Introduction to PSpice

PSpice is a version of the standard circuit simulator Spice. The student edition of PSpice, which is limited to small circuits, has been installed on the PCs in the circuits lab. You may also download or order your own free copy of the student edition from Orcad by going to `http://www.electronics-lab.com/downloads/schematic/013/`.

Creating a Circuit Description File

You can run PSpice by going to **Programs** → **PSpice Student** → **PSpice AD Student** from the Start menu. Next, we have to create a text file that describes our circuit and the simulation protocol. Create a new text file (**File** → **New** → **Text File**) and immediately save it (**File** → **Save As**) with the extension .cir (e.g., test.cir). Now you must open the file (**File** → **Open**, and change "Types of Files" to "Circuit Files") before PSpice will recognize it as a valid circuit description file.

Writing and Simulating a Circuit

Spice is a bit like a programming language, once you learn the syntax and conventions it is quite easy to use. In addition to the tutorials associated with this lab, you can access the complete PSpice user's guide at `http://www.electronics-lab.com/downloads/schematic/013/tutorial/PSPICE.pdf`. Another useful resource is `http://www.seas.upenn.edu/~jan/spice/spice.overview.html`; especially the section on "Most Common Mistakes". For now, let's type in the following text:

```
* Test circuit.  Remember, the first line is ALWAYS a comment.
* We can add additional comments by beginning a line with an asterisk.

* AC Voltage source, 1V magnitude, from node 1 to node 0 (ground):
Vin 1 0 AC 1

* 1 kOhm resistor from node 1 to node 2:
R1 1 2 1k

* 0.47 uF capacitor from node 2 to node 0 (ground):
C1 2 0 0.47u

* AC analysis, calculating 50 points per decade from 1 Hz to 1MHz:
.AC DEC 50 1 1MEG
* Enable plotting program:
.PROBE
.END
```

Now save the file, and run the simulation (**Simulation** → **Run**). A new window should appear with a blank plot. Note the tabs at the bottom left that you can click on to get back to your circuit file. Go to **Trace** → **Add Trace** and type `VDB(2)-VDB(1)`. This displays a magnitude Bode plot of the simple RC circuit you described from 10 Hz to 1 MHz. The output is node 2, and the input is node 1, so the transfer function is given by the ratio of the two voltage magnitudes, but since we.re using a logarithmic measure (dB), we subtract instead of dividing.

Now click on **Plot** → **Add Plot** to Window. A new box should appear above the old plot. Now add a trace to this plot: `VP(2)-VP(1)`. This is the phase Bode plot of our circuit! So now we know how to do frequency-domain analysis. What about timedomain analysis? Let's look at how this circuit responds to a square wave input. Go back to the circuit file and replace the old description of `Vin` with this one:

```
Vin 1 0 pulse(-1 1 0 0 0 3m 6m)
```

This represents a voltage source connected to nodes 1 and 0 that generates pulses from .1 V to +1 V with a pulsewidth of 3 milliseconds and a repetition period of 6 milliseconds. (Remember, 3m = 3M = 0.003; 3meg = 3MEG = 3,000,000.)

Now replace the .AC statement with the following statement:

```
.TRAN 12u 12m
```

This tells PSpice to do a transient (time-domain) analysis over 12 milliseconds, plotting points every 12 microseconds. Run the simulation, and add the traces of `V(1)` and `V(2)` to the same plot. You can print the circuit files or the plots (**File** → **Print**).

Transistors

Transistors are a bit more complex than passive elements such as resistors and capacitors. The format for entering a MOSFET in a circuit file is:

```
Mname drain gate source body modelname
```

For example:

```
M1 4 7 6 6 2N7000
```

Where the model definition for the transistor must be included in the file (see example below). Notice that the source and body are connected to the same node. Here is an example of what the model definition for a transistor might look like:

```
.MODEL EXAMPLE_TRANSISTOR NMOS (LEVEL=3 RS=0.205 NSUB=1.0E15 DELTA=0.1
KAPPA=0.0506
+TPG=1 CGDO=3.1716E-9 RD=0.239 VTO=1.000 VMAX=1.0E7 ETA=0.0223089
+NFS=6.6E10 TOX=1.0E-7 LD=1.698E-9 UO=862.425 XJ=6.4666E-7 THETA=1.0E-5
+CGSO=9.09E-9 L=2.5E-6 W=0.8E-2)
```

Notice that the + symbol is used to continue a statement on multiple lines. Don't be intimidated by the model definition; it just contains a bunch of parameters used by PSpice to simulate this particular transistor accurately. For example, `TOX` is the oxide thickness, and `W` and `L` are the dimensions of the device.

In this class, we will be using a more complex transistor model to accurately model the 2N7000 transistors used in the lab. The more complex model we will use is implemented as a subcircuit that starts with a SPICE transistor model at its core and builds up additional circuitry around it (resistors, capacitors, and diodes) to more accurately model the packaging parasitics and overall behaviour of the discrete 2N7000 transistors that we will be using. The definition file for this circuit is provided on the class website, the text in this file must be included at the start of your SPICE `.cir` file, and transistors can then be defined with the statement:

```
X1 2 1 0 2N7000
```

Here `X1` is the reference designator, and `2`, `1`, and `0` are the drain, gate, and source respectively.