وَمَا أُوتِيتُمْ مِنَ الْعِلْمِ إِلَّا قَلِيلًا

Ain Shams University – Faculty of Engineering – ECE Dept. – Integrated Circuits Lab.
Dr. Hesham Omran

Analog/Mixed-Signal Simulation and Modeling
Lab 02
Create Your Own Circuit Simulator!

Objectives

1. Parse a netlist description of a linear circuit with arbitrary number of nodes. The circuit may contain resistors, voltage sources, and current sources.
2. Build the modified nodal analysis (MNA) equation system of the circuit using element stamps.
3. Solve the circuit (linear DC and AC analysis) and print the following results for the user:
   a. Symbolic matrix equation.
   b. Numerical matrix equation (substitute with the values given in the netlist).
   c. Symbolic solution of every unknown.
   d. Numerical solution of every unknown (substitute with the values given in the netlist).

Instructions

1. Use Octave or Matlab to write your program (use the Symbolic Toolbox and define symbolic variables using 'syms' command).
2. Use LTSpice for design entry and simulation. Use this link: https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html
3. You may use Notepad++ to write your netlist (it has SPICE highlight mode). Do NOT use a schematic entry GUI. Do NOT use any other program.
4. If necessary, make any reasonable assumptions.
5. Check LTSpice help if needed.
6. Submit your solution on Canvas as a single zip file that contains all the deliverables.

Part 1

Index	Deliverable
1.	Complete the missing lines in "Solve_Circuit.m". This is a program written as a single function. The function should accept the netlist as an input (a path to a text file). The function should return the symbolic and numerical results as explained in the objectives section. You may assume the following for the netlist: • Node 0 is the ground node. • Other nodes are named as positive integers.
2.	Complete the missing lines in "Solve_Circuit_Examples.m". This is a script that calls your function and prints the output clearly in the command window for the two given netlist files ("circuit_1.cir" and "circuit_2.cir").
3.	Simulate the netlists and compare LTspice output with your simulator output in a table. Comment.

Part 2

Index	Deliverable
1.	Create "Solve_AC_Circuit.m" to add support for inductors, capacitors, and AC analysis.
2.	Create a script ""Solve_AC_Circuit_Examples.m" to test your code with three netlists of an RLC circuit to cover three cases: overdamped, critical damped, and underdamped. Your simulator should perform AC analysis and plot magnitude and phase vs frequency.
3.	Report your simulator results.
4.	Simulate the netlists using LTspice and compare with the results of your simulator. Comment.

Part 3

Index	Deliverable
1.	Create "Solve_AMP_Circuit.m" to add support for VCCS and VCVS.
2.	Create a script ""Solve_AMP_Circuit_Examples.m" to test your code with the op-amp circuit you created in Lab 01 connected in unity-gain feedback configuration. Your simulator should perform AC analysis and plot magnitude and phase vs frequency.
3.	Report your simulator results.
4.	Simulate the netlist using LTspice and compare with the results of your simulator. 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.