

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	5
4.1	Trace Expressions in PSpice	6
5	Transient Analysis in PSpice	7
6	Practice Example	8
7	Exploration: Thévenin Equivalent Circuits	9
7.1	Purpose	9
7.2	Introduction	9
7.3	Exercise	9
7.4	Practice Exercise: Thévenin Equivalent Circuit	10
	Grading Rubric	11

List of Figures

1	Blank Schematic	2
2	Example Circuit	3
3	Result of DC Analysis	4
4	Circuit for AC Analysis	5
5	Settings for AC Analysis	6
6	Result of AC Analysis	6
7	Circuit for Transient Analysis	7
8	Result of Transient Analysis	8
9	Practice Circuit	8
10	Thévenin Equivalent Circuit	9

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

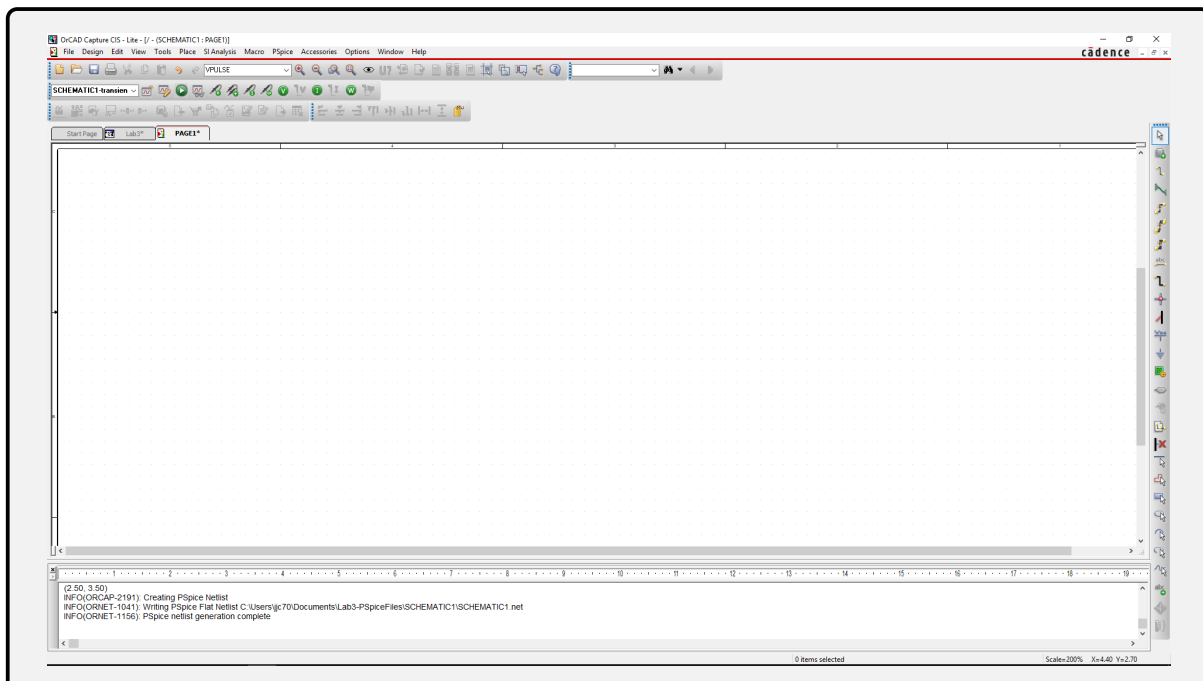
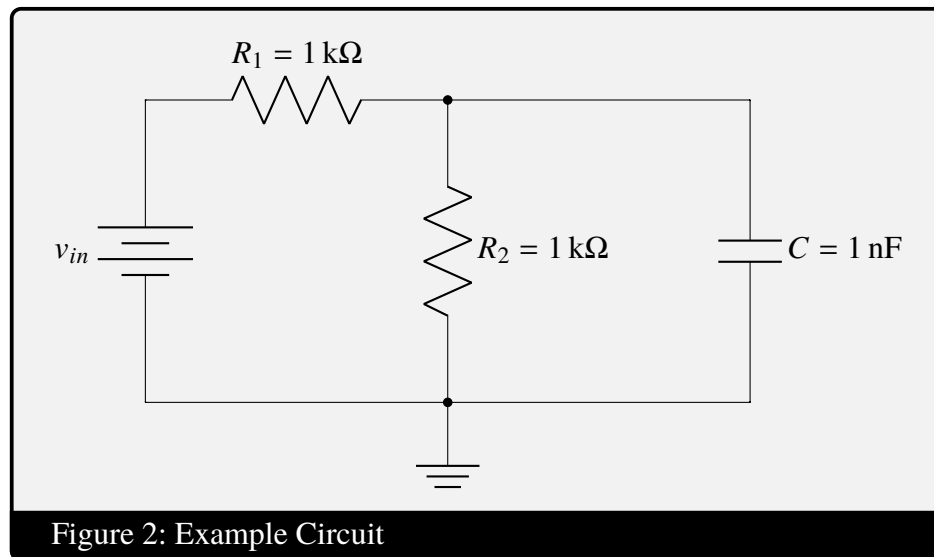


Figure 1: Blank Schematic

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. Plot V(R1:1), V(R1:2), I(R1), and I(R2). Figure 2 shows the circuit schematic and Figure 3 shows the result of DC analysis (top plot: current, bottom plot: voltage).

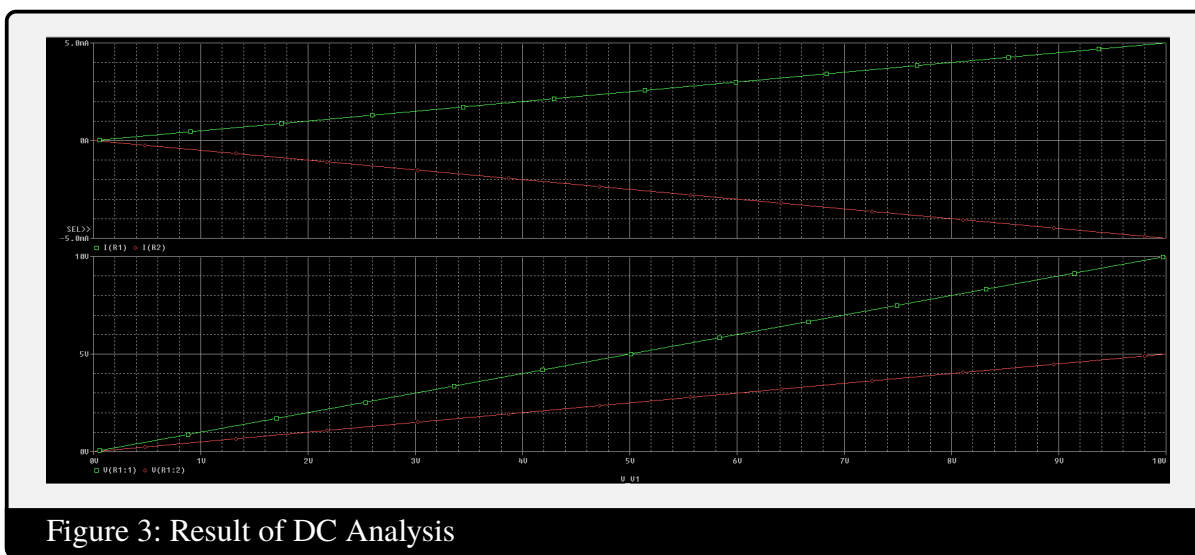


Figure 3: Result of DC Analysis

4 AC Analysis in PSpice

Before performing an AC analysis a new AC voltage source has to replace the DC source. To change the source, delete the DC source and follow **Place** → **PSpice Component** → **Source** → **AC**. We include the circuit in Figure 4. The following parameters will be used:

- DC Value: 10
- AC Amplitude: 1

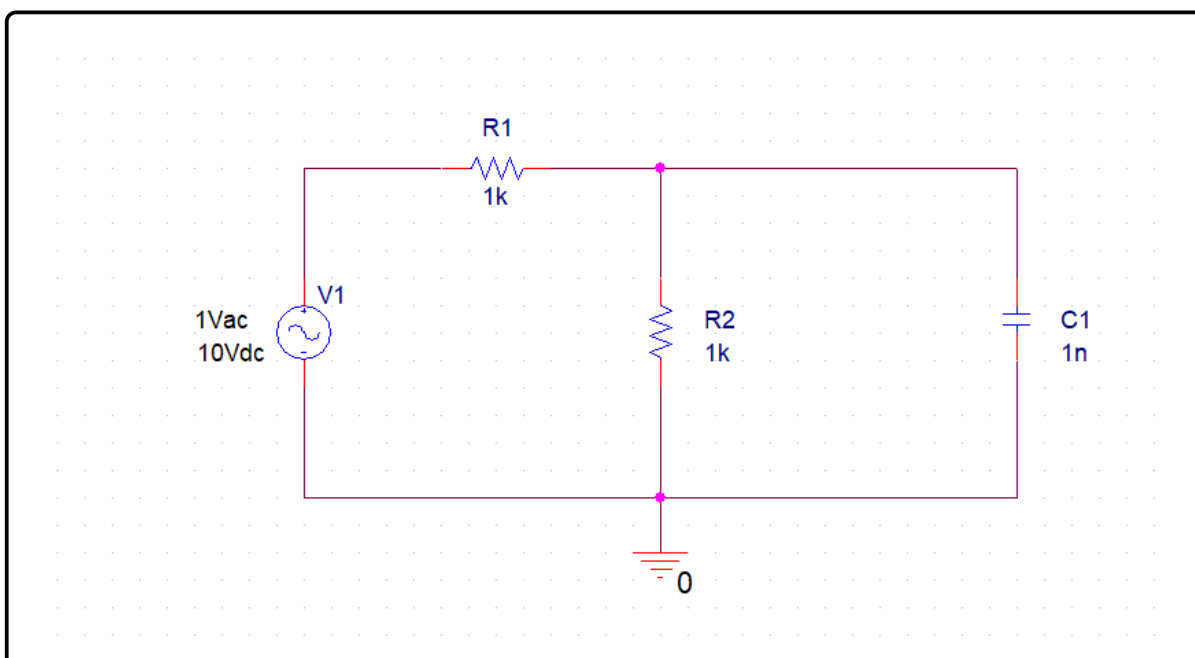


Figure 4: Circuit for AC Analysis

After the voltage source properties have been changed, AC analysis can be performed. First, create a new simulation profile called 'ac'. Next, select **AC Sweep/Noise** in the **Analysis Type** drop down menu and use the following parameters:

- AC Sweep Type: Logarithmic
 - Select **Decade** from drop down box below
- Start Frequency: 1k
- Stop Frequency: 1Meg
- Number of Points per Decade: 10

Press **Apply** and **OK** to save the profile settings. Begin the simulation by selecting **PSpice** → **Run**. A screenshot of the settings for the AC Analysis is included in Figure 5.

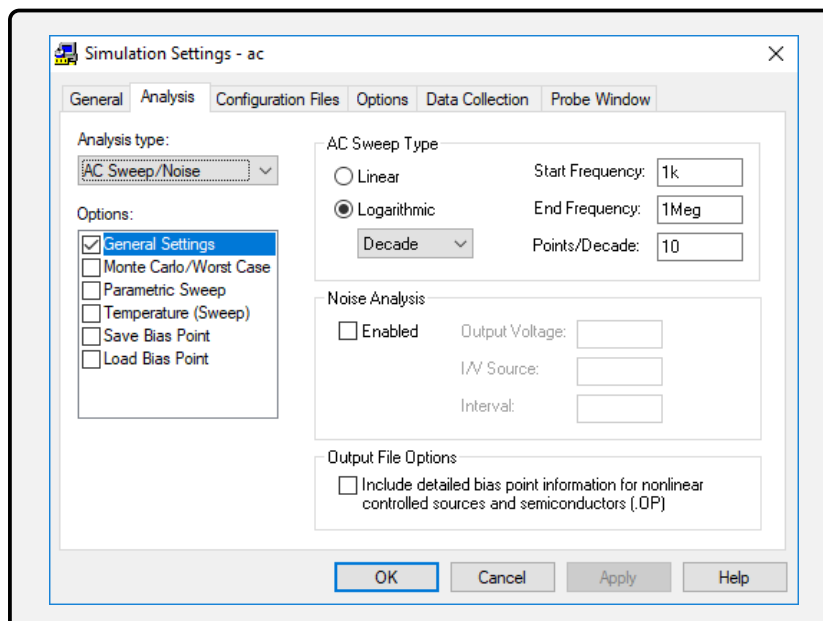
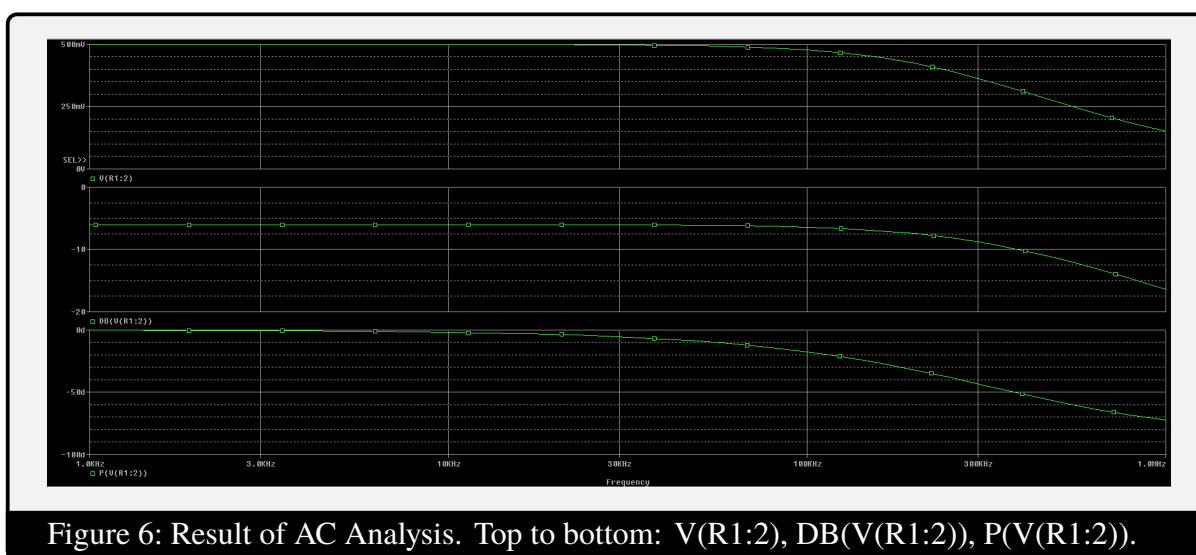


Figure 5: Settings for AC Analysis

4.1

Trace Expressions in PSpice

Trace expressions can be used to plot the phase of a desired value, a parameter in units of dB, or plot other useful mathematical operations on circuit parameters. Trace expressions are available under the **Trace** → **Add Trace** menu. Select **Plot** → **Add Plot to Window** and plot the value of $V(R2)$ in dB. Use the trace expression $DB(V(R1:2))$. Next, add another plot to the window and plot the phase of $V(R2)$. Use the trace expression $P(V(R1:2))$. The circuit for AC analysis is shown in Figure 4 and the resulting plots are shown in Figure 6.

Figure 6: Result of AC Analysis. Top to bottom: $V(R1:2)$, $DB(V(R1:2))$, $P(V(R1:2))$.

5 Transient Analysis in PSpice

Before performing a transient analysis replace the AC source with a Pulse source (Place-> PSpice Component-> Source-> Pulse). The following parameters will be used to set up the pulse:

- V1: 0
- V2: 5
- TD: 10n
- TR: 20n
- TF: 20n
- PW: 500n
- PER: 2u

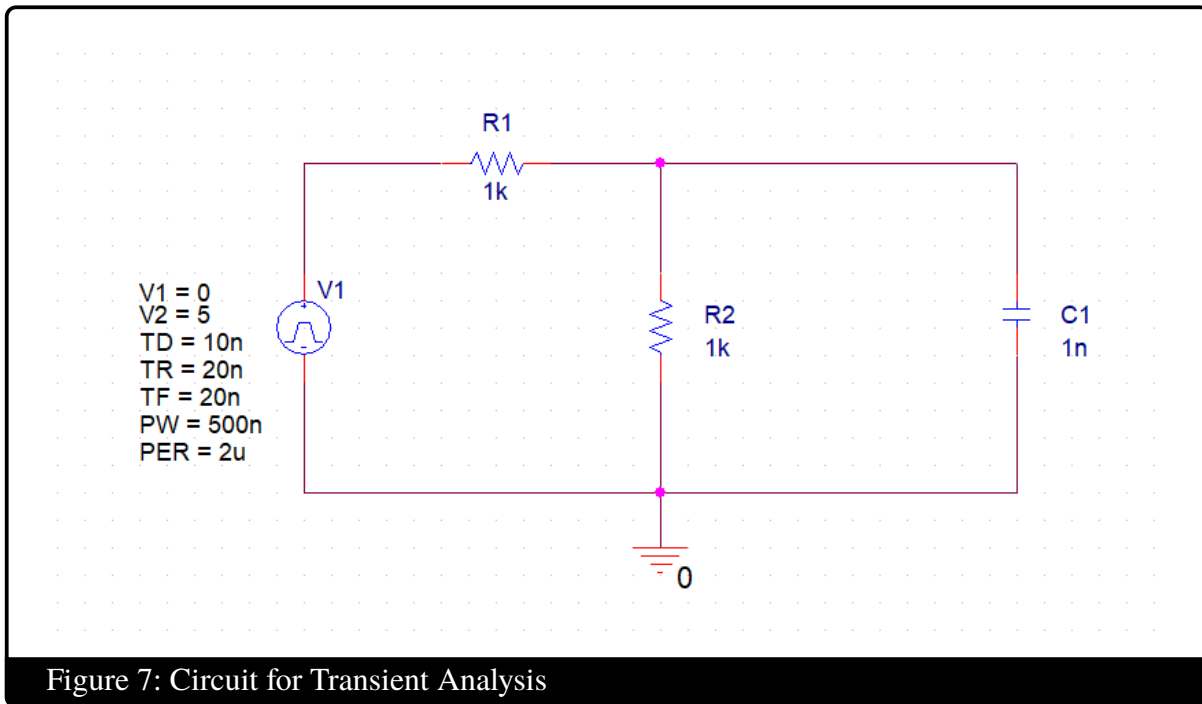


Figure 7: Circuit for Transient Analysis

Create a new simulation profile called ?Transient?. To analyze the example circuit, select ?Time Domain (Transient)? in the Analysis Type drop down menu and use the following parameters: ? Run to Time: 2u ? Start saving data after: 0 ? Maximum step size: 10n 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 the source voltage and V(R2). The circuit for transient analysis is shown in figure 7 and the resulting plot is shown in figure 8.

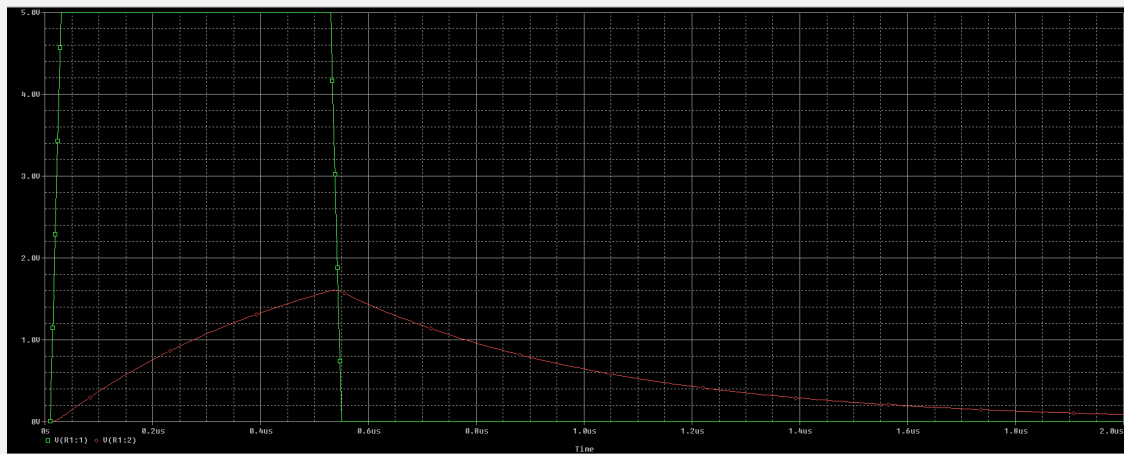


Figure 8: Result of Transient Analysis

6 Practice Example

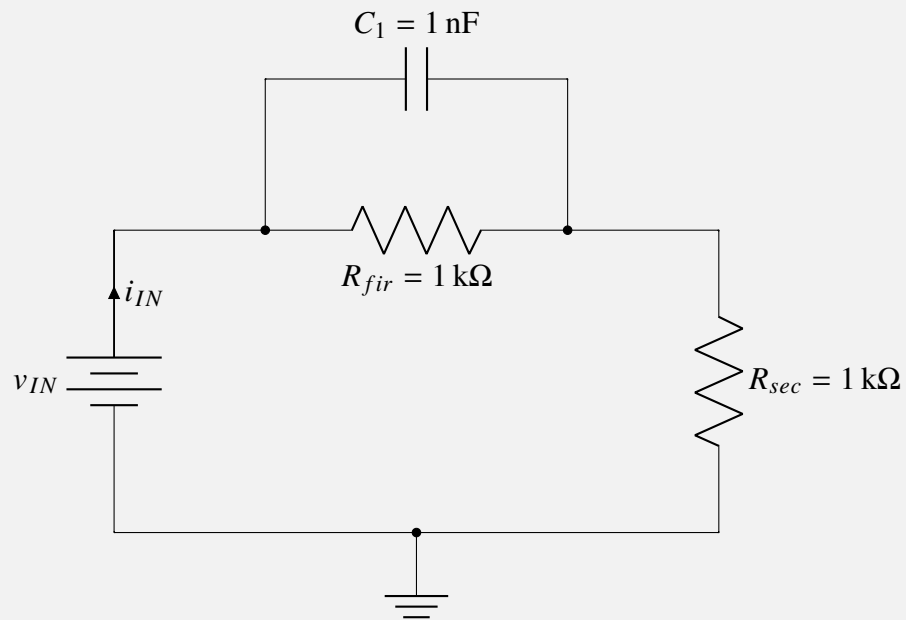


Figure 9: Practice Circuit

7 Exploration: Thévenin Equivalent Circuits

7.1 Purpose

The purpose of this exercise is to learn how to form a Thévenin Equivalent circuit by using circuit parameters obtained during simulation.

7.2 Introduction

Any linear DC circuit as seen at a pair of terminals can be reduced to a practical voltage source (an ideal voltage source in series with a resistor).

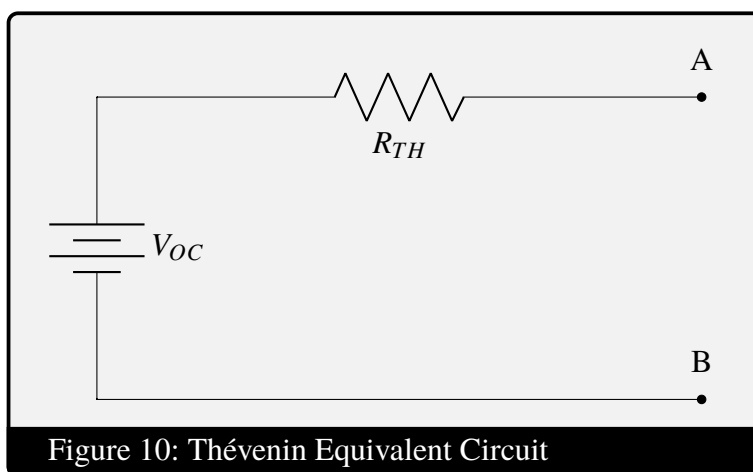


Figure 10: Thévenin Equivalent Circuit

To form a Thévenin Equivalent circuit, two quantities must be calculated, measured, or simulated:

1. v_{oc} : The open circuit voltage drop from terminals A to B
2. i_{sc} : The short circuit current from terminals A to B

Once the values for v_{oc} and i_{sc} have been obtained, the Thévenin resistance R_{TH} can be determined using the relation:

$$R_{TH} = \frac{v_{oc}}{i_{sc}} \quad (1)$$

If the circuit contains no dependent sources, then R_{TH} may also be found by turning off all of the independent sources and using resistance reduction at terminals A and B.

7.3 Exercise

You will simulate the circuit in figure 11 and form its Thévenin Equivalent circuit as seen at a pair of terminals.

7.4**Practice Exercise: Thévenin 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
Thévenin Equivalent Example Circuit	20
Circuit Diagram	10
V_{OC} and I_{SC} Labeled	5
Correct R_{TH} Value	5
Thévenin Equivalent Challenge Circuit	15
Circuit Diagram	5
V_{OC} and I_{SC} Labeled	5
Correct R_{TH} Value	5
Total	100