UNIVERSITY OF CALIFORNIA AT BERKELEY
College of Engineering
Department of Electrical Engineering and Computer Sciences
EE105 Lab Experiments

Lab 2: Non-Ideal Op-Amps

Contents

1. Introduction 1
2. Pre-Lab 2
2.1. DC current consumption 2
2.2. DC Open Loop Transfer Characteristic 4
2.3. Nulling the Offset Voltage 5
2.4. Slew Rate in Unity Gain Configuration 6
2.5. Gain and Bandwidth in Unity Gain Configuration 6
2.6. Gain and Bandwidth in Non-Inverting Amplifier Configuration 8
3. Lab 9
3.1. DC current consumption 9
3.2. DC Open Loop Transfer Characteristic 9
3.3. Nulling the Offset Voltage 10
3.4. Slew Rate in Unity Gain Configuration 10
3.5. Gain and Bandwidth in Unity Gain Configuration 11
3.6. Gain and Bandwidth in Non-Inverting Amplifier Configuration 11

1. Introduction

In this lab, you will characterize a LM741C operational amplifier. In the pre-lab we will simulate the following non-ideal properties of op-amps: input offset voltage, slew rate, and finite gain and bandwidth. In the lab we will measure these parameters. The datasheet of the operational amplifier is in the zip file you have downloaded from bCourses.

Make sure to submit the pre-lab worksheet on Gradescope before the beginning of your lab section.

2. Pre-Lab

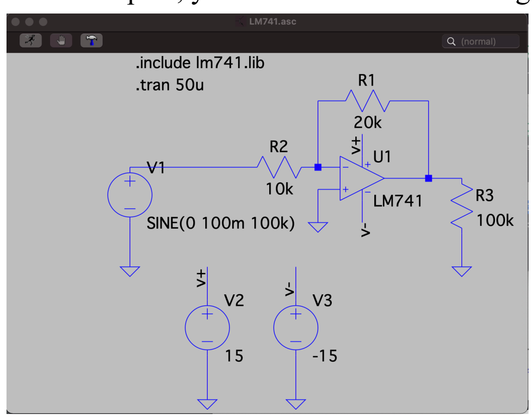
For the pre-lab, you will be doing LTSpice simulations of the measurements that you will be doing in your lab section to help you prepare for the lab. In the lab folder (which includes this file), you should have the necessary components to perform the pre-lab.

To perform the pre-lab you will need to download and install LTSpice (available for MAC, PC and Linux). You can find the link to do that here:
https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

2.1. Introduction to LTSpice

LTSpice is a SPICE ("Simulation Program with Integrated Circuit Emphasis") simulator developed by linear technologies as a teaching and open-source simulation tool. The original SPICE simulator was developed at Berkeley in the 1960s and 70s, and has been widely modified, extended, and built upon, and is the standard for circuit simulation today. At the entrance to Cory Hall from Hearst Avenue, opposite to the wall of pictures of current professors there is a plaque commemorating the creators of SPICE, a program we still use today! How cool!

If you open the LM741.asc file in LTSpice, you should see the following window.



There are a few things to point out in this diagram.

".include lm741.lib" ← this tells the simulator to include the special LM741 library file (included in the zip file you downloaded). Every opamp is different, so you need a special model for the LM741! You should see a file called "lm741.lib" in the zip folder

".tran 50u" ← This is the "simulation command", it tells the simulator to run a transient simulation for 50us. A transient simulation is a a time domain simulation to plot voltage/current vs. time. This will look exactly like what your oscilloscope outputs when you probe it in lab.

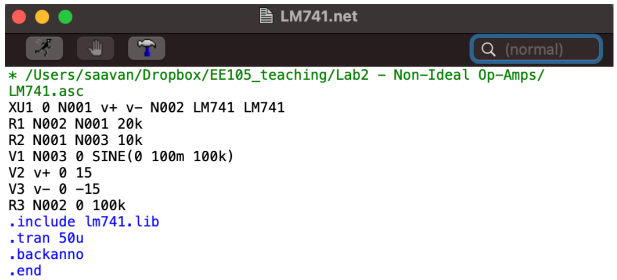
V1 SINE(0 100m 100k) ← This is a sinusoidal voltage source with DC value of 0V, 100mV amplitude, and 100khz frequency. You can modify this voltage source to be anything else (a DC value, a pulse source, etc.) by right clicking and changing the voltage values.

V2 ← This supplies a +15V voltage to the LM741 in simulation

V3 ← This supplies the -15V voltage to the LM741 in simulation

 ← this is the “run simulation” button, and is how you can get started analyzing your data

If you right click open window and go to View → SPICE Netlist, a window that looks something like this should pop up:



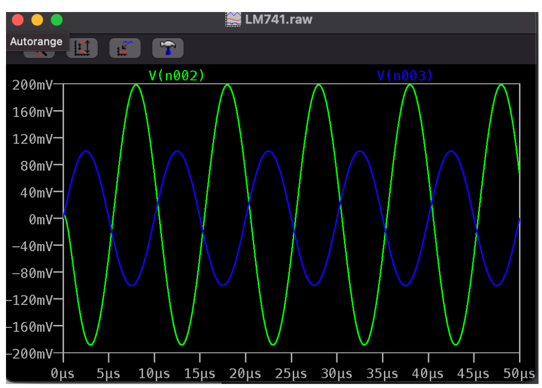

```
* /Users/saavan/Dropbox/EE105_teaching/Lab2 - Non-Ideal Op-Amps/
LM741.asc
XU1 0 N001 v+ v- N002 LM741 LM741
R1 N002 N001 20k
R2 N001 N003 10k
V1 N003 0 SINE(0 100m 100k)
V2 v+ 0 15
V3 v- 0 -15
R3 N002 0 100k
.include lm741.lib
.tran 50u
.backanno
.end
```

This is the compiled form of the schematic you just drew, and it is what is passed into the simulator. The SPICE netlist is a human readable format, and with some effort you can understand it too.

2.1.1. Running your first simulation

Click the run icon (the  in the top right). A blank black screen should pop up. This is your plotting window for the simulation.

From here, click on the wires in the original schematic that you want to view. Start by clicking the wire coming out of the voltage source, and the wire coming out of the op-amp into the resistor load. Your plotting window should look like below:



To modify values in the circuit, go back to the original schematic, right click and change values. When you re-run the circuit (click the running man again), plotting window should change.

To run transient simulations, this is the syntax for the command (the “.tran 500u” in the top left corner)

```
.TRAN <Tstop> [modifiers]
```

Where Tstop is how long you want to run the simulation for

Q: Now modify the feedback resistor to be achieve a gain of 10 (the output should be 10x the input). What feedback resistor value gave that? What kind of circuit are you simulating? Attach the plot to your pre-lab report.

2.1.2. Building your own simulations

Now, how do you modify the simulations to run something else?

Here are some tips and tricks:

- Right click any of the components, and you will be able to change their value, and what they do.
- If you right click the schematic anywhere else, the “draft” submenu will allow you to place components, wires, and directives. Use this sub-menu to place components and modify them
- The op-amp we have placed here is the LM741, you need to make sure to copy and paste this op-amp everywhere so that you get the correct simulation parameters!
- We have included a shortcuts flyer to help you get started on shortcuts and spice commands.
- The LTSpice help menu is very good! If you get stuck, type in the error/command in the help menu and it will pop a message up that will give you the needed details.

2.2. DC current consumption

For the configuration in Figure 1 with Vin=0 simulate the DC current consumption of the opamp.

To do this, you will want to run a DC operating point simulation “.op”. Right click the “.tran 50us” directive, and change it to “.op”.

Q: Is the DC current consumption in the range defined in the datasheet?

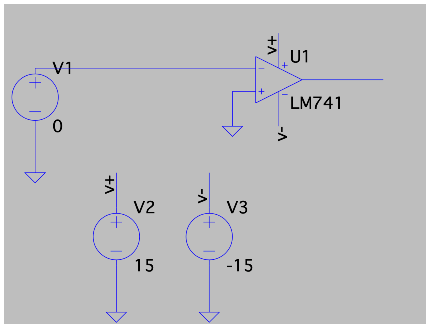


Figure 1: Open-loop amplifier

2.3. DC Open Loop Transfer Characteristic

For this portion of the pre-lab you will need to perform a “DC source sweep analysis" with the “.dc” command. You can do this by modifying the command directive (the “.op” text in the top left corner of the schematic) from part 2.2 to read “.dc V1 -5m 5m 10u”. This will sweep the input DC value of the source V1 to go from -5mV to 5mV in increments of 10uV.

The dc command syntax is as follows:

```
.dc <srcnam> <Vstart> <Vstop> <Vincr>
```

Q: Simulate the circuit in Figure 1, sweeping Vin from −5mV to +5mV. Plot Vout as a function of Vin and determine the open loop gain of the op-amp and the input offset voltage.

2.4. Nulling the Offset Voltage

Q: Null the offset voltage by adding a DC component Vshift to the input voltage source to cancel the offset. Is the offset voltage within the specification of the datasheet? While this works in simulation, in the lab you will use the offset null terminal pins on the op-amp to null the offset voltage. Check the datasheet for the circuit used to null the offset voltage and draw the circuit connected to the pinout diagram for the LM741 on your pre-lab worksheet. Why do we use a potentiometer to null the offset voltage in the lab rather than the Vshift voltage?

2.5. Slew Rate in Unity Gain Configuration

Simulate the circuit in Figure 2 to look at the transient response of Vout to the square wave voltage source Vin. Modify Vin to be a "pulse" source by right clicking and changing the "style" from "sine" to "pulse".

To perform a slew rate calculation, you will need to change the command directive to ".tran" again.

Set Vin to have the following characteristics: amplitude of ±1V, 10µs delay, 100ns rise and fall times, pulsewidth (Ton) of 100µs, and period of 200µs. Make sure to simulate at least one period of the square wave.

Q: Plot Vout and Vin versus time and determine the slew rate from the plot.
Is the simulated slew rate reasonable based on the datasheet?
Is the slew rate different for rising and falling voltages?

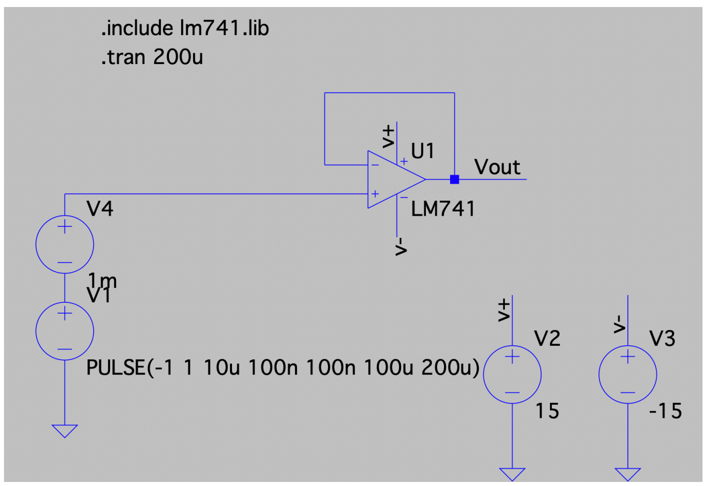


Figure 2: Unity-gain amplifier with square wave input

2.6. Gain and Bandwidth in Unity Gain Configuration

For this part, you will be performing AC simulations of the opamp. An AC simulation will generate something that looks like a bode plot, and give you the magnitude and phase change of signals with respect to their frequency. To perform AC simulations you will need to change the command directive to ".ac". The syntax for performing AC simulations is below:

```
.ac <oct, dec, lin> <Nsteps> <StartFreq> <EndFreq>
```

The first piece after "ac" is the number of steps for octave/decade/linear spacing. Nsteps is the number of steps, Start Frequency is the first frequency to start with, and Stop Frequency is the last frequency to end the simulation.

Q: Simulate the circuit in Figure 3, performing an AC analysis from 1Hz to 100MHz. Use logarithmic scale with enough points per decade to get a smooth curve. Plot the magnitude response db(Vout/Vin|) in log scale, and find the gain and 3dB bandwidth of the amplifier.

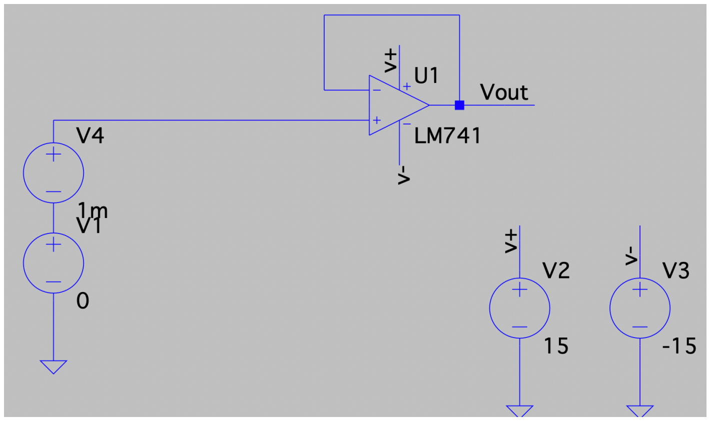


Figure 3: Unity-gain amplifier with sinusoidal input

AC analysis linearizes the circuit around the DC operating point. Non-linear behaviors such as slewing cannot be observed in this simulation. To capture it, we should run a transient analysis. For slewing we need to satisfy:

$$\frac{dV_{out}}{dt} = 2\pi A \cdot f \geq Slew\ Rate$$

Q: Based on your slew rate simulation, for 100KHz input at what amplitude the amplifier will start slewing?

Plot the input and the output for this amplitude, and for twice this amplitude, at the same plot. Use transient simulation with 4 time periods.

2.7. Gain and Bandwidth in Non-Inverting Amplifier Configuration

Simulate the circuit in Figure 4 with R=100kΩ and R=10kΩ, performing AC analyses from 1Hz to 1GHz. Plot the magnitude response of both circuits in log-log scale on the same plot as Problem 2.6, and find the gain and 3dB bandwidth of both amplifiers.

Q: At approximately what frequency and gain do the three curves intersect on the plot? What does this mean? If you are unsure, compute the product of the gain and the 3dB frequency for the three amplifiers.

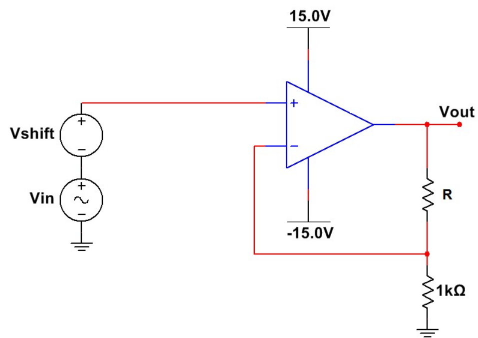


Figure 4: Non-inverting amplifier with sinusoidal input

Q: With R=10kΩ, for 10KHz input at what amplitude the amplifier will start slewing? *Hint:* don't calculate it based on slew rate definition, think about the difference from the unity gain case.

Plot the input and the output for this amplitude, and for twice this amplitude, at the same plot. Use transient simulation with 4 time periods.

3. Lab

3.1. DC current consumption

For the configuration in Figure 5 with Vin=0 measure the DC current consumption of the opamp. Is it in the range defined in the datasheet?
Tip: It is useful to measure the current of the opamp when debugging, to make sure that the opamp is properly connected.

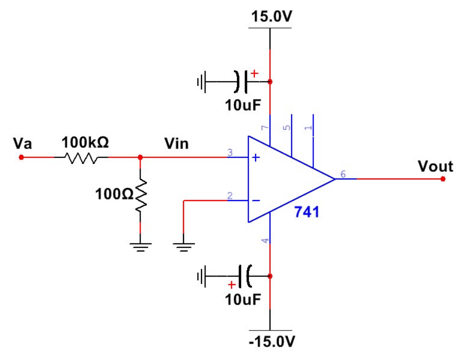


Figure 5: Open-loop amplifier

3.2. DC Open Loop Transfer Characteristic

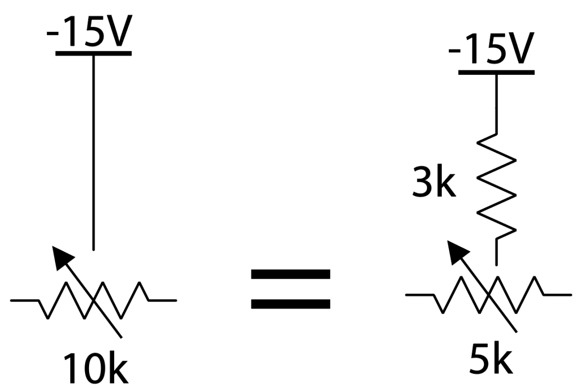
The op-amp gain is very high, so the output voltage can swing from −rail to +rail when sweeping the input voltage by 1mV, the smallest voltage step on the parameter analyzers. Thus the direct gain measurement of the op-amp using the parameter analyzers is not feasible. We first must attenuate the input voltage before feeding it into the input of the op-amp. Before wiring up the circuit in Figure 5, measure the resistance of the two resistors in the voltage divider circuit using the multimeter and write the resistance values in your lab worksheet.
To perform the DC sweep with the parameter analyzer, use the "Generic 2-port model" tab in SpaZilla (instead of the default "2-terminal device"). Connect Va to the Input SMU, Vout to Output SMU, and the ground to Ground SMU.
You will want to sweep Va over a large voltage range to start out with and then sweep Va over a smaller range so you can get enough points to be able to accurately calculate the gain. Determine the gain and the input offset voltage Voffset from the plot of Vout versus Va and write them in your lab worksheet. Remember: the parameter analyzer is plotting Vout versus Va. Make sure to back-calculate Vin from Va before determining the gain and input offset voltage!

Q: What is the open loop gain? Does it match up with what you saw in simulation? How about the datasheet?

3.3. Nulling the Offset Voltage

Connect the op-amp in the configuration shown in Figure 6. Make sure to connect the middle lead ("the wiper") of the 10kΩ potentiometer to the −15V power supply rail. Ground Vin, and measure the output voltage Vout of the op-amp.

Note: We don't have 10kΩ potentiometer in lab, so we will be modifying it to be a 3.6kΩ resistor and a 5kΩ potentiometer. Circuit Below:



Q: Adjust the potentiometer until the measured output voltage is nulled as best as you can. Measure the resistances between the wiper and the two outside leads of the potentiometer and write them in your lab worksheet. Does this make sense based on the datasheet?

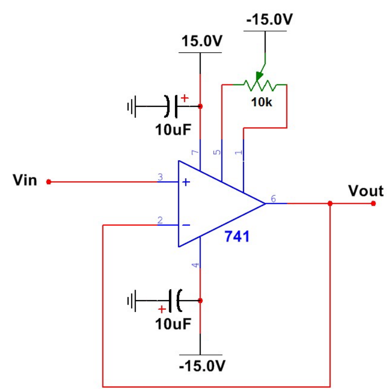


Figure 6: Unity-gain amplifier

3.4. Slew Rate in Unity Gain Configuration

Keep the op-amp in the configuration as shown in Figure 6. Apply a ±10V amplitude square wave signal to the input of the op-amp by connecting Vin to the function generator. Monitor Vin and Vout on the oscilloscope on separate channels. Make sure that the input impedances of the channels of the oscilloscope are set to 1MΩ so that the oscilloscope doesn't load your circuit.

Q: Determine the slew rate for both the rising and falling edge from the oscilloscope trace. Is it reasonable based on the datasheet? How about the simulation? Write the slew rates in your lab worksheet, and attach the oscilloscope trace(s) to your lab worksheet.

3.5. Gain and Bandwidth in Unity Gain Configuration

With your op-amp in the same configuration as shown in Figure 6, change the function generator to generate a sine wave signal. We would like to make sure that the amplifier is not slewing. For slewing we need to satisfy:

$$\frac{dV_{out}}{dt} = 2\pi A \cdot f \geq Slew\ Rate$$

Based on your slew rate measurements, for 100KHz input at what amplitude the amplifier will start slewing? Now double that amplitude, you should see a slewing output of the amplifier. Save the oscilloscope trace of the slewing output voltage.

Q: Now that you know what a slewing output voltage looks like, reduce the amplitude of the sine wave signal to ensure that it will not produce a slewing output. Measure the magnitude response of the voltage gain of the circuit from 100Hz to 10MHz, making sure to plot at least 3 points per frequency decade. If you can measure out to f3dB, write it in your lab worksheet. Also, write the measured gain A0 in your worksheet. Does the gain match up with what you expected from simulation?

3.6. Gain and Bandwidth in Non-Inverting Amplifier Configuration

Now connect your op-amp in the configuration shown in Figure 7. For both R=10kΩ and R=100kΩ, measure the magnitude response of the voltage gain of the circuit from 100Hz to 10MHz, making sure to plot at least 3 points per frequency decade. Make sure that the output is not slewing!!

Q: If you can measure out to f3dB, write it in your lab worksheet. Also, write the measured gain A0 in your worksheet. Plot the magnitude response of the voltage gain in log-log scale for the two non-inverting amplifier circuits and the circuit from Problem 3.5 on the same plot and attach the plot to your lab worksheet.

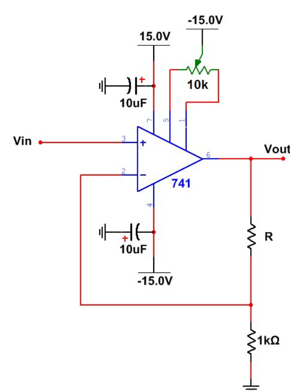


Figure 7: Non-inverting amplifier