

Digital Electronics Project 5 – SPICE Simulation 2

Due: 23:59, Apr. 21, 2022

In this project, you are going to simulate SPICE circuit netlist using ngspice software on your system. We will have SPICE simulations on RC and RL circuit networks. Please perform transient SPICE simulations on the following four circuits in our Lecture07 and Lecture08 slide files: (1) L07:P15, (2) L07:P17, (3) L08:P11, and (4) L08:P13. Since the values of voltage, resistance, capacitance, and inductance are not assigned, please assign your own values during simulations and observe the changes of corresponding time constants (τ) in both RC and RL circuits. For the capacitor/inductor in each circuit, you need to report the voltage across that component and the current through it.

Please submit your simulation report according to the following rules:

- 1- The font size of your report is 12 in PDF format.
- 2- The filename is your student ID (e.g., B12345678.pdf).
- 3- Post the SPICE circuit netlists under simulations.
- 4- Plot the voltage and current transient behaviors of the capacitor and inductor.
- 5- Analyze the time constants when you alter the values of R, C, and L.
- 6- Screenshots of your simulation results.

Hint: Initial condition of capacitor/inductor may be required for transient simulations.

Reference:

[1] Ngspice user's manual,

URL: <http://ngspice.sourceforge.net/docs/ngspice-manual.pdf>