

Laboratory 2: Characterization of the LMC6482 Op Amp

Preliminary Exercises

This lab will characterize an actual LMC6482 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 Texas Instruments for the LMC6482 op amp. It appears on-line in the file 'lmc6482.mod' in the website for this class. Since this is the official model given by Texas Instruments in association with their LMC6482 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 LMC6482 op amp model in various circuits using SPICE.

At the end of this document is a section with specific tips for the HSPICE simulations these exercises will entail. Please consult this section as you work through the questions below.

1. Open Loop Characteristics

- (a) Generate the netlist for the circuit of Fig. PL2.1. Then use SPICE to determine the voltage transfer characteristic (VTC) V_o versus V_i for $V_i = (-10\text{mV to } 10\text{mV in } 10\mu\text{V steps})$. Again, for the LMC6482 op amp, use the SPICE model given on-line on the course website in the file 'lmc6482.mod', and bias it with $\pm 5\text{ V}$ supplies. Note the warning in the model file and make sure to set g_{min} in the `.options` line of your netlist.

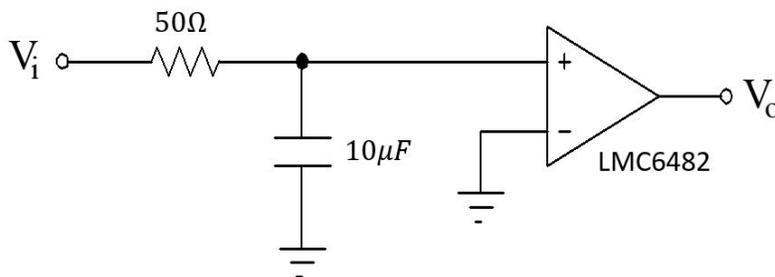


Figure PL2.1

- (b) From the simulated curve, determine the open loop gain A_o for $V_o = 0\text{ V}$.

- (c) What is V_i when $V_o = 0$ V?
- (d) Neglecting bias current, what does the SPICE model assume as a value for offset voltage?
- (e) Now replace the $50\ \Omega$ resistor in Fig. PL2.1 with a $10\ \text{G}\Omega$ 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 'lmc6482.mod' model.
- (f) What percent error in offset voltage estimation (mV) does the circuit of Fig. PL2.1 incur with the $50\ \Omega$ 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. PL2.2 uses the same op amp model as defined in 'lmc6482.mod', what do you expect for the value of V_o ? What is the source of this voltage?

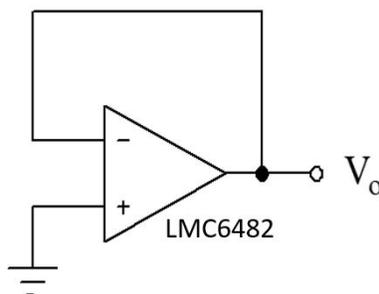


Figure PL2.2

- (i) Does bias current influence this value?

2. Frequency Response

Consider the circuit in Fig. PL2.3

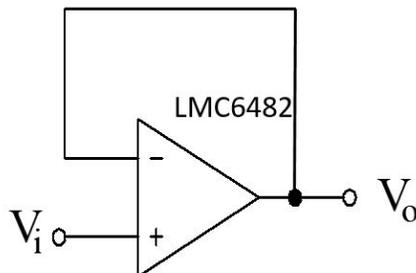


Figure PL2.3

- Using SPICE, plot the frequency response $V_o(s)/V_i(s)$.
- From the plot in (a), what is the unity gain bandwidth f_t of the op amp modeled by 'lmc6482.mod'?
- 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 op amp modeled in 'lmc6482.mod'.

If you use SPICE to simulate the circuit to frequencies higher than ~ 10 MHz, you will find that the performance deviates from that predicted by the single pole model we have assumed. This happens because the actual LMC6482 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 the most gain). In the lab, you will make bandwidth measurements at several closed loop gains to verify that the single pole model is justified.

- 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 LMC6482 op amp modeled by 'lmc6482.mod'. [Hint: You must eliminate offset voltage by applying an appropriate compensating voltage to get a proper op amp response.]

3. Large Signal Characteristics

- Using SPICE, plot the step response of the LMC6482 configured as a unity gain buffer (see Fig. P2.2). Apply a -5V to 5V step.
- 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 'lmc6482.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/LMC6482.MOD'
```

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

```
.SUBCKT LMC6482A/NS 1 2 99 50 40
```

The '.SUBCKT' command defines a sub-circuit called 'LMC6482A/NS' with five ports, connected to port 1, 2, 99, 50, and 40 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:

```
XLMC6482 P1 P2 P3 P4 P5 LMC6482A/NS
```

Replace 'XLMC6482' with a name of your choosing (it must begin with an 'X'), and 'P1' – 'P5' with your node names 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'. Make sure to set your y-axis scale to 'dB20' for the magnitude response.

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 LMC6482 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 LMC6482 op amp _____
- (c) Predicted 3dB bandwidth of LMC6482 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 = _____