

Using LTSPICE to Analyze Circuits

Overview:

LTSPICE is circuit simulation software that automatically constructs circuit equations using circuit element models (built in or downloadable). In its modern form the software uses a graphical user interface to enter the circuit (the jargon is a schematic capture tool) which it then translates into equations. The software then simulates the behavior of the circuit by integrating the equations forward in time. Finally it provides an interface for measuring currents, voltage, amplitudes, etc. Other SPICEs exist and have differing strengths.

Using the lab computers

Logging in


Each electronics lab computer has a local account that you should use

Login: zm****/electronics

Password: student


You will need to replace the *s in zm**** using the rest of the number on the PROPERTY OF SUNY CORTLAND sticker on the *computer tower*.


Running LTSpice

The LTSpice program  is in the bar at the bottom of the screen.

Circuit schematic entry

You will enter your circuit's schematic using menus to choose circuit elements from. Some common elements (wire, ground, resistor, capacitor, inductor, etc.) are on the bar at the top, a portion of which appears in Fig. 1. Others such as

Transistors (nnp, pnp, MOSFETs), Op Amps, voltage sources, current sources, etc. are available under the  symbol (an AND gate).

Symbols that have not yet been placed can be rotated  and/or

transposed  using buttons.

Once you place the circuit elements assign their values (e.g., resistance, capacitance, rating, precision) by right clicking on the element. For place value SPICE understands SI prefixes with some modification

mega=1E6=10⁶

k=10³

m=10⁻³

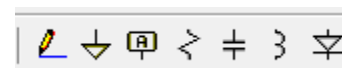



Figure 1 LTSpice menu bar, the symbols are wire, ground, label, resistor, capacitor, inductor, and diode respectively.

$u=10^{-6}$


The simulation is a computer code and as with all code is clearer if you provide some explanation in the form of comments. Comments are added to your schematic using the  tool.

Simulation selection

With a schematic drawn it is time to configure the simulation you wish to run simulation. The simulation command does this configuration. Under the Simulation menu by select Edit Simulation Command and select Transient analysis, this is the most common simulation we will do. While there are many values that you could set, the minimum that you must set are

- Stop Time: the duration of the simulation and
- Time to Start Saving Data: which specifies how much of the transient response to throw away.

The program composes a .tran command and clicking OK allows you to place it in the schematic window.

You are now ready to run the simulation. Do this by clicking on the icon of the running person .

Accessing the output

Once LTSpice has finished the simulation a blank graph will pop-up ready to display traces of voltage and current. Hover the cursor over a wire or terminal and the program shows a voltage probe icon. Hover over a circuit element (e.g. a resistor) or a tap for an element and the program shows a current clamp (with an arrow specifying the assumed direction of the current). Press alt over an element and a temperature gauge appears, click with this gauge to display the power dissipated by the circuit element. In all cases a single click adds the trace to the graph and a double click makes that the only trace on the graph.

Once you have added a trace displayed you can manipulate it.

- To focus on a particular region of the trace by highlighting the region you are interested in (click and drag a box around the part of the trace you wish to zoom in on. Zoom back out to view everything with 'ctrl e'.
- To find the numerical value of a particular moment, left clicking on a trace's name to brings up a cursor and a dialog box with the numerical value of the trace at the moment selected. Click and drag the center of the cursor to change the moment with information displayed.
- To do algebra on the trace displayed or change the color of the trace, right clicking on the trace's name.
- To find integrals (e.g., average voltage, RMS voltage, integrated power) of the trace control left click on the trace. Change the limits of integration by changing the region in view (see above).

Today's tasks

Task 1

By Hand:

In your lab groups sketch on piece of paper the schematic for voltage divider with:

- One 10V DC voltage source
- One 1kOhm resistor
- One 2kOhm resistor

Be explicit about where you have placed your ground.

By hand calculate the voltage across each element, the current through each element and the power dissipated by each element.

Spice:

Construct an LTSPICE schematic for the circuit you have drawn.

Run a simulation for 10s collecting data for the last 9 seconds.

Use the software to display traces of the voltage across, current through, and power dissipated by each element. Compare them to the results of your hand calculation.

Task 2

By Hand:

Sketch on piece of paper the schematic for voltage divider with:

- 1 8V, 1kHz AC voltage source
- 1 1kOhm resistor
- 1 3kOhm resistor

Be explicit about where you have placed your ground.

By hand calculate the voltage across each element, the current through each element and the power dissipated by each element.

Spice:

Construct an LTSPICE schematic for the circuit you have drawn.

Run a simulation for 0.52s collecting data for the last 0.02 seconds.

Use the software to display traces of the voltage across, current through, and power dissipated by each resistor and find their RMS values. Compare the LTSpice results to those from your hand calculations.

Task 3

By Hand:

Construct an LTSPICE schematic placing the following three things in series

- 1 3V, 1kHz AC voltage source
- 1 1kOhm resistor
- 1 2 μ F capacitor.

Situate your ground at the negative end of the voltage source.

By hand calculate the voltage across each element, the current through each element and the power dissipated by each element.

Spice:

Run a simulation for 0.52s collecting data for the last 0.02 seconds.

Use the software to display traces of the voltage across, current through, and power dissipated by each resistor and find their RMS values. Compare the LTSpice results to those from your hand calculations.

Make two sketches

1. voltage across the resistor and current through the resistor as a function of time
2. voltage across the capacitor and current through the capacitor as a function of time

Question:

Explain what the difference is between the two sketches you just made that causes the large difference in power dissipated by the resistor vs the capacitor.

Results

Task 1

Hand sketch of schematic

	Hand calculation result	LTSpice result
V_{R1}		
V_{R2}		
I_{R1}		
I_{R2}		
P_{R1}		
P_{R2}		
P_{source}		

Task 2

Hand sketch of schematic

(RMS voltages and currents)	Hand calculation result	LTSpice result
V_{R1}		
V_{R2}		
I_{R1}		
$\langle P_{R1} \rangle$		
$\langle P_{R2} \rangle$		

Task 3

Hand sketch of schematic

(RMS voltages and currents)	Hand calculation result	LTSpice result
V_R		
V_C		
I_R		
$\langle P_R \rangle$		
$\langle P_c \rangle$		
$\langle P_{source} \rangle$		

Sketch of V&I vs t for R

Sketch of V&I vs t for C