EE 220/220L Circuits I (Fall 2019)
Laboratory 6 PSpice Demonstration and Use

Background

The goal of this lab is to become familiar with the circuit simulation program called PSpice.

Preliminary

Your lab instructor(s) will demonstrate the PSpice program during the lab period. Take careful notes! This program is available in the computer labs on campus. A copy of the program (version 17.2) is available on the course web page along with some installation instructions. A wealth of information on PSpice is available on the web, e.g., check course web page and/or use an internet search engine with "orcad pspice tutorial" as the search topic.

Experiment

1) In PSpice, draw/compose the circuit from Lab #4 (*Nodal Analysis*) using **measured** source and resistor values. Add text showing EE 220L- lab number & title, the *date* of simulation, and *your name*. Run the PSpice simulation. Display all the DC voltages and print a hard copy. Then, display all the DC currents and print a **separate** hard copy (manually indicate current directions). Affix the two hardcopies in your logbook. Use the currents and voltages found using PSpice to calculate the power dissipated by each component or display the DC powers and print another hardcopy.

2) Repeat 1) for the circuit of Lab #5 (*Mesh Analysis*). In the circuit, use **measured** source voltage and resistor values. Also, adjust the transistor values (e.g., β_{meas}, V_{BE}) to match the values you **measured**/calculated as closely as possible. Add text showing EE 220L- lab number & title, the *date* of simulation, and *your name*. Run the PSpice simulation. Display all the DC voltages and print a hard copy. Then, display all the DC currents and print a **separate** hard copy copy (manually indicate current directions). Affix the two hardcopies in your logbook. Use the currents and voltages found to calculate the power dissipated by each component or display the DC powers and print another hardcopy. [Hint: To get transistor model, search for Q2N2222.]

Analysis and Conclusions

1) In two tables (one each for Labs 4 & 5), tabulate and compare (i.e., % difference with respect to your measured values) the **measured** current, voltage, and power values from Labs 4 and 5 with the results from your PSpice simulations. Table format: quantity description/name/variable in column 1, measured values in column 2, PSpice values in column 3, and % differences in column 4.

2) Discuss your results.