

Digital Electronics Project 4 – SPICE Simulation 1

Due: 23:59, Mar. 31, 2022

In this project, you are going to simulate SPICE circuit netlist using ngspice software on your system. We will have SPICE simulations on resistive circuit networks with DC voltage/current source. A comprehensive ngspice user manual is available online [1]. Please perform SPICE simulations on the following four circuits in our Lecture06 slide file: (1) L06:P16, (2) L06:P20, (3) L06:P22, and (4) L06:P26. For every component in each circuit, you need to report the voltage across individual 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- List the voltage and current values of all components.
- 5- Screenshots of your simulation results.

Hint: Dummy voltage sources may be required to complete this project assignment.

Reference:

[1] Ngspice user's manual,

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