
Experiment 2 - Introduction to PSpice

W.T. Yeung and R.T. Howe

UC Berkeley EE 105

1.0 Objective

One of the tools you will be using as an electrical engineer is SPICE. SPICE is essential to the analysis and design of complex circuits. In this experiment, you will be introduced to the basics of SPICE. You will learn techniques in representing circuit elements, constructing the circuits, and finally simulating the circuits.

2.0 Material

The version of SPICE you will be using is PSpice. PSpice is available in room 123 Cory Hall.

3.0 Prelab

- Reading: Vladimirescu, Chapters 1.1-1.3, 2.1-2.3
- Those who have personal computers can enter your SPICE decks. Save the file as <your-filename.cir>

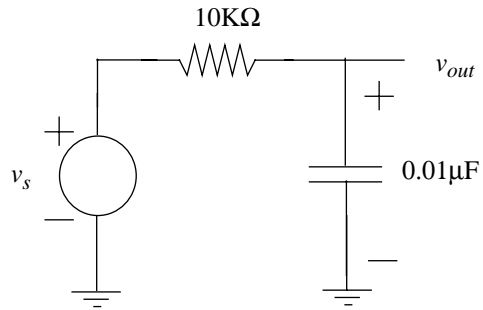
4.0 Procedure

4.1 AC Analysis and Transient Analysis

1. Construct the lowpass filter shown below. The small signal voltage source has no DC value. Let the waveform be a 1kHz square wave that ranges from 0 to 5V.

FIGURE 1.

Lowpass Filter



2. Your SPICE deck should contain the following information:

- Circuit information
- .ac statement for frequencies ranging from 10Hz to 100kHz on a decade scale.
- .tran statement for the times $t=0$ to $t=10\text{ms}$
- .probe

Lab Tip

A voltage source has the following format:

$Vname$ +node -node dc <dc/tran> ac <acmag acphase> transient information

The value for acmag is **NOT** the ac amplitude. It is the magnitude of the phasor used in finding the network function. (see p 48 of *The SPICE Book*) You should set the ac argument to be 1,0. Any small signal amplitude (of a waveform) should be included in the transient information.

The square wave in the above example can be modeled as either a *pulse* or a *piecewise linear function*.

3. Launch the PSPICE program by double-clicking on its icon.
4. Click on *File* and *Open*.
5. Select the filename you assigned to your SPICE deck and SPICE will analyze the circuit.
6. Launch Probe by double-clicking on its icon.
7. Open the data file by clicking on *File* followed by *Open*. Probe data files have a .dat extension.

Procedure

8. From the *Plot* menu, select AC. This will tell Probe to plot ac data and not transient data.
9. Click on the *Trace* command followed by *Add*. This will allow you to plot the voltages and currents available in the circuit.
10. Obtain the Bode plot (phase plot is optional) for the transfer function $\frac{v_{out}(j\omega)}{v_s(j\omega)}$.
How do the results compare to hand calculations? Label the -3dB point.

Lab Tip

Probe can also plot mathematical expressions involving the voltages and currents.

Voltages in decibel scale can be plotted by using Vdb(node number).

You can use the cursor command from the *Tools* menu in probe to get x and y coordinates from the graph.

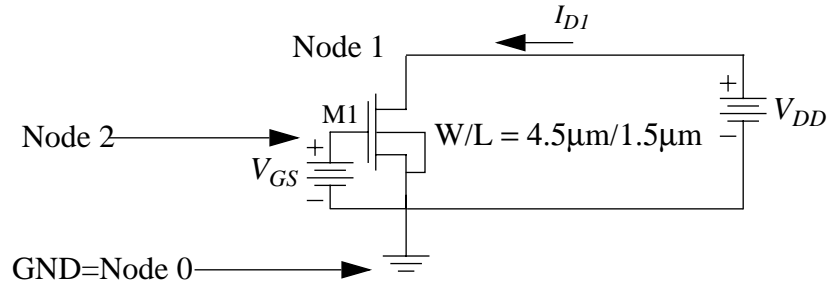
Labels and Titles can be inserted into your plots for clearer understanding. These are accessible from the *Tools* menu.

11. Print out your plot by clicking on *File* followed by *Print*.
12. From the *Plot* menu, select Transient. This will tell probe to plot transient data and not ac data.
13. Obtain a printout for both $v_{out}(t)$ and $v_s(t)$.

4.2 DC Analysis

1. The following circuit is a MOS field effect transistor. You do not need to know the internal operations of the MOSFET in order to complete this part of the experiment. Enter the circuit below in a SPICE deck.

FIGURE 2. MOSFET Circuit to Generate its I-V Characteristics



2. Include in your SPICE deck the following:

- `.dc lin VDD 0 5 .1 VGS 0 5 1` (This sweeps V_{DD} from 0 to 5V in .1V interval for each value of V_{GS} . (0 to 5V in 1V intervals))
- `.probe`
- `Kp=60e-6 Vto=1 Lambda=.05` for `.model` parameters

3. Run PSPICE and Probe and plot the drain current of your transistor I_{D1} . This will give you the MOSFET's IV characteristics. Print a hardcopy. Always include your input deck in your lab report.

5.0 Appendix

There are many versions of SPICE available (PSPICE, SPICE3, HSPICE, etc.). Regardless of the version, the structure of an input file is the same. A SPICE simulation consists of 4 parts:

2. Constructing the circuit
3. Specifying the type of analysis
4. Performing the simulation
5. Evaluating the Data

5.1 Constructing the Circuit

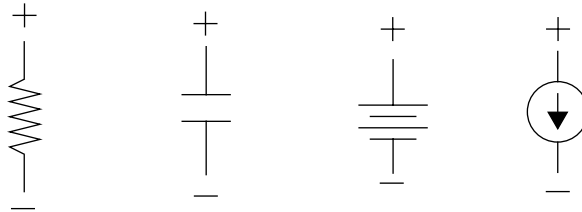
SPICE can simulate a wide variety of circuit elements. In this tutorial, we will introduce the elements which you will encounter in EE105.

5.1.1 Two Terminal Elements

Two terminal elements are specified by the + and - terminals. In cases of passive element, such as resistors and capacitors, there is no distinction between the + and - terminals. For voltage sources and current sources, the order of the nodes determines the polarity of the voltage or the direction of the source.

FIGURE 3.

Passive Two Terminal Elements

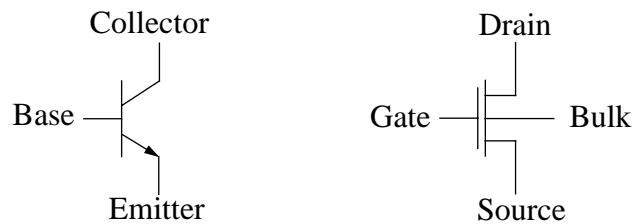


- Resistors (*Rname* <+ node> <- node> value)
- Capacitors (*Cname* <+ node> <- node> value)
- Voltage Source (*Vname* <+ node> <- node> value)
- Current Source (*Iname* <+ node> <- node> value)

5.1.2 Multi-Terminal Elements

FIGURE 4.

Examples of Multiterminal Elements: npn BJT and n-channel MOSFET



- **Bipolar Transistors** (*Qname* <collector node> <base node> <emitter node> model type)

A bipolar transistor is specified by its collector, base and emitter terminals. The model type tells SPICE to refer to that particular model with its specific parameters.

- **MOSFETS** (*Mname* <drain node> <gate node> <source node> <body node> model type, geometry)

A MOS transistor is specified by its drain, gate, source and body terminals along with the name of a model. If the geometry is not specified, it will have a gate length and

width of $1\mu\text{m}$. It is important to note that on a circuit, the body is often not labeled as a “terminal”. Most often, for a NMOS, the body is either tied to ground or to the source and for the PMOS, the body is either tied to the power supply or the source.

Note that whenever you are modeling an active device, you must specify it with a .model statement in SPICE. For example, if you want to model an npn BJT with the following properties: β of 100, V_A of 100V, and $I_S = 1 \times 10^{-15}\text{A}$.

You would include the following statement:

```
.model n1 npn Bf=100 Vaf=100 Is=1e-15
```

Consult the PSPICE manual for the form of .model lines for other active devices.

5.2 Entering the SPICE Deck

You will be using PSPICE for Windows in room 123. In order to enter the SPICE deck, you will need to use a text editor. The editor you should use is **Notepad** found in the **Accessories** group. If you are unfamiliar with the Windows environment, ask your TA for assistance.

Once you have finished entering your SPICE deck, save the file. The filename should be in the format *<your filename>.cir*. The filename should be no more than 8 characters long, not including the cir extension.

Your SPICE deck should have the following format:

- The first line must be a title.
- The end of the deck is defined with a **.end** statement.
- It is often a good practice to group the netlist with the following format:
 1. Title
 2. Circuit Elements
 3. .model statements
 4. analysis and options statements
 5. .end
- note: Any statement that is preceded by “*” is recognized as a comment. It is often a good idea to comment your netlist so that others can understand what you’re doing.
- Nodes can be represented as numbers as well as strings. Use meaningful nodenames. Ground is *always* node 0.

5.3 Running PSPICE

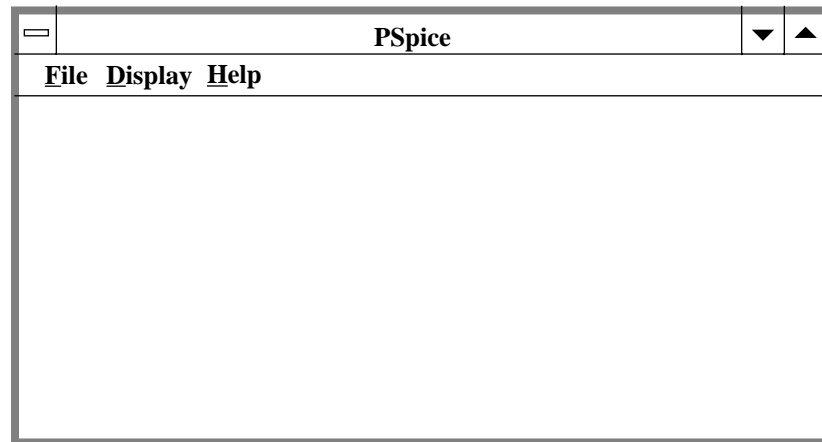
The file you have created in **Notebook** is ready to be simulated. You can open the PSPICE windows by double-clicking its icon. A window which resembles figure 5 will appear.

To simulate your SPICE deck,

1. Click on *File* then *Open*.
2. Select the SPICE deck you want to simulate.

FIGURE 5.

The PSpice Window



Lab Tip

Note the path in which you have saved your SPICE deck. If you don't see your **Notebook** file, select *All Files* in the filter within the dialog box. If the file is still not there, then PSPICE is not looking in the directory in which you saved your .cir file. You will have to traverse up or down the directory tree until you reach the correct directory.

If the circuit contains no error, PSPICE will display a message telling you that the simulation was successful. If there were errors, then you will need to fix it within your SPICE deck. You can view your errors by clicking *File* and then *Examine Output*.

5.4 Running Probe

Probe is a graphing program which plots data obtained from a PSPICE simulation. You can launch Probe by double-clicking its icon. A window which resembles figure 6 will appear.

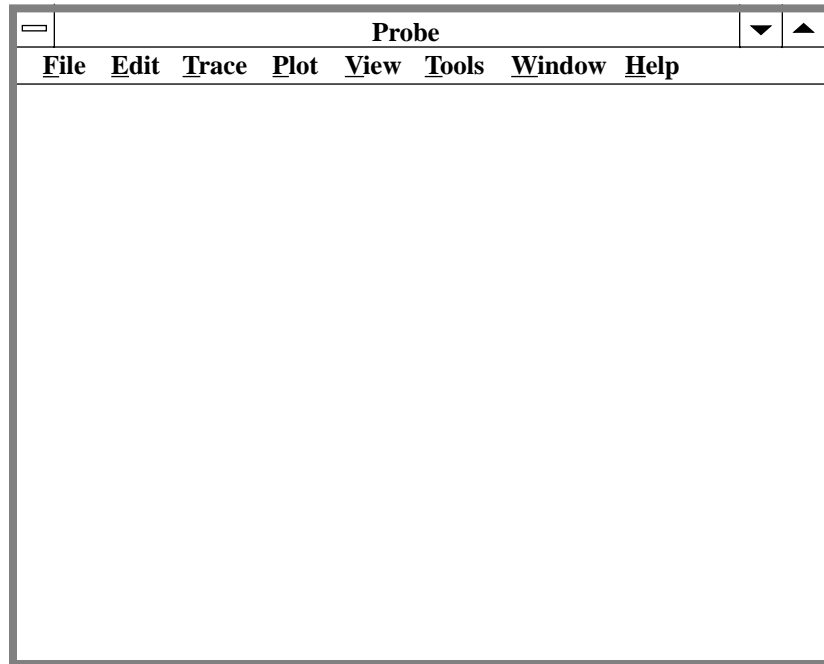
In order to run probe, you must include the following line in your SPICE deck:

```
.probe
```

Data for Probe is stored with the same name as your SPICE deck file, with the .dat extension. For example, if your SPICE deck was named *test.cir*, the data file for Probe will be named *test.dat*.

FIGURE 6.

The Probe Window



Before plotting your data, you must open the .dat file. This can be done in the *File* menu.

If your simulation contains multiple types of analysis (dc, ac or transient) you can specify which type of data to load into probe in the *plot* menu.

Data is plotted using the *Add* function in the *Trace* Menu.

You can obtain a hardcopy of the plot by using the *Print* function in the *File* Menu.

Lab Tip

Label your plots with insightful comments. Somewhere on your plot you should include your name. This can make identifying your plot an easier job.

If the print queue is long, you can convert the plot in *postscript* format. Use the *Printer Select* function in the *File* Menu.
