

Laboratory 2: Characterization of the 741 Op Amp

Preliminary Exercises

This lab will characterize an actual 741 operational amplifier with emphasis on its non-ideal properties, such as finite gain and bandwidth. To prepare you for this lab, the following preliminary exercises use an assumed non-ideal op amp model that includes finite gain, slew rate, bandwidth, offset voltage, and bias current. The model to be used is actually the simulation (SPICE) model given by National Semiconductor for the 741 op amp. It appears on-line in the file 'lm741.mod' in the website for this class. Since this is the official model given by National Semiconductor in association with their 741 op amps, the parameters in the model should closely approximate the performance of your actual laboratory op amps. (Don't be surprised, however, if there are deviations!) Use this model when doing both *hand calculations* and *SPICE simulation*.

Since the measurement of input bias current is difficult, the circuits used in our measurements are (on purpose) relatively insensitive to their effects. We will show this in the discussions below, which deal to a large extent with simulation of circuits involving the described 741 op amp model in various circuits using SPICE.

1. Open Loop Characteristics

- (a) Generate the netlist for the circuit of Fig. PL2.1(a). Then use SPICE to determine the voltage transfer characteristic (VTC) V_o versus V_i for $V_i = (-10\text{mV to } 10\text{mV in } 0.1\text{mV steps})$. Again, for the 741 op amp, use the SPICE model given on-line on the course website in the file 'lm741.mod', and bias it with $\pm 18\text{V}$ supplies.

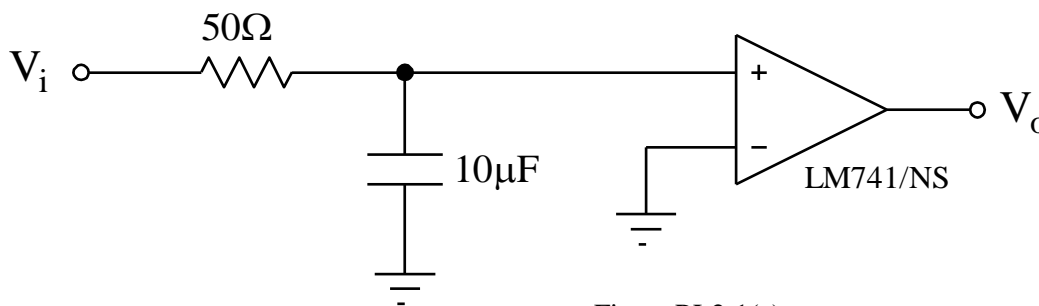


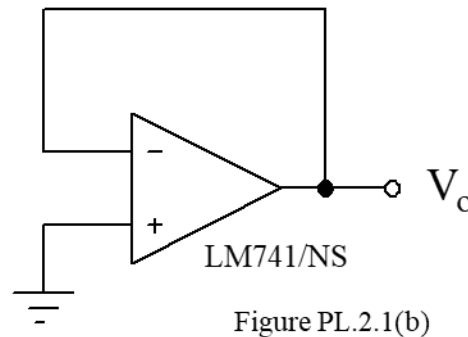
Figure PL2.1(a)

- (b) From the simulated curve, determine the open loop gain A_o for $V_o=0\text{V}$.
- (c) What is V_i for $V_o=0\text{V}$?
- (d) Neglecting bias current, what does the SPICE model assume as a value for offset voltage?
- (e) Now replace the 50Ω resistor in Fig. PL2.1(a) with a $10\text{M}\Omega$ resistor and determine the resultant effective value of offset voltage using SPICE. From this value, determine the input bias current into the positive terminal of the op amp assumed by the 'lm741.mod' model.

- (f) What percent error in offset voltage estimation (mV) does the circuit of Fig. PL2.1(a) incur with the 50Ω resistor if one neglects bias current? (Use the value of bias current determined in part (e).)
- (g) Propose a simple circuit modification (add a single resistor) to eliminate the effects of bias current.

Although this method of eliminating bias current is typically used in practice, in this case the offset voltage is so large and the input resistors are so small that it is impractical. We will therefore just neglect bias current in our lab measurements, keeping in mind the small error incurred.

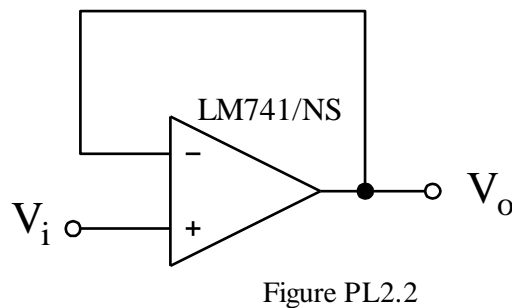
- (h) Assuming that the circuit depicted in Fig. PL2.1(b) uses the op amp model defined in 'lm741.mod', what is V_o ? What is the source of this voltage?



- (i) Does bias current influence this value?

2. Frequency Response

Consider the circuit in Fig. PL2.2



- (a) Using SPICE, plot the frequency response $V_o(f)/V_i(f)$.
- (b) From the plot in (a), what is the unity gain bandwidth f_u of the op amp modeled by 'lm741.mod'?

- (c) Using the open loop gain determined using SPICE in part 1(b) and the unity gain bandwidth determined in part 2(a) above, compute the dominant pole frequency (i.e., the 3dB frequency, f_b) for the 741 modeled in 'lm741.mod'.

If you use SPICE to simulate the circuit to >50 MHz, you will find that the performance deviates from that predicted by the single pole model we have assumed. This happens because the actual 741 op-amp has additional high frequency singularities. Still, the single pole model works reasonably well, since the dominant pole is several decades away from the other singularities and we are typically interested in frequencies where the additional poles have little effect (frequencies where the op-amp has gain). In the lab, you will make bandwidth measurements at several closed loop gains to verify that the single pole model is justified.

- (d) Now use SPICE to directly determine the open-loop unity gain frequency and open-loop 3dB-down corner frequency (i.e., f_b) of the 741 op amp modeled by 'lm741.mod'. [Hint: You must eliminate offset voltage by applying an appropriate compensating voltage to get a proper op amp response.]

3. Large Signal Characteristics

- (a) Using SPICE, plot the step response of the 741 configured as a unity gain buffer (see Fig. P2.2). Apply a -10V to 10V step.
- (b) Based on the output, determine the slew rate of the op-amp SPICE model.

Helpful Hints for HSPICE Simulation

The Op Amp Model to Use:

To characterize the op amp, you will need to include the model file 'lm741.mod' in the netlist. You can find the model file on the course website. To include the model file, use the following command at the beginning of the netlist:

```
.include 'path_to_file/LM741.MOD'
```

Open the .mod file with your favorite text editor. Scroll down until you see the following content:

```
.SUBCKT LM741/NS      1  2  99  50  28
```

The '.SUBCKT' command defines a sub-circuit called 'LM741/NS' with five ports, connected to port 1, 2, 99, 50, and 28 defined in this sub-circuit. You will find the definition of all ports above the name of each port.

To characterize the op amp described in the model file, you will need to create an instance of it and wire the ports. To create an instance, use the following command:

```
XLM741  P1  P2  P3  P4  P5  LM741/NS
```

Replace P1~P5 with nodes accordingly.

Important: Whenever wiring a circuit, be sure to include a DC path to ground on the output. HSPICE needs a ground reference to converge on a DC bias point to simulate your circuit. For

your op amp circuit, do this by connecting a large resistor (e.g., 100meg) to ground on the output of your circuit.

Simulation:

In this Pre-Lab, you need to do three HSPICE analyses: a DC analysis for part 1, an AC analysis for part 2, and a TRAN (i.e., transient) analysis for part 3. The following gives descriptions of the commands for DC, AC, and TRAN analyses in HSPICE. They assume that you have correctly created the netlist for your circuit with appropriate syntax. You will need to use different types of voltage source and simulation commands for the three analyses.

DC Analysis:

To get the voltage transfer characteristics (VTC), you need to use the DC sweep analysis. In particular, for this Pre-Lab, you need to plot V_o vs. V_i . To sweep V_i , you will need to connect a DC voltage source to the node:

```
vs vi gnd 1V
```

The command above creates a voltage source called v_s across V_i and GND with a level of 1V. DC simulation requires the `.dc` command at the end of the netlist:

```
.dc <SOURCE> <START> <STOP> <STEP>
```

Since we are sweeping V_i , `<SOURCE>` should be V_i . Choose the start, stop and step values such that you can get at least 10 data points within the transition region of the VTC.

To plot the VTC, follow these steps:

(1) Start WaveView using the following command:

```
> awaves &
```

(2) A 'CustomExplorer Console' window will pop up. Click 'Open Waveform File' under 'File'. A 'Custom WaveView' window and an 'Open: Waveform Files' window will pop up.

(3) Find 'file_name.sw0' in the 'Open: Waveform Files' window. Double click the file.

(4) Under the 'Output View' section of the 'Custom WaveView' window, click the '+' sign before the filename. An object called 'toplevel' will show up.

(5) Click on 'toplevel', and you will see a list of signals within the circuit in the section below.

(6) Double click on the output voltage, and you will see the VTC plotted on the right.

(7) Zoom in on the plot as necessary to answer the questions.

AC Analysis:

To run the necessary AC analyses in part 2, replace the DC source and the simulation commands with the following:

```
vs vi gnd ac 1V
```

The command above creates an AC voltage source across V_i and GND with a 1V peak voltage. To get HSPICE to do an AC analysis, use

```
.ac <LIN|DEC|OCT> <NUMBER OF SAMPLES> <START> <STOP>
```

Replace <LIN|DEC|OCT> with LIN, DEC or OCT. LIN performs a linear sweep, DEC performs a decade sweep, and OCT performs an octave sweep. Replace <NUMBER OF SAMPLES> with the number of samples, <START> with the starting sweep frequency and <STOP> with the ending sweep frequency.

To plot the frequency response in WaveView, follow the same steps as described in DC Analysis. The new filename is 'file_name.ac0'.

TRAN Analysis:

To run the necessary TRAN analyses in part 3, replace the DC source and the simulation commands with the following:

```
vs vi gnd pwl (t1 V1 t2 V2 t3 V3 t4 V4)
```

The command above creates a piece-wise linear source. At time t_i the voltage source generates V_i , and the transition between V_i and V_j has a linear dependence on time.

```
.tran <STEP> <STOP>
```

Replace <STEP> and <STOP> with the step of time and the total time accordingly.

To plot the transient response in WaveView, follow the same steps as described in DC Analysis. The new filename is 'file_name.tr0'.

Laboratory 2: Characterization of the 741 Op Amp
Results Sheet for Preliminary Exercises

NAME: _____

LAB SECTION: _____

1. Open Loop Characteristics

(a) Attach plot.

(b) $A_o =$ _____(c) $V_i =$ _____(d) $V_{offset} =$ _____(e) $V_{offset,eff} =$ _____ $I_{bias} =$ _____

(f) % error = _____

(g) Sketch your modified circuit and briefly explain its operation.

(h) $V_o =$ _____ What is the source of this voltage? _____

(i) _____ Explain your answer.

2. Frequency Response

(a) Attach plot and annotated SPICE file.

(b) Unity gain bandwidth of the 741 op amp _____

(c) Predicted 3dB bandwidth of 741 op amp _____

(d) Open-loop unity gain frequency f_u (from SPICE) _____

Open-loop corner frequency f_b (from SPICE) _____

3. Large Signal Characteristics

(a) Attach annotated SPICE plot.

(b) Slew Rate = _____

Laboratory 2: Characterization of the 741 Op Amp
Laboratory Exercises

INTRODUCTION

Objectives

In this lab, we will measure the parameters of an actual 741 op amp. The measurement of these parameters will provide insight into the limitations of real world op-amps in various circuit applications.

Summary of Procedures

Using a 741 op amp, measure the following parameters:

- (i) DC open loop gain
- (ii) Input offset voltage
- (iii) Dominant pole frequency
- (iv) Unity gain bandwidth
- (v) Slew rate

Materials Required

- Oscilloscope
- HP 4155B Semiconductor Parameter Analyzer
- Proto-board
- Function Generator
- DMM
- 741 Op Amp
- Assorted Resistors and Capacitors

PROCEDURE

1. Open Loop Characteristics

This part of the lab will use the HP 4155B Semiconductor Parameter Analyzer to measure the open-loop characteristics of the 741 op amp. Fig. L2.1 shows the measurement circuit. Construct this circuit on your proto-board. (The pinout for the 741 op amp can be found in the part datasheet or in the General Lab Instruction Manual.)

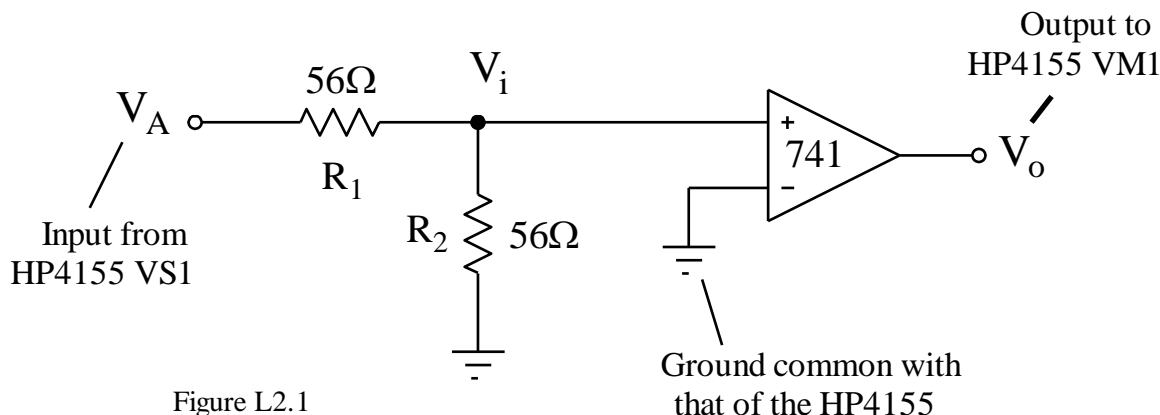


Figure L2.1

- (a) Before beginning, measure the values of R_1 and R_2 with the DMM. Use these measurements to accurately compute the ratio of V_i to V_A . Be sure to take care with these computations as the accuracy of the following measurements depends on this data. Wire the circuit on the Global proto-board (including the power supply connections to the op amp).
- (b) The HP 4155B can be used to measure the open loop gain A_o and the offset voltage V_{offset} . Take DC measurements of V_o for the input V_A ranging from -100mV to $+100\text{mV}$.

Take your proto-board with the circuit from Fig. L2.1 to the HP 4155B. Hook up the circuit to the proper signals on the HP 4155B— V_A to **SMU1** (Source Measurement Unit 1), ground to the **GND**, and V_o to **VM1** (Voltage Monitor 1). Measure A_o and V_{offset} by following the instructions on Page 10.

Important: You must take into account the scaling factor k when determining A_o and V_{offset} . The input voltage from the HP 4155B calculates A_o from the slope of the measured characteristic, it actually calculates

$$A_o(\text{HP4155B}) = \frac{\Delta V_o}{\Delta V_A}$$

However, we want

$$A_o = \frac{\Delta V_o}{\Delta V_i} = \frac{\Delta V_o}{\Delta V_A} \times \frac{\Delta V_A}{\Delta V_i} = A_o(\text{HP4155B}) \times k, \text{ where } \frac{V_i}{V_A} = \frac{R_2}{R_1 + R_2} = \frac{1}{k}$$

Analogously,

$$V_{offset} = \frac{1}{k} \times x - \text{intercept.}$$

2. Closed Loop Characteristics


Using the same op amp you used in Part 1, build a unity gain buffer on your proto-board.

- (a) Ground the input and measure the output voltage with a DVM set to the mV scale.
- (b) Connect the input of the amplifier to the function generator and the output to the oscilloscope. Carefully measure the 3dB frequency of the amplifier.

- (c) Connect the input of the amplifier to a 10Vpp, 10-kHz square wave. Use the response of the amplifier to this input to estimate the slew rate of the op amp.
- (d) Reconnect the amplifier as a non-inverting amplifier with a gain of 11. Recall that the feedback network of this amplifier has a gain of 1/11, therefore, the loop gain will reach unity when the op amp has gain 11, not 10. Carefully measure the 3dB frequency of the amplifier.
- (e) Reconnect the amplifier as a non-inverting amplifier with a gain of 101. Carefully measure the 3dB frequency of this amplifier.

HP 4155B Instructions for Measurement of Op Amp Open-Loop Characteristics

This lab uses the HP 4155B Semiconductor Parameter Analyzer to measure characteristics of the 741 operational amplifier. The following procedure will make the necessary measurement and provide the required data plot. The HP 4155B has two types of keys: “hard” keys, which are dedicated buttons on the front panel, and a column of “soft” keys, just to the right of the screen. In the procedure below, a KEY is a hard key, and a KEY is a soft key. Note that you may have to use the EXTN (extension) soft key, which pages from one soft key menu to the next, to find some of the indicated soft key entries.

- (1) MENU ; NEXT to get the *****CHANNEL DEFINITION ***** window. Set the **NAME** of **SMU1** as ‘**V1**’ and ‘**I1**’ and the **SOURCE** of **SMU1** as ‘**V**’ and ‘**VAR1**’. This sets **SMU1** as a swept voltage excitation port. Set **SMU2~SMU4** as ‘**NOT USE**’.
- (2) NEXT to get the *****SOURCE SET UP***** window. Set **VS1** to 18V and **VS2** to -18V. This sets **VS1** as a constant 18V voltage source and **VS2** as a constant -18V voltage source. These two ports are the DC supplies of the circuit.
- (3) Set **SMU1** as ‘**LINEAR START = -100mV STOP = 100mV STEP = 1mV COMPLIANCE = 100mA**’. This sets **SMU1** signal as a linearly swept voltage excitation from -100mV to 100mV with 1mV step with a 100mA compliance.
- (4) NEXT to get the *****MEAS & DISP MODE SET UP***** window. Set the **X axis** as ‘**V1 LINEAR**’ and **Y1 axis** as ‘**VM1 LINEAR**’.
- (5) MENU ; NEXT enough times to get the *****GRAPHICS PLOT***** window.
- (6) SINGLE to perform the measurement.
- (7) To get a measurement line ($A_0 = k \times \text{GRAD}$, $V_{offset} = \frac{1}{k} \times x - \text{intercept}$):
 - (a) EXTN MARKER to turn marker on; move it to first point with the  knob.

- (b)

EXTN

SHORT CURSOR

 to turn on cursor.
- (c)


EXTN

LINE ON

 to turn line on; the line will be drawn between the cursor and the marker.
- (d)

CURSOR →o

CHANGE POINT

.
- (e) Move cursor to second point with the  knob;

CURSOR →o

 to move line. (Note: the two points you choose should be the most symmetric about the x -axis.)

Results Sheet for Laboratory Exercises

NAME: _____

LAB SECTION: _____

1. Open Loop Characteristics(a) $R_1 =$ _____, $R_2 =$ _____ $k =$ _____(b) Attach HP 4155B plot, annotated with V_{offset} and A_o . Show all calculations on the plot. $V_{offset} =$ _____, $A_o =$ _____**2. Closed Loop Characteristics**(a) $V_o =$ _____ What is the source of this voltage?

Contrast the value of V_{offset} measured in this way to V_{offset} estimated from the open loop plot measured in Part 1. What factors induce error in each measurement?

(b) 3dB Frequency (attach oscilloscope plot) = _____

What is the open loop gain of the op amp at this frequency? _____

(c) Slew Rate (attach oscilloscope plot with $\Delta V/\Delta t$ markers) _____

Was the distortion on the output totally due to slew rate limitations? How can you tell the difference between a square wave limited by slew rate and one limited by frequency response? Sketch an example of each below.

(d) 3dB Frequency (attach oscilloscope plot) = _____

Define β as the voltage V_f/V_o of the resistive feedback network. What is the loop gain $|\beta A(j\omega)|$ at the 3dB frequency measured above?

(e) 3dB Frequency (attach oscilloscope plot) = _____

What is the open-loop gain of the op amp at this frequency?

3. GENERAL QUESTIONS

- (a) Using the DC open loop gain and all the 3dB frequencies measured above:
- (i) Create a detailed plot of the open loop frequency response of the op amp. Be sure to indicate all the above data points on your plot. You should additionally indicate the dominant pole frequency and unity gain frequency on your plot. You may assume a single pole model of the op amp.
 - (ii) Based on your gain plot, create an open-loop phase plot of the 741 op amp. Compare your results to a simulation of the phase plot for the open-loop op amp modeled by 'lm741.mod' using SPICE. How well does your plot agree with the data sheet plots? *Analytically* explain any discrepancies between your plot and the data sheet plots. (You should estimate the location of any additional poles or zeros that you postulate.)
- (b) Could you conveniently make a slew rate measurement using a sinusoidal input for the 741 op amp used in this lab with the circuit given below? Why or why not?

