# LTSpice and Lab Orentation – Instruments and Measurements

# Lab 1

EEE 334
Circuits II

Instructor: ***Dr. Leslie Hwang***


| | |
|---|---|
| Student Name: | ***Matthew Heusmann*** |
| Student ID: | *********** |


| | |
|---|---|
| Report Completed on: | ***31/08/2023*** |


Arizona State University

# INTRODUCTION

This labs' main focus is to allow the student to become more comfortable with analyzing various circuits such as simple DC, simple AC, Low Pass Filter, and lastly High Pass Filter. This lab allows the student to understand the basics of each of these circuits while getting valuable practice utilize the ADK software, LTSpice Software, physically building circuits, and practice with a handheld multimeter.

**Figure 1 was removed due to just being a picture of myself.**

# EQUIPMENT and COMPONENTS USED

For this lab the following items were used to conduct the experiments and findings:

- *Digital Multimeter*
- *ADK (Analog Discovery 2)*
- *Large Breadboard*
- *Analog Parts Kit*
- *TI-nspire CX II Calculator*
- *2x 1kΩ resistor, 3x 2.2kΩ resistor, 2x 10kΩ resistor, 20kΩ resistor, 0.01µF capacitor*

# Course of Action

This section covers the processes that need to be done for each part. There will be a detailed description of what needs to be found and accomplished. The completed results mention in this section will be found in the results section.

## 1.1 Simple DC Circuit

| | |
|---|---|
| VS1 | 3V DC |
| VS2 | 4V DC |
| R1 | 2.2kΩ |
| R2 | 2.2kΩ |
| R3 | 2.2kΩ |

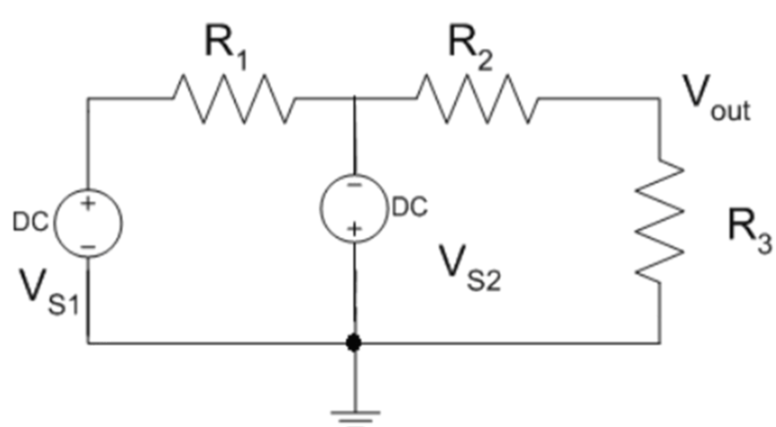



*Figure 1 Simple DC Circuit*

This part focuses on analyzing the circuit using appropriate methods such as Superposition, Thevenin, or whatever way that will give the results for the unknown parts of this circuit. The main focus is to find the current through $R_2$ and the $V_{out}$ voltage. With these found then the circuit will be simulated in LTSpice to find the supposed values the circuit should contain. To finish up this part this circuit will be built physically and measured using the ADK (Analog Discovery Kit) to verify the simulated results from LTSpice. Once results are taken the error will be calculated to see how close or how far off the simulated results are from the physical results.

## 1.2 Simple AC Circuit

| | | |
|---|---|---|
| **VS** | **2Vpp** | **f = 100Hz** |
| **R1** | 1kΩ | |
| **R2** | 1kΩ | |
| **R3** | 10kΩ | |
| **R4** | 10kΩ | |

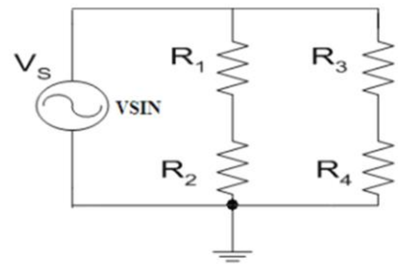



*Figure 2 Simple AC Circuit*

For 1.2, this part is similar to 1.1 but now it is a simple AC circuit that will be analyzed. The process that will be taken, is first completing hand calculations determining what the current will be through $R_2$ and $R_4$. Then hand calculating the voltage across $R_1$, $R_2$, $R_3$, and $R_4$. Once the previous values have been found $V_{rms}$ will be defined, and hand calculated for $R_2$.

Next a simulation of this circuit will be built in LTSpice to verify the hand calculated results and the simulation circuit will be displayed along with the graphs of the desired results.

Lastly, the simulated circuit will be physically built and tested/compared with the hand calculations along with the simulated results. The circuit will also be tested against a Triangular wave, and Square wave. These different waves will be cross checked across multimeter readings along with the oscilloscope function on the ADK. The items being looked at here will be the Voltage waveforms across $R_2$ and $R_4$ and the $V_{pp}$ along with the $V_{rms}$ of $R_2$ and $R_4$. All of these results will be checked against hand calculation, simulation on LTSpice, and physical circuit to obtain the percentage error between all of these.

## 1.3 Low Pass Filter Analysis

$$\frac{V_o}{V_i} = \frac{1}{j\omega RC + 1} \qquad (1.1)$$

| | | |
|---|---|---|
| VS | 2 Vpp | f=10kHz |
| R | 1kΩ | |
| C | 0.01µF | |

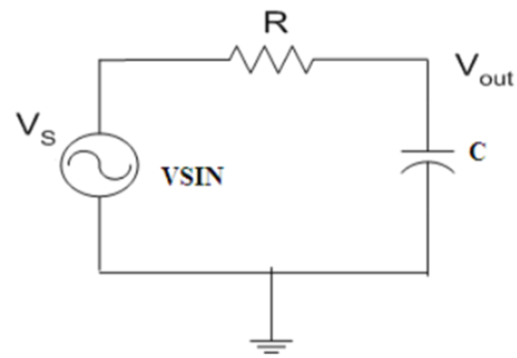



Fig.1.3 Low Pass Filter Network

*Figure 3 Low Pass Filter*

This section focuses on the basics of understanding and analyzing a simple low pass filter. Similar to how the last two sections were laid out this section will be the same. First, the voltage gain of the filter will be calculated using equation (1.1), and the $\omega_{3dB}$ will be determined through the formula $\omega_{3dB} = 1/RC$. Subsequently, the circuit will be simulated in LTSpice. A Transient Analysis will be done to find the anticipated gain and phase shift of the circuit, along with the max gain in dB. Then, an AC Analysis will be conducted to predict the frequency response of the circuit’s gain and phase.

For the hands-on portion, the circuit depicted in Fig 4 will be constructed on a breadboard. Before assembly, the values of the resistance and capacitance will be confirmed using a multimeter and listed in the results. Once verified, the ADK Function Generator will be set up to produce a 2 Vpp sine wave at 10 kHz. Each of the ADK oscilloscope channels will then be placed to monitor specific points: Channel 1 at VS and Channel 2 at Vout, ensuring accurate readings of the waveforms, including Vpp, Vrms, and the Phase Shift of Vout.

In the final steps, oscilloscope readings from the ADK will be used to measure and calculate the circuit's gain and phase. Additionally, the ADK's network analyzer will be employed to gauge the frequency response of the circuit’s gain and phase. Lastly, the percentage error will be taken to compare the LTSpice results to the physical results.

## 1.3 High Pass Filter Analysis

$$\frac{V_o}{V_i} = \frac{1}{(1/j\omega RC)+1} \qquad (1.2)$$

| | | |
|---|---|---|
| VS | 2Vpp | f=1kHz |
| C | 0.01uf | |
| R | 20 kΩ | |

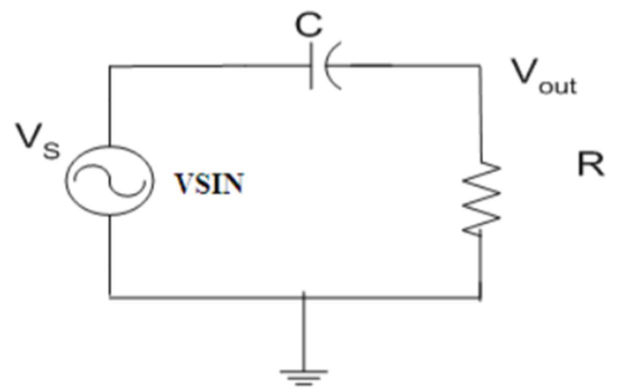



Fig.1.4 High Pass Filter Network

*Figure 4 High Pass Filter*

As for the last section, this will be focusing on the basics of understanding and analyzing a simple high pass filter. This will be very similar to the low pass filter with some differences. First, The voltage gain of the filter associated with figure 5 will be determined by using the equation (1.2). Simultaneously, the $\omega_{3dB}$ will be hand calculated from the equation $\omega_{3dB}$ = 1/RC. Following this step, a simulation of the circuit will be tested in LTSpice. A Transient Analysis will be taken to predict the gain and the phase shift between Vout and Vin. Then an AC Analysis will be examined to offer insights into the frequency response of the circuit, particularly focusing on the gain and phase of this circuit.

During the hands-on section, the schematic shown in Fig 4 will be physically constructed. Prior to full assembly, the resistance and capacitance component values will be taken using the multimeter. The ADK Function Generator will then be set to generate a 2 Vpp sine wave at 1 kHz. To ensure precise data, the ADK oscilloscope will be set with Channel 1 at VS and Channel 2 at Vout. With this configuration the measurements requested from the lab manual such as Vpp, Vrms, and the Phase Shift of Vout can be taken and logged in the results section.

Lastly, the oscilloscope readings from the ADK will be used for measuring and calculating the circuit's gain and phase. Then, the ADK's network analyzer will be used to graph out the frequency response of the circuit's gain and phase over a bandwidth ranging from 10Hz to 100kHz. Along with percentage errors will be taken to compare the LTSpice results to the physical results.

# RESULTS

## 1.1 Simple DC Circuit

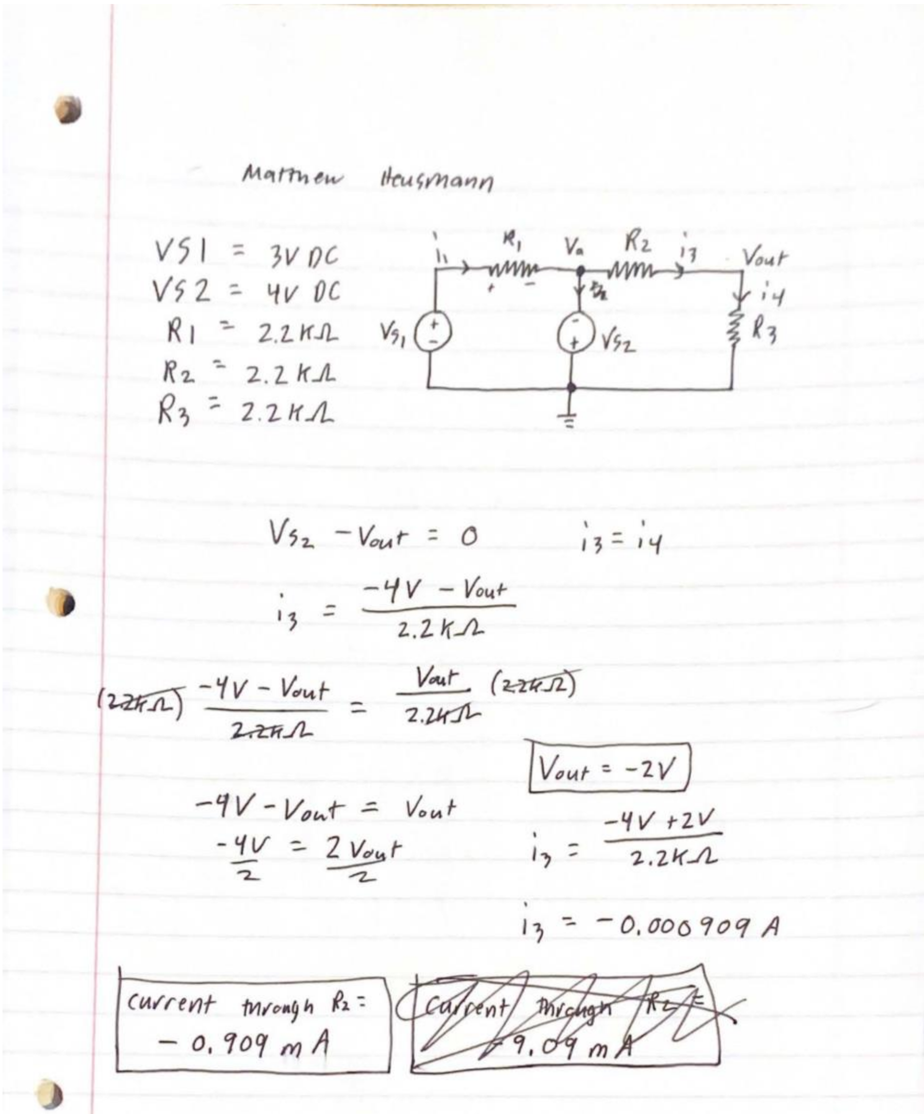


*Figure 5 Simple DC Circuit Hand Calculation*

From the figure above it can be seen that the hand calculations came out to be $\mathbf{V_{out} = -2V}$ **DC** and the **current at $\mathbf{R_2}$ to be -0.909mA**. When comparing this hand calculation to the LTSpice simulation shown below:

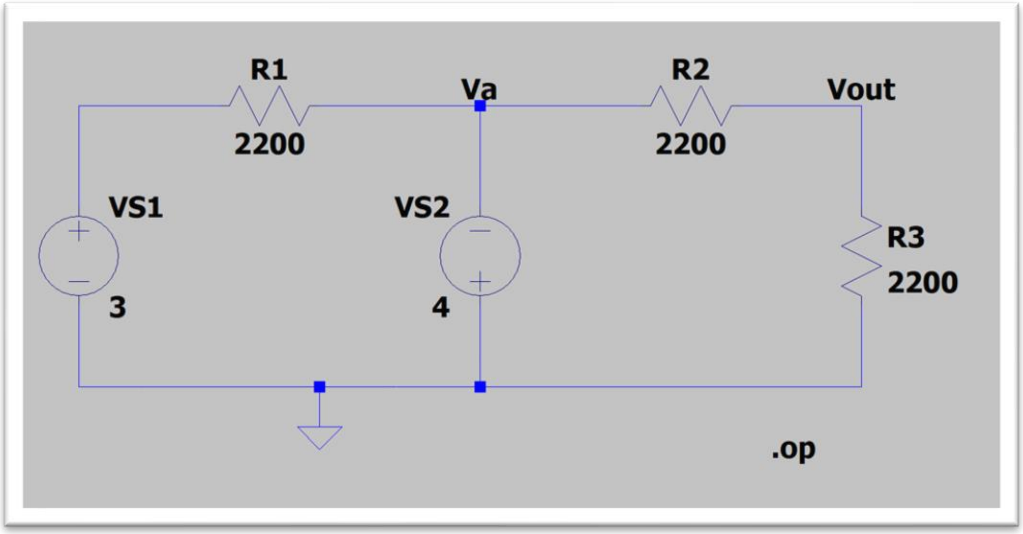



*Figure 6 LTSpice DC Simple Circuit*

```
        --- Operating Point ---

V(n001):          3               voltage
V(va):            -4              voltage
V(vout):          -2              voltage
I(R3):            -0.000909091    device_current
I(R2):            0.000909091     device_current
I(R1):            -0.00318182     device_current
I(Vs2):           -0.00409091     device_current
I(Vs1):           -0.00318182     device_current
```

*Figure 7 LTSpice Simulation Results*

From the LTSpice results it can be seen that **$V_{out}$ = -2V DC** and the **current at $R_2$ = -0.909mA**. These are exactly the same as the predicted results when completing the hand calculations.

Once the physical circuit was built the desired values were measured and the results were **$V_{out}$ = -1.998V DC** and the **current at $R_2$ was read to be -0.90mA**. These results are very close to the desired values from the hand calculations and the simulated results on LTSpice.

Measure Resistors used: $\mathbf{R_1 = 2172\ \Omega, R_2 = 2177\ \Omega, R_3 = 2170\ \Omega}$

A percentage error calculation was run from the simulation results and the physical results. The equation used is displayed below:

$$\text{Percentage error} = \frac{|measured - simulated|}{simulated} \times 100\%$$

$$\frac{|-1.998V - (-2.000V)|}{-2.000V} * 100\% = \mathbf{0.1}\%$$

**$V_{out}$ Percentage Error**

$$\frac{|-0.90mA - (-0.909)|}{-0.909V} * 100\% = \mathbf{0.99}\%$$

**Current at $R_2$ Percentage Error**

The simulated and physical circuits measurements were very accurate compared to the calculated measurements. All this is very accurate to conclude that it is correct.  However, due to physical resistor limitations not being perfectly accurate this is the cause for the slight disparity from it not being 100% accurate.

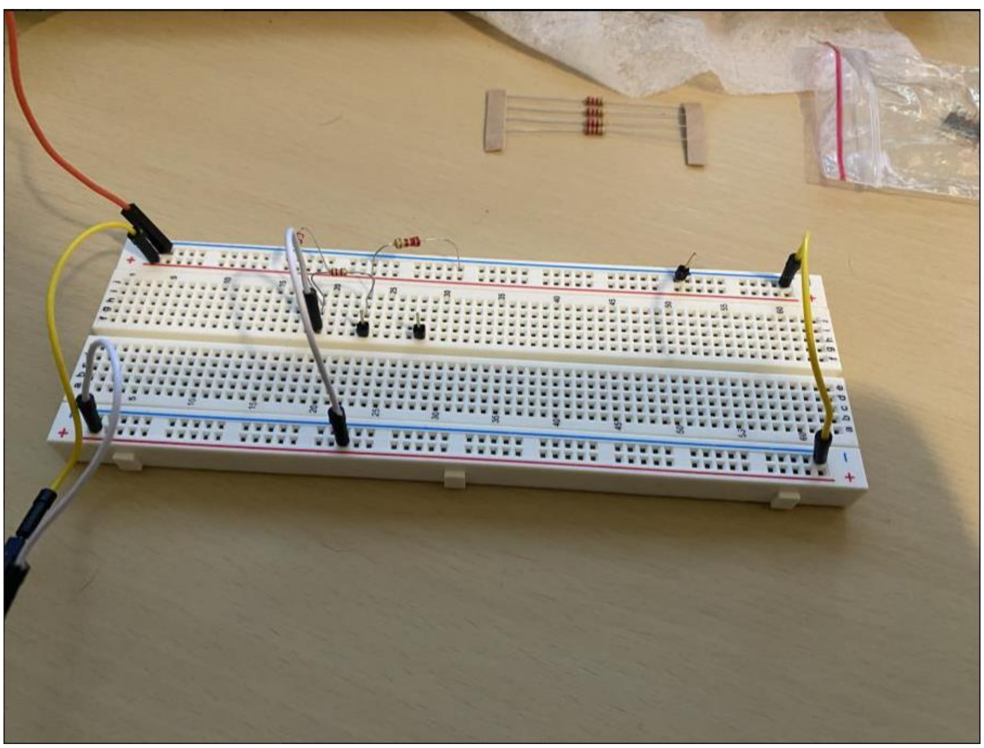
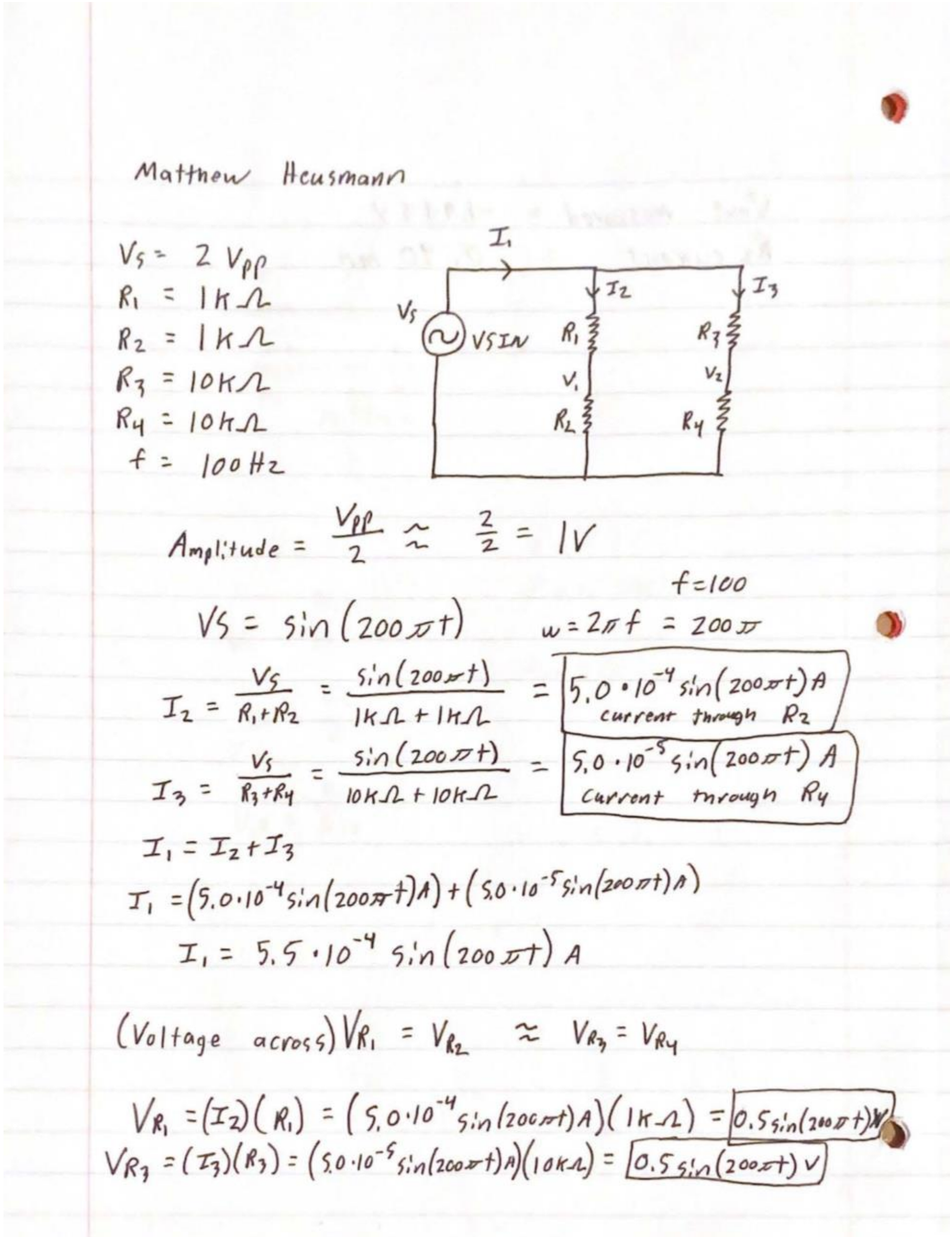


*Figure 8 Physical Simple DC Circuit*

## 1.2 Simple AC Circuit



Matthew Heusmann

$V_S = 2\ V_{pp}$
$R_1 = 1k\Omega$
$R_2 = 1k\Omega$
$R_3 = 10k\Omega$
$R_4 = 10k\Omega$
$f = 100Hz$



$$\text{Amplitude} = \frac{V_{pp}}{2} \approx \frac{2}{2} = 1V$$

$$V_S = \sin(200\pi t) \qquad \omega = 2\pi f = 200\pi \quad f = 100$$

$$I_2 = \frac{V_S}{R_1 + R_2} = \frac{\sin(200\pi t)}{1k\Omega + 1k\Omega} = \boxed{5.0 \cdot 10^{-4} \sin(200\pi t) A \text{ current through } R_2}$$

$$I_3 = \frac{V_S}{R_3 + R_4} = \frac{\sin(200\pi t)}{10k\Omega + 10k\Omega} = \boxed{5.0 \cdot 10^{-5} \sin(200\pi t) A \text{ Current through } R_4}$$

$$I_1 = I_2 + I_3$$

$$I_1 = (5.0 \cdot 10^{-4} \sin(200\pi t) A) + (5.0 \cdot 10^{-5} \sin(200\pi t) A)$$

$$I_1 = 5.5 \cdot 10^{-4} \sin(200\pi t) A$$

$$(\text{Voltage across})\ V_{R_1} = V_{R_2} \approx V_{R_3} = V_{R_4}$$

$$V_{R_1} = (I_2)(R_1) = (5.0 \cdot 10^{-4} \sin(200\pi t) A)(1k\Omega) = \boxed{0.5 \sin(200\pi t) V}$$

$$V_{R_3} = (I_3)(R_3) = (5.0 \cdot 10^{-5} \sin(200\pi t) A)(10k\Omega) = \boxed{0.5 \sin(200\pi t) V}$$

*Figure 9 Simple AC Circuit Hand Calculations*

From the figure above it can be seen that the current $I_2$ is the current that flows through $R_2$ while the current $I_3$ is the current that flows through $R_4$. So, the **current calculated through $R_2$ is $\mathbf{5.0 * 10^{-4} sin(200\pi t) A}$** while the **current through $R_4$ is $\mathbf{5.0 * 10^{-5} sin(200\pi t) A}$.** Also, from the hand calculations above the **voltage across each resistor was calculated to be $\mathbf{0.5 sin(200\pi t) V}$**.

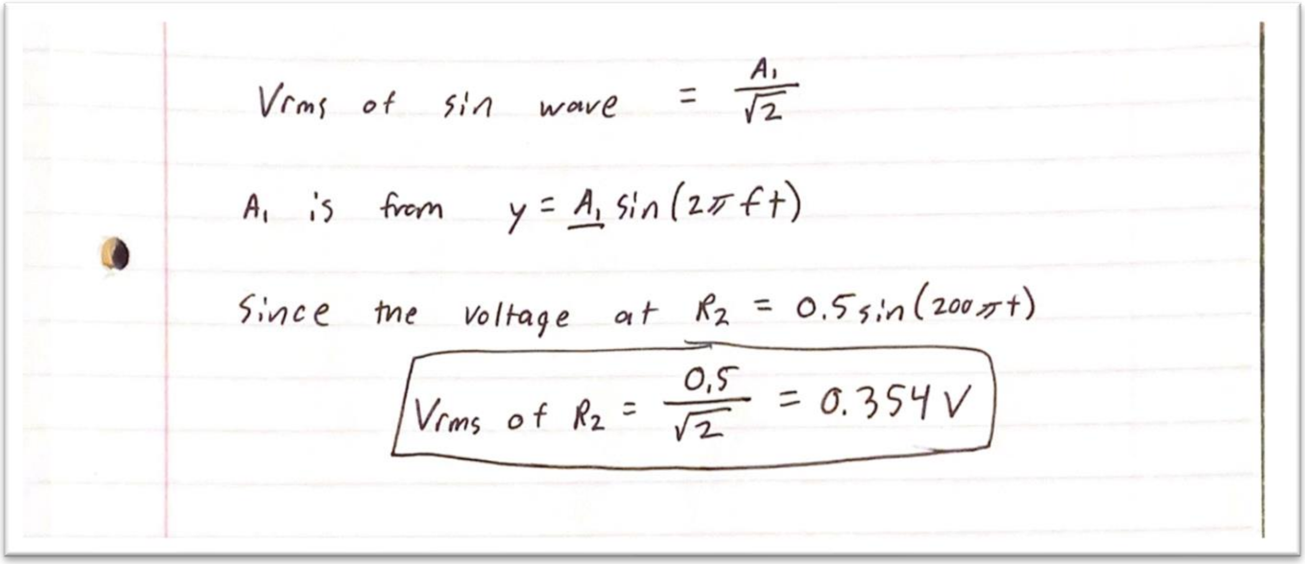



*Figure 10 Vrms Hand Calculation for AC Circuit*

This leads to what actually is $V_{rms}$? Well, $V_{rms}$ is the root mean square voltage of the presented sin wave to the circuit. So, it is the square root of the average squared voltage for one period. This can be found by $V_{rms} = \frac{A_1}{\sqrt{2}}$. Now this makes sense from the results calculated. Since the voltage came out to be $\mathbf{0.5sin(200\pi t)V}$ over $R_2$ from the work in the above figure the results were **0.354V for the $V_{rms}$ for $R_2$.**

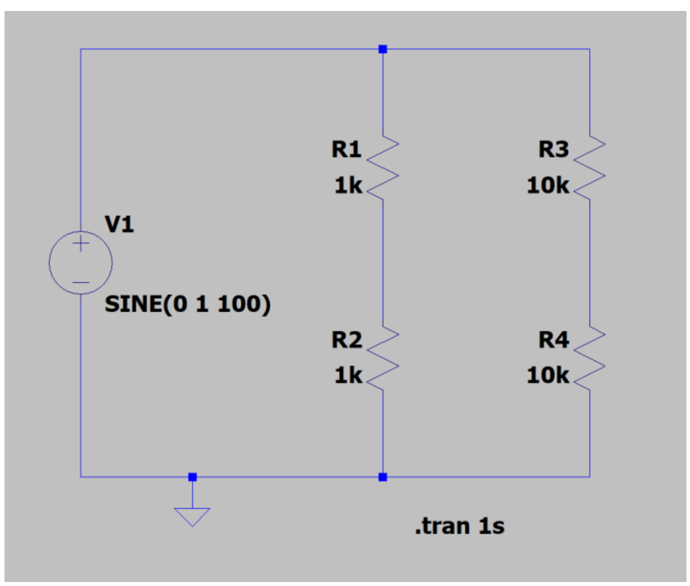



*Figure 11 LTSpice AC Simple Circuit*

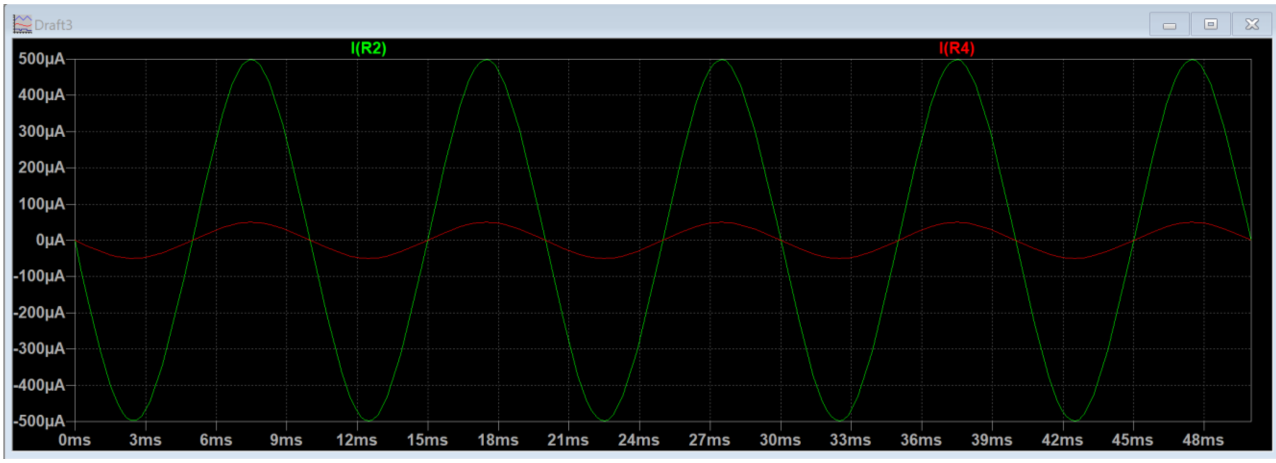



*Figure 12 Current Waveforms through R2 and R4*

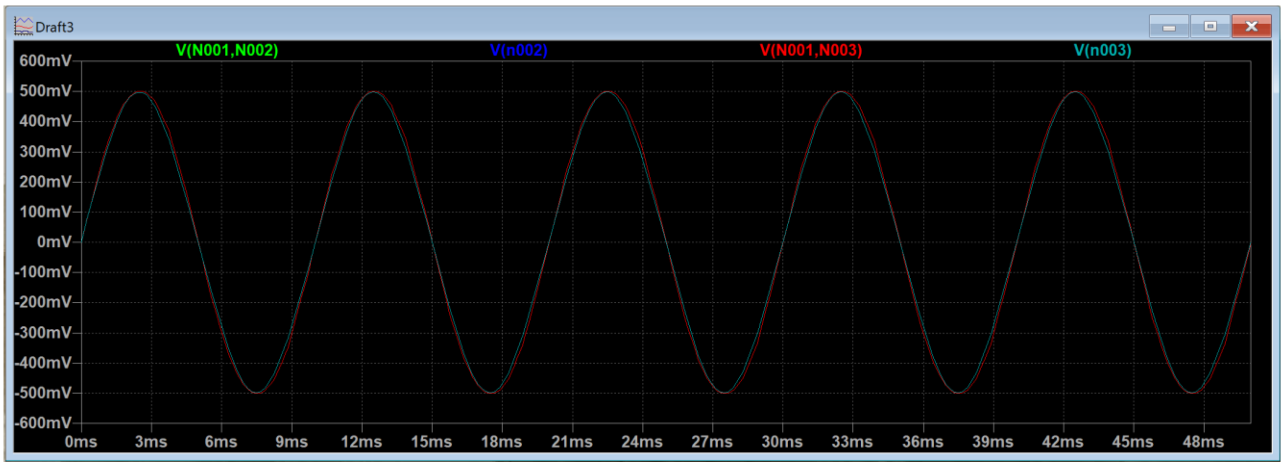



*Figure 13 Voltage Waveforms through R1, R2, R3, and R4*

*In figure 14 it should be noted that V(N001, N002) represents voltage across $R_1$, V(n002) is voltage across $R_2$, V(N001,N003) is voltage across $R_3$, and lastly V(n003) is voltage across $R_4$.

Measured Resistors used for the physical circuit are:
**$R_1$ = 978 Ω,$R_2$ = 986 Ω, $R_3$ = 9840 Ω, $R_4$ = 9870 Ω**

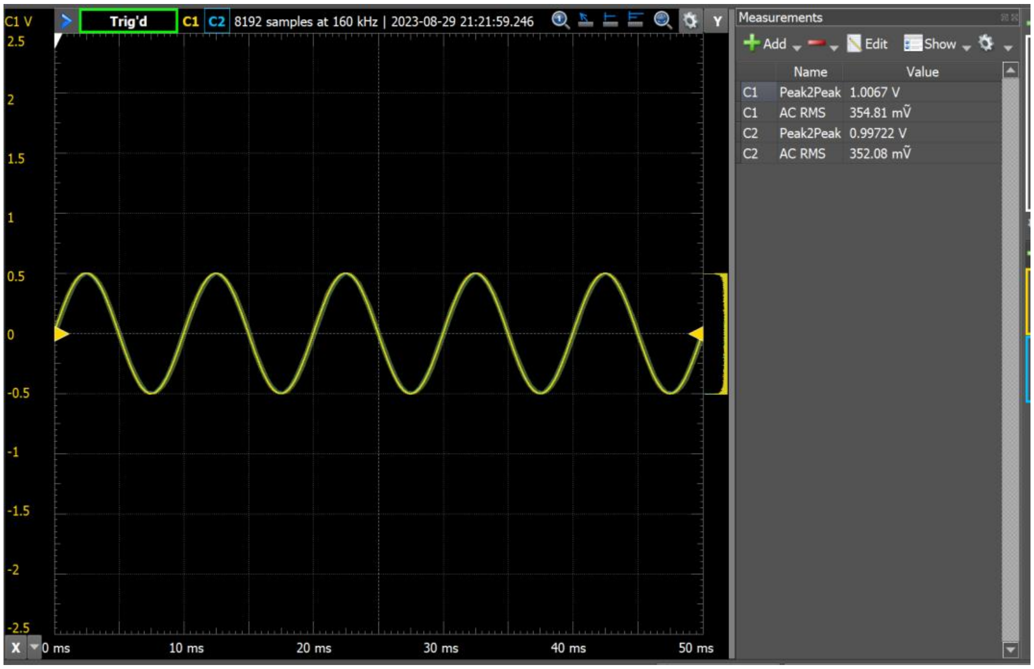



*Figure 14 Voltage waveforms across R2 and R4 of a Sine Wave*

From figure 15, this image was taken from the ADK oscilloscope sine wave to obtain the voltage waveforms across $R_2$ and $R_4$. Channel 1 was set to measure $R_2$ while channel 2 was set to measure $R_4$. The results were:

**$R_2$ $V_{pp}$ = 1.0067V and $R_2$ $V_{rms}$ = 354.81mV**
**$R_4$ $V_{pp}$ = 0.9972V and $R_2$ $V_{rms}$ = 352.08mV**

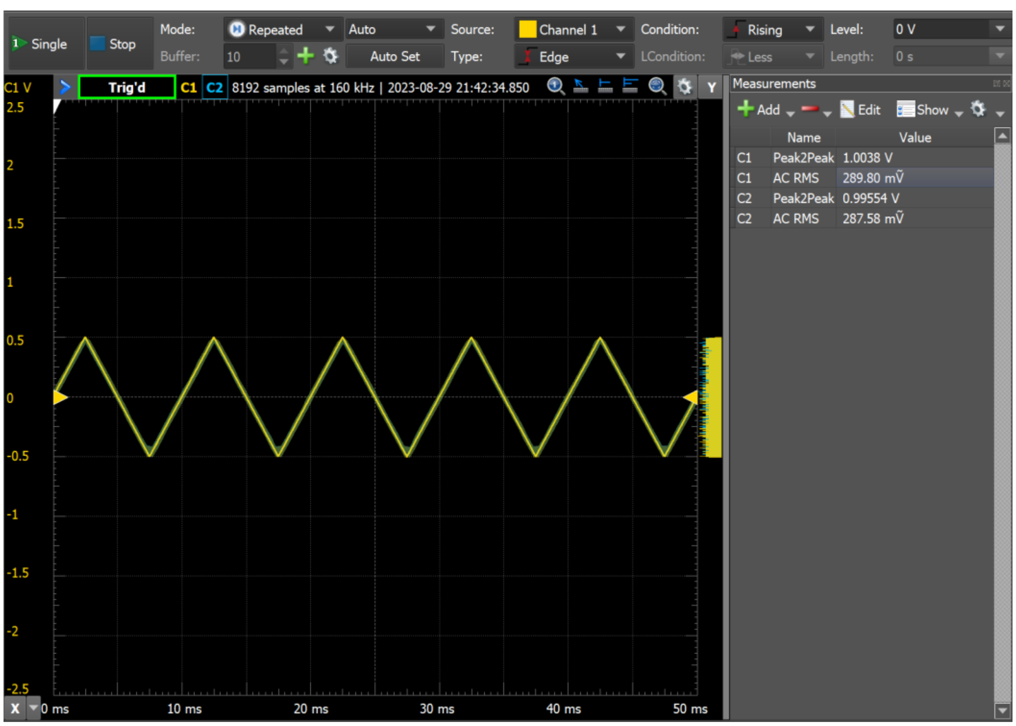



*Figure 15 Voltage waveforms across R2 and R4 of a Triangle Wave*

From figure 16, this image was taken from the ADK oscilloscope triangle wave to obtain the voltage waveforms across $R_2$ and $R_4$. The channels were set to the same resistors as in the sine wave. The results were:

**$R_2$ $V_{pp}$ = 1.0038V and $R_2$ $V_{rms}$ = 289.80mV**
**$R_4$ $V_{pp}$ = 0.9955V and $R_2$ $V_{rms}$ = 287.58mV**

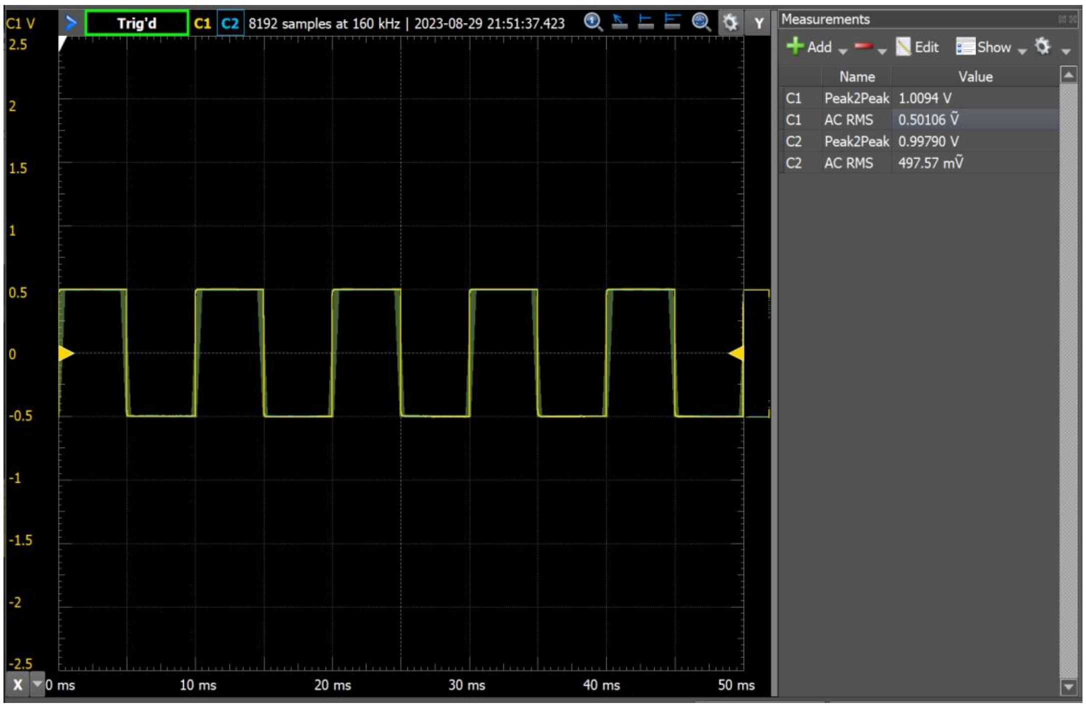


*Figure 16 Voltage waveforms across R2 and R4 of a Square Wave*

From figure 17, this image was taken from the ADK oscilloscope square wave to obtain the voltage waveforms across $R_2$ and $R_4$. The channels were set to the same resistors as in the sine wave. The results were:

**$R_2$ $V_{pp}$ = 1.0094V and $R_2$ $V_{rms}$ = 501.06mV**
**$R_4$ $V_{pp}$ = 0.9979V and $R_2$ $V_{rms}$ = 497.57mV**

With all this information multiple tables were made to present all the information in a more pleasant way. Also, it should be noted that a single voltage value is present in the table due to the fact that the measured and calculated voltages were identical and if not identical for some parts they were extremely close down to less than 1% percent error.

*Table 1 Sine Wave Results*

| Results From Sine Wave | | | | | |
|---|---|---|---|---|---|
| | **Hand Calculated** | **LTSpice Simulation** | **ADK Measured** | **Multimeter Measured** | **% Error** |
| $\mathbf{V_{pp}}$ | 1V | 1V | 1.0067V | 0.9984V | 0.16% |
| $\mathbf{V_{rms}}$ | 0.354 | 0.354 | 0.354 | 0.353 | 0.28% |
| **Peak $\mathbf{I_{R2}}$** | $5.0 * 10^{-4}A$ | $5.0 * 10^{-4}A$ | N/A | $4.8 * 10^{-4}A$ | 4% |
| **Peak $\mathbf{I_{R4}}$** | $5.0 * 10^{-5}A$ | $5.0 * 10^{-5}A$ | N/A | $4.9 * 10^{-5}A$ | 2% |
| **Time** | 10ms | 10ms | 10ms | 10ms | 0% |

*Table 2 Triangular Wave Results*

| Results From Triangular Wave | | | |
|---|---|---|---|
| | **ADK Measured** | **Multimeter Measured** | **% Error** |
| $\mathbf{V_{pp}}$ | 1.0038V | 0.9734V | 3.07% |
| $\mathbf{V_{rms}}$ | 289.80mV | 281.0mV | 3.04% |
| **Peak $\mathbf{I_{R2}}$** | N/A | $3.6 * 10^{-4}A$ | N/A |
| **Peak $\mathbf{I_{R4}}$** | N/A | $3.8 * 10^{-5}A$ | N/A |
| **Time** | 10ms | 10ms | 0% |

*Table 3 Square Wave Results*

| Results From Square Wave | | | |
|---|---|---|---|
| | **ADK Measured** | **Multimeter Measured** | **% Error** |
| $\mathbf{V_{pp}}$ | 1.0094V | 1.008V | 0.14% |
| $\mathbf{V_{rms}}$ | 501.06mV | 504.0mV | 0.59% |
| **Peak $\mathbf{I_{R2}}$** | N/A | $7.5 * 10^{-4}A$ | N/A |
| **Peak $\mathbf{I_{R4}}$** | N/A | $7.6 * 10^{-5}A$ | N/A |
| **Time** | 10ms | 10ms | 0% |

**Triangular $\mathbf{V_{rms}}$ = $\frac{V_{peak}}{\sqrt{3}}$** **Square wave $\mathbf{V_{rms}}$ = $\mathbf{V_{peak}}$** This information was taken from the Wikipedia chart located on Ed discussion.

From the results of each of the tables it can be seen that everything should be as expected, however there are some slight variances when it comes to the percent error. Everything is below 5%, however the ones with the highest percent errors were due to physical errors. This could be the resistor discrepancy from the resistors not being the true value in the simulation or hardware issues with varying multimeter leads. It was noticed that one of the leads connected to the multimeter was giving a different reading from another set of leads.

*Table 4 Ratio of Vrms/Vpeak for Triangular and Square Waves*

| Ratio of $V_{rms}/V_{peak}$ | | | |
|---|---|---|---|
| | Theoretical | Measured | % Error |
| **Triangular Wave** | $\frac{1}{\sqrt{3}} = 0.577$ | 0.577 | 0% |
| **Square Wave** | 1 | 1 | 0% |

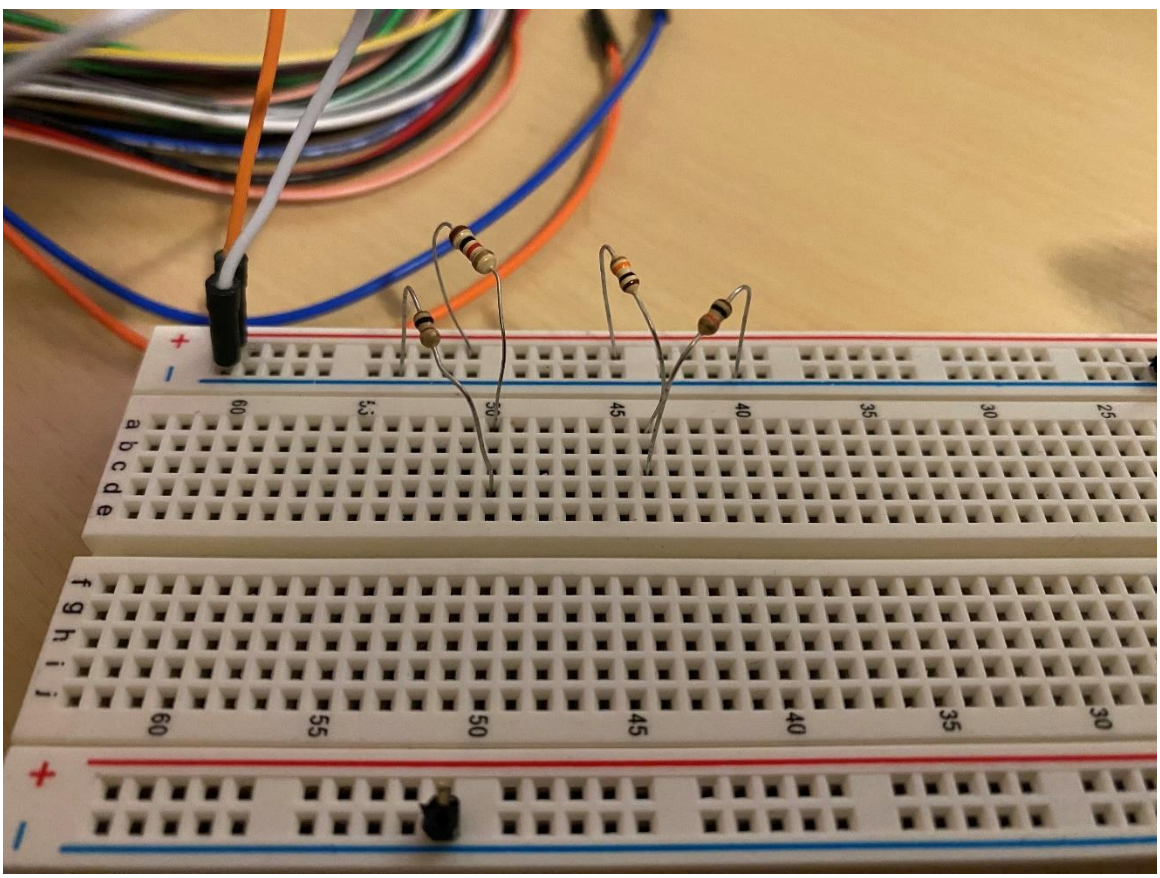



*Figure 17 Simple AC Circuit*

## 1.3 Low Pass Filter Analysis

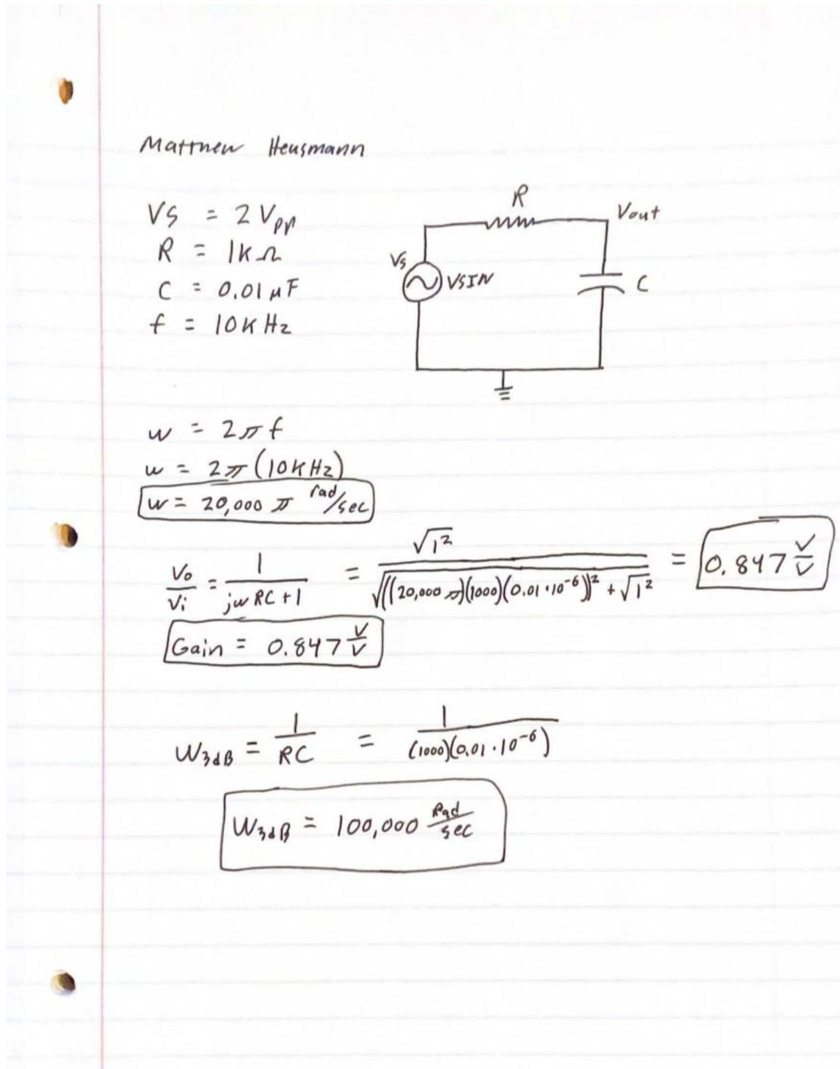



*Figure 18 Low Pass Filter Hand Calculations*

From the hand calculations above it can be seen that three items were focused to be found. They were $\omega$, the gain for the low pass filter circuit, and the $\omega_{3dB}$. The results were **ω = 20000π rads/sec, Gain = 0.847V/V, and $\omega_{3dB}$ = 100,000rads/sec.**

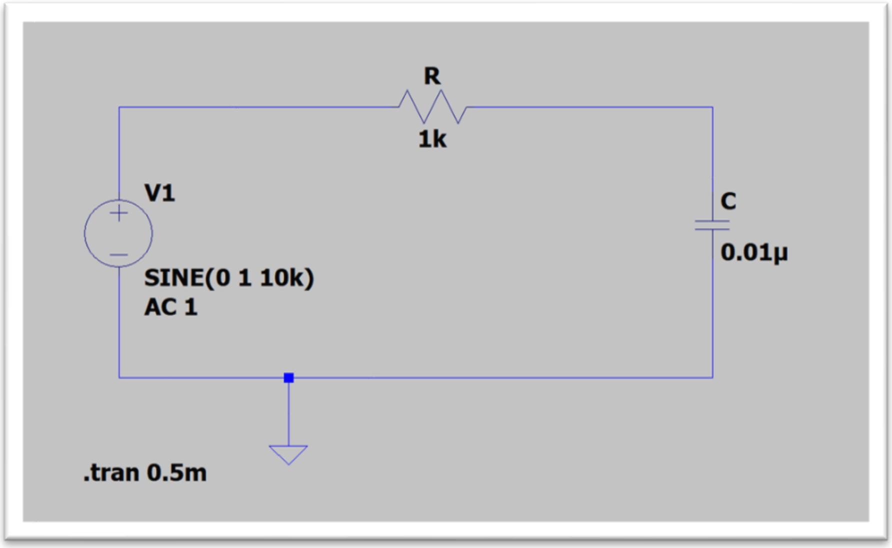



*Figure 19 LTSpice Low Pass Filter*

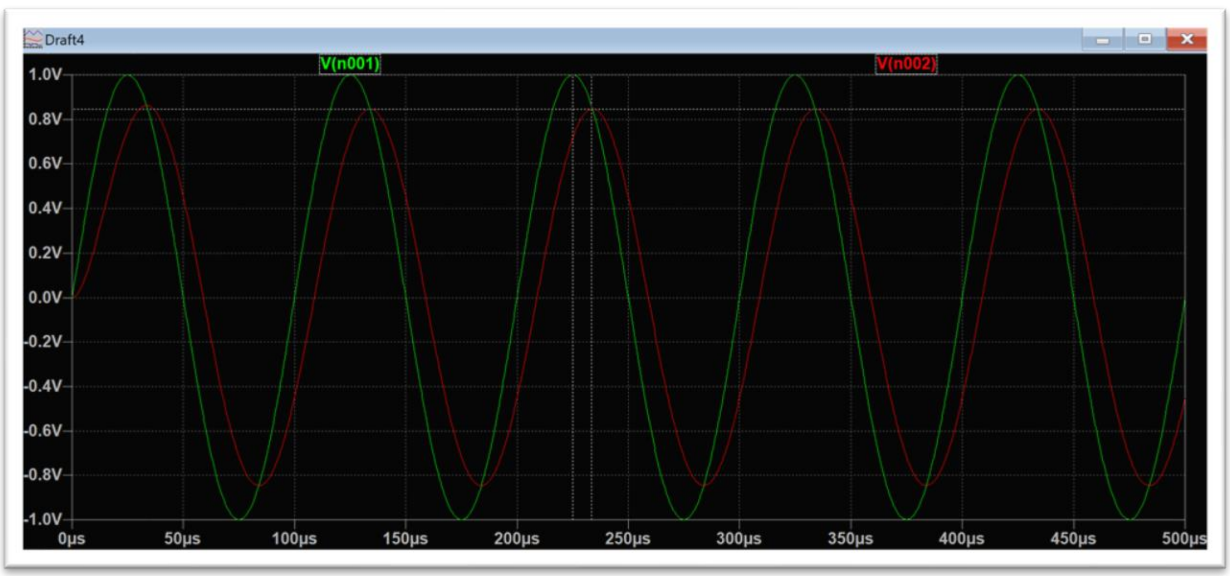



*Figure 20 Transient Analysis of Low Pass Filter*

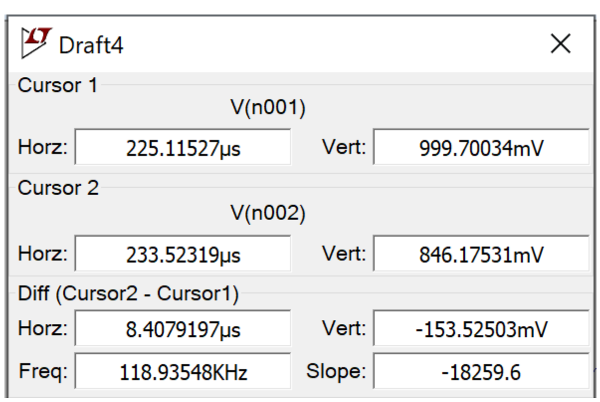



*Figure 21 Transient Analysis of Low Pass Filter Cursor Readings*

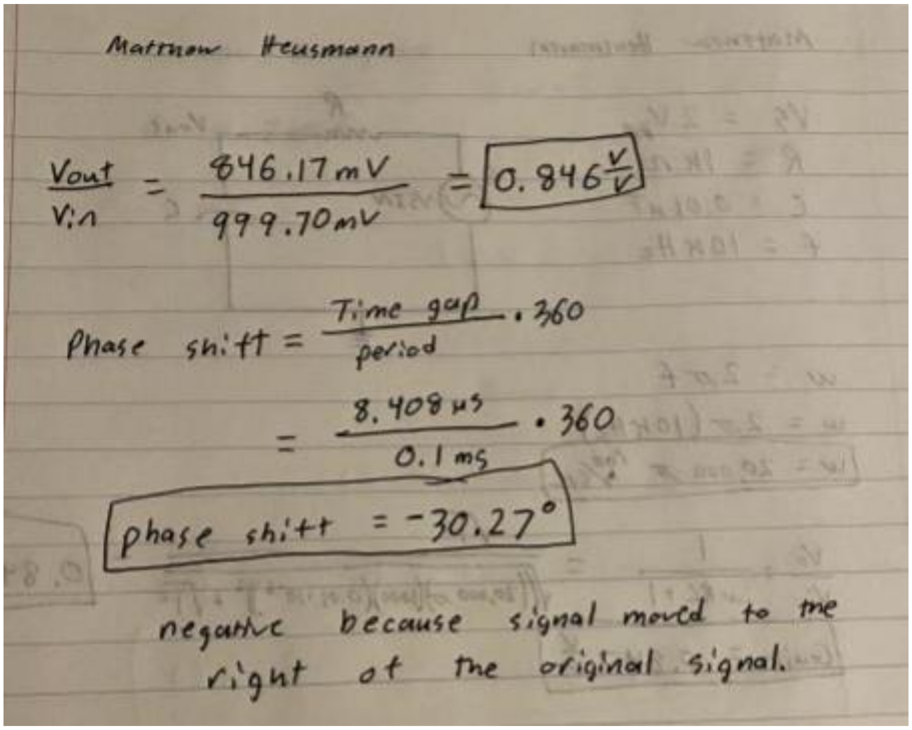


*Figure 22 Calculations for Gain and Phase Shift for Low Pass Filter*

From the figures above this is the result from the Transient Analysis of the Low Pass Filter. From analyzing the figures, it can be seen that V(n001) is the Vin, and this resulted in a peak voltage of **999.70mV** while V(n002) is representing $V_{out}$ this value was **846.17mV**. Each signal has a 100µs period however it can be seen that Vout leads $V_{in}$ by about 8.408µs. With this time difference the phase shift can be calculated, as this was done above in figure 23. After the calculations it can be seen that $V_{out}$ is phase shifted by -30.27°. The reason for the negative is due to the fact the $V_{out}$ is to the right of the original signal.

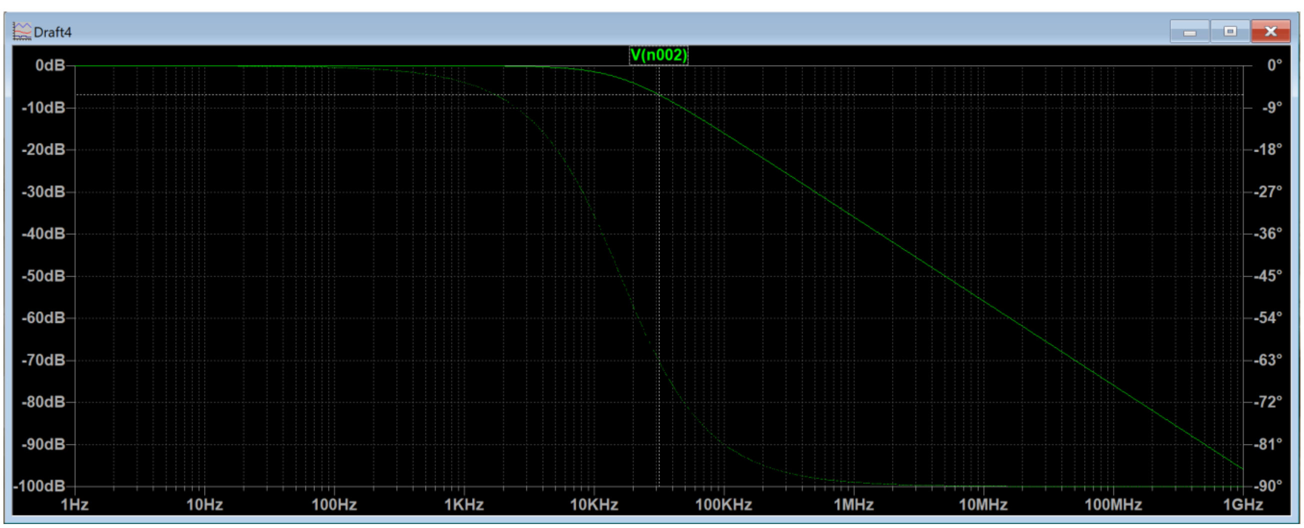


*Figure 23 AC Analysis of the Low Pass Filter*

From figure 24 it can be seen that the maximum gain of this circuit would be at the lower frequencies, this would be $A_0$ = 0dB. Now as for finding the $f_{3dB}$ point this can be found in the LTSpice graph by trying to get as close as possible to the -3dB the result was 15883Hz. Referencing figure 25 below this is where the calculations were run. This results in a $\omega_{3dB} = 99795.8\ \frac{rad}{sec}$.

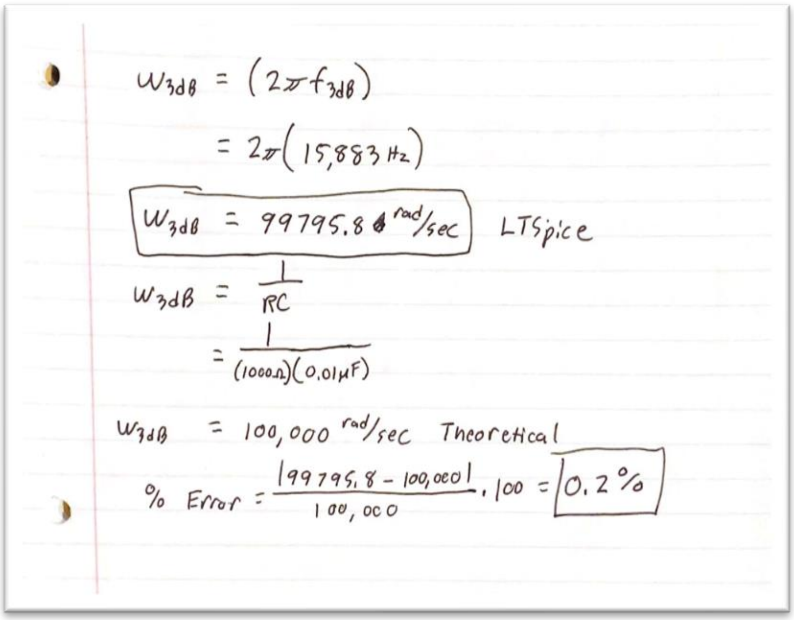



*Figure 24 LTSpice AC Analysis Calculations*

Measured values for components: **R = 9820Ω , C = 0.0089µF**

*Figure 25 ADK Analysis and Measurements*

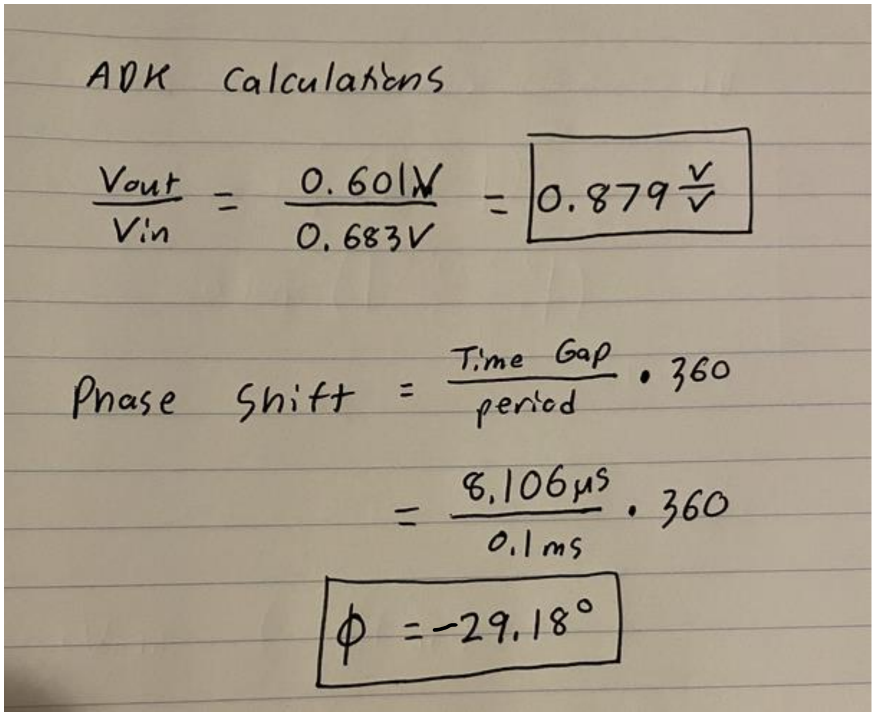



*Figure 26 ADK Calculations from Figure 25*

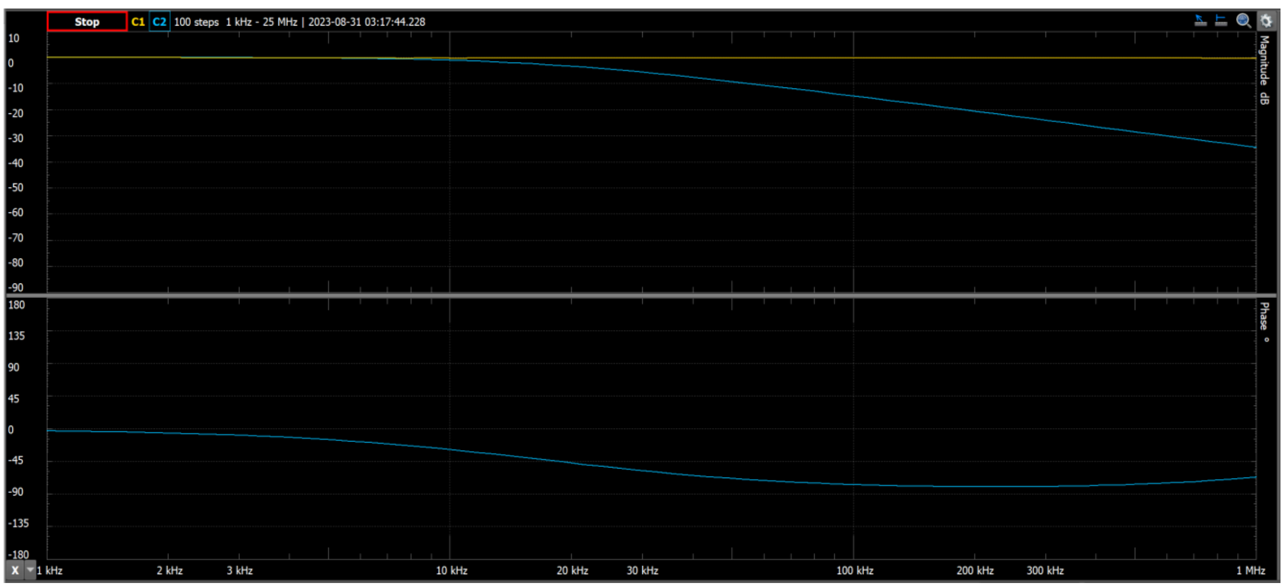



*Figure 27 ADK Network Analyzer of Low Pass Filter*

From figure 27 above it can be seen that **Vout/Vin = 0.879**$\boldsymbol{\frac{V}{V}}$ this is slightly off from the theoretical calculations and similar results from the phase shift that equals = **-29.18°**. The reason for this disparity from the theoretical calculations is due to the physical limitations of the hardware in the breadboard such as the resistor used and the capacitor. Each of these values are off from the expected values and this slight variation results in small but not negligible percent errors.

*Table 5 Results of 3dB Frequencies for the Low Pass Filter*

| Comparison of 3dB frequencies for the Low Pass Filter | | |
|---|---|---|
| **Theoretical** | **Simulated** | **% Error** |
| 100,000 rad/sec | 99,796 rad/sec | 0.2% |

*Table 6 Results of Gain and Phase Shift for Low Pass Filter*

| Gain and Phase Shift Results for Low Pass Filter | | | | |
|---|---|---|---|---|
| | **Theoretical** | **Simulated** | **Measured** | **% Error** |
| **Gain** | $0.847\,\frac{V}{V}$ | $0.846\,\frac{V}{V}$ | $0.879\,\frac{V}{V}$ | 3.9% |
| **Phase Shift** | N/A | -30.27° | -29.18° | 3.6% |

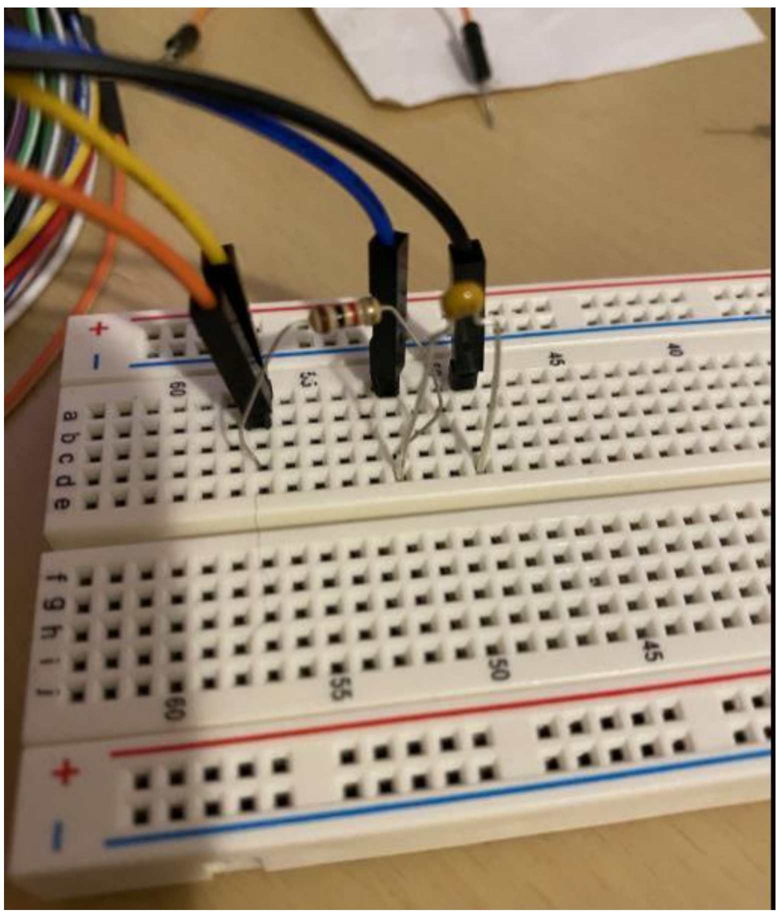


*Figure 28 Low Pass Filter Hardware*

## 1.4 High Pass Filter Analysis

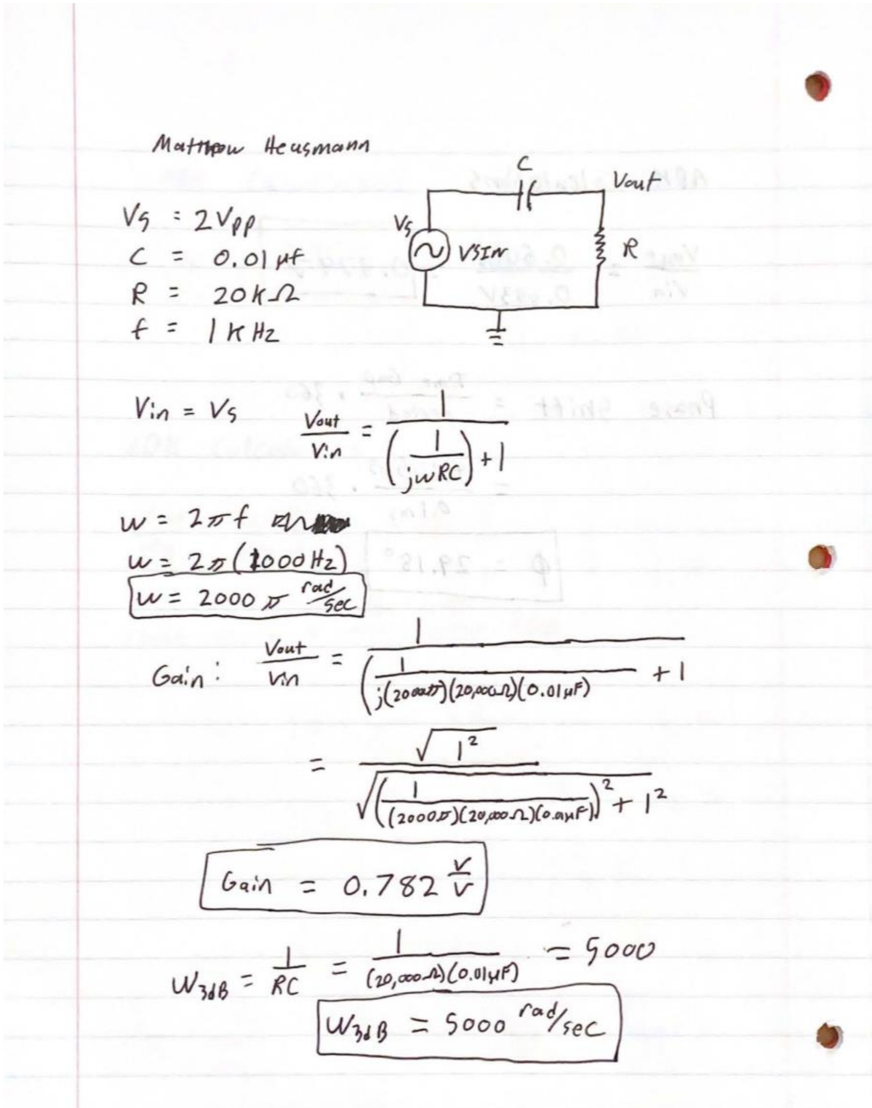



*Figure 29 Hand Calculations for High Pass Filter*

From the hand calculations above it can be seen that three items were focused to be found. They were $\omega$, the gain for the high pass filter circuit, and the $\omega_{3dB}$. The results were **ω = 2000π rads/sec, Gain = 0.782V/V, and $\omega_{3dB}$ = 5000 rads/sec.**

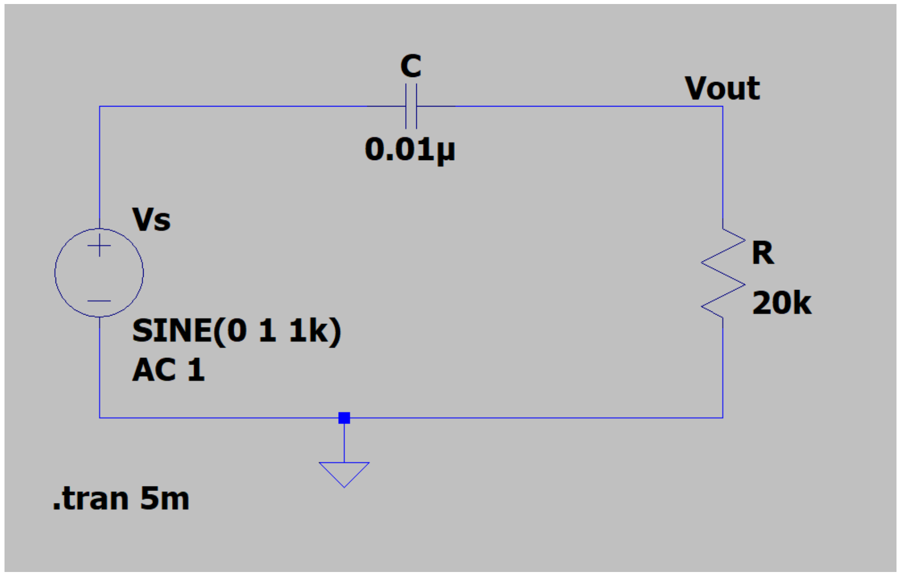



*Figure 30 LTSpice High Pass Filter*

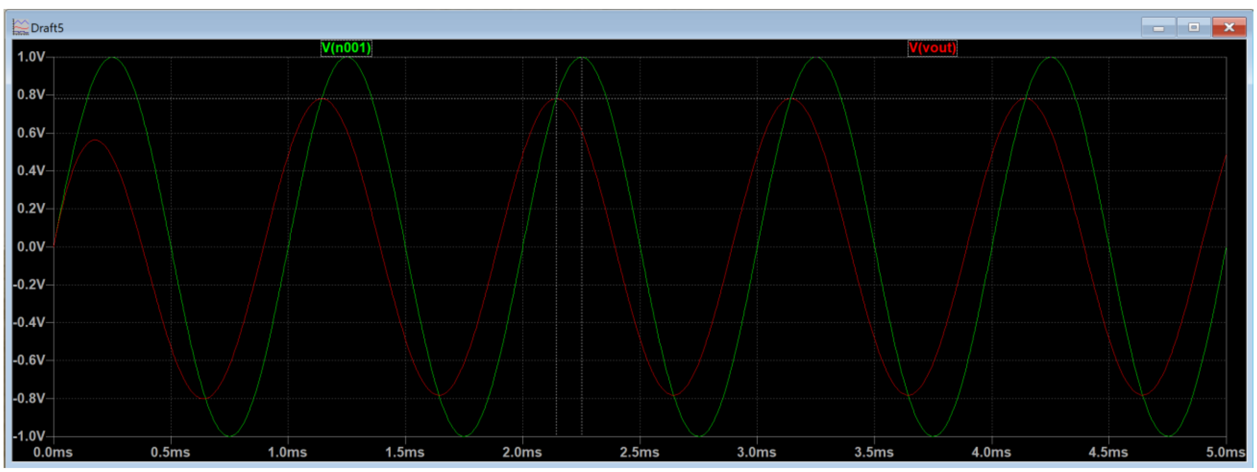



*Figure 31 Transient Analysis of High Pass Filter*

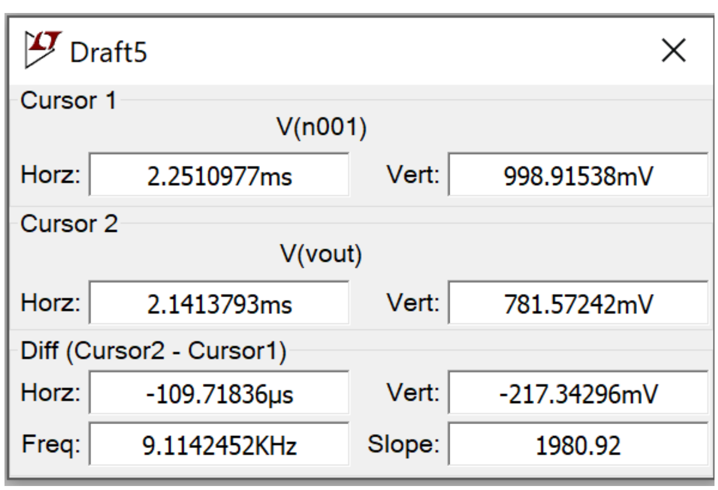
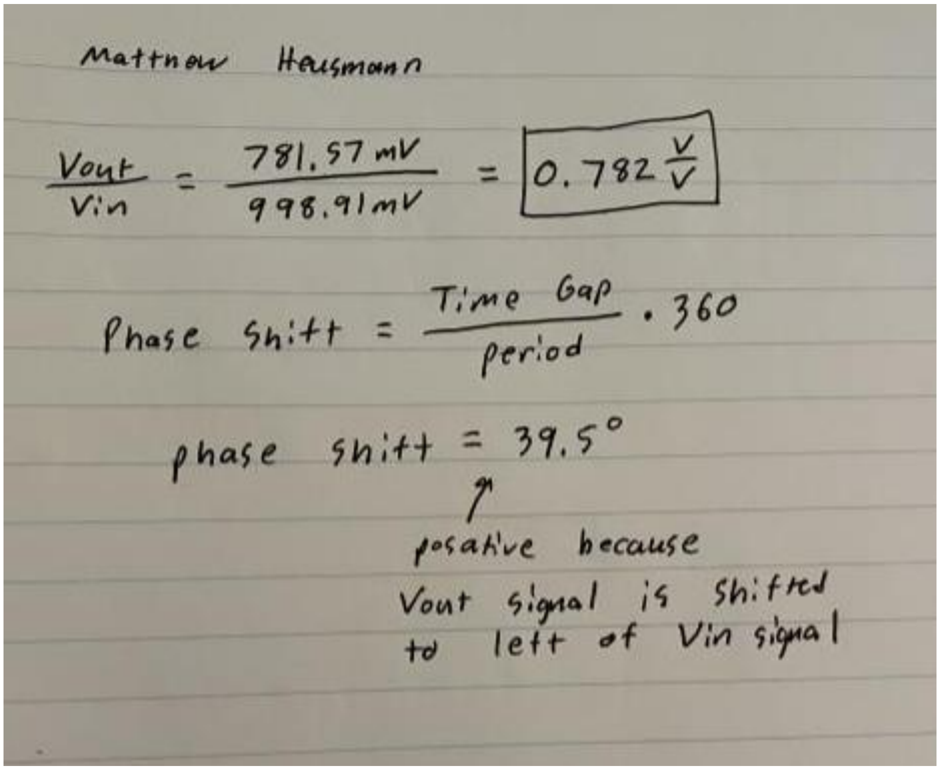



*Figure 32 Transient Analysis of High Pass Filter Cursor Readings*



Matthew Hausmann

$$\frac{V_{out}}{V_{in}} = \frac{781.57\,mV}{998.91\,mV} = \boxed{0.782\,\frac{V}{V}}$$

$$\text{Phase Shift} = \frac{\text{Time Gap}}{\text{period}} \cdot 360$$

$$\text{phase shift} = 39.5^\circ$$

posative because
Vout signal is shifted
to left of Vin signal

*Figure 33 Calculations for Gain and Phase Shift for High Pass Filter*

From the figures above this is the result from the Transient Analysis of the High Pass Filter. From analyzing figures 32-34, it can be seen that V(n001) is the Vin and this resulted in a peak voltage of **998.91mV** while **Vout was 781.57mV**. Each signal has a 0.001s period however it can be seen that $V_{out}$ lags Vin by about 109.7µs. With this time difference the phase shift can be calculated, as this was done above in figure 34. After the calculations, it can be seen that $V_{out}$ is phase shifted by +39.5°. The reason for the positive this time is due to the fact the $V_{out}$ is to the left of the original signal Vin.

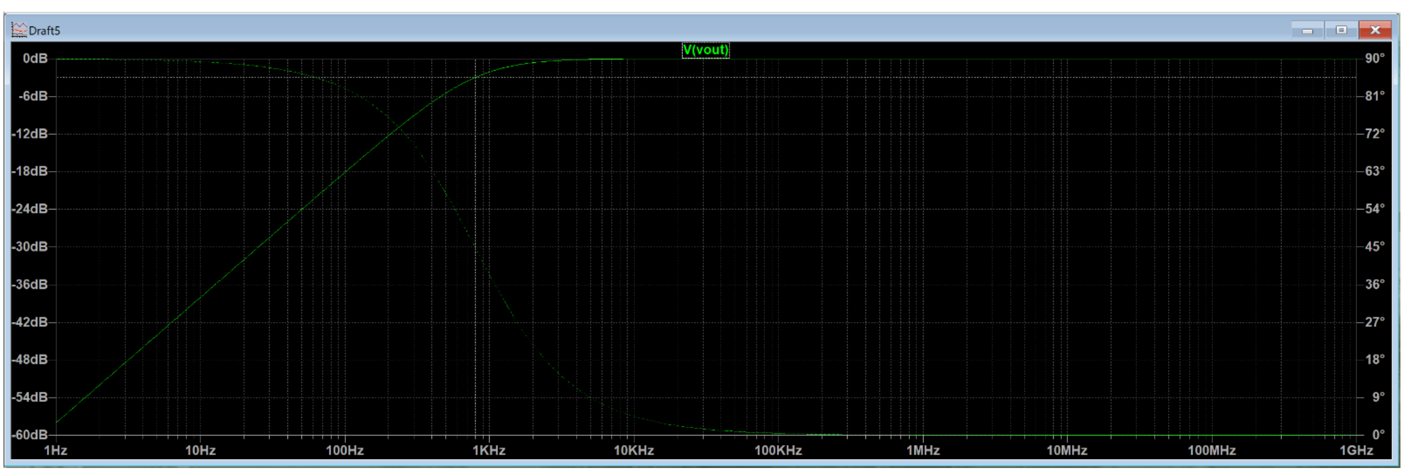



*Figure 34 AC Analysis of the High Pass Filter*

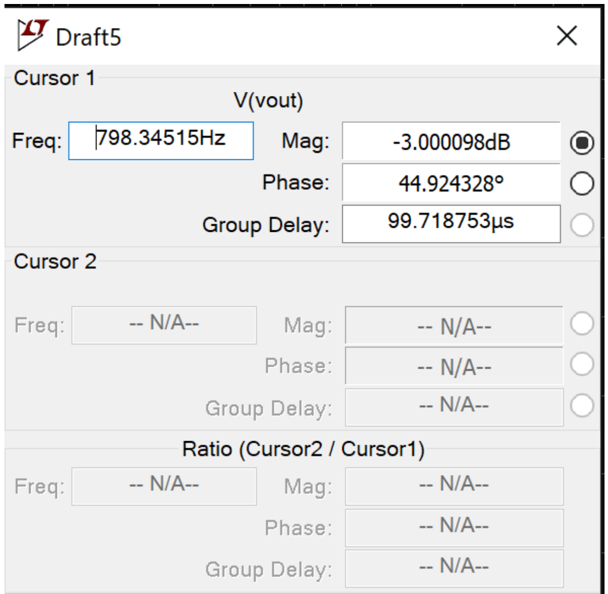



*Figure 35 AC Analysis of the High Pass Filter Data*

From figure 35-36 it can be seen that the maximum gain of this circuit would be at the higher frequencies, this would be $A_0$ = 0dB. Now as for finding the $f_{3dB}$ point this can be found in the LTSpice graph by trying to get as close as possible to the -3dB the result was 798Hz. Referencing figure 37 below this is where the calculations were run. This results in a $\omega_{3dB} = 5014\ \frac{rad}{sec}$.

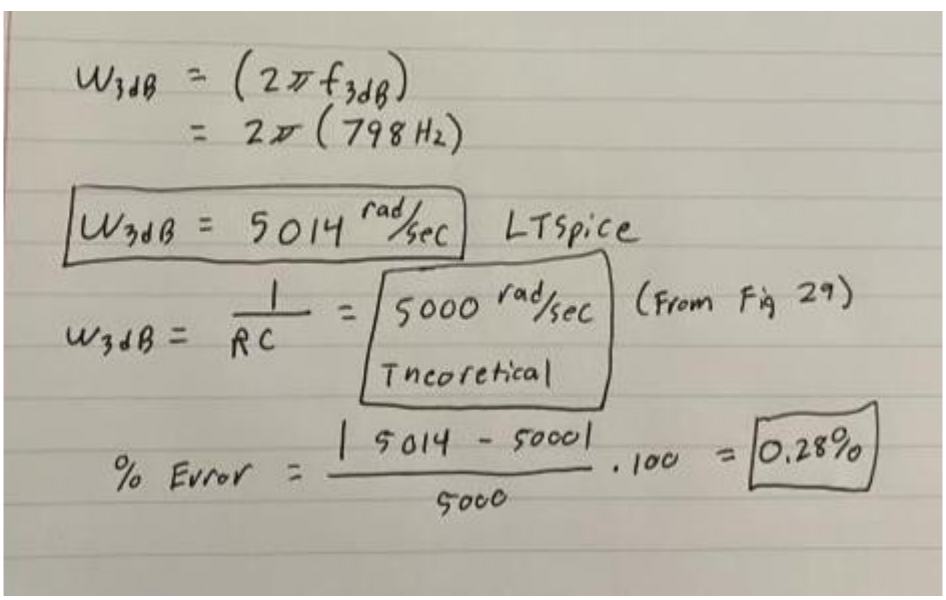



*Figure 36 LTSpice AC Analysis Calculations High Pass Filter*

Measured values for components: **R = 19860Ω , C = 0.0089µF**

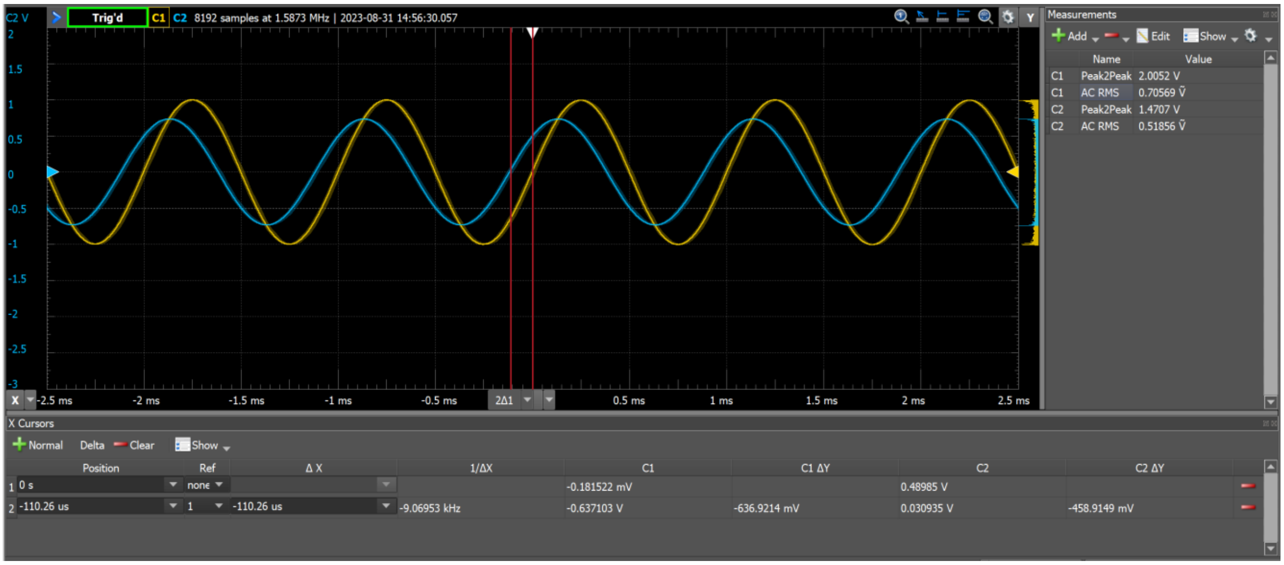


*Figure 37 ADK Analysis and Measurements High Pass Filter*

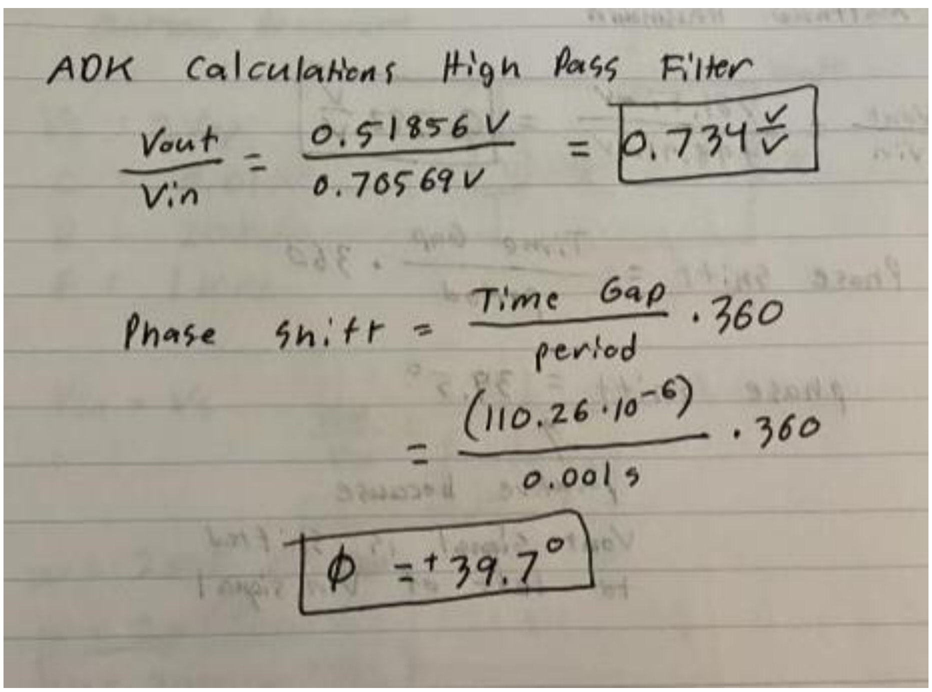


*Figure 38 ADK Calculations from Figure 37*

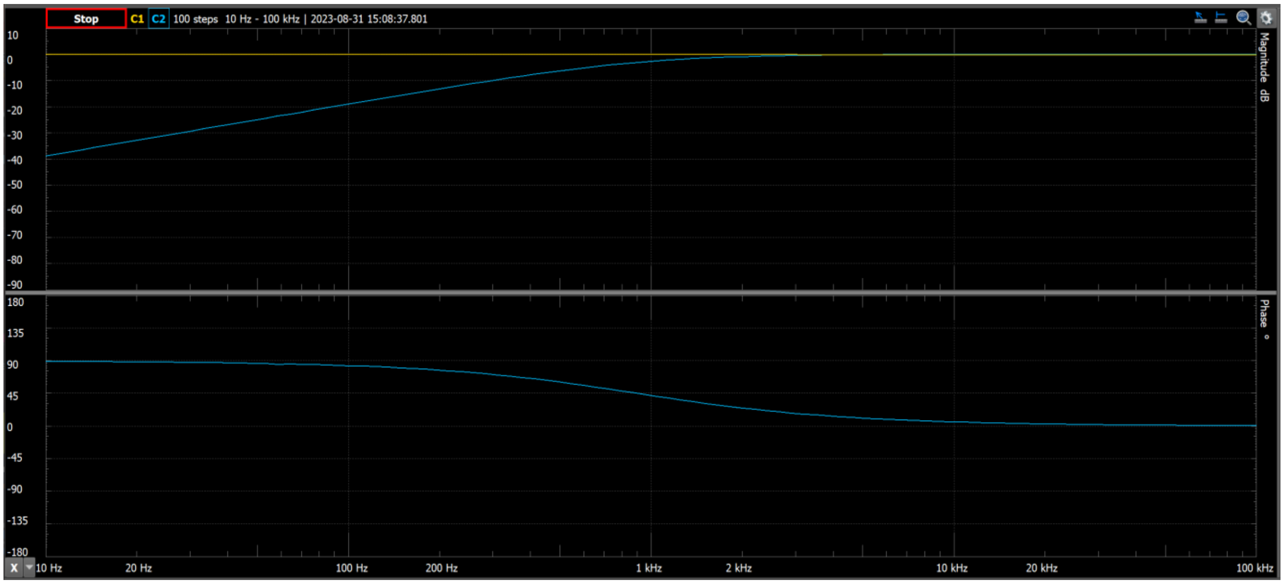



*Figure 39 ADK Network Analyzer of High Pass Filter*

From figure 39 above it can be seen that **Vout/Vin = 0.734**$\boldsymbol{\frac{V}{V}}$ this is slightly more off from the theoretical calculations and similar results are seen from the phase shift that equals = **39.7°**. The reason for this greater disparity from the theoretical calculations is as stated above to the physical limitations of the hardware in the breadboard such as the resistor used and the capacitor. Each of these values are off from the expected values and this variation results in small but not negligible percent errors. Also, it was noted that the breadboard would have similar but varying results when moving the circuit around to different pin slots. This is also an issue that would cause this error. This is further discussed in the conclusion section.

*Table 7 Results of 3dB Frequencies for the High Pass Filter*

| **Comparison of 3dB frequencies for the High Pass Filter** | | |
|---|---|---|
| **Theoretical** | **Simulated** | **% Error** |
| 5,000 rad/sec | 5,014 rad/sec | 0.28% |

*Table 8 Results of Gain and Phase Shift for High Pass Filter*

| **Gain and Phase Shift Results for High Pass Filter** | | | | |
|---|---|---|---|---|
| | **Theoretical** | **Simulated** | **Measured** | **% Error** |
| **Gain** | 0.782 $\frac{V}{V}$ | 0.782 $\frac{V}{V}$ | 0.734 $\frac{V}{V}$ | 6.1% |
| **Phase Shift** | N/A | 39.5° | 39.7° | 0.5% |

**Please see paragraph above table 7 and conclusion for reason for the 6.1% error in gain from table 8.

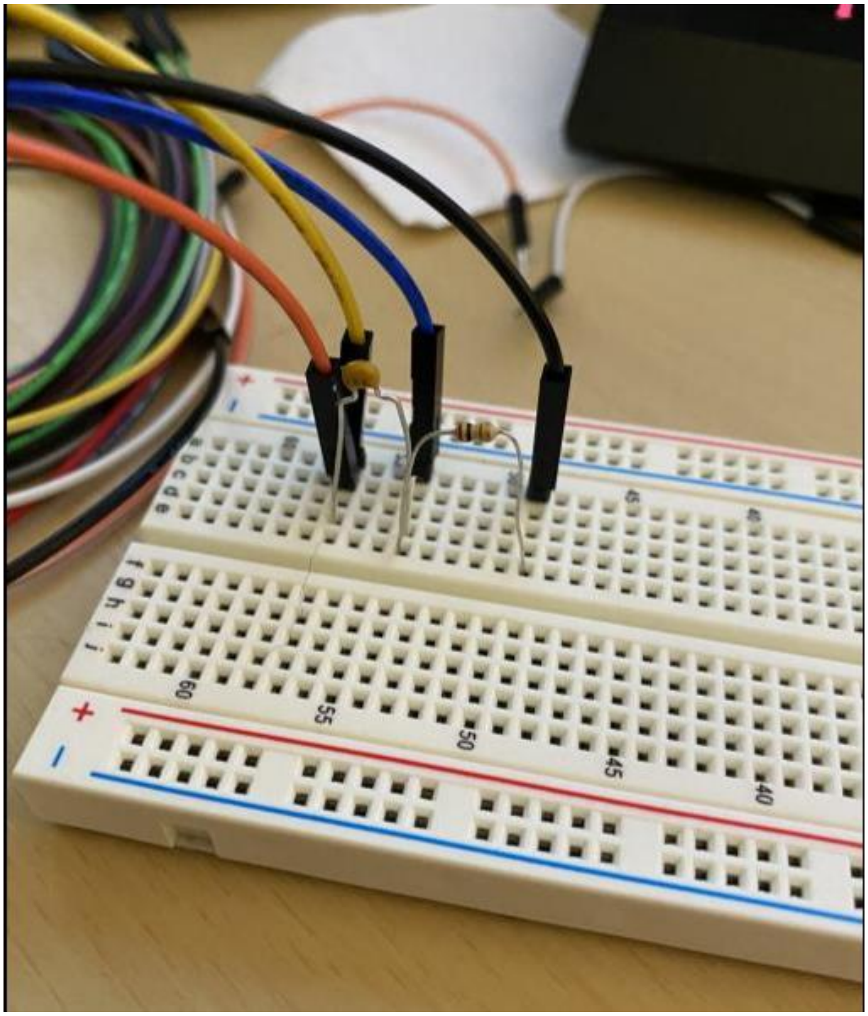


*Figure 40 High Pass Filter Hardware*

# CONCLUSION

For the first part of the lab the main focus is to get a basic understanding of a simple DC circuit. This was done by first analyzing the drawing of the schematic and hand calculating the expected values for the circuit. Once this was complete the circuit was then built in LTSpice, and circuit analysis was completed to verify that the hand calculations were correct or at least a very small percent error such as less than 1%. The last part of the section was building the physical circuit on a breadboard to be analyzed by waveforms and a multimeter. The results yielded a 0.1 error for voltage and a 0.99 error for the current through $R_2$. These are at acceptable limits and the reason for this error is caused by the physical characteristics of the components used such as the resistors not being at the expected 2.2kΩ.

Moving into part 1.2 of the lab the focus was shifted to analyzing a simple AC circuit. This section of the lab was familiarizing the idea that voltage and current read from the circuit was in the form of $V_{rms}$ and $I_{rms}$. Again, this section was very similar to the previous section where the circuit values were hand calculated, built and simulated in LTSpice, and physically built for analysis with waveforms oscilloscope and a handheld multimeter. This section hammers the idea that voltage and current division are also effective for analyzing AC circuits. Towards the end of this part not only was a basic sine wave analyzed but also a triangular wave along with a square wave. This section was allowing practice and saw how the $V_{rms}$ differs from the basic sine wave. At the end of the part all values calculated were entered in tables. From the percentage error calculations, it can be seen that no values exceeded a 3% error. This allows the error to be attributed to the tolerances of the components used such as the resistors.

In part 1.3 the lab shifts to analyzing a low pass filter. This was to familiarize how exactly the gain is calculated in a filter along with seeing that low pass filters are fairly ideal. Again, this section was completed as above with hand calculations, LTSpice built and simulated, and physically built to be analyzed against LTSpice values. The main learning points here were how to calculate the gain and phase shift from the ADK waveforms software. This was completely new, however now fairly comfortable with completing this task. There are some faults here with the software such as placing the cursors in the exact right locations to receive the delta time between the two signals for calculating phase shift. This along with physical hardware limitations resulted in a percent error under 4% this is starting to approach the higher end of allowable error. The main cause of this error is the capacitor values were drastically different from each other in the ADK parts kit the one chosen was the closest to the desired value. Along with an issue inside the breadboard that was discovered in the high pass filter section. With all this information 4% is not horrible but starting to be on the high side of allowable.

As for the last section 1.4, this was focused on a simple high pass filter. This was very similar to the low pass filter and the steps were almost exactly the same. Again, the section was laid out like the previous where hand calculations were run, circuit was built in LTSpice and simulated, and lastly the physical circuit was built and analyzed. Once all this was accomplished the results were logged in tables 7-8 for easy viewing. Now the error for this high pass filter came out on the high end of the lab results. With a gain % error of around 6% this is definitely higher than desired. However, as mentioned above there was an issue found with the breadboard, that as the circuit was moved around the breadboard, varying results would happen. The percent error would move from 4-8% so the middle values were taken for the final results. These issues accompanied with physical component error would result in this higher 6% error. The action that will be taken for future labs will be that the breadboard will be replaced with a new one.

To sum up the lab, this lab was mainly focused on introducing basic circuit familiarization and gearing towards the labs to come. This was a great refresher to circuits 1 that was taken a long time ago. Each of these four circuits gave a better understanding of how to utilize LTSpice when it comes to building the circuit and utilizing transient analysis, AC analysis, and the network analyzer on waveforms. Also getting a refresher on physical building of the circuit on a breadboard allows a better grasp at errors that can occur from physical hardware. This lab was extremely long and tedious but was a good refresher starting into the class.

# POST LAB QUESTIONS

1. Digital multimeters mostly measure and display rms values for AC currents and voltages. As for oscilloscopes, they are primarily used to measure voltages, often neglecting direct current measurements. Yet, oscilloscopes offer a comprehensive insight into AC parameters by showcasing details like period, peak-to-peak values, and phase shifts as shown from this lab.

2. When a voltage source is shorted, it is connected to an extremely low resistance, causing it to draw an immense amount of current to maintain its potential difference. This large current dissipates a vast amount of energy rapidly, often as heat. This energy release can damage surrounding circuit components and poses a direct threat to people, potentially causing burns or shocks.

3. A.) Making all the resistors the same resistance values being 10kΩ for $R_1$ through $R_4$ this will make the circuit be $1V_{pp}$ across all resistors. Thus, the result of having the same current across all the resistors. The current being 0.1mA peak to peak. Also, with a frequency of 100Hz would make a period of 10ms. Figures are displayed below.

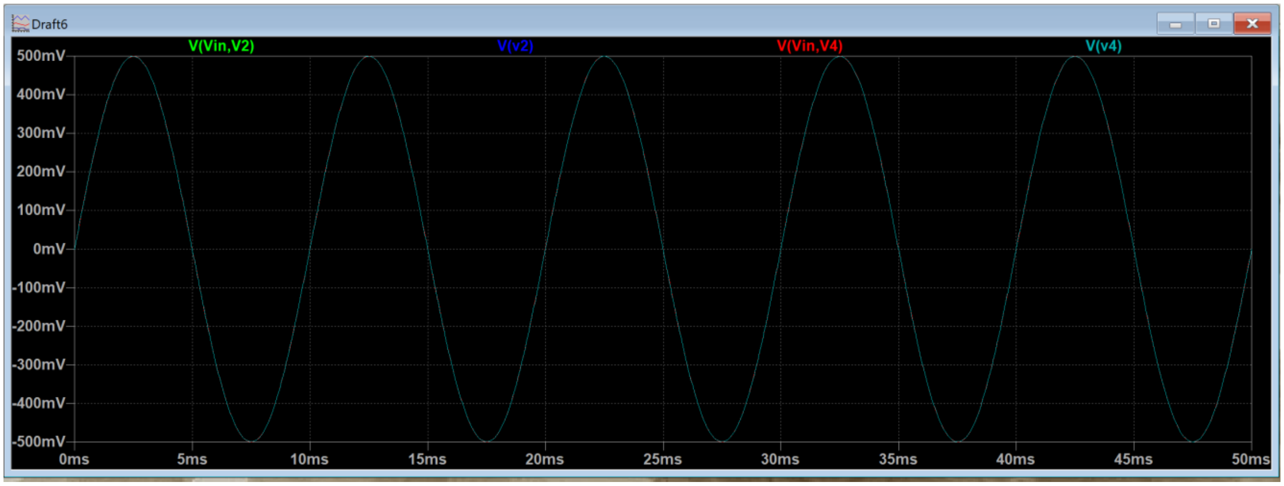



*Figure 41 Transient Analysis of Modified AC Circuit Voltages Across Each Resistor*

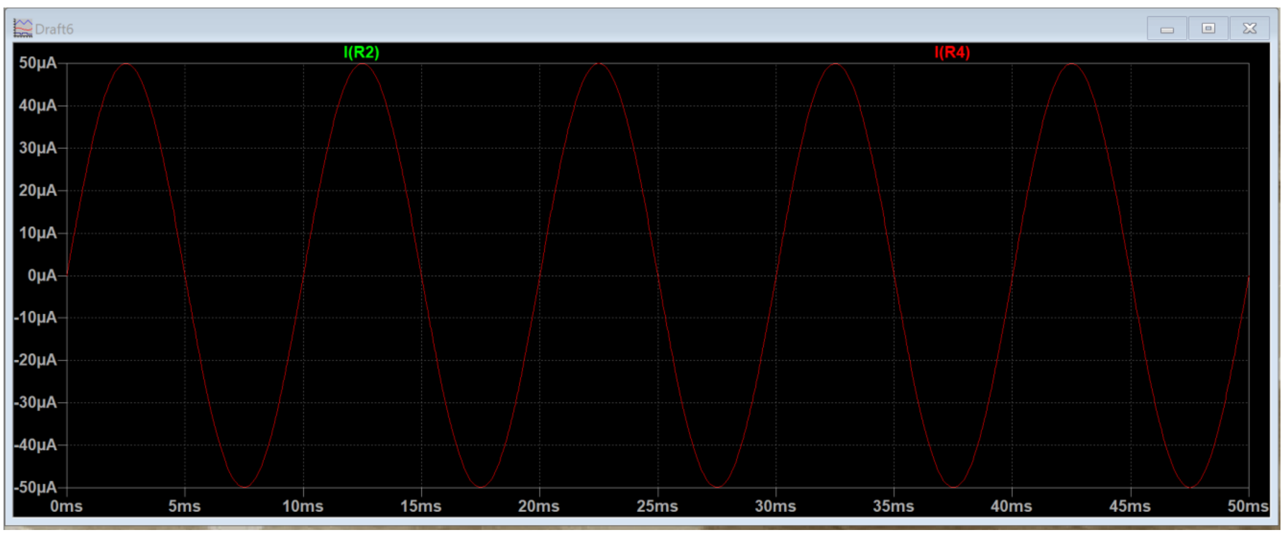



*Figure 42 Figure 42 Transient Analysis of Modified AC Circuit Current Across Resistor R2 and R4*

B.)

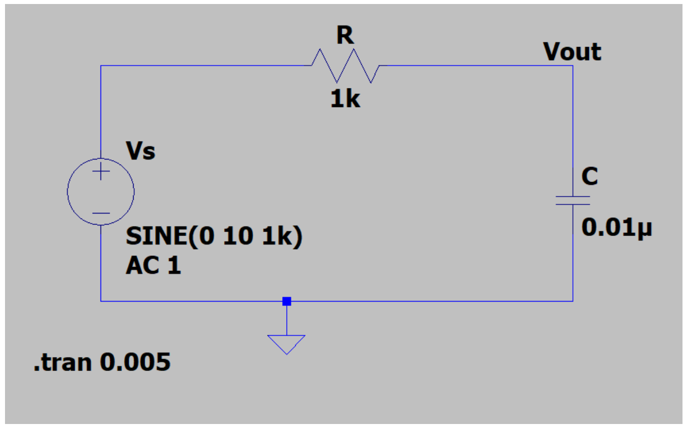



*Figure 43 Modified Low Pass Filter*

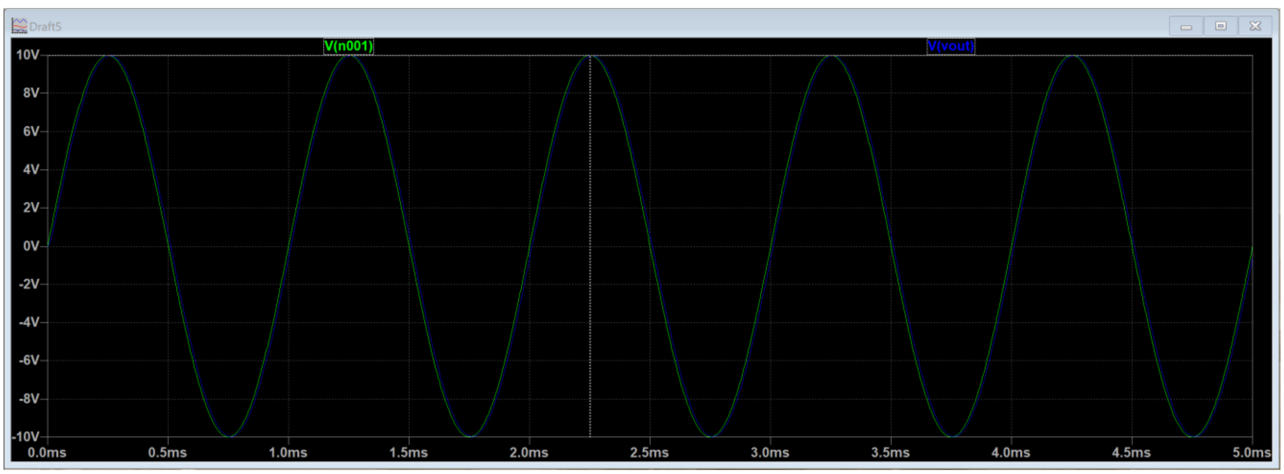



*Figure 44 Modified Low Pass Filter Transient Analysis*

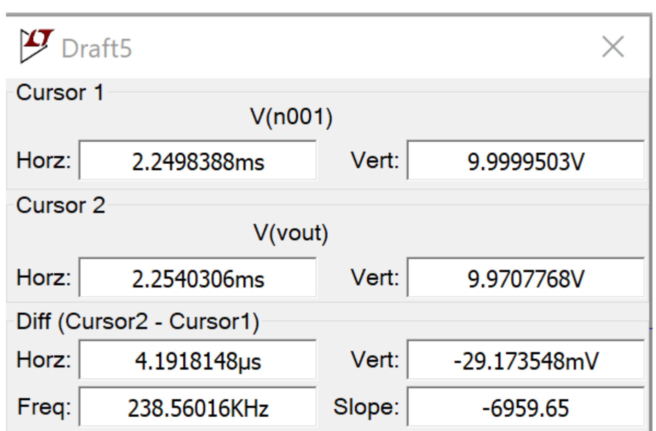



*Figure 45 Modified Low Pass Filter Transient Analysis Data*

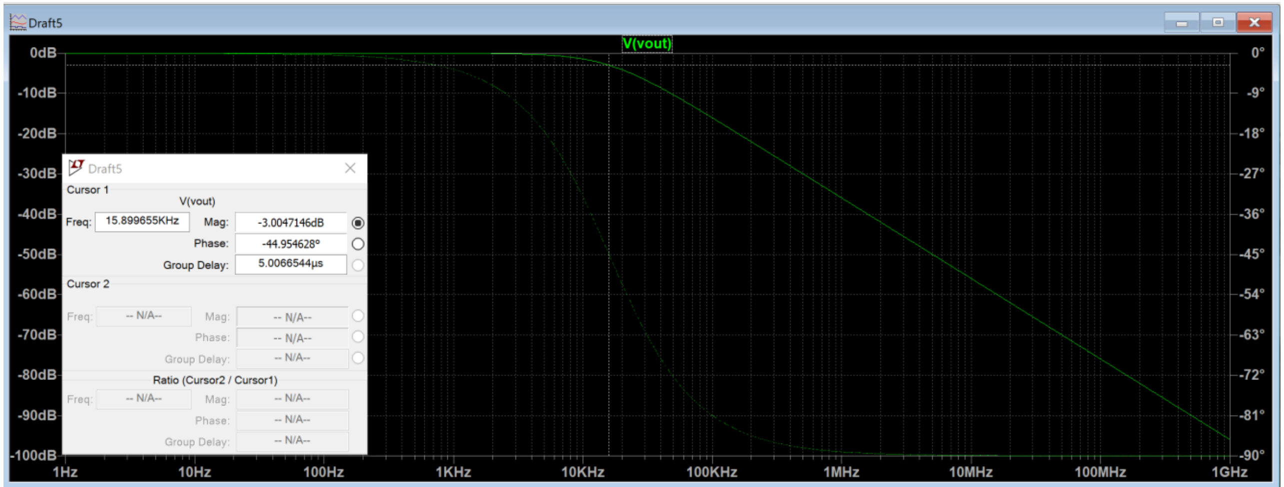



*Figure 46 Modified Low Pass Filter AC Analysis*

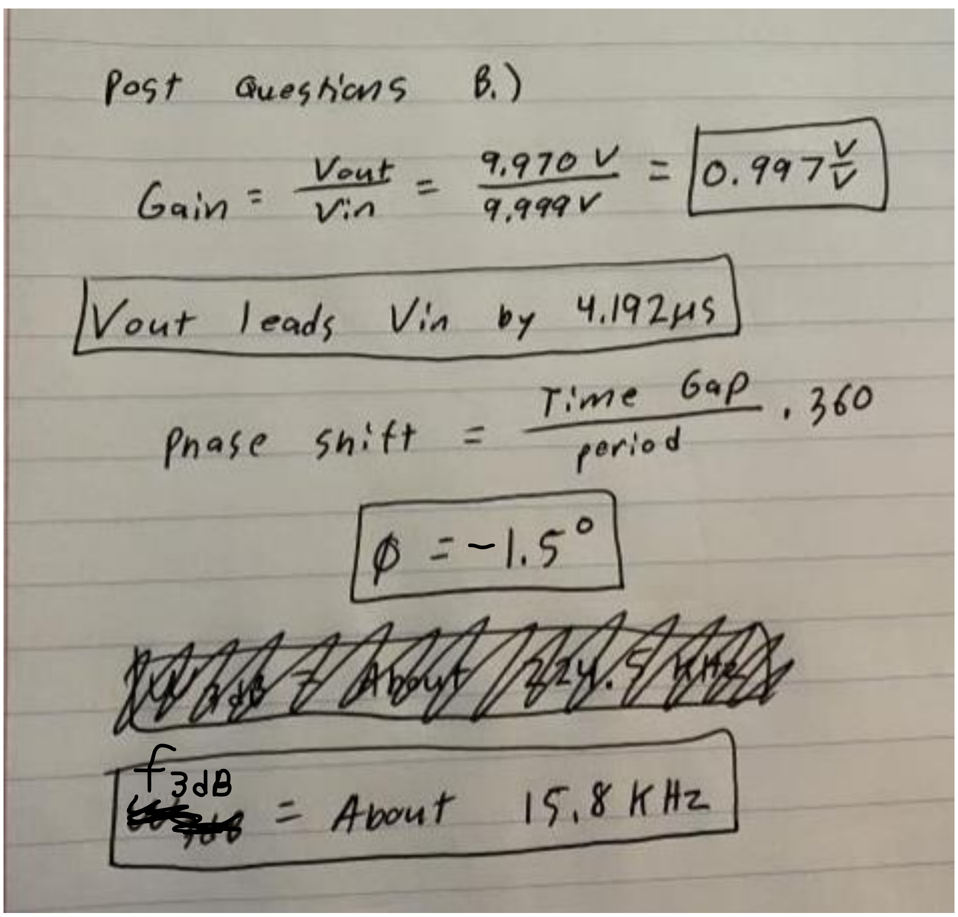



*Figure 47 Calculations for Post Question 3b*

From the calculations above it can be seen that with the changes implemented the most apparent changes are a higher gain and a very small phase shift when compared to the Low Pass Filter Analysis in the Lab.

C.)

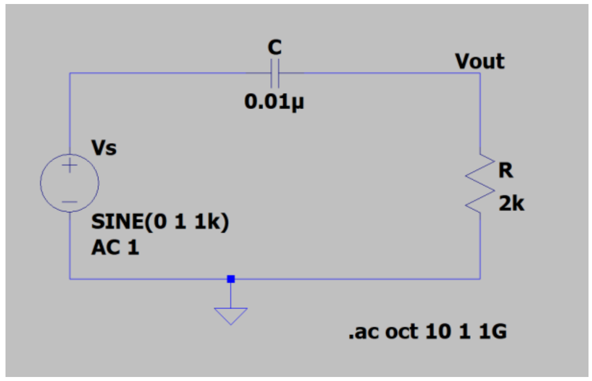



*Figure 48 Modified High Pass Filter*

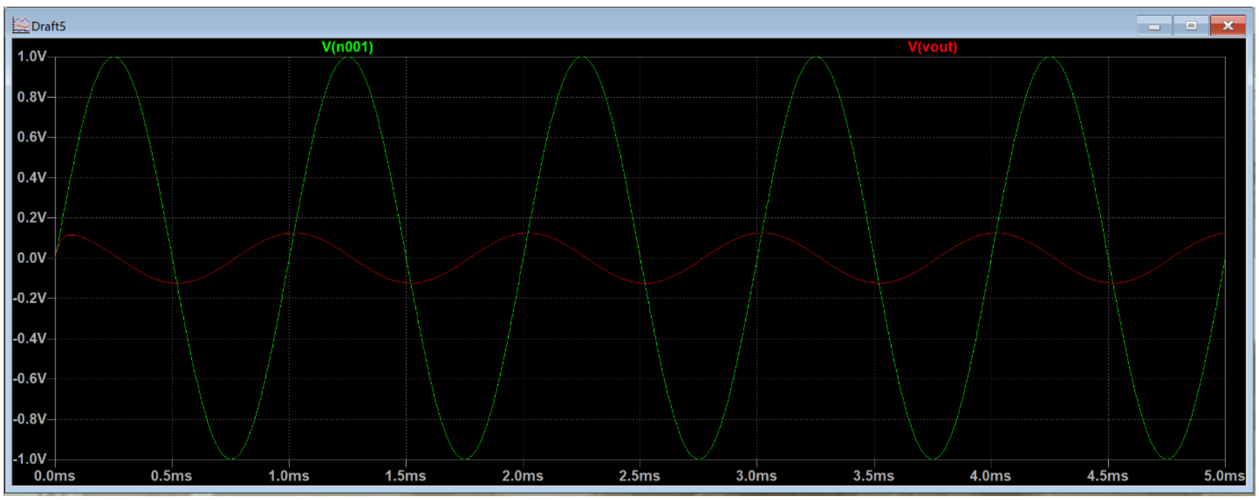



*Figure 49 Modified High Pass Filter Transient Analysis*

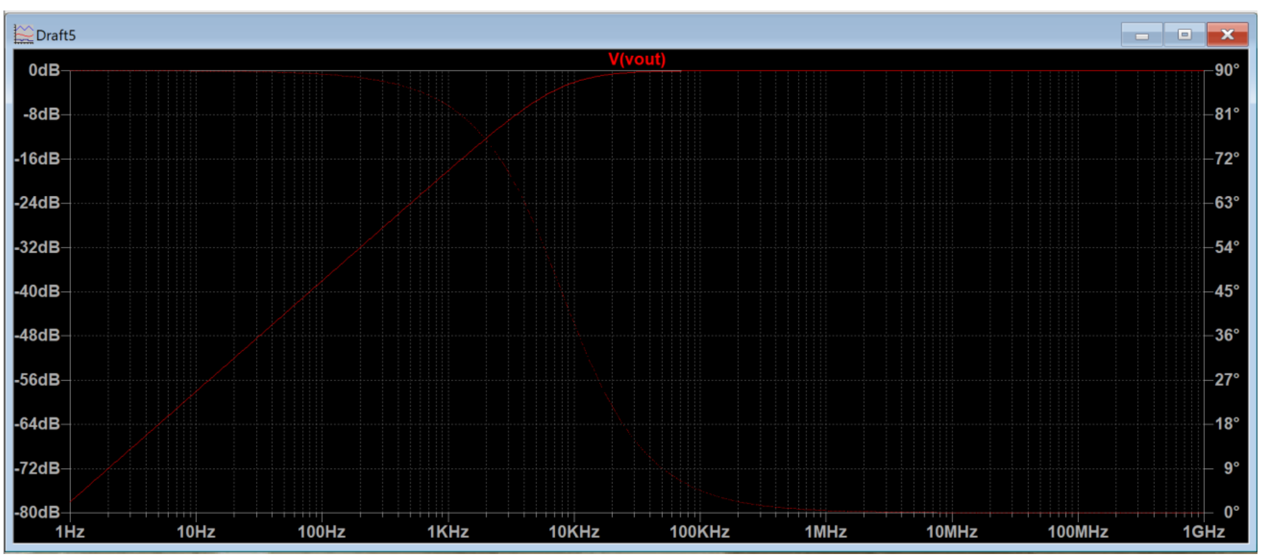



*Figure 50 Modified High Pass Filter AC Analysis*

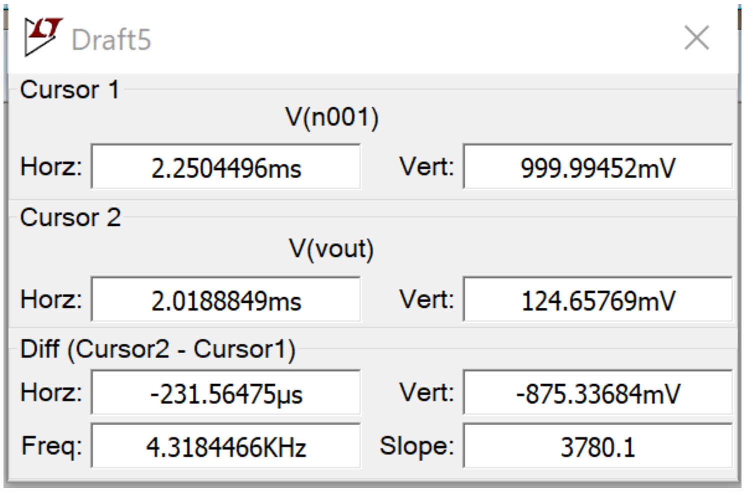



*Figure 51 Modified High Pass Filter Transient Analysis Data*

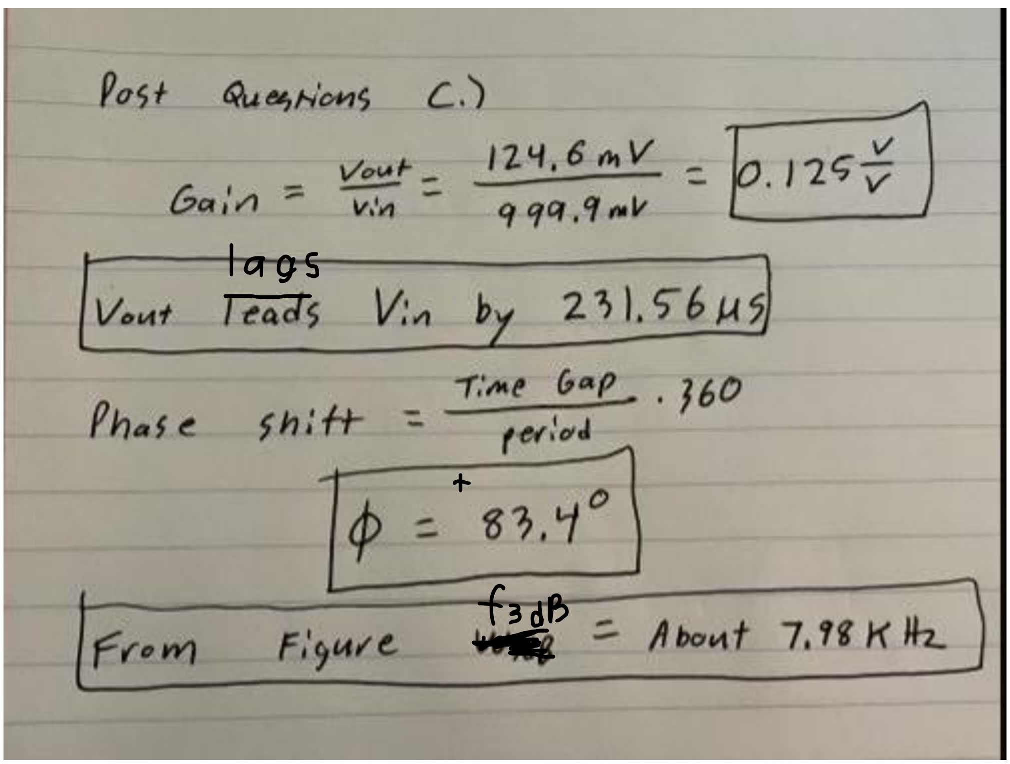



*Figure 52 Calculations for Post Question 3c*

From the calculations above it can be seen that with the changes implemented to the high pass filter the changes are a very low gain and a very large phase shift when compared to the High Pass Filter Analysis in the Lab. This could issues resulting in a non-ideal high pass filter.

4. i. $R = \left(10 * (1) + (0)\right)(100) \pm 5\%$

**R = 1kΩ** $\pm$ **50**Ω

ii. $R = \left(10 * (4) + (4)\right)(10000) \pm 5\%$

**R = 440kΩ** $\pm$ **22*k***Ω

iii. $R = \left(10 * (1) + (0)\right)(1) \pm 5\%$

**A = Brown, B = Black, C = Black, D = gold, White, or Green**