LABORATORY 3 – PSpice Circuit Simulations

A. Lab Goals

The main purpose of the lab is to learn how to use the software tools Schematic Capture, PSpice, and Probe to model analog circuit performance both to design better circuits and to analyze the performance of existing circuits.

B. Background Reading

Read Appendix C in the lab manual. Also, look up some information on Cadence PSpice either on-line or from a book. Possibilities include the Cadence website, Wikipedia, etc. Finally, read and complete the Pre-lab BEFORE going to lab. You will need to turn it in to the LA before the lab in order to complete the lab.

C. Definitions

Probe – is a waveform viewer and data analyzer that displays the output values from PSpice in appropriate graphical forms.

PSpice – (Personal Simulation Program with Integrated Circuit Emphasis). PSpice is a mixed signal (analog and digital) circuit simulation package that runs on personal computers. PSpice has been available for over 25 years and has gone through many versions. PSpice can perform many types of analyses, including DC, AC (sinusoidal) steady state and transient analyses. It can also perform parametric analyses and Monte-Carlo simulations. Circuit components can be linear or nonlinear; ideal or non-ideal.

Schematic Capture – a front end program to PSpice in which one can draw a schematic diagram of a circuit and set up the type and parameters of a PSpice simulation. The program outputs text files that are read into PSpice to control the circuit analyses.

D. Laboratory Equipment

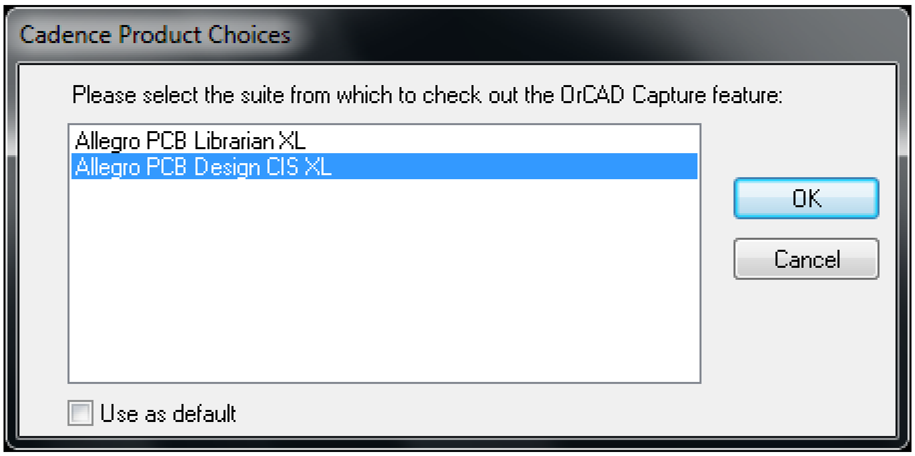
The only pieces of equipment you will use during this lab is a computer (and possibly a printer).

E. New Hardware

There is no hardware to be used in this lab, as it is purely a simulation lab.

F. Helpful Hints

1. You will have to perform PSpice simulations yourself, for both pre- and post-lab analyses in future labs, so make sure that you know where you can have reliable access to PSpice and related codes, and make sure you know how to run the code and debug circuit schematics by yourself.
2. You may run into slightly different versions and settings for PSpice on different computers. The main version on the ECE computers is version 17.2. When you open PSpice in the 205 Lab room, you get an additional dialog box asking you to choose between Cadence products – choose Allegro PCB Design CIS XL.



3. The most useful libraries are listed below:

ABM	BREAKOUT	EVALP	SPECIAL
ANALOG	EVAL	SOURCE	
ANALOG_P	EVALAA	SOURCESTM	

The full version has many more libraries, spread out over several directories. When you go to open a new library, you should see a directory called “PSpice.” These standard libraries are in there, along with dozens of other libraries.

4. The delete button gets rid of components or wires incorrectly/accidently placed. The component highlighted (purple) will be eliminated. If you want to eliminate some component that is not highlighted, you simply move the cursor over the component and left-click to highlight it.

Laboratory 3 Description - PSpice Circuit Simulations

Objective:

The objective is to use PSpice to design and analyze analog circuits.

Pre-lab preparation

Part 1 – Pre-lab discussion / calculations

1. Calculate the current supplied by the voltage source shown in Fig. C.1.
2. Calculate the voltage v(t) across the capacitor shown in Fig. C.12. Assume $v_c(0) = 0V$. The capacitor current can be found to be:

   $i_c(t) = 4.13\cos(60t) + 2.18\sin(60t) + 4.129e^{-\frac{t}{.0088}}\, mA.$
3. Explain in words how you think a diode placed between the two resistors in Fig. C.12 will change the performance of that circuit for t>0.

Experimental procedure:

Follow Appendix C, step-by-step.

Part I – DC circuit

1. Include the screen dump that corresponds to Fig. C.10.
2. Include the screen dump that corresponds to Fig. C.11.

Part II –Transient circuit

1. Include the printed trace that corresponds to Fig. C.13.
2. Include the printed trace that corresponds to Fig. C.17.

Part III – AC circuit

1. Include the printed trace that corresponds to Fig. C.19.
2. Include the printed trace that corresponds to Fig. C.21.

Part IV – Diode operation

1. Add a diode between the two circuit resistors from Part II (in series with the 500Ω resistor in Fig. C.12). Print the new schematic.
2. Rerun the circuit simulation and plot the output voltage across the capacitor as in Fig. C.15.

Post-lab analysis:

Generate a lab report "following" the sample report available in Appendix A. Mention any difficulties encountered during the simulations. Describe any results that were unexpected and try to account for the origin of these results (i.e. explain what happened). In ADDITION, answer the following questions:

1. Did the current you measured in the first circuit simulation agree with the pre-lab calculation (Question #1)? Prove your answer, e.g. what is the measured error?
2. Did the voltage you measured in the second circuit simulation agree with the pre-lab calculation (Question #2)? Prove your answer, e.g. what are the errors in amplitude and phase? (N.B. this question is more difficult than the last one…)
3. Measure the period of the sine wave in Fig. C.17 using the cursors in Probe. What is the error between your measurement and the actual period?
4. For the third simulated circuit, in what frequency range is the voltage across the inductor small, and why?
5. How would the answer to the previous question change if the inductance were decreased by an order of magnitude?
6. For the third simulated circuit, in what frequency range is the voltage across the capacitor small, and why?
7. How would the answer to the previous question change if the capacitance were decreased by an order of magnitude?
8. Looking at the plots in Fig. C.21, clearly the sum of the three voltages in the figure do not sum to one volt at most, if not all, frequencies. Reconcile this apparent contradiction with Kirchhoff's Voltage Law.
9. Did the diode perform as you expected it to according to your Pre-lab question #3 answer? Please give both quantitative and qualitative similarities and differences between your expected performance and actual performance.