CMPE 306

Lab VI: Op Amp Circuits, Part 1

Created by: EFC LaBerge based on 2008 lab by Dr. L. Yan and Ryan Helinski

March 2014

1. Purpose and Introduction

The purpose of this lab is to familiarize students with circuits containing operational amplifiers, known in class as "op amps".

By the end of this labs session, students will be able to perform the following tasks:

1. Simulate an op amp circuit in LTSPICE using the .lib opamp.sub model
2. Properly power an op amp chip in a breadboard circuit.
3. Measure and illustrate the limits of op amp voltage gain.
4. Combine op amps in a multi-stage amplifier design.

2. Pre-Lab

The prelab work for this experiment consists of simulating the circuits of Figure 3 and Figure 5 in LTSPICE or equivalent software. As with earlier labs in this series, LTSPICE is used as the standard.

To place an ideal op amp and use it in your LTSPICE simulation, complete the following procedure.

1) Open LTSPICE and start a new circuit.
2) From the Component list, select the library [Opamps]. Within the library, scroll down past the dozens of op amp devices offered by Linear Technologies to the entry "opamp" for an ideal single pole operational amplifier. Select the "opamp" device and place it on your circuit diagram.
3) Whenever you use this model, you must include a SPICE directive for how to treat the model. Enter a new SPICE directive as ".lib opamp.sub" and place the directive on your circuit diagram. If you don't do this, your op amp simulation will not work.
4) Place the other elements of Figure 3, using $R_1 = 10\text{k}\Omega,\ R_f = 20\text{k}\Omega$. You do not need to worry about the pin numbers (labels 8, 9, 10) show in Figure 3. Label and place the node Vo as shown.
5) To illustrate the gain characteristic, we'll sweep the voltage input. This, too, is a new SPICE directive. Add the SPICE directive ".dc V1 1 5 0.1" to your circuit. This will sweep the value of voltage source V1 from 1 volt to 5 volts in steps of 0.1 volts. Your LTSPICE circuit should look like Figure 1.

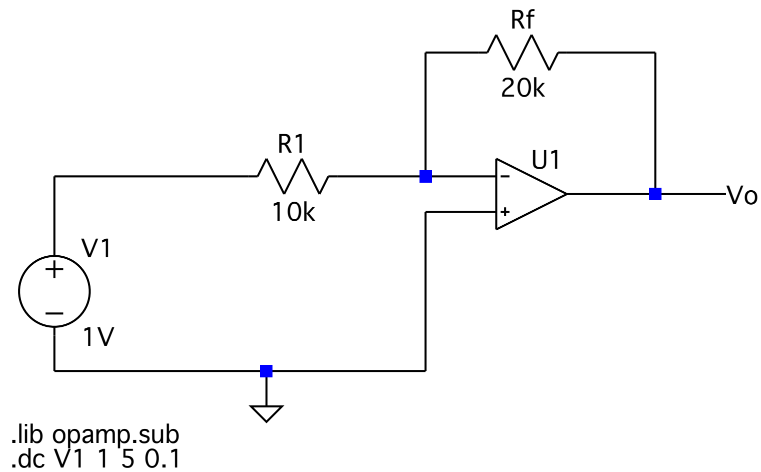

Figure 1 Simple Op Amp Simulation in LTSPICE

6) Save your circuit and run the simulation. Because this is not a DC operating point simulation, the results are best displayed using the graph capabilities. Click on the graph icon if the window doesn't automatically open. Click on the window and add traces V1 and Vo. You should see two linear traces. Note that the trace corresponding to V1 goes smoothly from 1V to 5V volts, while the trace for Vo goes smoothly from -2V to -10V. This demonstrates that the gain of this ideal op amp configuration is $G = -\frac{R_f}{R_1} = -\frac{20\text{k}\Omega}{10\text{k}\Omega} = -2$. The output curve should look like

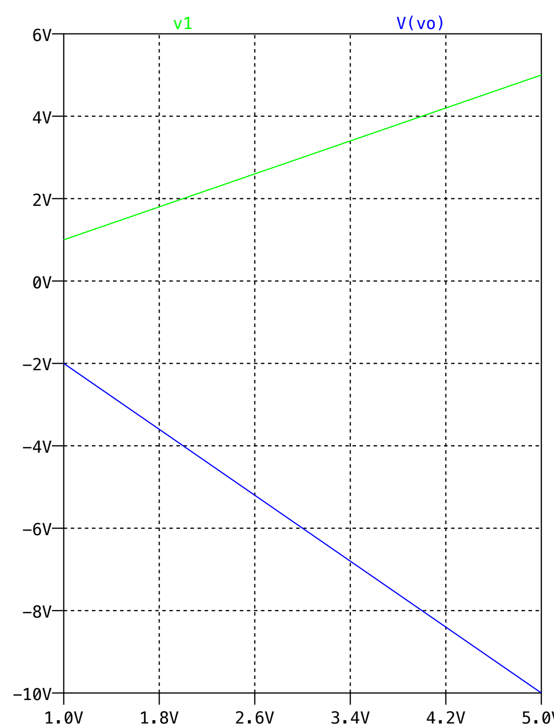

Figure 2 Output Traces from Step 6

7) Construct and simulate the circuit for Figure 5.

8) The prelab verification consists of the circuits and figures corresponding to Figure 3 and Figure 5. For the circuit of Figure 3 you may choose any of the resistor combinations required in the lab. You need not simulate every combination in the lab, only one configurations for Figure 3. and one configuration for Figure 5

3. Equipment

This lab exercise uses the following equipment:

1) Tektronix AFG310 Arbitrary Function Generator
2) Tektronix 2012 Digital Storage Oscilloscope
3) BNC-to-BNC cable
4) Two BNC-to-alligator cables
5) Agilent programmable voltage source.
6) $2\times 20\mathrm{k}\Omega, 10\mathrm{k}\Omega, 51\mathrm{k}\Omega$.
7) 14 pin Opamp IC from your lab kit.

4. .Procedure

4.1. Single Op Amp and Voltage Gain

For this procedure, you will construct the circuit shown in Figure 3, substituting different resistor combinations for R_1 and R_f and evaluating the resultant voltage gain, $G = \frac{V_o}{V_f}$.

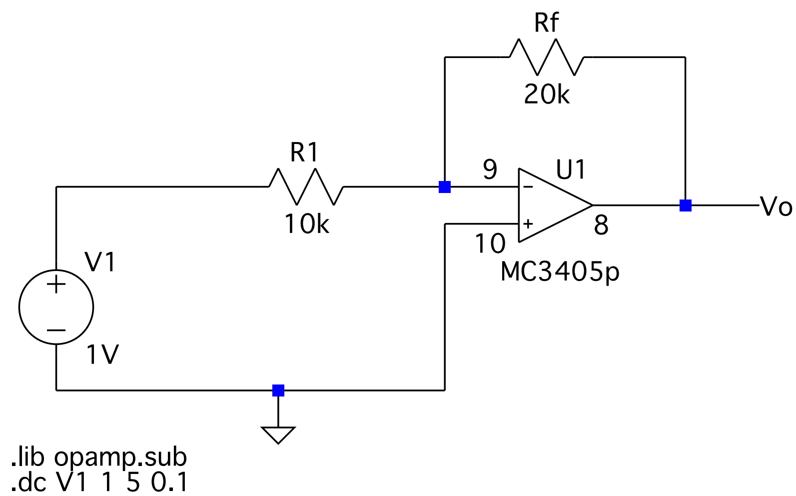

Figure 3 Single Op Amp Circuit for Voltage Gain Measurement

To construct this circuit, consider the MC3405p 14-pin op amp data sheet posted on Blackboard and associated with this lab. The data sheet includes a configuration diagram of the dual in-line package (DIP) shown in Figure 4. The "top" of the chip is identified by a small notch, as shown in Figure 4. This lab uses Op Amp 1 (pins 12, 13, and 14, numbered counterclockwise around the chip starting in the upper left), and Op Amp 2 (pins 7, 8, and 9).

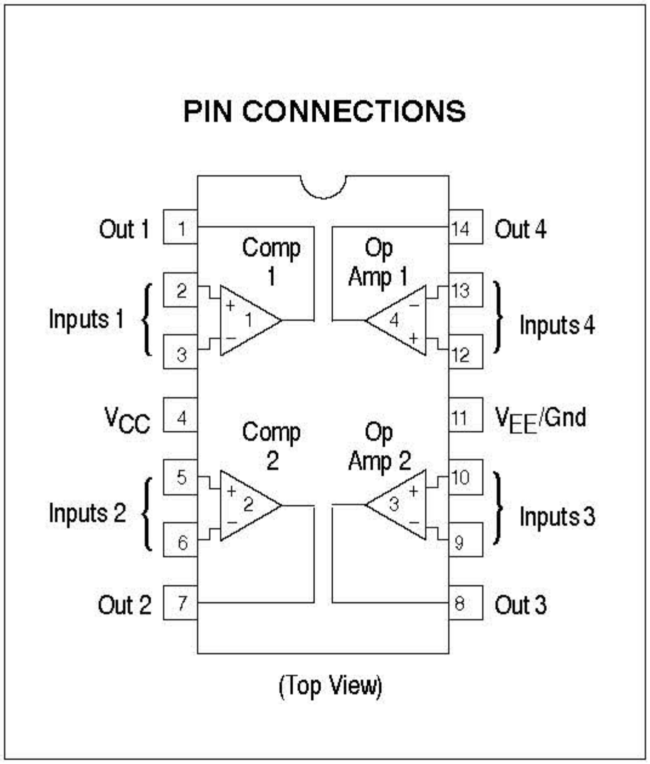

Figure 4 Connections for 14-pin Op Amp Chip

Use the following procedure to construct the circuit of Figure 3.

1) Set the Agilent voltage source to provide $+12\text{V}$ and -12V from the positive and negative power supplies. Turn the Agilent supply output off and connect the ground (black) lead to the ground of your circuit, the negative lead (red) to the V_{EE} pin 11, and V_{CC} to pin 4.
2) Connect Op Amp 2 (pins 8, 9, 10) as shown in Figure 3. Use $R_1 = R_f = 20\text{k}\Omega$. Check your circuit thoroughly before energizing the Agilent voltage source.
3) Connect the AFG as V_1, that is, to open end of R_1. Adjust the value of the AFG output to +2, +4, +10, -2, -4, -10 volts. Use the oscilloscope to measure the corresponding output voltages at pin 10. For best results, set the oscilloscope to DC Coupling. In the data section of your lab report, plot the input voltage as the independent variable and the output voltage as the dependent variable.

4) Using the AFG, create a 5 V (peak to peak) 500Hz square wave input to the circuit and verify that $V_o = -V_1$. Use the scope to capture and print the $V_o(t)$ and $V_1(t)$ simultaneously as PRINTOUT1.
5) Change the $\frac{R_f}{R_1}$ ratio to $\frac{20\text{k}\Omega}{10\text{k}\Omega}$, $\frac{20\text{k}\Omega}{51\text{k}\Omega}$, and $\frac{51\text{k}\Omega}{20\text{k}\Omega}$. In each case verify $V_o(t) = -\frac{R_f}{R_1}V_{in}(t)$ and print out the Vo(t) and Vi(t) simultaneously as PRINTOUT, PRINTOUT3, and PRINTOUT4.
6) Use a gain of –5 (what resistors should you use?) and square wave input (500Hz). Increase the amplitude in increments of 0.5 V from 0.5 V to 3V. Observe both the input and the output on the oscilloscope. Record the positive and negative input voltages and the corresponding output voltages until the amplifier saturates[1]. Record the saturated values of the output. Plot the results in lab report. Verify the gain in the linear region.
7) Show the raw data and print-out (PRINTOUT1 through PRINTOUT4) to lab instructor.
8) In your lab report, discuss the similarities or differences between the real op amp device and the simulated device you used in the prelab.

4.2. Cascaded Op Amps

1) Extend the circuit from Figure 3 as shown in Figure 5, using the other op amp circuit (pins 12, 13, 14) in the MC3405p package. Because the Agilent supply powers the entire chip, you do not need to provide any additional power connections to the Agilent supply.

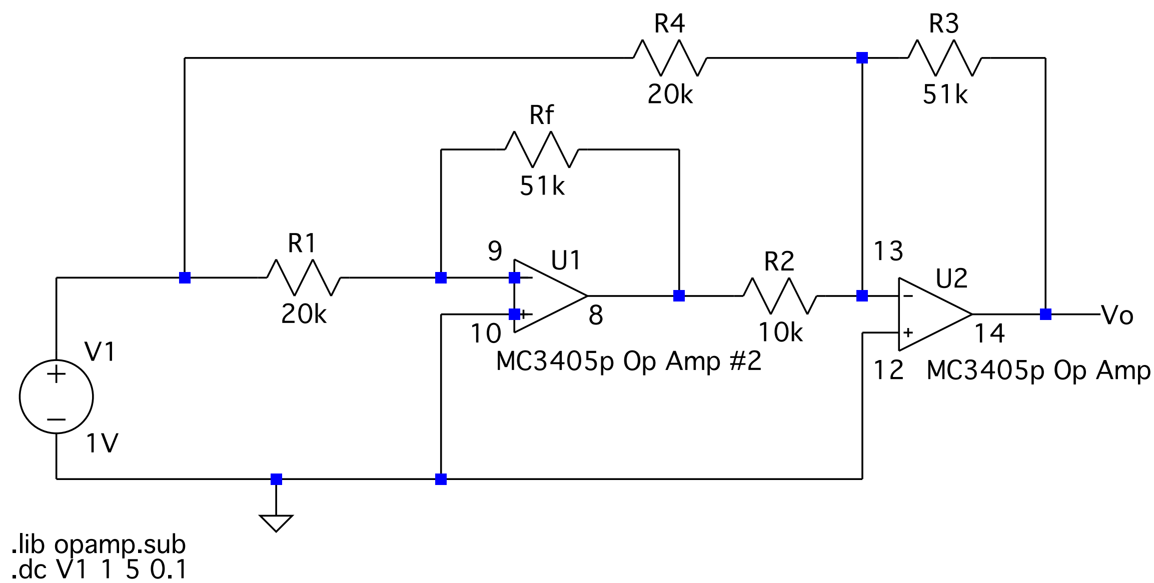

Figure 5 Cascaded Op Amp Circuit

[1] We say that a linear amplifier device *saturates* when the output voltage no longer increases as the input voltage is increased.

2) In the computation portion of your lab report, derive an analytical expression for the overall voltage gain, $G = \frac{V_o}{V_1}$ of this cascaded amplifier configuration. Take your time and follow the op amp rules. Lab partners may assist in this effort, regardless of who is writing the report.
3) Use a 0.3 V (amplitude, 0.6 volt peak-to-peak) 500 Hz square wave input to the circuit. Capture and print out the $V_o(t)$ and $V_1(t)$ PRINTOUT5
4) Increase the input level V_1 from the AFG at an increment of 0.3 V until it reaches 1.8 V (amplitude) and record the outputs of every increment. Plot the results. Compare the slope of the line to the gain value computed in step 2.

4.3. Preparation for Next Lab

Next week, your Lab VII will make use of resistors and capacitors to demonstrate the time-varying response to a changing input voltage. Please look at the Lab VII writeup and use LTSPICE to simulate the circuit. Dr. LaBerge will post some guidance about using LTSPICE with capacitors and time-varying voltage sources. Other instructional videos are readily available on the web. The pre-lab will consist of simulating the various circuits required in Lab VII in LTSPICE or its equivalent.

For the lab report for this week, please include all of the plots that you were asked to save, and all of the values you were asked to record.

5. Tear Down and Clean Up

1. Turn off the power supply, AFG, and oscilloscope and set the multimeter to the OFF position. Return the multimeter to your TA for storage.
2. Save your images or data to your memory stick. Then close the program and sign off of the computer.
3. Put your resistors and op amp chip back in your lab kit. Return your lab kit to the TA for storage.
4. Return the BNC cables and BNC-to-alligator cables and hang them neatly in their proper rack.
5. Police your lab area: leave it neat and clean.
6. If you're using your own laptop, there's nothing else to clean up.
7. If you're using the lab computer, save whatever work you want to your USB drive. Close LTSPICE if necessary. Eject your drive.