EEE 117L Laboratory – Network Analysis
Lab #6: Operational Amplifiers (Week II)

Lab Day and Time: Wednesday 1:30 - 4:10

Group Number: # 03

Group Members: (Last Name, First Name)

Member #1: Algador, Vigomar Kim

Member #2: Chan, Casey

Member #3: Bon, Trinh

Total Score: /100

Work Breakdown Structure: It is important that every group member do their share of the work in these labs. Remember that you will receive no credit for the lab worksheet if you did not contribute. Write in the Table provided below, which group member(s) contributed to the solution of each problem in the lab worksheet. Also remember that only on lab worksheet per group will be turned in to Canvas. If there was any group member that did not contribute, then write their name in the space provided below.

Problem Number	Group member(s) that worked on the problem.
Part Ia	Algador, Vigomar Kim Chan, Casey Trinh, Bon
Part Ib	Algador, Vigomar Kim Chan, Casey Trinh, Bon
Part Ic	Algador, Vigomar Kim Chan, Casey Trinh, Bon
Part IIa	Algador, Vigomar Kim Chan, Casey Trinh, Bon
Part IIb	Algador, Vigomar Kim Chan, Casey Trinh, Bon
Part IIc	Algador, Vigomar Kim Chan, Casey Trinh, Bon
Part III	Algador, Vigomar Kim Chan, Casey Trinh, Bon

Absent member(s):____

General Instructions:

1) To work properly, operational amplifiers (op-amps) must be powered externally by both a positive and a negative voltage supply, typically labelled $+\ V_{CC}$ and $-\ V_{CC}$ respectively. When connecting the power supplies to the op-amp, DO NOT mix the two power sources up or it is almost guaranteed that you will irreversibly damage the op-amp and potentially other circuit components including the power supply.

2) If you ever smell something funny or burning, turn off the voltage sources from the AD2 immediately!

3) If the voltage source does not allow you to increase the voltage using the dial, then there is something wrong with your circuit and you have to go back and check your connections and components. This is usually an indication that too much current is flowing through your circuit and it could potentially damage it if it isn't fixed.

4) For all circuits that contain op-amps, it is usually a good idea to connect the external power supplies before connecting other components. After the power supplies, it's a good idea to then connect any ground wires. To connect the rest of the components (especially for the more complex circuits), it is usually a good idea to identify and label nodes in the original circuit rather than trying to link up components.

5) Remember that Ch1 and Ch2 refer to channels one and two of the oscilloscope, respectively.

6) A voltage gain, sometimes labelled A_V, is by definition the ratio of the voltage output to the voltage input $A_V = \frac{V_{out}}{V_{in}}$.

Part I. Strain-Gauge Amplifier Total Score: ____/60

a) Preliminary Measurements: Score: ____/5

This section of the lab involves the circuit shown in Figure 1 below. This is what is known as a strain-gauge amplifier circuit. Before simulating or building the circuit, the values of the resistors R_0 and R_1 must be measured, along with the varying values of ΔR. Do so and record the values in Table 1. Use the average values of R_0 and R_1 in the Table and in the simulations.

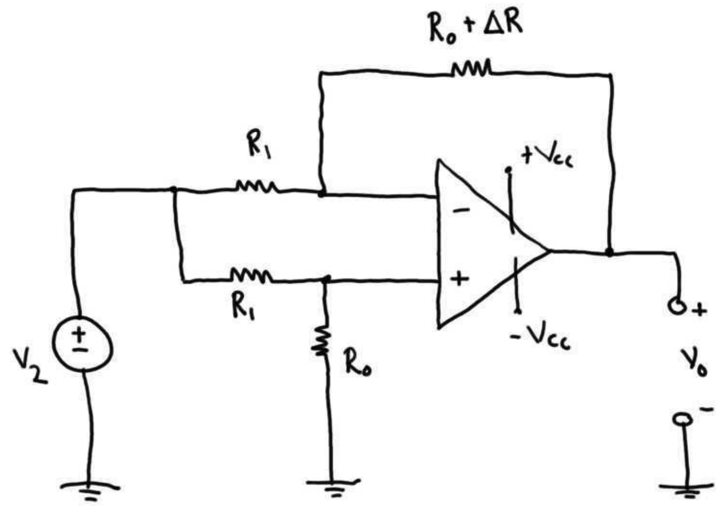


Figure 1. Strain-Gauge Amplifier Circuit

Strain-gauge Amplifier	Theoretical Value	Experimental Value
R_0	1 kΩ	0.996 0.995
R_1	2 kΩ	1.990 1.979
ΔR_1	0.2 kΩ	0.198
ΔR_2	1 kΩ	0.990
ΔR_3	4.3 kΩ	4.292

Table 1. Resistor Values for Strain-Gauge Amplifier

b) Prediction: Simulation with a Parametric Sweep Score: ____/20

Using PSpice, simulate the circuit shown in Figure 1 using a transient analysis from $0 \leq t \leq 3T$, where T is the period of the input voltage V_2. Choose the time step such that there are at least 500 data points in the simulation. In your simulation set the external power supplies to $+V_{CC} = 5\ V$ and $-V_{CC} = -5\ V$, while setting the input voltage V_2 to a sinusoidal signal with a frequency of 500 Hz and a peak-to-peak value of 2.0 V. Use the "Vsin" voltage source in PSpice to generate the sinusoidal voltage. Also use the experimental values of the resistors found in the previous section. Perform a parametric sweep across the three values of ΔR. For convenience, a brief tutorial explaining how to perform a parametric sweep is included in the appendix at the end of this worksheet. For the output, on the same graph show the input voltage V_2 and the output voltages V_0 for each value of ΔR. Include pictures of 1) the circuit schematic and 2) the output graph of the simulation below.

1) Circuit Schematic:

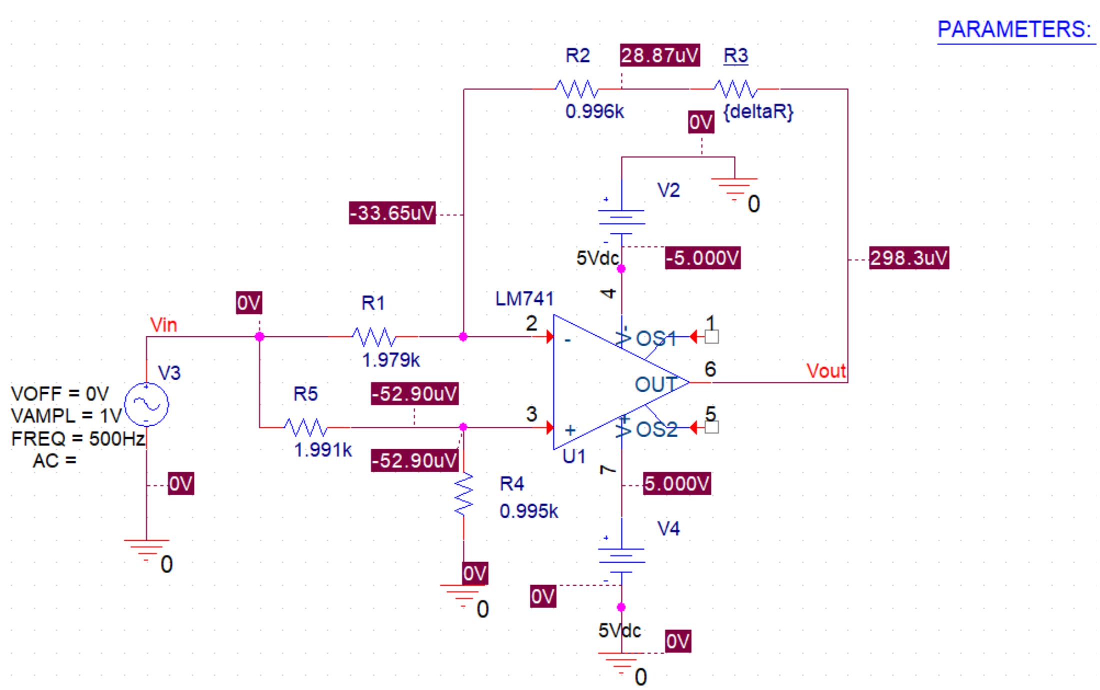


Figure 1.1. Pspice Circuit Schematic

2) Output:

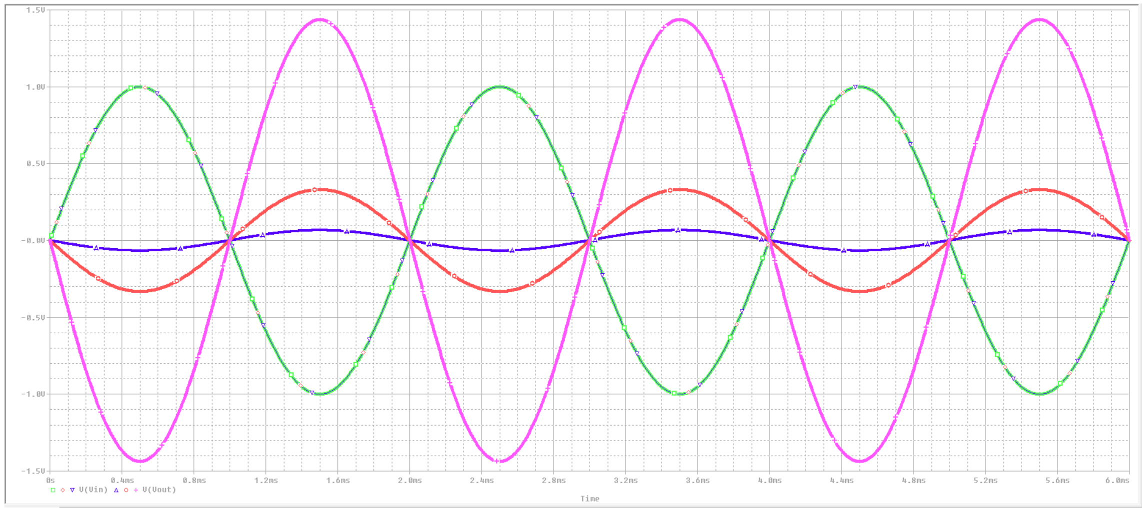


Figure 1.2. Pspice Simulation

In figure 3, We can see V_{in} as the green colored wave and V_{out} for blue, pink, and red. For V_{out}, we used ΔR_1 to output the blue colored wave, ΔR_2 to output the pink colored wave, and ΔR_3 to output the red colored wave.

c) Data Collection and Analysis Score: ____/35

Assemble the circuit shown in Figure 1 on your breadboard. Use the function generator to generate the input voltage V_2 which is a sinusoidal signal with a frequency of 500 Hz and a peak-to-peak value of 2.0 V. Set the external power supplies to $V+ \quad = + V_{CC} = 5\,V$ and $V- \quad = - V_{CC} =- 5\,V$. Use Ch1 to monitor the input voltage V_2 and Ch2 to monitor the output voltage V_0. Set the time base on the oscilloscope so that three full waveforms are shown. Show the experimental output coming from the oscilloscope below.

Experimental output: $\Delta R = 0.2\ k\Omega$

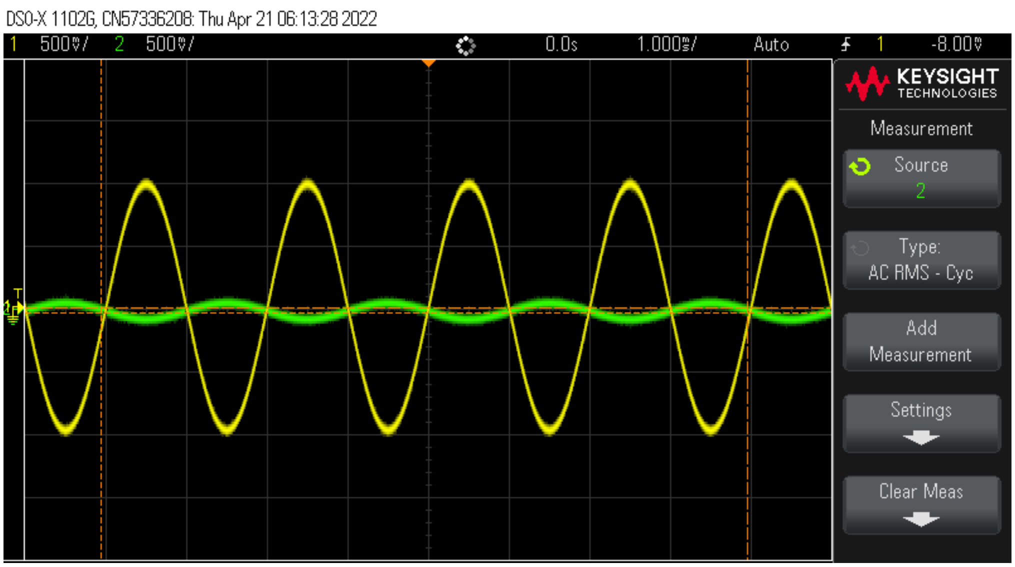


Figure 1.3. Oscilloscope waveform output for $\Delta R = 0.2\ k\Omega$

Experimental output: $\Delta R = 1\ k\Omega$

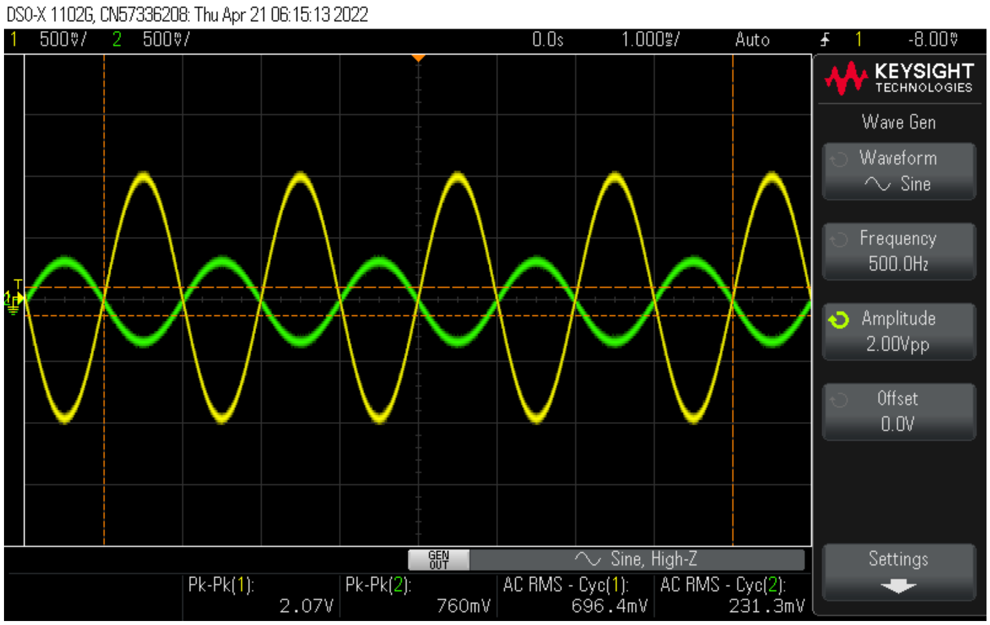


Figure 1.4. Oscilloscope waveform output for $\Delta R = 1\ k\Omega$

Experimental output: $\Delta R = 4.3\ k\Omega$

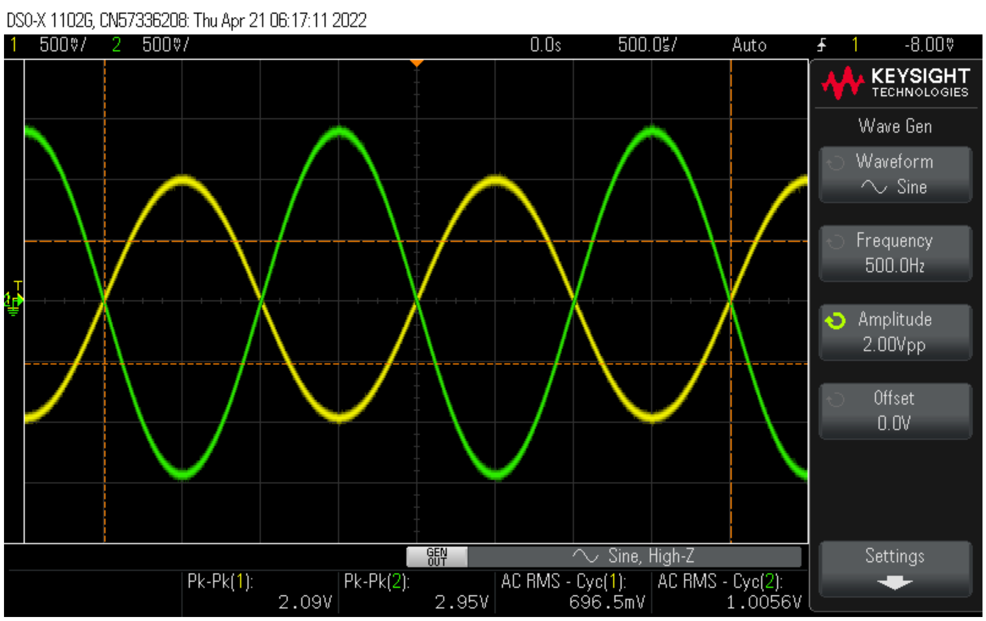


Figure 1.5. Oscilloscope waveform output for $\Delta R = 4.3\ k\Omega$

Question: How does this compare to the simulated results?

The experimental outputs done on the oscilloscope shown in figures 1.3, 1.4, and 1.5 matched with the simulated results on PSpice shown in figure 1.2.

Using your results from the pre-lab, calculate the theoretical values of the missing parameters in Tables 2a-2c for each value of ΔR. Using the oscilloscope, take the measurements missing from Tables 2a-2c. Calculate the percent error for each parameter and enter it into the last column of Table 2. Show all your work. Include more pages if needed.

$\Delta R = 0.2\ k\Omega$	Theoretical Value	Experimental Value	Percent Error
f (Hz)	500	500	0%
Amplitude of V_2 - Scope	1.0 V	1.03 V	0.03%
Amplitude of V_0 - Scope	66.7 mV	68.2 mV	2.25%
Gain - Scope	0.0667	0.0659	1.20%

Table 2a. Comparison of theoretical results to experimental results

$\Delta R = 1\ k\Omega$	Theoretical Value	Experimental Value	Percent Error
f (Hz)	500	500	0%
Amplitude of V_2 - Scope	1.0	1.035	3.5%
Amplitude of V_0 - Scope	333 mV	333.7 mV	0.21%
Gain - Scope	0.333	0.3332	0.06%

Table 2b. Comparison of theoretical results to experimental results

$\Delta R = 4.3\ k\Omega$	Theoretical Value	Experimental Value	Percent Error
f (Hz)	500	500	0%
Amplitude of V_2 - Scope	1.0	1.04	4%
Amplitude of V_0 - Scope	1.43	1.47	2.79%
Gain - Scope	1.43	1.46	2.098%

Table 2a. Comparison of theoretical results to experimental results

Theoretical Calculations:

a) For $\Delta R = 0.2\ k\Omega$

$V_0 = \frac{\Delta R}{R_0 + R_1} * V_{in} = \frac{0.2k}{1k + 2k} * 1 = 0.0667$ V

Gain = $\frac{V_{out}}{V_{in}} = \frac{0.0667}{1} = 0.0667$

f(Hz): $\frac{|500-500|}{|500|} = 0\%$

Amplitude of V_2: $\frac{|1.03-1.0|}{|1.0|} = 0.03\%$

Amplitude of V_0: $\frac{|68.2-66.7|}{|66.7|} = 2.25\%$

Gain: $\frac{|0.0659-0.0667|}{|0.0667|} = 1.20\%$

b) For $\Delta R = 1\ k\Omega$

$V_0 = \frac{1k}{1k + 2k} * 1 = 0.33$ V

Gain $= \frac{0.33}{1} = 0.33$

Amplitude of $V_2 = \frac{|1.035-1.0|}{|1.0|} = 3.5\%$

Amplitude of $V_0 = \frac{|0.-0.33|}{|0.33|} = 56.66\%$

Gain = $\frac{|0.3332-0.333|}{|0.333|} = 0.06\%$

c) For $\Delta R = 4.3\ k\Omega$

$V_0 = \frac{4.3k}{1k + 2k} * 1 = 1.43$ V

Gain $= \frac{1.43}{1} = 1.43$

f(Hz) = $\frac{|500-500|}{|500|}$ = 0

Amplitude of V_2 = $\frac{|1.04-1.0|}{|1.04|}$ = 4%

Amplitude of V_0 = $\frac{|1.47-1.43|}{|1.43|}$ = 2.79%

Gain = $\frac{|1.46-1.43|}{|1.43|} =$ 2.098%

Part II. Weighted Summer Total Score: ____/30

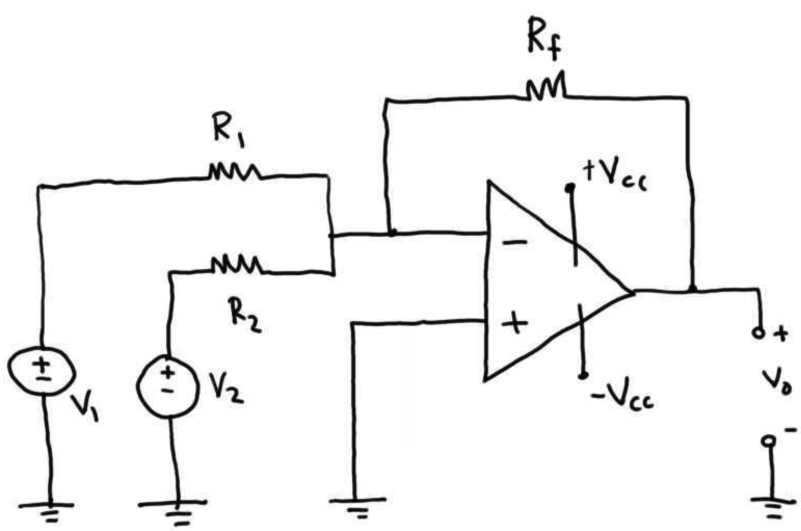


Figure 2. Weighted Summer Amplifier Circuit

a) Preliminary Measurements: Score: ____/5

This section of the lab involves the circuit shown in Figure 2. Before simulating or building the circuit, the values of the resistors R_1, R_2, and R_f must be measured. Do so and record the values in Table 3. Use these experimental values in your simulations and theoretical calculations.

Weighted Summer	Theoretical Value	Experimental Value
R_1	2 kΩ	1.990k
R_2	1 kΩ	0.995k
R_f	1 kΩ	0.994k

Table 3. Resistor Values for Weighted Summer Amplifier Circuit

b) Prediction: Simulation Score: ____/10

Using PSpice, simulate the circuit shown in Figure 2 using a transient analysis from $0 \le t \le 3T$, where T is the period of the input voltage V_1. Choose the time step such that there are at least 500 data points in the simulation. In your simulation set the external power supplies to $+V_{CC} = 5\ V$ and $-V_{CC} = -5\ V$, while setting the input voltage V_1 to a sinusoidal signal with a frequency of 500 Hz and a peak-to-peak value of 2.0 V. Use a constant DC voltage of 2.0 V for the input voltage V_2. Use the "Vsin" voltage source in PSpice to generate the sinusoidal voltage. Also use the experimental values of the resistors found in the previous section. For the output, on the same graph show the input voltage V_1 and the output voltage V_0. You need not include the input voltage V_2. Include pictures of 1) the circuit schematic and 2) the output graph of the simulation below.

1) Circuit Schematic:

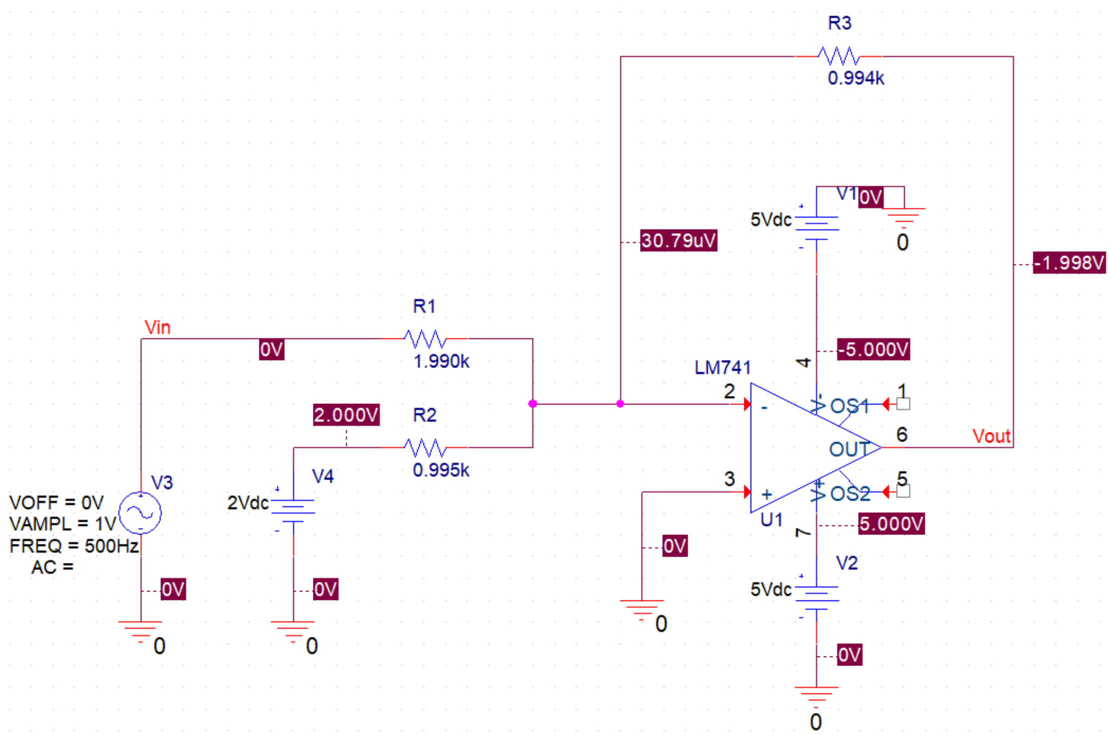


Figure 2.1. Pspice Circuit Schematic

2) Output:

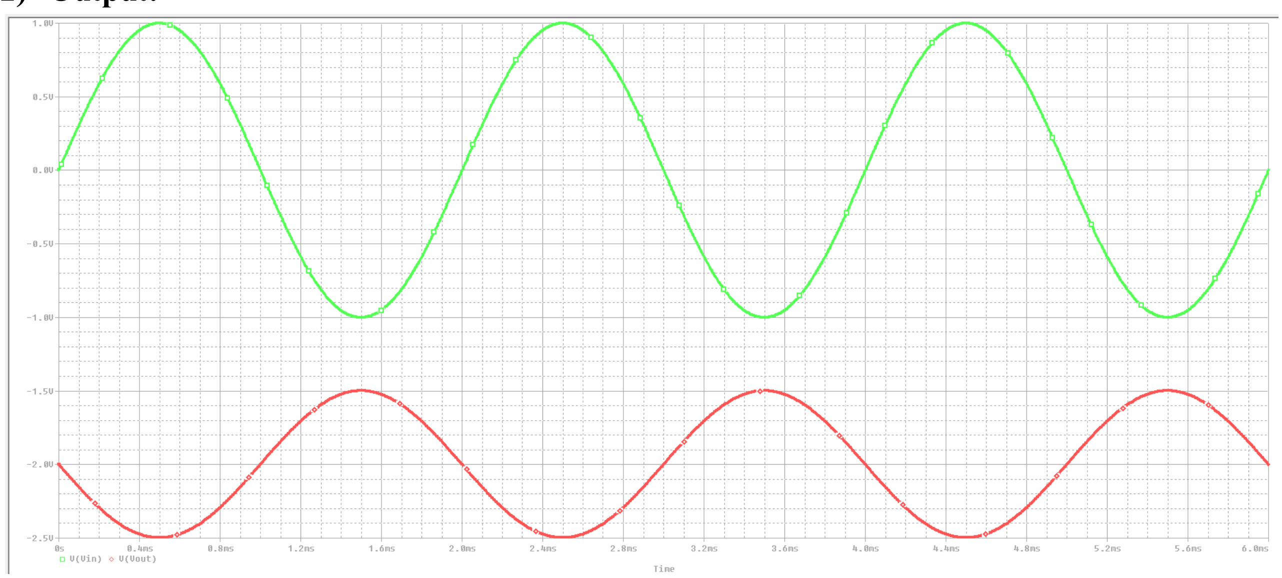


Figure 2.2. Pspice Simulation

c) Data Collection and Analysis Score: ____/15

Assemble the circuit shown in Figure 2 on your breadboard. Use the function generator to generate the input voltage V_1 which is a sinusoidal signal with a frequency of 500 Hz and a peak-to-peak value of 2.0 V. Use the 6-V power supply to generate the input voltage $V_2 = 2.0\,V$. Set the external power supplies to $V+ \;\; = +V_{CC} = 5\,V$ and $V- \;\; = -V_{CC} =- 5\,V$. Use Ch1 to monitor the input voltage V_1 and Ch2 to monitor the output voltage V_0. The input voltage V_2 does not need to be monitored. Set the time base on the oscilloscope so that three full waveforms are shown. Show the experimental output coming from the oscilloscope below.

Experimental output:

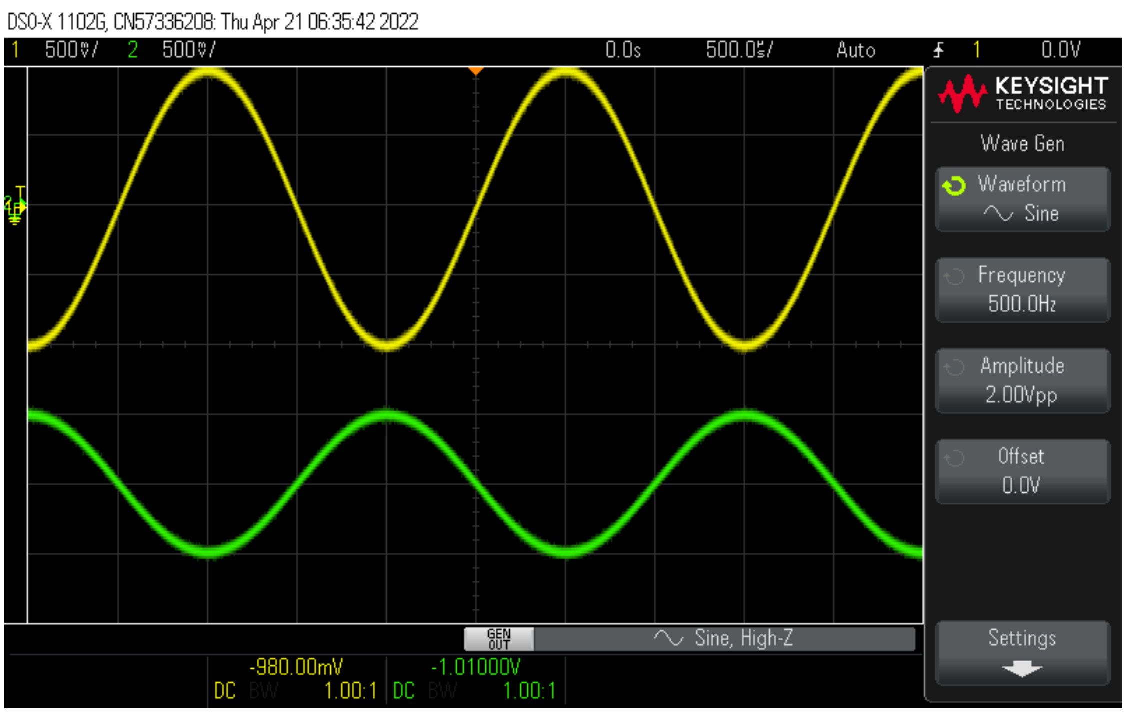
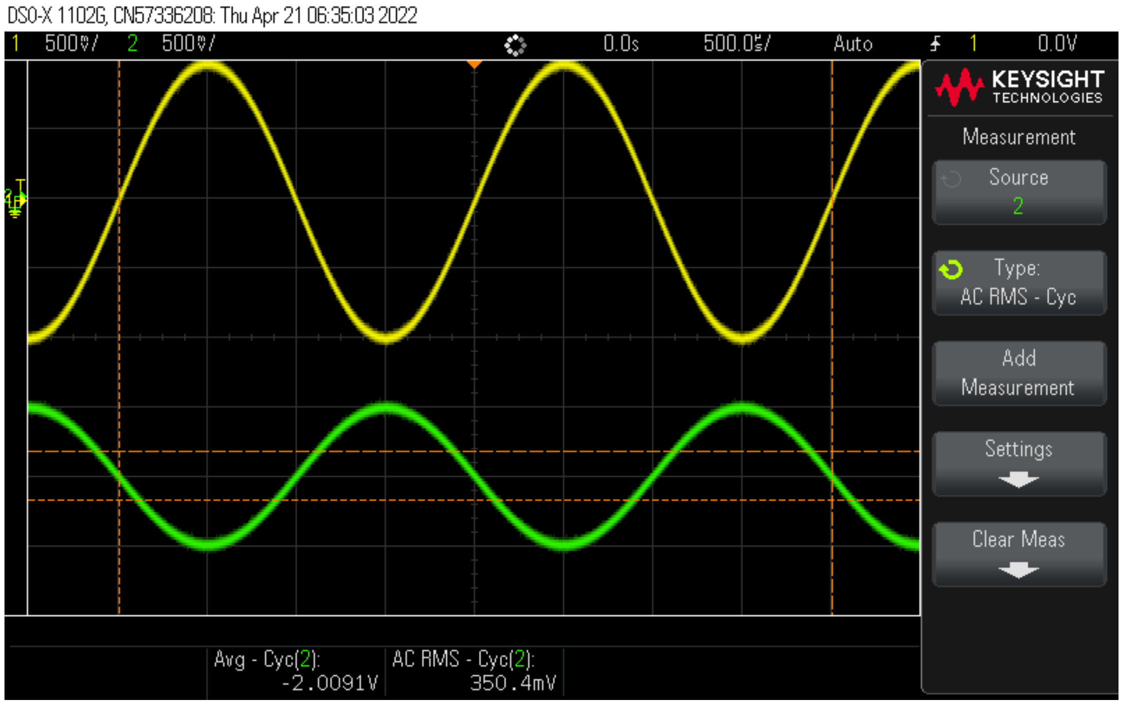


Figure 2.3. Oscilloscope waveform output



Question: How does this compare to the simulated results?

The waveform from the function generator matched with the simulated results from PSpice shown in figure 2.2.

Using your results from the pre-lab, calculate the theoretical values of the missing parameters in Tables 3. Using the oscilloscope and DMM take the measurements missing from Table 3. Calculate the percent error for each parameter and enter it into the last column of Table 3. Show all your work. Include more pages if needed.

Weighted Summer	Theoretical Value	Experimental Value	Percent Error
Average V_0 (V) - Scope	-2 V	-2.0091 V	0.0045%
RMS V_0 (V) - Scope	353.55 mV	350.4 mV	0.89%
DC Setting V_0 (V) - DMM	-2 V	-1.99 V	0.5%
AC Setting V_0 (V) - DMM	0 V	0.345 V	undefined

Table 3. Comparison of theoretical results to experimental results

Discuss the various measurements above. In particular, compare the measurements taken by the oscilloscope and the measurements taken by the DMM. Which ones match?

The measurements taken by the oscilloscope and measurements taken by the DMM were all within an acceptable percent error.

Calculations:

RMS $V_0 = \frac{1}{2\sqrt{2}} = 0.353$

Average $V_0 = \frac{|-2.0091-(-2)|}{|-2|} = 0.0045\%$

RMS $V_0 = \frac{|350.4-353.55|}{|353.55|} = 0.89\%$

DC Setting $V_0 = \frac{|-1.99-(-2)|}{|-2|} = 0.5\%$

AC Setting $V_0 = \frac{|0.345-0|}{|0|} = \text{undefined}$

Part III. Conclusions Total Score: /10

Explain in a few paragraphs the purpose of the lab, the experimental set up and methodology, and central results of the lab and these experiments. **You should be quantitative** in this summary. Include any important equations used and explain their significance. Write the conclusion as if you were writing an English essay. This is an important portion of the lab, so make sure to do a good and thorough job.

In this lab, we are continuing to work with operational amplifiers. For the first part of the lab, we worked with a strain-gauge amplifier. ΔR has three varying values, 0.2 kΩ, 1 kΩ, and 4.3 kΩ. For the simulation, we used a parametric sweep so that all three results will be on the graph at the same time. We then used a function generator to generate the input and output voltage onto the oscilloscope. Our experimental results matched with our predicted simulation on PSpice. When building the circuit we had a problem with the output voltage not being displayed correctly. This was due to the circuit not being connected to the ground from the voltage generator.

In the second part of the lab, we worked with a weighted summer amplifier circuit. This circuit has two voltage sources. We only had to keep track of input voltage V_1. When building the circuit on PSPice, V_1 would be a sinusoidal signal and V_2 would be a DC voltage source. Transient analysis was used when simulating the circuit. We then built the circuit on a breadboard and used a function generator to generate the input and output voltage onto the oscilloscope. The results from the function generator and the simulation were the same.

Appendix: Parametric Sweep Tutorial

Below can be found two YouTube tutorials for PSpice that may be useful in helping you perform the transient analysis with a parametric sweep across the values of ΔR for the strain-gauge amplifier circuit. Also included is an algorithm that you can follow to perform the same simulation. Use whichever you find the most useful. Also remember that the Zoom session for the lab is being recorded and will be uploaded to Canvas a few hours after the lab ends.

YouTube Tutorials:

1. Basic one: RLC circuit
   https://www.youtube.com/watch?v=7Fsrn_f5154
2. A little more sophisticated: Common emitter amplifier circuit
   https://www.youtube.com/watch?v=vaiRDVM9oy8

Algorithm: Parametric Sweep

1. Choose PARAM under Parts List
2. Add "PARAMETERS" to workspace
3. Double click on the value of the variable you wish to vary (Not the part itself)
4. In the Value box type {Variable Name} with curly brackets. You can choose any variable name and must include the curly brackets.
5. Double click on the PARAMETERS in the workspace
6. Find "Add New Property" at the top left of the menu that shows up. You will need to scroll until you find it.
7. In the Name type in the Variable Name (with no curly brackets)
8. Type "Initial Value" in the Value box (which will be changed later). You can type in "1" for convenience, if you like. No quotation marks should be included in the value you type in.
9. Define a simulation profile and make sure that "Parametric Sweep" box is checked off under the simulation type (Transient Analysis, DC Sweep, AC Sweep, Bias Point)
10. Under "Sweep Variable" make sure that it is defined as a "Global Variable" and under "Parameter Name" make sure you find the Variable Name that you chose.
11. In the Value list box go ahead and specify the values to be swept through. The values should be listed with a comma to separate the values and with no space. For example, you could type in the following: Value list - 0.2k,1k,4.2k
12. When you plot the output graphs the traces for each of the values of the sweep should automatically appear on the same graph.