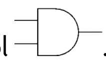
Using LTSPICE to Analyze Circuits

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 constructs those equations using a graphical user interface (the jargon is a schematic capture tool). Other SPICEs exist and have differing strengths.

You will construct your circuits on a schematic using menus to choose circuit elements from. Some common elements (wire resistor, capacitor, inductor, ground etc.) are on the bar at the top, others such as Transistors (npn, pnp, MOSFETs), Op Amps, voltage sources, current sources, etc. are available under the AND gate symbol . Symbols that have not yet been placed can be rotated and transposed using 'control r' and 'control e' respectively.

Once you place the circuit elements you give them 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=10^6

k=10^3

m=10^{-3}

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 bar Aa tool.

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.

Running the simulation pops up a blank graph where you can 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).

The tasks for today.

Task 1

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.

Construct an LTSPICE schematic for the circuit you have drawn.

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

Graph traces of the voltage across each resistor and the current through each resistor. Calculate the voltage across each resistor the current through each resistor by hand and compare them to the LTSpice results. Display the power dissipated by each resistor and compare it to a calculation by hand.

Task 2

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

- One 3V, 1kHz AC voltage source
- One 1kOhm resistor
- One 2kOhm resistor

Be explicit about where you have placed your ground.

Construct an LTSPICE schematic for the circuit you have drawn.

Run a simulation for 1.02s collecting data for the last 0.02 seconds.

Graph traces of the voltage across each resistor and the current through each resistor. Calculate the RMS voltage across each resistor the RMS current through each resistor by hand and compare them to the LTSpice results.