EE4350 Lab 3
Spring 2025

Current-Mirror Operational Amplifier Design

The current mirror op amp is shown below. This can be used as a standard operational amplifier in a feedback configuration though when it was introduced the primary focus was on using it in open-loop applications as an Operational Transconductance Amplifier (OTA).

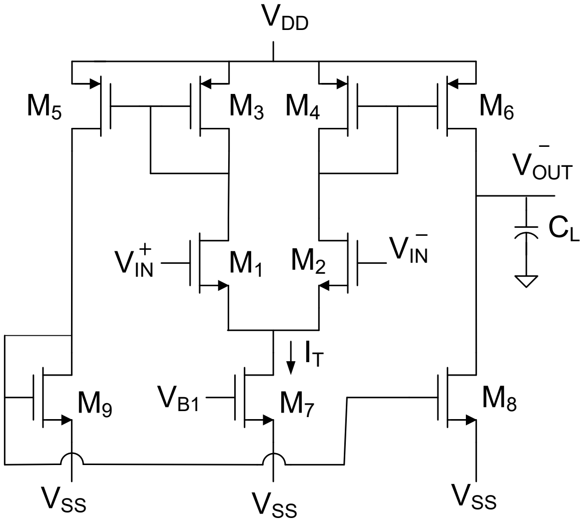


Figure 1: Current Mirror Op Amp Schematic

1. Design this op amp in a 0.18μm process. In this design:
   - $V_{DD} = 1.65V$ and $V_{SS} = -1.65V$
   - The lengths of all devices should be $5xL_{min}$
   - The mirror gains M_{35} and M_{46} should be 10
   - The mirror gain M_{98} should be 1
   - The power dissipation should be 1mW
   - $V_{EB} = 100mV$ for all devices

Note: In the .18μ process we are working with there are two types of transistors. One type is comprised of low-threshold transistors and the other of high-threshold transistors. The total supply voltage for circuits using low-threshold transistors for this process is 1.8V and for circuits using high-threshold transistors it is 3.3V. Please use the high-threshold devices for this design.

Low threshold devices = nmos2v/pmos2v

High threshold devices = nmos3v/pmos3v

2. In Part 1 you were given several constraints in this design. Identify the number of degrees of freedom you have in the design of the current mirror op amp and a complete set of design variables for this design using the practical design variables. Determine the number of degrees of freedom you have left if you satisfy the constraints given in Part 1.

3. Assuming C_L=20pF, for this design, analytically determine the:
   - Differential voltage gain $A_V(s)$
   - Bandwidth (BW)
   - Gain-Bandwidth (GB)
   - Slew Rate (SR)
   - Output signal swing (assume a common-mode input V_{CM} of -200mV)

   Use the value of λ you obtained for this process as part of the Homework 4 assignment.

   Note: If you solved HW 4 using low-threshold devices, you will need to re-run your λ extraction algorithm for the high-threshold devices.

4. Compare the results obtained in Part 3 with simulations in Cadence. Remember to use a C_L of 20pF.

5. In Cadence, connect the op amp in a unity gain buffer configuration. Use a C_L of 20pF. Using simulations, determine the following for this configuration:

   - What is the 3dB bandwidth (BW)?
   - What is the transient response to a 100mV step input with a V_{CM} of -200mV?

6. Using the small signal testbenches for the open loop configuration and unity gain buffer configuration from parts 4 and 5, determine the poles and zeros of the op amp. Plot the poles and zeros in the complex plane and mark the two dominant poles.

   Note: Poles and zeros can be found using a "pz" analysis in Spectre. See Appendix I for details on running this simulation.

Appendix I: Spectre "pz" Analysis Instructions

Spectre "pz" analysis uses a small signal simulation to determine the poles and zeros of a circuit. Often when designing circuits, we are only interested in the low frequency poles and zeros. In real circuits, many poles and zeros exist though most of them are at very high frequencies.

Spectre "pz" analysis is useful for finding the actual values of the low frequency poles as well as for seeing the higher frequency poles and zeros. "pz" analysis is particularly useful for compensating op amps (talked about later in this class) or for understanding unexpected transfer function behavior from high frequency poles and zeros.

For running "pz" simulations, use the same testbench used for AC analysis of the small signal op amp parameters. The testbench used for determining the small signal open loop characteristics of the Lab 2 op amp is shown in Figure 2 below. This will be used again for determining the open loop poles and zeros.

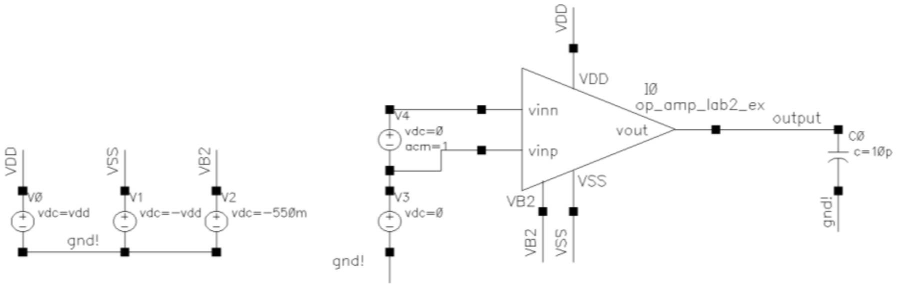
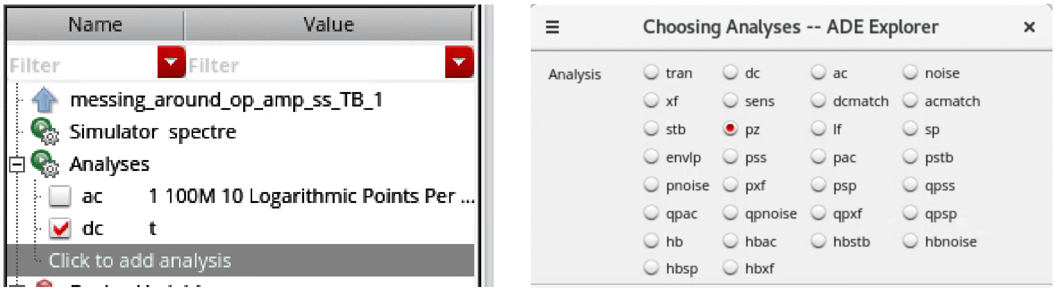


Figure 2: Small Signal Open Loop Characteristics Test Bench Schematic

Open ADE Explorer. Create a new test (See Figure 3a) and select "pz" from the list (See Figure 3b).

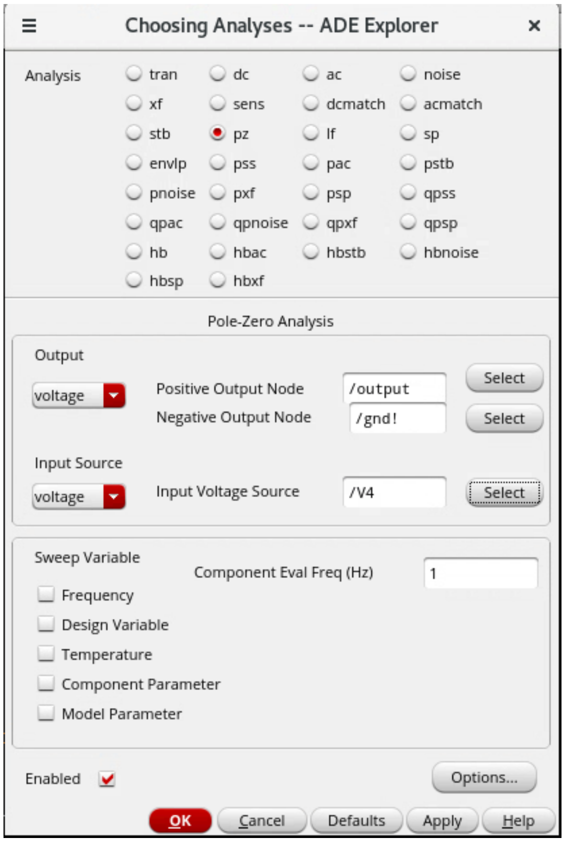


messing_around_op_amp_ss_TB_1
Simulator spectre
Analyses
☐ ac 1 100M 10 Logarithmic Points Per ...
☑ dc t

○ tran ○ dc ○ ac ○ noise ○ xf ○ sens ○ dcmatch ○ acmatch ○ stb ● pz ○ lf ○ sp ○ envlp ○ pss ○ pac ○ pstb ○ pnoise ○ pxf ○ psp ○ qpss ○ qpac ○ qpnoise ○ qpxf ○ qpsp ○ hb ○ hbac ○ hbstb ○ hbnoise ○ hbsp ○ hbxf

Figure 3: a) Click to add another test b) Select "pz"

In the "pz" analysis setup box, click select next to "Positive Output Node". Select the output of the op amp on the testbench. Similarly, click select next to "Negative Output Node" and select your ground node. Lastly, next to "Input Voltage Source", click select. Select the voltage source that supplied the AC input (differential input) in your AC simulation. Do not choose any sweep variables. The setup window should now look similar to that in Figure 4.



Analysis: ● pz

Output: voltage
Positive Output Node: /output
Negative Output Node: /gnd!

Input Source: voltage
Input Voltage Source: /V4

Component Eval Freq (Hz): 1

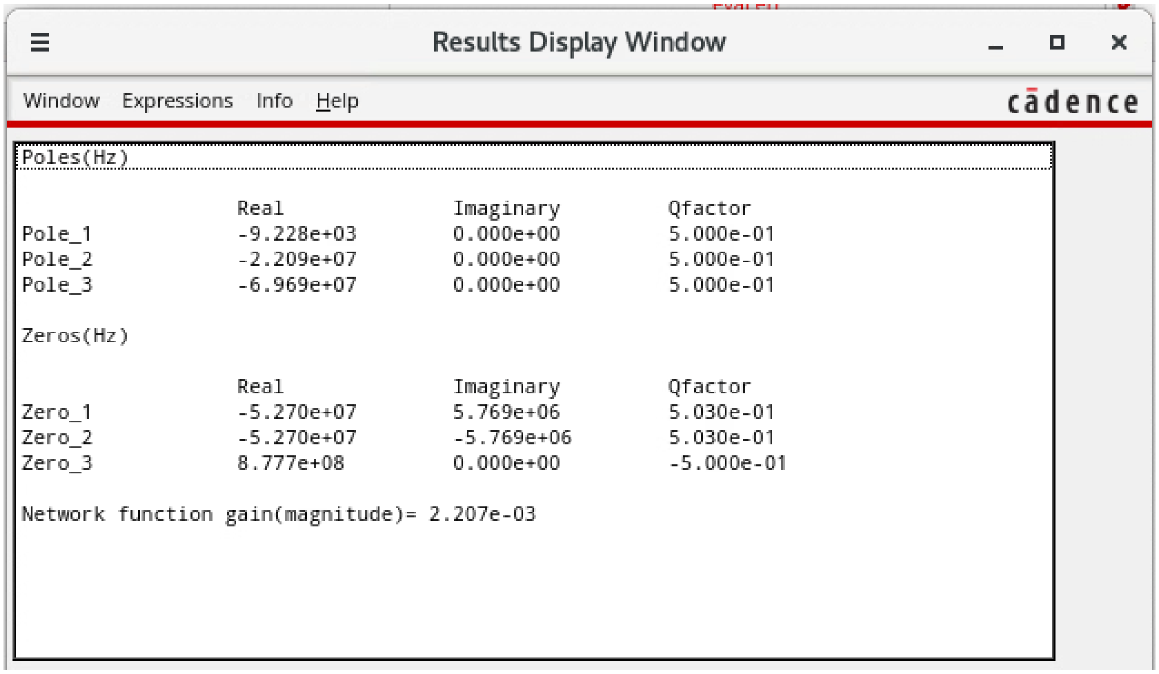
☐ Frequency
☐ Design Variable
☐ Temperature
☐ Component Parameter
☐ Model Parameter

Enabled ☑

Figure 4: "pz" Analysis Completed Setup Window

Click "OK" and run the simulation. No outputs will plot. To see a list of all the poles and zeros, click on Results → Print → Pole-Zero Summary…

You can choose to print poles, zeros, or both. You can leave the other options as the default. A list of poles and zeros will be printed as shown in Figure 5.

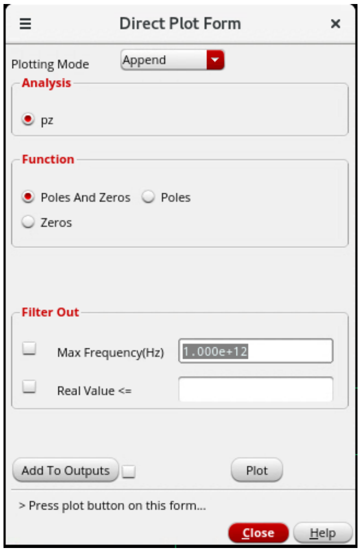


```
Poles(Hz)

                    Real            Imaginary       Qfactor
Pole_1              -9.228e+03      0.000e+00       5.000e-01
Pole_2              -2.209e+07      0.000e+00       5.000e-01
Pole_3              -6.969e+07      0.000e+00       5.000e-01

Zeros(Hz)

                    Real            Imaginary       Qfactor
Zero_1              -5.270e+07      5.769e+06       5.030e-01
Zero_2              -5.270e+07      -5.769e+06      5.030e-01
Zero_3              8.777e+08       0.000e+00       -5.000e-01

Network function gain(magnitude)= 2.207e-03
```

Figure 5: Pole-Zero Summary Window

The poles and zeros can also be plotted in the complex frequency plane. To do so, in ADE Explorer, click Results → Direct Plot → Main Form.

In the window that pops up choose whether you want to plot the poles, zeros, or both and click "Plot" at the bottom of the window (See Figure 6).



Plotting Mode: Append

Analysis

● pz

Function

● Poles And Zeros ○ Poles

○ Zeros

Filter Out

☐ Max Frequency(Hz) 1.000e+12

☐ Real Value <=

> Press plot button on this form...

Figure 6: Pole-Zero Direct Plot Form

The plot will show the poles as crosses and the zeros as circles. The x-axis is the real frequency and the y-axis is the imaginary frequency (See Figure 7). Usually in our analysis, we determine the poles in rad/s. To compare analysis and simulation results, the necessary conversion must be made.

Markers can be places on the poles and zeros to see their values.

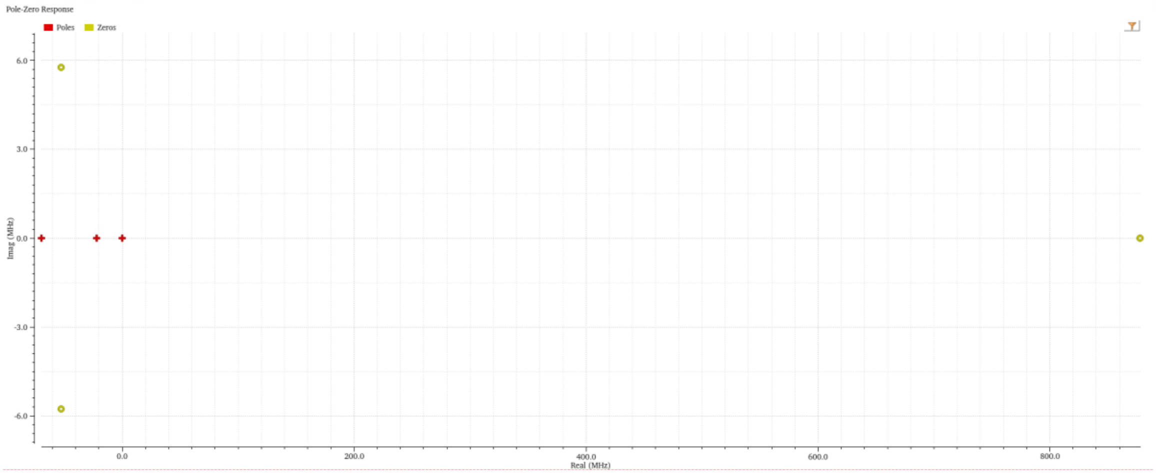


Figure 7: Pole-Zero Plot on the Complex Frequency Plane