

EE1002 Lab1: SPICE Simulation of Resistive Circuits

I. Objectives

1. To get familiar with the procedures and some basic functions of SPICE circuit simulation;
2. To validate the Ohm's law via simple resistive circuits;
3. To explore the way for finding equivalent circuits based on SPICE simulations.

II. Equipment and Software

Networked computer, PartSim (online SPICE circuit simulator, available at <https://www.partsim.com/simulator>). PartSim is the simplest online version of SPICE that does not require any installation. You may also download some other version of SPICE such as LTSpice or PSpice which are also free for certain limitations but need some installations.

III. Background

Before conducting this experiment, you should study carefully the theoretical background involved in this experiment. You can get familiar with SPICE and PartSim through various online tutorials or even experience yourself with the software used in the experiment: PartSim, in advance.

IV. Procedures

1. In this experiment, the circuit we are going to study is shown in Fig.1. Follow the steps (2 to 5) given below to input the circuit to PartSim.
2. Pick four resistors and a DC voltage source from the "Components" menu on the left-hand side of the window. The resistor is under "Passives" and DC voltage source under "Sources". You can rotate or flip a component by placing your mouse cursor on it and right-click to pop up the edit menu. Pick "voltmeter" from "Test Equipment" menu for " V_{out} ".
3. Connect the components and voltage source by placing the cursor to one of the terminals of the resistor, voltage source or other components. Holding left button of mouse and dragging the cursor you will see a dashed green line extending from the starting point to where the mouse you moved. If you want to change the direction and bending the line, left click once and drag to your desired direction. Draw the whole circuit as shown in Fig.1. Remember to place a "ground" symbol which can be found from "Ports" under the "Components" menu.
4. Change the values and the label of the resistors in accordance to those given in Fig.1. That can be done by right-click each of the resistors and select "Edit Properties". The items you can change are displayed on the right-hand side of the window. (Note: SPICE simulator is able to recognize prefixes and units we often used. Capital and smaller letters are treated as the same; so "m" and "M" both mean "milli". For "mega", you should type "MEG" or "meg". No Greek letter; so use "U" or "u" for "micro".) You

can view the change on the schematic diagram when you editing the properties. Also change the voltage V_s to 12 V in the same way.

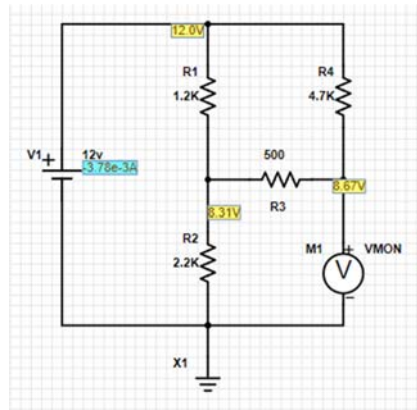


Fig.1

5. Go to “Probes” menu to add the current of all resistors [$I(R1)$, $I(R2)$, $I(R3)$, and $I(R4)$] and all net voltages so that they will be displayed on the circuit after the simulation.
6. Now, you can “Run” the simulation from the top menu bar icon under “spice” group. Choose “DC Sweep”, with V_s as DC source, filling the fields “start voltage” = 6 V, “stop voltage” = 12 V, “step voltage” = 6V, and then press “Run”. If you got error messages, check the circuit if all the connections are in good shape or not.
7. The resistor currents and net voltages should be graphically displayed on the probe panel as curves. Record, print or save the simulated results for source voltage at both 6 V and 12 V. You can save your simulation results by pressing the “triple dash” icon next to “Legend Info” near top right corner. You can also print the circuit with simulation results by using the “print” function shown on the top menu bar.
8. Now short circuit V_s by replacing V_s with a wire. Add a 5 V DC voltage source to the terminal of V_{out} . Run “DC Sweep” with V_{out} as DC source, input 0V, 5V, and 1V, respectively, for “start voltage”, “stop voltage” and “step voltage”. Record the $I(R3)$ and $I(R4)$ value at $V_{out} = 5V$.
9. Calculate the equivalent resistance value $R_{eq} = 5 \text{ V (DC voltage source)} / (I(R3) + I(R4))$.
10. Now construct a new circuit with 12 V DC source in series with “ R_{eq} ” as calculated in Step 9 and an ammeter at the output port (see Fig.2). Run “DC Sweep” simulation with “start voltage” = 6 V, “stop voltage” = 12 V, and “step voltage” = 6V. Check the output voltage and output current at $V_s = 6 \text{ V}$, and 12 V.

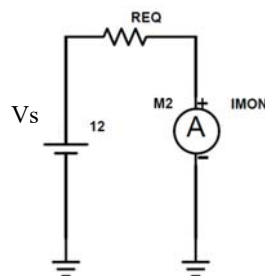


Fig.2

V. Results and Discussion

1. Compare the results given in steps 6, 7 and 10.
2. Draw an equivalent circuit for Fig.1 if V_s is in short-circuited (what we did in step 10). The circuit should become much simpler now. You should be able to express R_{eq} in terms of R_1 , R_2 , R_3 , and R_4 based on the equivalent circuit. Check if the value is same as that calculated in Step 10 using the simulated currents or not.
3. Could you explain what are the logics or principles behind the manipulation given in step 9? Are there any assumption being made for this treatment? Are there any situations the treatment cannot be applied? Can the same approach be applied to a circuit with current source?
4. In week 8 or later, you should be able to analysis Fig.1 by applying KVL or KCL for nodal or mesh analysis. By that time, check if the SPICE simulation results agree with your calculation results or not. You can always verify you calculation results with the SPICE simulation.