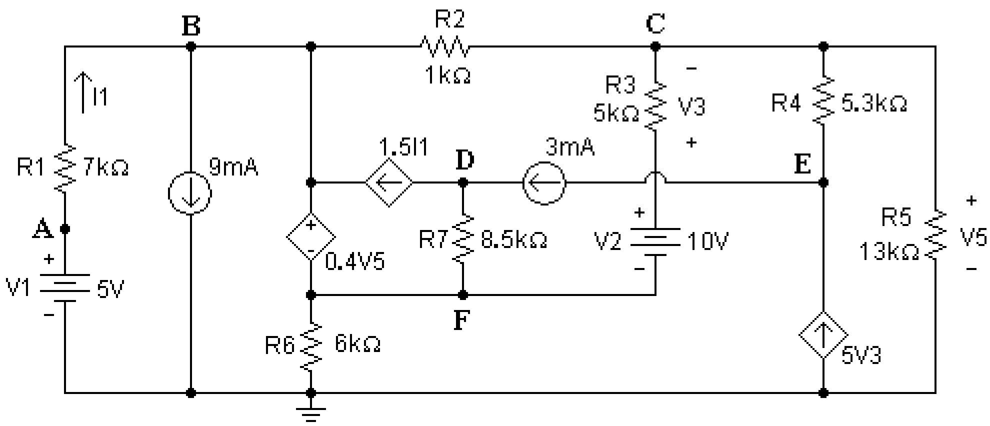
EE 206 Simulation 4

# Introduction to PSpice Schematic Capture

PSpice permits circuit schematics to be entered graphically. See the information provided on the course web site and in the text for more details.




For the circuit given above:

1. Enter the circuit using PSpice Schematics. Note that you will have to use PSpice devices E, F, and G to simulate the controlled sources. Then simulate the circuit.
   a. Have PSpice display all branch currents and node voltages on the schematic itself.
   b. Vary V2 from 10 V to 15 V using an increment of 0.1 V. Have PSpice display the following six values in six separate graphs: (1) voltage at node B, (2) voltage at node C, (3) voltage at node E, (4) current through R3, (5) current through R6, and (6) current through R7.

2. Compare the method of schematic capture with the method of writing a .cir file used in Simulation 2. Explain the "pros" and "cons" of writing a .cir file compared to editing a .sch file.

**Report**

Your report should include the printouts of steps 1a and 1b and the discussion of step 2. Your report is due at the beginning of the next lab session.