

## LABORATORY 1 – CIRCUIT SIMULATION WITH SPICE

### OUTLINE

#### 1 Introduction

- Drawing a Circuit Schematic
- Running a Simulation
- Analyzing Results

#### 2 Pre-Lab Assignment

- Lamp Circuit

#### 3 In-Lab Work

- Dependent Source Circuit
- Lamp Circuit
- LED Circuit

#### 4 Post-Lab Assignment

- Dependent Source Circuit
- Lamp Circuit
- LED Circuit

### 1. INTRODUCTION

LTSpice is a circuit simulation package that is based on SPICE (Simulation Program with Integrated Circuit Emphasis). The software includes a graphical interface that can be used in Windows and Mac OS X, is freeware, and is available on CAEN computers. LTSpice is available at the link below, which also includes documentation on a ‘Getting Started Guide’ and shortcuts for Mac OS X:

<http://www.linear.com/designtools/software/#LTspice>

**Mac OS X users:** the user interface is not as intuitive as the Windows version, so you will need to become familiar with shortcuts, and a method to ‘right click’. Right click can usually be accomplished using (Ctrl)+click with a mouse, or two-finger click with a track pad.

Let us get started by analyzing the basic circuit in Figure 1, consisting of a voltage source, resistor, and a lamp (light bulb that has a circuit equivalent of a resistor). The objective is to find the current through the lamp  $I_{out}$  and the voltage across the lamp  $V_{out}$ .

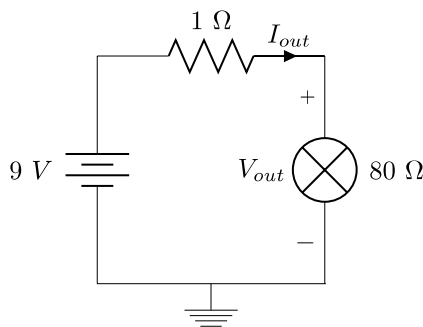


Figure 1: Circuit with a resistor and lamp.

### Drawing circuit schematic

Begin by creating a new schematic (New -> Schematic). Add components and wire them together using the procedure below.

- Add new components using Draft-> Component. For this example, we will choose voltage source (voltage) and resistors (res). Use edit tools to move or delete components, and the shortcut Ctrl+R to rotate objects.
- Add a reference ground node, using Draft -> Net Name. Select GND (global node 0) and place.
- Connect the components with wires using Draft -> Wires.
- Define component values by right clicking on each component and changing the value.

Your circuit should then look very similar to Figure 2.

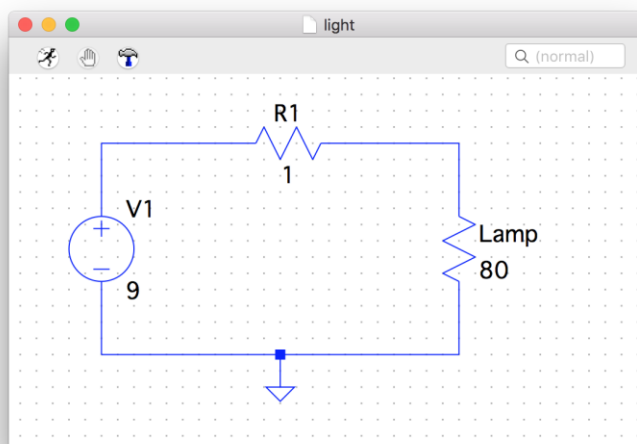


Figure 2: Circuit schematic of lamp circuit drawn in LTSpice.

### Running a simulation

To run a simulation, click Draft -> SPICE Directive. In the window, type the command '.op', which refers to a DC operating point of the circuit, and place the text anywhere on the schematic. There are six different types of analyses that can be performed, each with a different analysis command. The syntax for each may be found in the user manual (and will be given to you for cases used in EECS 215).

.tran	Transient analysis
.ac	Small signal AC
.dc	DC sweep
.noise	Noise
.tf	DC transfer function
.op	DC operating point

Now run the simulation by clicking 'Run'!

## Analyzing results

Following a successful simulation, you will see a new window of Waveform Data. Resulting values from the simulation will be shown here, but now you need to specify the variables that you are interested in viewing. There are at least three methods of specifying output, and displaying either in the Waveform Data window or directly on the circuit schematic:

- On the circuit schematic, use Draft -> .op Data Label. Place the label on a node of interest, and the DC operating point will appear. Note that this only works for node voltage, not current. Example results for data labels are also shown in the schematic above.
- On the Waveform Data window, click on 'Add Trace(s)'. Select the variables of interest to show on the Waveform Data. Moving the cursor on the circuit schematic will also show a 'voltage probe' when the cursor is on a node, or a 'current probe' when the cursor is on a circuit element. Clicking on the desired value will add to the display in the Waveform Data window. Note that the x-axis of the Waveform Data window is time or frequency, and will be much more 'interesting' for transient and AC analysis.
- In the log file (shortcut Command+L), all node voltages and element currents should be listed.

The resulting output using these three methods are shown in Figure 3, where we find  $I_{out} = 0.1111$  A and  $V_{out} = 8.8889$  V.

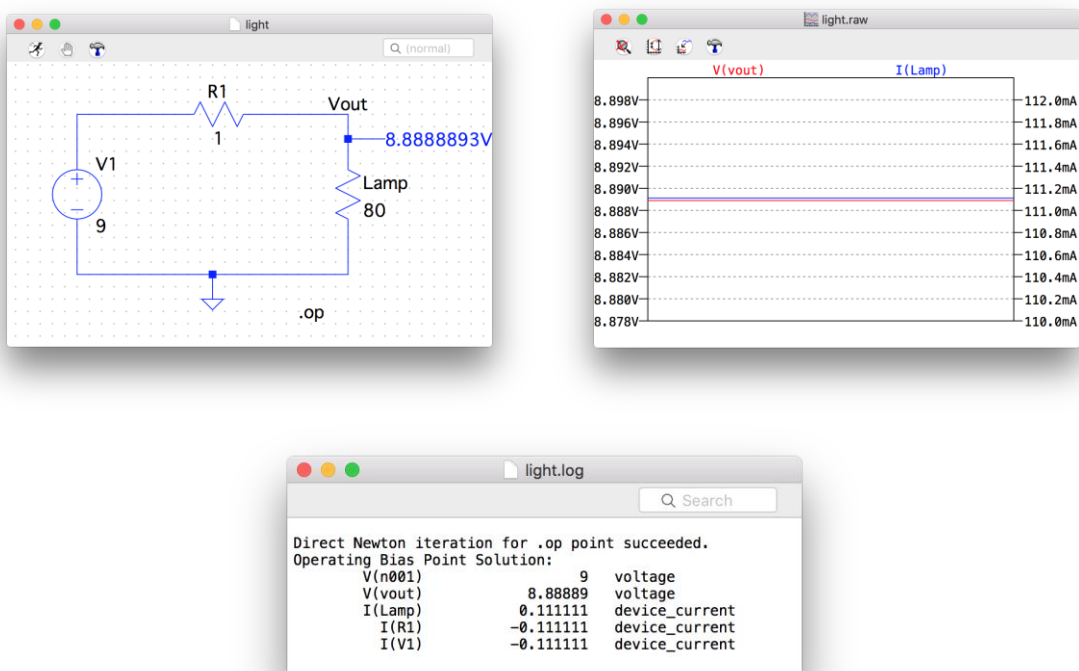


Figure 3: LTSpice output for the lamp circuit.

## 2. PRE-LAB ASSIGNMENT

Simulate the circuit in Figure 1 and obtain the output for the three graphs in Figure 3.

### 3. IN-LAB WORK

#### Dependent Source Circuit

Now let us look at a more complicated circuit that includes a dependent source, as shown in Figure 4. For the dependent source, we need to insert a function that references another value on the circuit. Use the component `bv`, and note that a right click will show a screen with Spice Model Value  $V=F(\dots)$ . We need to change the value to the desired function. Let us choose the current through the battery labeled `V1`. We can then change the `bv` Spice Model Value to  $V=-500*i(V1)$ . The label on the schematic will not automatically change, so you will need to also edit the label.

\* Note that LTspice assumes a current direction flowing into the positive terminal of an element (same as passive sign convention). In the case of  $i(V1)$ , the reference current  $i$  in the figure above is the opposite direction of the current flowing into the positive terminal of `V1`.

Set up a DC operating point simulation to determine all node voltages in the circuit.

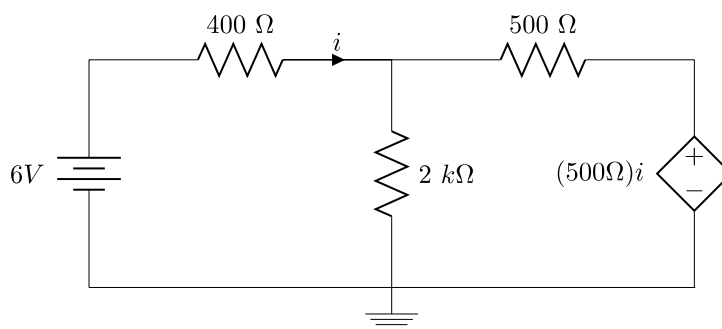


Figure 4: Circuit with a dependent source.

#### Lamp Circuit

Now let us return to the basic lamp circuit of Figure 1. From our DC operating point simulation, we find that the current through the circuit is 111.1 mA. Suppose that the lamp manufacturer recommends that current should be maintained at 100 mA since this is where the lamp was designed for safe operation, optimal performance, and long lifetime. If we continue to use a 9 V source, we can reduce the current flow by changing the resistance of the 1  $\Omega$  resistor. A resistor used to reduce current or voltage in a circuit is often called a **ballast resistor**. We can use SPICE to sweep the resistor value to determine what is needed to provide a current of 100 mA.

- Change the resistance value to the text `{ballast}`. This will set the resistance value to a variable named `ballast`.
- Define a new SPICE directive to step through the parameter `ballast`. Use the command `.step param ballast 1 20 0.5`, where the syntax is `.step param variable start end stepsize`. The command therefore steps through a resistance from 1  $\Omega$  to 20  $\Omega$  in steps of 0.5  $\Omega$ .
- Run the simulation and add the trace `I (Lamp)` in the waveform window. You should then see a plot of `I (Lamp)` versus the value of `R1`, as shown in Figure 5.

From the simulation, we should see that a resistance of 10  $\Omega$  will limit current to 100 mA. Of course by inspection, we know  $I=V/R$ , and  $100 \text{ mA} = (9 \text{ V})/(10 \Omega + 80 \Omega)$

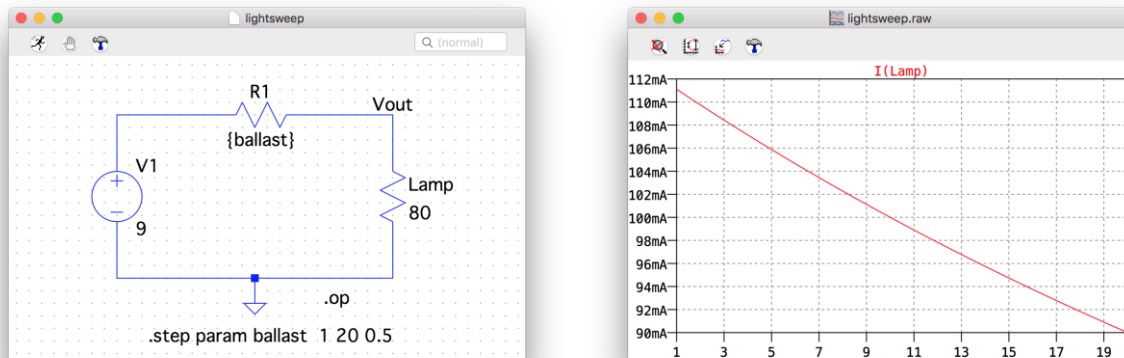


Figure 5: Circuit with lamp and stepped resistor values and corresponding simulation output.

### LED Circuit

Cases like the lamp circuit above could be calculated simply by hand, or perhaps with a calculator. However, many circuits have much more complex connections, and potentially components that are non-linear such as a diode. Let us examine the circuit in Figure 6 that substitutes the lamp for an LED.

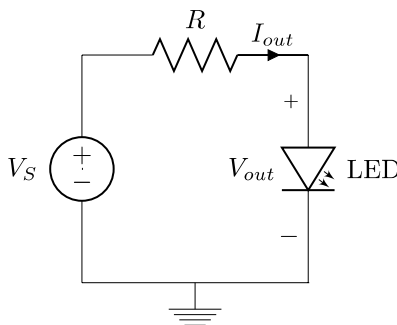


Figure 6: Circuit with a resistor and LED.

- Draw the circuit above, using the component LED.
- There are many types of light emitting diodes, with widely varying performance characteristics. SPICE has many built-in commercial components, as well as the ability to define new components or revise models for existing components. For this example, right-click on the LED and choose Pick New Diode to select the diode model LXHL-BW02. This is a highly efficient white light LED used for a variety of applications.
- Define the resistance  $R$  as  $0.1\ \Omega$ .
- Similar to the parameter sweep for the lamp circuit, perform a DC sweep that varies the voltage source. Use the SPICE directive `.dc v1 0 4 0.01`, which sweeps the voltage source labeled `v1` from 0 V to 4 V in steps of 0.01 V.
- Plot the current through the LED in the waveform window, and change the scale of the current axis to values between 0-0.5 A.

The resulting circuit and output should look similar to Figure 7.

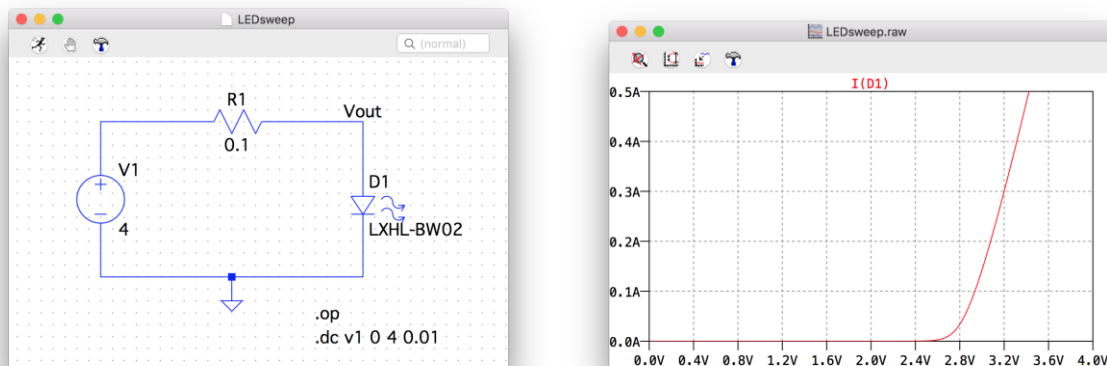


Figure 7: Circuit with lamp and stepped resistor values and corresponding simulation output.

Note that the resistance is very small for R1, and that current rises exponentially with increasing source voltage V1. The LED is clearly nonlinear, and if you are not careful, too much current will flow through the LED, burning out your LED or voltage source! A ballast resistor can be used to protect your circuit. Properly design an LED circuit (same circuit, determine value of ballast resistor) to achieve the following:

- 9 V power supply
- Operating current of 50 mA.
- Resistance between 1  $\Omega$  and 1 k $\Omega$ .
- Plot Current versus value of ballast resistor. Use a logarithmic scale for both current and resistance axes. In defining parameter sweeps, you can use `.step dec param variable start end steps_per_decade`, where the inclusion of `dec` defines the parameter steps logarithmically in the defined number of steps per decade.
- Plot the power consumed by the LED, power consumed by the resistor, and power supplied by the battery versus resistance (on log-log scale), all on the same plot. Note that when you add traces to the waveform window, you can enter algebraic expressions to plot in terms of current and/or voltage.

#### 4. POST-LAB ASSIGNMENT

##### Dependent Source Circuit

1. Submit circuit schematic and output results.
2. Calculate the node voltages using nodal analysis and compare results.

##### Lamp Circuit

1. Submit circuit schematic and output results.

##### LED Circuit

1. Submit circuit schematic and output results.