وَمَا أُوتِيتُمْ مِنَ الْعِلْمِ إِلَّا قَلِيلًا

Analog/Mixed-Signal Simulation and Modeling
Lab 01
SPICE

Objectives

1. Write and analyze SPICE netlists.
2. Run and analyze different types of simulations (TF, AC, TRAN).
3. Create a behavioral model for an op-amp.
4. Explain the relation between negative feedback closed-loop parameters and open-loop parameters.
5. Perform basic hand analysis and compare it with simulation results.

Instructions

1. Use LTSpice for design entry and simulation. Use this link: https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html
2. It is recommended to use Notepad++ to write your netlist (use SPICE highlighting mode: Language -> S -> Spice). Do NOT use a schematic entry GUI.
3. If necessary, make any reasonable assumptions.
4. Check LTSpice help if needed.
5. If you do not know how to perform basic hand analysis, you must study it on your own. A suggested reference is Sedra/Smith.
6. Submit your solution on Canvas as a single pdf file that contains all the deliverables.

Part 1

Index	Deliverable
1.	Complete and run “vdivider.cir”. Report the netlist and the results.
2.	Complete and run “simple_rc_tran.cir”. Report the netlist and the results.
3.	Complete and run “simple_rc_ac.cir”. Report the netlist and the results.

Part 2

Index	Deliverable
1.	Write a SPICE subcircuit that describes an op-amp with an open-loop gain of 1e4 and a UGF of 10MHz. Use comments generously to describe every line of the netlist. Report the SPICE subcircuit and explain how you chose the circuit parameters.
2.	Use the previous subcircuit to write a netlist of a **non-inverting** amplifier. The feedback resistance is 9kOhm and the other resistance is 1kOhm. Use a 1V DC input. Use comments generously to describe every line of the netlist. **Run transfer function (TF) analysis.** Report a snapshot of the SPICE output file. Justify the output.
3.	Change the input to be a sine wave with 1V amplitude and 1kHz frequency. Run transient analysis for two complete periods. Use a time step = period/50. Report results (Vsig and Vout vs time). Clearly annotate the peak value of Vsig and Vout in the figure.

4.	How much is the voltage gain? Compare the voltage gain acquired from hand analysis, TF analysis, and TRAN analysis in a table. Comment.
5.	Report the waveform of the differential input of the op-amp. Clearly annotate the peak value in the figure. What is the amplitude of this signal? Why (explain with hand-analysis)?
6.	Repeat the previous step but with input frequency equal to the UGF. What is the amplitude of this signal? Why (explain with hand-analysis)?
7.	Run AC analysis to plot the frequency response of the previous non-inverting amplifier (use an AC source). **Use parametric sweep for the feedback resistance with two values (9k and 4k).** Report the gain in dB vs frequency (log-log scale). Clearly annotate the DC gain, the 3dB high cutoff frequency, and the GBW (UGF) in the figure.
8.	If you increase the input amplitude in AC analysis and transient analysis, do you expect to see clipping in the output? Why?
9.	Compare the DC gain, the 3dB high cutoff frequency, and the GBW (UGF) from hand analysis and AC analysis in a table. Comment.

Thanks to all who contributed to these labs. If you find any errors or have suggestions concerning these labs, please contact Hesham.omran@eng.asu.edu.eg.