Summary: PSPICE Essentials

Introduction

Most universities have access to some form of the Windows version of PSPICE. It has been available as freeware to institutions of higher education for many years. There are many comprehensive textbooks available that cover all aspects of its operation. This Summary will briefly include only those parts of this versatile software package which are necessary to simulate and analyze the circuits used as examples in your laboratory assignments. The processes of putting parts on a Schematic and wiring them together are obvious enough that they will not be presented here.

A1. Analyses

Your lab assignments utilize only four of the Analyses available in PSPICE. For more detailed explanations, consult one or more of the references in the Reference Table below:

Reference:	Transient Analysis	DC Sweep Analysis	AC Sweep Analysis	Parametric Analysis
R. W. Goody, *MicroSim PSPICE for Windows, 2nd ed.,* Prentice- Hall	Chs. 3 & 7	Chs. 2 & 4	Chs. 4 & 5	Ch. 6

1. Transient Analysis: This analysis produces time-domain plots in PROBE of any legal variable in the PSPICE Schematic. It works by solving in the time-domain the differential equations governing the voltage across and the current through each part on the Schematic. To produce an output from a circuit in the time-domain, you must have at least one voltage or current source whose attributes are also specified in the time domain. Once plotted, the resulting time waveforms can be Fourier transformed to yield frequency-domain information.
2. DC Sweep Analysis: This analysis produces plots of DC voltages and currents in the circuit as functions of another DC voltage and/or current.
3. AC Sweep Analysis: This analysis produces frequency-domain plots of the amplitudes and phases of voltages and currents. The horizontal axis in these plots is frequency.
4. Parametric Analysis: This analysis can be done in conjunction with the others. It allows you to vary a parameter in the circuit. The Parametric Analysis essentially causes the other analysis paired with it to be repeated for different values of the parameter being varied.

You select the analysis/analyses to be performed in the Analysis->Setup menu shown in Fig. A1.

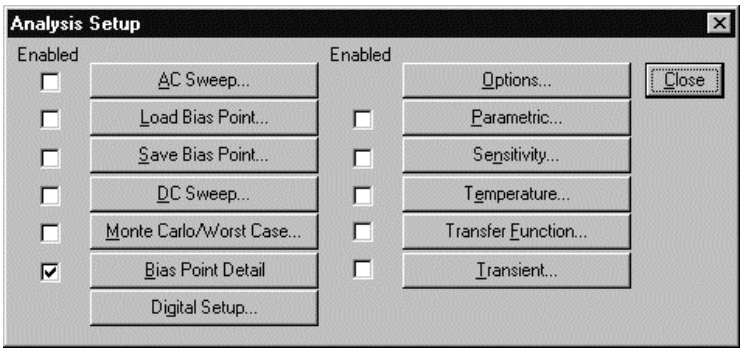

Fig. A-1.
The Analysis Setup Menu

A1.1. Transient Analysis. Figure A2 shows the Transient analysis setup menu. You need to set the **Final Time** at the time your analysis is to end. **Step Ceiling** can keep the program from running too long by limiting the total number of time steps allowed. The **Fourier Analysis** section at the bottom of the menu lets you calculate a Fourier Series for a periodic waveform. You must give the program the **Center Frequency** (the frequency of the fundamental harmonic) and the number of harmonics to be calculated, as well as the **Output Variables** for which you want Fourier Series components calculated.

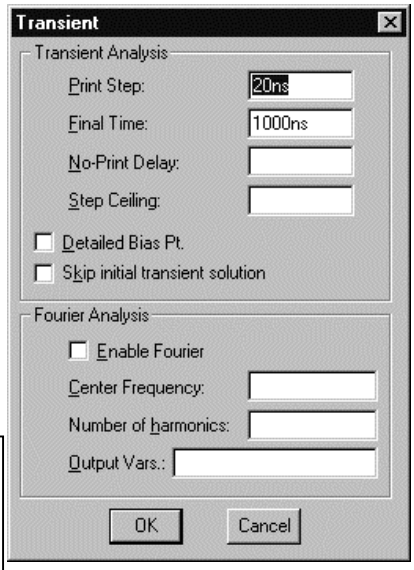

Fig. A-2.
Transient Analysis setup menu.

The resulting Fourier Series doesn't appear in PROBE, but at the end of the PSPICE output file. You get to this file by selecting **Analysis->Examine Output.**

A1.2. DC Sweep Analysis. Figure A-3 shows the DC Sweep setup menu. In the figure, this menu is set up the way you will use it most often. It will cause the voltage source V1 to sweep from 0 to 12 volts in steps of 0.01 volt.

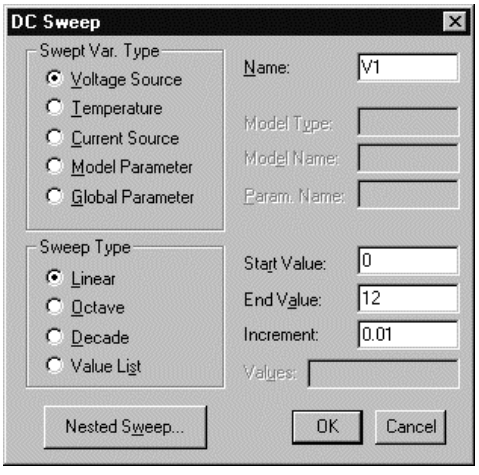

Fig. A-3.
DC Sweep setup menu

A1.3. AC Sweep Analysis. Figure A-4 shows the AC Sweep setup menu. You must define the start and end frequencies for the frequency domain plot, and the type of scale. The Noise Analysis feature is beyond the scope of this summary.

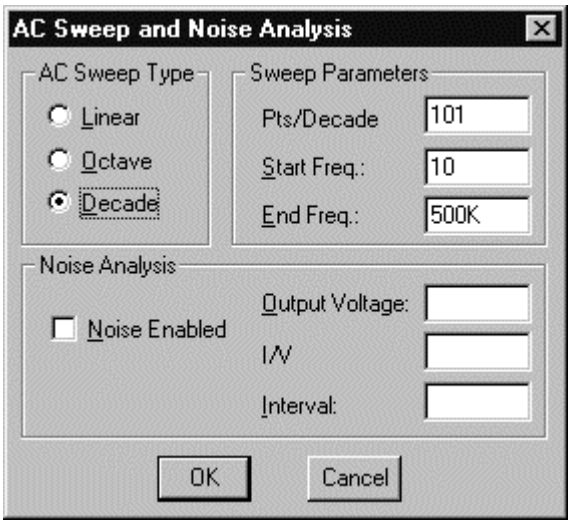

Fig. A-4.
AC Sweep setup menu.

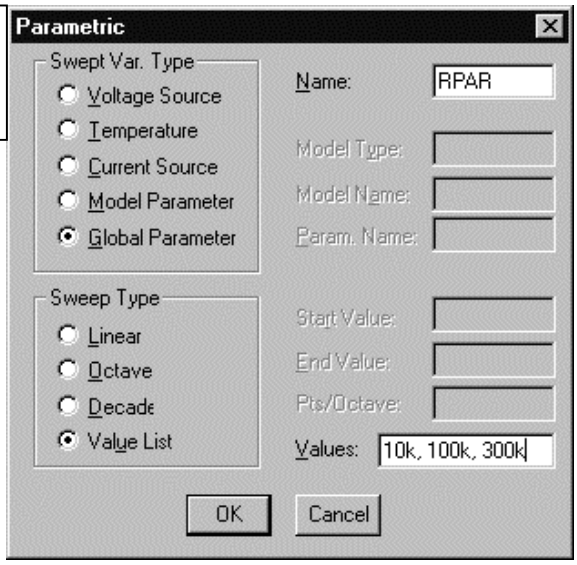

Fig. A-5.
Parametric Analysis setup menu.

A.1.4. Parametric Analysis. Figure A-5 shows the Parametric Analysis setup menu. In the figure, the value of a component will take on three values, 10, 100k, and 300k. The value of the particular component in the schematic diagram has been replaced by **{RPAR}**, and the name of the parameter, RPAR, has been added as a **NAME** in a part on the schematic called a **Parameter List.** Figure A-6 shows how the schematic looks in a parametric analysis where the value of a resistor is to be the parameter varied. Component values such as resistor or capacitor values are "**Global Parameters**" in Fig. A-5.

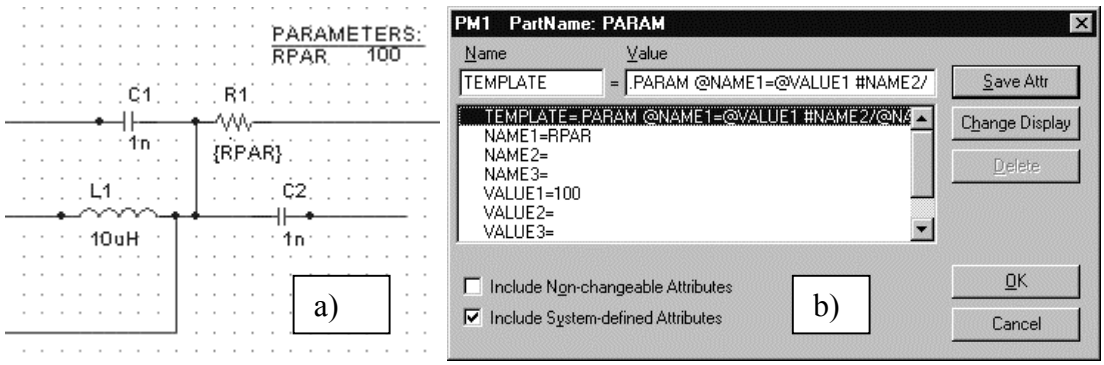

Fig. A-5. a) Schematic with the value of resistor R1 made a parameter, b) the Parameter List.

You "Get" the **Parameter List** like any other part, then double-click on it in Fig, A-5a) to open the menu in Fig. A-5b). The value you enter in Value1 in Fig. A-5b) will only be used if you disable **Parametric Analysis** in the main Analysis Setup menu, Fig. A-1. If you check the **Parametric…** box there, Fig. A-5 determines what values R1 will take on in the analysis.

A2. Some PSPICE signal sources

One of the parts libraries in PSPICE, called the **source.slb**, contains a wide variety of independent and dependent voltage and current sources. Your lab assignments only utilize four of them: the independent voltage sources, VSRC, VPULSE, VPWL, and VSIN. Once you understand how these work, it is easy to learn to use the other sources.

A2.1. The simple voltage source, VSRC. This voltage source provides the following voltages:

- a constant DC level in a Transient analysis, set by its attribute, **TRAN**.
- a sine wave in an AC Sweep analysis, whose amplitude is set by the attribute, **AC**.
- a DC voltage in a DC analysis, set by the attribute **DC**.

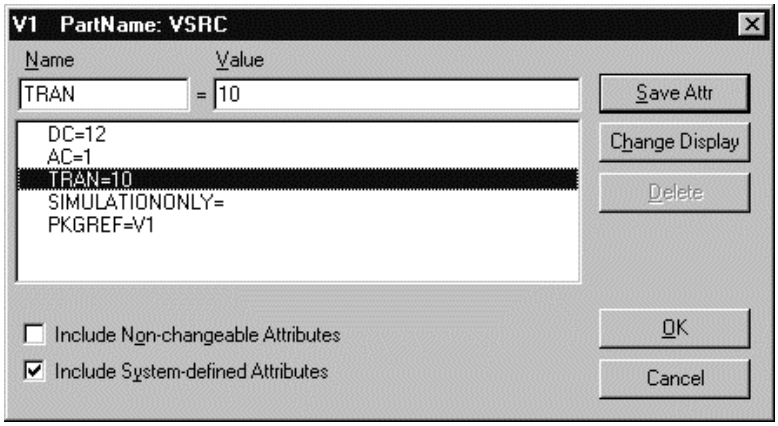

Fig. A-6. The Attributes of a simple voltage source, VSRC.

Figure A-6 shows the menu for setting the attributes for the VSRC source. In a Transient Analysis, this source would be a DC level of 10 volts, while in an AC sweep, it would be a sine wave of amplitude one volt, whose frequency would vary over the range specified in the AC Sweep setup menu.

A2.2. The pulsed voltage source, VPULSE

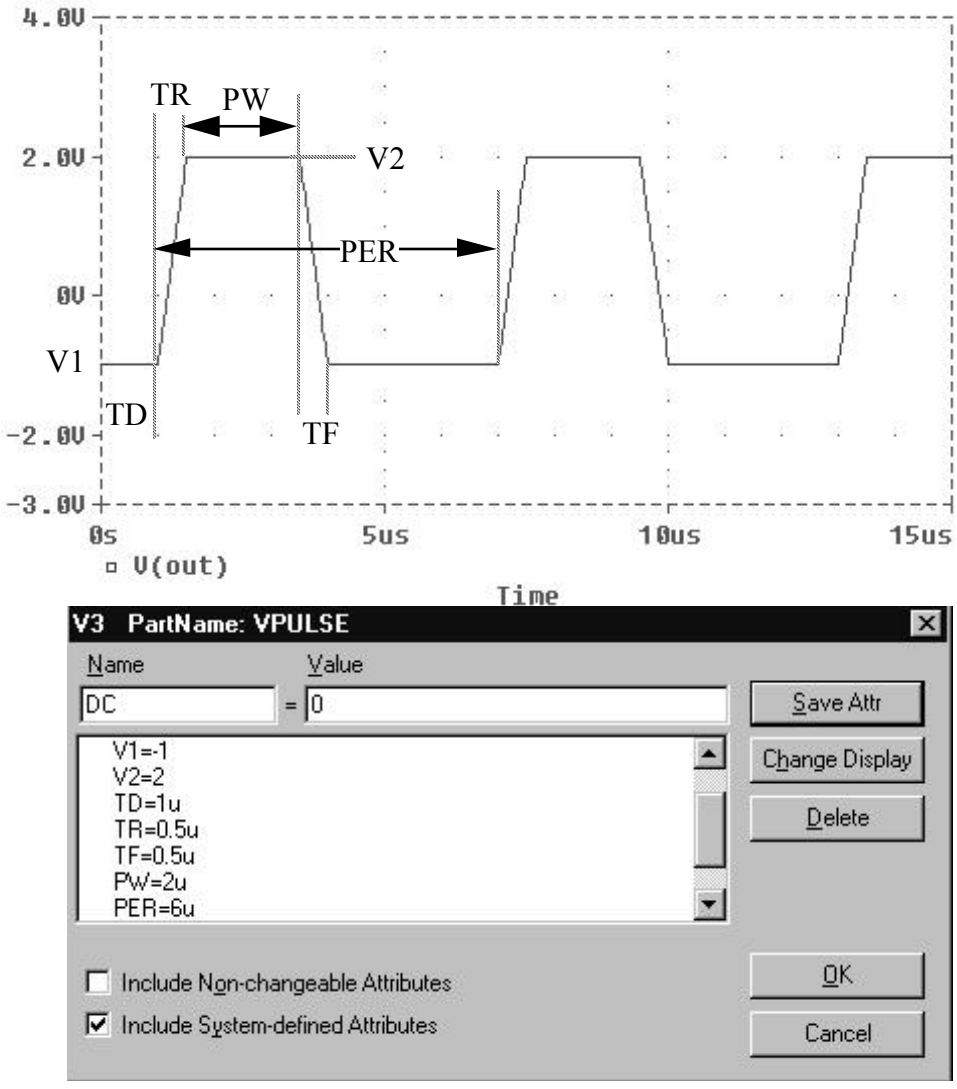

Fig. A-6. Time waveform and attributes of the VPULSE voltage source in PSPICE.

This voltage source provides various pulsed waveforms where up and down transitions are made between two voltage levels. Figure A-6 shows how its attributes relate to its time waveform. You can use this source to make pulse trains, square waves and triangle waves.

A2.3. The piecewise linear voltage source, VPWL

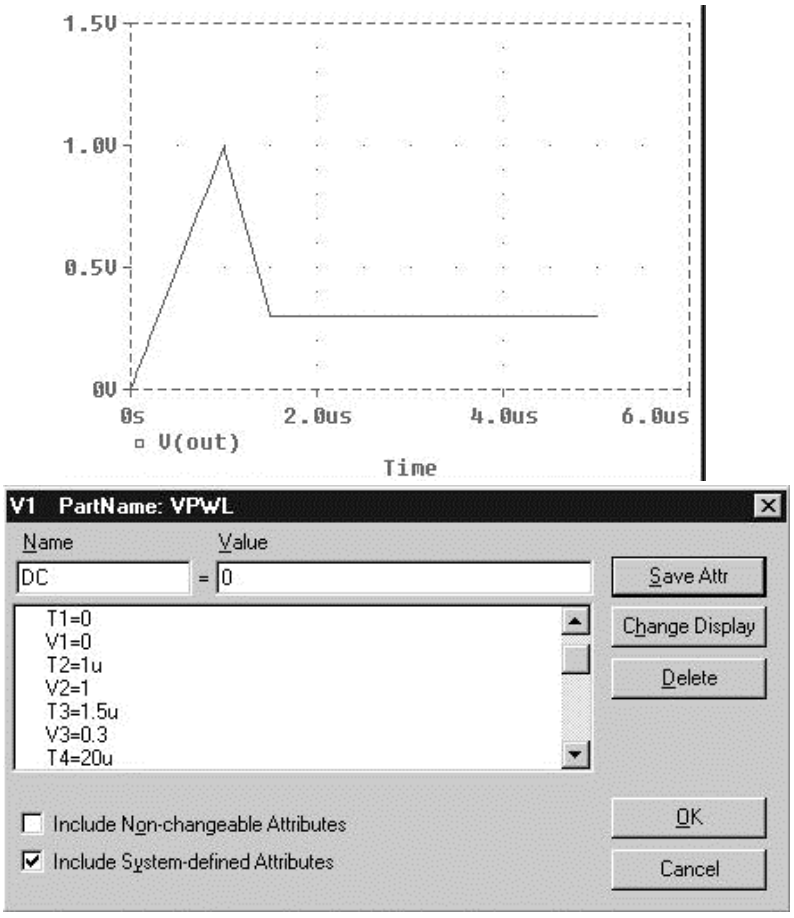

Fig. A-7. Waveform and attributes of the VPWL voltage source.

In principle, you could make any waveform you wanted with this voltage source. Its time waveform is a series of straight lines connecting the points (Vi,Ti) in order of the index, *i*. Figure A-7 shows its attributes and time waveform.

A2.4. The sine-wave source, VSIN

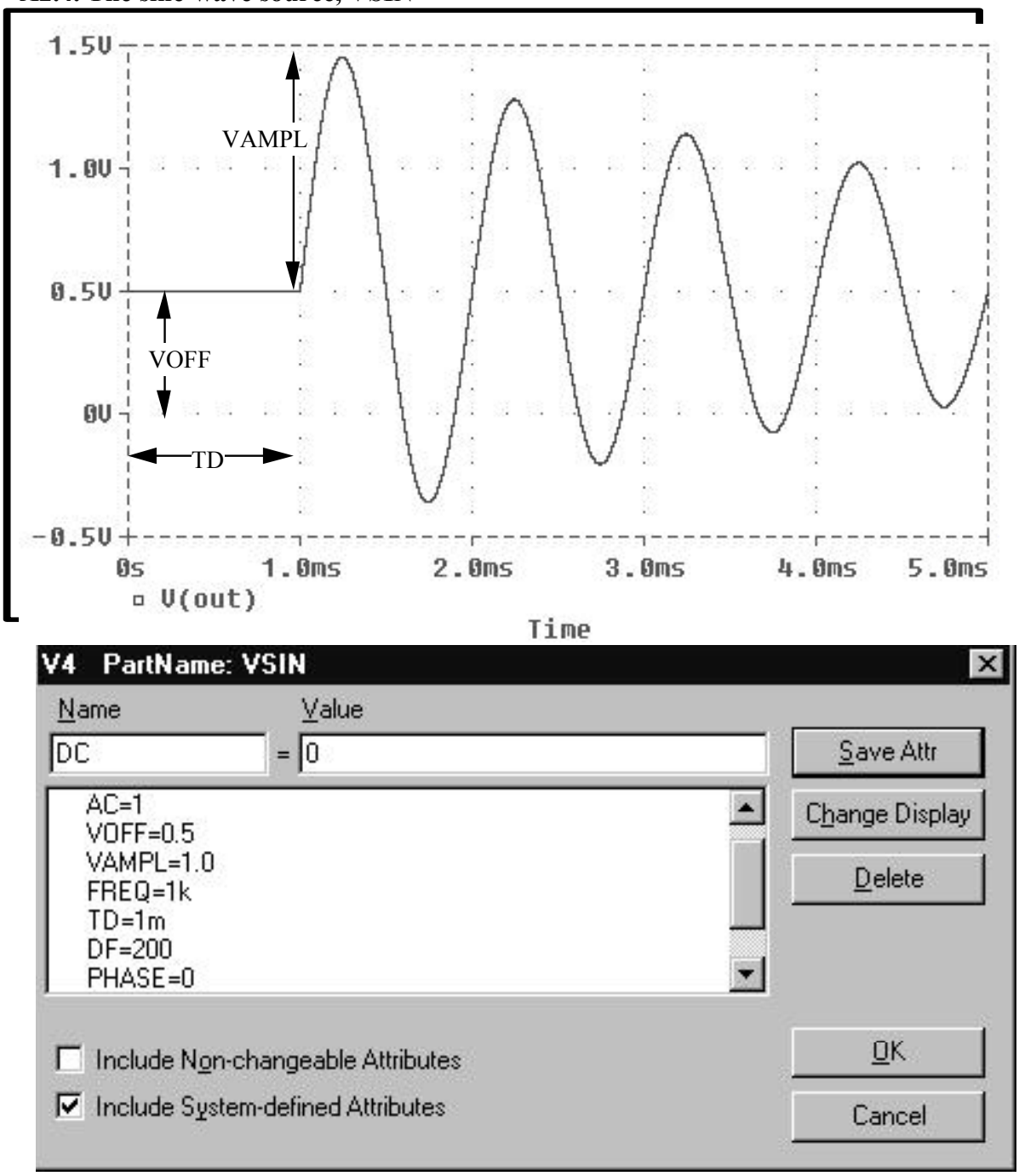

Fig. A-8. Time waveform and attributes of the VSIN source.

Use this source to make sine waves. You can set every parameter you see in the attribute list in Fig. A-8. The attribute, **DF**, is a damping factor and has the units of reciprocal seconds.