ENGR 206

Experiment #4
Circuit Analysis by Node and Loop Methods and Superposition

Objective
To use the techniques of circuit analysis – node method, loop method and superposition – and circuit simulation software (PSpice or LTSpice) to verify measured voltages and current of resistive circuits.

Introduction
The node-voltage, loop-current and superposition techniques are among the most useful methods to analyze the voltages and current in circuits. In this laboratory, we will do three things:

- **Theoretical analysis:** You will use node method, loop method and/or superposition to find the voltages and currents of the circuits.
- **PSpice simulation:** You will simulate the DC voltages and current in the circuits using PSpice.
- **Experimental measurement:** You will measure the voltages and current in the circuits and compare the measured values to the results of your theoretical calculations and PSpice simulations.

Pre-laboratory work
For each of the circuits shown in Figures 1, 2 and 3:

1. Use the node method to find all the node potentials, e_1 and/or e_2. From these values and the values of the resistors, calculate the voltage across and current through each element of the circuit.
2. Use the loop method to find all the loop currents, I_1, I_2 and/or I_3. Again, from these values and the values of the resistors, calculate the voltage across and current through each element of the circuit.
3. Create a PSpice simulation to obtain all the node potentials, as well as the branch currents using the techniques discussed in the tutorial.

In addition, for the circuit in Figure 1 only, use the superposition method to find the node potential e_1.

Note that the only difference between the two circuits in Figure 2 is the placement of the ground (COM).

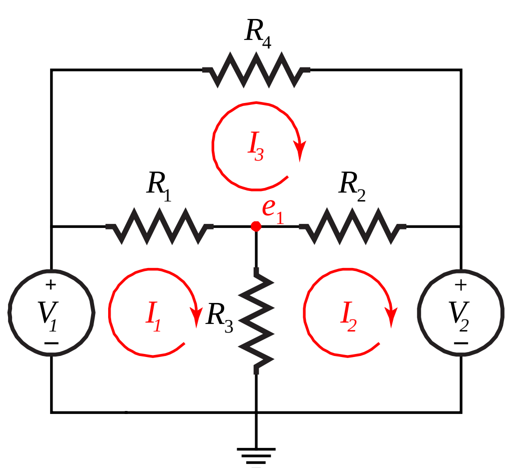

Figure 1

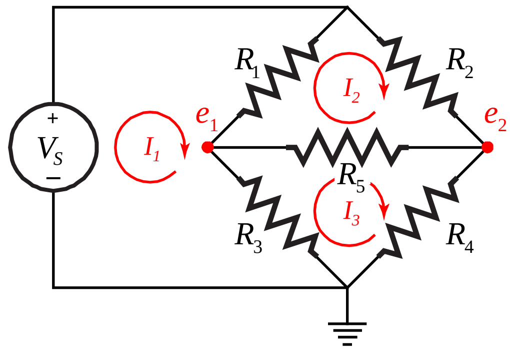

Figure 2

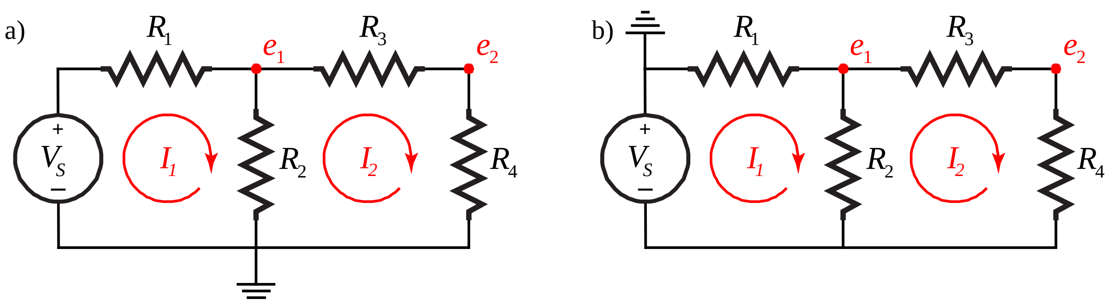

Figure 3

Laboratory work

1. The purpose of this part is to measure node potentials and loop currents and compare them to PSpice simulations. Construct the circuit shown in Figure 1. Let $V_1 = -10V$ (note the minus sign!), $V_2 = +15V$, $R_1 = 5.1k\Omega$, $R_2 = 2.2k\Omega$, $R_3 = 3.3k\Omega$ and $R_4 = 6.8k\Omega$. The two voltage sources are obtained from the triple-output power supply's -25V and +25V outputs, respectively.
   a. Measure the node potential e_1 with the DMM. From the value of node potential and the source voltages, calculate what the voltage should be across all the elements. Measure these values with the DMM and compare with the values obtained from the PSpice simulation.
   b. Measure the loop currents, I_1 , I_2 and I_3 with the DMM. From the loop currents, calculate what the current through each element should be. Measure these current values with the DMM and compare with the values obtained from the PSpice simulation. When you measure the loop currents, make sure that you break the circuit in exactly the right places. For example, when measuring I_1 , break the circuit at the top of V_1 , not at R_1 or R_3 . Why?
2. Since the circuit of Figure 1 is a linear circuit, predict (without going through circuit analysis again) the expected voltage values across the four resistors if the two source voltages are reduced to 50 % of their original values. Now, adjust both of the outputs of power supply to 1/2 their original values (i.e. $V_1 = -5V$ and $V_2 = +7.5V$). Measure and record voltages across each resistor and compare with calculated and simulated values.
3. We will now test the superposition principle using the circuit of Figure 1.
   a. Reset $V_1 = -10V$ and $V_2 = 15V$.
   b. Remove voltage source, V_1 ,from the circuit and replace it with a wire. This is equivalent to setting $V_1 = 0$ (short circuit). Measure the voltage and current through resistor R_3 .
   c. Replace voltage source, V_1 , in the circuit and remove voltage source, V_2 . Again measure the voltage and current through resistor R_3 . Does superposition of voltages and currents apply?
4. Construct the bridge circuit shown in Figure 2. Let $V_S = 10V$, $R_1 = 5.1k\Omega$, $R_2 = 3.3k\Omega$, $R_3 = 6.8k\Omega$, $R_4 = 10k\Omega$ and $R_5 = 2.2k\Omega$.
   a. Measure the node potentials e_1 and e_2 . From the value of node potentials and the source voltages, calculate what the voltage should be across all the elements. Measure these values with the DMM and compare with the values obtained from the PSpice simulation.

b. Measure the loop currents, I_1, I_2 and I_3 with the DMM. From the loop currents, calculate what the current through each element should be. Measure these current values with the DMM and compare with the values obtained from the PSpice simulation.

5. Construct the circuit shown in Figure 3a. Let $V_S = 10V$, $R_1 = 1k\Omega$, $R_2 = 2.2k\Omega$, $R_3 = 1k\Omega$ and $R_4 = 3.3k\Omega$.
   a. Measure the node potentials e_1 and e_2 . From the value of node potentials and the source voltages, calculate what the voltage should be across all the elements. Measure these values with the DMM and compare with the values obtained from the PSpice simulation.
   b. Measure the loop currents, I_1 and I_2 with the DMM. From the loop currents, calculate what the current through each element should be. Measure these current values with the DMM and compare with the values obtained from the PSpice simulation.
   c. Repeat with the circuit shown in b (note the position of the ground, COM). Are there any differences?