Laboratory 3: Feedback Amplifier Compensation (Spice Component)

Due Week 10 (Oct. 22-26)

1 Introduction

The purpose of this assignment is to carry out SPICE simulations on feedback amplifiers, to observe the effects of different feedback networks and compensation techniques on amplifier stability. The values from this assignment will be compared to the measurements in the experimental part of Lab 3. You should complete the simulations before attempting the lab measurements to help you know what to expect in Lab 3.

2 Procedure

Create a .cir file for the circuit shown in Fig. 1. **Note the configuration of the input terminals of the third opamp** (its terminals are flipped with respect to the other opamps). The model for the 741 opamp is available on the class website. Number your nodes as indicated to maintain consistency with the rest of the class, and to make it easier to help you if you have questions.

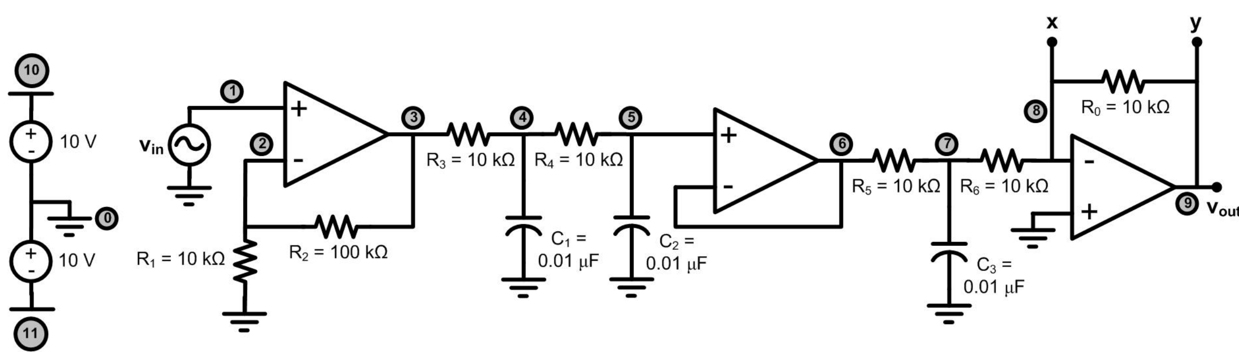

Figure 1: Open-loop amplifier schematic.

2.1 Open-loop Amplifier Characterization

The circuit in Fig. 1 models an op-amp with multiple gain stages, where v_{in} is the negative input, and the positive input is implicitly grounded. This circuit has a high input resistance, low output resistance, and reasonably high gain. The capacitors $C_1 - C_3$ limit the

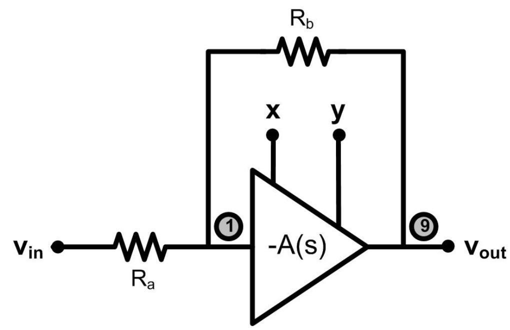

Figure 2: Schematic for uncompensated amplifier with feedback.

high-frequency response of this amplifier, playing the role that internal transistor parasitic capacitances play in actual IC (integrated circuit) op-amps. Run the following simulations:

1. Run an AC simulation and plot the magnitude and phase response.
2. Simulate the transient response to a 50 mV, 100 Hz square wave (use the PULSE voltage source in SPICE for the v_{in} source). Report the rise and fall times of the output signal (**note:** the rise time is the time it takes the signal to increase from 10% to 90% of its final value).

2.2 Closed-loop Amplifier Characterization (No Compensation)

In the next part of the experiment we will close a feedback loop around the amplifier and characterize its performance for different feedback factors and compensation methods. Connect the feedback network around the amplifier, as shown in Fig. 2. The triangular block represents the entire 3-stage amplifier of Fig. 1 (our "op-amp"). Note that this feedback configuration corresponds to the familiar "inverting amplifier" op-amp topology. What do you expect the gain to be in this case? Terminals x and y should be left open, but the resistor labeled R_0 in Fig. 1 should not be removed.

Now run the following simulations:

1. Let $R_a = 1$ kΩ and $R_b = 3.3$ kΩ. Ground the input (remove the v_{in} source in Fig. 1 and connect the positive input of the first opamp to node 0 (ground) instead of node 1)and run a transient simulation **with a length of 1 ms**. Plot the output voltage. Does the feedback amplifier oscillate? If so, plot the waveform at v_{out}, and report the approximate period. Is this the expected period based on your bode plots for the open-loop amplifier?
2. Now let $R_a = 1$ kΩ and $R_b = 15$ kΩ. Ground the input and run a transient simulation **with a length of 1 ms**. Plot the output voltage. Does the feedback amplifier oscillate? Keep in mind that any non-periodic disturbance that you observe is probably just a numeric artifact.

3. Change the input to a 0.1 V, 100 Hz square wave and plot the output for two periods. Report the low-frequency gain of the amplifier based on this plot. Is there any ringing in the output signal? Report the rise and fall times of the output signal, and the frequency of any ringing effects that are present.

2.3 Amplifier Compensation

We will now compensate the second version of the amplifier (with $R_a = 1$ kΩ, $R_b = 15$ kΩ) to increase its stability (and thereby reduce the ringing effects observed in the transient simulation). We will do this using dominant pole compensation, as described in pp. 848-851 of the text. Run the following simuations:

1. Re-run the AC simulation for the open-loop amplifier and plot the magnitude and phase responses. What is the phase margin of the amplifier for the given feedback factor (β)? Recall that $A_{CL} \approx 1/\beta$ for large open-loop gains.

2. Now incrementally increase the value of C_2 (the dominant pole in the amplifier) until the amplifier has a phase margin of about 70°. **Record the final value of** C_2.

3. An alternate way to compensate the amplifier is by introducing a Miller capacitance across terminals x and y. As discussed in class, the advantage of this approach is that compensation can be achieved with a smaller capacitor. Change C_2 back to its original value, and add a capacitance C_C across terminals x and y in Fig. 1. Adjust this capacitance until the amplifier has a phase margin of about 70° (again, taking into account the β that results from the closed-loop configuration that we are using). **Record the value of** C_C, and compare how much capacitance we have added to the circuit in this case to the case of increasing the size of C_2.

4. Now re-run the transient simulation (with the 0.1 V, 100 Hz square wave input) for the closed-loop amplifier with the Miller compensation (with C_C included across terminals x and y). Do you see any ringing? Report the rise and fall times and overshoot of the output waveform.

5. Now double the Miller capacitance C_C and re-run the transient simulation (with the 0.1 V, 100 Hz square wave input). Report the rise and fall times and overshoot of the output waveform. Why should we avoid "over-compensating" our circuits?

3 Lab Report

Include the following in your lab report:

1. Your netlist (.cir file).
2. AC and transient response plots for the open-loop amplifier without compensation.
3. AC response plots for the open-loop amplifier with both types of compensation (increasing C_2 and Miller compensation). Mark the phase margin on these plots.
4. Transient response plots to the square wave input for the Miller compensated amplifier for the both values of C_C. Mark the overshoot and rise and fall times on these plots.