

### EE1002 Lab 3: PSPICE Workshop - Circuit Simulation Using PSPICE

<b>Name:</b> _____  <b>Matric No.</b> _____  <b>Group:</b> _____	<b>Activities Completed</b>	<b>Verified By</b>	<b>Marks from 3</b>
	<b>1</b>		
	<b>2</b>		
	<b>3</b>		
	<b>4</b>		
	<b>5</b>		

## 1. Objective

This lab session is for students to be given exposure to try out PSPICE functions and the features which will be required in other labs.

This lab session will teach students to create and simulate circuit schematics with **Orcad Capture CIS** (a PSPICE tool), to understand various stimulus types and their corresponding analysis types, and to verify theoretical results using PSPICE simulations. Students will also learn to write SPICE netlists to represent electrical circuits and simulate them.

## **2. Introduction to PSPICE**

SPICE (Simulation Program with Integrated Circuit Emphasis) was developed at the University of California at Berkeley for computer simulation of analog circuits. Several companies have developed graphical user interfaces for Spice, which make it much easier to use. One of the most popular is PSPICE. PSPICE takes in a circuit netlist description file in ASCII format (SPICE deck) and constructs the corresponding Kirchhoff Current Law (KCL) and Kirchhoff Voltage Law (KVL) equations automatically. The program then solves the corresponding equations to find the desired operating current and voltage associated with the circuit.

### 3. PSPICE using Orcad Capture CIS

#### 3.1. Opening Orcad Capture CIS

- a. Go to Start → All Programs → Orcad 10.5(16.0) Demo → Capture CIS Demo.
- b. Select File → New → Project. From the New Project window, give your project a name, select “Analog or Mixed A/D”, and select its location (see Figure 1). Change the “location” at the bottom to “C:\OrCAD” or another proper directory (You may need to create a new directory first).

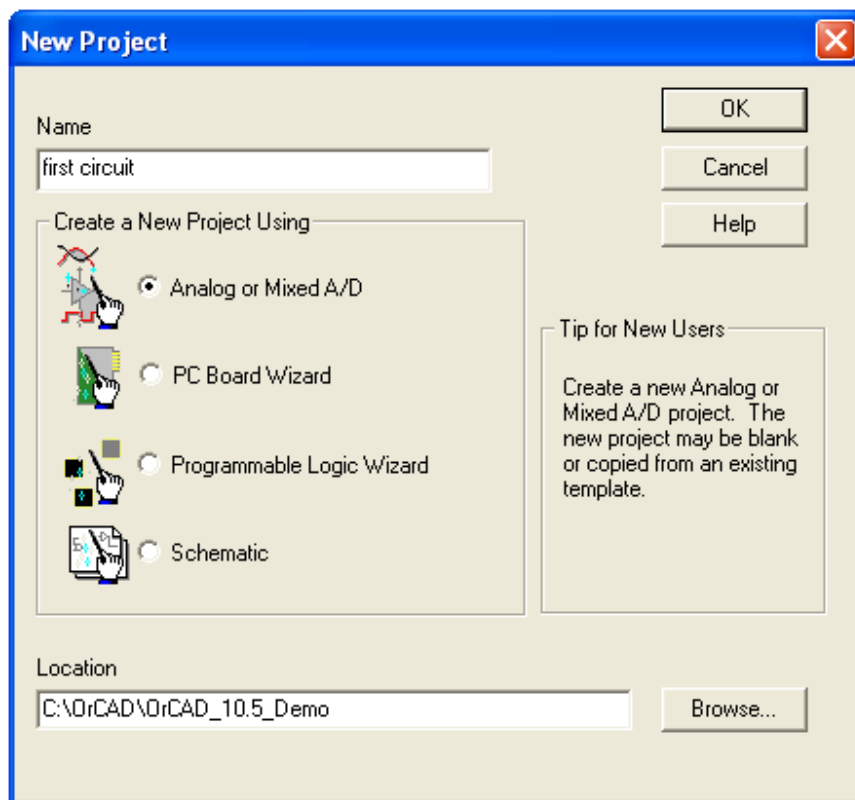


Figure 1 – New Project Window

- c. Choose “Create a blank project” and then click OK. You’re now ready to create your schematic.

### 3.2. Creating Schematics with Capture CIS

- a. Next we must draw the desired schematic. To get parts, go to Place -> Part...(see Figure 2). There is also shortcut icon on the right side of the window.

**Note:** If no parts are visible in the “Place Part” dialog, simply click on the “Add Library...” button. Inside the Browse File dialog, select all the “.olb” files and hit the “Open” button to add all libraries.

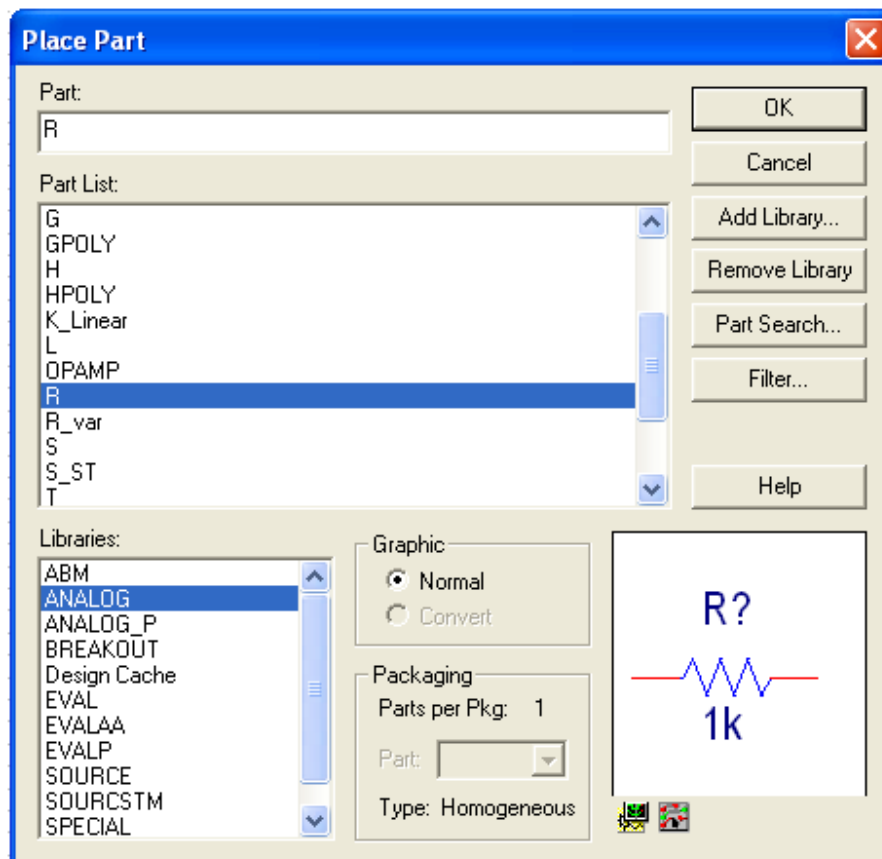


Figure 2 – Place Part Dialog

- b. Place a resistor (R) in the ANALOG Library into the schematic page. You can change the part's attributes by clicking on its name (R1) or value (1k), as shown in Figure 3. Change the resistance value to 5k. Maintain the name as R1.

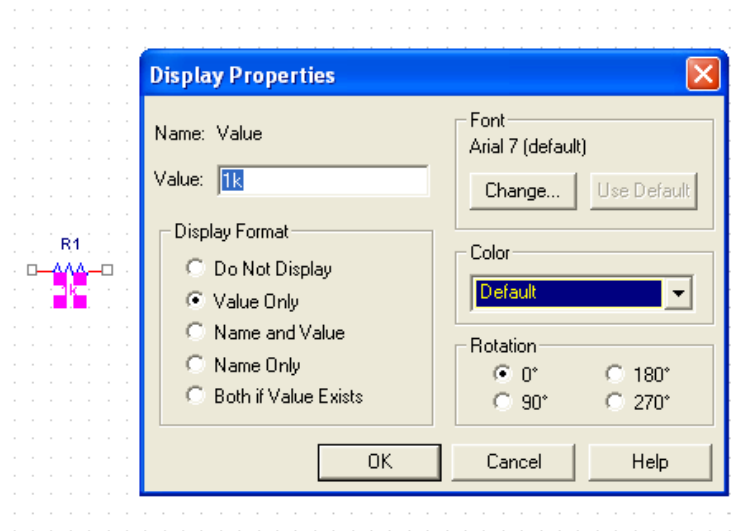


Figure 3 – Resistor's Attributes Dialog

- c. Place a DC voltage source (VDC) in the SOURCE Library. Set the voltage value to be 1V as shown in Figure 4.

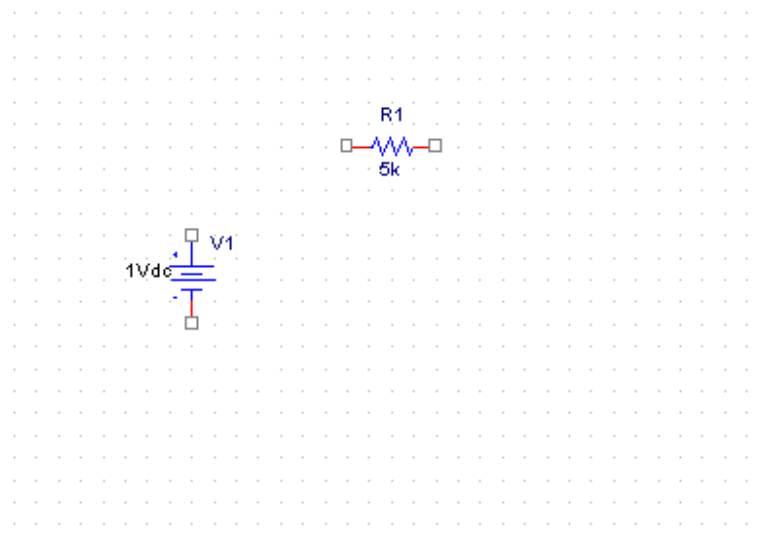



Figure 4 – Add 1V DC Voltage Source

- d. Remember in PSPICE we **ALWAYS** need a ground in the circuit, this can be found in the Place Ground menu, or by clicking the  button on the right side of the screen. The ground we will use is 0/SOURCE as shown in Figure 5.

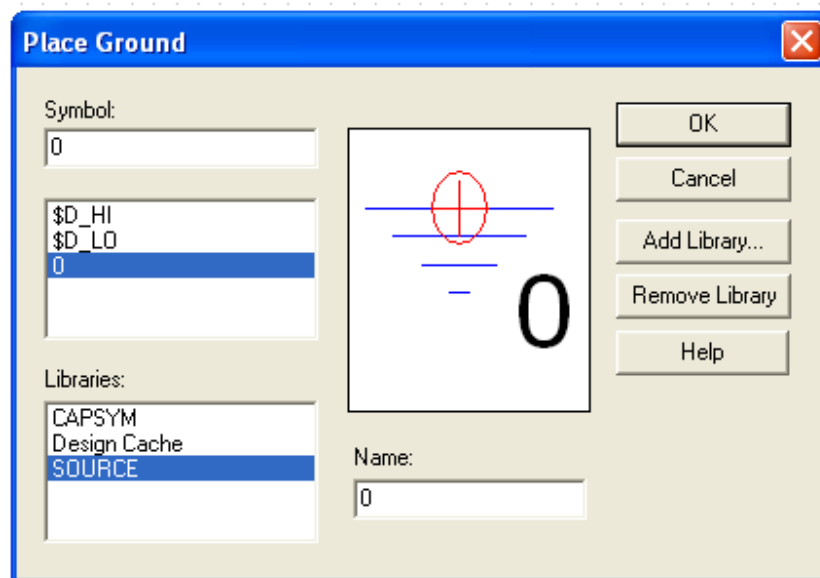


Figure 5 – Place Ground Dialog

- e. Wire the parts together as shown in Figure 6. Use “w” for shortcut. Now the circuit is ready for simulation.

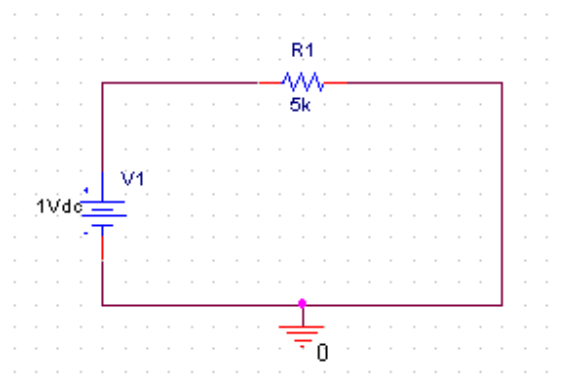


Figure 6 – Complete first circuit after wiring

- f. Moving, rotating, and other part manipulation functions can be accomplished through the Edit menu at the top of the screen or right click with part selected.

**Warning:** When making changes, be careful - parts may become disconnected.

### 3.3. Circuit Simulation

- a. After connecting the circuit parts, go to PSpice → New Simulation Profile. (see Figure 7). Name your simulation profile and click on Create, then the “Simulation Settings” (see Figure 8) dialog will appear.

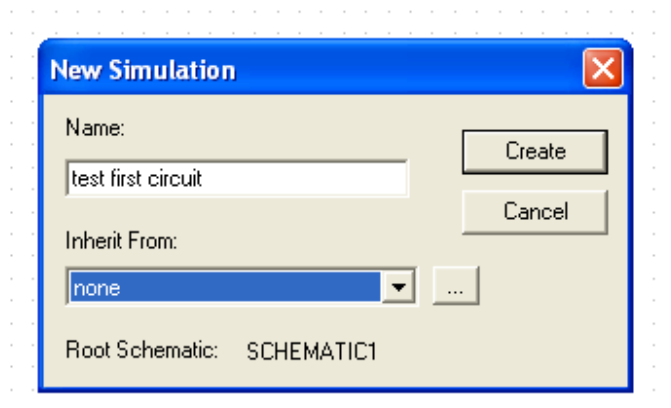


Figure 7 – Naming Simulation

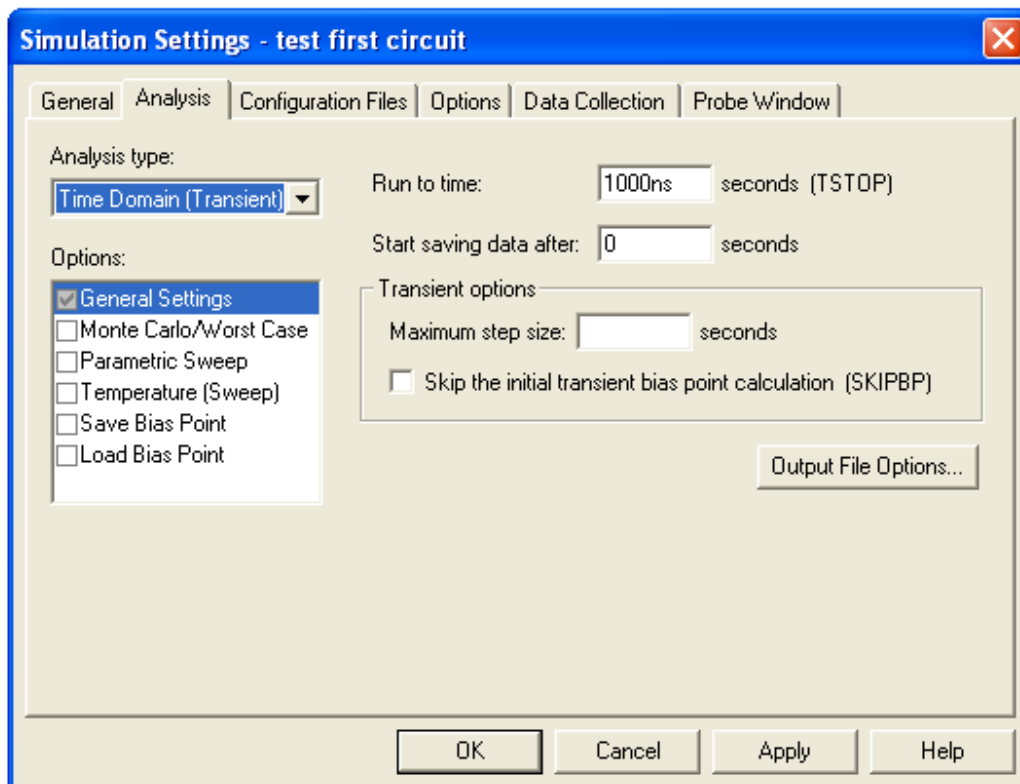




Figure 8 – Simulation Settings

- b. Time domain analysis or transient analysis is a type of simulation in which the x-axis is time. Usually this type of simulation starts from  $t=0$ s to a certain time set by the user. By default, TSTOP is set to 1000ns. Leave the simulation settings as it is and click OK.
- c. Simulate the circuit according to the above simulation settings by selecting PSPICE → Run or by clicking the  button. PSPICE A/D will appear.
- d. Select Trace → Add Trace or click the  button. Once the add trace dialog is open, find I(R1). Click OK as shown in Figure 9. Simulation waveform will be displayed as shown in Figure 10.



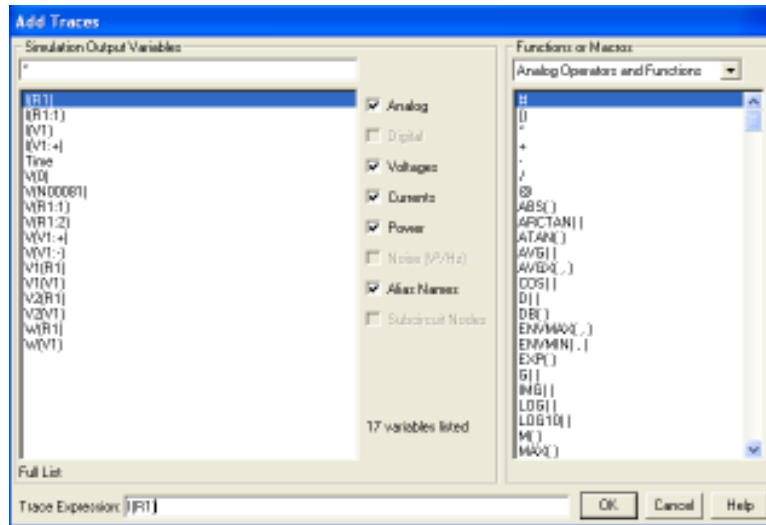


Figure 9 – Add Trace Dialog

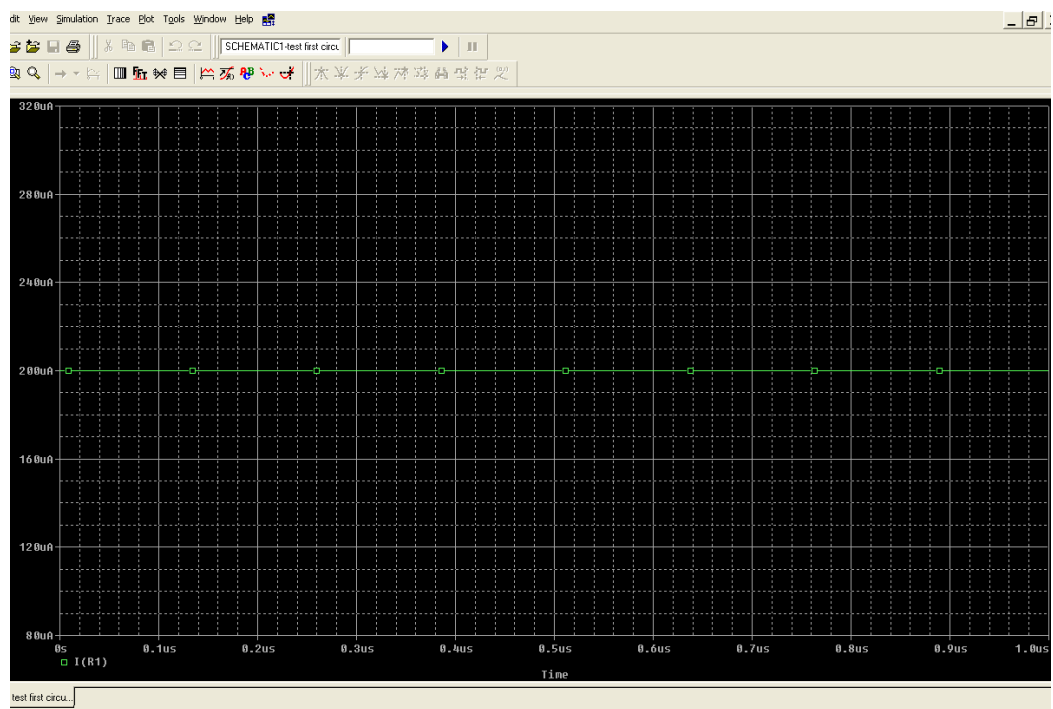


Figure 10 – Simulation Waveform

- e. Close the PSPICE A/D window. Go back to the Capture CIS window, select Pspice → Edit Simulation Profile.

- f. Change the Analysis Type to “DC sweep”. DC sweep allows you to vary the source(s) values and monitor circuit operation according to the variations. In the “Sweep Variable” part, choose “Voltage source” and Name: V1. In the “Sweep Type” part, choose “linear”, fill in start value 0V; stop value 5V and increment 0.001V. (You don’t need to indicate “V” in the textbox as volt is default unit.) See Figure 11 as well.

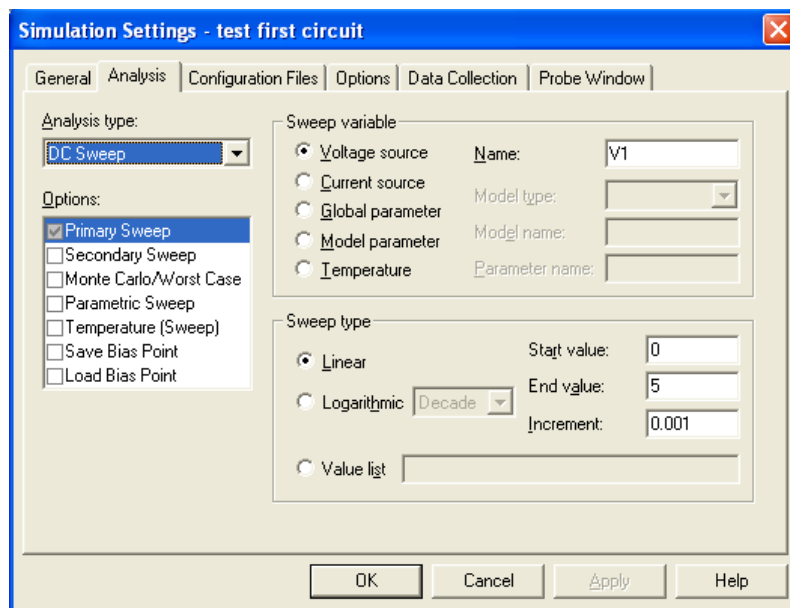


Figure 11 – DC Sweep simulation settings

- g. Click Ok. Follow Procedure c and d to run the simulation again. You should see the simulation waveform as shown in Figure12. **Based on the waveform, what principle are we verifying?**

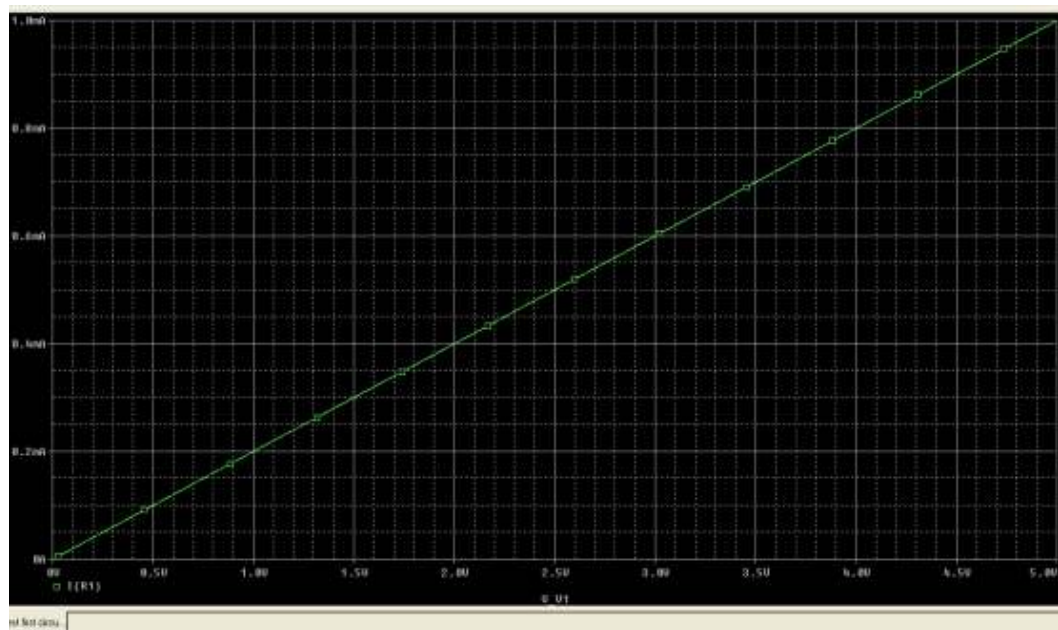


Figure 12 – DC Sweep waveform

**Exercises** (Show your simulation results to your lab GAs after you finish **EACH** question.)

1. The circuit shown below has an ideal current source  $I1 = 10\text{mA}$ ,  $R1 = 30\text{k}\text{-ohm}$ . Simulate the circuit using **both transient analysis and DC sweep** ( $I1$  start from 0 and stop at  $100\text{mA}$  with increment  $0.1\text{mA}$ .) to plot the voltage across  $R1$ .

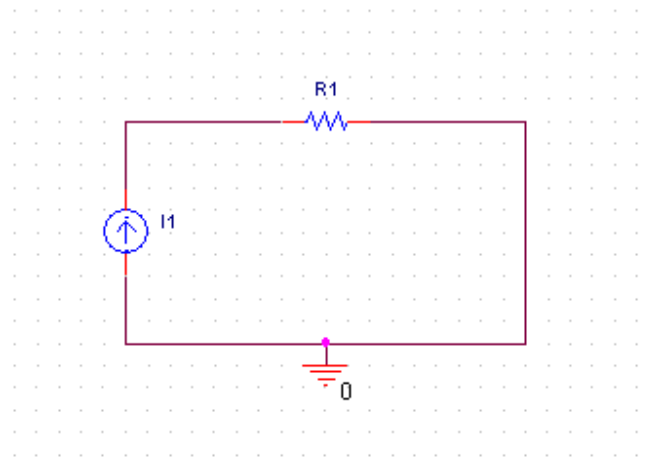


Figure Q1 – Ideal Current Source Circuit

2. (a) Find the equivalent resistance between Node A and Node B by using PSPICE simulation. (Hint: You need to connect test voltage or current sources for the simulation.)

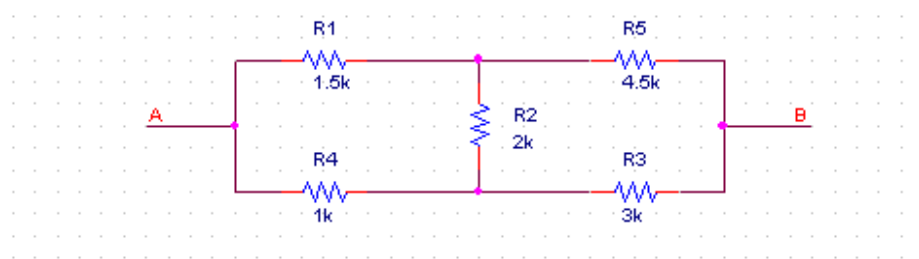
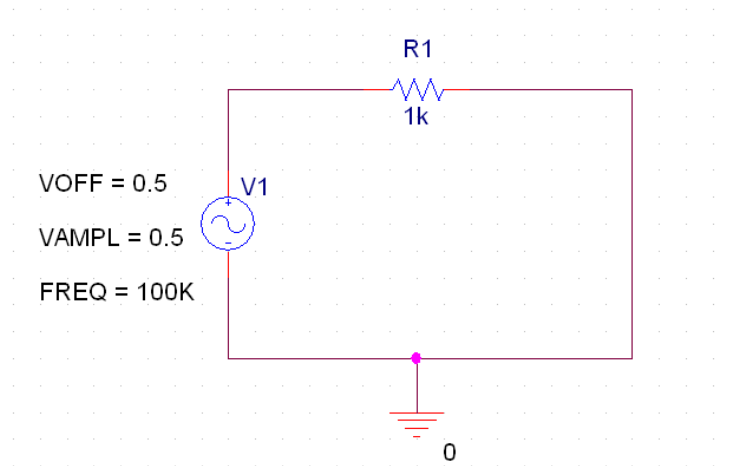


Figure Q2 – Equivalent Resistance Circuit

- (b) Eliminate  $R2$  in Figure Q2 and run the simulation again. What difference of equivalent resistance from (a) are you getting? Why?

3. Draw circuit schematic according to Figure Q3. Given that the voltage source V1 is a sinusoidal wave (VSIN in SOURCE library) with offset voltage 0.5V, amplitude 0.5V, and frequency 100k Hz. Run the simulation using **transient analysis** and plot the current flowing through R1.



4. Draw circuit schematic according to Figure Q4. Verify, using PSPICE simulations, KVL in all of the three loops L1, L2 and L3. Also, verify KCL for Node A, B and C.

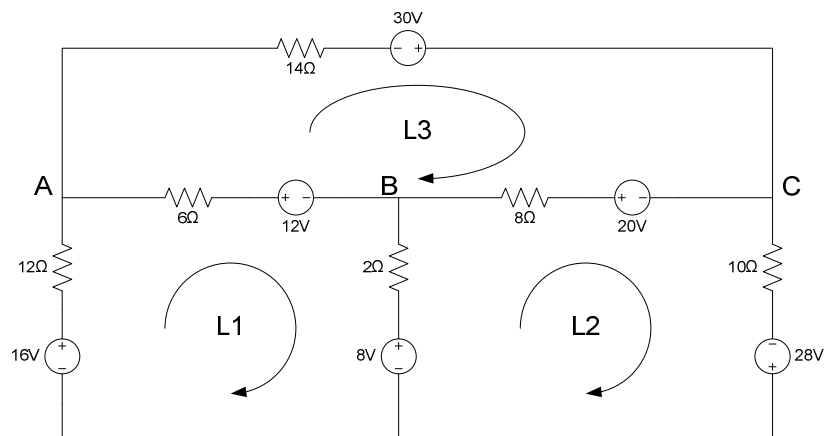


Figure Q4

5. Find, using PSPICE simulation, the **Thevenin equivalent voltage** and **Thevenin equivalent resistance** that the load  $R_L$  sees for the circuit of Figure Q5. Complicated calculation should not be involved.

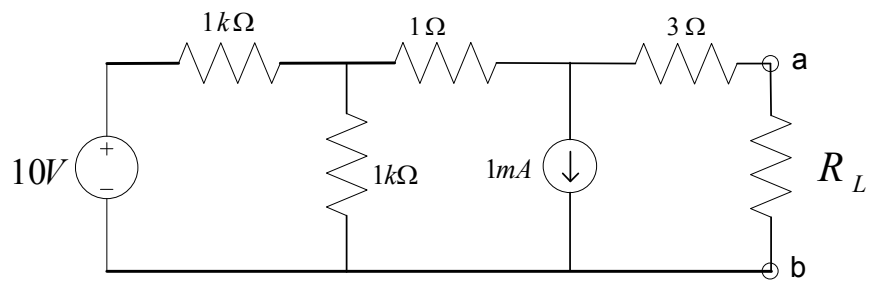


Figure Q5