Purpose

The purpose of this experiment is to introduce the students to PSPICE. The most common forms of analyses will be considered. The students will be required to copy schematics and graphs from PSPICE and paste them into a word processing program.

History

SPICE (Simulation Program Integrated Circuit Emphasis) was developed at UC Berkeley in the 70’s to provide an efficient method to analyze large arrays of transistors. In the early versions of SPICE, each circuit element and each desired form of analysis was a card in a computer deck. This quickly changed from a card to a line of code. The original version from Berkeley had no graphical input or output capabilities but rather the output was a string of numbers. However, most of the first commercial versions of SPICE had some graphical output package contained as part of the program. These early versions of SPICE required large, main frame computer to run. The first successful version of SPICE to run on a PC was PSPICE developed by MicroSim. MicroSim originally supported the PC as well as the Macintosh, and the Sun systems. When MicroSim introduced the schematic capture version of PSPICE they stopped supporting any operating system except Windows.

Over the past several years, MicroSim sold out to ORCAD who in turn sold out to Cadence. In this laboratory (as well as in EEE 108L and EEE 109 labs) you will be using the latest student version by Cadence. This is the version contained in your Herniter text. When installing PSPICE on your computer you are encouraged to use the method for library installation outlined in Appendix A of the text. This will make your computer compatible (maybe?) with the computers in RVR 3017.

There are three main forms of analysis in PSPICE. They compare to the three sections of ENGR 17. The first form of analysis is the DC analysis. This is the usual DC analysis (resistive circuits) with inductors behaving like short-circuits, and capacitors like open-circuits. The second form of analysis is the TRANsient analysis. This analysis uses numerical methods to solve the nonlinear equations that describe the circuit. The third method of analysis is the AC analysis. This method uses phasor analysis on the linearized circuit.

The instructor will demonstrate in class how to place the components on the schematic diagram, and adjust the “Simulation Profile” for the desired analysis. Note that PSPICE has the capability to specify an initial condition on the capacitor (IC). Remember that initial condition on a capacitor is the voltage, which is related to the initial energy stored. If no initial condition is specified, the capacitor would behave like an open circuit, given that the circuit is driven by a DC source. Realize that by setting the initial condition to zero, the circuit in Figure 1 is equivalent to that in Figure 2, with the switch originally closed, then opening up at t=0s.

The Simulation Profile is found under “PSPICE” on the tool bar. Transient is the default mode. Having calculated the time constant, \( \tau \), for the circuit (pre-lab), a “Run to time” of \( 5 \tau \) should provide sufficient time for the circuit to reach steady state. PSPICE uses numerical analysis to find the output of the circuit. Hence, the output is a series of straight lines and not a continuous function. If a sufficient number of these straight lines is used, the graph appears to be a continuous curve. A few hundred points is usually used. For this example 500 should be plenty. Therefore, adjust the “Maximum step size” to \( \frac{\text{Final Time}}{500} \). Click on “OK” to close the simulation profile box.

The circuit is now ready to be simulated. If the simulation is not successful, some error message will appear. The actual error messages are found in the “OUTPUT FILE” which appears under “PSPICE” on the tool bar and in the Session Log found under Windows. If the simulation is successful “PROBE” (a graphical interface) should open. (Probe might be “hidden” behind the schematic page.) Add the graph of the output by selecting “ADD” under “TRACE” on the toolbar. When ADD is opened, two columns of functions appear. The right-hand column contains the mathematical functions that PROBE
can perform. The left-hand column contains the output variables that are available for plotting. For this example V(VOUT) is chosen. The solution of the circuit should now appear on the graph.

Next, construct a new circuit by interchanging the resistor and the capacitor in the circuit shown in Figure 1. Repeat the simulation this time with VOUT being the voltage across the resistor.

The final part of this experiment is to copy the circuit schematic and the graph from PSPICE (for the simulation of the circuit in Figure 1 only) into a word processor. The graph is copied from PROBE by using “COPY TO CLIPBOARD” under “WINDOWS” on the tool bar. The schematic is copied by selecting the entire circuit and using “COPY” under “EDIT” on the tool bar. These two pictures should be included in your laboratory report due next week.

AC Analysis

This analysis uses the two circuits simulated in the transient analysis section. However, the DC source is replaced with an AC (VAC) source. Adjust the source for 1 Volt with 0˚ phase shift. In the Simulation Profile use the AC Sweep. For sweep type, select decades. The sweep parameters should be 50-points/decade, with a start frequency of 1Hz and an end frequency of 100KHz. PSPICE uses the phasor method to calculate the amplitude and phase of the circuit responses at each of the 50 points in the frequency range specified.

The first circuit (Figure 1) is now ready for simulation. For the first trace, use the magnitude of the output voltage. Select M (for magnitude) from the functions and V(VOUT) from the variables. The next trace will be the phase. Since the values of the magnitude and phase are so different, a different Y-axis should be used for the phase. Use ADD y-axis (under PLOT on the tool bar.) Select the new Y-axis and add P[V(VOUT)]. (If the simulation is correct this RC circuit should have a magnitude of 0.707 and a phase shift of -45˚ at $\omega = 1$Krs {f = 1000/2$\pi$ Hz.}) Note that PSPICE plots the graphs in Hertz and not radians per second.

Repeat the simulation and plot of magnitude and phase for the modified circuit (simulating the resistor voltage, having swapped the capacitor and the resistor).

This graph and schematic (for the first circuit) should also be copied and included in the laboratory report.

Pre-Lab and Lab Report

Solve for the capacitor voltage (for t>0) in the circuit shown in Figure 2. Calculate the time constant, $\tau$. Plot the capacitor voltage for 0<t<5$\tau$. The lab report should contain only the two schematics and the two graphs. No other write-up is required. This lab report is worth zero points but must be turned in to pass the course.
This is a very brief description of the steps necessary to get started in Cadence PSPICE. For more detail you are referred to the text.

Drawing a Schematic

2. Open "New Project".
3. Select "Analog or Mixed A/D".
4. Select an appropriate name (such as RC-Circuit).
5. Select "Create a Blank Project".
6. To begin placing parts use "Part" under the "Place" menu –left click to place the part and use ESC to change to a new part.
7. For ground use "0/Source". Every PSPICE program must have a ground.
8. To change a part name or value click on the desired name or value.
9. To add or change the IC on a capacitor click on the circuit symbol.
10. Use "NET ALIAS" to add names to frequently used nodes (such as VOUT).
11. After you have the schematic in the desired form do a "SAVE".

Doing a Circuit Simulation

1. Under PSPICE on the menu bar choose "NEW SIMULATION PROFILE".
2. Choose a name for the simulation such as RC-TRANSIENT or RC-AC.
3. For the transient response choose “RUN TO” 5ms with a maximum step size of 5us. For the AC sweep use a logarithmic sweep with a starting frequency of 1Hz and a final frequency of 100KHz. Use 50 steps per decade.
4. Choose “RUN” under PSPICE on the menu bar.
5. If you have a successful simulation PROBE should open. Use ADD under TRACE on the menu bar to add the desired traces.

Copying to a Word Processor

1. For PROBE graphs use “Copy to Clipboard” under Windows.
2. For schematic diagrams use the mouse to select the desired part of the circuit. Next use copy under Edit.

Trouble Shooting

1. The major reasons for a failed simulation are shown in the “Session Log” found under “Window.”
2. Other errors may be found in the “Output File” found under “PSPICE.”