

Laboratory 1

Circuit Simulations in Matlab

Objectives

- Learn how to use Simscape within Matlab and Simulink environment to model and simulate circuits.
- Observe the responses of the circuits when the circuits and parameters are changed.

Equipment and components

- A desk computer
- Matlab software

Preliminary

Simscape™ provides an environment for modeling and simulating physical systems in mechanical, electrical, hydraulic, and other physical domains. It is a set of block libraries and special simulation features for modeling these physical systems. You can parameterize your models using MATLAB variables and expressions, and design control systems for your physical systems in Simulink.

Simscape provides fundamental building blocks from these domains that you can assemble into models of physical components. These blocks represent basic mathematical operations. When you connect these blocks together, the resulting diagram is equivalent to the mathematical model, or representation, of the system under design. These connection ports are nondirectional. They mimic physical connections between elements.

For circuit simulations, Simscape provides building blocks for circuit elements, such as DC/AC voltage sources, DC/AC current sources, resistors, inductors, capacitors, and switches.

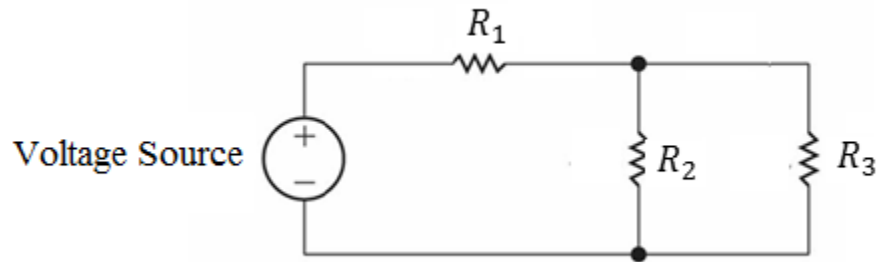
Read the “User’s Guide of Simscape” file attached with the lab if necessary.

Fill up the table below based on the given circuit and the value of the voltage source.

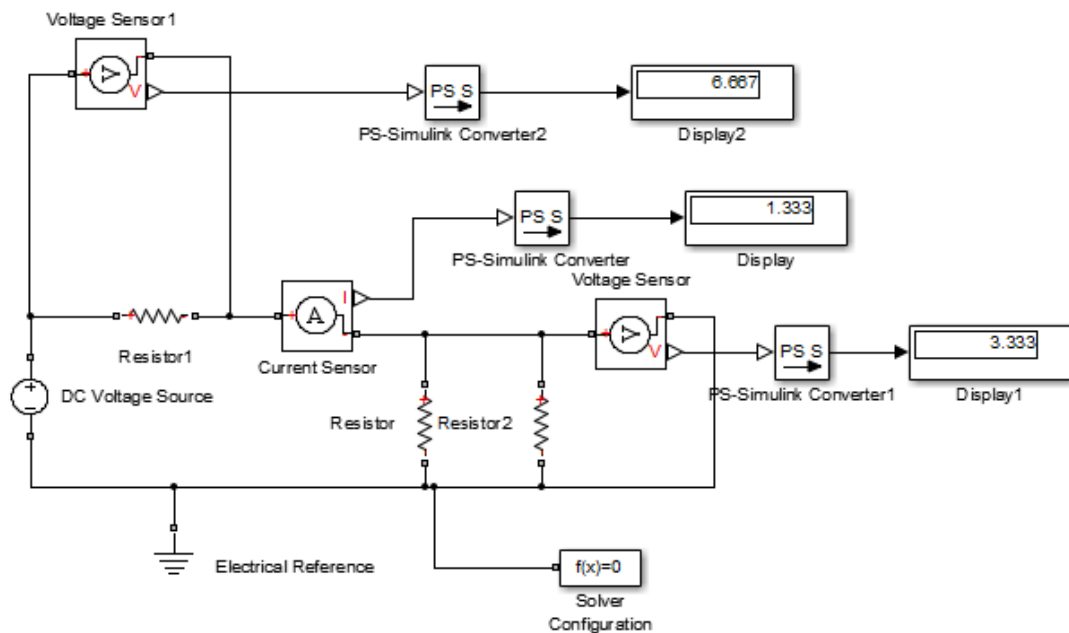
Procedure

1. Go to <https://it.ucmerced.edu/software-list/> to download Matlab. Make sure you have selected Simulink and checkmark Simscape when you install Matlab.
2. Open Matlab from the All Program.
3. Type simulink in the Matlab Command Window and Simulink Library Browser pops up.

- Expand the Simscape entry in the contents tree. The Simscape has two top-level libraries: Foundation and Utilities.
- Click Foundation library, you will see the libraries for different systems.
- Click Electrical, you can find three sub-libraries: Electrical Elements, Electrical Sensors, and Electrical Sources.
- Use the blocks provided in these three sub-libraries and the Utilities to build the following circuit.



- Add Current Sensor and Voltage Sensor into your model.
- Select your Display from the Sink under the Simulink entry. An example of the circuit model is shown below.



- Set $R_2 = R_3 = 100 \, \Omega$. Click the "Resistor2" in your model. Fill the value of 100 and unit of "ohm" in the "Resistance" window. Do the same thing for "Resistor3".
- Set the voltage source to 10 V.
- Change the resistance value of the "Resistor1" by clicking it in the model. Fill up the following table.

| R1 (Ω) | Voltage across R1 (V) | Voltage across R2 (V) | Current flowing through R1 (A) |
|-----------------|-----------------------|-----------------------|--------------------------------|
| 10 | | | |
| 20 | | | |
| 30 | | | |
| 40 | | | |
| 50 | | | |
| 60 | | | |
| 70 | | | |
| 80 | | | |
| 90 | | | |
| 100 | | | |

What do you observe from the results in the above table? For example, what is the sum of the voltages on R1 and R2? Are the currents in R1 and R2 different? Explain them by using what you have learned in the lectures.

Questions and conclusions

- Summarize your findings and explanations in this lab.