➢ Installing PSPICE 9.1

- To download PSPICE 9.1 student version, visit the following link: http://www.electronics-lab.com/downloads/circutedesignsimulation/?page=5
  You will find a list of programs, scroll down until you see **PSPICE 9.1 Student Version**, then click download. The setup is straight forward, however, in case you needed help check the following video and follow steps. https://www.youtube.com/watch?v=tCFjjHY94Ro

➢ Building and simulating circuits

- To open PSPICE 9.1, search your computer for **"Schematic"** and open the program. This is shown in figure.1 for windows 10 users. **Note: zoom in to see figure details if not clear.**

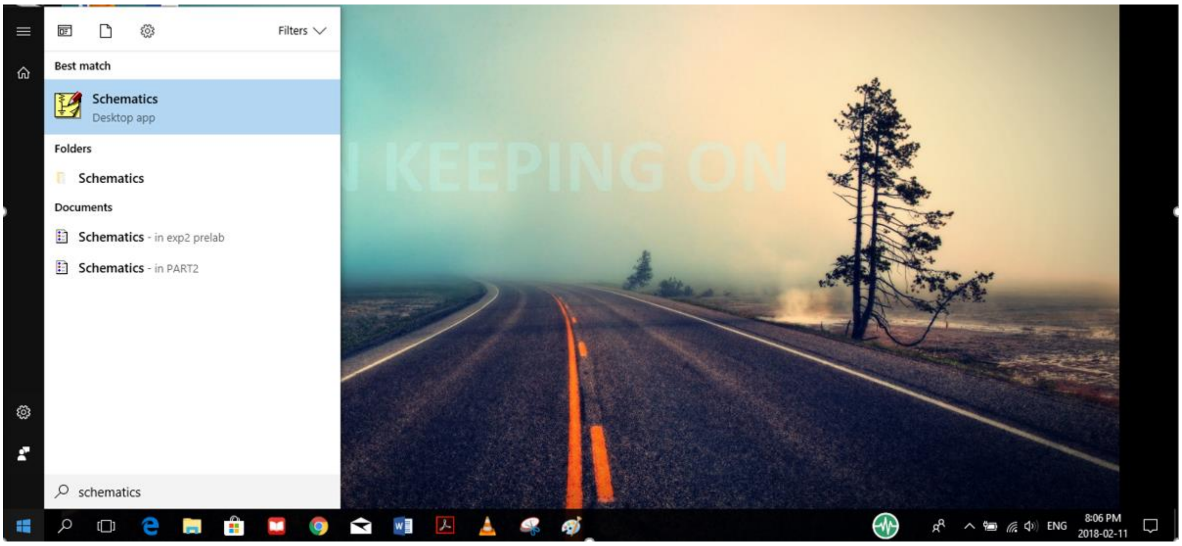


Figure.1

- Click on Schematics icon, then you will see the window shown on figure.2

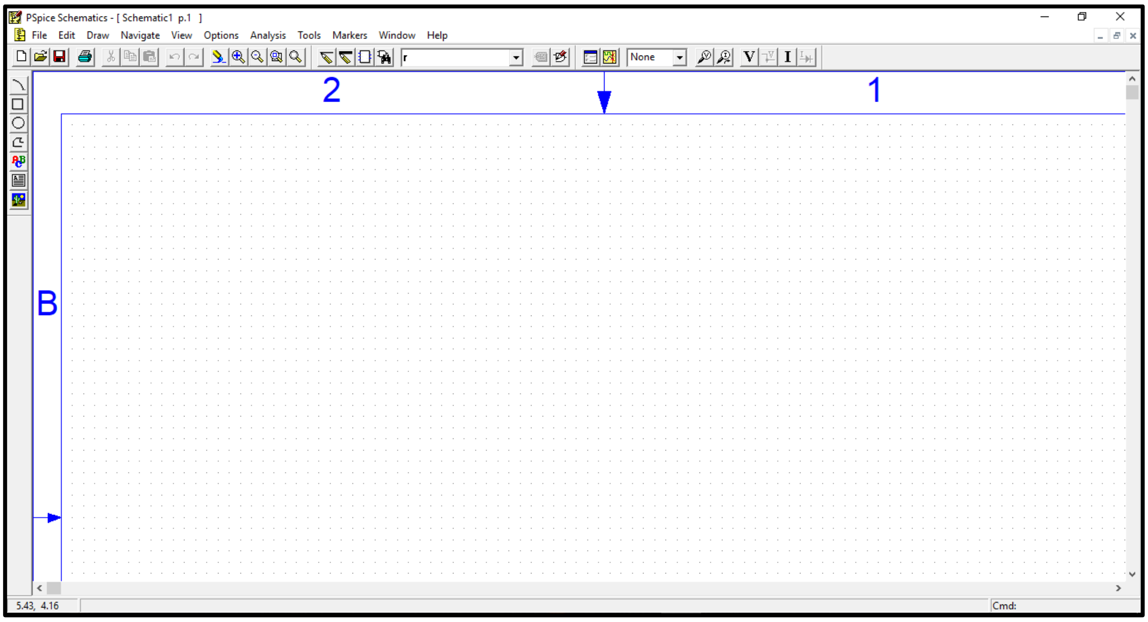


Figure.2

- To build the circuit of figure.3 on PSPICE, click on **"get new part"** icon shown in figure.4:

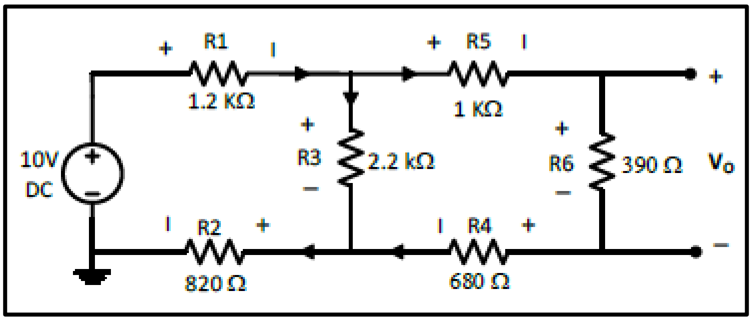


Figure.3

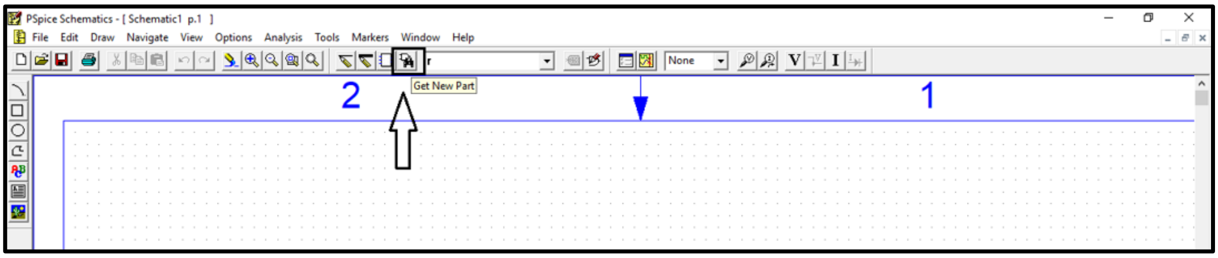


Figure.4

- When you click on get new part icon, you will see the window shown on the left side in figure.5. This is a list of all components available in the program. Click on "advanced" so you can see the picture of part you want to add as shown on the right side in figure.5.

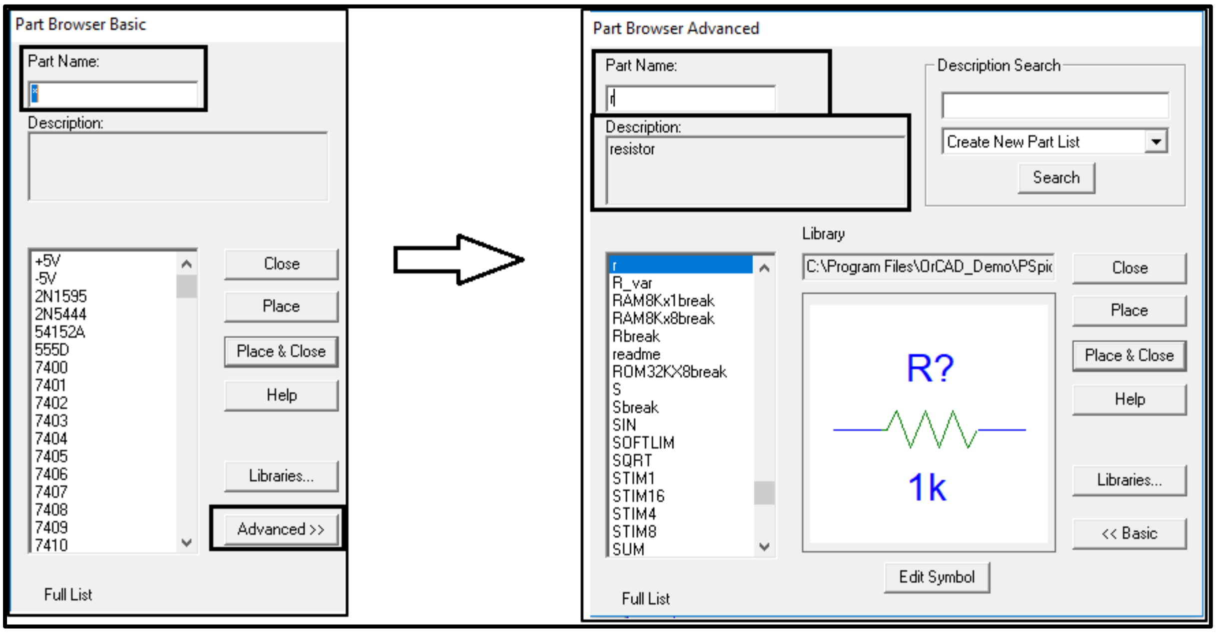


Figure.5

- To add a circuit part, you can search by typing part name in the specified field. After finding the part you want, click on "place and close", then click wherever you want to add a part. After you finish adding components, click on "esc" on your keyboard to end the mode.

- Note that each circuit element in PSPICE have a specific name that might be different from its name in circuit analysis. Table.1 provides the names of mostly used circuit elements in the circuit lab.

Table.1

Circuit element	Part name in PSPICE
Resistor	R
Capacitor	C
Inductor	L
DC Voltage source	vdc
DC Current source	Idc
Periodic square voltage source	vpulse
AC Voltage source: two types	vsin (sinusoidal voltage source used in transient analysis)
	vac (variable frequency source used in ac sweep analysis)
ground	gnd_analog
Operational amplifier	ua741

- Note that when you place parts, each circuit part in PSPICE has a name and a value as shown in figure.6 Always make sure to place circuit parts in PSPICE in a similar way to the circuit given in manual. For example, check the way the parts are placed in figure.6.

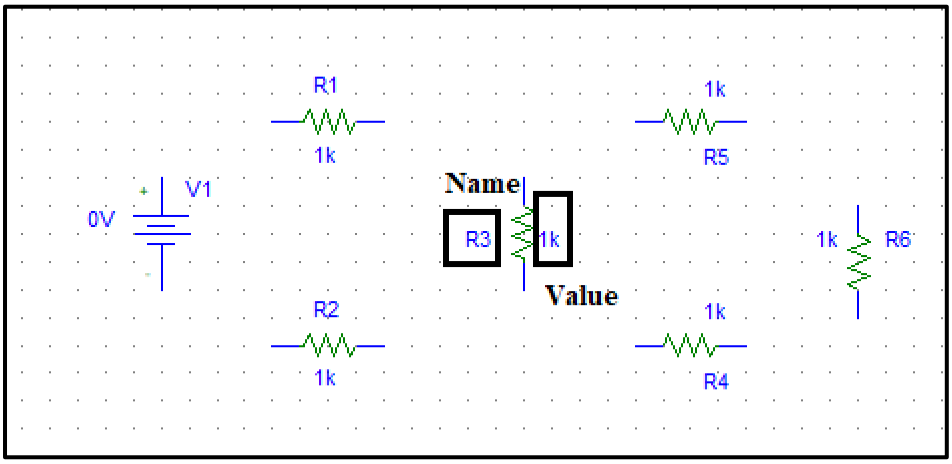


Figure.6

- To connect the components, click on "draw wire" icon shown in figure.7, **a common mistake** that students do is clicking on "draw bus" icon which is next to "draw wire". Avoid doing this mistake! also avoid drawing wire over components (shorting parts). Click where you want each vertex of the wire. Each click ends a wire segment and starts a new one

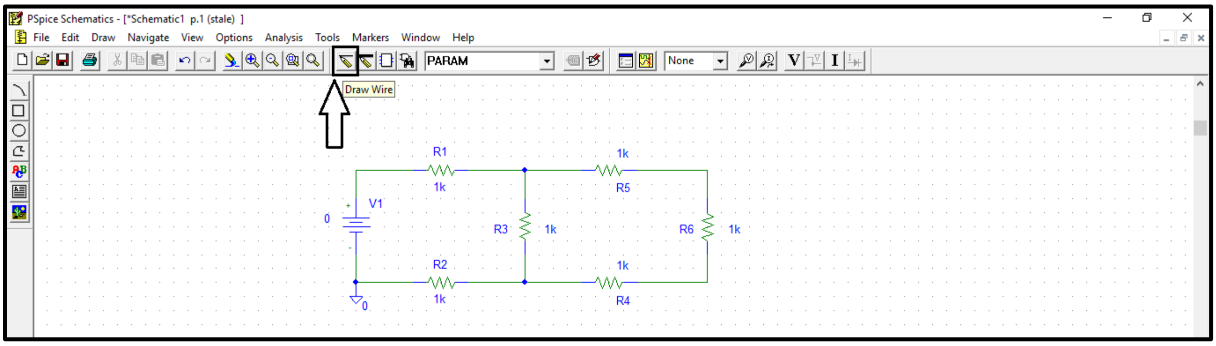


Figure.7

- To change a part value e.g. resistance, voltage source … etc. double click on its value and the window in figure.8 will show up, type the value you want inside the box then click ok.

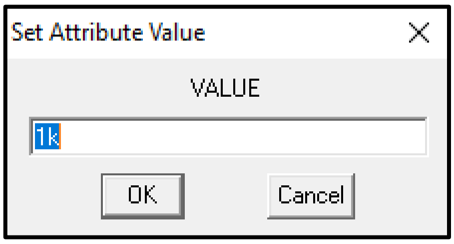


Figure.8

- When elements such as resistors and voltage sources are given values, it is convenient to use unit prefixes. PSPICE supports the prefixes listed in table.2. Note that the letter must immediately follow the value – no spaces. Also, PSPICE is case insensitive so, there is no difference between 1M and 1m in PSPICE.

Table.2

PSPICE Unit Prefixes		
K - kilo - 10^3	MEG - mega - 10^6	G - giga - 10^9
M - Millie - 10^{-3}	U - micro - 10^{-6}	N - Nano - 10^{-9}

- **PSPICE requires that all schematics have a ground**, the voltage there will be zero and all other node voltages are referenced to it. **If you do not place a ground, you will get an error and will not be able to simulate your circuit**. The part you need is either the analog ground (GND_analog) or the earth ground (GND_earth) which are equivalent, you can get them from “get new part option”. In this example, we used GND_analog.
- Your schematic is finished now and ready for saving, click on “save” then on “simulate” button as shown in figure.9.

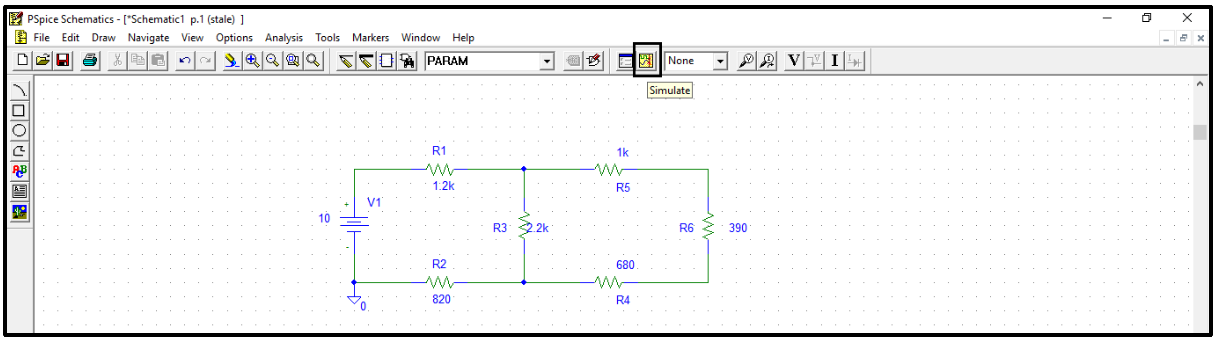


Figure.9

- When you click on "simulate" button the window shown in figure.10 will show up (simulation output window), if your circuit doesn't have any errors you will see the message in the box in figure.10.

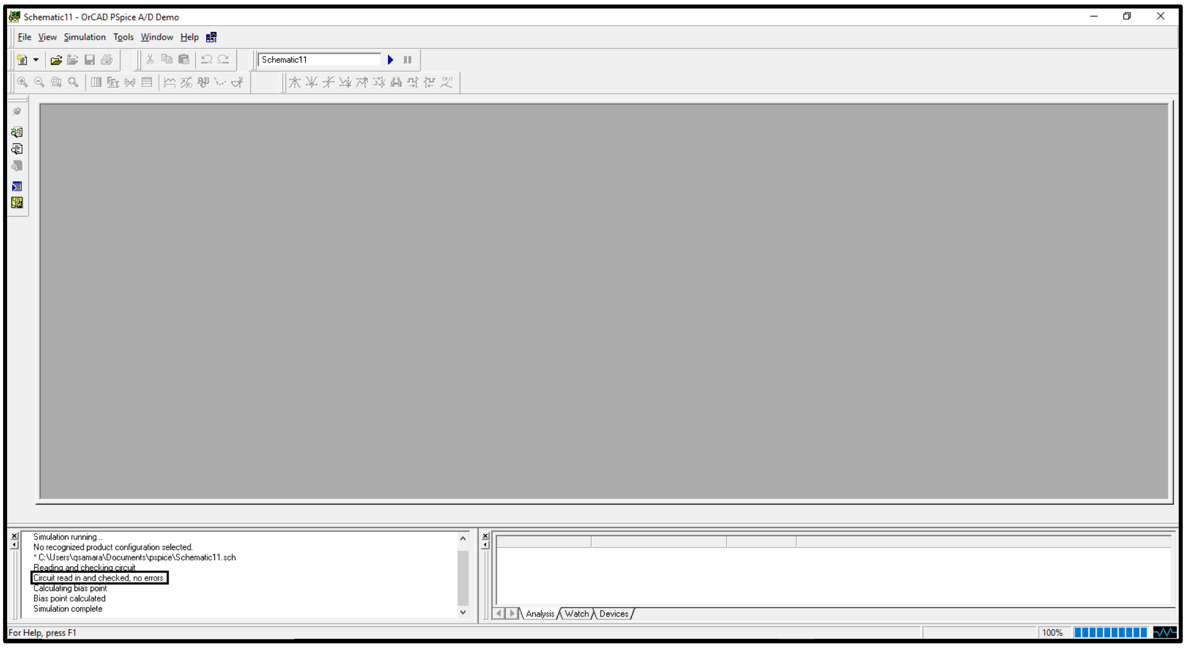


Figure.10

- Return to the schematics window and click on **"V"** and **"I"** buttons shown in figure.11. By clicking on these buttons, PSPICE displays the voltage on each node "**with respect to ground"** and the current on each branch.

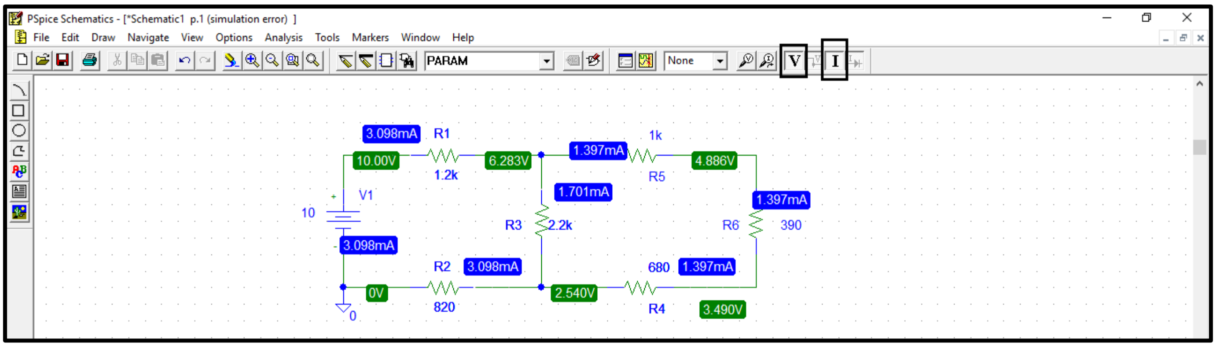


Figure.11

- **The Passive Sign Convention and PSPICE:**

  All currents and voltages in PSPICE and Schematics obey the passive sign convention shown in Figure.12. The voltage across the element is defined positive at node 1 with respect to node 2, and current is entering a device from its “1” end and leaving its “2” end.

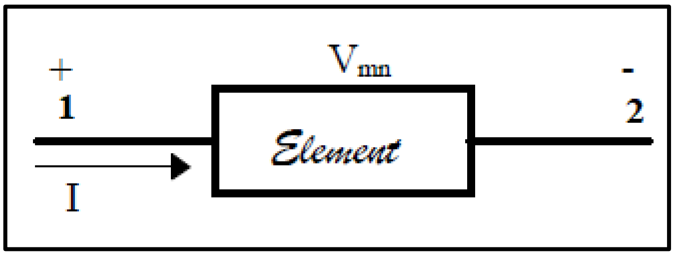


Figure.12

All two leaded passive components have an **implied** “1” end and a “2” end (not visible on schematic). Whenever you place a component, it takes a default position, for example, a resistor, capacitor, or inductor will take a default position with its “1” end to the left as shown in figure13-(a). A component may be rotated by activating it, then right-clicking and selecting Rotate, or by typing the letter “r” (see b). Each rotation moves the component counterclockwise by 90º. To get the “1” end facing up, you must rotate the component 3 times from its default position as indicated in (c).

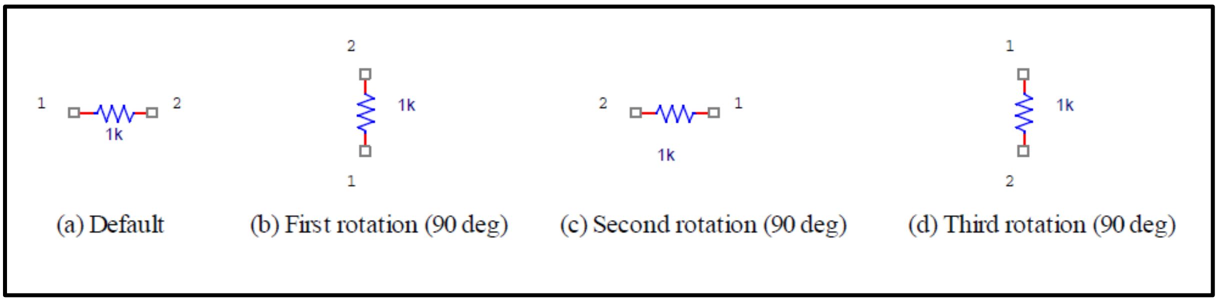


Figure.13

- Knowing about component layout is important when you are viewing your results in Probe to be discussed later.

➢ Types of Analysis Performed by PSPICE

PSPICE is capable of performing four main types of analysis: Bias Point, DC Sweep, AC Sweep/Noise, and Time Domain (transient).

- **Bias Point:**

  The Bias Point analysis is the starting point for all analysis. In this mode, the simulator calculates the DC operating point of the circuit. **This is the type which was explained in the previous section.**

- **DC Sweep**

  The DC Sweep analysis varies a circuit part e.g. voltage source, global parameter (resistor) over a specified range in an assigned number of increments in a linear or logarithmic fashion.

- **Time Domain (transient)**

  The Time Domain (transient) analysis is probably the most popular analysis. In this mode, you can plot the various outputs as a function of time.

- **AC Sweep/Noise**

  The AC Sweep/Noise analysis varies the operating frequency in a linear or logarithmic manner. It linearizes the circuit around the DC operating point and then calculates the network variables as functions of frequency.

➢ Example on DC Sweep

- To perform a DC sweep for the same circuit of the previous section, click on the "setup analysis" icon shown in figure 14.

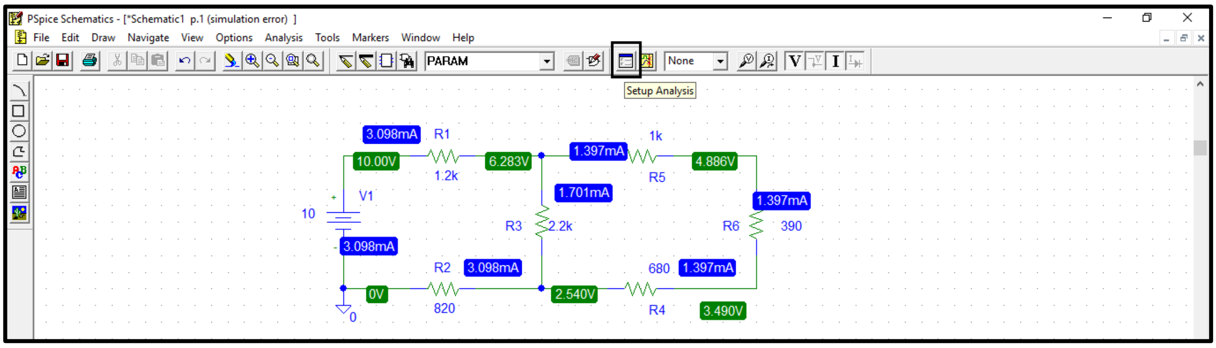


Figure.14

- The window shown in figure.15 will show up:

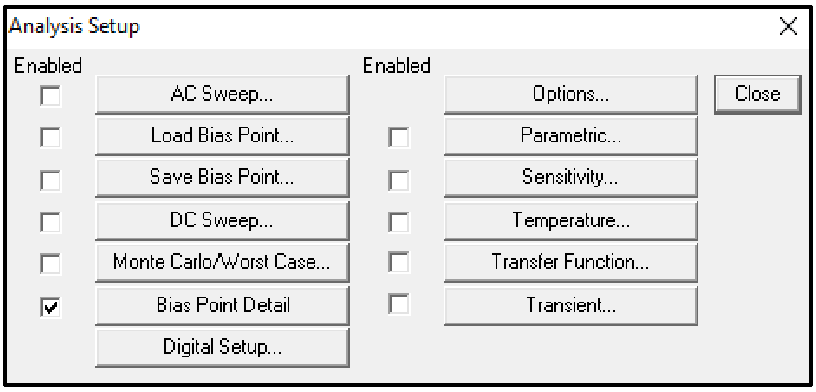


Figure.15

- Click on "DC sweep" and the window shown in figure 16 will show up, you have to fill the spaces indicated inside boxes in figure.16.

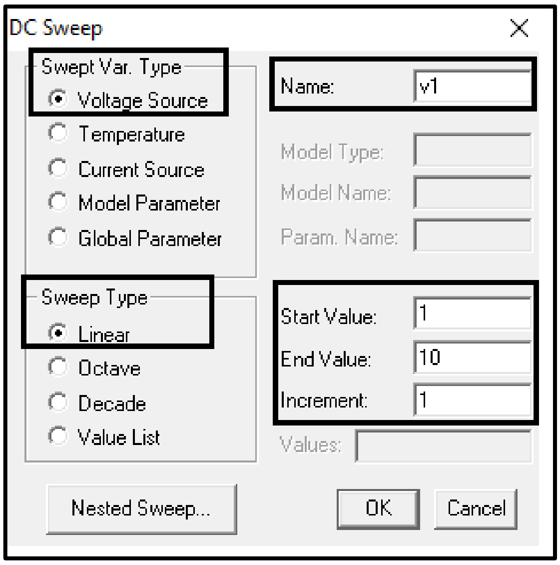


Figure.16

- Click on simulate, then the simulation output window will be as shown in figure.17:

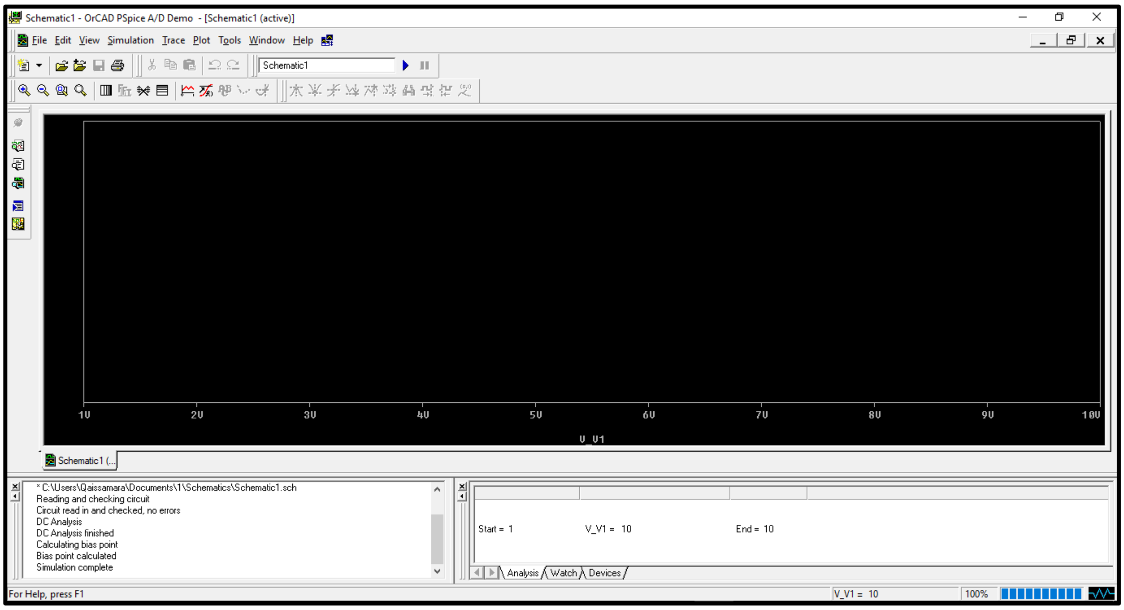


Figure.17

- The output now is displayed as curve rather than a point, to display the voltage across R6 for example click on "Voltage Marker" and place it where you want as shown in figure.18.

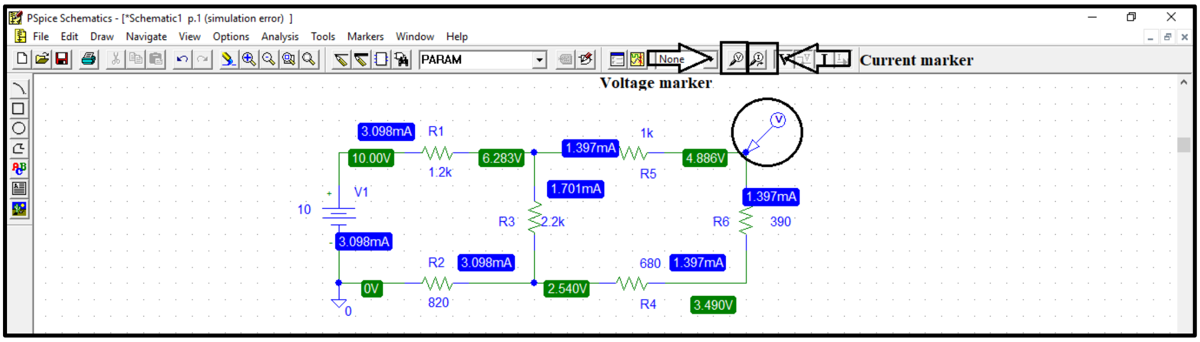


Figure.18

- When you place the marker, the output simulation window will be as shown in figure.19. this is a curve of the voltage at the node you placed the marker **with respect to ground.**

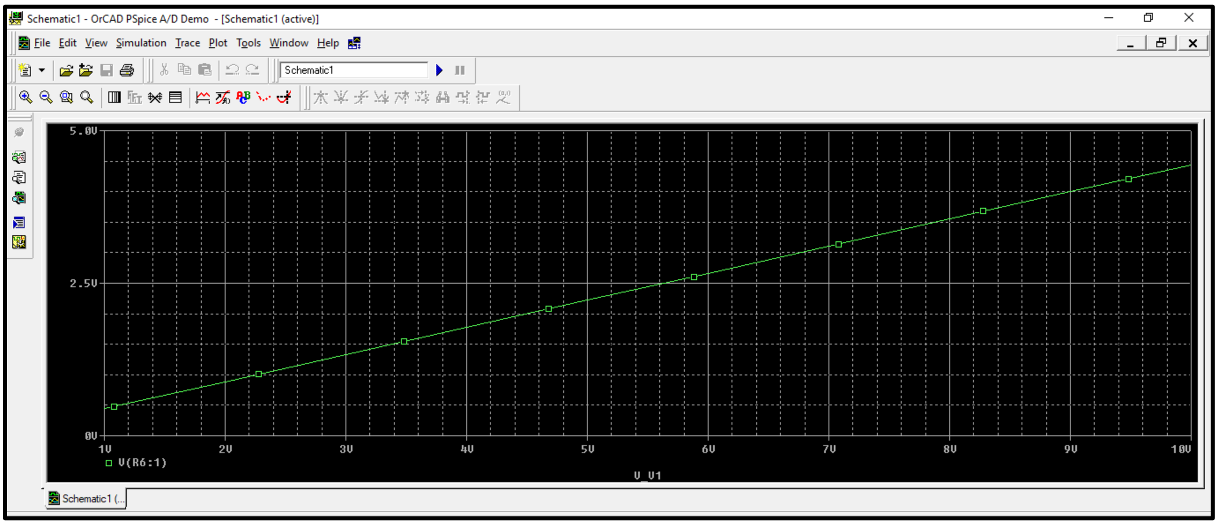


Figure.19

- If you want to display the voltage on R6 only, then go to schematic window, click on marker, and select "mark voltage differential" as shown in figure.20. (you will place two markers)

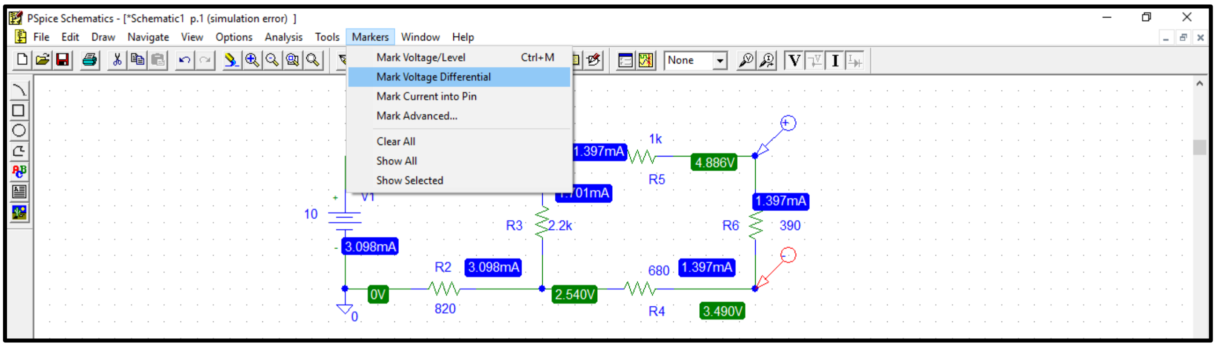


Figure.20

- The output now will be as shown in figure.21. **Note the difference on y-axis range!**

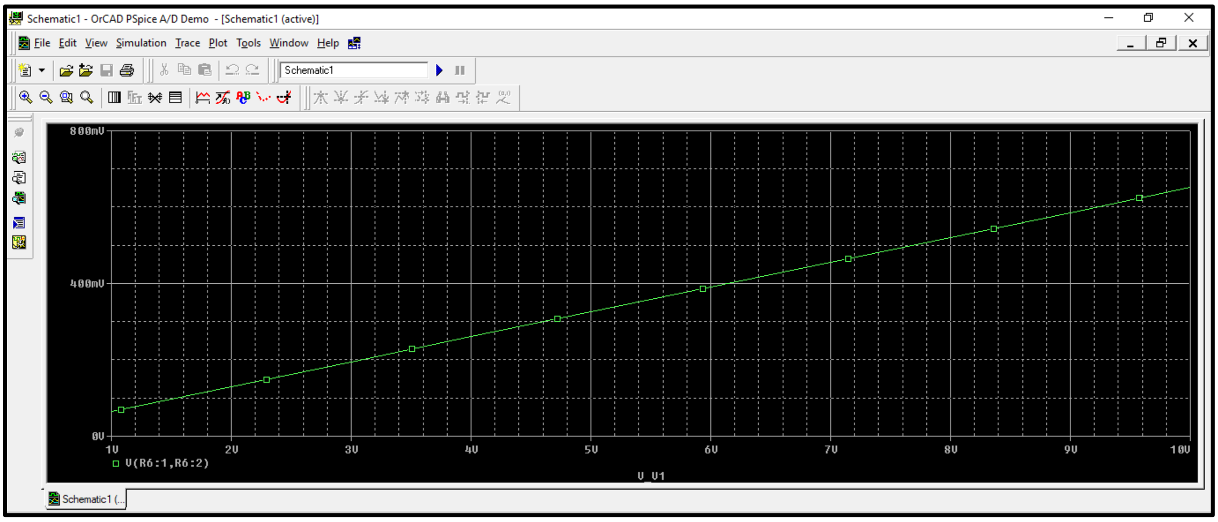


Figure.21

- You also can add curves, (traces) from simulation output window by clicking on traces and select add trace as shown in figure 22. You show current trace by clicking upon Current Maker (note that current marker must be placed on the terminals of a part to display current).

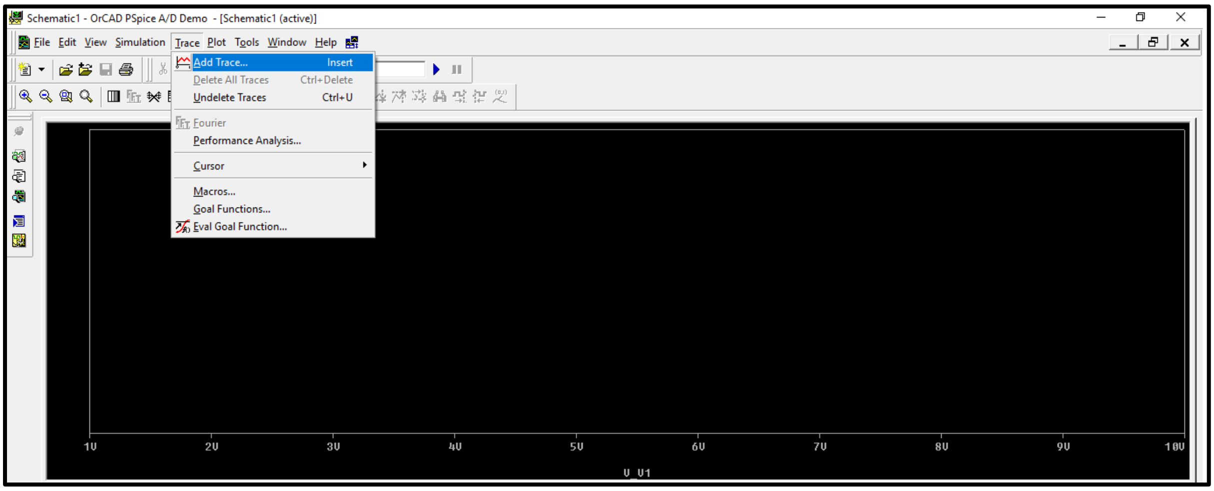


Figure.22

- Then the window shown in figure 23 will show up, in this window you will find a list of all variables in the circuit: (voltages and currents on all parts) to add a trace just click on it and it will show up in the field indicated in the box in figure.23.

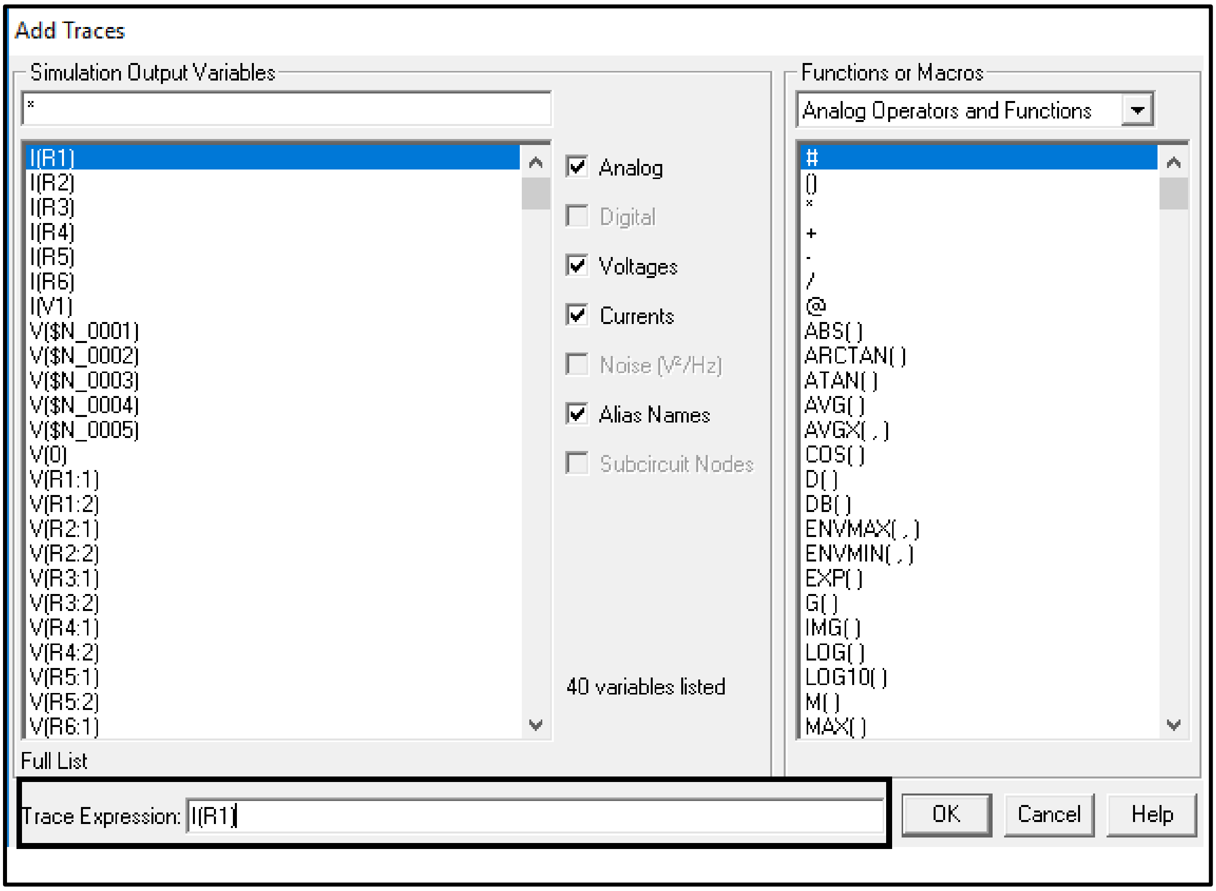


Figure.23

- When you finish selecting variables click on ok, and the trace you selected will be shown in simulation output window as shown in figure.24.

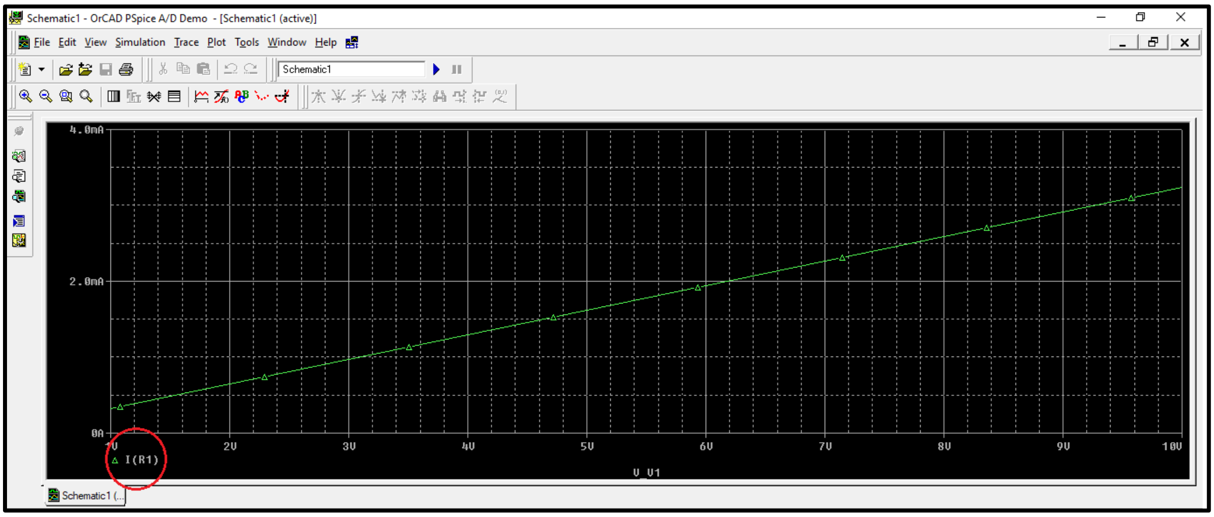


Figure.24

- In the "add trace window you can also add a trace that is a combination of more than one circuit variable e.g. to add a trace of the power dissipated in R6, note the equation inserted in figure 25.

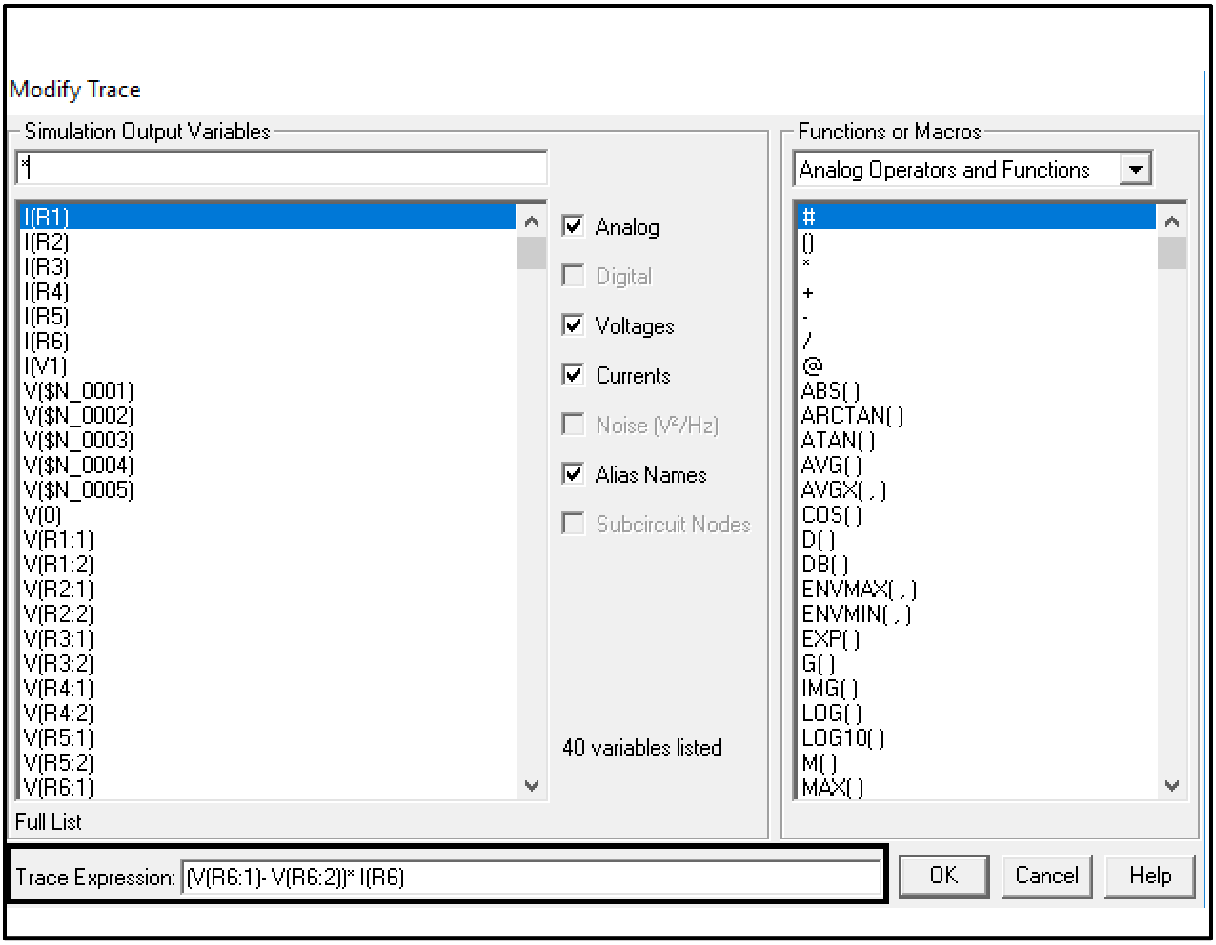


Figure.25

- The output is displayed in figure.26, note that the y-axis unit now is Watt.

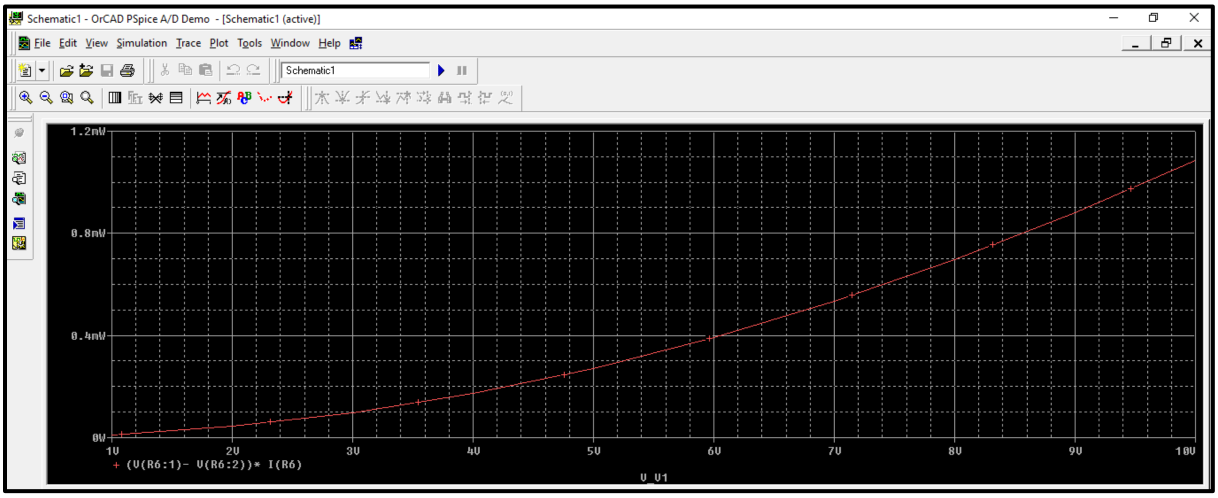


Figure.26

- **To perform a DC sweep for a resistor in the same circuit**, you need to add "param" part, go to "get new part" and search for "param", as shown in figure.27.

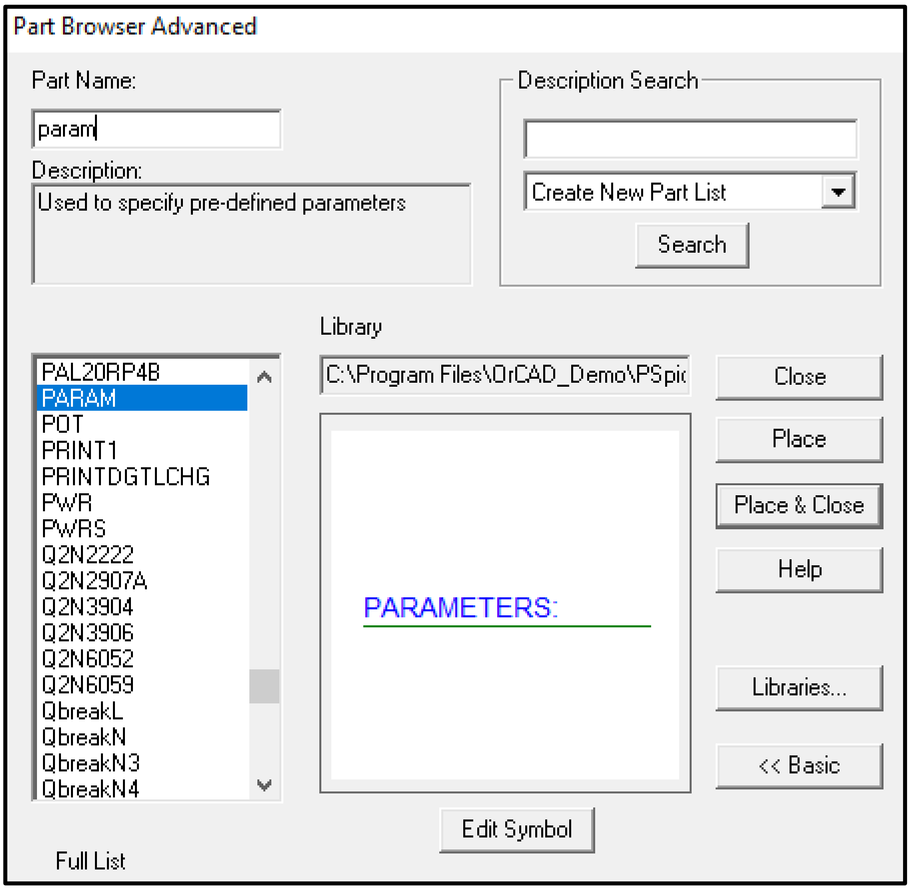


Figure.27

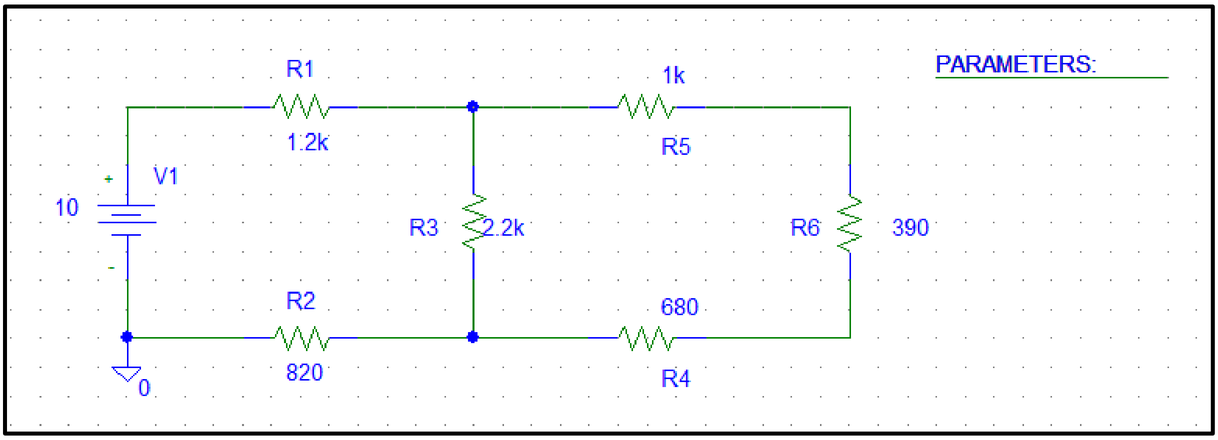


Figure.28

- Double click on “param”, and the window shown in figure.29 will show up, insert parameters as shown in figure.

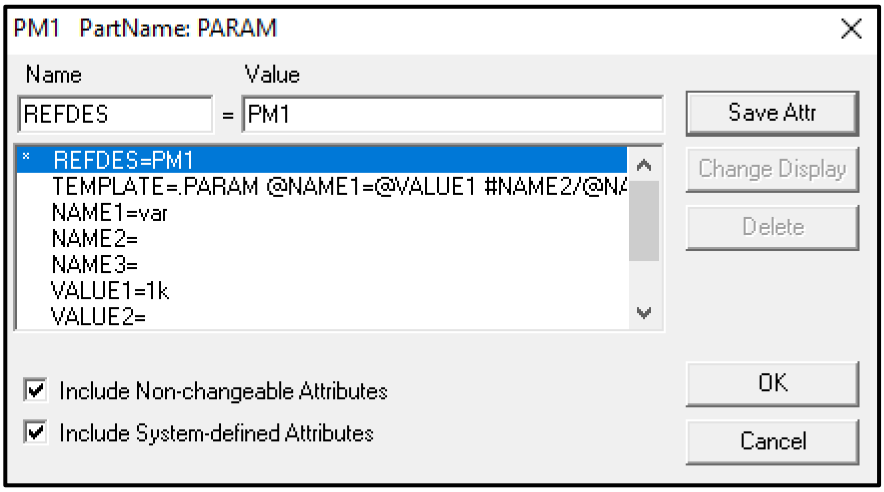


Figure.29

- Click on “setup analysis” and select DC sweep, fill in parameters as shown in figure 30.

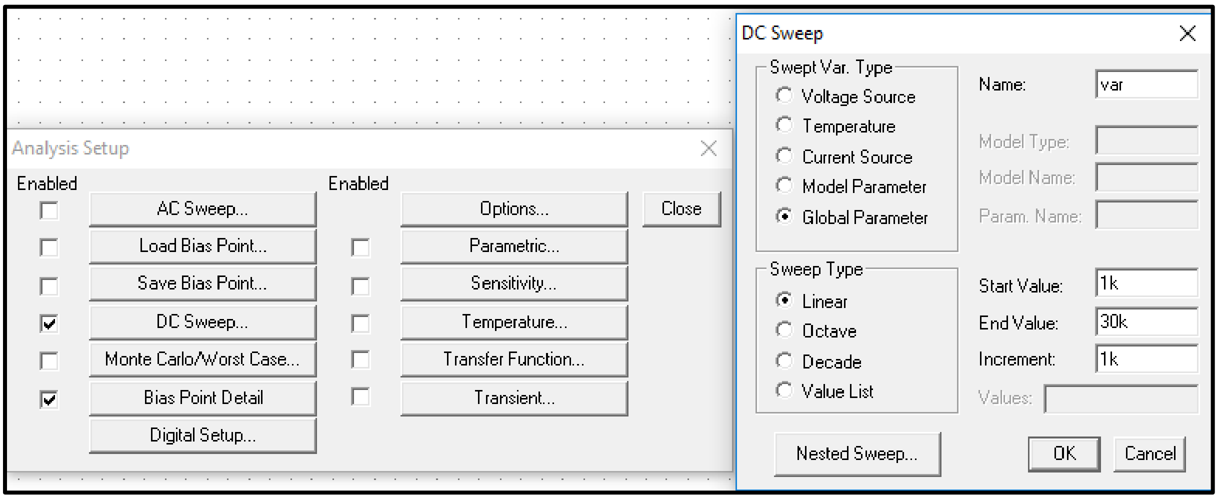


Figure.30

- Type the name of the global parameter inside curly brackets {}, as shown in figure 31.

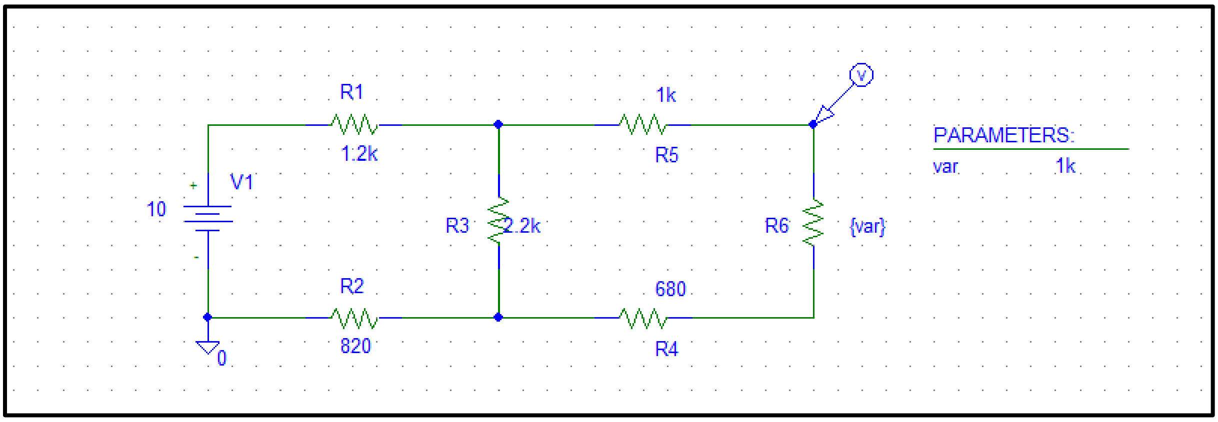


Figure.31

- Click simulate and check the output as shown in figure.32.

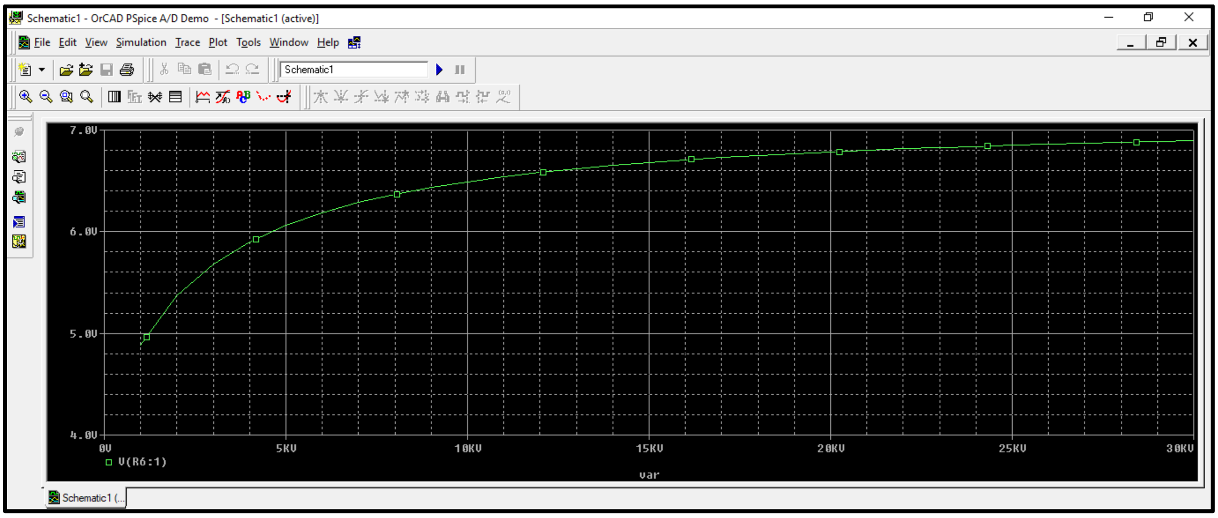


Figure.32

- **To change trace properties such as width and color, right click on the trace and select properties as shown in figure.33.**

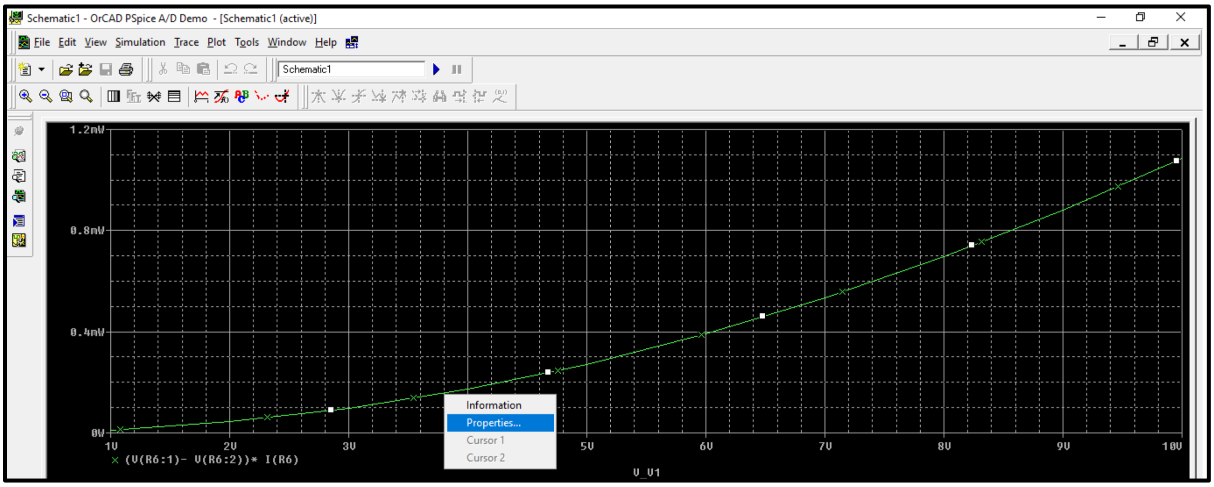


Figure.33

- The window shown in figure.34 will show up, you can change trace properties.

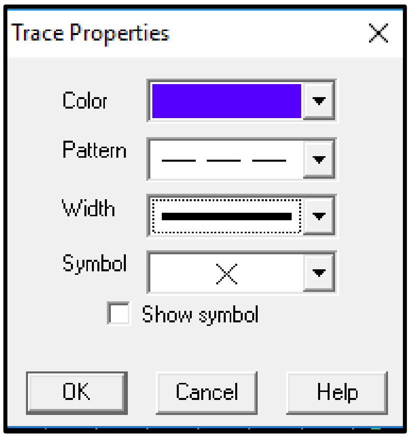


Figure.34

- After changing trace properties, the simulation output window will be shown in figure.35, **always make sure to increase the trace width in your prelab.**

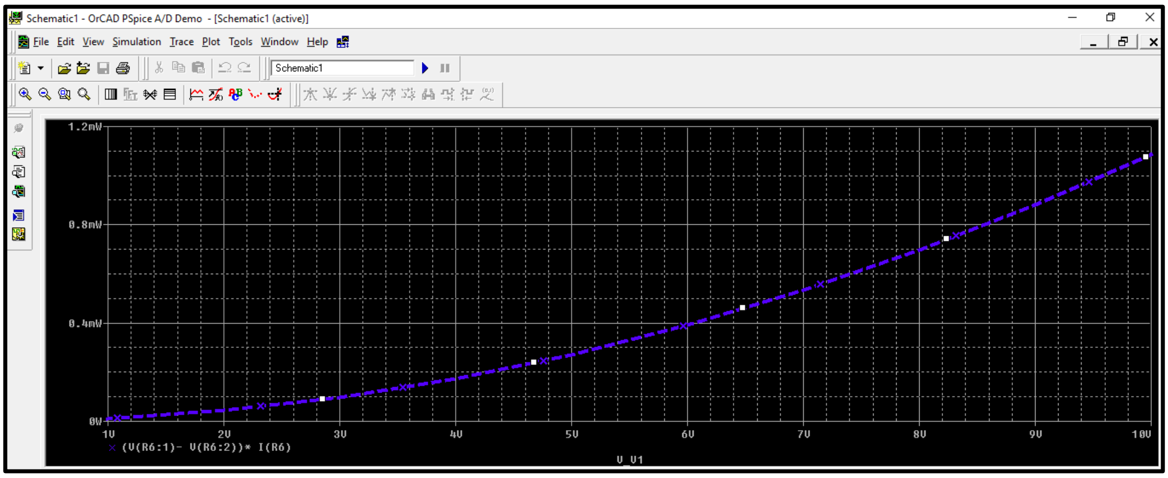


Figure.35

- **To copy the output curve to a word document**, go to the simulation output window and click on window then select "copy to clipboard", then select ok as shown in figure.36.

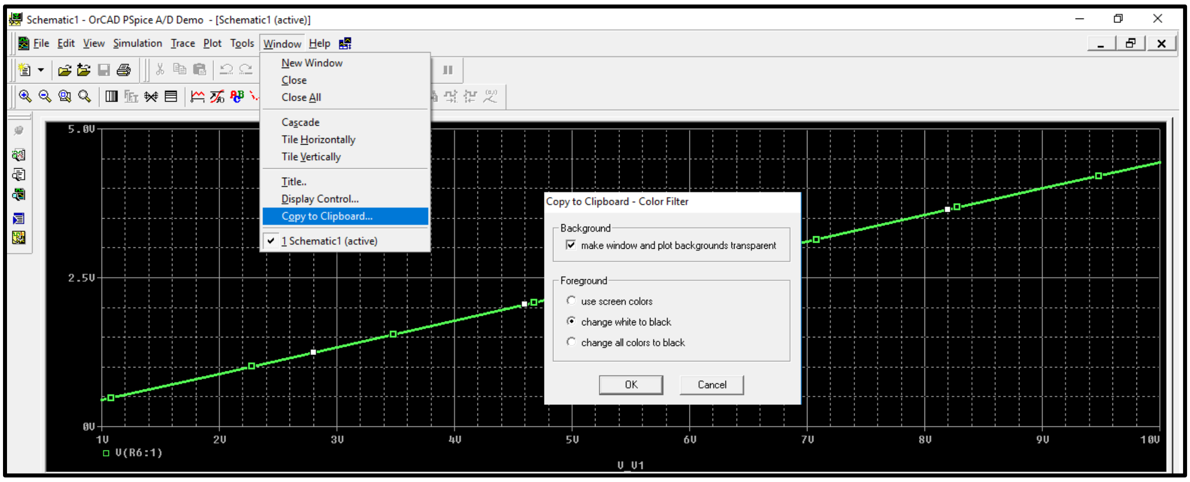
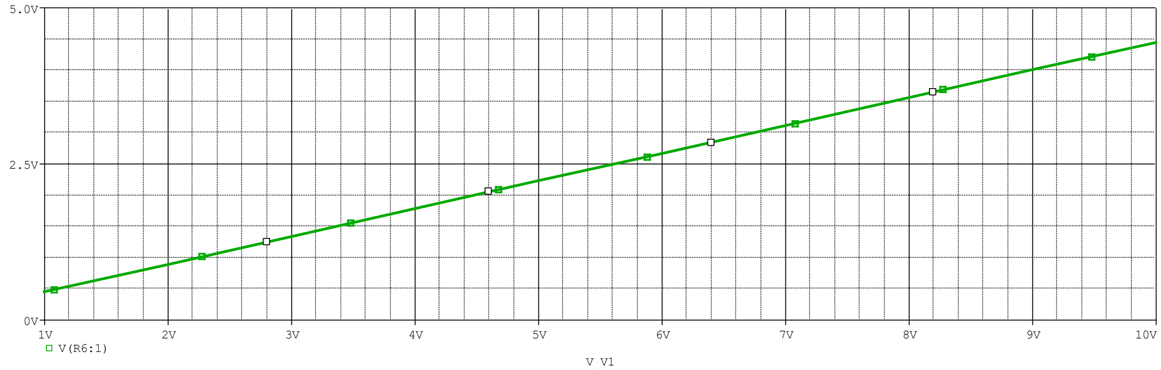


Figure.36

- Then click “paste” or “ctrl+v” inside word document and the output curve is copied a shown.



➢ Example on Transient analysis

- The circuit shown in figure.37 contains a square wave source which is a function of time, so we will use transient analysis.

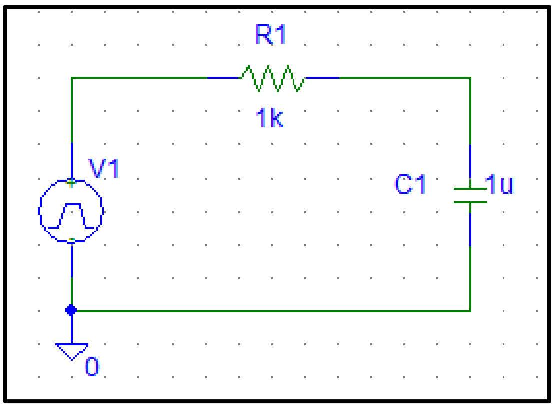


Figure.37

- The source name is “vpulse” and can be found on the components list as shown in figure.38.

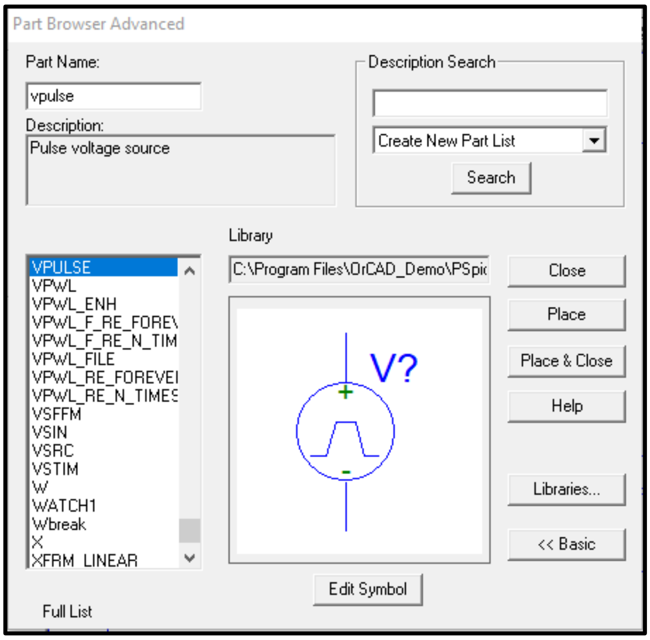


Figure.38

- Double click on the source to insert its parameters, the window in figure.39 will show up.

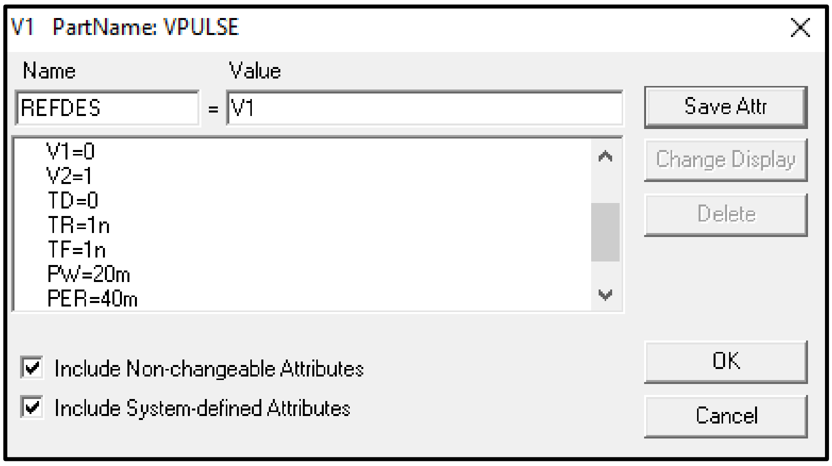


Figure.39

- Table.3 provides the details of this parameters.

Table.3

Parameter	Description	Value
V1	Lowest value	Usually 0
V2	Highest value	Depending on the question e.g. 1
TD	Delay Time	Usually 0
TR	Rise Time	1n
TF	Fall Time	1n
PW	Pulse Width	(1/ 2xFrequency) e.g. 20m
PER	Period	(1/ Frequency) e.g. 40m

- To perform a transient analysis for the circuit of figure.37, click on the “setup analysis” icon and select “Transient”, the window in figure.40 will show up.
- There are **two important parameters** that must be determined carefully in transient analysis so you can picture the output properly, these are: **Final time, and step ceiling.**
- **Final time** is the time till which you want to see the output signal, it is usually selected three times the period of the source.
- **Step ceiling** controls the accuracy **(smoothness)** of the output plots, it is usually obtained by dividing the final time on 10^4.

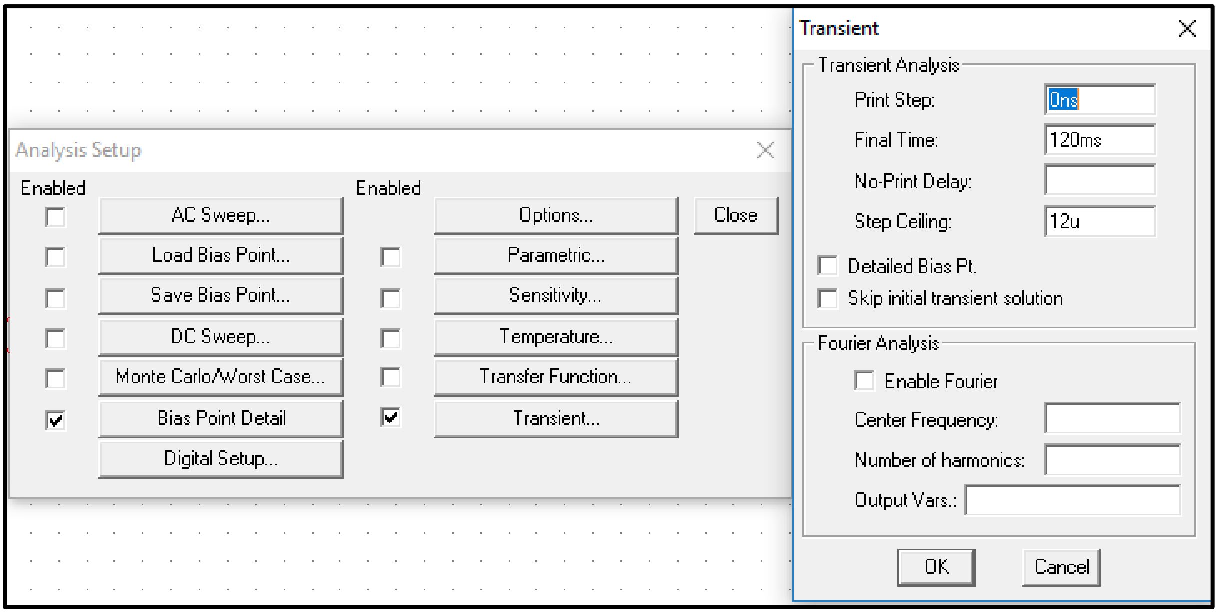


Figure.40

- The output is now displayed in figure.41.

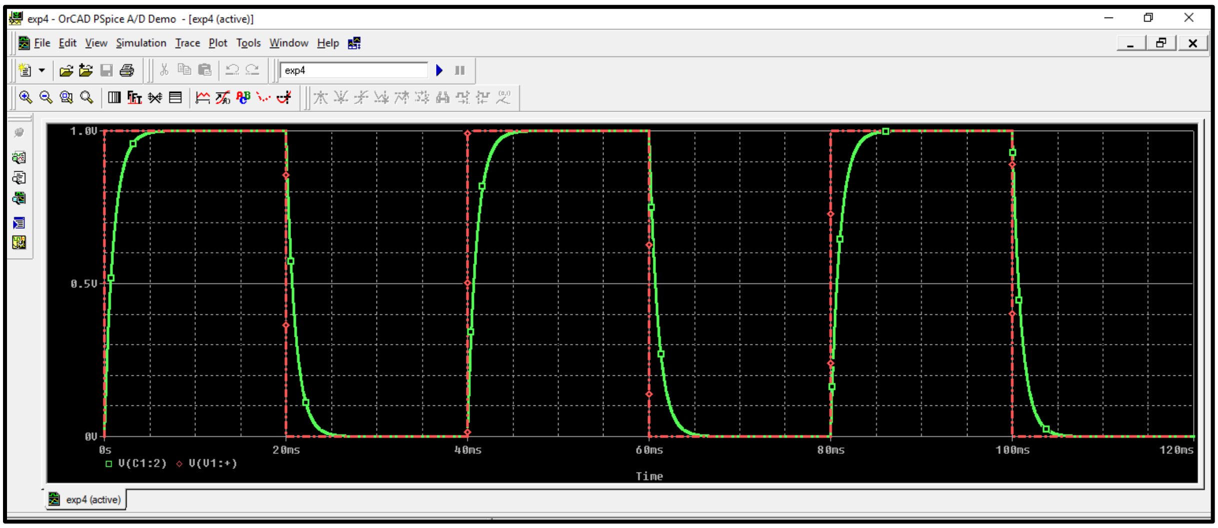


Figure.41

- Cursors are used often used to determine certain points on traces to use cursors click on “toggle cursors” icon shown in figure.42.
- There are two cursors in PSPICE (a1), and (a2), you can move between them by clicking the right and the left buttons of the mouse.
- Each cursor has a reading of x-axis and a reading of y-axis, look at figure.42.

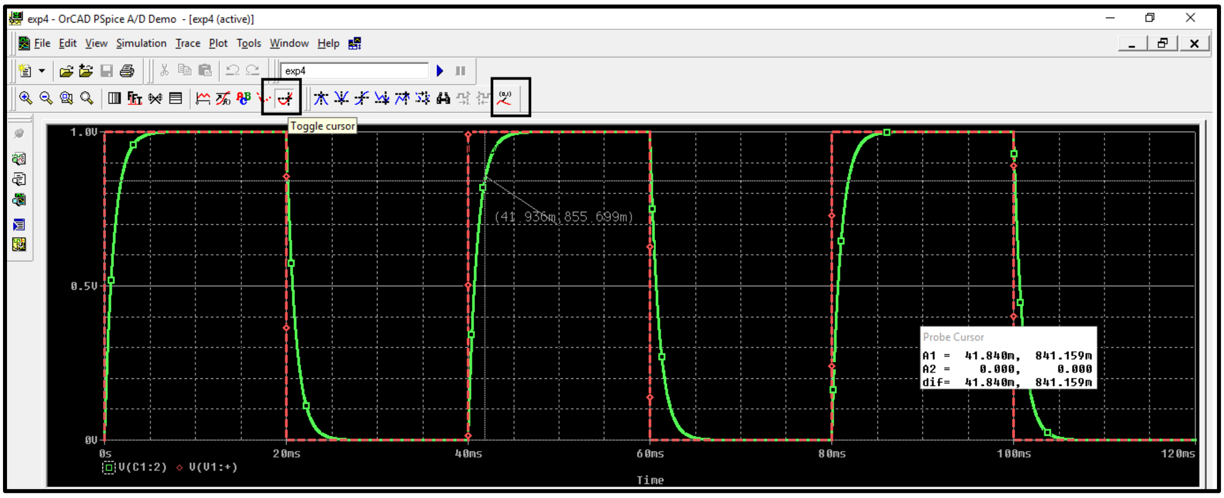


Figure.42

- Another example on transient analysis, sinusoidal voltage source (vsin).

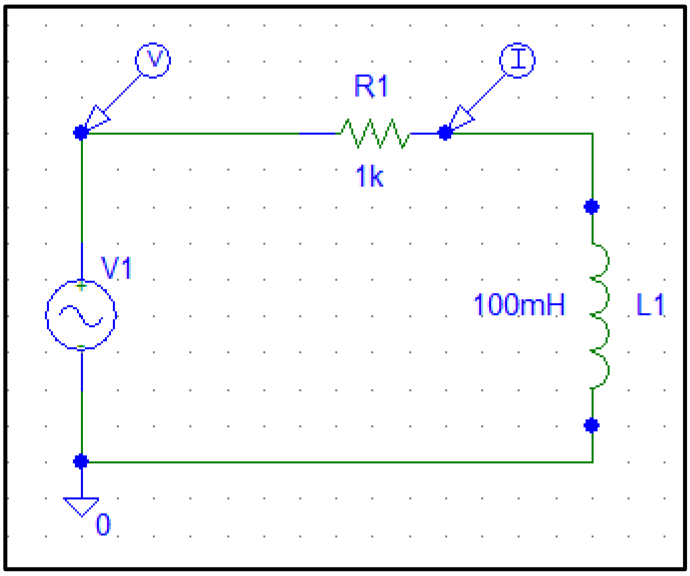


Figure.43

- Source settings.

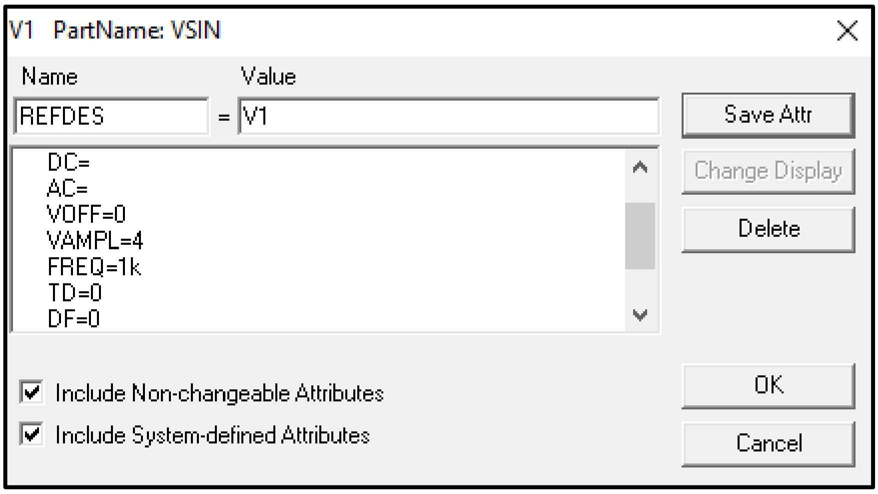


Figure.44

- In the simulation output window, we are viewing the current and voltage, but the voltage is in the range of (-4 to 4) and the current is in the range of (-4m to 4m) so we need **to add a separate y-axis to the output window.**
- Click on "plot", then select "add y-axis", note that you have different y-axes as shown in figure.45, you can move between axes by clicking on the axis you want to add a trace to.

Figure.45

- Voltage and current traces are shown in figure.46, each trace is on a different y-axis range.

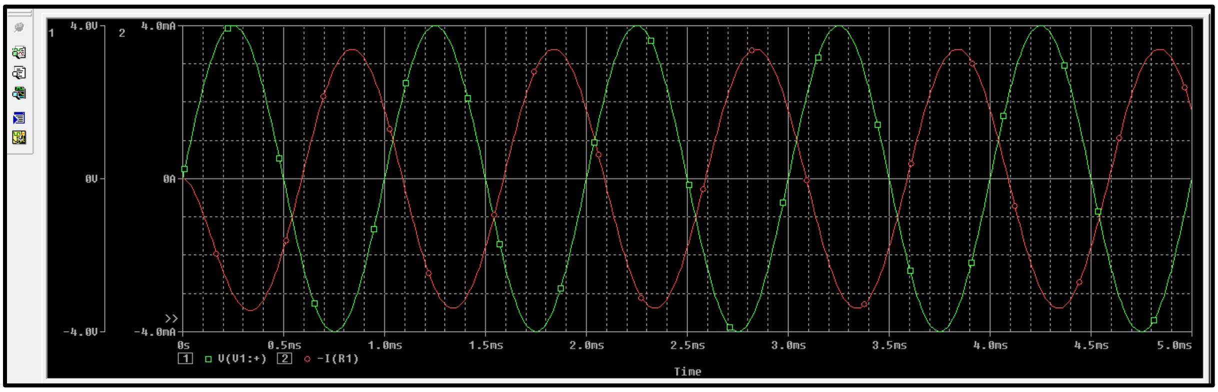


Figure.46

➢ Example on AC sweep analysis

- The circuit shown in figure.47 contains an (vac) source which is used for AC sweep analysis.
- Double click on the ac source to insert setting: only insert voltage magnitude to 1.

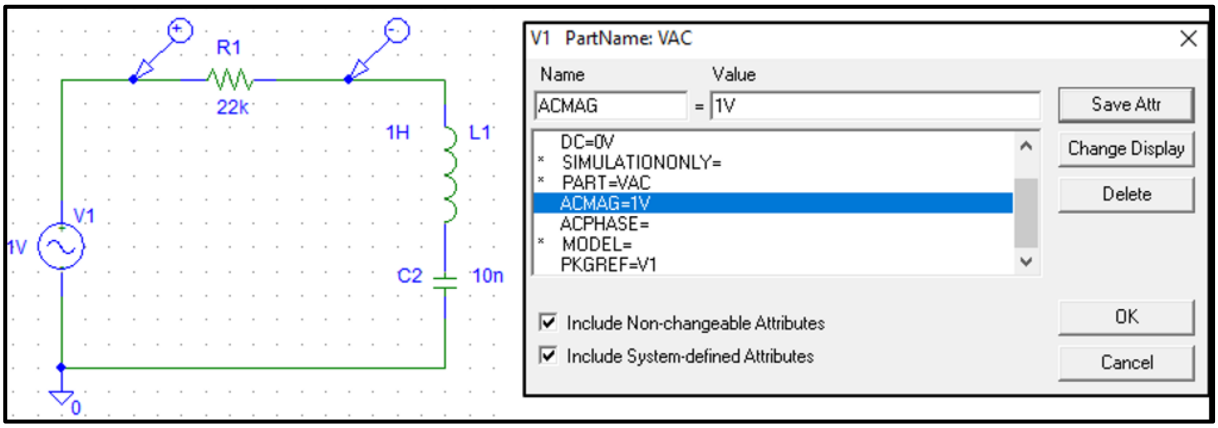


Figure.47

- Setup analysis setting: usually select "AC sweep type" to decade.

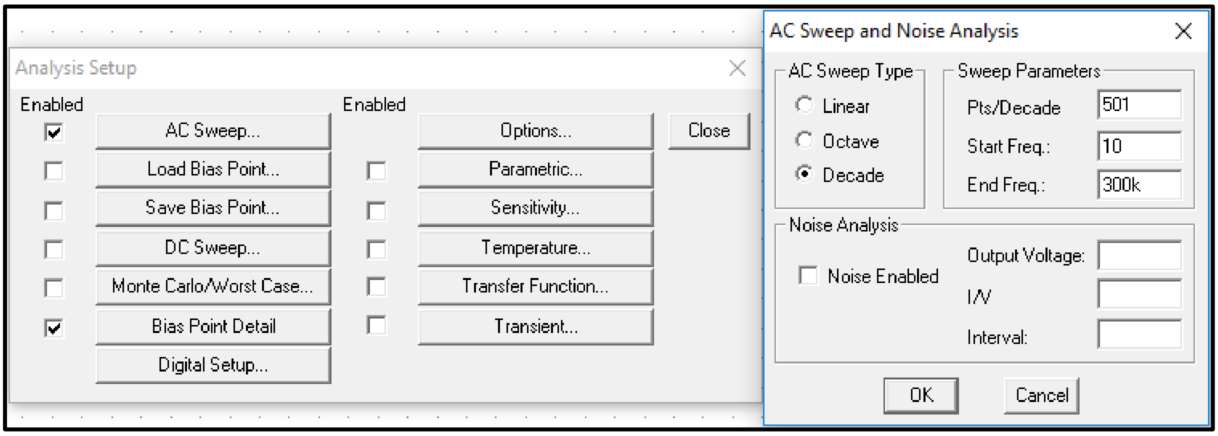


Figure.48

- Output window:

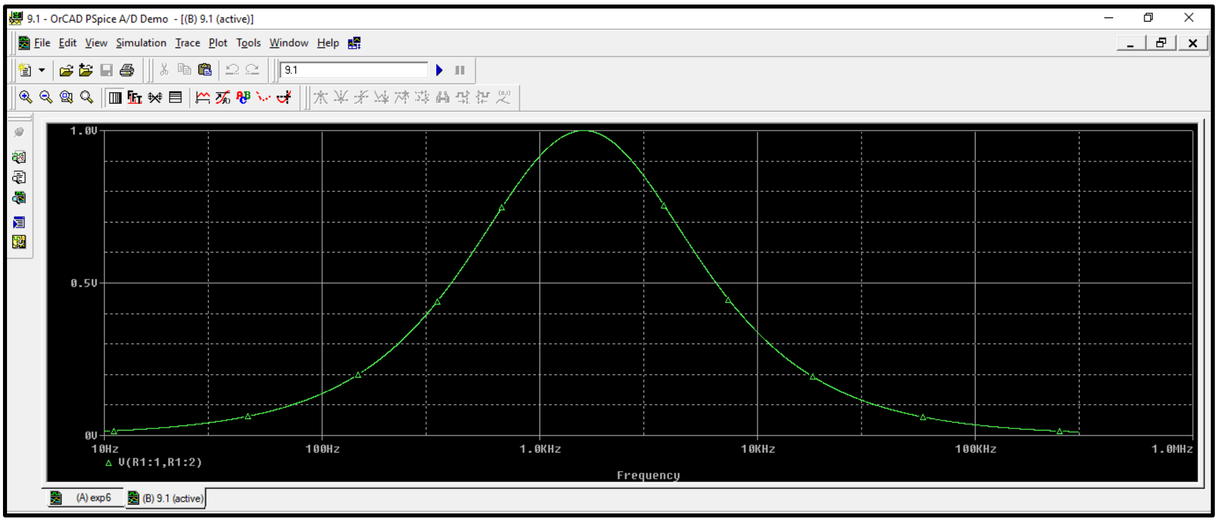


Figure.49