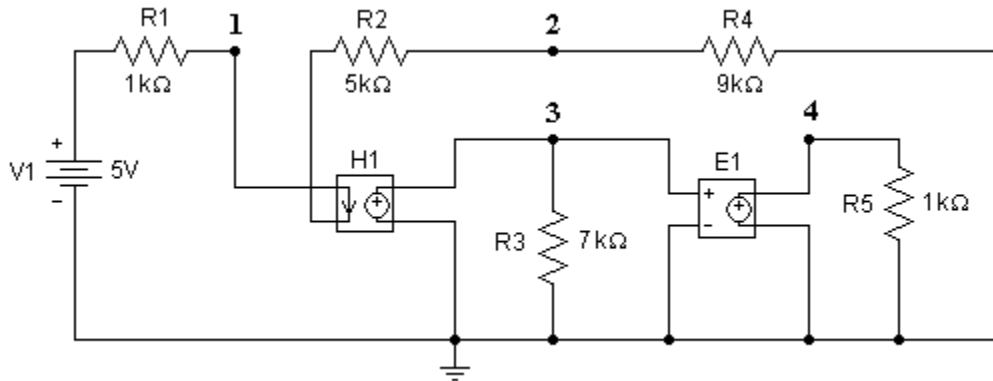


EE 206 Simulation 2

Nodal Analysis and Controlled Sources in PSpice

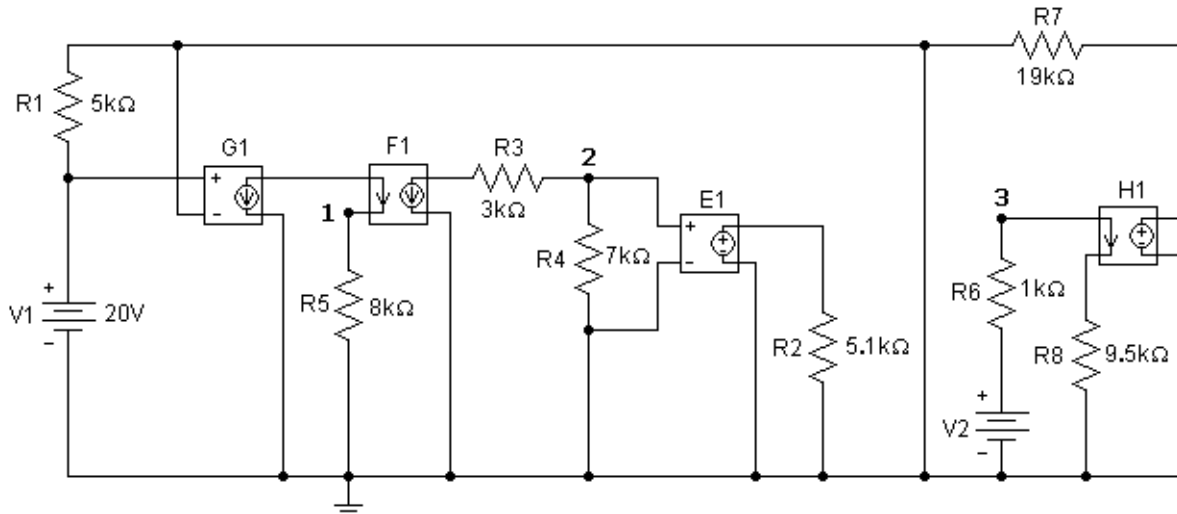
Part 1:

Write a PSpice .CIR file for the following circuit. Simulate your .CIR file. The gain of H1 is 100, and the gain of E1 is 2. Have PSpice display the following values: (1) the currents through R1, R3, and R5; and (2) the voltages at all labeled nodes with respect to the ground. Then solve the voltages of these nodes analytically and show that your solutions match the results from PSpice.



Part 2:

Write a PSpice .CIR file for the following circuit. The gains for E1, F1, G1, and H1 are 0.01, 0.25, 150, and 0.5, respectively. Simulate the circuit so that V2 has each of the following values: 10V, 15V, 20V, and 25V. Have PSpice output the following values: (1) the currents through R1, R2, R3, R6, and R7; and (2) the voltages at all labeled nodes with respect to the ground.



Your report should include the .CIR and .OUT files for both parts, the conventional schematics for both parts, and the analytical solutions for Part 1.