Digital Electronics Project 6 – SPICE Simulation 3
Due: 23:59, April 28, 2022

In this project, you are going to simulate SPICE circuit netlist using ngspice software on your system. We will have SPICE simulations on band-pass fliter, band-stop filter, and RLC circuit networks. Please perform transient SPICE simulations on (1) the two circuits drawn in our Lecture10 slide file: L10:P10, (2) the circuit drawn in our Lecture10 slide file: L10:P11, and (3) the RLC circuit drawn in our Lecture10 slide file: L10:P15. Since the style of voltage sources and the values of resistance, capacitance, and inductance are not assigned, please assign them during simulations and observe the voltage transient behaviors of the load resistance.

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 transient behaviors of the load resistance.
5- Analyze the transient behaviors when you alter the values of R, L, and C.
6- Screenshots of your simulation results.

Reference:

[1] Ngspice user's manual,

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