EECE 311 Laboratory Project #5

Instructor: Dr. Hede Ma

Modeling and Analyzing RLC Circuit with Pspice

1. **Objective**
   The objectives of this laboratory exercise are to learn and gain experience in analyzing and simulating the behavior of first order RC or RL circuits and second order RLC circuit under transient conditions using Pspice.

2. **Laboratory Procedure**
   Generate a PSPICE input file for the circuit shown as follows to find the capacitor voltage v_c from 0 to 20 ms with a time increment of 5 µs. The voltage-controlled switch S_1 is closed to position a at t=0 and then moved instantaneously to position b at t=10 ms. Assume that the model parameters of the switch are RON=0.01, ROFF=10E+5, VON=0.1v, and VOFF=0v.
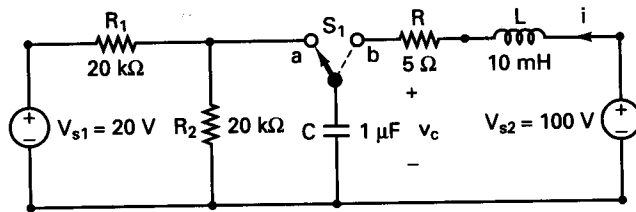
3. **Pspice Transient Analysis**
   1. Use the netlist.
   2. (optional) Construct the circuits using the Draw, Get New Parts, and Edit Attribute commands in the same manner as when doing a steady state DC circuit analysis. Select Transient under Analysis Setup. Click on the Transient button to select both the transient period ending time and data point time intervals to achieve the desired graph appearance. Simulate with the probe and order the desired trace variables with multiple Y-axis setting.