Simulating Electronic Circuits

Lab 11: Introduction to a Simulation Program with Integrated Circuit Emphasis (SPICE) and Simulation of Voltage Dividers and Diode Circuits

GOALS

Use LTspice, a SPICE program, to simulate voltage dividers and diode corcuits

Proficiency with new equipment:
- LTspice:
  - Install LTspice
  - Learn how to make a circuit, to simulate it, and to present the results

Applications:
- Simulate voltage dividers
- Simulate diode circuits

DEFINITIONS

SPICE: a general purpose open source Simulation Program with Integrated Circuit Emphasis.

LTspice: a SPCIE program developed by Linear Technology

Notice: This manual is prepared based on Windows system LTspice.

LAB PREP ACTIVITIES

Installing LTspice program: We can download LTspice simulation software V17 for Windows 7, 8, or 10 or Mac 10.9+ operating systems from the link below. Download the installation file for Windows or Mac in the link and install it.

https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html#

How to Use LTspice: The following tutorial video demonstrates basic functions of the program by simulating a voltage divider. It lasts about 12 minutes. If you have installed the program already, you can skip the installation part and watch starting from 2:30.

https://www.youtube.com/watch?v=JRcyHuyb1V0

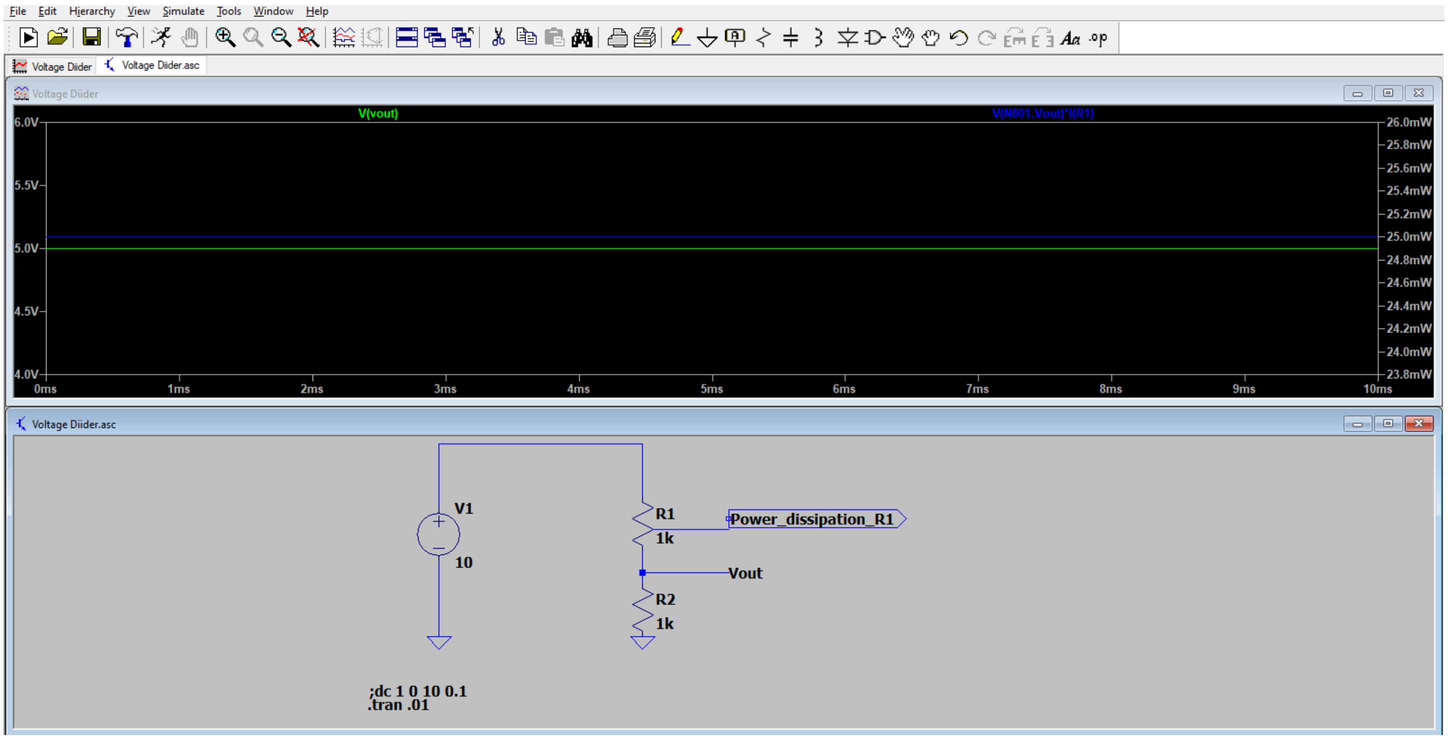
SIMULATING VOLTAGE DIVIDERS USING LTspice

Step 1. Making a circuit
1. Bring each electronic component, a power supply and two resistors from the tool bar
2. Wire them using the ‘pen’ tool on the tool bar
3. Set parameters of the components by right-clicking on each component as follows.

Case 1	Power Supply	Resistor 1 (Ω)	Resistor 2 (Ω)
	10 V, DC	1k	1k

Step 2. Running a simulation of the circuit

1. Set simulation parameters using 'Edit Simulation Command' from the 'Simulate' menu.
2. Choose Transient and set the Stop Time to 1s, see the tutorial video.
3. Run the simulation by clicking the 'running man'. Then you will see a graph window.
4. You can select a value of your interest in the circuit with the mouse. Choose the voltage.
    1) On a point of a wire for voltage
    2) On a component for current
    3) On a component with the alt key for power



Step 3. Displaying the Simulation Graph:

1. Adjust the scale of y-axes of the graph and select which data to present as shown in the video for a good presentation of the simulation. (Optional)
2. Choose power dissipated in R1. Notice that the graph show two values using the y-axes on the left for one value and the y-axes on the right for the other value.
3. You can zoom in the graph by selecting an area with the mouse (Optional)
4. You can select or deselect graphs using 'Pick Visible Traces' tool on the tool bar (Optional)

Step 4. Copying the Graph or Circuit on the Screen: Copy the bitmap of the graph or circuit to clipboard using the menu in the 'Tools'. Copy the result of the simulation showing the Vout and power at R1 and paste it on your electronic lab notebook.

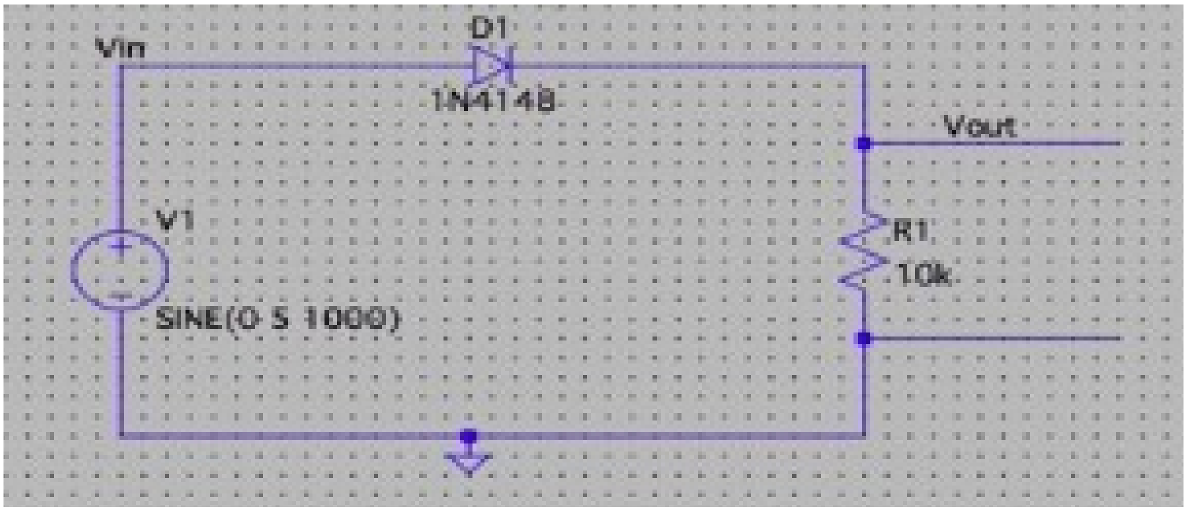
Step 5. Variation of the Simulation: Repeat Steps 1-4 for the two following cases

	Case 2	Case 3
Power Supply	10	10 Vpk-pk, 1 kHz, sine wave
Resistor 1 (Ω)	10k	10k
Resistor 2 (Ω)	90k	90k

Step 6. Save the circuit file with a good name to submit. The default extension of the saved file is "asc" when the circuit area is selected. Submit the file with the your electronic lab notebook.

SIMULATING DIODE CIRCUITS USING LTspice

Step 7. Make the following diode circuit made of a power supply, a 1N4148 diode, and R1=10kΩ using LTspice.



Step 8. Simulate Vout when Vin = 5 x sin(2π x 1000 t), or sine wave with amplitude of 5V and frequency of 1 kHz. Copy the simulated graph in your electronic lab notebook. Save the circuit file to submit.

Step 9. Repeat the steps 7 and 8 with the polarity of the diode, D1, reversed.