ECE 3410 – Microelectronics I
SPICE Handout I

SPICE – short for "Simulation Program with Integrated Circuit Emphasis" – is a powerful platform for simulating electronic circuits and systems. There are several different implementations of SPICE, starting with the original Berkeley SPICE program, which is open-source and designed for Unix environments. In this course we focus on PSPICE, a commercial implementation which runs on Windows. The basic structure and syntax varies little for different versions of SPICE.

SPICE File Structure

Every SPICE simulation is defined by a SPICE file. A SPICE file is organized as follows:

```
Title Line
[Include definitions]
[Options]
[Sub-Circuit Definitions]
Devices and Node Connections
Power and Signal Sources
Analysis Commands
Output Statements
.END
```

Each SPICE file must begin with a title line and end with a .END statement. The .END statement must be followed by a carriage return.

At minimum, the SPICE file must specify the structure of a circuit, power sources, and some analysis or analyses to perform on the circuit. Output statements are usually desirable in order to print out the results of the simulation.

Devices and Nodes

Passive devices are declared using the following syntax:

```
Device_Name node1 node2 value [Initial Condition]
```

The first letter of the device name indicates its type. Possible passive device types are (R) Resistors, (C) Capacitors and (L) Inductors. Nodes are usually designated by numbers. Node 0 is commonly reserved for ground. The initial condition is either a voltage or current through the device. The initial condition is useful for transient simulations, and is always optional.

Example:

```
R1 1 2 100kOhm
R1 1 2 100k
R1 1 2 100e3
```

These lines are all equivalent. Each declares a 100kΩ resistor between nodes 1 and 2. Values can be specified using scientific notation, as in 3.33e3, or using SI order-of-magnitude units, as in 3.33k. The order-of-magnitude units are as follows:

f (femto)	10E-16	k (killo)	10E+2
p (pico)	10E-13	Meg (Mega)[1]	10E+5
n (nano)	10E-10	G (giga)	10E+8
u (micro)	10E-7	T (tera)	10E+11
m (milli)	10E-4		

---

[1] Note that SPICE is not case-sensitive, so `m' is the same as `M'. You must type `MEG' to indicate "mega".

Independent Sources

Independent voltage and current sources are specified using similar single-line statements. SPICE statements can declare DC and AC sources, as well as transient sources such as step functions and piecewise-linear signals. For a DC source, the general format is

```
Name positive_node negative_node type value
```

where "type" is DC or AC.

Example:

```
V1 1 0 DC 5V
```

This line declares a DC independent voltage source of 5V, with its positive terminal connected to node 1 and its negative terminal connected to node 0. Independent current sources are declared similarly, only the name must begin with an `I'.

Example:

```
I1 2 3 DC 10mA
```

This line declares a DC current source which forces a current of 10mA to flow from node 2 to node 3.

Analyses

The most basic analyses available in SPICE are *operating point*, *DC analysis*, *AC analysis* and *transient analysis*. The operating point analysis computes the static DC state of the circuit, and is a prerequisite for the other analyses. You can request a simulation to compute nothing but the operating point using this simple statement:

```
.op
```

This statement causes SPICE to calculate the DC voltage of every node in the circuit and print it in the simulation's output file. For large circuits, the .op analysis tends to output more data than is desired.

A more interesting analysis is the DC analysis, which is requested using the .dc command. This command is used to sweep the value of an independent voltage or current source over a desired range. The syntax for the command is

```
.dc source_name start stop step
```

This tells SPICE to sweep "source_name" from "start" to "stop," in increments of "step."

Example:

```
.dc V1 0v 1v 0.1v
```

This tells SPICE to adjust the value of independent voltage source V1, starting at 0v and stepping to 1v with a step size of 0.1v.

Complete SPICE File

The following listing is a simple example of a SPICE simulation:

```
Example Circuit 1
R1 2 1 1kOhm
R2 1 0 1kOhm
V1 2 0 DC 1v
.op
.end
```

To simulate this file using the bundled PSPICE version (part of the OrCAD tools), begin by running the "PSPICE AD" program. The program opens with an active text window. Type the above text into that window, and save the file in your document directory (or on your personal disk) as "example1.cir". Then, under the "File" menu, select "Open Simulation" and choose the "example1.cir" file. Then choose "Run" from the "Simulation" menu.

The simulation status appears in the text frame on the lower left. When the text "Simulation complete" appears in this frame, you may view the output by pressing the "view output" button from the toolbar on the left (it is the third button down). Alternatively, you can select "Output File" from the "View" menu. The following text is displayed in a new tab:

```
**** 02/13/05 21:21:06 *************** PSpice Lite (Mar 2000) *****************
 Example Circuit 1

 ****      CIRCUIT DESCRIPTION
******************************************************************************

R1 1 2 1kOhm
R2 2 0 1k
V1 1 0 dc 1V

.op

.end

**** 02/13/05 21:21:06 *************** PSpice Lite (Mar 2000) *****************
 Example Circuit 1

 ****      SMALL SIGNAL BIAS SOLUTION       TEMPERATURE =   27.000 DEG C
******************************************************************************

 NODE   VOLTAGE      NODE   VOLTAGE      NODE   VOLTAGE      NODE   VOLTAGE

(    1)    1.0000  (    2)     .5000


    VOLTAGE SOURCE CURRENTS
    NAME           CURRENT

    V1             -5.000E-04

    TOTAL POWER DISSIPATION   5.00E-04  WATTS


**** 02/13/05 21:21:06 *************** PSpice Lite (Mar 2000) *****************
 Example Circuit 1


 ****      OPERATING POINT INFORMATION      TEMPERATURE =   27.000 DEG C
******************************************************************************

          JOB CONCLUDED

          TOTAL JOB TIME               .02
```

Exercises

1. Run "OrCad Capture CIS". Under the "Help" menu, select "Learning Capture." A menu of tutorial lessons appears. Step through the following tutorials: "About Capture," "Designs and Schematics," and "Navigating Designs."
2. Run "PSPICE AD" and step through the example given above. Verify that you get the expected results.
3. Create and execute a SPICE file to simulate this circuit:
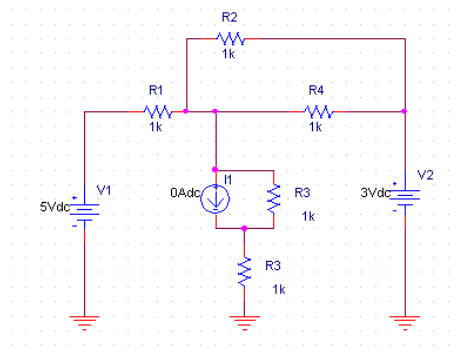
Turn in a hand analysis for this circuit, showing your expected values for all voltages and currents. Also turn in a printout of the SPICE output file, showing your original SPICE statements and the operating point results.

4. Draw a schematic diagram for the circuit described by the SPICE file below. Turn in a drawing of the schematic and the results of your hand analysis, solving for all the currents and voltages in the circuit. Also simulate the circuit in SPICE and turn in a printout of the output file.

```
Circuit for Problem 4
R1 1 2 10ohm
R2 2 0 1kohm
R3 2 3 20ohm
C1 2 0 1nF
V1 1 0 DC 1v
V2 3 0 DC -1v
.op
.end
```