

# ECE 230L - LAB 4

## INTRODUCTION TO CIRCUIT SIMULATION USING PSpICE

---

### Contents

<b>1</b>	<b>Objectives of this Laboratory</b>	<b>2</b>
<b>2</b>	<b>Setting Up a Project Using ORCAD Capture</b>	<b>2</b>
<b>3</b>	<b>DC Analysis in PSpice</b>	<b>3</b>
<b>4</b>	<b>AC Analysis in PSpice</b>	<b>5</b>
<b>5</b>	<b>Transient Analysis in PSpice</b>	<b>7</b>
<b>6</b>	<b>Practice Example</b>	<b>9</b>
<b>7</b>	<b>Exploration: Thévenin Equivalent Circuits</b>	<b>11</b>
7.1	Example Exercise . . . . .	12
7.2	Challenge Exercise: Thévenin Equivalent Circuit . . . . .	14
	<b>Grading Rubric</b>	<b>15</b>

### List of Figures

1	Blank Schematic . . . . .	2
2	Circuit for DC Analysis . . . . .	3
3	Result of DC Analysis . . . . .	4
4	Circuit for AC Analysis . . . . .	5
5	Result of AC Analysis . . . . .	6
6	Circuit for Transient Analysis . . . . .	7
7	Result of Transient Analysis . . . . .	8
8	Practice Circuit . . . . .	9
9	Thévenin Equivalent Circuit . . . . .	11
10	Example Circuit . . . . .	12
11	Open Circuit Voltage Schematic . . . . .	12
12	Short Circuit Voltage Schematic . . . . .	13
13	Thévenin Equivalent Schematic . . . . .	13
14	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 Project 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 4'
4. Choose **Analog or Mixed A/D** under the **Create a New Project Using** menu
5. Select **Create a blank project** when prompted

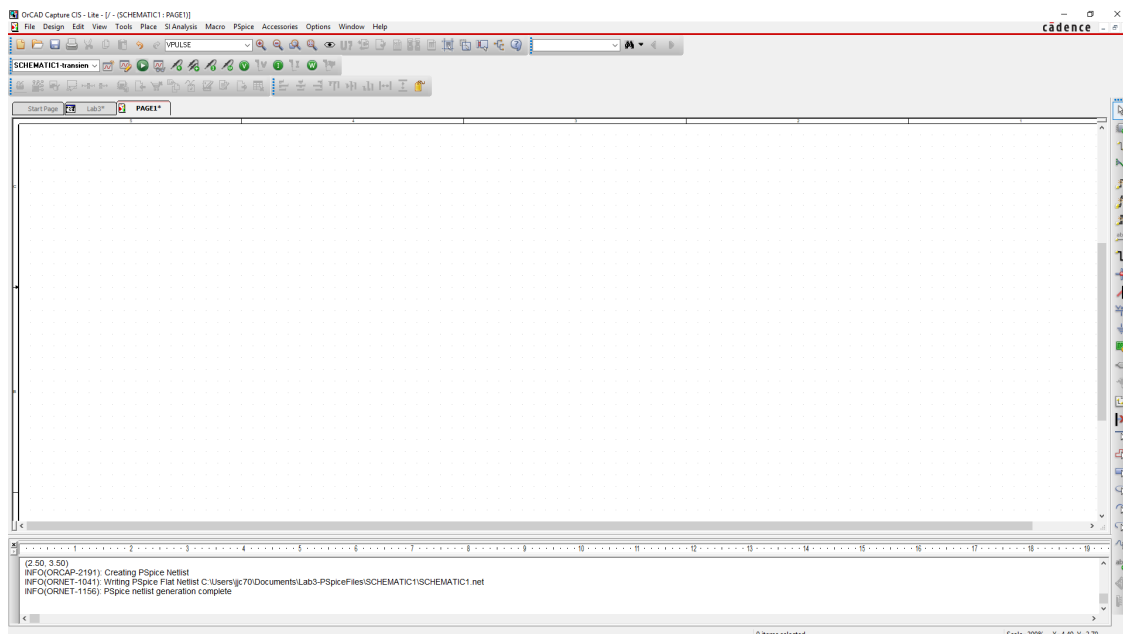


Figure 1: Blank Schematic

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

### 3 DC Analysis in PSpice

PSpice will be used to model and perform analyses on the circuit in Figure 2. To create the circuit, follow the steps below:

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**

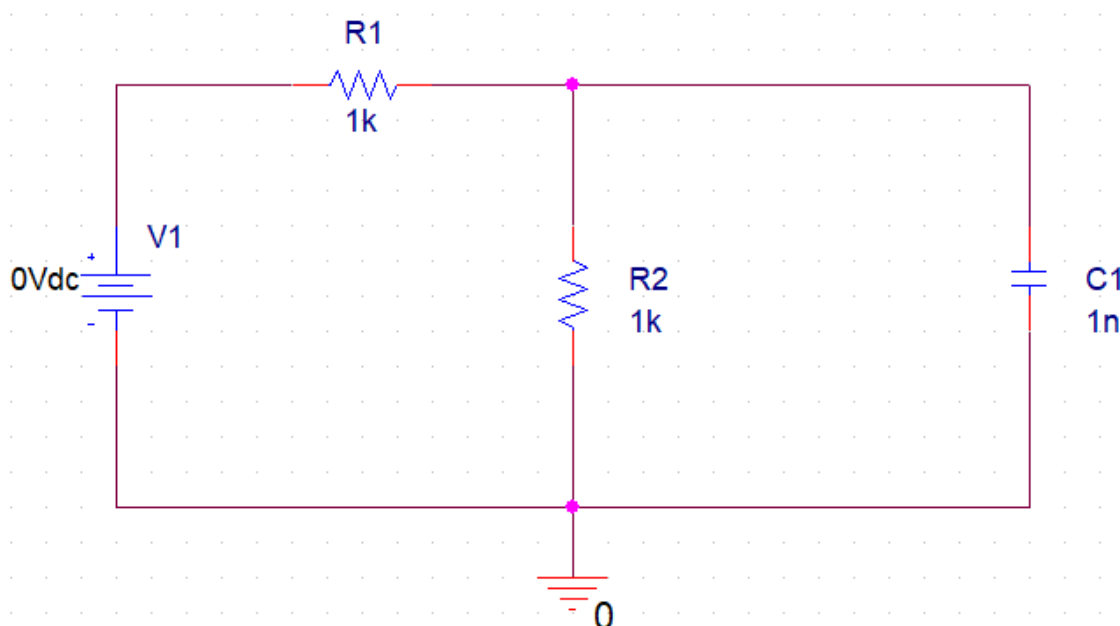


Figure 2: Circuit for DC Analysis

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.

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.

Begin the simulation by selecting **PSpice** → **Run**. To view the circuit behavior at a particular point in time, follow **Trace** → **Add Trace** to select different values to plot. Plot  $V(R1:1)$ ,  $V(R1:2)$ ,  $I(R1)$ , and  $I(R2)$ . Figure 3 shows the result of DC analysis.

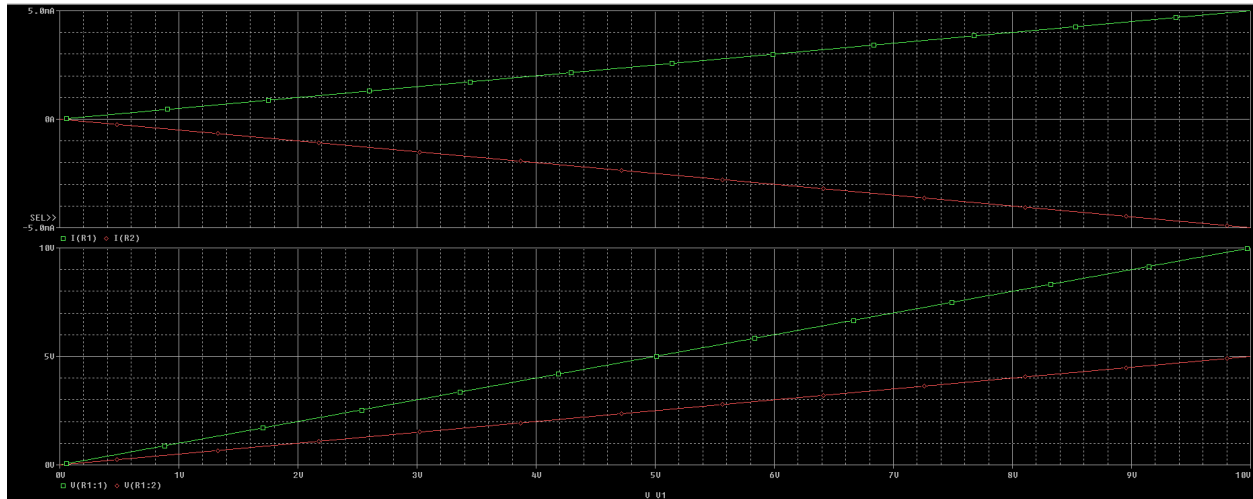


Figure 3: Result of DC Analysis. Top: Current; Bottom: Voltage

## 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

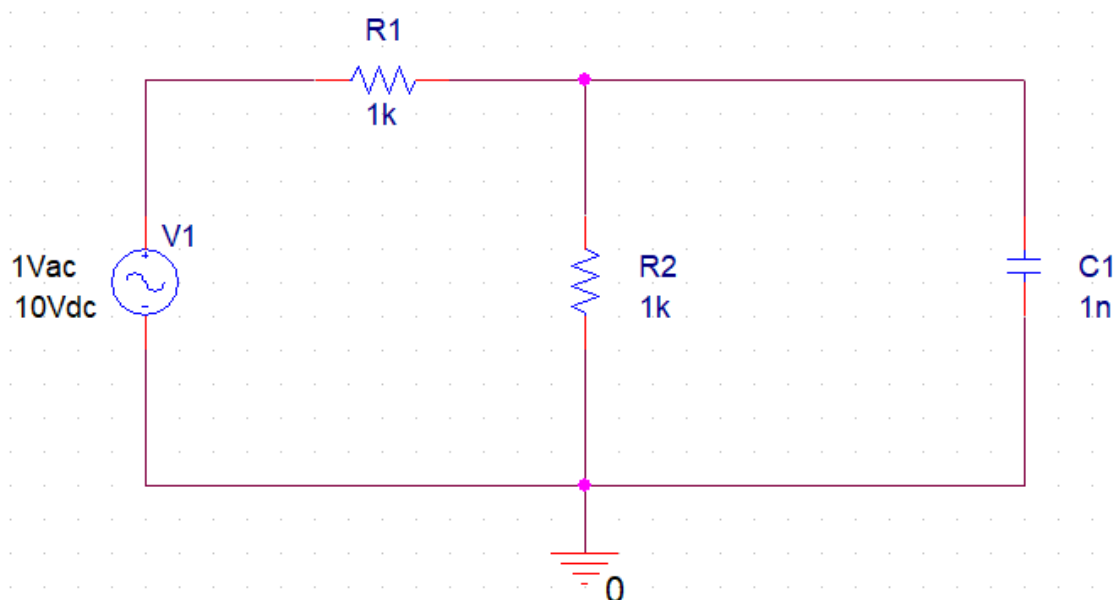


Figure 4: Circuit for AC Analysis

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 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 ??.

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 5.

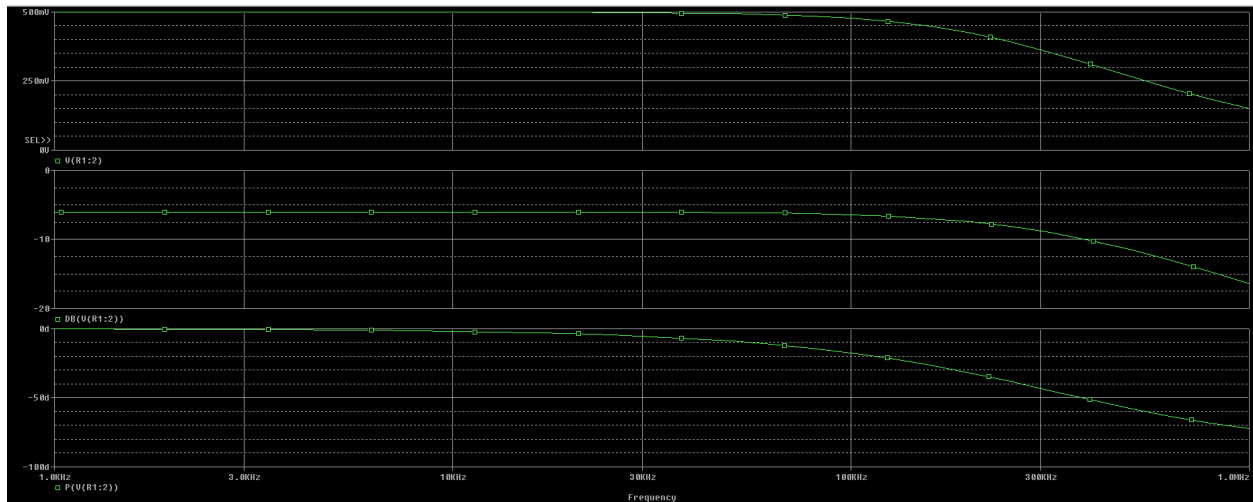


Figure 5: Result of AC Analysis

## 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:

- $V1 = 0$
- $V2 = 5$
- $TD = 10\text{n}$
- $TR = 20\text{n}$
- $TF = 20\text{n}$
- $PW = 500\text{n}$
- $PER = 2\mu$

We include the circuit in Figure 6.

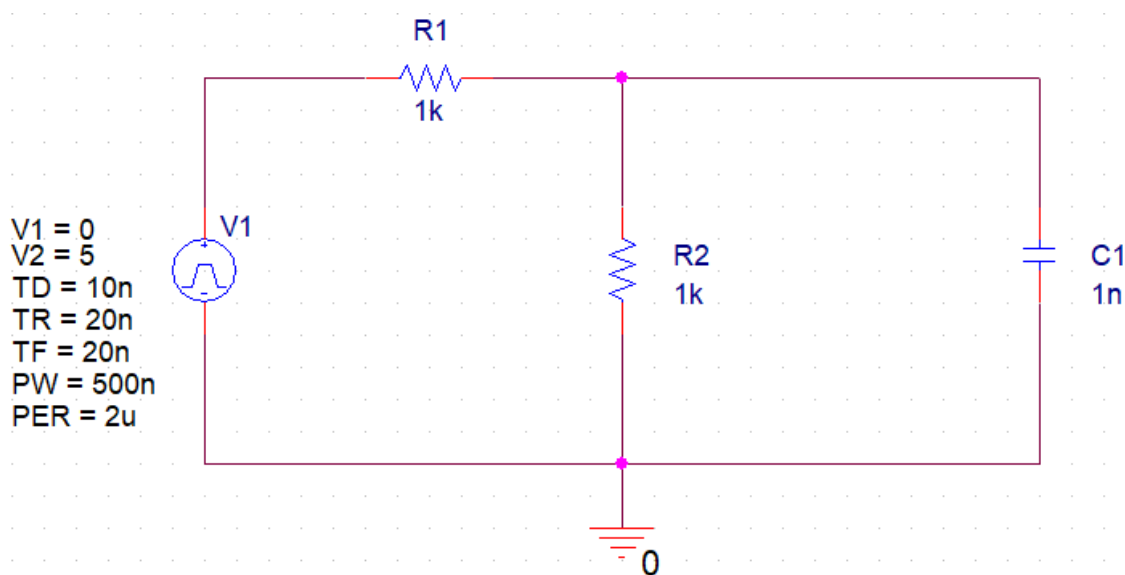


Figure 6: Circuit for Transient Analysis

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). The resulting plot is shown in Figure 7.

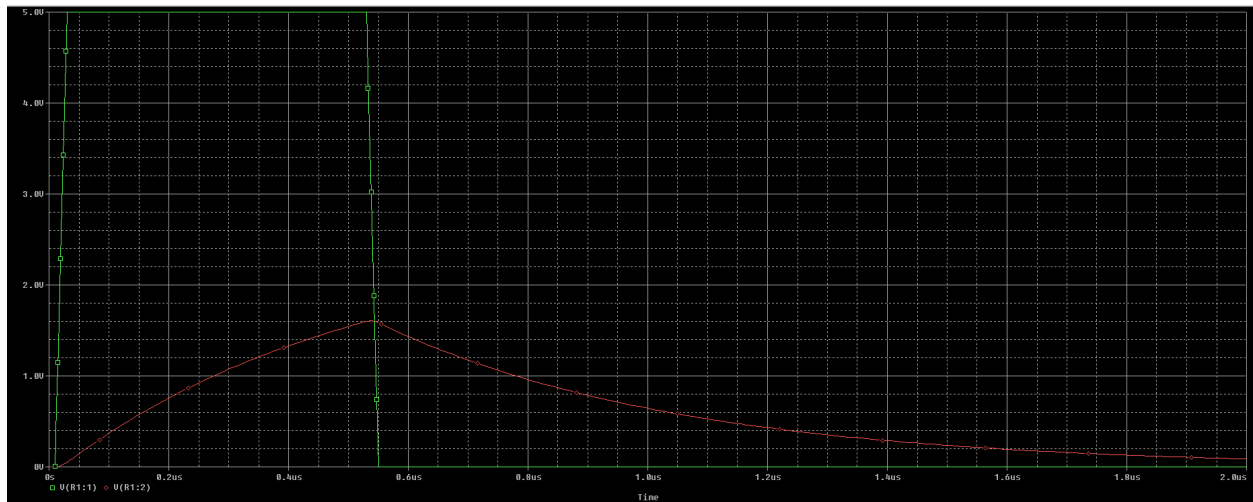


Figure 7: Result of Transient Analysis



## 6 Practice Example

Use PSpice to simulate the circuit shown in Figure 8 using DC, AC, and Transient analyses. Take a screenshot of a plot of the voltage across capacitor  $C_1$  in each analysis.

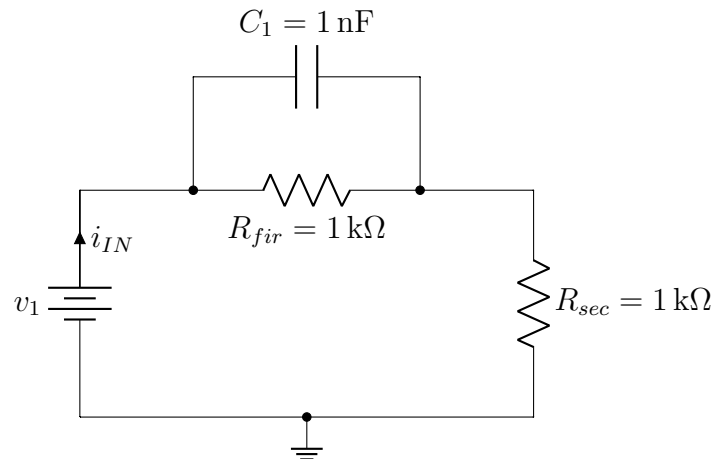


Figure 8: Practice Circuit

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

- $V1 = 0$ ,
- $V2 = 5$ ,
- $TD = 100\text{n}$ ,
- $TR = 40\text{n}$ ,
- $TF = 40\text{n}$ ,
- $PW = 500\text{n}$ ,
- $PER = 2\text{u}$

And use the following parameters:

- Run to Time:  $2\text{u}$
- Start saving data after:  $50\text{n}$
- Maximum step size:  $10\text{n}$

## 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).

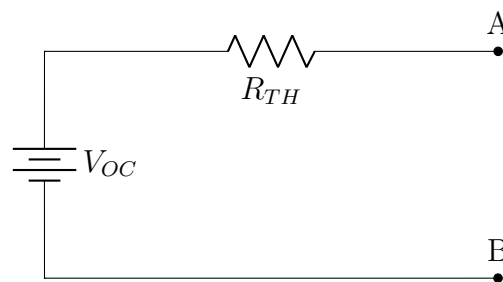


Figure 9: Thévenin Equivalent Circuit

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:

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

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.

## 7.1 Example Exercise

Simulate the circuit in Figure 10 and form its Thévenin Equivalent as seen from terminals A and B.

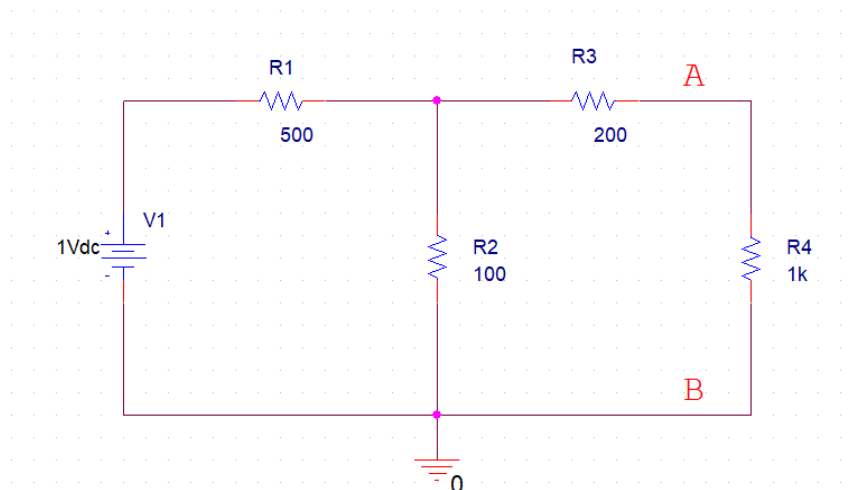


Figure 10: Example Circuit

To find the open circuit voltage, replace resistor  $R_4$  with a large “dummy” resistance (at least  $100\text{ M}\Omega$ ). 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.

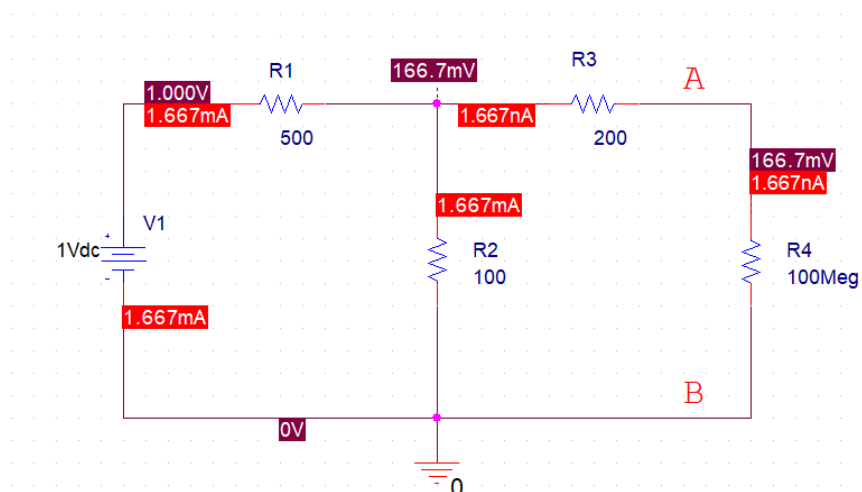


Figure 11: Open Circuit Voltage Schematic

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,  $i_{sc}$ .

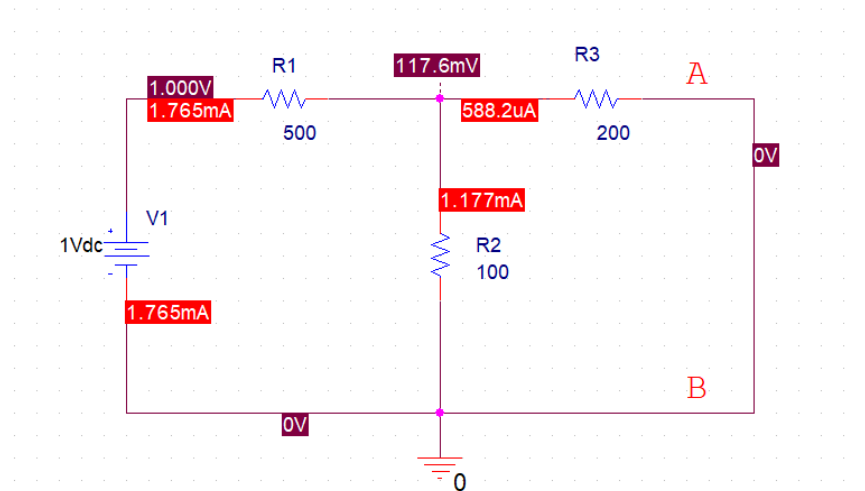


Figure 12: Short Circuit Voltage Schematic

Calculate the Thévenin resistance using Equation 1. Now, re-create Figure 9 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.

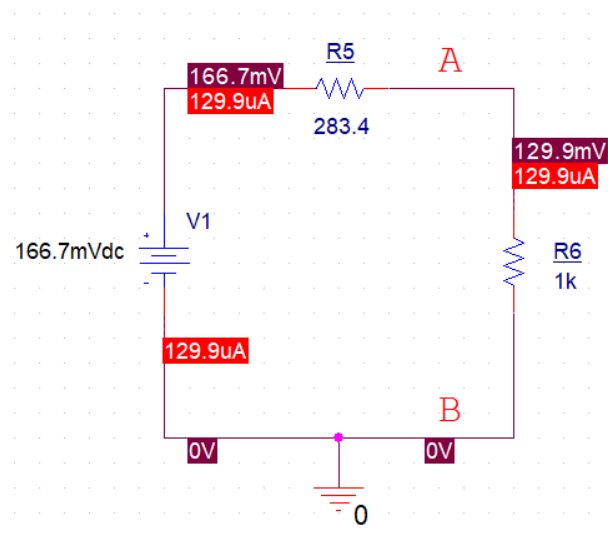


Figure 13: Thévenin Equivalent Schematic

## 7.2 Challenge Exercise: Thévenin Equivalent Circuit

Create the schematic in Figure 14 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.

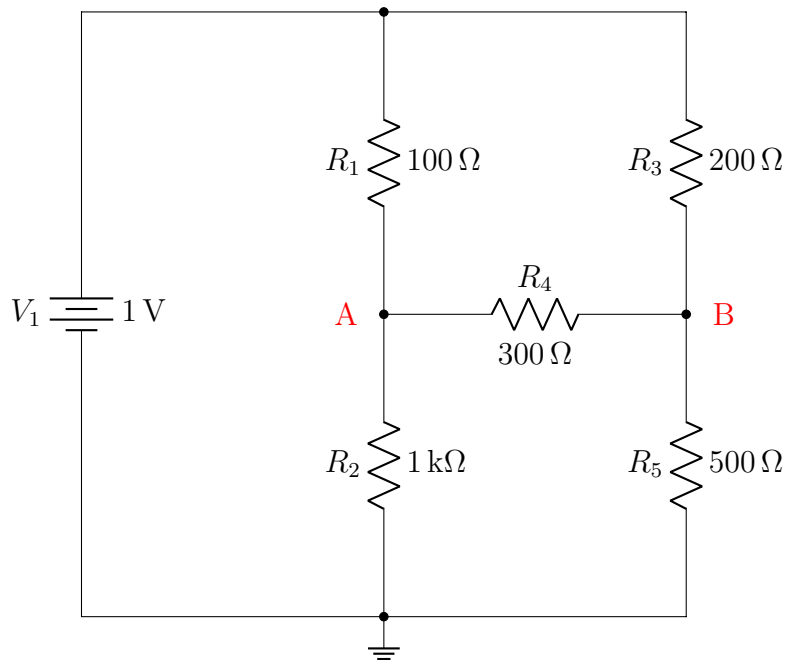


Figure 14: Exercise Circuit

Table 1: ECE 230L Laboratory 4 Grading Rubric

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