

ECE 230L - LAB 3

INTRODUCTION TO CIRCUIT SIMULATION USING PSpICE

Contents

1	Objectives of this Laboratory	2
2	Setting Up a Circuit Using ORCAD Capture	2
3	DC Analysis in PSpice	3
4	AC Analysis in PSpice	4
4.1	Trace Expressions in PSpice	4
5	Transient Analysis in PSpice	4
6	Practice Example	4
7	Exploration: Thevenin Equivalent Circuits	4
7.1	Purpose	4
7.2	Introduction	4
7.3	Exercise	4
7.4	Practice Exercise: Thevenin Equivalent Circuit	4
	Grading Rubric	5

List of Figures

1	Blank Schematic	2
2	Example Circuit	3

1 Objectives of this Laboratory

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

To create a circuit in a PSpice environment, one must first launch ORCAD:

1. Open ORCAD Capture CIS
2. Create a new project by selecting **File** → **New** → **Project**
3. Name 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

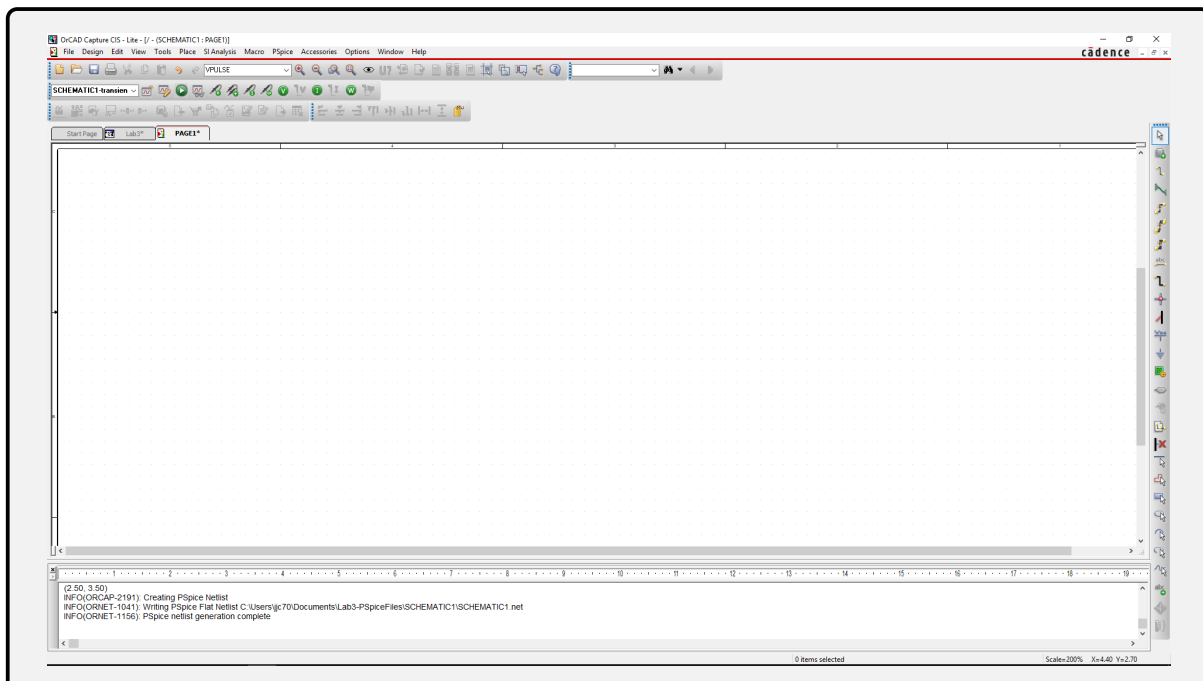
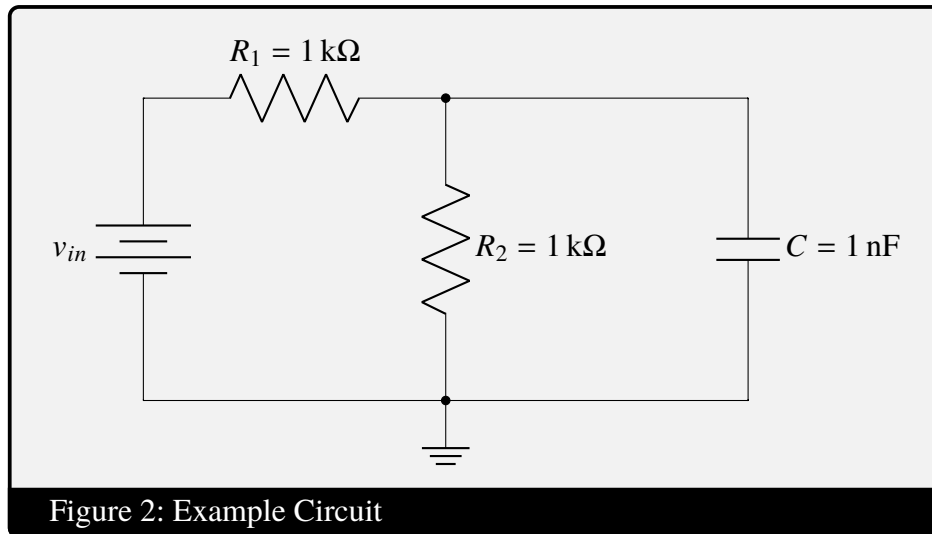


Figure 1: Blank Schematic

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 2 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.

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).

4 AC Analysis in PSpice

4.1 Trace Expressions in PSpice

5 Transient Analysis in PSpice

6 Practice Example

7 Exploration: Thevenin Equivalent Circuits

7.1 Purpose

7.2 Introduction

7.3 Exercise

7.4 Practice Exercise: Thevenin Equivalent Circuit

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