Lab Supplement

Circuit simulation using SPICE

SPICE is a Simulation Program with Integrated Circuit Emphasis. It is a set of algorithms for simulating circuit behavior from a list of nodes, circuit connections, and component properties. Although the SPICE engine is in the public domain, most people use it with a proprietary user interface in products such as OrCAD, TINA, 5Spice, WinSpice, HSpice, or PSpice. The separate products MultiSim and ISIS perform many of the same functions.

Spice is very convenient for varying circuit parameters and component values. It is widely used and models exist for a staggering number of components and circuits. The numerical models are sophisticated – for example, most include the effect of temperature variation – but they cannot replicate every aspect of a physical circuit. Computer simulations do not eliminate the need to test electronic prototypes, but they can reduce the number of prototypes that need to be tested.

This tutorial will guide you through the simulation of a few circuits with active components (transistors and op-amps) using PSPICE 9.1. Unfortunately, this version of PSPICE does not include Zener diodes, so it is difficult to model the resistance-controlled LED driver circuit. You can, however, model the Zener diode as a series of regular diodes (forward biased), assuming 0.6 V drop each.

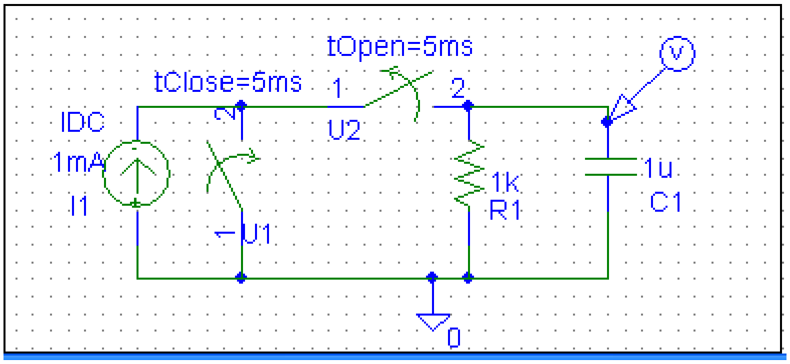
The first step in a circuit simulation is to draw the circuit in *PSPICE Schematics*, which is a GUI for the simulation engine. Simulations can then be set up and run for a variety of input signals. Current and voltage sources can be set up to provide DC, AC, or transient inputs, and switches are available if step changes are desired. There are also “sweep” functions that vary the input voltage over some specified range; we will use this function today. Later in the quarter we will use a related frequency sweep function to model the response of RLC circuits to sinusoidal inputs. The voltage or current at any node in the circuit can be recorded and plotted against time, input voltage, or input frequency.

Part 1: Draw and set up a practice circuit

1. Open *PSPICE > Schematics* from the Programs menu. A grid of dots should appear. The grid can be modified via *Options>Display Options*.

2. There are many built-in components in PSPICE. To find one for the first time, select *Draw>Get New Part…* or click the binoculars button on the toolbar. Select the part you want, click *Place part*, and click on the schematic where you want the part to be. Click again to place another copy of the same part, or right click to stop placing them. To rotate the part 90°, type ctrl-R. Once you have placed a part, it will join your short list on the toolbar.

Try drawing the following circuit schematic, which should provide a first-order natural response. You will need to get and place parts IDC, r, C, Sw_tClose, and Sw_tOpen. The wiring tool can be found through *Draw>>Wire* or ctrl-W. The value of the current source can be set by double clicking on the displayed value. Place a voltage probe at the upper terminal of the capacitor to observe the capacitor voltage (*Markers>Mark voltage/level*).



3. Once the circuit is complete, save the schematic and set up an analysis. Select *Setup…* from the *Analysis* menu, and enable *Transient*. Open the *Transient…* menu, and enter these values: Name=I1, Final time = 10ms.

4. Now you are ready to start the simulation. On the Analysis menu, select *Simulate*. A new window should appear that will show a graph with the voltage plotted against time. This graph should show that the voltage stays steady until 5 ms, after which it decays in a familiar way.

To view the voltage at another location, click "Add Trace" on the Trace menu. For example, the labels V[R1:1] and V[R1:2] mean the first and second nodes of resistor R1. In this case V[R1:1] would be the same as V[C1:1], but in most circuits there are a variety of voltages of interest. Observe and save the voltage trace that you see. The image can be copied by selecting *Window>>Copy to clipboard…* from the menu bar, and can be inverted for printing with a white background.

Part 2: Simulate the circuit you are going to build

1. If you are building a resistance-controlled circuit such as an LED driver with a transistor, Zener diode and potentiometer, it is useful to vary the resistance of the potentiometer using a parametric sweep. This is a little tricky because it requires the creation of a global variable and – in my experience at least – a concurrent voltage sweep at a single voltage value in order to get the simulator to store the data properly. If you are interested in this setup please the instructors know and we will talk you through it in lab.

Numerical notation

* SPICE recognizes most standard engineering notation multipliers, but the notation is not case sensitive.

T = 1E+12	G = 1E+9	Meg = 1E+6
K = 1E+3	f = 1E-15	p = 1E-12
n = 1E-9	u = 1E-6	m = 1E-3

* There can be no space between the value and the multiplier. For example, "1.00 uF" is considered to be 1 Farad.

* Do not confuse Farads with fempto. Example: "1.0 F" on a capacitor means 1 Farad while "1.0F" means 1 femto Farad (1E-15).

* Do not confuse milli with meg. In SPICE, the "m" or "M" multiplier means milli (10^{-3}). "Meg" is the proper multiplier meaning 1E+6. For example, 1MHz in SPICE means 1 millihertz.

"VSIN": sinusoidal source

"r": resistor

"c": capacitor

"GND_analog": ground

"Q2N3904" or "Q2N2222": npn transistor, similar to the 2N5769 / 2N5770

"555D": LM555 integrated timer circuit.

References

PSpice tutorials:

http://homepages.which.net/~paul.hills/Circuits/Spice/SpiceBody.html

http://dave.uta.edu/dillon/pspice/pspice01.htm; includes abbreviations for powers of 10

Download sites:

PSpice 9.1 (http://www.electronics-lab.com/downloads/schematic/013/)

5Spice (http://www.5spice.com)

WinSpice http://www2.eng.cam.ac.uk/~dmh/ptialcd/spice/spice.htm)

Orcad, the most popular system that contains PSPICE:
http://www.cadence.com/products/si_pk_bd/downloads/orcad_demo/index.aspx

MultiSim, an alternative to SPICE:
http://www.electronicsworkbench.com/products/proprod_dl.html

Component models:

Fairchild 2N5769: http://www.fairchildsemi.com/models/#PSPICE