

III. Using SPICE Simulation

The PCs in the Electronics Laboratory are loaded with the student version of PSpice, a commercial version of the SPICE simulation package. This software may be copied, and is widely available. Students are encouraged to use PSpice to simulate the lab experiments, and to compare with the laboratory data. (This is only required if explicitly requested.) To this end, SPICE models for most of the components used in the lab are found in the appendix of this manual, and included in the PSpice model library. Groups working in the lab may find it convenient to have one member run a simulation of an experiment while the other connects the hardware. Simulation results can often be used as an aid to learning how a circuit would work if the components were ideal and correctly connected. A simulation also allows access to difficult-to-make measurements; for example, a current waveform for which there is no laboratory current probe available. One should never confuse simulation results with actual laboratory results, however! There are many non-ideal behaviors in the components and circuit layout which are not modeled in the simulation.

Because of the rapid evolution of the PSpice user interface, it is difficult to give detailed directions on how to use it. Older versions of SPICE expect an input file containing the circuit description in a format which has been standard since the 1970s. This input file generally carries the extension ".CIR". The newer Windows versions of PSpice include a schematic editor which allows the user to draw a schematic diagram using library component symbols (found in *.SLB). The model corresponding to each symbol must be in the model library (*.LIB). The program then automatically generates its own input file, following the normal conventions. This is a process which involves creating a netlist (*.NET), which is a description of the schematic in SPICE conventions using automatically assigned node numbers. The results of running a simulation, including an echo of the input file, any error statements, and the results of print/plot commands, are written to a file with a ".OUT" extension. In PSpice, the utility named "PROBE" is used to produce an oscilloscope-style plot of the simulation results. If PROBE is called in the .CIR file, a PROBE data file with the ".DAT" extension will be generated.

The user interface in the PSpice version in the Electronics Laboratory expects the user to open the application "Schematics" first, draw a schematic, and assign component values. The command "Create Netlist" calls a netlist generator which produces the part of the .CIR file which describes the circuit. The "Setup" command adds statements indicating the desired type of simulation (dc, ac, transient, etc.). The "Simulate" command runs the simulation, and starts PROBE automatically, if this has been enabled in "Probe Setup." It is also possible to use any text editor to manually create the .CIR file, and then directly go to "Simulate." This is the process described in most texts on SPICE. The following libraries have been added to PSpice as loaded in the Electronics Lab: USER.LIB, USER.SLB, and USER.PLB. These are libraries containing device models, circuit symbols, and device packages for some of the standard components used in this lab. Their accuracy cannot be guaranteed.

As an example, a circuit pertinent to the diode experiment (Experiment 3) is simulated.

Using SPICE Simulation

Fig. 1. Diode clipper drawn in PSpice schematic editor.

Fig. 1 shows a diode clipper, drawn using PSpice schematic editor. V1 is an 23-V-peak 60-Hz ac sinusoidal voltage source. This is set in a dialog box that appears upon double-clicking on V1 with the mouse. The diode, a 1N4148, is described by a diode model which is in the library included with PSpice. The markers tagged with a circled "V" indicate which voltages will be automatically displayed when PROBE is run. The simulation type used here is a parametric transient simulation. This means that the dc source V2 will be given three different values: 0 V, 5 V, and 10 V. For each of these three values there will be a transient simulation of the circuit. This setup was defined through the "Setup" dialog box. The two marked voltages will be displayed by PROBE automatically; all the others are available by asking. Upon running the "Create Netlist" command, the following file is created. This follows the normal SPICE conventions. The node numbers are prefixed with "\$" and are automatically assigned. Recall that "0" is defined by SPICE to be the global reference point (ground).

```
* Schematics Netlist *
```

```
V_V1      $N_0001 0 DC 0  SIN(0 23V 60)
R_R1      $N_0001 $N_0002  10K
R_R2      0 $N_0003  1K
D_D1      $N_0002 $N_0003  D1N4148
V_V2      $N_0003 0 5V
```

The "Setup" dialog box will add the simulation commands to this file. After running the simulation, any error statements generated, as well as the simulation statistics, may be found in the .OUT file. This file is as follows:

Using SPICE Simulation

**** 07/26/96 12:55:44 ***** Win32s Evaluation PSpice (April 1996) *****

* C:\MSIMPR63\EVAL\EXAMPLES\DIODEX\EXP3.SCH

**** CIRCUIT DESCRIPTION

* Schematics Version 6.3 - April 1996

* Fri Jul 26 12:53:49 1996

** Analysis setup **

.tran 200us 20ms 0 10u SKIPBP

.STEP V_V2 LIST

+ 0 5 10

* From [SCHEMATICS NETLIST] section of msim.ini:

.lib nom.lib

.INC "EXP3.net"

**** INCLUDING EXP3.net ****

* Schematics Netlist *

V_V1 \$N_0001 0 DC 0 SIN(0 23V 60)

R_R1 \$N_0001 \$N_0002 10K

R_R2 0 \$N_0003 1K

D_D1 \$N_0002 \$N_0003 D1N4148

V_V2 \$N_0003 0 5V

**** RESUMING EXP3.CIR ****

.INC "EXP3.als"

**** INCLUDING EXP3.als ****

* Schematics Aliases *

.ALIASES

V_V1 V1(+= \$N_0001 -=0)

R_R1 R1(1=\$N_0001 2=\$N_0002)

R_R2 R2(1=0 2=\$N_0003)

D_D1 D1(1=\$N_0002 2=\$N_0003)

V_V2 V2(+= \$N_0003 -=0)

.ENDALIASES

**** RESUMING EXP3.CIR ****

.probe

.END

**** 07/26/96 12:55:44 ***** Win32s Evaluation PSpice (April 1996) *****

* C:\MSIMPR63\EVAL\EXAMPLES\DIODEX\EXP3.SCH

Using SPICE Simulation

**** Diode MODEL PARAMETERS

```
      D1N4148
IS     2.682000E-09
N      1.836
ISR    1.565000E-09
IKF    .04417
BV     100
IBV    100.000000E-06
RS     .5664
TT     11.540000E-09
CJO    4.000000E-12
VJ     .5
M      .3333
```

JOB CONCLUDED

TOTAL JOB TIME 10.77

Using SPICE Simulation

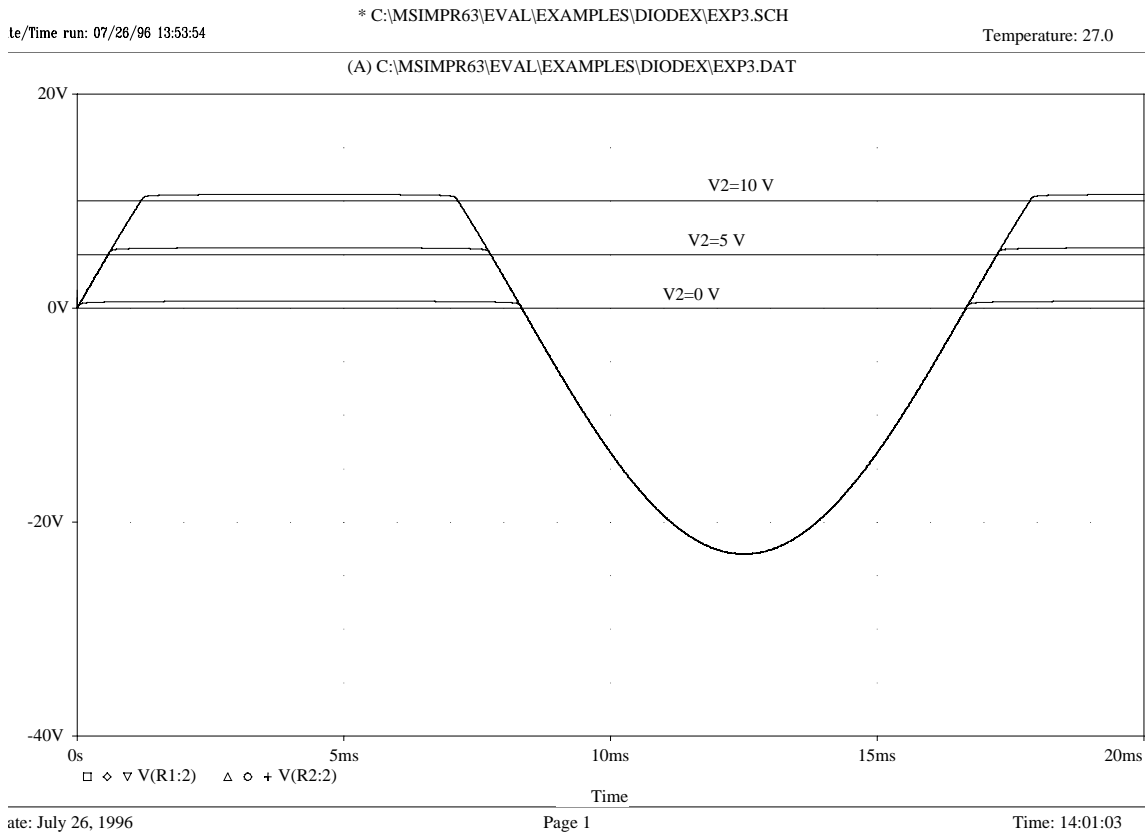


Fig. 2. The initial PROBE display. The labels were added manually.

PROBE begins running immediately upon completion of the simulation. The start-up display by PROBE appears as shown in Fig. 2. The labels "V2=0 V" etc. were added manually in PROBE. The three different clipping levels, corresponding to three different reference values for V2, can be seen above. The effects of diode forward drop (about 0.6 V) can also be seen.

Using SPICE Simulation