ECE 230L - LAB 3

INTRODUCTION TO CIRCUIT SIMULATION USING PSPICE

Contents

List of Figures

1 Objectives of this Laboratory

Circuit simulation is an important tool in the analysis and design of microelectronic circuits. Spice is a general-purpose circuit simulation program in which nonlinear dc, nonlinear transient, and linear ac analyses of electronic circuits are carried out. Circuits may contain resistors, capacitors, inductors, mutual inductors, independent voltage and current sources, four types of dependent sources, lossless and lossy transmission lines, switches, uniformly distributed RC lines, and the five most common semiconductor devices: diodes, BJTs, JFETs, MESFETs, and MOSFETs. The version of Spice used in the ECE Department at Duke is PSpice. The objectives of this laboratory session are to introduce you to the basics of PSpice by learning:

- How to set-up your PSpice simulation environment,
- How to represent the circuit elements,
- How to construct the circuits, and
- How to simulate the circuits.

2 Setting Up a Circuit Using ORCAD Capture

1. Open ORCAD Capture CIS
2. Create a new project by selecting File → New → Project
3. New your project 'Lab 3'
4. Choose Analog or Mixed A/D under the Create a New Project Using menu
5. Select Create a blank project when prompted

Once the new project has been created, circuit design can begin. Sources, components, ground nodes, and wires can be selected using the Place menu.

PSpice will be used to model the circuit in Figure 1 and perform DC, AC, and transient analysis on the circuit.

To make the circuit,

1. Add a DC Voltage Source by following Place → PSPice Component → Source → Voltage Source → DC

Add a DC voltage source to the circuit by following Place → PSpice Component → Source → Voltage Sources → DC. After adding the voltage source to the schematic, use the Place → PSpice Components → Passives menu to insert the remaining resistors and capacitors. Use Ctrl-R to rotate the components. Use Place → Wire to connect the circuit nodes. To change values of circuit elements, double click on the element and adjust the desired properties. Finally, add a ground node to the circuit schematic. Follow Place → Ground and select 0/SOURCE as your ground node.

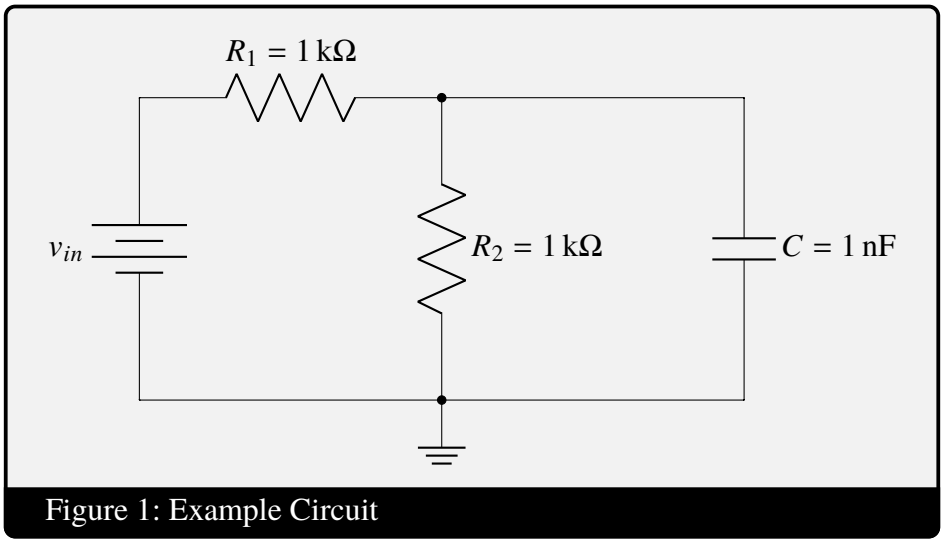

Figure 1: Example Circuit

3 DC Analysis in PSpice

To perform a DC analysis of the circuit, you will create a new simulation profile. To create a new profile select PSpice → New Simulation Profile. Name the new profile 'dc' and press Create. To analyze the example circuit, select 'DC Sweep' in the Analysis Type drop down menu and use the following parameters:

- Sweep variable > Voltage source: V1
- Sweep Type: Linear
- Start Value: 0
- End Value: 10
- Increment: 0.01

Press 'Apply' and 'OK' to save the profile settings. Begin the simulation by selecting PSpice-> Run. To view the circuit behavior at a particular point, follow Trace-> Add Trace to select different values to plot or use the voltage and current markers indicated in Figure 1. Plot source voltage, V(R2), I(R1), and I(R2). Figure 3 shows the circuit schematic and figure 4 shows the result of DC analysis (top plot: current, bottom plot: voltage).

Grading Rubric

Table 1: ECE 230L Laboratory 3 Grading Rubric

Criteria	Points Possible
DC Analysis	**10**
Circuit Diagram	5
Waveforms	5
AC Analysis	**10**
Circuit Diagram	5
Waveforms	5
Transient Analysis	**10**
Circuit Diagram	5
Waveforms	5
Practice Exercise	**35**
Circuit Diagram	5
DC Analysis	10
AC Analysis	10
Transient Analysis	10
Thevenin Equivalent Example Circuit	**20**
Circuit Diagram	10
V_{OC} and I_{SC} Labeled	5
Correct R_{TH} Value	5
Thevenin Equivalent Challenge Circuit	**15**
Circuit Diagram	5
V_{OC} and I_{SC} Labeled	5
Correct R_{TH} Value	5
Total	100