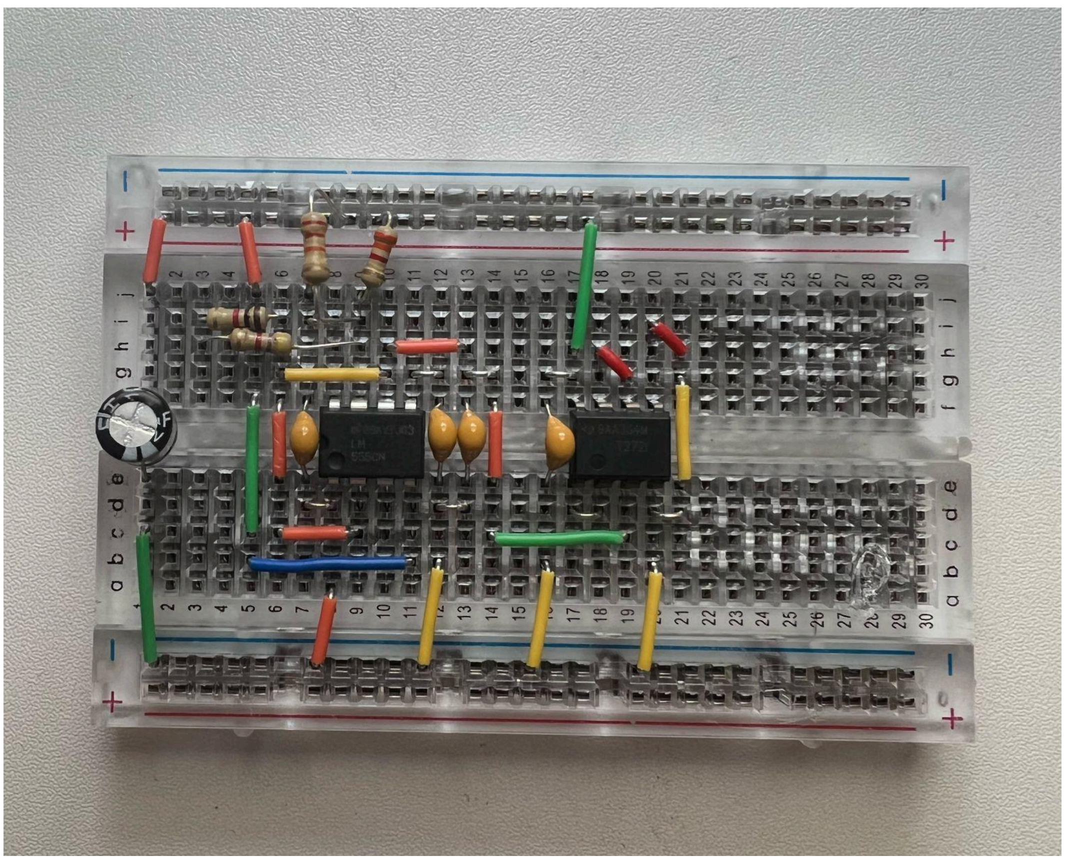
Once again, a transient of the circuit can be done to observe the important characteristics at circuit startup and as it approaches DC steady state.



Simulated Output of Pre-lab Circuit

This voltage output makes sense looking at what components are used for the circuit. The LMC555 op amp acts as a clock voltage, so it makes sense to have a rectangular waveform. Additionally, the charging and discharging waveform offers a good idea of what to expect when looking at this same circuit constructed on a breadboard.

1.B.5 Construct the Circuit on the Breadboard

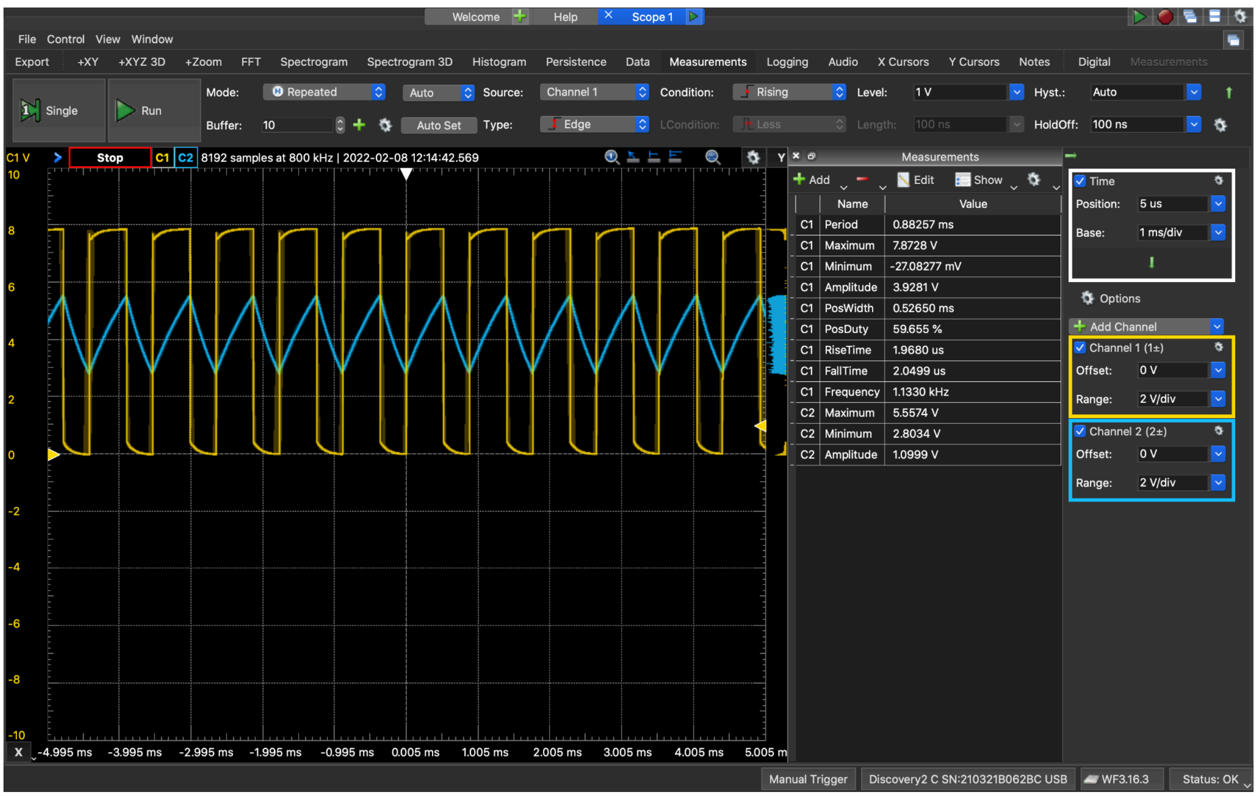


Our Groups Breadboard Circuit for 1.B.5

1.B.6 Test the Circuit Using the AD2

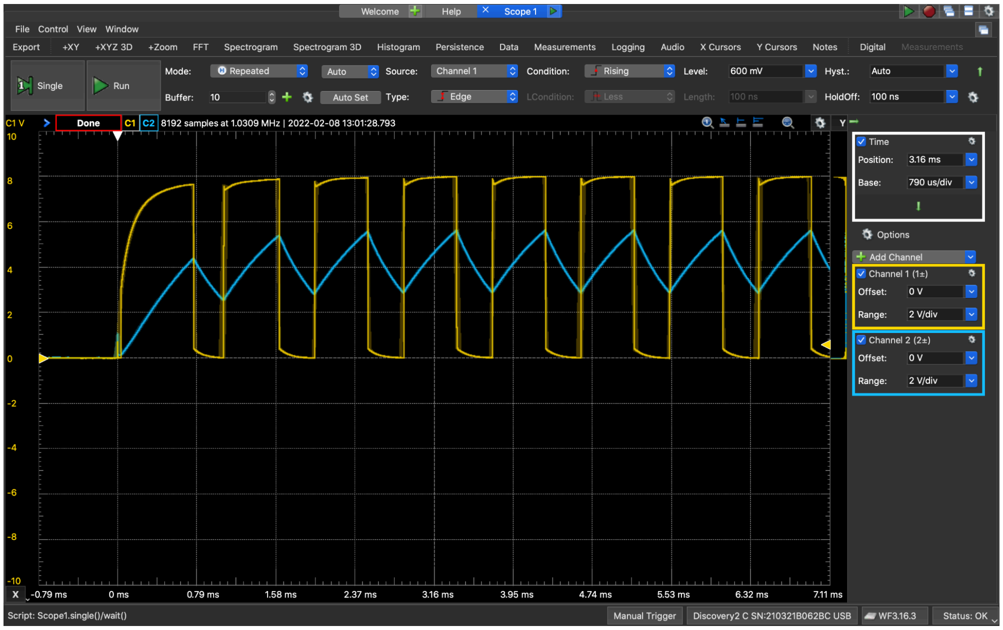
To test the circuit, we used a 9V alkaline battery. We used our AD2 BNC probes to measure our output voltage waveforms, and took measurements directly on the Waveforms software.

We had little to no trouble debugging, one thing we noticed and had some trouble with was that two members had their signal minimum at ~ -4V, and the other had his minimum at 0V. The amplitude was the same, they were just offset differently. Since our LTSpice analysis had the minimum at 0V, we figured that was what our signal should look like, especially given our circuit setup of 0V at the ground rail and ideally 9V at the power rail. The problem occurred because two of us had our BNC attachments set on AC analysis, instead of DC analysis. Once we switched the pins to DC on the AD2 BNC attachment, our signals all matched the spice graphs.



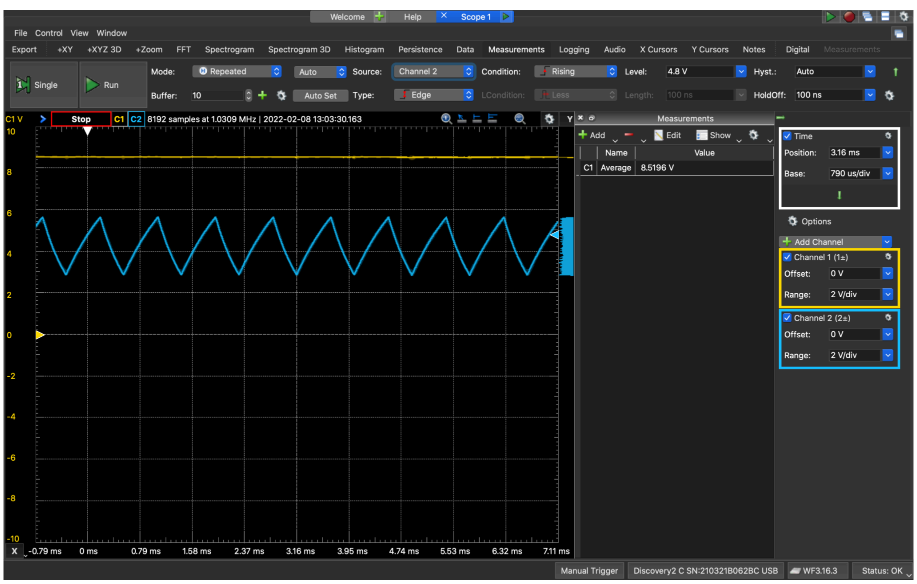
Circuit output voltage waveforms

We then analyzed the two output waveforms in the steady state domain as well as the transient domain. We were able to capture the transient response by using the single acquisition control, with a low trigger value, so that when we powered up our circuit Waveforms automatically captured the transient response. The trigger value had to be in a specific range, because if it was too low it would get triggered by noise, and if it was too high we would miss part of the transient signal.



Transient response of 1.B.5 Breadboard Circuit

After we captured the transient response we probed the V_{cc} power supply as well as the pulsating output waveform.



Supply voltage V_{cc} and output waveform

This voltage reading makes sense since the V_{cc} power supply is a constant DC value of 9V.

All of these readings and measurements have helped to verify that our computer simulation of the prelab circuit and constructed circuit do indeed match what we expected.

Conclusion

The main concepts explored in this lab had to do with learning the ins and outs of basic electrical lab measurement tools. First we investigated the properties and got familiarized with the Waveforms software and the Analog Discovery 2. Through these tools we were able to learn how to generate waveforms of sinusoidal, pulsating, and custom voltage waveforms.

Next we learned how to measure real world circuits through oscilloscope probes. Ideas that were important for this part of the lab were how to properly trigger the desired channel, add a measurement, and adjust the oscilloscope settings to make sure the measurements were of proper resolution. We learned that an oscilloscope cannot directly find current through a component, rather we have to add in resistors to indirectly solve for current. We also used the waveforms software to directly analyze frequency responses through the network analyzer.

We applied these practical understandings to the provided LMC555 timer circuit. We first simulated this circuit in LTSpice, to determine our desired output voltage waveforms. We then practiced proper breadboard design skills to build this circuit, and then used our AD2 BNC probes to measure the real world output. We had some struggles with incorrect readings, which we knew were not correct because of our LTSpice models. In conclusion, we were able to simulate, build, and analyze the LMC555 timer circuit because of ideas that were built upon throughout the entire lab. Overall through the course of the lab, we learned a lot of useful techniques for circuit measurement, and got some good practice with simulating, then breadboarding and testing more complex circuits. These practices will continue to help us through future lab reports.