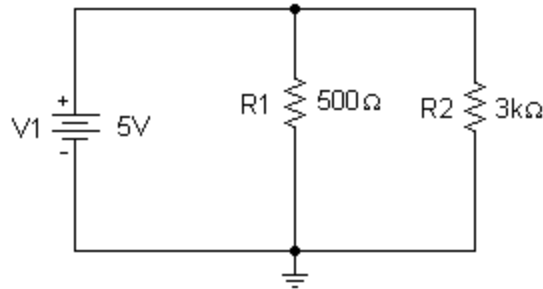


EE 206 Simulation 1

Basic PSpice Statements

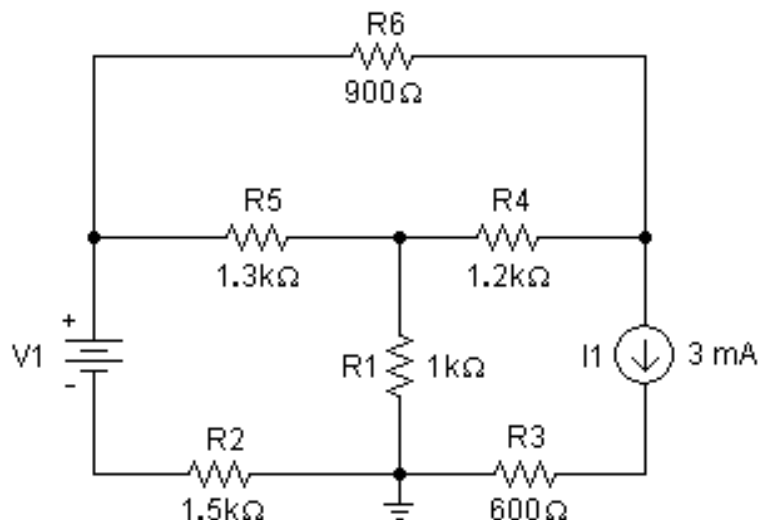
Part 1:

Write a PSpice .CIR file for the following circuit. Simulate your .CIR file using PSpice. Have PSpice display the following values: (1) the current through R1, (2) the current through R2, (3) the current through V1, and (4) the power generated by V1. Calculate all four of these values analytically and compare them to your simulation results. Also calculate the power in resistors R1 and R2 analytically and compare the sum to the power generated by V1.



Part 2:

Write a PSpice .CIR file for the following circuit. Vary V1 from 1V to 10V inclusive using a 1V increment. Simulate your .CIR file. Have PSpice display the following values: (1) the current through R6, (2) the current through R2, (3) the current through V1, (4) the voltage across R4, and (5) the voltage across R3.



Your report should include your source files (.CIR), output files (.OUT), and analytical calculations. Turn in your report at the beginning of lab next week.