Lab 3—ECE 230L

Introduction to Circuit Simulation Using PSpice

Table of Contents

List of Figures

Goals of this lab

Circuit simulation is an important tool in the analysis and design of microelectronic circuits. Spice is a general-purpose circuit simulation program in which nonlinear dc, nonlinear transient, and linear ac analyses of electronic circuits are carried out. Circuits may contain resistors, capacitors, inductors, mutual inductors, independent voltage and current sources, four types of dependent sources, lossless and lossy transmission lines, switches, uniformly distributed RC lines, and the five most common semiconductor devices: diodes, BJTs, JFETs, MESFETs, and MOSFETs. The version of Spice used in the ECE Department at Duke is PSpice. 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.

Setting up a circuit using ORCAD Capture

Open ORCAD Capture CIS and create a new project by selecting File-> New-> Project. Name your project 'Lab 3' and choose the 'Analog or Mixed A/D' option under the 'Create a New Project Using' menu. Select 'Create a blank project' when prompted by the 'Create PSpice Project' menu. Once the new project has been created, circuit design can begin. Sources, components, ground nodes, and wires can be selected using the 'Place' menu.
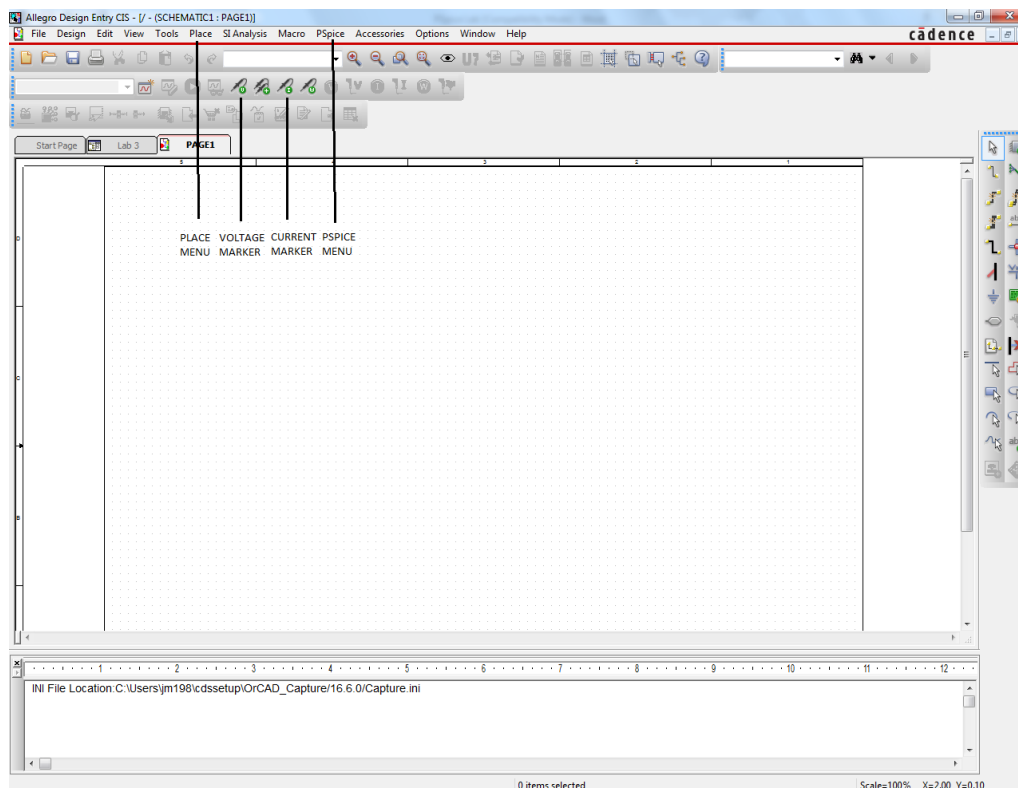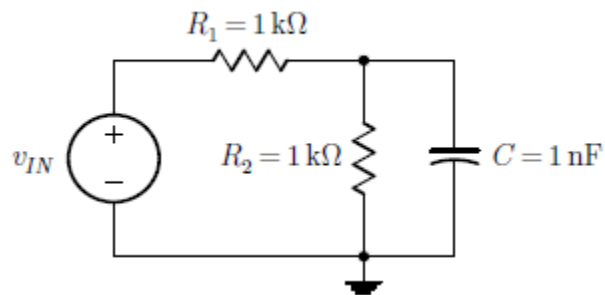

Figure 1: Blank Schematic

PSpice will be used to model the circuit in figure 2 and perform DC, AC, and transient analysis on the circuit.

$$R_1 = 1\,\mathrm{k\Omega}$$

v_{IN} $R_2 = 1\,\mathrm{k\Omega}$ $C = 1\,\mathrm{nF}$

Figure 2: Example Circuit

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.

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).
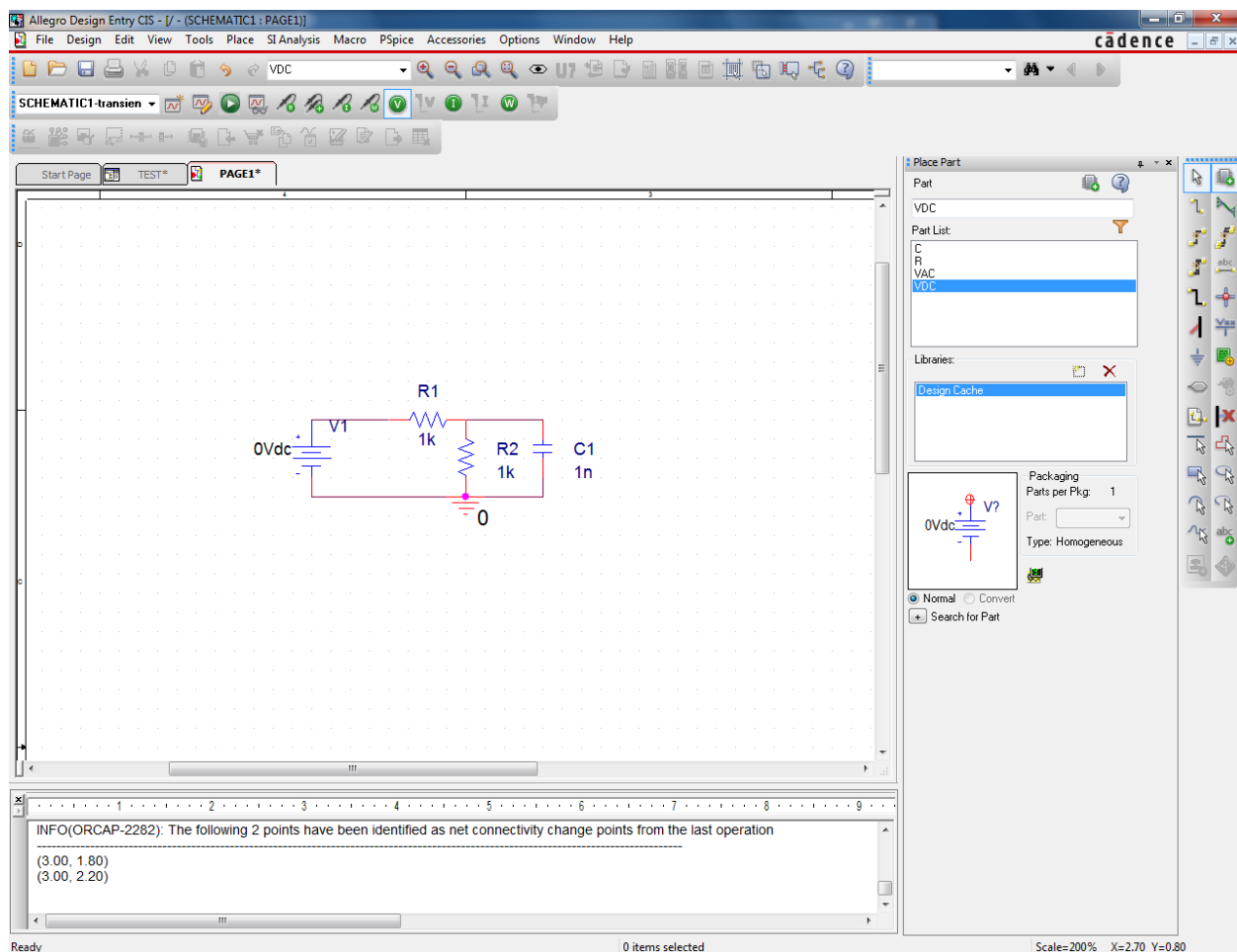

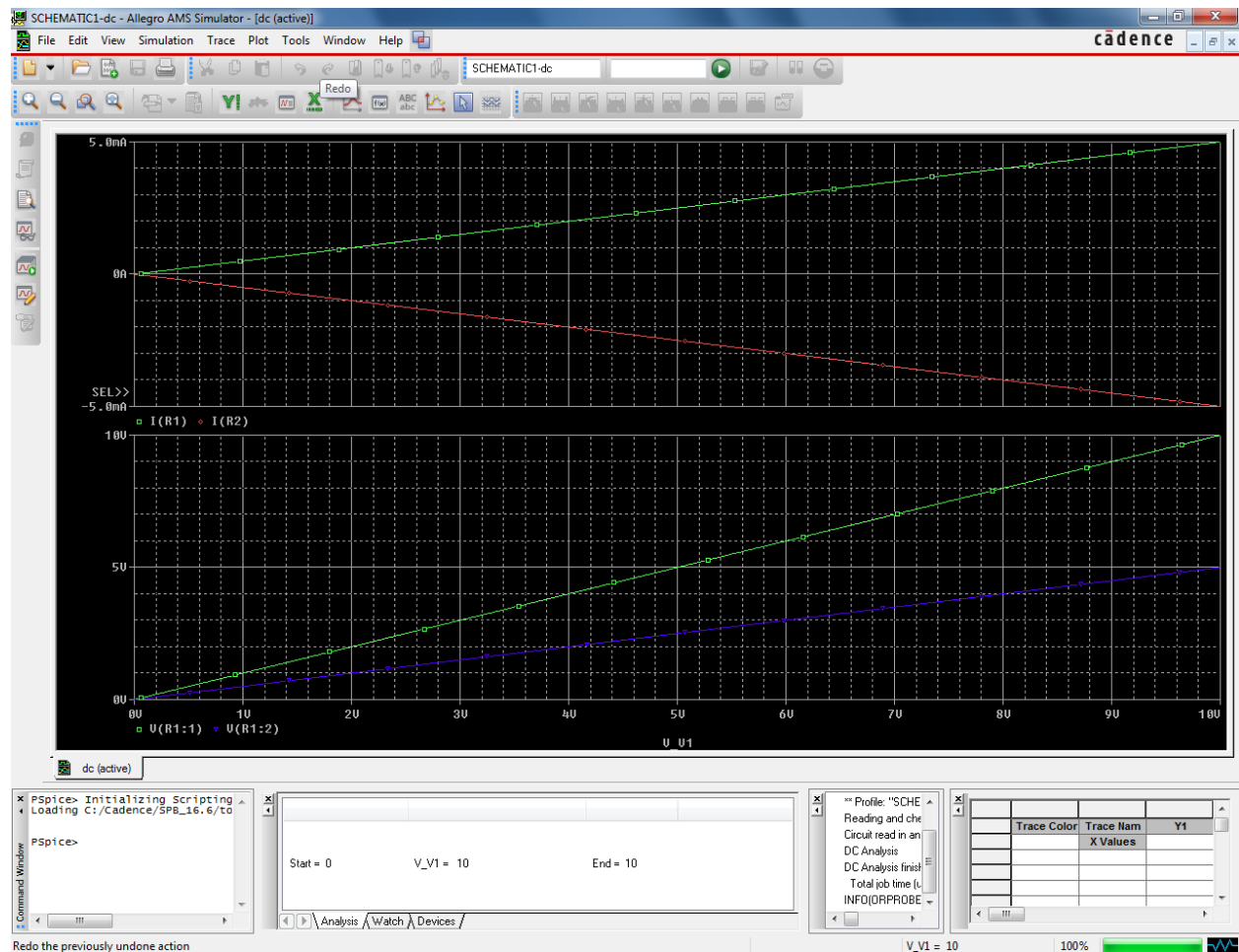*Figure 3: DC Analysis Circuit*

Figure 4: Results of DC Analysis

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

After the voltage source properties have been changed AC analysis can be performed. To perform an AC analysis of the circuit, create a new simulation profile called 'ac'. To analyze the example circuit, select 'AC Sweep/Noise' in the Analysis Type drop down menu and use the following parameters:

- AC Sweep Type: Logarithmic and 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. 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 V(R2).

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 5 and the resulting plots are shown in figure 6 (top plot: V(R2) , middle plot: DB(V(R2)) , bottom plot: P(V(R2))).
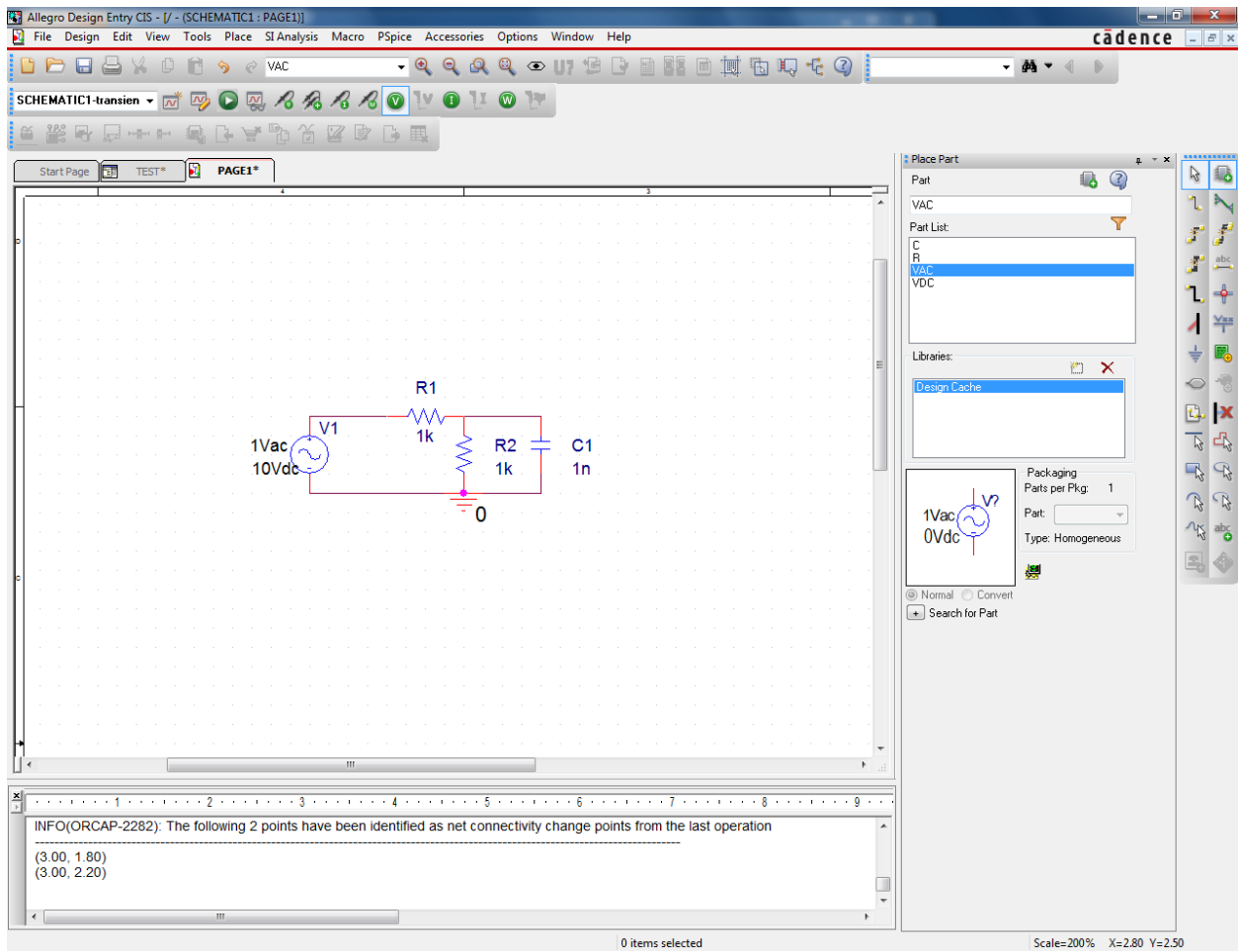
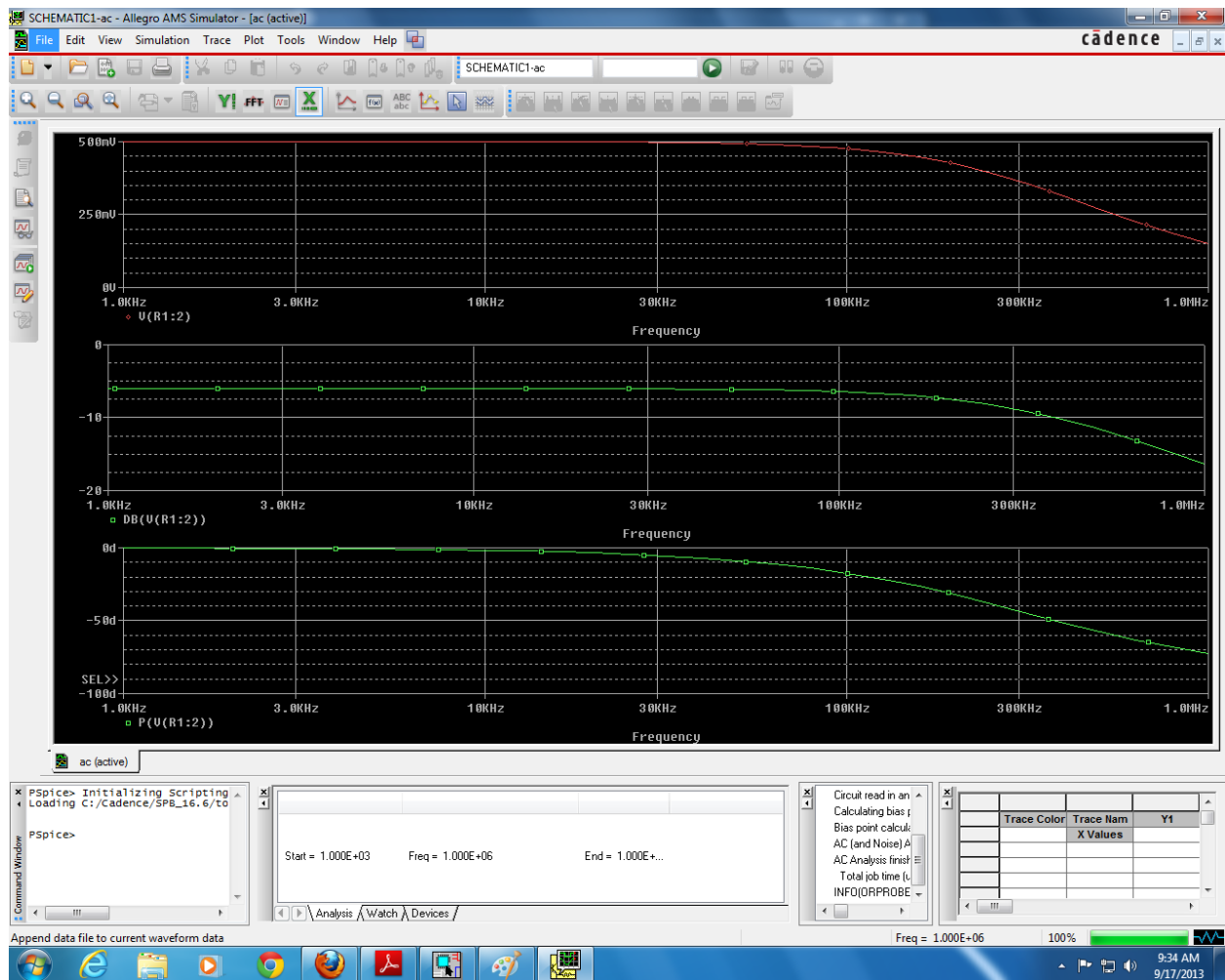*Figure 5: AC Analysis Circuit*

Figure 6: Results of AC Analysis

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

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.
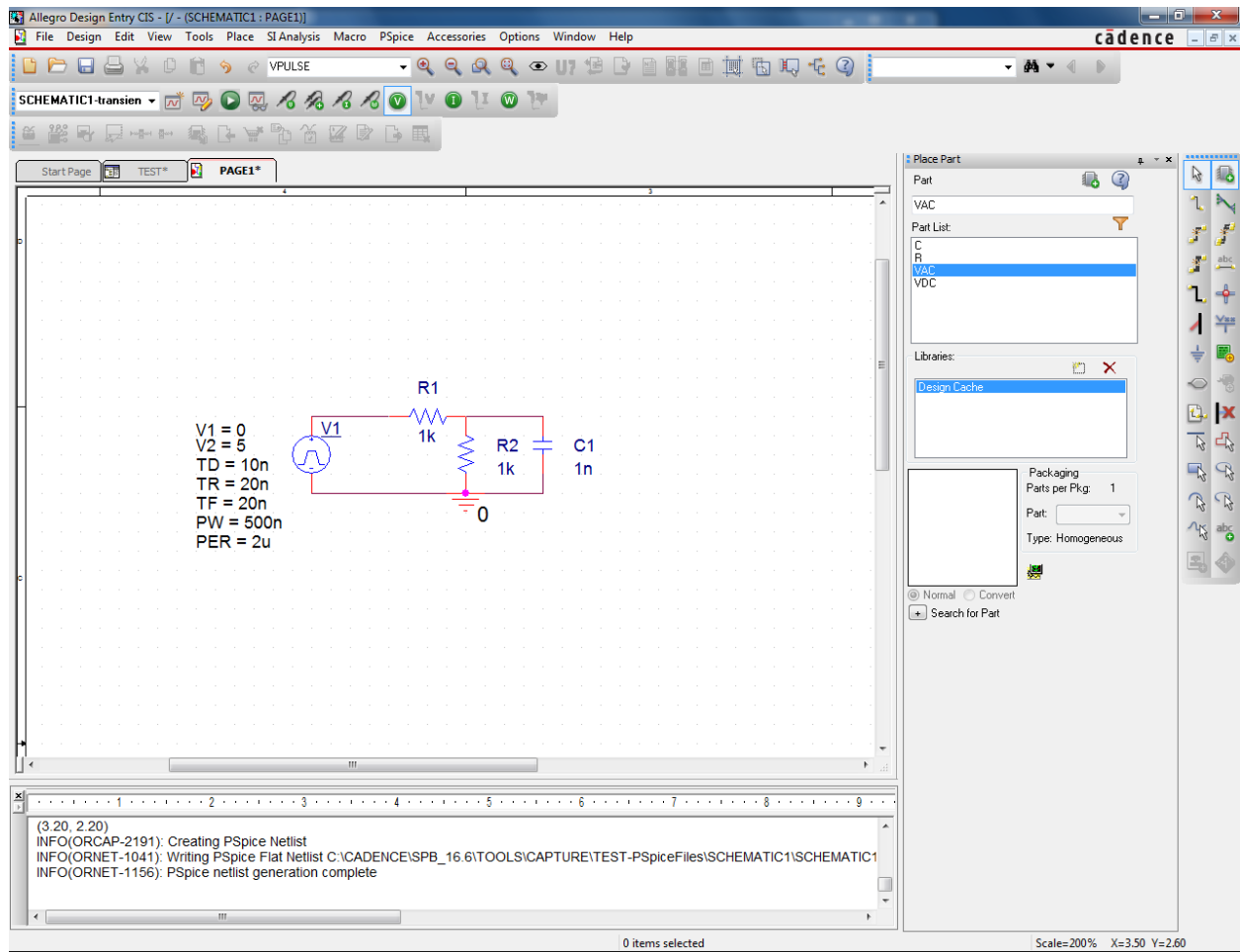
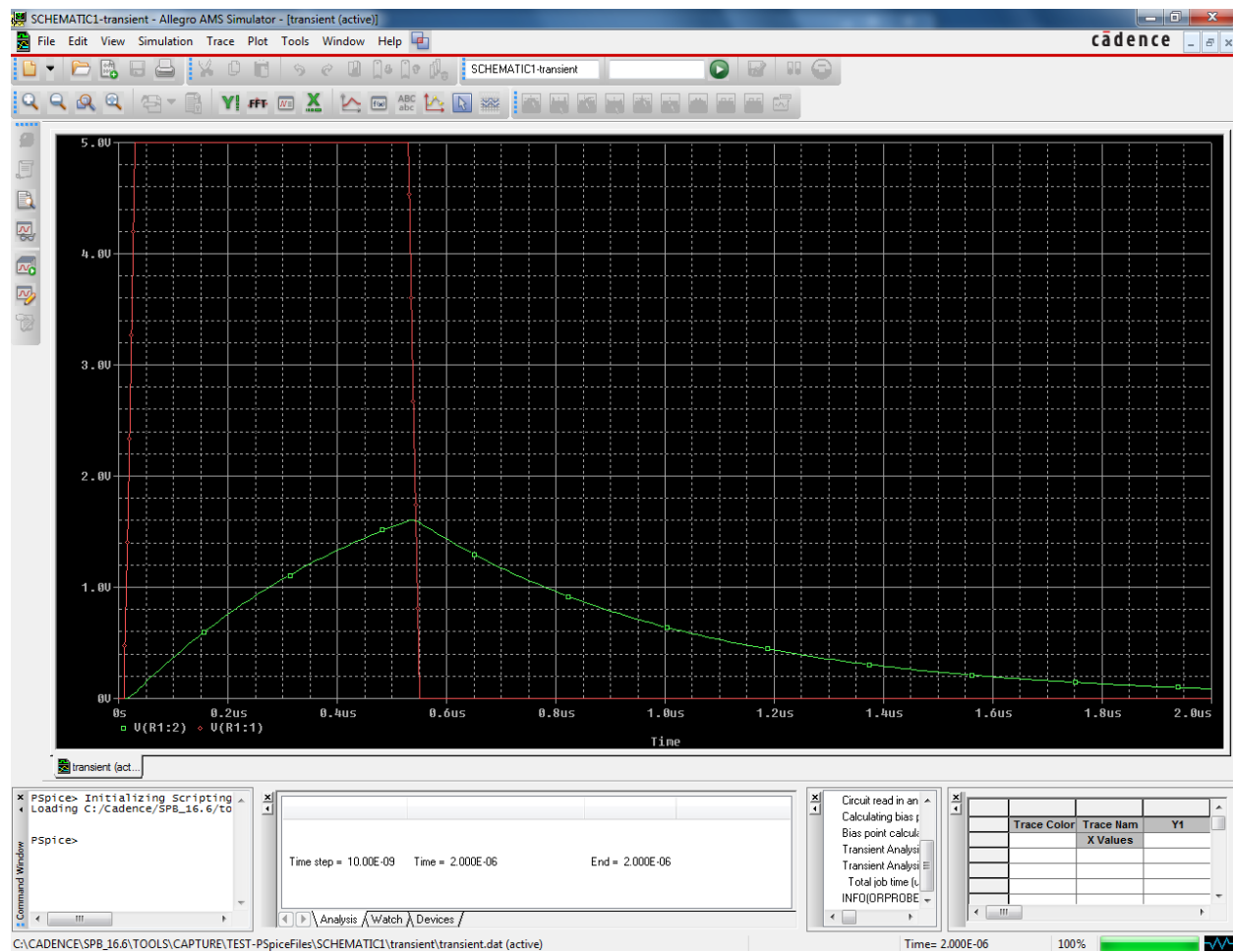*Figure 7: Transient Analysis Circuit*

Figure 8: Results of Transient Analysis

Practice Example

Use PSpice to simulate the circuit shown in figure 8 using DC, AC, and transient analysis. Use the element values that are specified in the figure.

1. In the DC analysis, sweep Vi from 0V to 10V in 0.01V increments
2. In the AC analysis, Vi have a DC component of 5v and an AC component of 1v. The voltage source is swept from 10Hz to 1MHz using 10 data points in every decade.
3. In the transient analysis, let Vi be a pulse input with initial voltage of 0v, maximum voltage of 5v, rise and fall times of 40ns each, initial delay of 100ns, pulse width of 500ns, and a period of 2us. The analysis is done between 50ns and 2us in 10ns increments.

Print out plots of the voltage across the capacitor C1 in each analysis.
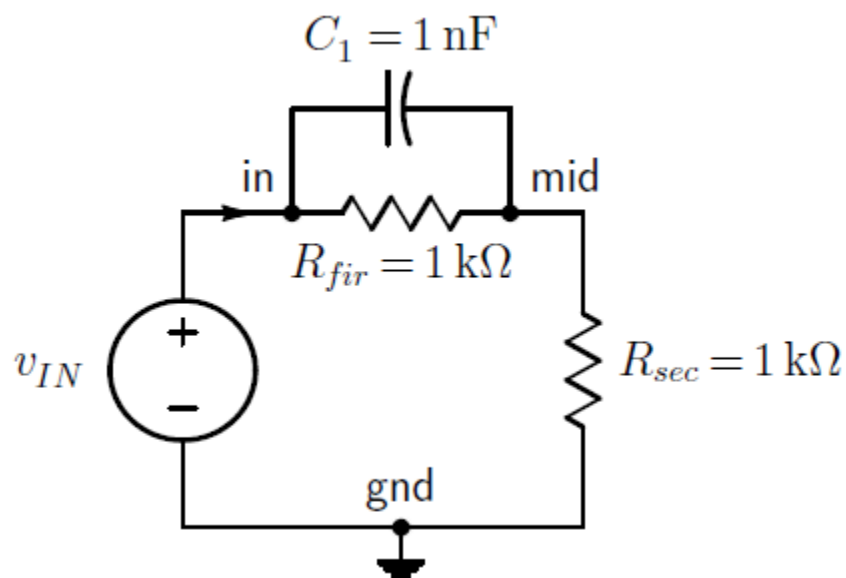

Figure 9: Practice circuit

Exploration: Thevenin Equivalent Circuits

Purpose

The purpose of this exercise is to learn how to form a Thevenin 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). See figure 10.
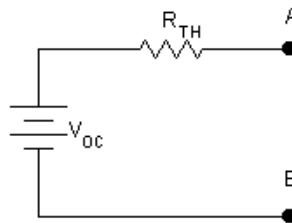

Figure 10: Thevenin Equivalent Circuit

To form a Thevenin 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 Thevenin resistance R_{Th} can be determined using the relation:

$$R_{Th} = \frac{v_{oc}}{i_{sc}} \qquad (1)$$

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.

Exercise

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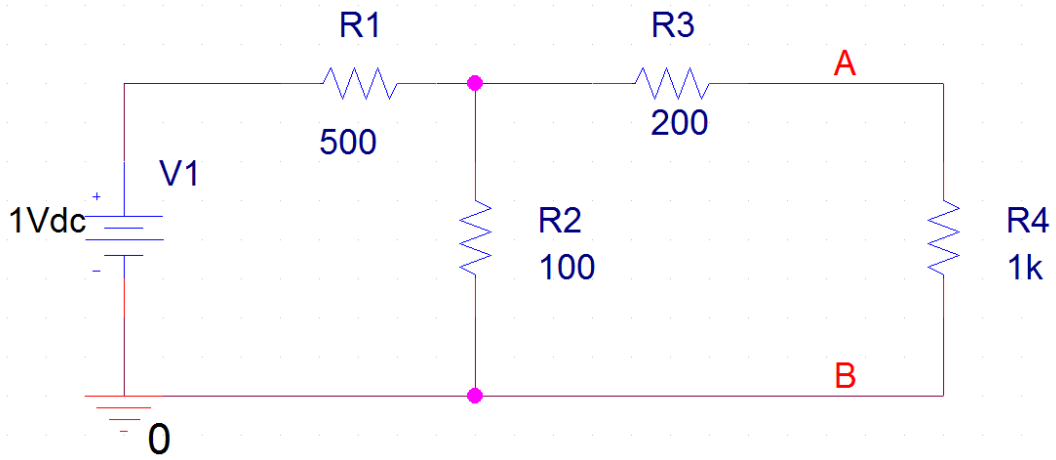

Figure 11: Example circuit

Create the schematic in figure 11 using PSpice. You will need to manipulate the circuit to obtain the necessary v_{oc} and i_{sc} values as follows:

To find the open circuit voltage, replace resistor R_4 with a large "dummy" resistance (at least 100MΩ). 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. This will show the node voltages and currents in throughout the circuit. Record the voltage across terminals A and B as your open circuit voltage.
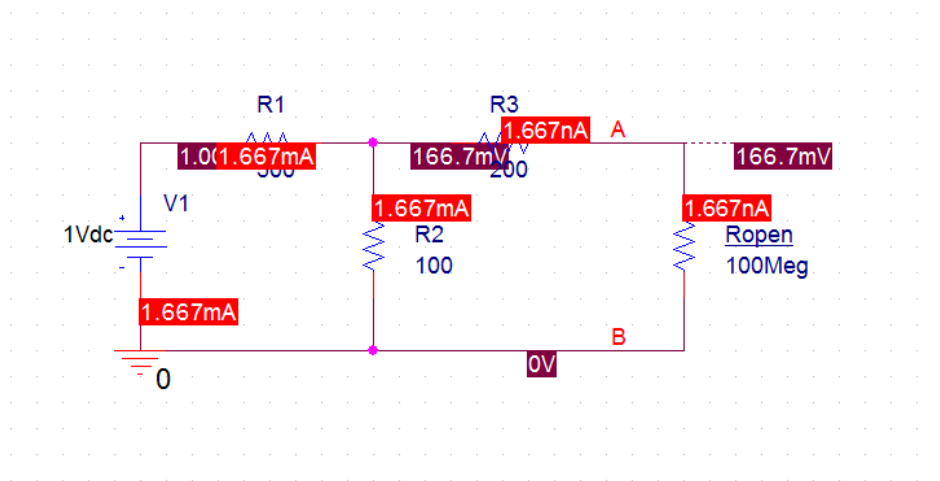

Figure 12: Open circuit voltage schematic

To find the short circuit current, remove R_4 and connect terminals A and B with a wire (short circuit). Use the same "thev" simulation profile that was created to find v_{oc}. Run the simulation. Once the simulation is complete, follow PSpice-> Bias Points and click Enable. This will show the node voltages and currents in throughout the circuit. Record the current between terminals A and B as your short circuit current.
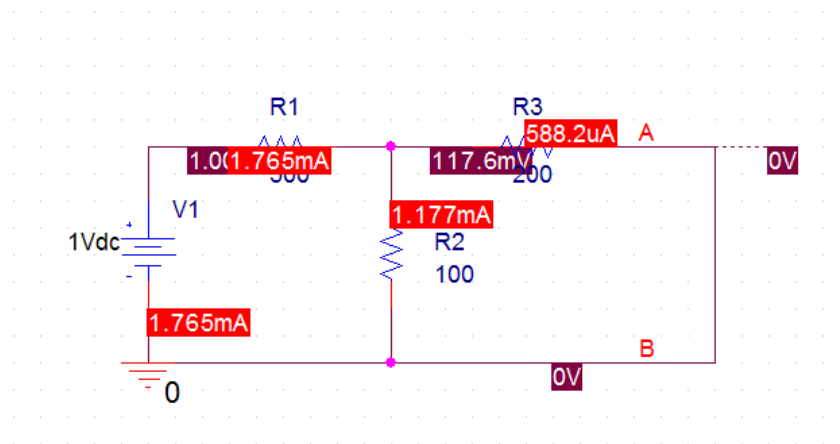

Figure 13: Short circuit current schematic

Calculate the Thevenin resistance using (1). Now, re-create figure 10 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 Thevenin Equivalent. Run the simulation using the same "thev" profile. Once the simulation is complete, follow PSpice-> Bias Points and click Enable. This will show the node voltages and currents in throughout the circuit. Check to make sure the terminal voltages and currents match for both circuits.
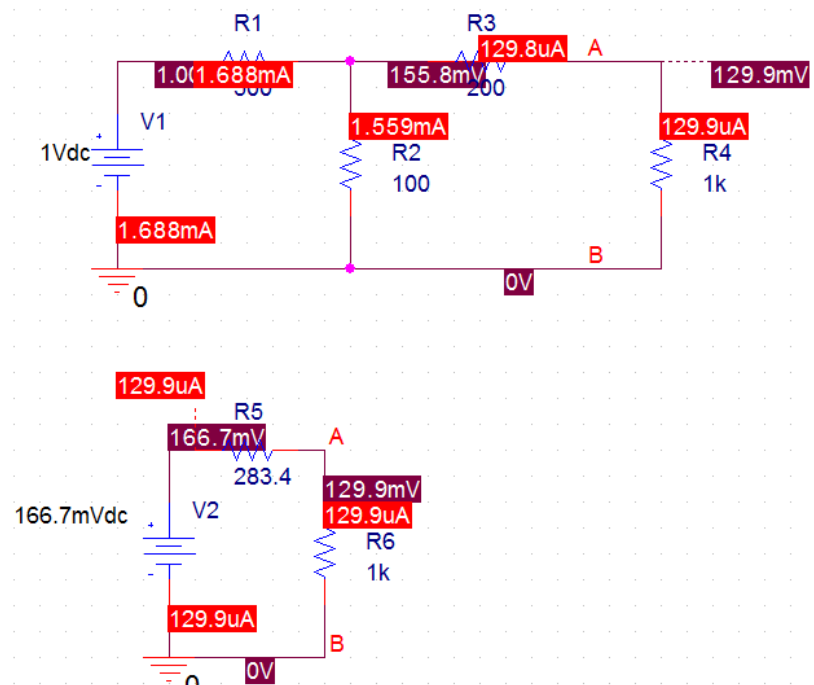

Figure 14: Original circuit (top) versus Thevenin Equivalent circuit (bottom)

Practice Exercise: Thevenin Equivalent circuit

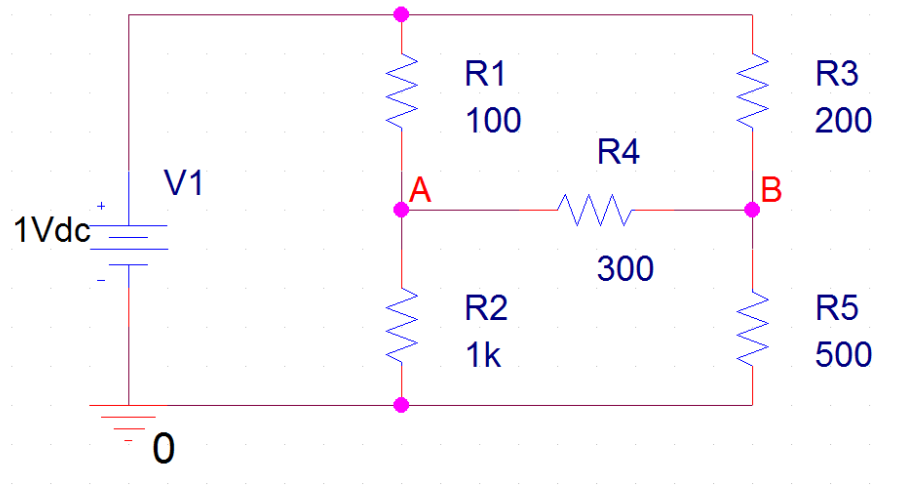Create the schematic in figure 15 using PSpice and find the Thevenin Equivalent circuit. You will need to manipulate the circuit in figure 6 to obtain the necessary v_{oc} and i_{sc} values.

Figure 15: Practice circuit