ECE 230L - LAB 3

INTRODUCTION TO CIRCUIT SIMULATION USING PSPICE

Contents

List of Figures

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

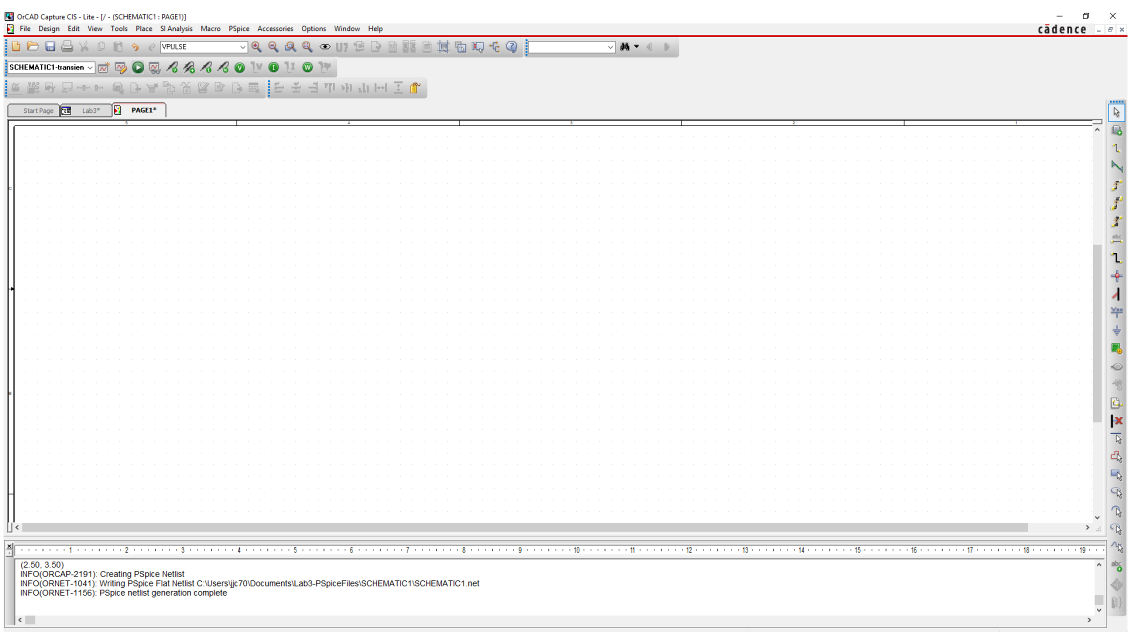

Figure 1: Blank Schematic

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.

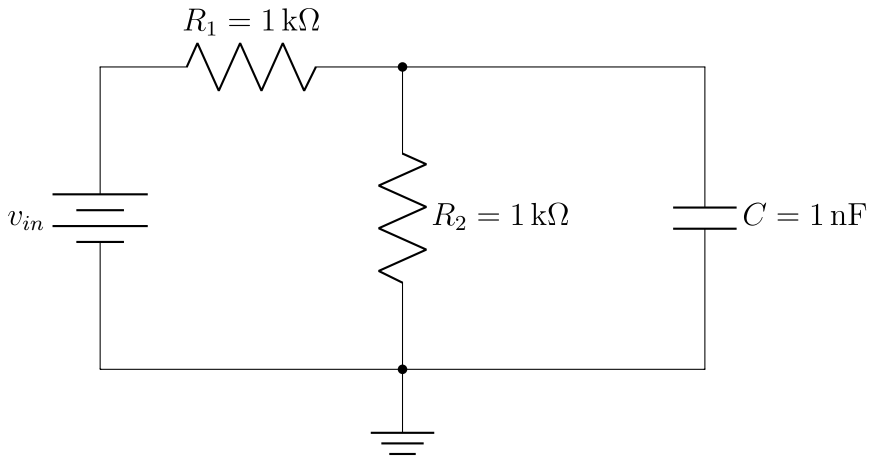

Figure 2: Initial Circuit

To make the circuit,

1. Add a DC Voltage Source by following $\boxed{\text{Place}} \rightarrow \boxed{\text{PSPice Component}} \rightarrow \boxed{\text{Source}} \rightarrow \boxed{\text{Voltage Source}} \rightarrow \boxed{\text{DC}}$

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 6 shows the circuit schematic and Figure **??** shows the result of DC analysis (top plot: current, bottom plot: 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**. We include the circuit in Figure 13. The following parameters will be used:

- DC Value: 10
- AC Amplitude: 1

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 screencap of the settings for the AC Analysis is included in Figure 3.

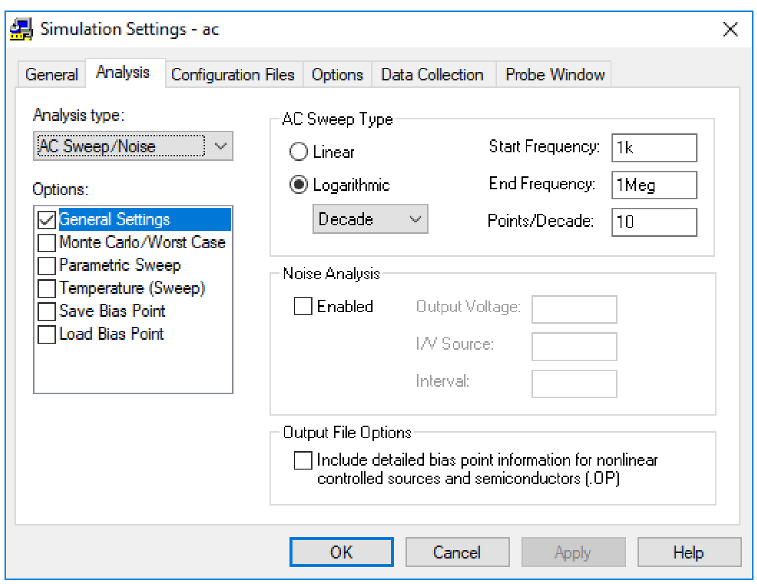

Figure 3: Settings for AC Analysis

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 **??** and the resulting plots are shown in Figure 14.

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

A schematic for the transient analysis circuit can be found in the appendix.

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).

6 Practice Example

Use PSpice to simulate the circuit shown in Figure 4 using DC, AC, and Transient analyses. Take a screencap of a plot of the voltage across capacitor C_1 in each analysis.

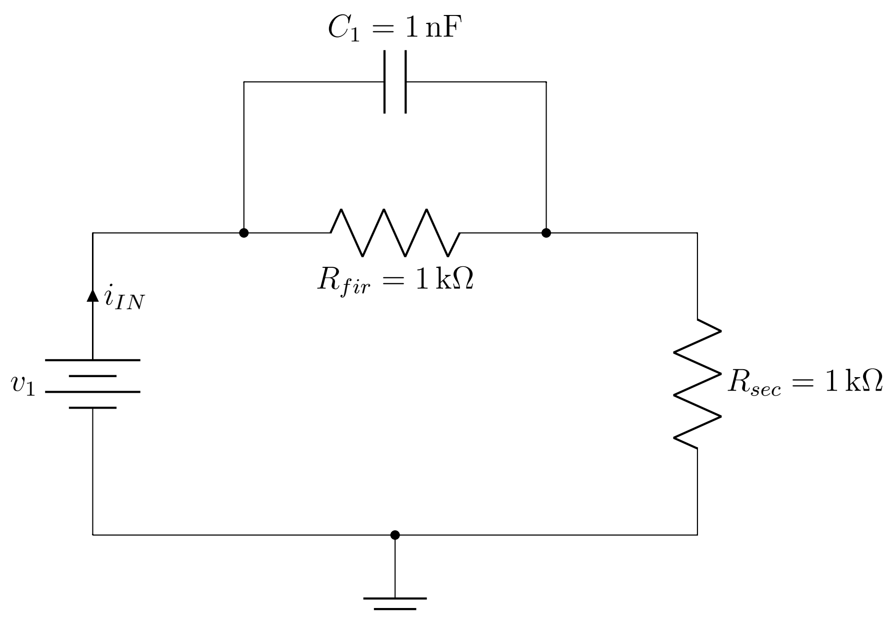

Figure 4: Practice Circuit

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 = 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).

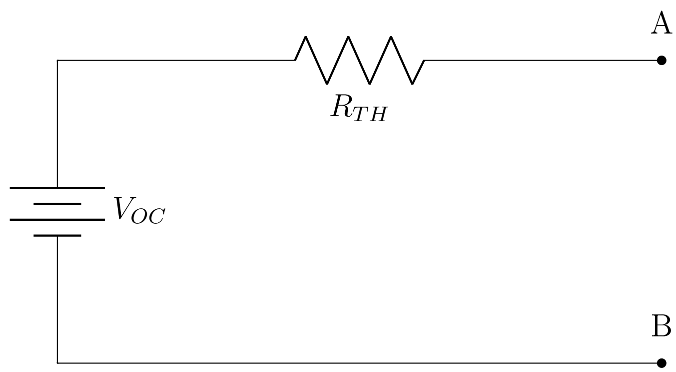

Figure 5: Thévenin Equivalent Circuit

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 6 and form its Thévenin Equivalent as seen from terminals A and B.

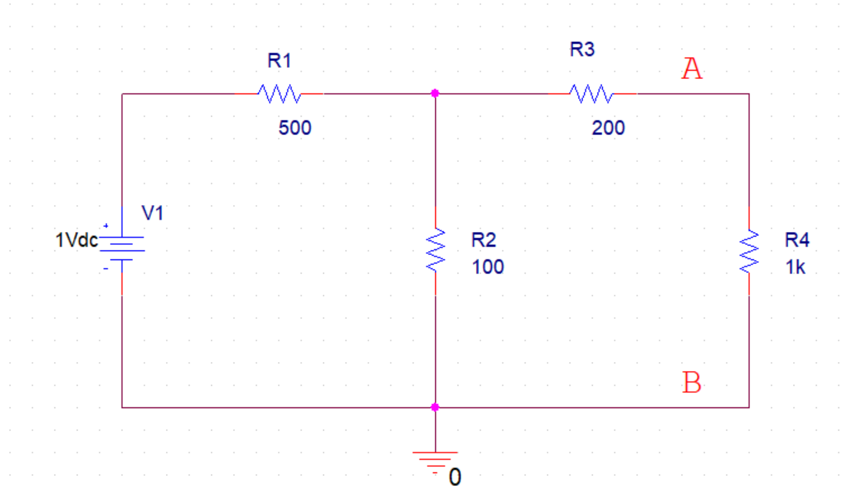

Figure 6: Example Circuit

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.

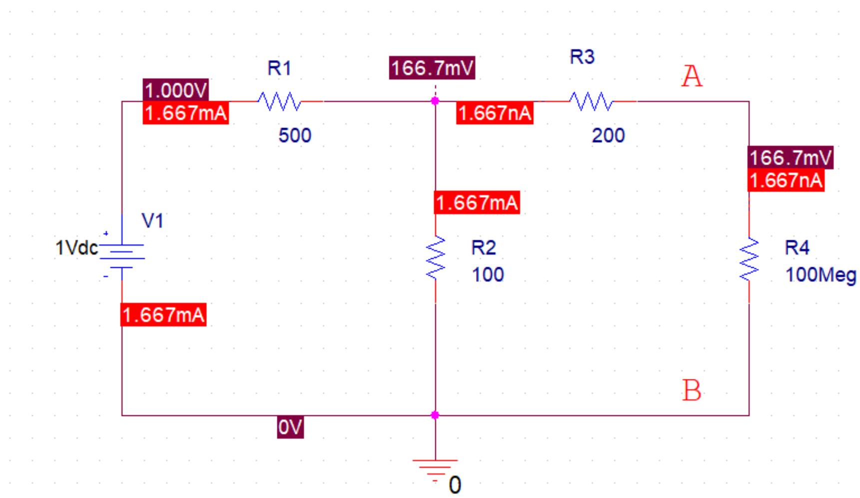

Figure 7: Open Circuit Voltage Schematic

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}.

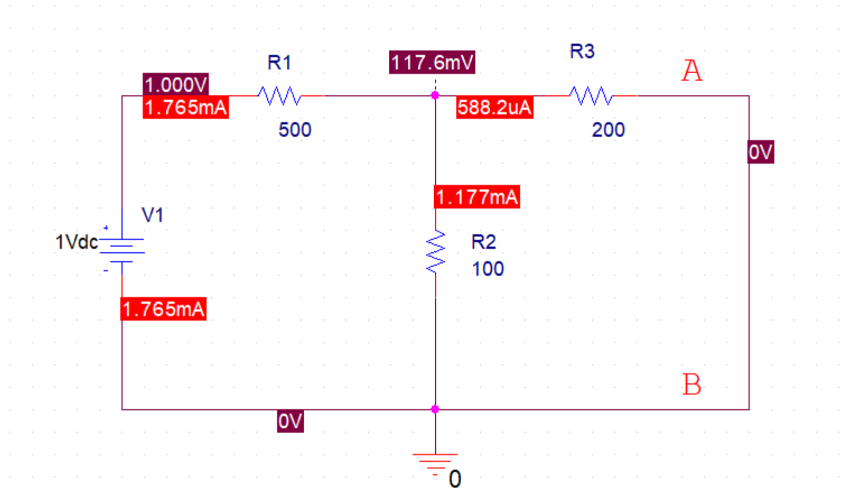

Figure 8: Short Circuit Voltage Schematic

Calculate the Thévenin resistance using Equation 1. Now, re-create Figure 5 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.

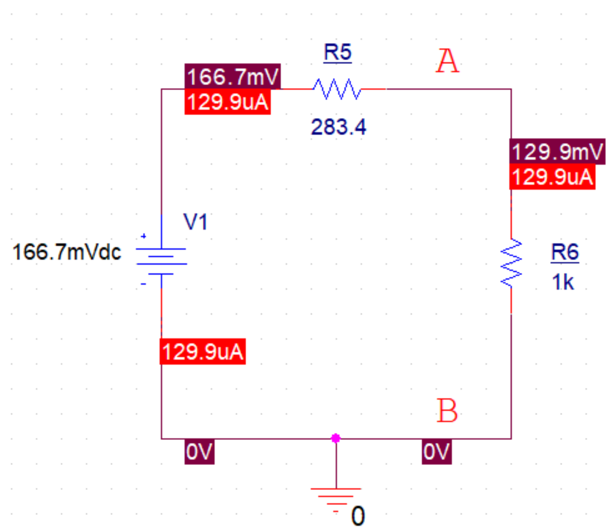

Figure 9: Thévenin Equivalent Schematic

7.2 Challenge Exercise: Thévenin Equivalent Circuit

Create the schematic in Figure 10 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.

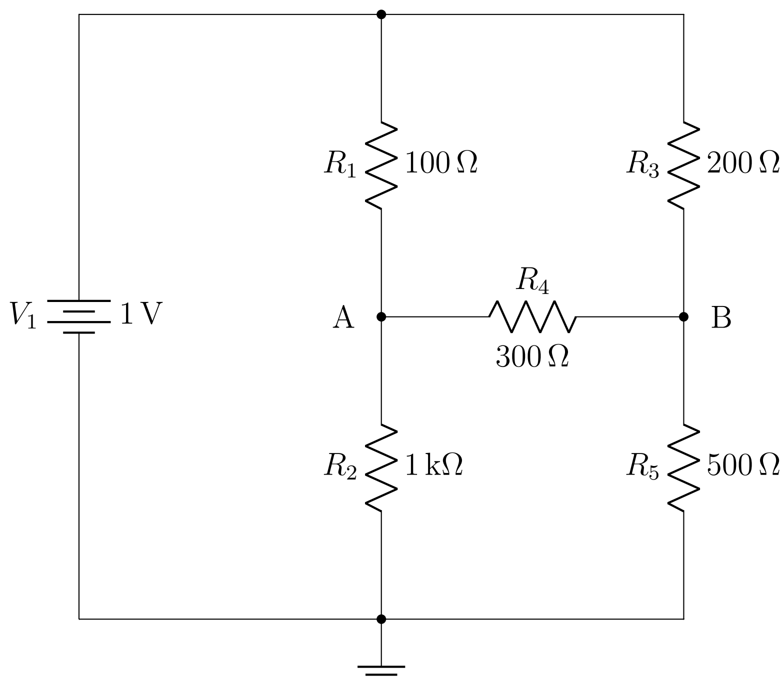

Figure 10: Exercise Circuit

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

Appendices

A DC Analysis

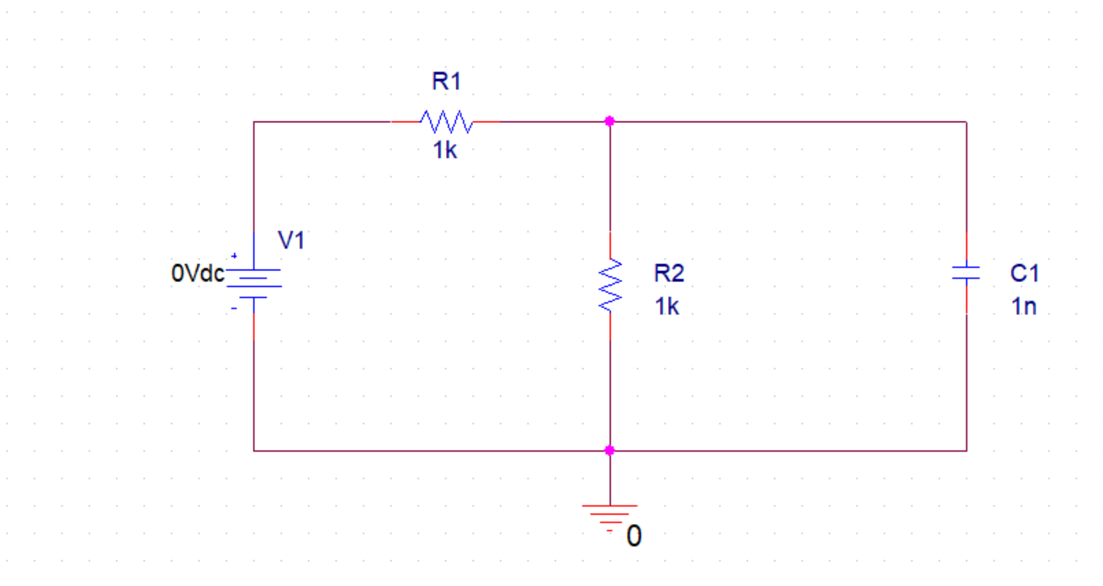

Figure 11: Circuit for DC Analysis

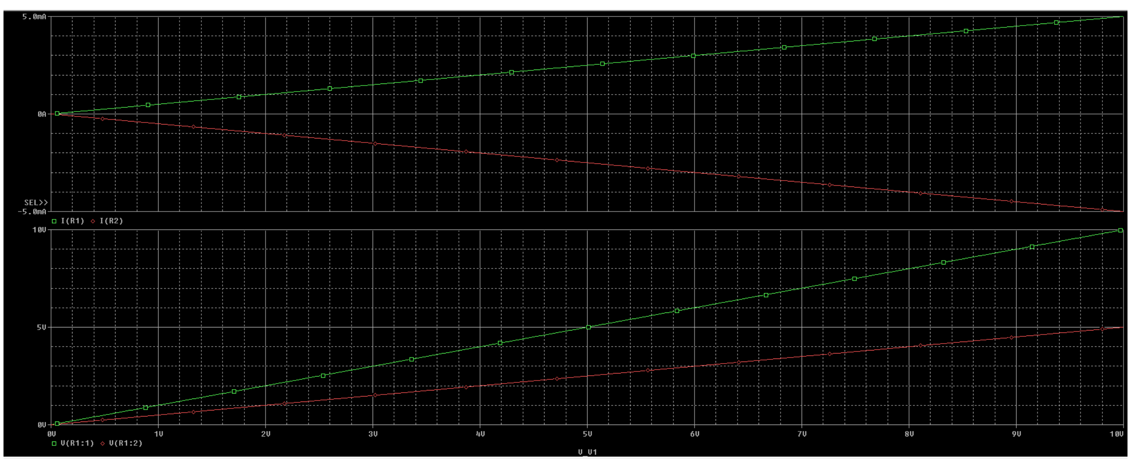

Figure 12: Result of DC Analysis

B AC Analysis

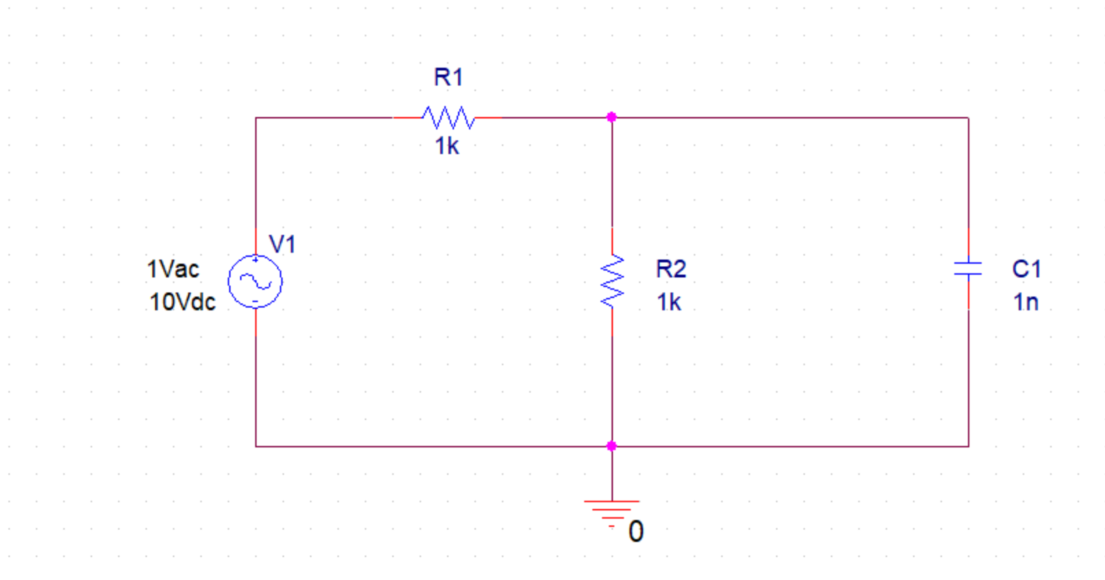

Figure 13: Circuit for AC Analysis

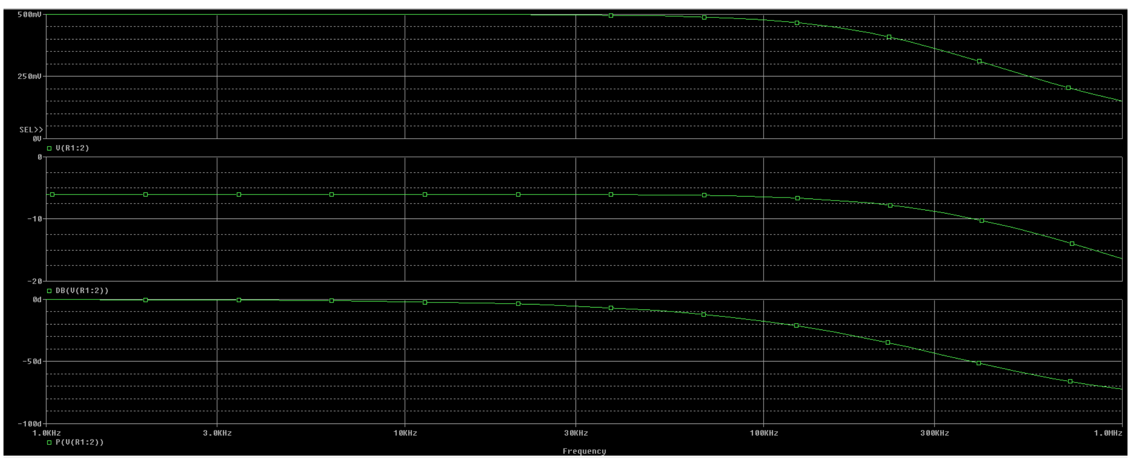

Figure 14: Result of AC Analysis

C Transient Analysis

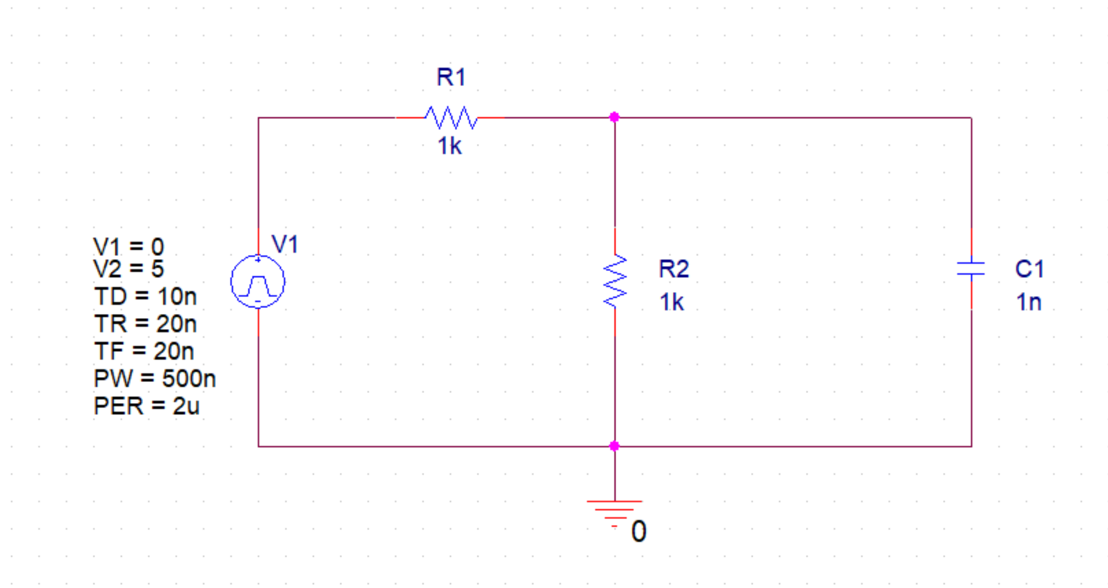

Figure 15: Circuit for Transient Analysis

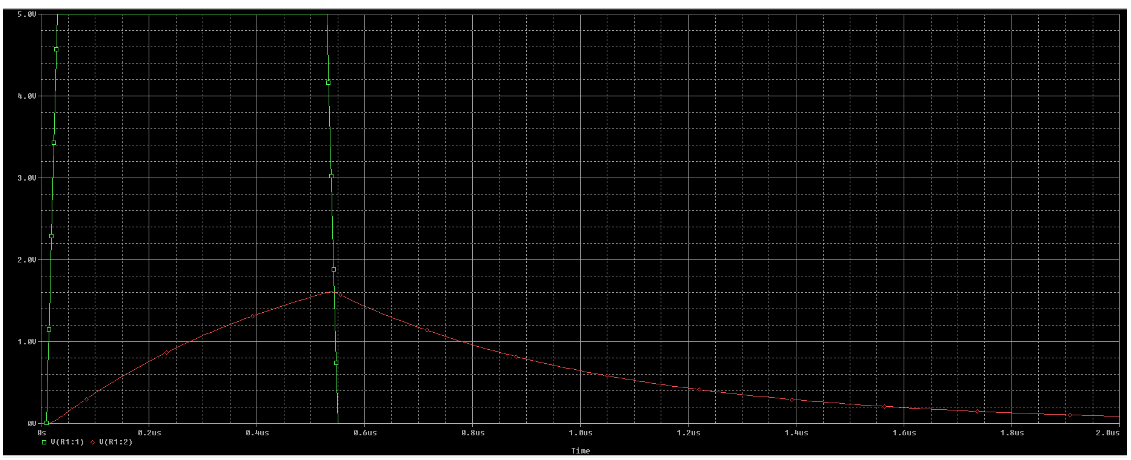

Figure 16: Result of Transient Analysis