EXPERIMENT IV

BASIC APPLICATIONS OF OPERATIONAL AMPLIFIERS

OBJECTIVE: In this experiment, you will analyze basic circuit configurations with operational amplifiers and resistors. The following concepts will be covered:

1. Controlling the transfer characteristics of circuits using different resistor configurations around a basic operational amplifier (op-amp);
2. Applying node voltage circuit analysis to op-amp circuits,
3. Using feedback techniques to deliver op-amp circuits with predictable performance,
4. Fundamental specifications and limitations of practical operational amplifiers,
5. Use of function generator to deliver, and oscilloscope to monitor AC signals.

EQUIPMENT & COMPONENT LIST:

DC Power Supply, Function/Arbitrary Waveform Generator, Digital Oscilloscope
Op-Amp (μA741), Resistors (2x100 Ω, 4 x 1kΩ).

PRELIMINARY WORK:

0. μA741 specifications and LTSPICE model installation

A. Go to Texas Instruments manufacturer's web site for μA741 operational amplifier (http://www.ti.com/product/UA741/). You may download 'μA741 General-Purpose Operational Amplifiers datasheet' under Technical Documents tab, and review the specifications. Here is the interpretation of a small subset of the specifications that may be helpful in completing the rest of your preliminary work:

Electrical Parameter	Meaning
Large Signal Differential Voltage Amplification	The open-loop (no feedback) gain of the op-amp at DC (low frequency operation). The typical gain for μA741 is listed as 200 V/mV (200,000). This figure is conservative at minimum supply voltage level, and can be larger for higher supply voltages. Note: Many op-amps have gains over 1,000,000. Based on 'test conditions' column, amplification depends on load R_L.
Maximum Peak Output Voltage Swing	The output voltage saturates short of power supply rails. The max output voltage also depends on the load current. With a smaller load (i.e. a big load resistor drawing little current) the output can go higher than with a large load (i.e. a small load resistor requiring more current). Most op-amps can swing the output to within a few volts of the power supply rails. Note: There are special op-amps called "Rail-to-Rail" op-amps that can swing the output to within 100mV of the supply rails. μA741 is not one of those.
Input Resistance	The resistance seen looking into the input pins. The μA741 has a typical input resistance of 2 MΩ. This value tends to scale up for input voltage much lower than supply voltage. Note: 2 MΩ is considered low. Many op-amps have input resistances over 1 GΩ.
Short Circuit Output Current	How much current the op-amp can source or sink from the output pin. Note: The output voltage could drop near zero volts when delivering the maximum current. Typically the op-amp can't deliver more than 25 mA (but always can deliver at least 10mA). See "Output Voltage Swing" to get an idea of how much voltage you can put across various sized resistors.
Supply Current	The current drawn from the power supply with no load on the op-amp. Note: There are low power op-amps available that run on less than 10uA, but μA741 is not one of those. Usually the faster the op-amp the more power it requires.

B. Spice models are available for many common electrical or electronic off-the-shelf commercial components in order to help integrate them into system simulation models. In this part of the pre-work, you will acquire a spice model for the popular μA741 operational amplifier component, and incorporate it into the LTspice library. Once the circuit model of a component is in your library, you can then instantiate it into your schematics just like any other components you have used so far (voltage sources, resistors, etc.)

i. Create a new folder in the LTSPICE subcircuit directory, which is typically generated at: C:\Program Files\LTC\LTspiceXVII\lib\sub on your computer after you install LTspice. You may call the new folder **EEE201** to distinguish your course libraries from the rest. You should end up with a folder: **C:\Program Files\LTC\LTspiceXVII\lib\sub\EEE201**

ii. Create a new folder with the same name (EEE201) in the LTSPICE symbol directory at: C:\Program Files\LTC\LTspiceXVII\lib\sym on your computer. You should end up with the folder: **C:\Program Files\LTC\LTspiceXVII\lib\sym\EEE201**

iii. In the Texas Instruments manufacturer's web site for UA741 operational amplifier (http://www.ti.com/product/UA741/), you may click on UA741 PSPICE Model under 'Tools & software' tab to download the spice model for the component. After you unzip the file, you should get a small spice model file. Copy this to **sub\EEE201** you created above.

iv. You may need to change security permissions for the LTspice folder and the subfolders under C:\Program Files\LTC in order to be able to modify directories and files. Rename the file as **UA741.lib**. LTSPICE sub circuit files need the .lib extension.

v. Now you need a symbol for the spice model of the 741 op amp you downloaded. You can create your own symbols from scratch, but it is easier in this case to just copy an existing symbol of a similar component and modify it. The symbol for the generic part opamp2 in the LTSPICE symbol directory will do nicely. On your computer it is probably located at:

C:\Program Files\LTC\LTspiceXVII\lib\sym\Opamps\opamp2.asy

Copy this file to the symbol directory you created in step 2 and rename it to UA741.asy. This folder should now contain:

C:\Program Files\LTC\LTspiceXVII\lib\sym\EEE201\UA741.asy

vi. Start up LTSPICE and select File->open. Change the "Files of type:" to Symbols (*.asy) and navigate to the sym\EEE201 directory you created in step 2. Open the file UA741.asy that should now be in this directory. You should now have an exploded drawing of a basic op amp on your screen.

vii. From the LTSPICE main menu bar, choose Edit->Attributes->Edit Attributes (or just press Ctrl A). This should bring up a Symbol Attribute Editor window. On the third line, change the Value attribute from opamp2 to UA741. On the eighth line (ModelFile attribute), you need to type in the name of the model file LTSPICE will use for the UA741 symbol. This is the file you downloaded, edited and saved as UA741.lib in step 4. Specify the directory (EEE201) and the filename (UA741.lib) by adding the text EEE201\UA741.lib to the ModelFile attribute line. Do not include a "\" before the EEE201. You can also change the Description attribute on line 7 to whatever you want, or leave it as is. From the LTSPICE main menu bar, choose File->Save. Close all LTSPICE windows.

viii. Your new component should now be in a new component subdirectory called EEE201. Start LTSPICE and select new schematic (if you did everything correctly, when you restart LTSPICE, it will spend few seconds updating its component libraries). Choose component and click the EEE201 directory. You should be able to select a UA741 op amp and place it on the schematic.

1. Open-loop operation (without feedback)

Given the following circuit with μA741 op-amp (component pins labeled in the figure) in open-loop configuration and the DC power supply to provide power to the circuit:

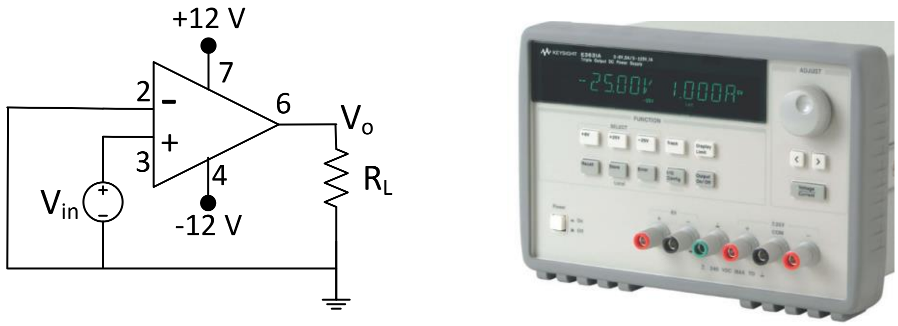

Figure 4.1 Op-amp in open-loop configuration (left) and DC supply front panel (right).

A. Sketch and roughly show how you would wire the DC supply front panel outputs (on the right) to the μA741 op-amp positive and negative supply, and as the V_{in} input*.* *Hint: Op-amp positive and negative supplies can be connected to* + and – 25 V, and V_{in} can be connected to 6 V (how about ground?)

B. Calculate $\boldsymbol{V_o}$ and $\boldsymbol{i_+}$ for the following parameters, and answer the questions in the table, given the specifications in the μA741 datasheet and information in Section 0 above. (**DONOT** assume ideal op-amp in this part of the pre work).

Table 4.1 Parameter calculations and comments for open-loop operation

V_{in} (V)	R_L (Ω)	V_O (V)	Input current, i_+ (mA)	Is output voltage saturated (Yes/No)?	Is output voltage limited by op-amp output current limit (Yes/No)?
0.00001	**2000**				
0.002	**3000**				
0.002	**200**				
2	**200**				

C. Describe in two or three sentences how you would measure the total power dissipation of the circuit of Figure 4.1 in the lab environment.

D. **(LTspice)** Run a DC op pt (DC operating point) analysis for the circuit in Figure 4.1 to obtain simulation results for $\boldsymbol{V_o}$ and $\boldsymbol{i_+}$ calculated in Table 4.1, and op-amp power dissipation for each scenario. Make sure all circuit nodes, resistor and supply instances are clearly labelled in your schematic in order to make it easier to follow the simulation output report. Include LTspice schematic and simulation result window snapshots to the prelab report. Generate another table similar to Table 4.1 to show your '**Parameter simulation results and comments for open-loop operation**'. This table should have an additional column for '**Op-amp power dissipation**'. Comment on possible reasons for any discrepancies between calculations and simulations.

2. Basic amplifier configurations with feedback

A. Assume ideal op-amp model in your calculations for the following circuits. Derive an expression that relates the output voltage to the input voltage for each of the circuits in Figure 4.2 to Figure 4.4. If $v_{in}(t)=(3\sin 1000\pi t)$ V, plot $v_o(t)$ versus $v_{in}(t)$ for each:

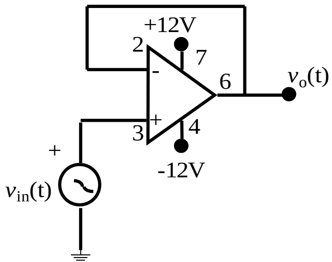

Figure 4.2 Voltage follower (buffer)

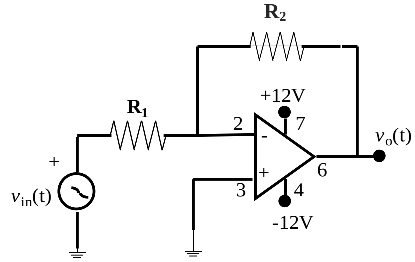

Figure 4.3 Inverting Amplifier

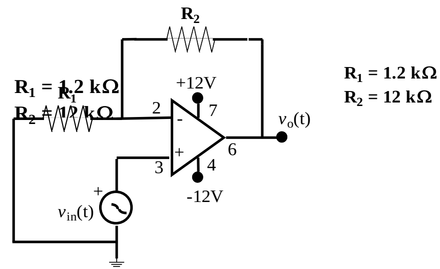

Figure 4.4 Non-inverting Amplifier

B. $v_{in}(t)$= 1 V (DC) for the voltage follower in Figure 4.2, and the circuit drives a load resistance of R_L = 12 kΩ (tied between $v_o(t)$ and ground).

i. Does the transfer function (v_o in terms of v_{in}) you derived in part 2(A) above change?
ii. What is the output voltage in this configuration?
iii. Calculate the power dissipated at load resistor, R_L.
iv. Do you expect the power dissipated at R_L to be equal to the sum of power delivered by v_{in}, +12 V and -12 V sources in a practical circuit? Explain.
v. What is the energy dissipated at load resistor, R_L at the end of 1 second?

3. Summer and Subtractor (Difference) Circuits

A. Assume ideal op-amp model in your calculations for the following circuits. Derive an expression that relates $v_o(t)$ to $v_a(t)$ and $v_b(t)$ for each of the circuits in Figure 4.5. If $v_a(t)$=(2sin1000πt) V, and $v_b(t)$=2 V, plot $v_o(t)$ for each:

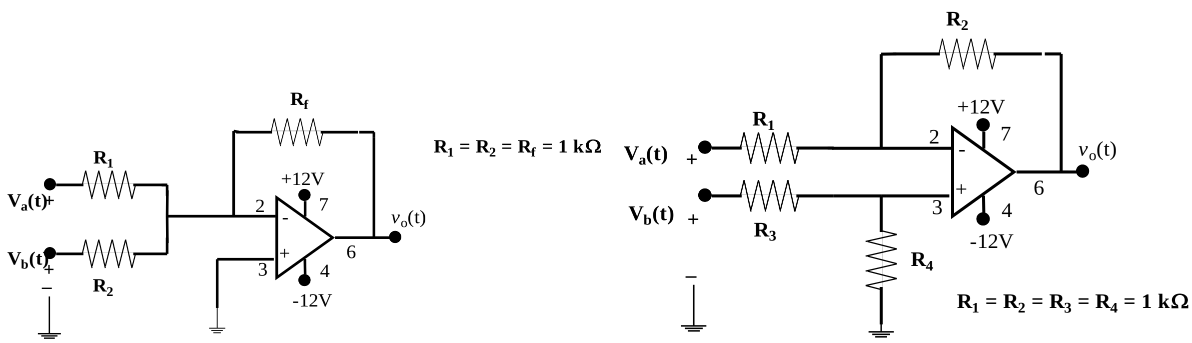

Figure 4.5 Summer (left) and difference (right) circuits

B. **(LTspice)** Construct the circuits in Figure 4.5 using two LTspice schematics, and analyze using a transient simulation that lasts as long as **two periods (T)** of the input $v_a(t)$. Make sure all circuit nodes, resistor and supply instances are clearly labelled in your schematic in order to make it easier to follow the simulation output report. Include LTspice schematic and simulation result window snapshots to the prelab report. Are there any discrepancies between your calculations from part 3(A) and simulation results? If yes, comment on possible reasons.

<u>EXPERIMENTAL WORK</u>:

Note: There are four demonstrations in this laboratory. Once you are done with any of the demos, please have one of the lab instructors take a look at your setup and measurement in order to get credit for this portion of the lab. You may be asked questions about your demonstration.

Hint: **Do not forget to set the function generator to the "High Z" mode, and use DC supply compliance limits to avoid excessive currents during the experiment.**

1. **Open-loop operation (without feedback)**

   Setup the circuit of Figure 4.1. Try to get as close as possible to the $\boldsymbol{v_{in}}$ and $\boldsymbol{R_L}$ values in the **last three rows** of Table 4.1 (skip V_{in}=10μV case in the first row since this is too low for the equipment), and make measurements for output voltage $\boldsymbol{v_o}$, input current $\boldsymbol{i_+}$, and total power dissipation for each case. **Make a table of measurements** using the **same number of columns** you used for simulations in preliminary work part 1(D). **Explain** any discrepancies between simulations and measurements.

   Demo 1: Demonstrate your circuit setup, voltage, current and power measurements for V_{in} = 0.2 mV and R_L = 200 Ω to the lab instructor.

2. **Voltage follower (buffer) characterization**

   Build the circuit of Figure 4.2.

   A. Observe $\boldsymbol{v_{in}(t)}$ and $\boldsymbol{v_o(t)}$ on the oscilloscope using $v_{in}(t)=(3\sin 1000\pi t)$ V. Increase the amplitude of the sinusoidal input $\boldsymbol{v_{in}(t)}$ until you observe saturation effects at the output on the oscilloscope. Note down the maximum amplitude of $\boldsymbol{v_{in}}$ before output saturates. Comment on this value with respect to the op-amp datasheet. Which is the relevant specification?

   B. Obtain and plot $\boldsymbol{v_o}$ versus $\boldsymbol{v_{in}}$ using the XY mode on the oscilloscope.

   Demo 2: Demonstrate your circuit setup, saturation, and XY mode measurements to the lab instructor.

   C. Change the configuration of the circuit for DC characterization with values provided in preliminary work section 2(B). Complete the measurements that correspond to preliminary work 2(B) parts (ii), (iii), and (v). Make a table that lists the calculated parameter, measured parameter, and % difference in different columns. Three different parameters should be listed in different rows.

   Demo 3: Demonstrate your DC circuit setup and parameter measurements to the lab instructor.

3. **Difference circuit**

   Setup the difference circuit of Figure 4.5 (right). Using $\boldsymbol{v_a(t)}=(2\sin 1000\pi t)$ V, and $\boldsymbol{v_b(t)}$=2 V, monitor $\boldsymbol{v_a(t)}$ and $\boldsymbol{v_o(t)}$ on the oscilloscope. Sketch your observed signals. How does your measurement compare to the calculations and simulations from preliminary work Section 3? What happens as you slowly reduce $\boldsymbol{v_b(t)}$?

   Demo 4: Demonstrate your circuit setup and observations to the lab instructor.

4. **Summer circuit (bonus)**

   Repeat Section 3 above with the summer circuit, if time allows. If you have finished the lab experiments early, you may get bonus points by completing this part.