

**Passive Sign Convention and LTspice® Introduction**  
ECE 2100 Circuit Analysis Laboratory  
version 22 September 2021

## Equipment and Supplies

LTspice®	Available <a href="#">here</a>
----------	--------------------------------

## Learning Outcomes

Students will:

1. Use LTspice® for DC and transient simulations; and
2. Explore the passive sign convention for Ohm's Law and power computations by analyzing and simulating electrical circuits.

## Pre-Laboratory Assignment

Read the following.

Note: Typically, a laboratory safety quiz is completed prior to the first experiment. Since this experiment is simulation only, the quiz will be completed at a later date.

### *The Passive Sign Convention*

Ohm's Law states that  $V = I R$ . *This is not a complete definition of Ohm's Law since*

- voltage is ACROSS and has a POLARITY; and
- current is THROUGH and has a DIRECTION.

We must establish a directional relationship between voltage polarity and current direction. That relationship is known as the PASSIVE SIGN CONVENTION (PSC).

See Figure 1a. The PSC for the resistor stipulates that the reference direction for  $I_1$  is that  $I_1$  flows from the positive to negative voltage polarity designators for  $V_1$ . If you select a current direction, you have also selected the voltage polarity, or vice versa.  
THEY ARE NOT INDEPENDENT.

So, in Figure 1a, if we connect a voltmeter to read  $V_1$  (positive lead on top, negative lead on bottom),  $V_1 = 1V$  and by the PSC  $I_1 = V_1 / 1\text{ohm} = 1A$ .

The computation of power  $P = V I$  also follows the PSC.

The power of the resistor is  $P = V_1 * I_1 = 1V * 1A = 1W$ .

The power of the voltage source is  $P = V_1 * (-I_1) = 1V * (-1A) = -1W$ .

**NOTE THAT** to compute the power of the voltage source, if we select  $V_1$  as the voltage, we need the current flowing DOWNWARDS through the voltage source, which is  $-I_1$ .

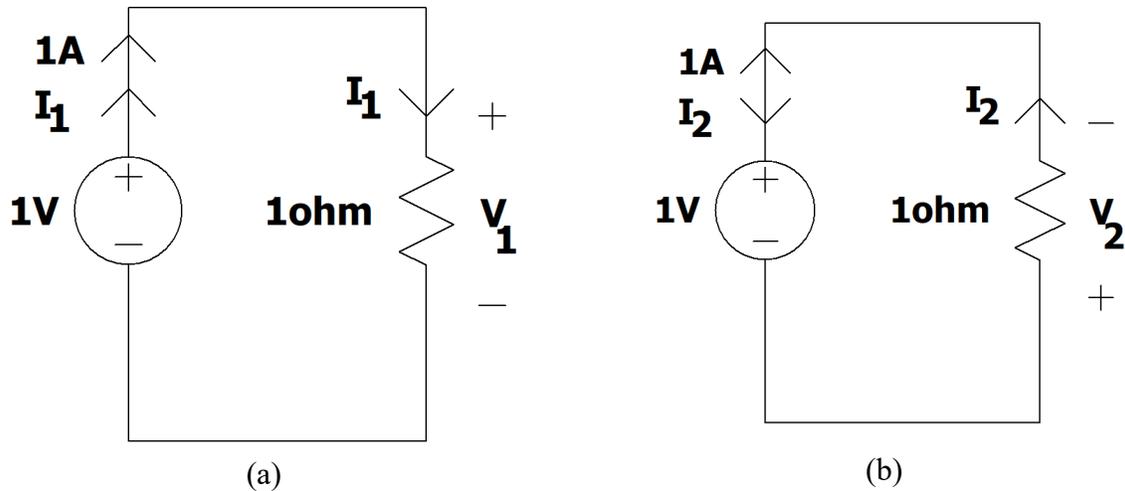


Figure 1. Circuits to illustrate passive sign convention

Now consider the circuit of Figure 1b. **This is the same circuit.** The only thing that changed is how the voltmeter was connected to measure the resistor voltage.

Now

$$I_2 = V_2/1\text{ohm} = (-1\text{V})/1\text{ohm} = -1\text{A}$$

$$\text{The power of the resistor is } P = V_2 * I_2 = (-1\text{V}) * (-1\text{A}) = 1\text{W}$$

$$\text{The power of the voltage source is } P = (V_2) * (-I_2) = (-1\text{V}) * (1\text{A}) = -1\text{W}$$

Note that  $I_2 = -I_1$ ;  $V_2 = -V_1$ ; and the powers are the same in both cases (a) and (b) and add to zero.

## Procedure

### Lab Notebook

1. As always, carefully document your work in your lab notebook. Begin each lab on a new page. Include the date and title of the experiment.

Use a pen for all notebook entries. If you make an error, simply cross out the mistake, but do not make the mistake illegible. Often times what was thought to be an error turns out to be correct! Such notebook procedures may be required for your lab work as an engineer, particularly for intellectual property documentation.

### LTspice® Setup

The next two steps eliminate the need for color printing and make your report easier to view. These steps are required before including LTspice® schematics and results in your lab report. One simple way to use copy schematics and waveforms from LTspice® to another application is using Tools->Copy bitmap to Clipboard. There are ways to obtain better quality images, such as printing to a PDF printer.

2. Open LTspice(®). Using Tools->Color Preferences->Schematic set the color of all items to black (RGB values to 0), except the background, which is set to white (RGB values to 255).
3. Using Tools->Color Preferences->Waveform make all trace colors and labels black.

### ***Schematic Drawing***

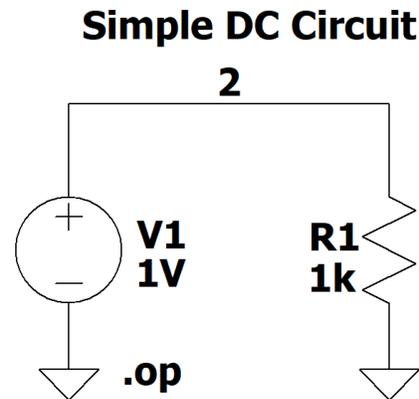
4. Create a new schematic using File->New Schematic.
5. Draw the circuit of Figure 2.
  - Components are selected using the gate symbol in the toolbar. The gate symbol is just to the left of a 'hand'. Click on the gate symbol and enter 're' in the box. A resistor symbol will appear. Click 'OK' and place the resistor on your schematic.

#### **Use CTRL-R to rotate the resistor during placement.**

The name of the resistor is R1. Its value is R. Right click on 'R' and set the resistor value to  $1k\Omega$  by typing '1k'.

- Use the leftmost 'hand' symbol to select and move the resistor around. Try it. You can also reposition component names and values.
- Now place a voltage source on your schematic by using the gate symbol and typing 'vo'. Set its value to '1V' by clicking on the 'V'.
- Use the wiring tool (the 'pencil' symbol) to connect the components as shown in Figure 1. Simply click where the wire starts, click for intermediate corners, then click at its endpoint.
- LTspice® REQUIRES a reference node. Use the ground symbol (by the 'pencil') to add connections to a reference node as shown. This means that the 'bottom' of V1 and R1 are connected together. To improve spacing use a wire to connect the 'ground' to the components as shown. Component labels and values can be moved using the 'hand'.

Figure 2. Simple DC circuit



- Try View->Zoom->Zoom to Fit. You may then want to Zoom->Zoom Back.
- SAVE YOUR SCHEMATIC using File->Save As. Use 'ECE2100Lab1Fig2.asc' as the filename. **You should frequently save your work.**
- The circuit has only two nodes. The 'bottom' node is the reference node. LTspice® measures all voltages with respect to the reference node. Let's label the top node as node 2; otherwise, LTspice® will name the node for us. The node labeling tool is to the right of the ground symbol. Select that symbol, enter '2', and position the '2' on top of the wire to label that as node 2.
- Add a title to your schematic. Use the 'Aa' symbol. Be sure to make the text a Comment and **NOT** a SPICE directive.

### *DC Operating Point Simulation*

6. We will now perform a DC (direct current or constant) Operating Point Simulation. LTspice® computes voltages and currents at DC and prints them to a file.
  - Use Simulate->Edit Simulation Cmd. Select 'DC op pnt' and click OK. Place the resulting '.op' directive on the schematic.
  - To run the simulation, press the 'running person' symbol next to the 'hammer' symbol. If all is well, a window will pop up:

```
--- Operating Point ---  
V(2) :      1          voltage  
I(R1) :     -0.001     device_current  
I(V1) :     -0.001     device_current
```

Figure 2. DC operating point simulation result

This shows that:

- The voltage from node 2 to the reference node is 1V. This voltage is ACROSS

node 2 and 'ground'.

- The current THROUGH V1 is labeled  $I(V1)=-1\text{mA}$ . **LTspice® follows the passive sign convention; the reference direction for  $I(V1)$  is from the positive terminal of V1 to the negative terminal of V1.**
- It is not easy to tell the reference direction for  $I(R1)$ . But here we can see that current  $I(R1)$  must be in the same direction as current  $I(V1)$  since they have the same sign. If your  $I(R1)$  current is positive, that means that the reference direction for  $I(R1)$  was assigned in the opposite direction of  $I(V1)$ . To see this, select the resistor with the hand tool, rotate it, and redo the simulation. The current  $I(R1)$  will now be negative. We can use a 0V voltage source to eliminate this potential confusion as described next.

Look at View->netlist and see the effect of rotating the resistor. Do you see the difference before and after rotating the resistor? How does LTspice® assign the current direction? The netlist is what LTspice® uses to simulate the circuit. In the 'old' days you did not draw the circuit, you just typed in the netlist!

### ***Add an Ammeter to Measure Current***

7. Next a 0V voltage source will be added IN SERIES with R1, see Figure 3.

This will simplify measurement of the current  $I_R$  through R1. Of course, we know this current already, but this is a very useful 'trick' in LTspice®. Based on the orientation of V2,  $I(V2)$  is  $I_R$ .

**The direction of  $I_R$  was selected based on the polarity of  $V_R$  to follow the PSC for R1.** Add the 0V source as shown in Figure 3. *Since V2 is a short, it has no effect on the circuit.* Also label node 1. Rerun the simulation. Figure 3 also shows the result.

Note:

- Node voltage  $V(1)$  is zero since V2 is a short to ground. There is no voltage across voltage source V2.
- Since  $V(2)$  and  $V(1)$  are node voltages,  $V_R=V(2)-V(1)=1-0=1\text{V}$ .
- $V(2)$ ,  $I(R1)$ , and  $I(V1)$  are the same as before V2 was added.
- $I(V1)$  is  $-1\text{mA}$ . **WHY?**

8. Be sure to save your schematic.

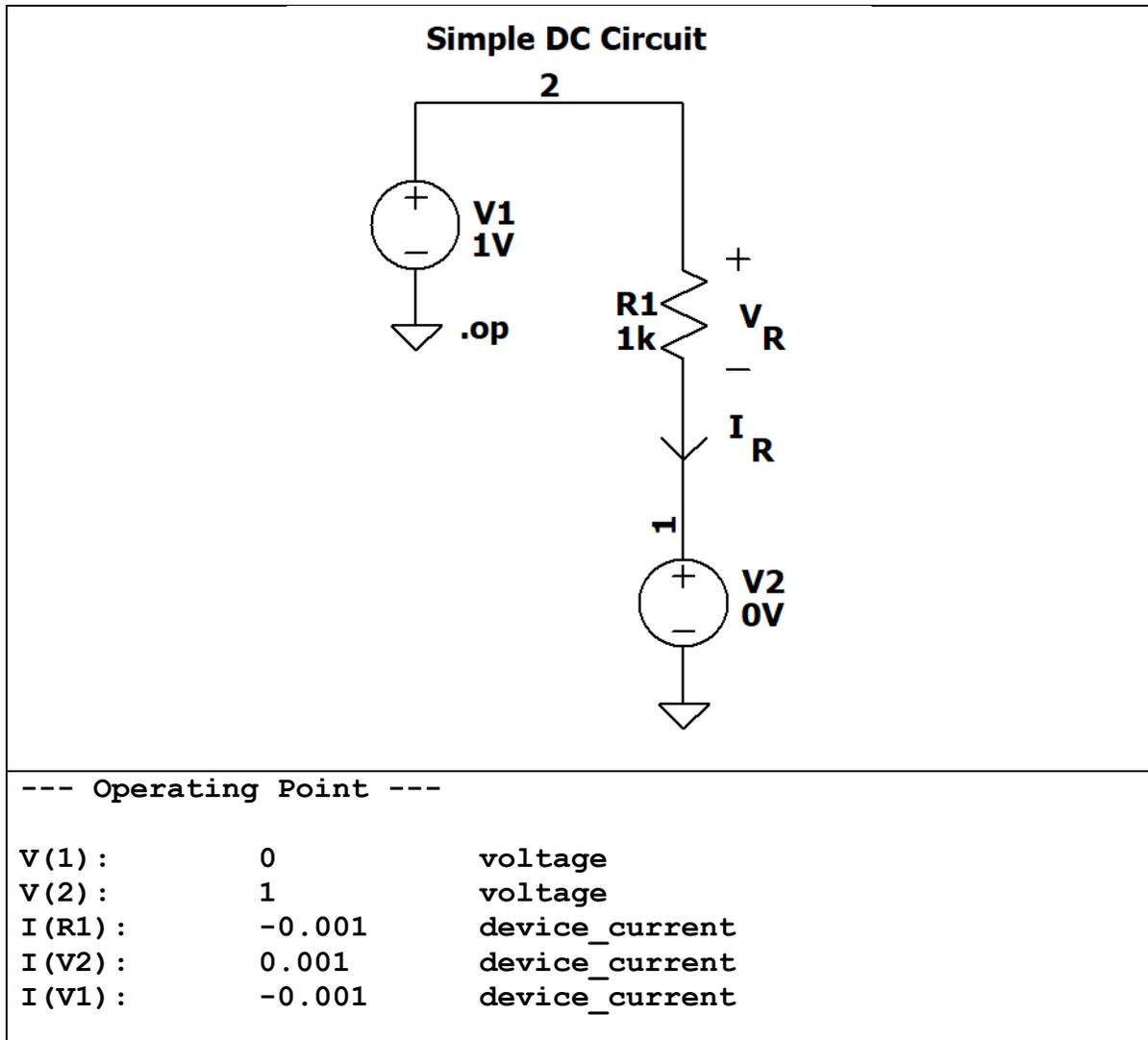


Figure 3. Circuit simulation result after adding ammeter

### *Transient Simulation*

9. A DC operating point analysis is sufficient for a DC circuit. By using a transient analysis, we can plot voltages, currents, and other useful quantities as a function of time.
  - Save your schematic under a different name. Right click on the .op statement and change the analysis type to 'Transient' with a stop time of 1s. Run the simulation. A waveform window will pop up.
  - Right click in the waveform window and select Add Traces. Click on V(2). This will plot the voltage of node 2 (measured with respect to ground). Of course, the graph is a constant 1V.
  - Add a trace for I(V2). Plot the power of R1 as follows. Right click in the waveform viewer. Select Add Traces. In the Expressions box add 'I(V2)\*(V(2)-V(1))'. This is

the current through R1 times the voltage across R1, of course following the PSC. The power is 1mW.

10. Right click on V1 the click on 'Advanced'. Change V1 to a SINE wave

(click on the SINE button):

DC offset (V): 0V

Amplitude (V): 1V

Freq (Hz): 1 (this means one cycle per second).

Click OK.

See Figure 4.

Rerun the simulation.

- Node voltage  $V(2)$  now is sometimes positive and sometimes negative, so  $i_R(t) = v_R(t)/(1k)$  is sometimes positive and sometimes negative as well. In electrical engineering we use small variables to denote AC (alternating current or non-constant) quantities.
- The power of R is ALWAYS positive since a resistor can only absorb power. Note the power is at a different frequency than the source. **WHY?**
- To measure the power at any time, right click on the power trace label at the top of the graph and set 'Attached Cursor' to '1<sup>st</sup>' then click OK. A window will appear. By moving the cursor you can find the power at any time t.

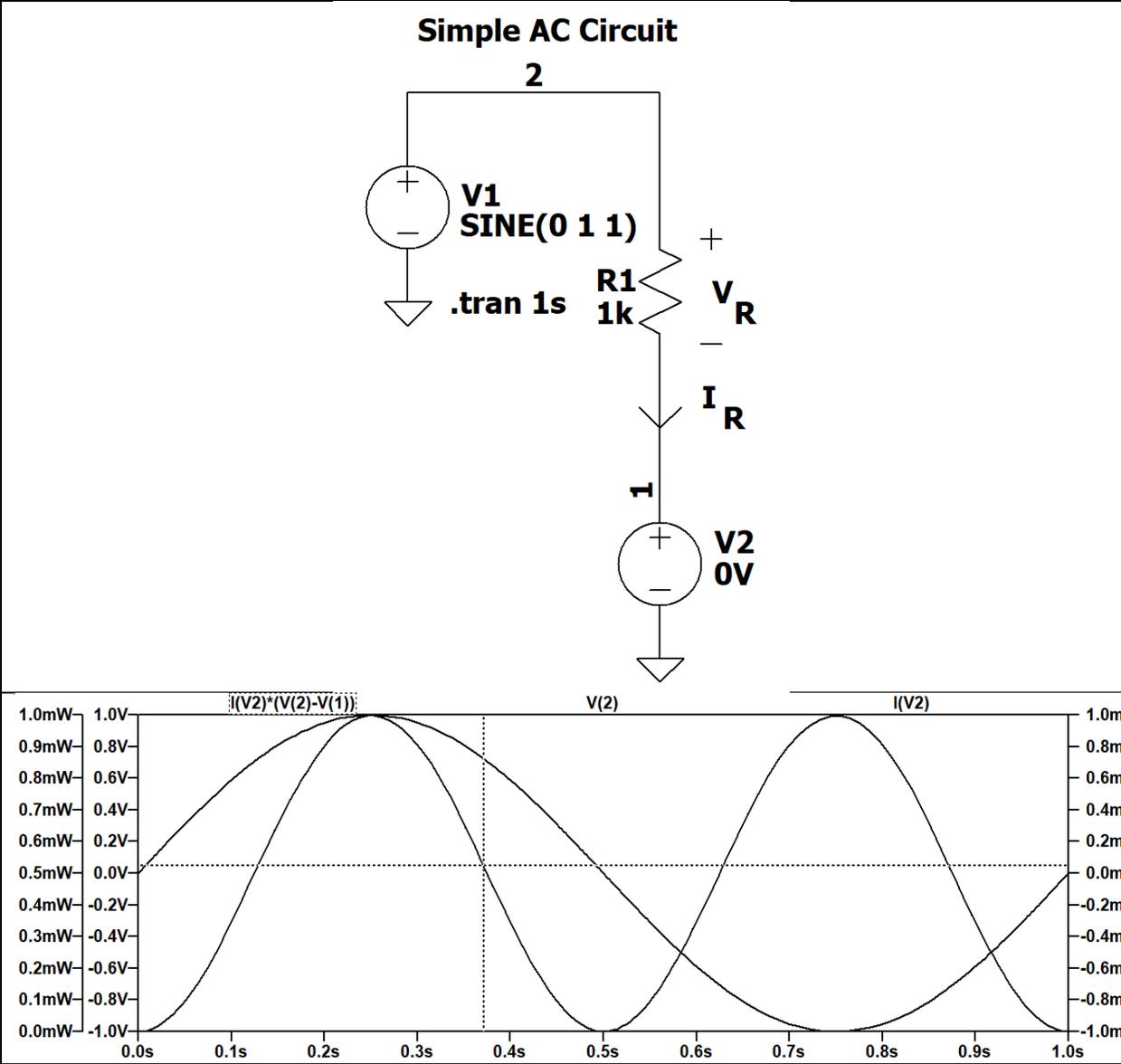


Figure 4. Simple AC circuit

*A More Complex Circuit*

11. Draw the circuit of Figure 5 in LTspice®. Save using a new file name.

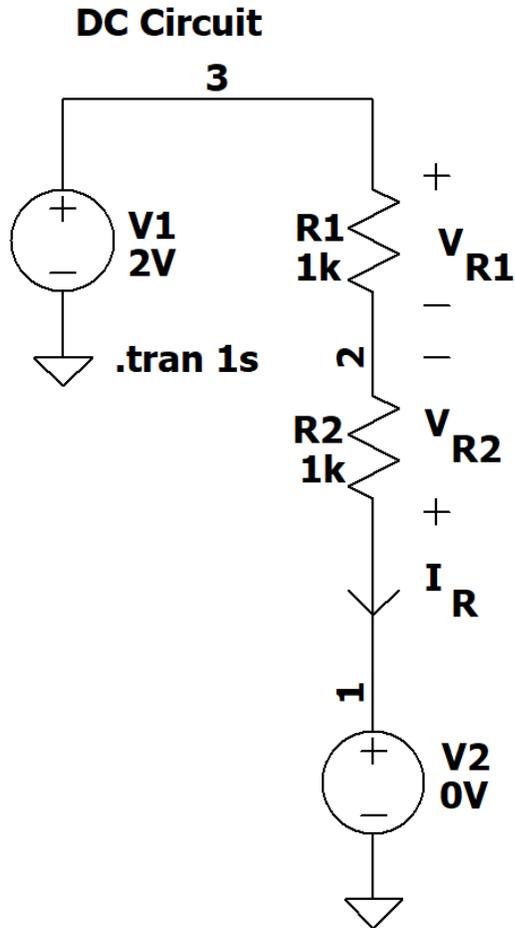


Figure 5. A more complex circuit

Compute the indicated quantities and fill in Table 1.  
Then simulate the circuit in LTspice® and record simulated values.  
Be sure to use proper units. Resolve any discrepancies. Be careful with  $V_{R2}$ !

Quantity	Computed Value	Simulated Value
V(3)		
V(2)		
V(1)		
V <sub>R1</sub>		
V <sub>R2</sub>		
I <sub>R</sub>		
P <sub>R1</sub>		
P <sub>R2</sub>		
P <sub>V1</sub>		

Table 1. Analysis and simulation results for the circuit of Figure 5.

### Analysis

None

### Credits and Copyright

Simin Masihi, Masoud Panahi, and Mohammad El Yabroudi provided helpful feedback that was incorporated into this lab.

© 2021 Damon A. Miller. All rights reserved. Not for use outside of ECE 2100 without explicit written permission.