

Lab 7: Simulations and IV Curves

Prelab

You’ve been using EveryCircuit for simple simulations, but it has significant limitations in regards to circuit size and realistic component behavior. The industry standard for circuit simulation is SPICE, and the various incarnations thereof. You’ll be using a free version provided by Linear Technologies, LTSpice.

At least one parter needs to install LTSpice on their laptop. The interface for the Windows version is distinctly better than the Mac version. You can install the windows version on Linux using WINE.

Part I: Using LTSpice

1.1 Tutorial

SPICE (Simulation Program with Integrated Circuit Emphasis) is a general-purpose, open source analog electronic circuit simulator. It is a program used in integrated circuit and board-level design to check the integrity of circuit designs and to predict circuit behavior.¹ It has its roots in a class project and grew into Laurence W. Nagel’s PhD project at Berkeley. It was announced to the world in 1973 and quickly grew in popularity.

Many derivatives of SPICE have been created which use the same data structures and commands, one of the most widely used is the version created at Linear Technologies, LTSpice. LTSpice has both a graphical front end (which you will be using) and a text based interface in which circuit “netlists” can be defined as a list of components connected to numbered “nodes.”

This is an example of how a circuit can be represented. This circuit has one dc voltage source and three resistors.

```
v1 1 0 dc 24
r1 1 2 10k
r2 2 0 8.1k
r3 2 0 4.7k
.end
```

There are text based commands called “SPICE directives” which are used to control the parameters of the simulation. Much of the functionality is replicated in the graphical interface, but some isn’t.

Unfortunately the graphical interface is not very user friendly. Figure 1 shows part of the menu with labels for the various buttons. Here is a list of some of the more unintuitive aspects which

¹Wikipedia

may help you navigate the software. When you aren't sure about something, glance back over this list.

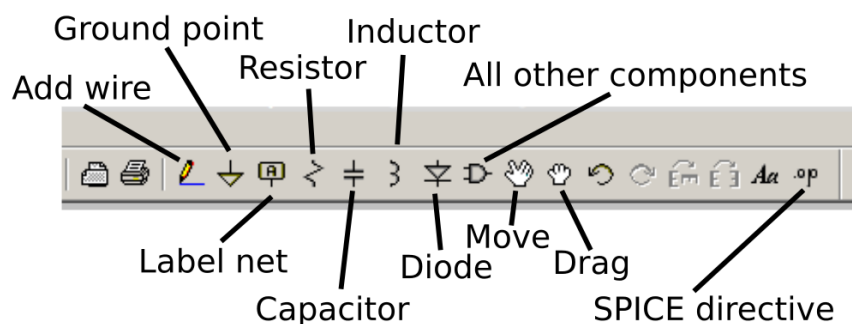


Figure 1: LTSpice menu bar

- The general way to interact with a command is to click the icon, then click a location on the screen or click a component. Hit escape to exit out of a command mode. Clicking and dragging mostly isn't a thing. The faster you starting thinking "click, move mouse, click again, hit escape" as your general work flow, the better.
- Create an empty document using File → New Schematic
- All circuits MUST have a ground point
- "Nets" are the same thing as nodes (constant voltage regions where components connect together). You can label these to help keep measurements organized.
- "Move" will allow you to move a single component while breaking its connections. Also use this mode to rotate components.
- "Drag" allows you to move components while keeping the wires attached.
- Right clicking on components brings up edit menus. Note that right clicking on the component image brings up a different menu than right clicking on the text.
- To set up an analysis, click the running person icon and fill in the relevant values. This creates a text SPICE directive which you must place anywhere on the page. Right clicking this allows you to edit the simulation parameters.
- Once you place the directive text box somewhere, clicking "Run" again will run the analysis and pop up a window showing an empty graph.
- Now hovering the mouse over wires or components will show a probe icon. Click on wires to plot voltage (relative to ground) and on components to plot current. Click and hold on one side of a component and drag to the other side to measure the voltage across a component.
- Left clicking on the trace label (Voltage or Current) gives access to a cursor which can be dragged along the trace, and a window showing the value at the cursor.
- Right clicking on the trace label lets you edit properties of the trace, and delete the trace.
- To avoid printing out plots with black backgrounds, go to Tools → Color Preferences. Select "Background" from the drop down menu and change the RGB values all to 255.

1.2 Practice

Create the circuit shown in figure 2. **You and your partner must take turns using the software.**

Be sure you've read through all of the previous section. Mouse over the remaining icons to see the pop up text describing what they do. Use "Cut" to delete any objects or wires you want to remove.

Be sure to label the input and output nodes/nets to help keep your graph organized.

Click the Run icon and set the stop time to 1 sec, the time to start saving data to zero seconds, and the max time step to 0.1 sec.

Graph the input and output voltage, as well as the current through each resistor. (Enable a grid through the Plot Settings menu, which will be available after clicking on the plot.) *Do the values agree with what you expect from Labs 1 and 2? Include the circuit and graph in your report.*

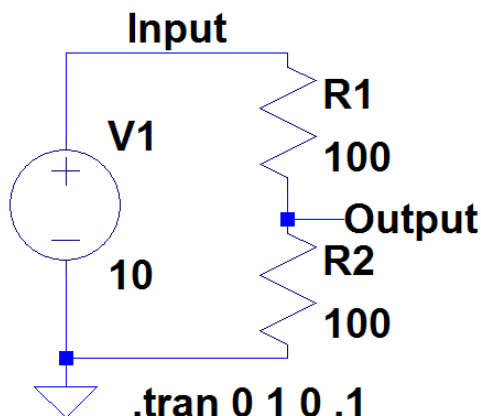


Figure 2: Voltage Divider

Part II: Simulating Filters

2.1 Transient Analysis

Construct the filter circuit shown in figure 3. To obtain a sinusoidal voltage source, right click the voltage source and go to "Advanced". Under Functions, select SINE. Set the amplitude to 10 volts and the frequency to the cutoff frequency of the filter.

Set the transient analysis (time domain) to run for 4 milliseconds, to start saving data at 1 ms, and to have a max time step of 0.1 ms. (Enter 4 milliseconds as "4m".)

Plot the input and output voltages.

What do you expect the max output to be?

Find the max output on the plot using the cursors and compare.

Now you'll find the max value using the MEASURE directive. Click the SPICE directive icon and enter the following into the text box:

```
.meas TRAN maxVal MAX V(Output)
```

This will measure the maximum value of the output voltage during a transient analysis, and assign the result the name maxVal.

Run the analysis again.

For some reason, the result is stored in the error log. Navigate to View → SPICE Error Log (or use the shortcut CTRL+L). *What value do you get from this method and how does it compare?*

Include a screen shot of the graph in your report.

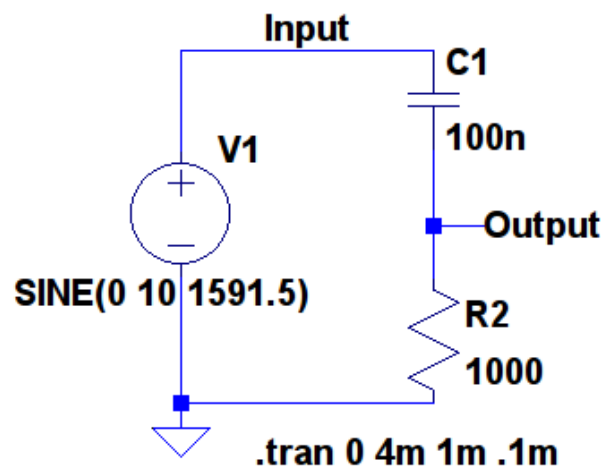


Figure 3: High Pass Filter

2.2 Bode Plots

To create a plot of attenuation and phase vs frequency, you must change the type of analysis and the signal source.

- Change the voltage source function to “None” and set the “Small Signal AC Analysis” amplitude to 1 volt.
- Edit the simulation direction and change the type to “AC Analysis”
 - Set the type of sweep to octave
 - Number of points per octave: 20
 - Start freq: 10
 - Stop freq: 100k

Run the simulation and confirm that the phase angle is 45 degrees at the cutoff frequency.

Use the MEASURE directive to check the cutoff frequency. Add a new SPICE directive and enter in the following:

```
.meas AC Fcut when mag(V(output))=1/sqrt(2)
```

What frequency does the simulation find to be the cutoff? How does that compare to your calculation?

Include a screen shot of the circuit and of the Bode plot in your report.

Part III: Diode IV Curves

Current vs voltage plots are a common method of characterizing a component. For a Ohmic component (like a resistor) the plot is just a straight line where the slope is the inverse of the resistance (since usually current is plotted on the vertical axis).

For a diode the curve is rather more interesting.

3.1 Circuit Construction

Look at the datasheet (on Moodle) for the 1N4148 diode. *What is the maximum continuous current rating?* **Set the current limit on your DC power supply to that value.**

Set the power supply to zero volts. Using both multimeters, one in voltage mode and one in current mode, construct the circuit shown in figure 4.

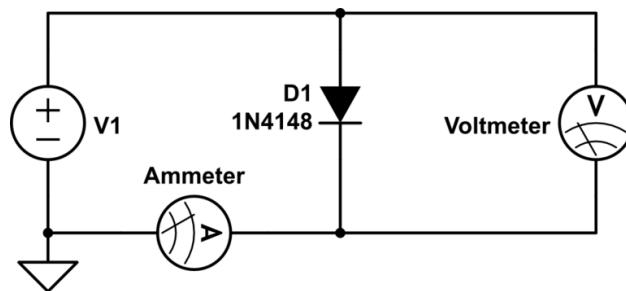


Figure 4: Measure the voltage and current relationship of a diode

Collect voltage and current data points every 0.05 volts starting at $V = 0.3$ volts until you hit the current limit. Create three plots of current vs voltage as described below.

- The first plot should have linear axes (I vs V).
- The second plot should look just like the IV curve plot in the diode datasheet.
- The third plot should be like the first, but only include the data points in the linear region (after the ‘knee’).

From your real life data, what slope do you obtain for plot 3? What “resistance” does this correspond to? If the diode could handle the it, what would be the voltage drop with 1 amp of current?

3.2 Circuit Simulation

One of the very powerful aspects of LTSpice is its vast library of specific components. Once you have created the diode, right click the symbol and choose the 1N4148.

Instead of a DC or AC voltage source, you are going to use a “pulsed” voltage source. In the voltage source menu, choose PULSE and set it to start at zero volts and rise to 2 volts over the span of 1 second.

Set the simulation parameters to also run for 1 second with steps of 0.01 seconds.

Run the simulation and graph the current through the diode. Right click on the time axis and change it to plot $v(\text{out})$. Resize the plot window to be a square and change the vertical axis to logarithmic (right click on it). Set the min and max values on the vertical axis to match those in the data sheet.

What current does the simulation show at 0.8 volts? What current does the data sheet show (note the weird divisions on the log scale)? What current did you get in your actual circuit?

To make the simulation more accurate, companies provide text files with component models. A model for the 1N4148 diode from NXP is available for you on Moodle.

- Download the file 1N4148_NXP.model and open it in any text editor.
- While holding down CTRL, right click on the diode symbol and change the value of the value field (no that wasn't a typo) to “1N4148NXP” and click Okay.
- Create a new SPICE directive. Copy and paste the context of the text file into the directive's text field. Click Okay and then click anywhere on the diagram to place the text.
- Run the simulation again. (You may need readjust the plot axes.)

What current do you now get at 0.8 volts? How does that compare to your previous result?

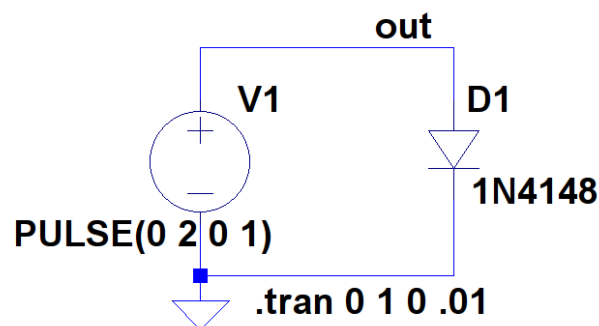


Figure 5: Simulate the voltage and current relationship of a diode