

1. (Lab #5) PSpice Analysis of DC Circuits

Objective:

Help you analyze a DC circuit. A DC circuit is the one in which the source of the circuit is of direct current rather than alternative current. At the start of the tutorial a brief introduction of DC (direct current) is provided after that a simple DC circuit is implemented in PSPICE and the results were analyzed precisely. At the end of the tutorial you are provided with an exercise to do it by yourself, and in the next tutorials I will assume that you have done those exercises and I will not explain the concept regarding them.

Introduction to a DC circuits

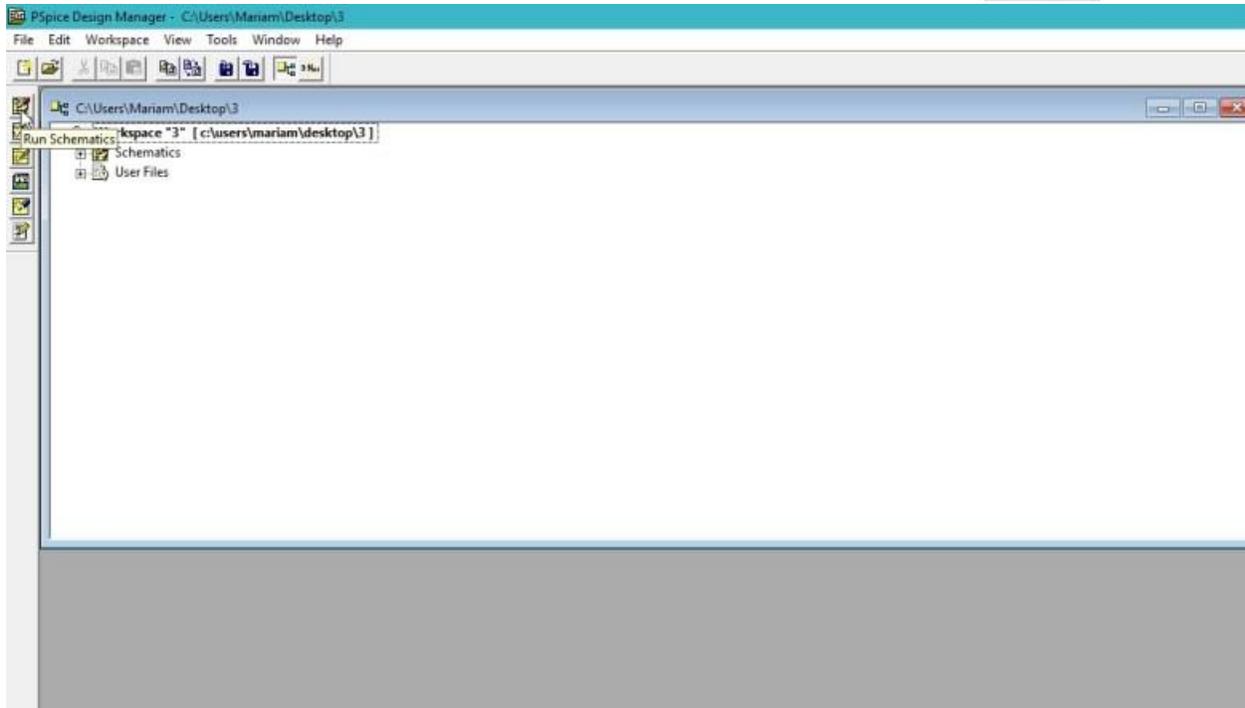
DC circuit is the circuit in which direct current source is used. Direct current is defined as the constant flow of electric charge traveling from high potential to low potential. Similarly, the circuit that contained electric current flow in only one direction is known as direct current circuit. The relationship between voltage, current and resistance is defined using ohms' law. Ohms' law states as

The voltage is directly proportional to the current in a circuit provided that the resistance (and the temperature) remains constant. In this tutorial this law is applied and verified using PSPICE as you will see shortly.

Tasks for the (Lab #5): PSpice Analysis of DC Circuits

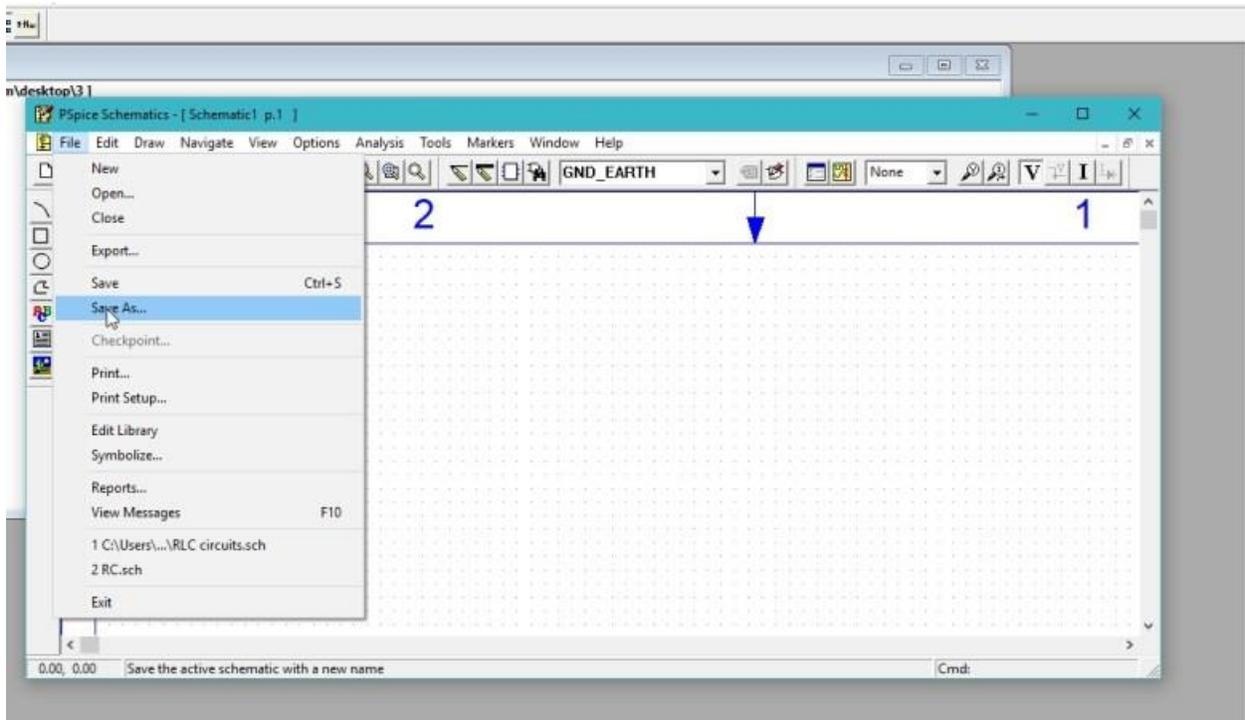
DC circuits analysis with PSpice

- Lets' design a simple DC circuit i.e. a circuit with DC source as a supply. Open the PSPICE design manager on your PC by typing design manager in the search bar. From the design manager click on the run schematic button to open a new blank schematic as shown in figure (5-1)



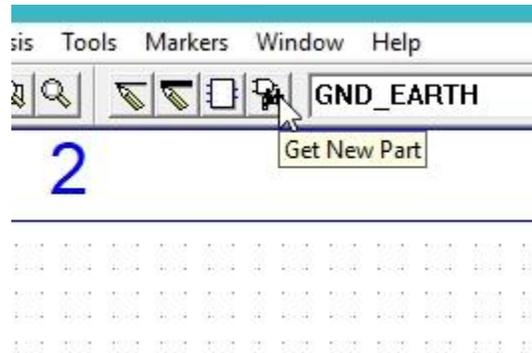
[Figure 5-1] Starting PSpice blank schematic

- After opening the new schematic before jumping into designing first save the schematic by clicking on the file button at the top left corner and then selecting save as so that we can access it anytime in the future. Refer to figure (5-2)



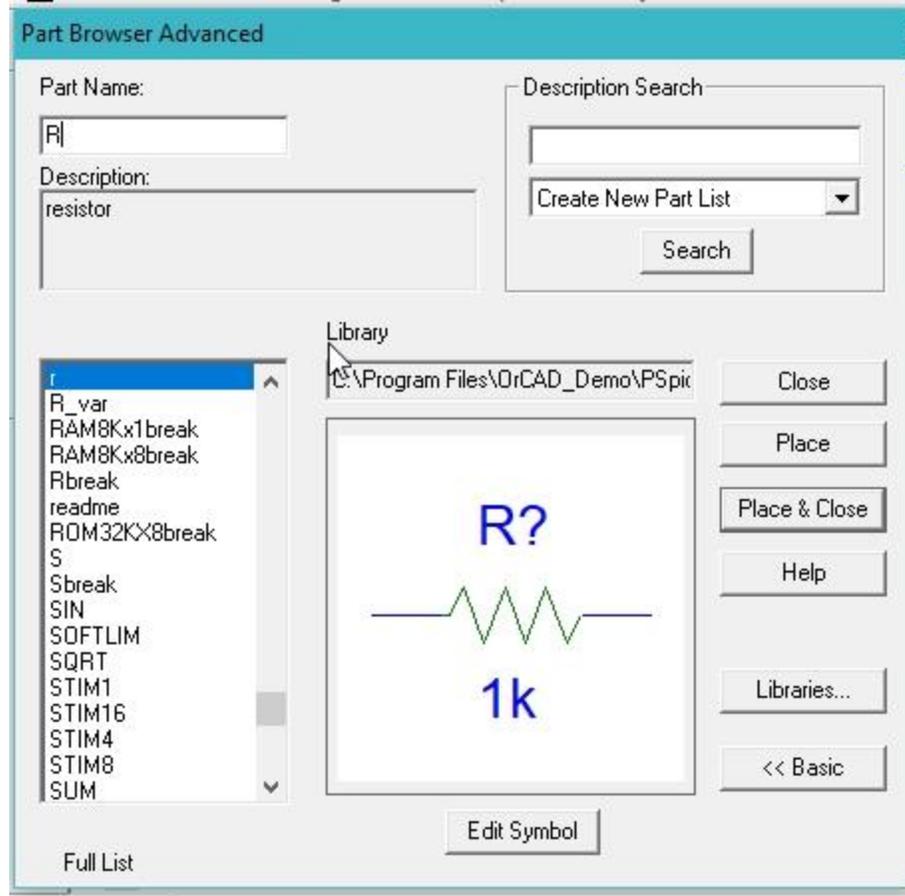
[Figure 5-2] Saving schematic

- Click on the get new part icon at the top bar of the schematic window in order to search for the components that are needed for circuit designing.



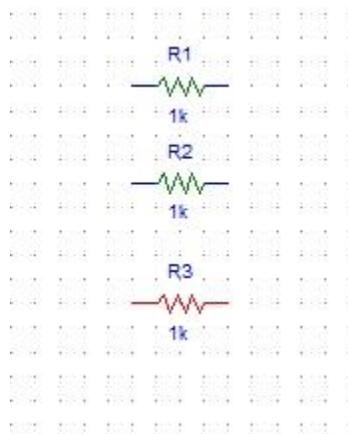
[Figure 5-3] Getting new part

- In the get new part window, type 'r' it will display a list of resistors available in PSPICE. From that list select a simple resistor as shown in figure (5-4)



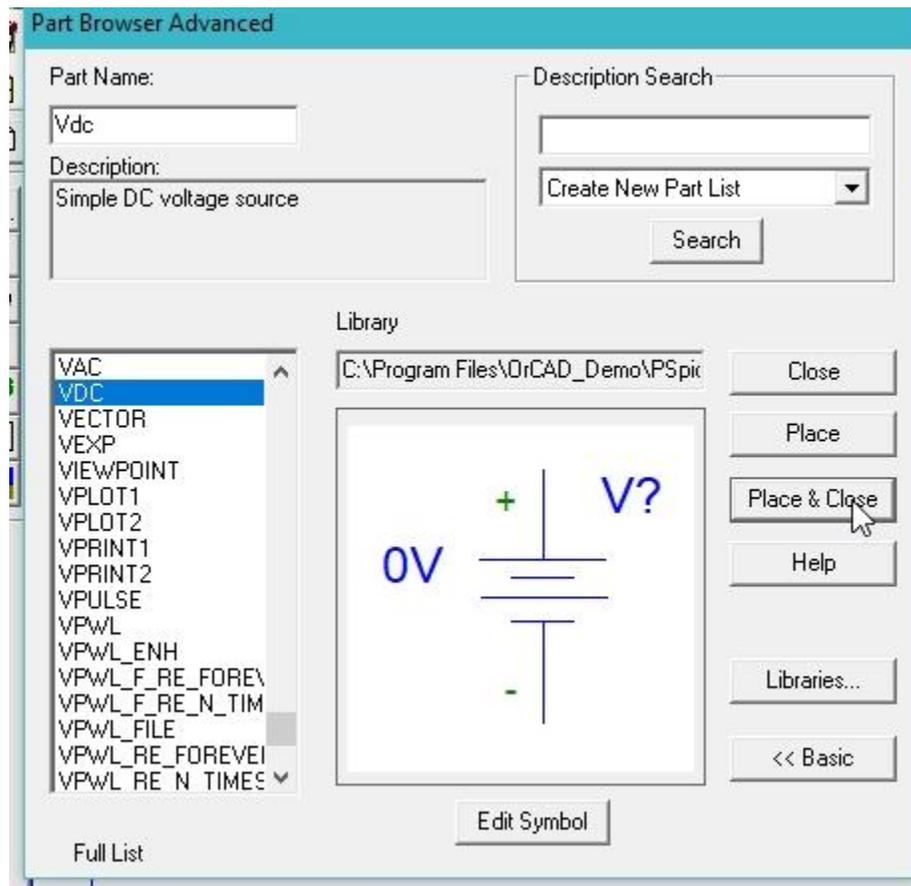
[Figure 5-4] Place Resistor

- After selecting the resistor place three resistors on the schematic you created previously in such a manner that each resistor is in parallel with each other. Refer to the figure below to see the arrangement of the resistors.



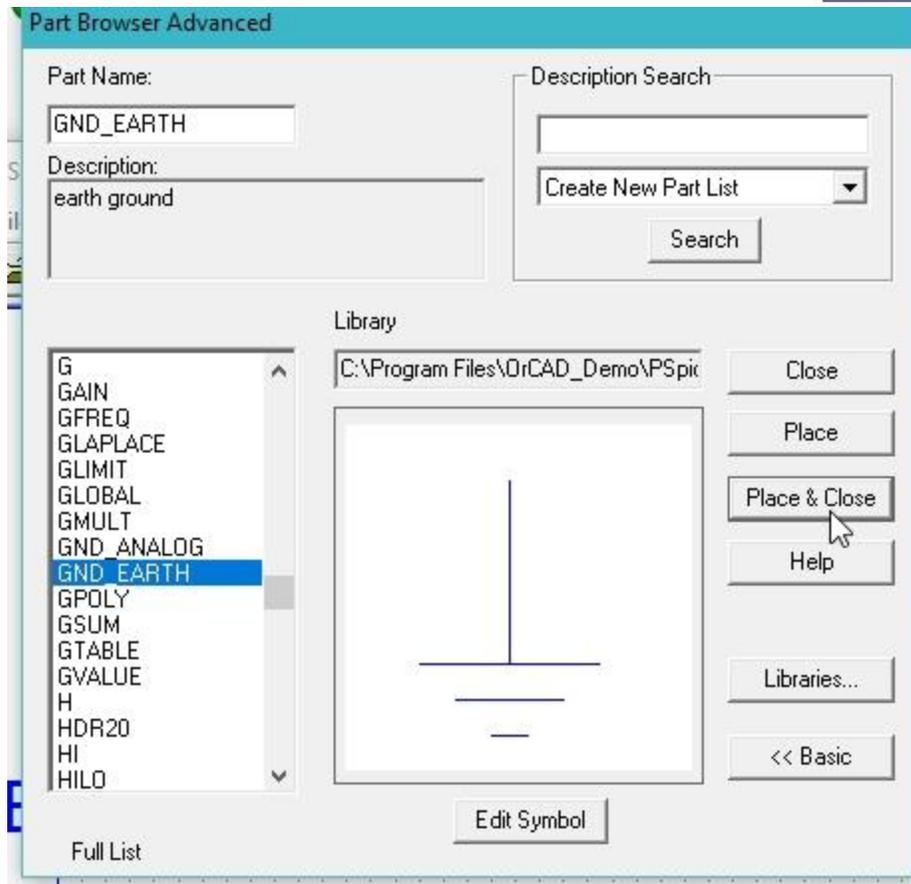
[Figure 5-5] Resistors placed

- Again open the get new part window and in the part name block type Vdc, select the supply from the list given and then click on place & close as shown in figure (5-6)



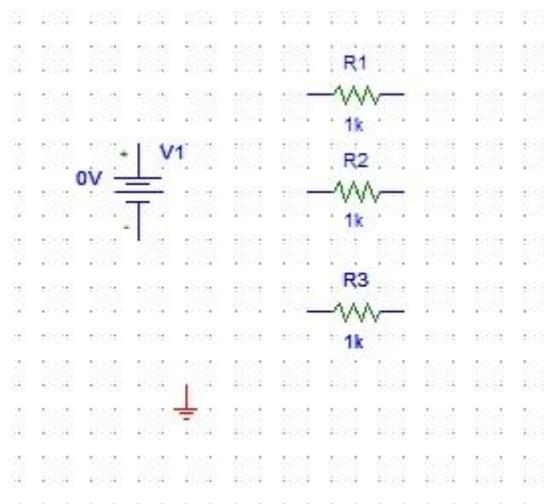
[Figure 5-6] Input dc placed

- Next step is to place a ground, do the same again and in the part name type Gnd and select the ground and then click on place & close as shown in Figure (5-7)



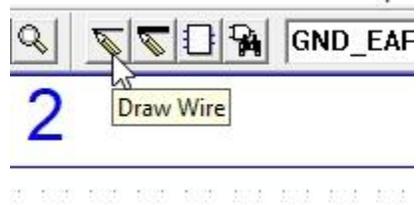
[Figure 5-7] Ground placed

- The placed components in the schematic window are shown in figure (5-8)



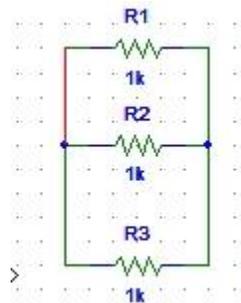
[Figure 5-8] Placed components

- Click on the draw wire icon at the top bar of the schematic window in order to connect the already placed components for circuit designing.



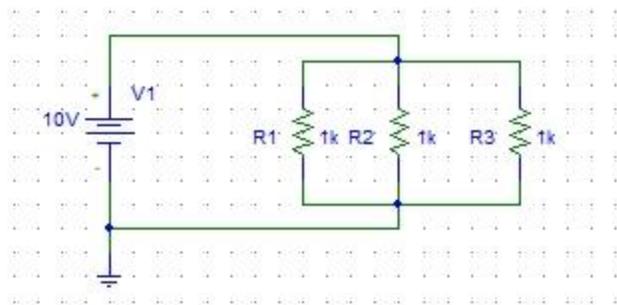
[Figure 5-9] Drawing wire

- Connect all the resistors in parallel with the wire as shown in figure (5-10)



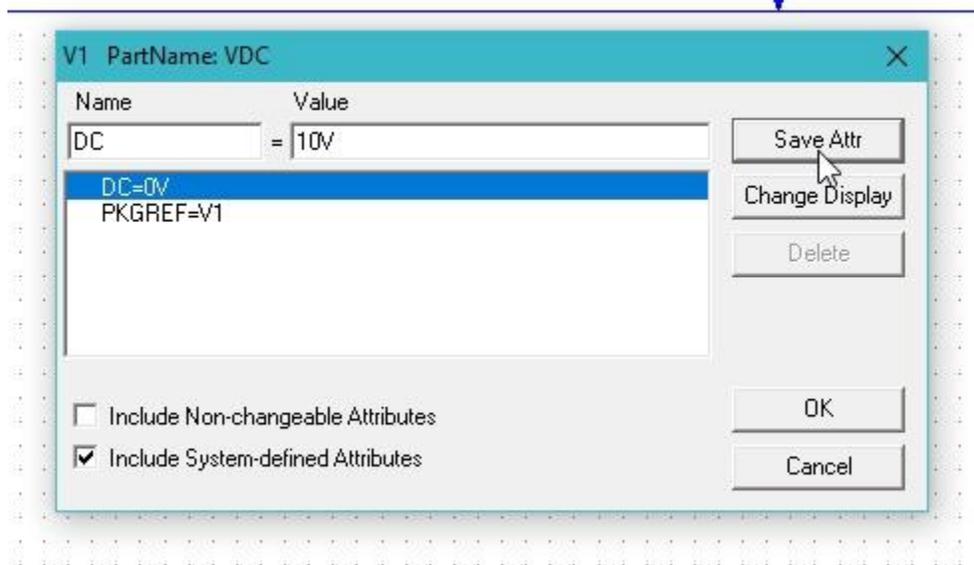
[Figure 5-10] Parallel resistor

- Connect the remaining components to complete the circuit diagram as shown in figure (5-11)



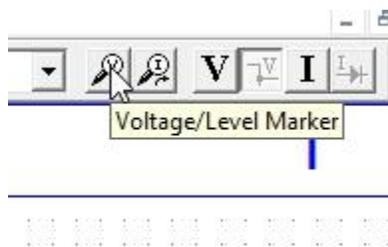
[Figure 5-11] Complete circuit diagram

- Next step is to set the attributes of the input dc supply. Double click on the dc supply you connected in the circuit previously and set the dc voltage of the supply to 10V as shown in figure (5-12)



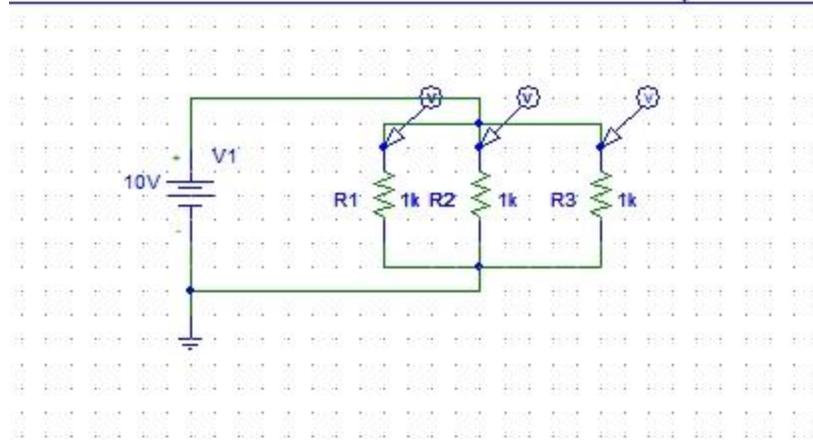
[Figure 5-12] DC attributes

- On the top of the schematic window, click on the Voltage/Level Marker button as shown in the figure (5-13)



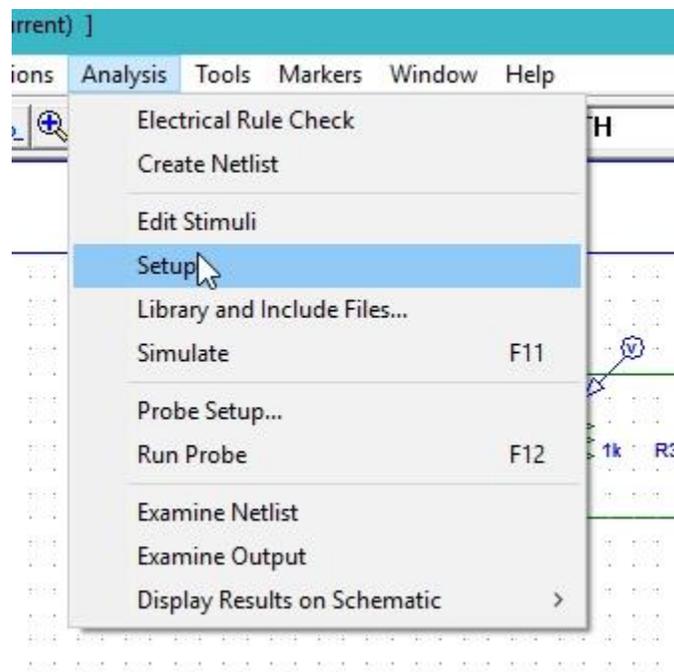
[Figure 5-13] Voltage marker

- Place it at each resistor node as shown in the figure (5-14) below,



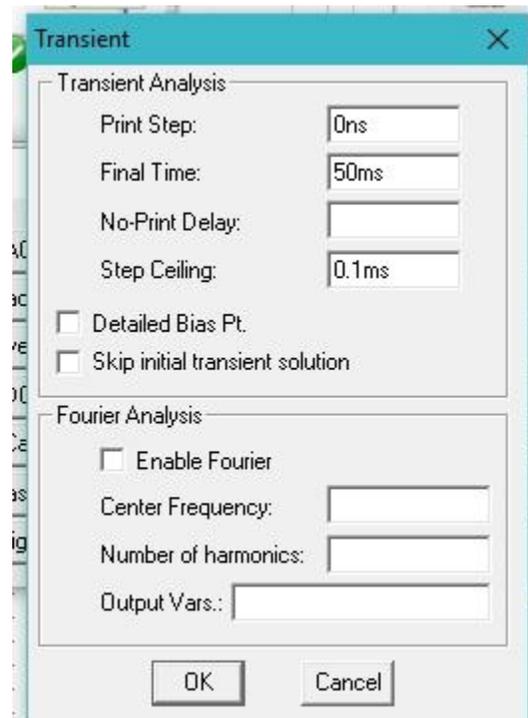
[Figure 5-14] Voltage marker placed

- Next step is to adjust the properties of the simulations in order to produce the graph of the voltage at the marker. Click on analysis and then click on Setup as shown in figure (5-15)



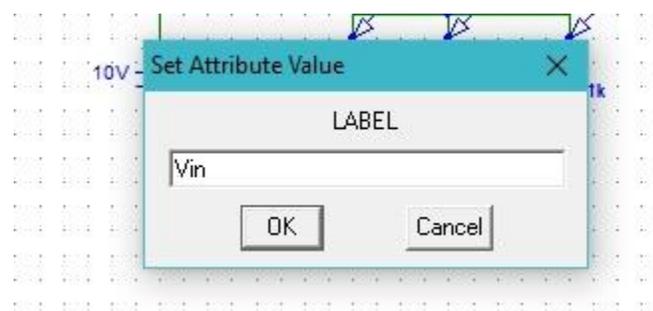
[Figure 5-15] Analysis setup

- A widow will appear, click on the transient block on the window and adjust the properties of the window according to your requirement, refer to figure (5-16)



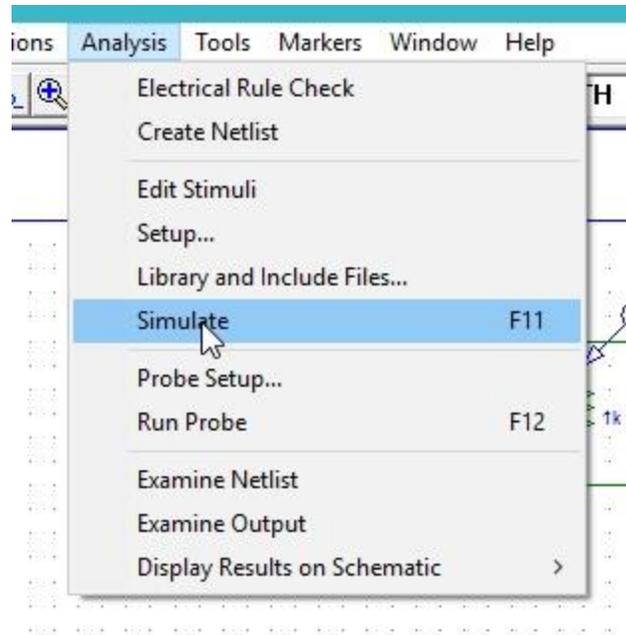
[Figure 5-16] Transient properties

- If we are interested in checking the voltage on a specific wire in spite of checking it at a node, double click on the wire and inn the window that appear as a result, type the name of the wire you want to label it with, as shown in figure (5-17)



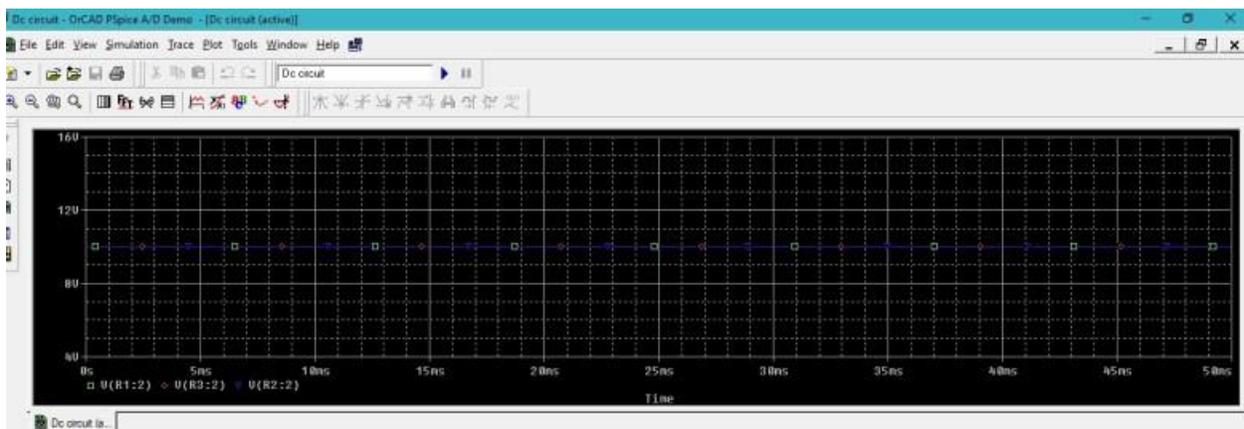
[Figure 5-17] Labeling the wire

- Now comes the simulation part, click on the analysis at the top bar of the schematic window and then click on simulate as shown in figure (5-18)



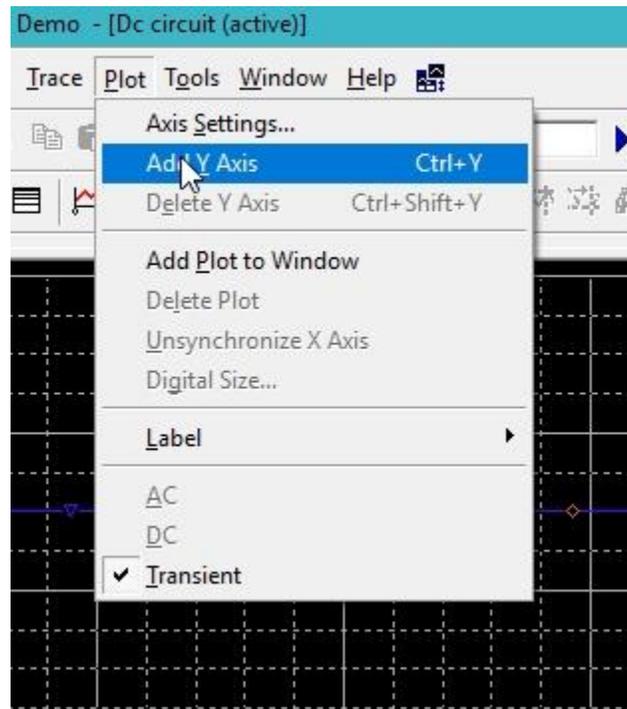
[Figure 5-18] Simulation

- As all the resistors are in parallel and we know that the voltage remains the same in parallel regardless of the value of the resistances connected in parallel. If two or more resistors are connected in parallel, then the very basic concepts of electrical engineering provide us with the information that the voltage across each of the parallel connected resistances remains same no matter what value we assign to the resistance. The voltage supplies to the parallel resistor appears across every resistor connected parallel to it. This concept is illustrated in the output of the circuit we simulated above. Refer to the figure below which shows the output of the circuit designed above.



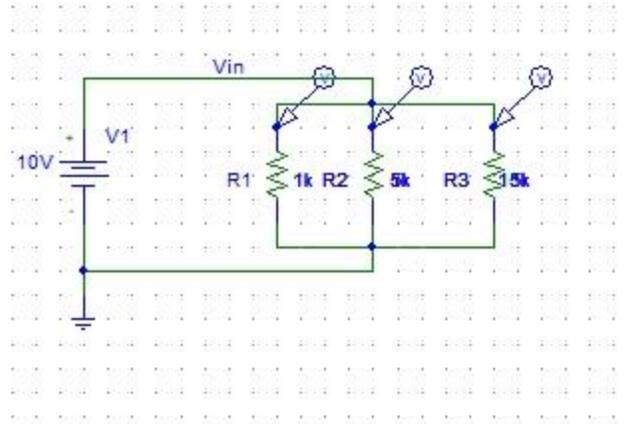
[Figure 5-19] Output Voltages

- The voltage at each node of the parallel connected resistor should be same and the simulation depicts the same as is obvious from the above figure. Now we need to plot the currents instead of the voltages hence the currents values will vary in parallel according to the value of the resistances using ohms' law stated in the introduction part.
- On the simulation window select plot and then select Add y axis as shown in figure (5-20)



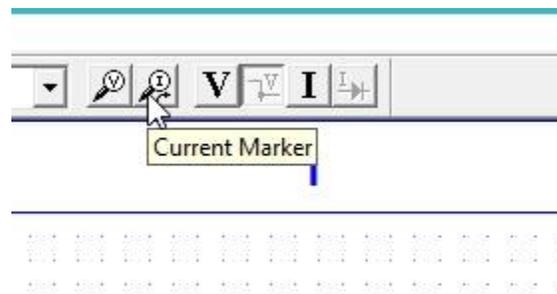
[Figure 5-20] Adding Y axis

- This will add a plot of variable Y axis according to the value of the trace we are going to plot. Now go the schematic and change the values of the resistances, so that the value of the current in each branch vary according to the value of the resistance using ohms' law as shown in figure (5-21)



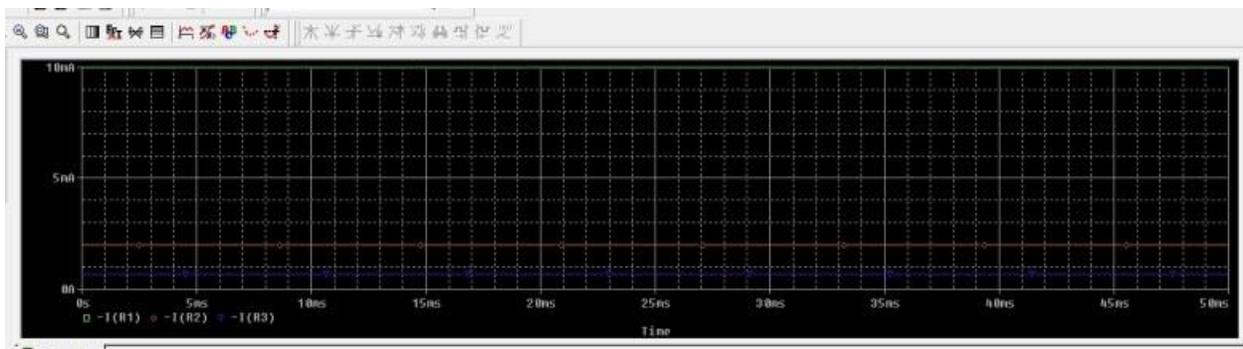
[Figure 5-21] Updating resistance value

- Also change the marker from the voltage marker to the current marker as shown in figure (5-22)



[Figure 5-22] Current marker

- Simulate the circuit as we have done before, now the output of the circuit will show three different current depending upon the value of the resistor as shown in figure (5-23)



[Figure 5-23] Current output