ROSE-HULMAN INSTITUTE OF TECHNOLOGY
Department of Electrical and Computer Engineering

EC 300 Signals and Systems | Fall 2000
B.A. Black

P-Spice Familiarization

by Bruce A. Black after a suggestion by Wayne Padgett

Objectives

PSpice is a circuit simulation package that is widely used in industry, particularly for designing circuits containing electronic devices. In this lab project we will become familiar with the AC (sinusoidal steady state) and transient analysis capabilities of PSpice. We will do this by modeling some passive and op-amp-based filter circuits.

Pre-Lab

PSpice can be obtained from the RHASP server. Before coming to lab, download PSpice and install it on your laptop computer. (You may already have PSpice installed from another course.) Record the date of your installation in your lab notebook.

The schematic diagram for the Orange Filter can be obtained from the instructions for the "Frequency Response of a Filter" lab. Prepare a simulation model of the Orange Filter using the MicroSim Schematics Editor. Use a "VAC" voltage source for the input, and be sure to include the 50 Ω load resistor. Print out a copy of the schematic and tape it in your lab notebook. You will do the actual frequency response simulation during lab. Hand in a photocopy of your pre-lab at the start of the class before lab.

Procedure

PSpice Demonstraton

At the start of lab, the instructor will demonstrate the following PSpice operations:

1. Setting up the Orange Filter model for frequency response analysis.
2. Using Probe to plot insertion loss vs. frequency.
3. Using the cursors in Probe to determine the 3 dB frequency.
4. Setting up a simulation of an active filter using an op-amp model from the PSpice parts library.
5. Using a voltage-controlled voltage source (VCVS) to produce an "ideal" op-amp model.
6. Comparing the frequency responses generated by the two models.
7. Setting up the active filter models for transient analysis.
8. Comparing the transient analysis results from the op-amp model circuit and the VCVS circuit.
9. Using multiple AC sources to represent harmonics.

Do the Following

1. Using your model of the Orange Filter, generate a plot of the insertion loss vs. frequency. Print out the plot and tape it in your lab notebook.
2. Redesign the example circuit shown in Eccles, Fig. 11.5-9 so that the low-frequency gain is 20 dB. (This only involves changing the gain of the final non-inverting amplifier stage.) Simulate the circuit using the µA741 op-amp model from the PSpice parts library. Find the frequency response and plot it in the format of Fig. 11.5-10. Print out your schematic, and also print out the frequency response plot. Tape the printouts in your lab notebook.
3. Replace the µA741 op-amp model with VCVS models and repeat step 2. Comment in your lab notebook on whether the frequency response is any different.
4. The *impulse response* of a circuit is the response of a circuit to a unit impulse $\delta(t)$. The impulse response can completely characterize the behavior of a linear time-invariant circuit. In this step you will use PSpice to find the impulse response of the circuit you set up in step 3. An actual unit impulse has an infinite voltage value for zero time, and cannot be generated even by a simulator. We will see that a short enough pulse will do the trick, however.
    - Begin by replacing the input voltage source by a VPULSE source. Set the pulse parameters so that V1=0 (pulse "off" value), TD=0 (time delay), TR=1ns (rise time), TF=1ns (fall time), PER=10ms (period).
    - Start by setting the pulse "on" value to V1=5000 and the pulse width to PW=200 µs. This will give the pulse a unit area.
    - Run the transient analysis with a Final Time of 200 µs and a Print Step of 1µs. Use Probe to observe the output voltage.
    - Repeat the transient analysis with V1=10,000 and PW=100 µs. Continue to repeat making the pulse higher and narrower (keeping the area equal to unity) until the output ceases to change significantly. Record the height and width of your "impulse" in your notebook. Print out the Probe plot of your impulse response, and tape the plot in your notebook.
    - Now use the same "impulse" to find the response of the circuit from step 2. Is the result the same? Explain briefly in your notebook.

Report

Record the results of all of your work in one partner's lab notebook. Tape any printout (graphs, for example) into the notebook as specified in the Lab Manual for the course. Be sure that all members of your lab group sign the lab notebook, and hand the notebook in at the end of lab.