ECE 230L - LAB 4

INTRODUCTION TO CIRCUIT SIMULATION USING PSPICE

Contents

1 Objectives of this Laboratory **3**

2 Setting Up a Circuit Using ORCAD Capture **3**

3 DC Analysis in PSpice **5**

4 AC Analysis in PSpice **6**

5 Transient Analysis in PSpice **7**

6 Practice Example **9**

7 Exploration: Thévenin Equivalent Circuits **11**
7.1 Example Exercise 12
7.2 Challenge Exercise: Thévenin Equivalent Circuit 14

Grading Rubric **15**

List of Figures

1 Blank Schematic 3
2 Initial Circuit 4
3 Circuit for DC Analysis 5
4 Result of DC Analysis 5
5 Circuit for AC Analysis 6
6 Settings for AC Analysis 7
7 Result of AC Analysis 7
8 Circuit for Transient Analysis 8
9 Settings for Transient Analysis 9
10 Result of Transient Analysis 9
11 Practice Circuit 10
12 Thévenin Equivalent Circuit 11
13 Example Circuit 12
14 Open Circuit Voltage Schematic 12
15 Short Circuit Voltage Schematic 13
16 Thévenin Equivalent Schematic 13

17 Exercise Circuit 14

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

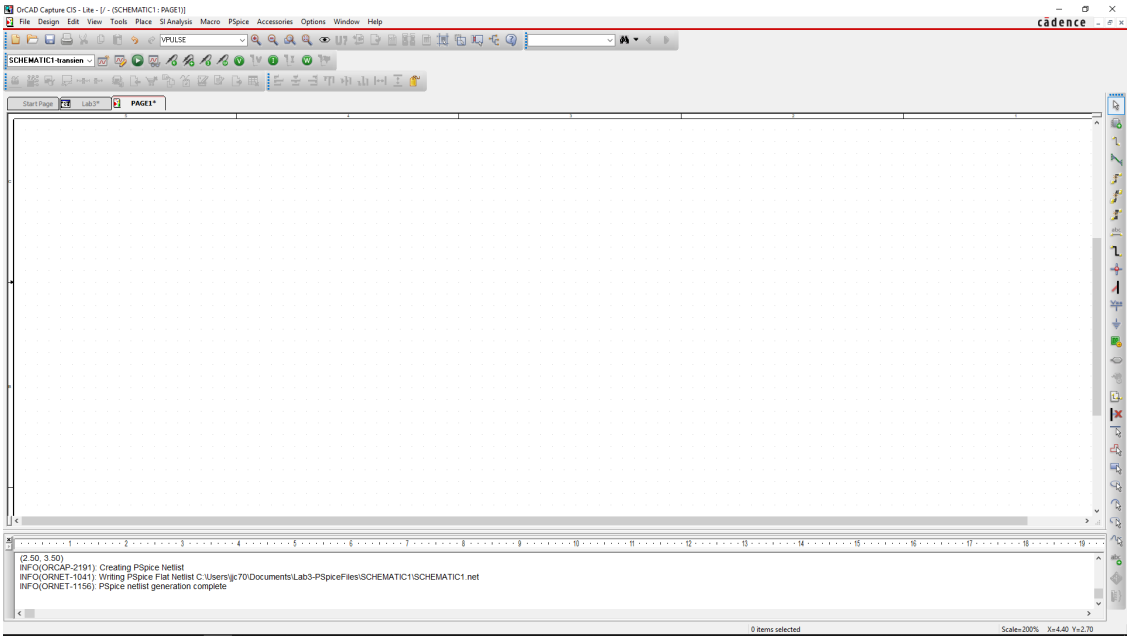

Figure 1: Blank Schematic

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 and perform analyses on the circuit in Figure 2.

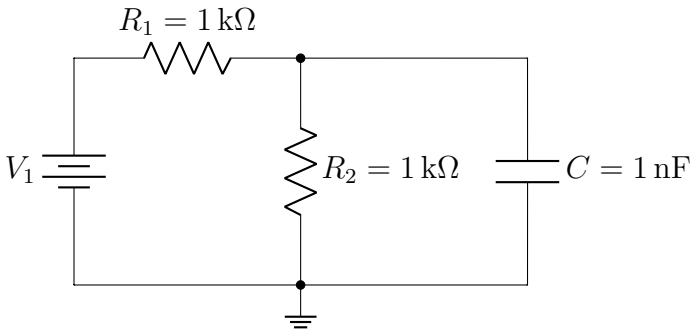

Figure 2: Initial Circuit

To create the circuit,

1. Add a DC Voltage Source: Place → PSPice Component → Source → Voltage Source → DC
2. Insert Resistors and Capacitors: Place → PSpice Component → Passives
3. Insert Wires and connect circuit nodes: Place → Wire
4. Insert a Ground: Place → Ground → 0/SOURCE

Note that you can use Ctrl-R to rotate the components. Also, to change values of any component, double click on the component and change the values using the pop-up menu. You may also change values by double clicking directly on the value and typing a new value.

3 DC Analysis in PSpice

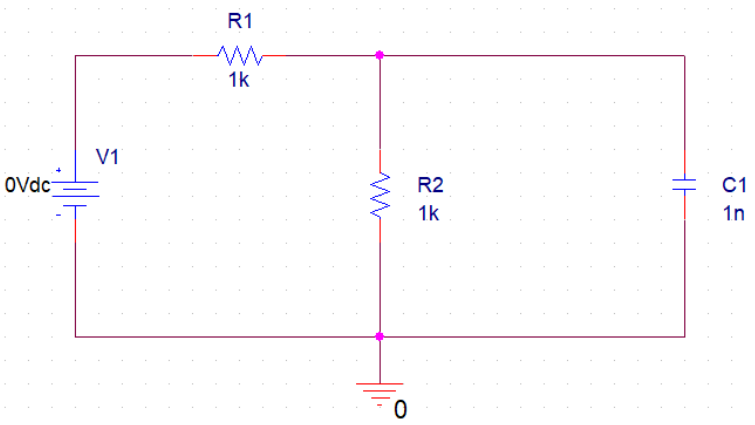

Figure 3: Circuit for DC Analysis

To perform a DC analysis of the circuit, create a new simulation profile by selecting **PSpice** → **New Simulation Profile**. Name the new profile 'dc' and press **Create**. Then, 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.

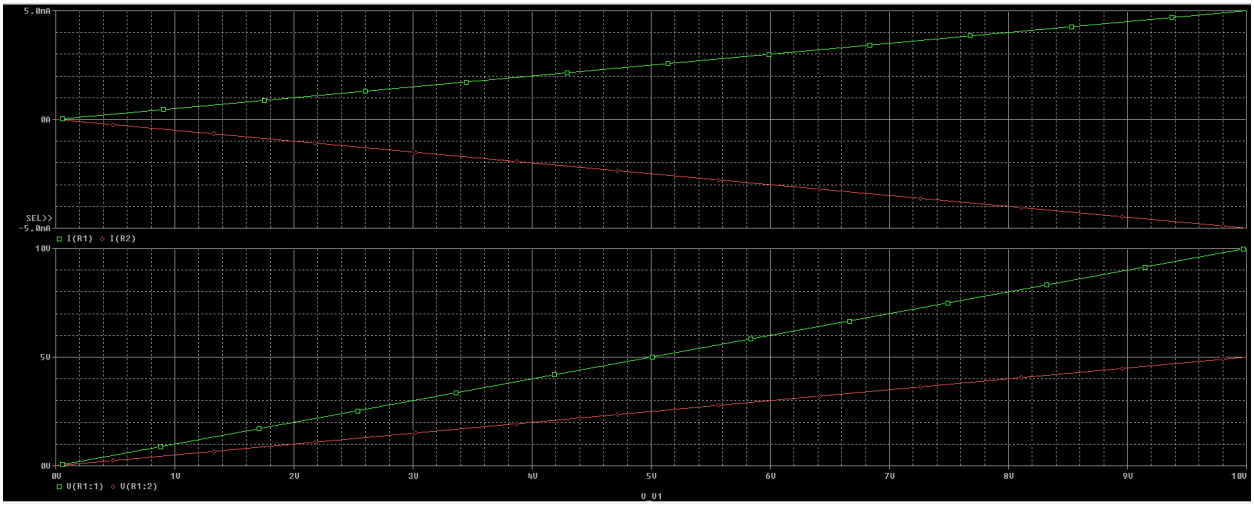

Figure 4: Result of DC Analysis. Top: Current; Bottom: Voltage

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**. The following parameters will be used:

- DC Value: 10
- AC Amplitude: 1

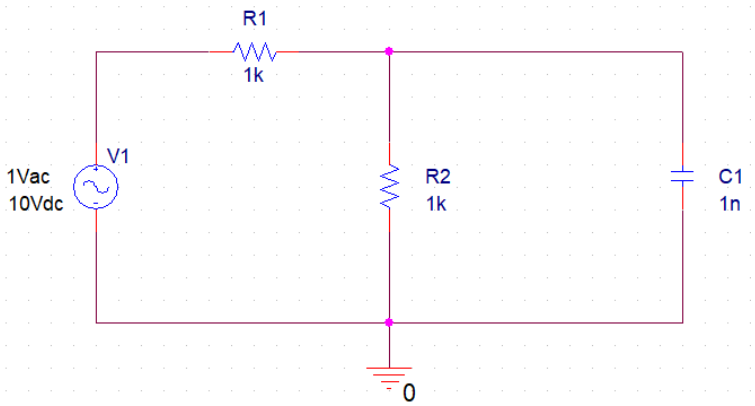

Figure 5: Circuit for AC Analysis

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 menu below
- Start Frequency: 1k
- Stop Frequency: 1Meg
- Number of Points per Decade: 10

Press **Apply** and **OK** to save the profile settings. They should look like Figure 6.

Now, begin the simulation by selecting **PSpice** → **Run**.

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(R1:2) in dB. Use the trace expression DB(V(R1:2)). Next, add another plot to the window and plot the phase of V(R1:2). Use the trace expression P(V(R1:2)). The resulting plots are shown in Figure 7.

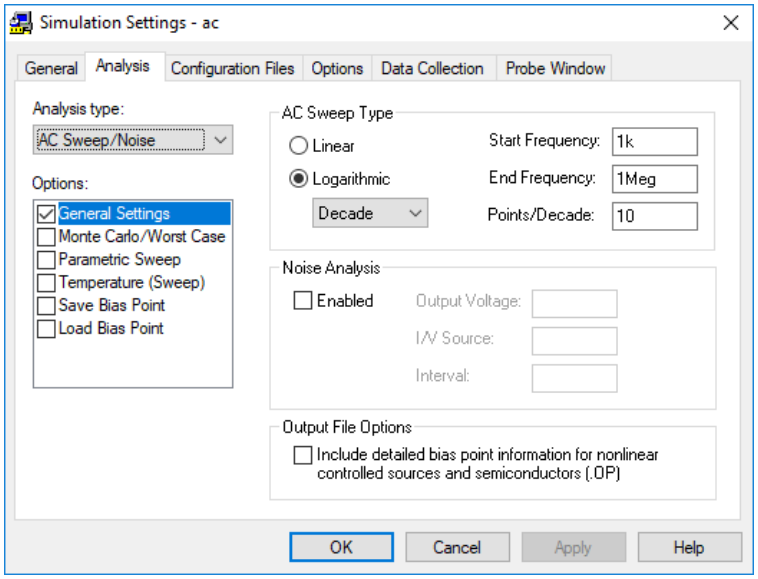

Figure 6: Settings for AC Analysis

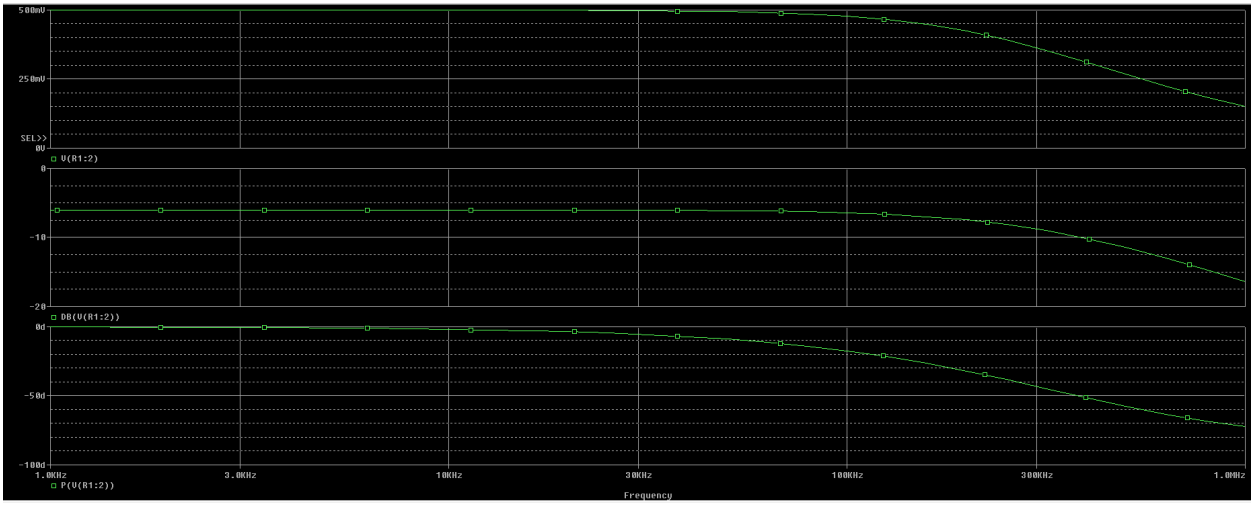

Figure 7: Result of AC Analysis

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

We include the circuit in Figure 8.

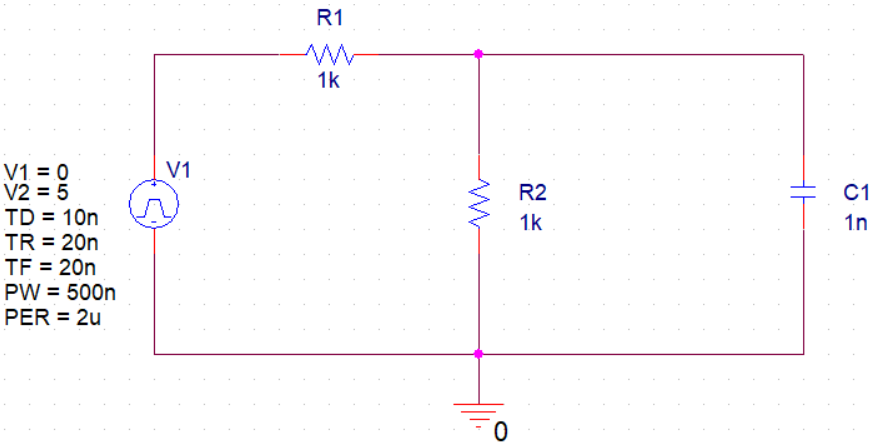

Figure 8: Circuit for Transient Analysis

After creating the circuit, create a new simulation profile called "Transient". To analyze the new 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**. Plot the source voltage, V(R1:1), and V(R1:2).

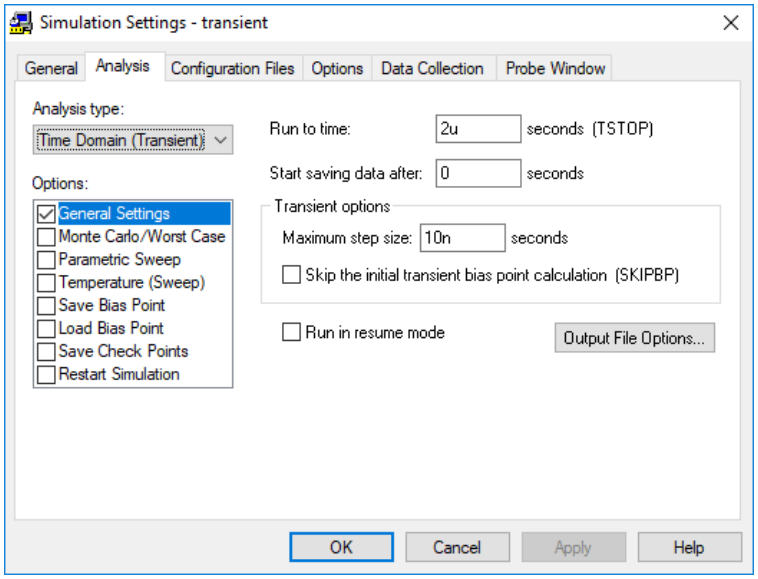

Figure 9: Settings for Transient Analysis

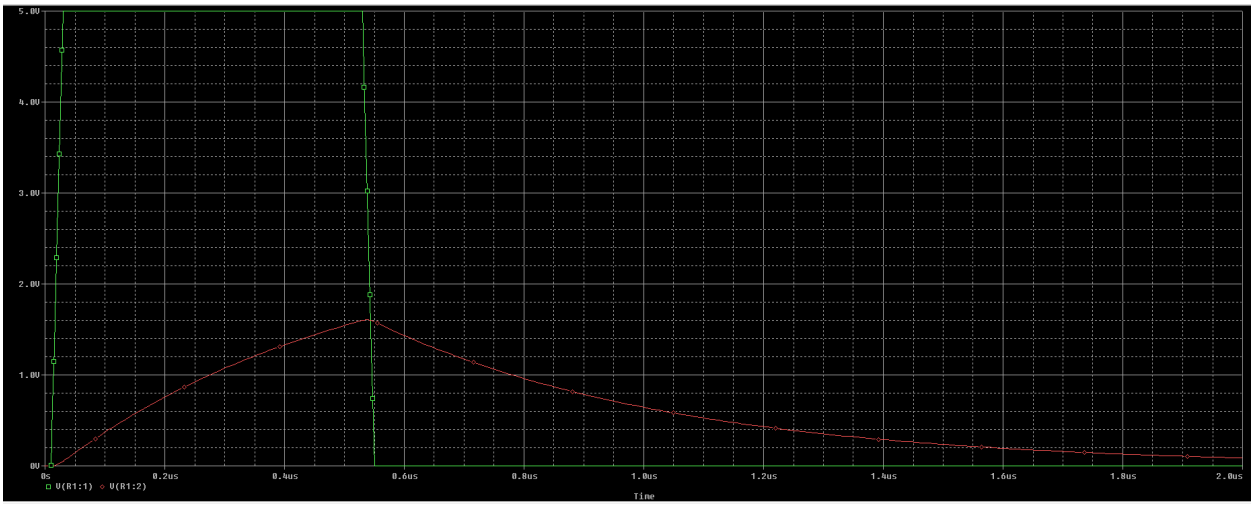

Figure 10: Result of Transient Analysis

6 Practice Example

Use PSpice to simulate the circuit shown in Figure 11 using DC, AC, and Transient analyses. Take a screencap of a plot of the voltage across capacitor C_1 in each analysis.

1. In the DC analysis, use the following parameters:
    - Sweep Type: Linear
    - Start Value: 0
    - End Value: 10
    - Increment: 0.01

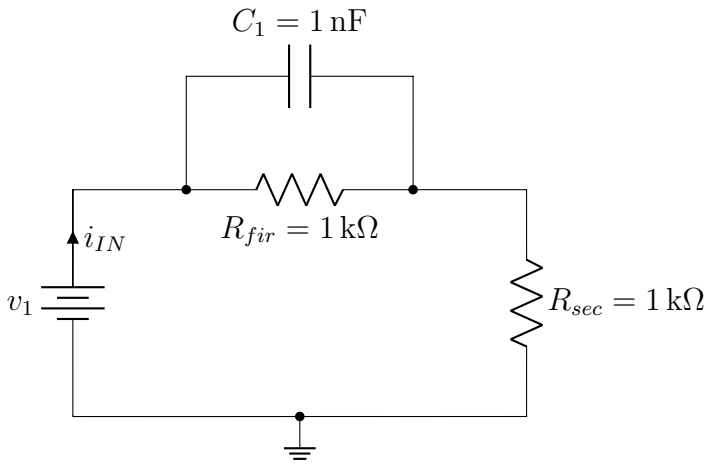

Figure 11: Practice Circuit

2. In the AC analysis, change the voltage source to have the following values:
   - DC Value: 5
   - AC Amplitude: 1

   And use the following parameters:
   - AC Sweep Type: Logarithmic
   - Start Frequency: 10
   - Stop Frequency: 1Meg
   - Number of Points per Decade: 10
3. In the transient analysis, change the voltage source to have the following values:
   - V1 = 0,
   - V2 = 5,
   - TD = 100n,
   - TR = 40n,
   - TF = 40n,
   - PW = 500n,
   - PER = 2u

   And use the following parameters:
   - Run to Time: 2u
   - Start saving data after: 50n
   - Maximum step size: 10n

7 Exploration: Thévenin Equivalent Circuits

Purpose

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

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).

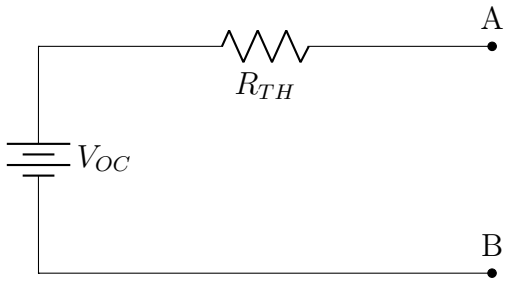

Figure 12: Thévenin Equivalent Circuit

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

- v_{oc}: The open circuit voltage drop from terminals A to B
- 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:

$$R_{TH} = \frac{v_{oc}}{i_{sc}} \tag{1}$$

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.

7.1 Example Exercise

Simulate the circuit in Figure 13 and form its Thévenin Equivalent as seen from terminals A and B.

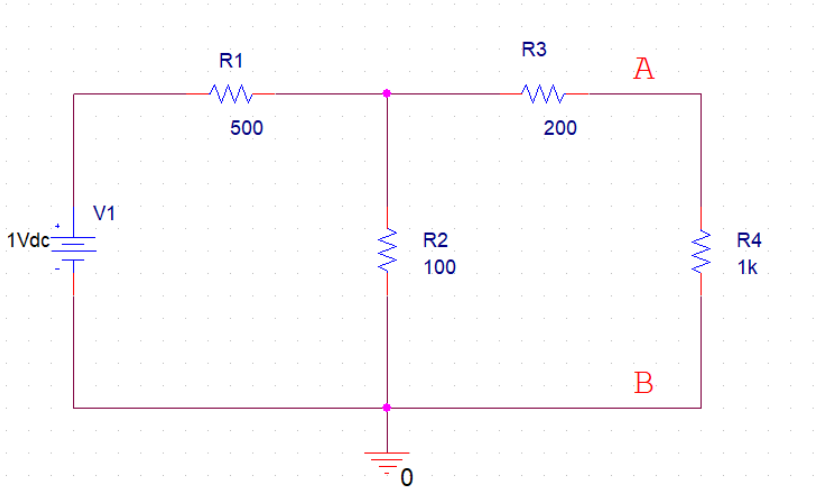

Figure 13: Example Circuit

To find the open circuit voltage, replace resistor R_4 with a large "dummy" resistance (at least 100 MΩ). Create a new simulation profile called "thev". Choose **Bias Point** under the **Analysis type** menu and check the box labeled **Include Detailed Bias Point Information** under the **Output File Options** heading. Press **Apply** and **OK** to save the profile. Now, run the simulation. Once the simulation is complete, follow **PSpice** → **Bias Points** and click **Enable**. Record the voltage across terminals A and B as your open circuit voltage.

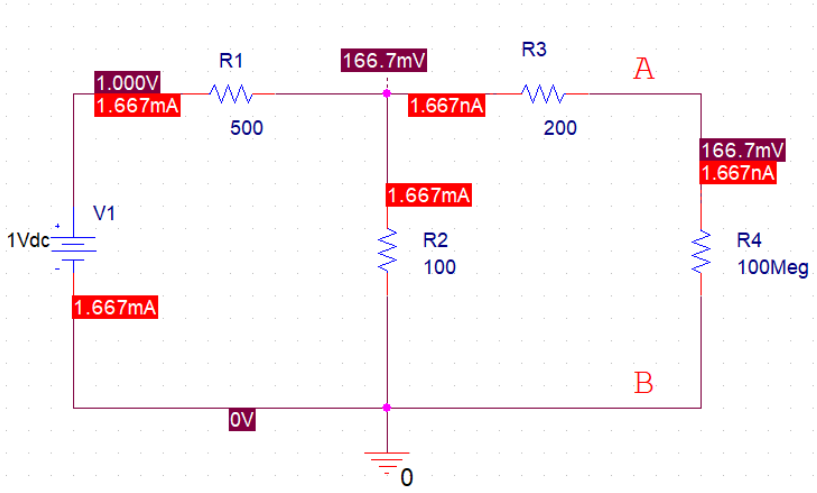

Figure 14: Open Circuit Voltage Schematic

To find the short circuit current, remove R_4 and connect terminals A and B with a wire. Use the same "thev" simulation profile that was created to find v_{oc}. Run the simulation. Once

the simulation is complete, enable bias points. This will show the node voltages and currents throughout the circuit. Record the current between terminals A and B as your short circuit current, i_{sc}.

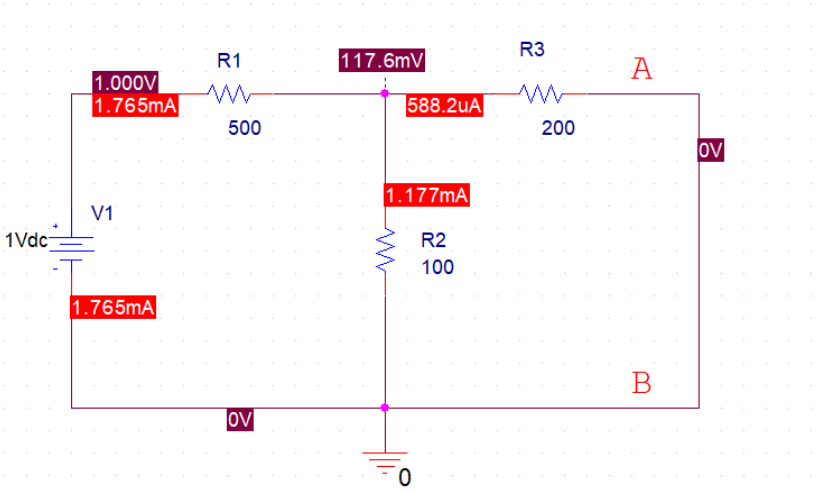

Figure 15: Short Circuit Voltage Schematic

Calculate the Thévenin resistance using Equation 1. Now, re-create Figure 12 in PSpice using your v_{oc} and R_{TH} values. Place R_4 back into your first circuit and place a resistor of equal value between terminals A and B in your Thévenin Equivalent. Run the simulation using the same "thev" profile. Once the simulation is complete, enable bias points. Check to make sure the terminal voltages and currents match for both circuits.

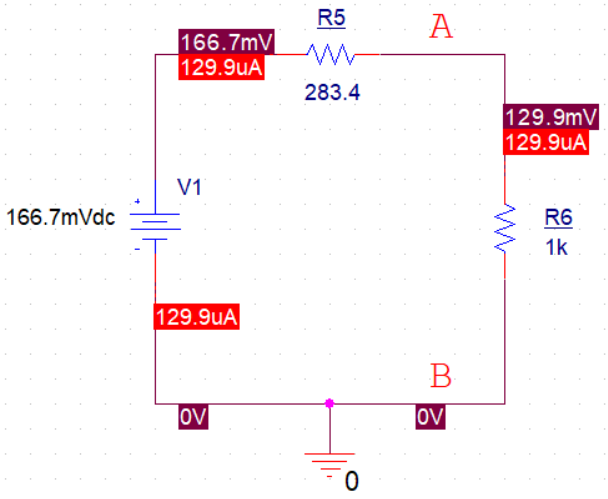

Figure 16: Thévenin Equivalent Schematic

7.2 Challenge Exercise: Thévenin Equivalent Circuit

Create the schematic in Figure 17 using PSpice and find its Thévenin Equivalent circuit as seen from nodes A and B. In your lab report, be sure to include all voltages and currents present in the Exercise Circuit and its Thévenin Equivalent, as well as a short explanation of each step you took in finding the Thévenin Equivalent.

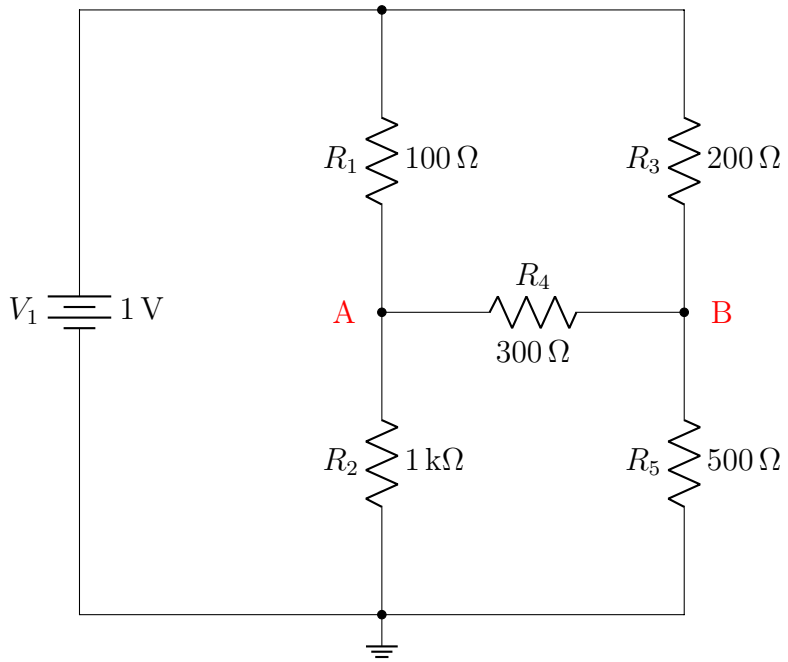

Figure 17: Exercise Circuit

Table 1: ECE 230L Laboratory 4 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