Experiment 7
The Operational Amplifier

Objectives
In this experiment, you will investigate several useful operational amplifier circuits, both analytically and via simulation with LTspice.

Note: Use netlists to "build" the circuits needed here, and please make sure that you include all netlists in your report.

Exercise 1
For the circuit shown in Figure 1, assume the OpAmp is ideal and calculate the output voltage for each of the six values of V_s listed in the first column of Table 1(a). Record your results in the second column of the table.

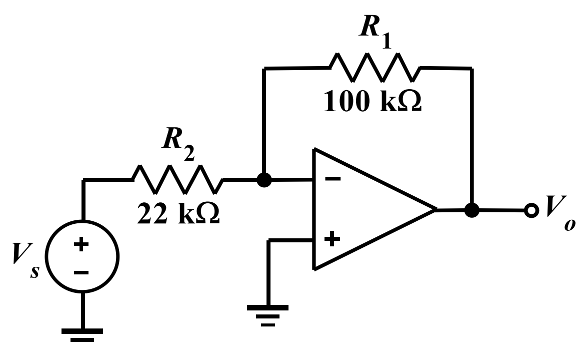

Figure 1

Now, do the following:
1. Build the circuit in LTspice. For the operational amplifier, use the LM741 model from the LTspice subcircuit library. Use $\pm 15\,\mathrm{V}$ supplies to power the OpAmp.
2. Perform an operating point simulation to determine V_o and V_- for each of the six values of V_s and compare the results by completing Table 1. Note that "% Error" is defined as:

$$\%\ Error = \frac{(Ideal\ V_o) - (V_o\ \ from\ LTspice)}{(V_o\ \ from\ LTspice)}$$

because the simulation result is assumed to be more accurate than the ideal estimate.

Table 1(a)

V_s **(V)**	**Ideal** V_o **(V)**	V_o **from LTspice (V)**	ΔV_o **(V)**	**% Error**
0.4				
2.0				
3.5				
-0.6				
-1.6				
-2.4				

3. What would you expect for the voltage at the inverting input (V_-) in each case if the OpAmp were ideal? What are the simulation results? Use Table 1(b) to record your results.

Table 1(b)

V_s **(V)**	**Ideal** V_- **(V)**	V_- **from LTspice (V)**
0.4		
2.0		
3.5		
-0.6		
-1.6		
-2.4		

4. Can you estimate the gain of the operational amplifier (not the circuit, but the OpAmp itself) based on the simulation results? (This is the A_v coefficient of the VCVS in the OpAmp model considered in class.)

   Estimated gain of the OpAmp: A_v = ______________________

   Explain how you determined this estimate:

5. Using your estimate from step 4, calculate the maximum allowed value of V_s that can be applied without causing amplifier saturation. Then perform a simulation to determine that limit, and compare it with your theoretical calculation.

Calculated V_s limit: ____________________

V_s limit from LTspice: ____________________

Exercise 2

Change V_s to a sinusoidal source as shown in Figure 2.

Table 2

Characteristics of v_s	
Sinusoidal with DC offset	
Peak-to-peak amplitude	400 mV
Offset	500 mV
Frequency	1 kHz
Phase angle	0°

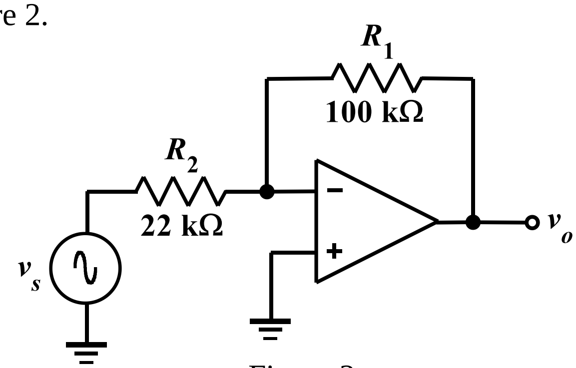

Figure 2

Now, do the following:

1. Using LTspice, again with the LM741 OpAmp model, perform a transient analysis for 10ms. Plot the output and the input in the same graph and explain your simulation results.
2. Change the offset to -500mV and redo the simulation. Plot the output and the input in the same graph and explain your simulation results.
3. Change the offset to 2 V and the peak-to-peak amplitude to 1V and redo the simulation. Plot the output and the input in the same graph and explain your simulation results.

Exercise 3

For the circuit shown in Figure 3, assume the OpAmp is ideal and calculate the output voltage for each of the six values of V_s listed in the first column of Table 3(a). Enter your results in the second column of the table.

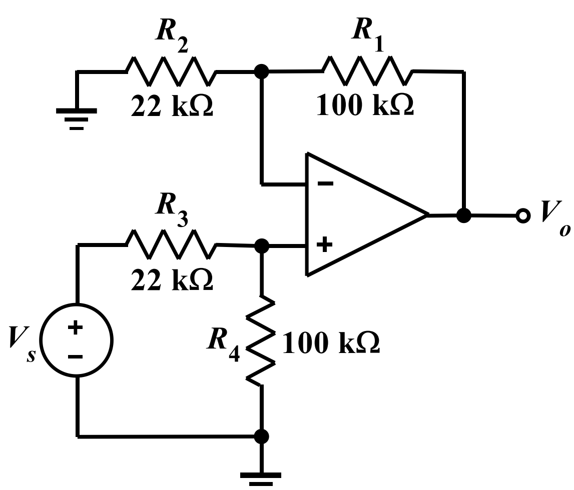

Figure 3

Table 3(a)

V_s (V)	Ideal V_o (V)	V_o from LTspice (V)	% Difference
0.4			
2.0			
3.5			
-0.6			
-1.6			
-2.4			

Now, do the following:

1. Build the circuit in LTspice using the LM741 OpAmp model.
2. Perform an operating point simulation for each of the six values of V_s and compare the results by completing Table 3(a).
3. What do you expect for the voltage at the inverting input for each of the values of V_s if the operational amplifier is ideal? What are the simulation results? Enter your results in Table 3(b).

Table 3(b)

V_s (V)	Ideal V_- (V)	V_- from LTspice (V)	% Difference
0.4			
2.0			
3.5			
-0.6			
-1.6			
-2.4			

4. Calculate the maximum allowed value of V_s that will avoid amplifier saturation. Then perform a simulation to determine that limit and compare it with your theoretical calculation.

Calculated V_s limit: ____________________

V_s limit from LTspice: ____________________

Exercise 4

For the circuit shown in Figure 4, assume the OpAmps are ideal and calculate the output voltage for each of the five values of R_7 listed in the first column of Table 4(a). Enter your results in the second column of the table.

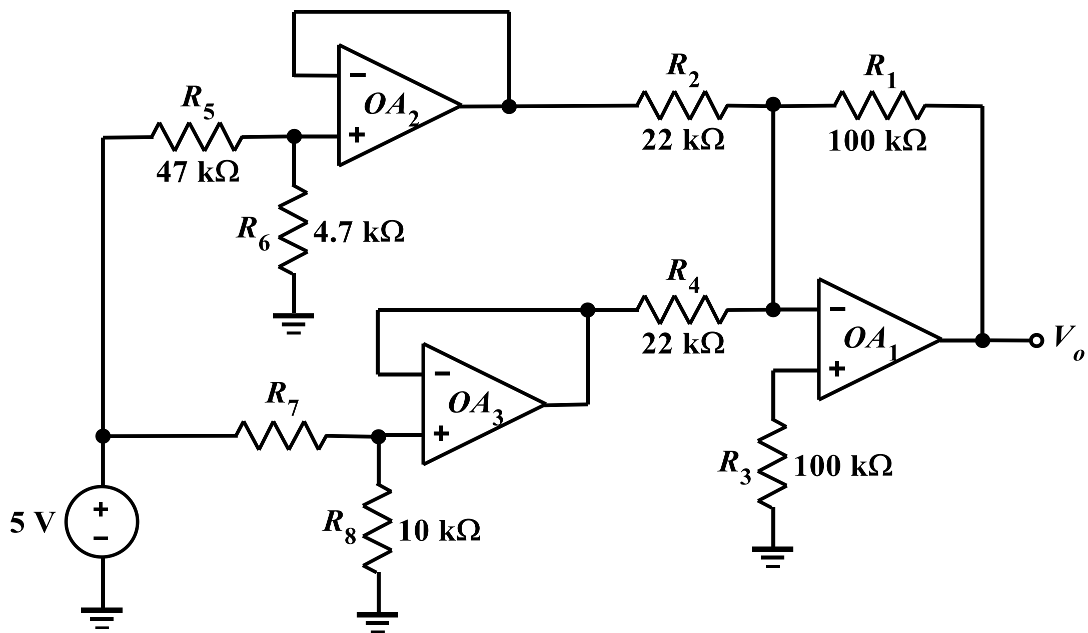

Figure 4

Table 4(a)

R_7 (Ω)	**Ideal V_o (V)**	**V_o from LTspice (V)**	**% Difference**
20 k			
40 k			
60 k			
80 k			
100 k			

Now, do the following:

1. Build the circuit in LTspice using the LM741 OpAmp model.
2. Perform an operating point simulation for each of the five values of R_7 and compare the results by completing Table 4(a).
3. Are V_+ and V_- always the same for each of the three OpAmps? Use your simulation data to fill in Table 4(b). Is it reasonable to assume zero volts difference between the two input terminals as we do with the ideal OpAmp model?

Table 4(b)

	***OA*1**			***OA*2**			***OA*3**		
$\mathbf{R_7}$ **(Ω)**	$\mathbf{V_+}$ **(V)**	$\mathbf{V_-}$ **(V)**	$\mathbf{\Delta V}$ **(V)**	$\mathbf{V_+}$ **(V)**	$\mathbf{V_-}$ **(V)**	$\mathbf{\Delta V}$ **(V)**	$\mathbf{V_+}$ **(V)**	$\mathbf{V_-}$ **(V)**	$\mathbf{\Delta V}$ **(V)**
20 k									
40 k									
60 k									
80 k									
100 k									

4. What is the purpose of operational amplifiers OA_2 and OA_3?

Please make sure that you include netlists for all circuits in your report.