BIRZEIT UNIVERSITY

Faculty of Engineering and Technology

Electrical and Computer Engineering Department

Introduction to PSPICE 9.1

2023

➢ 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 to the end of the page 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.

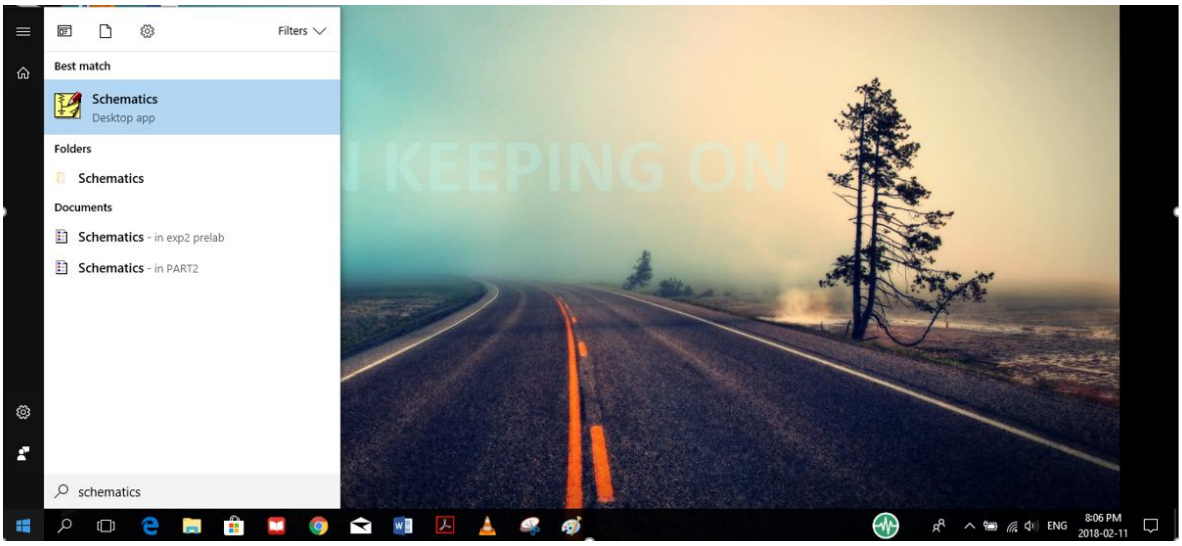

Figure.1

- **Note: Through this document you may need to zoom in to see figure details if not clear.**

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

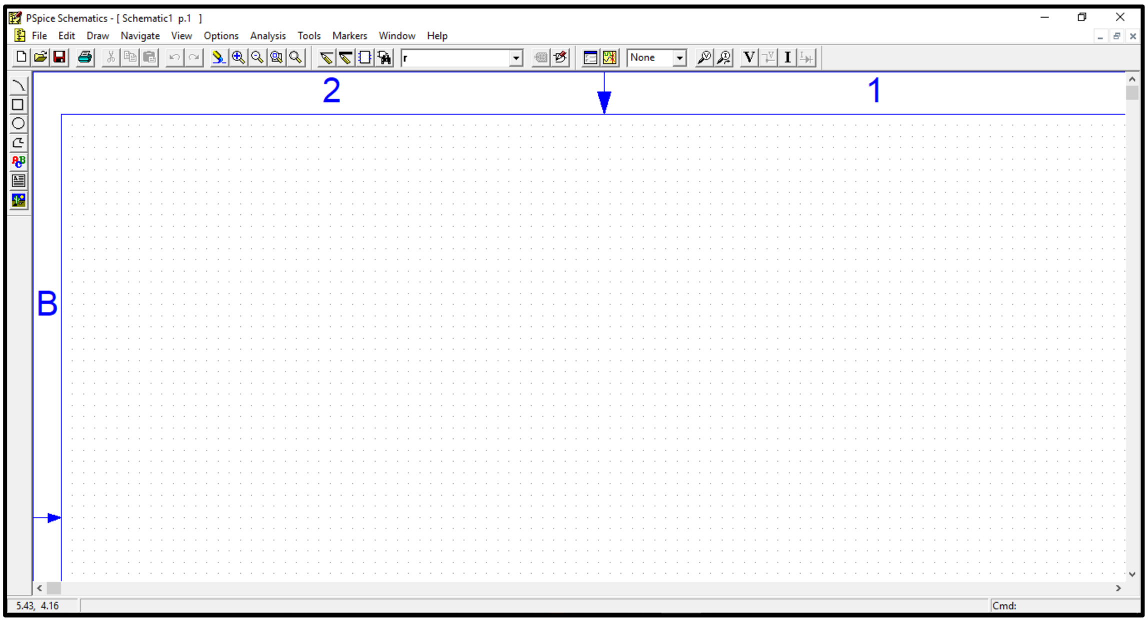

Figure.2

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

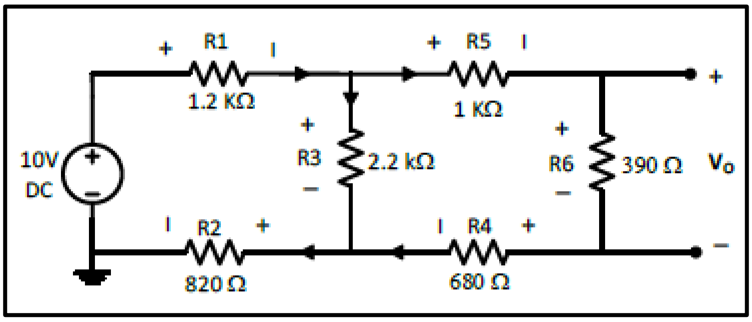

Figure.3

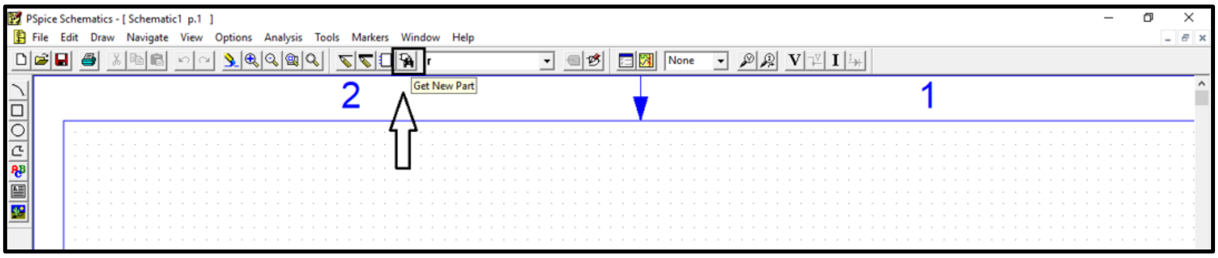

Figure.4

- 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.

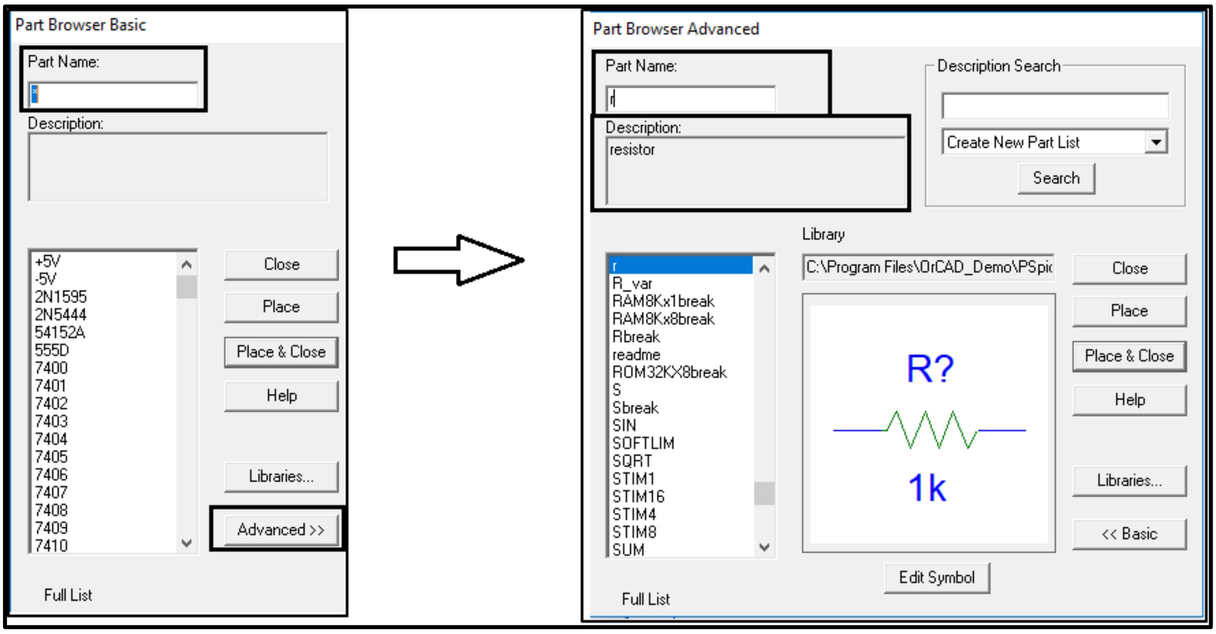

Figure.5

- 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.

Table.1

Circuit element	Part name in PSPICE
Resistor	R
Capacitor	C
Inductor	L
DC Voltage source	vdc
DC Current source	Idc
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
Switch closes at "t_{close}"	Sw_tClose

- Note that when you place parts, each circuit part in PSPICE has a name and a value as shown in figure.6.

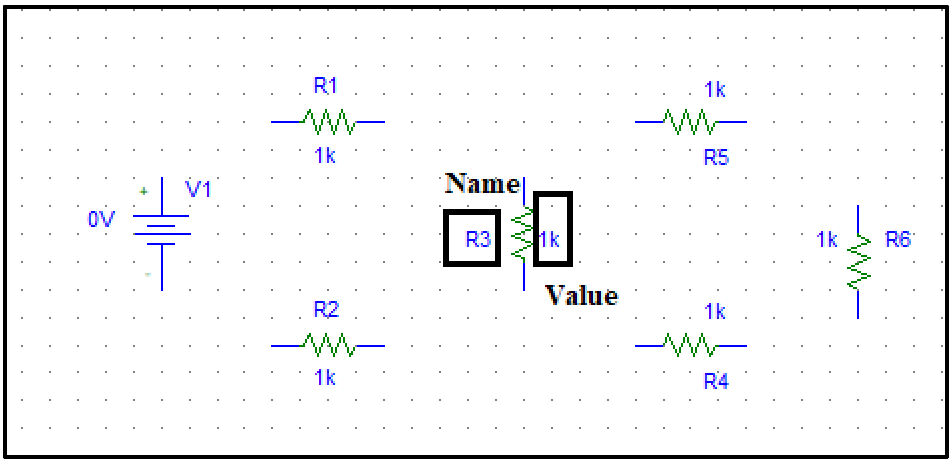

Figure.6

- 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

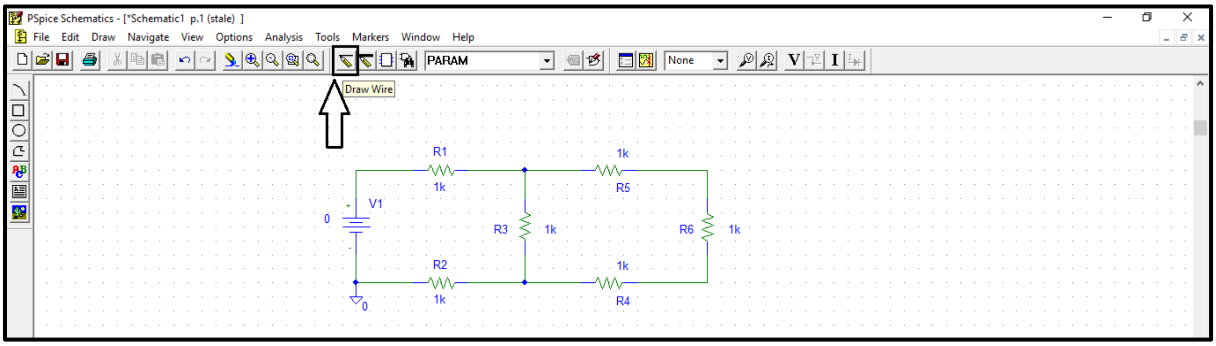

Figure.7

- 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.

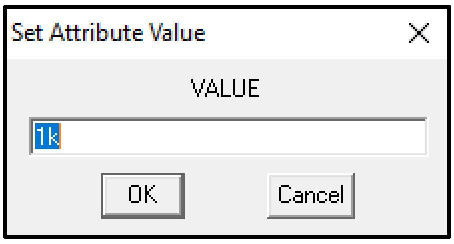

Figure.8

- 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 the analog ground (GND_analog), you can get it from "get new part option".
- Your schematic is finished now and ready for saving, click on "save" then on "simulate" button as shown in figure.9.

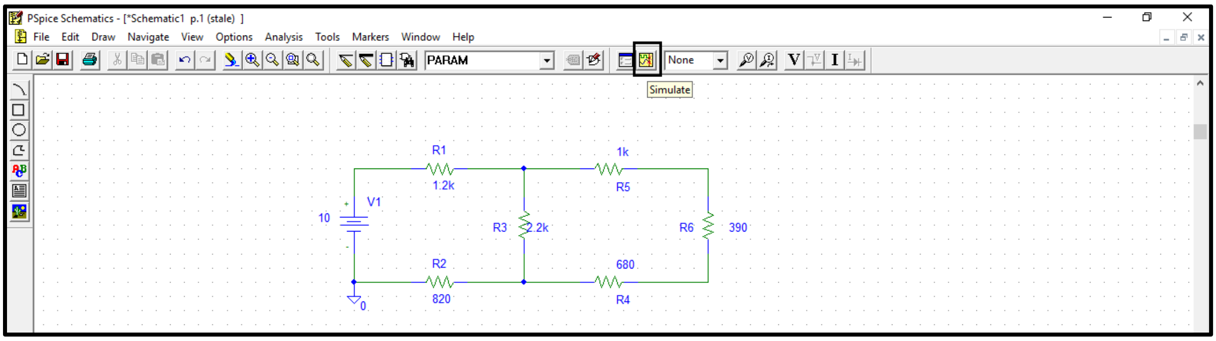

Figure.9

- 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.

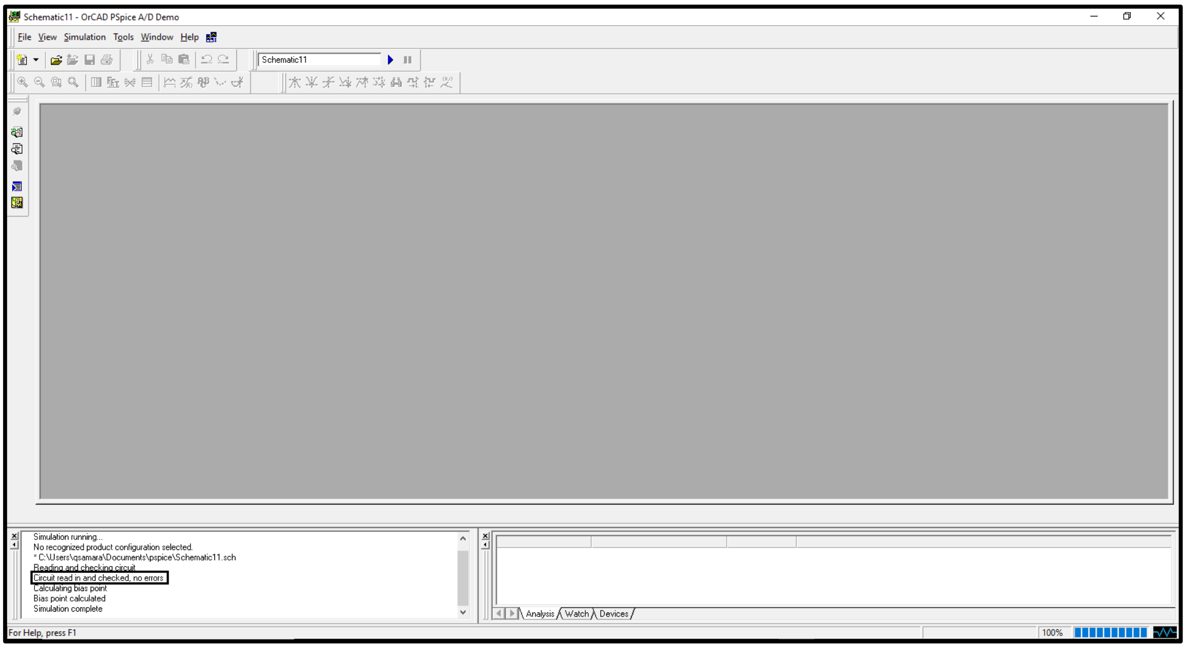

Figure.10

- 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.

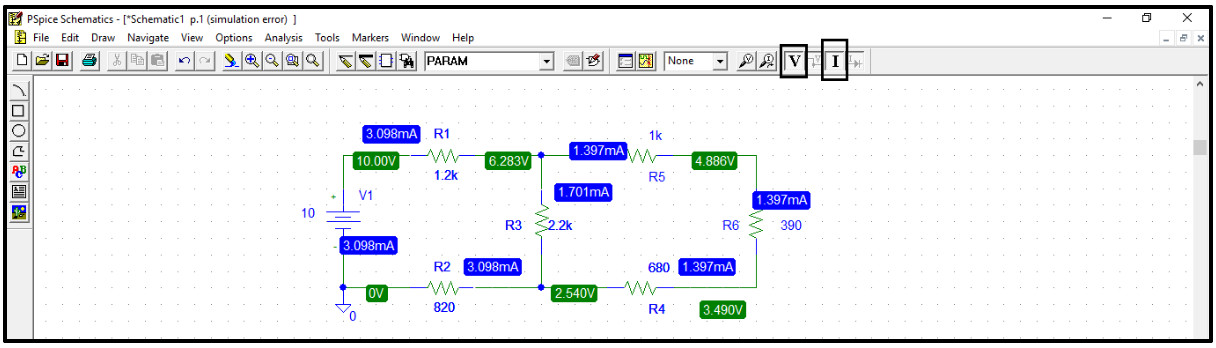

Figure.11

➢ Types of Analysis Performed by PSPICE

PSPICE is capable of performing four main types of analysis: Bias Point, DC Sweep, Time Domain (transient), and AC Sweep/Noise. We are concerned with the first 3 types here.

- **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.**

- **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.

➢ Example on Transient analysis

- The circuit shown in figure.12 contains a capacitor and a switch whose behavior depends on time, so we will use transient analysis.

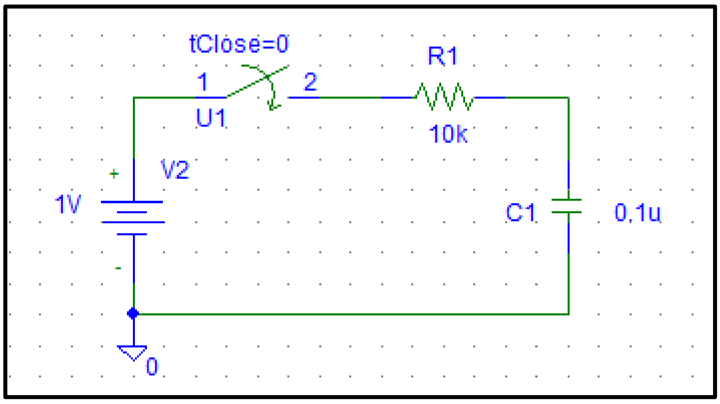

Figure.12

- The switch used here can be found on the components list as shown in figure.13.

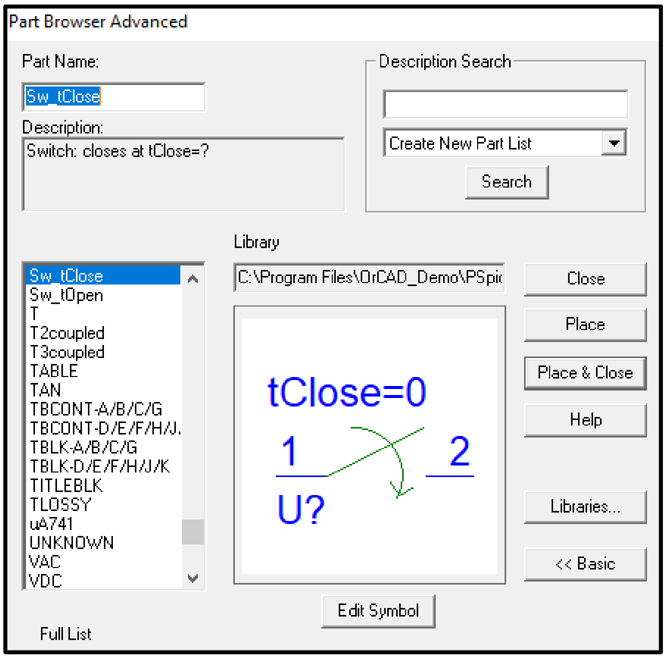

Figure.13

- To perform a transient analysis for the circuit of figure.12, click on the "setup analysis" icon and select "Transient", the window in figure.14 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 by multiplying the time constant (τ) of the circuit by 7 to 10 times.
- **Step ceiling** controls the accuracy **(smoothness)** of the output plots, it is usually obtained by dividing the final time on 10^4.
- Make sure to **tick the box** beside **"skip initial transient solution"** as shown in figure 14.

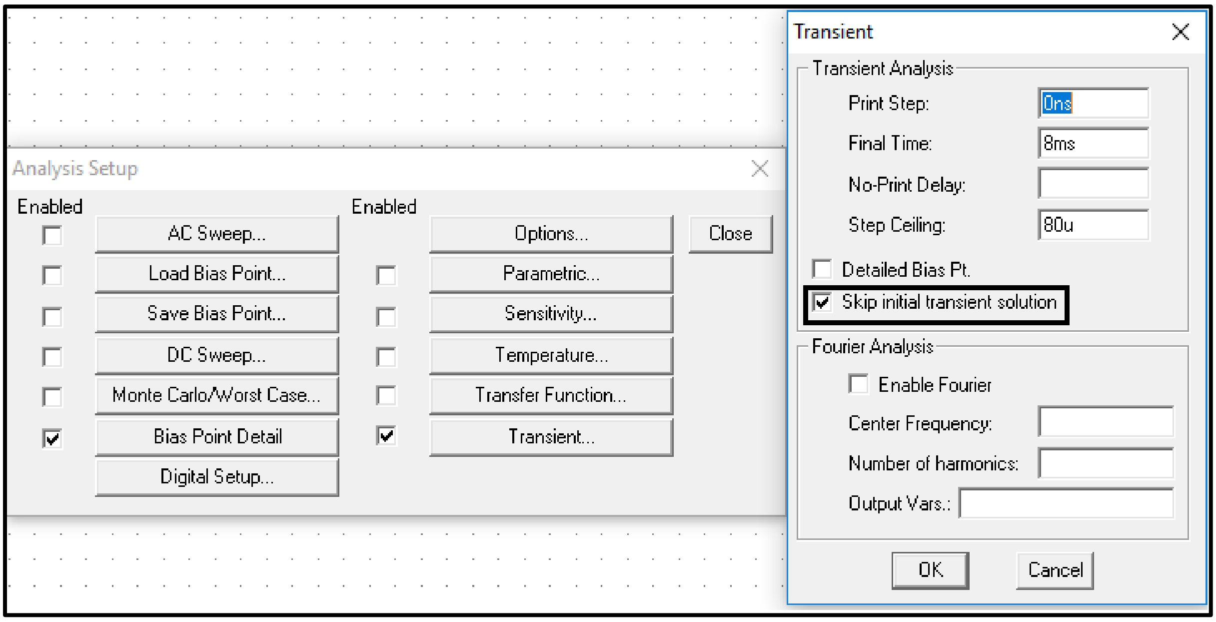

Figure.14

- The output is now displayed in figure.15.

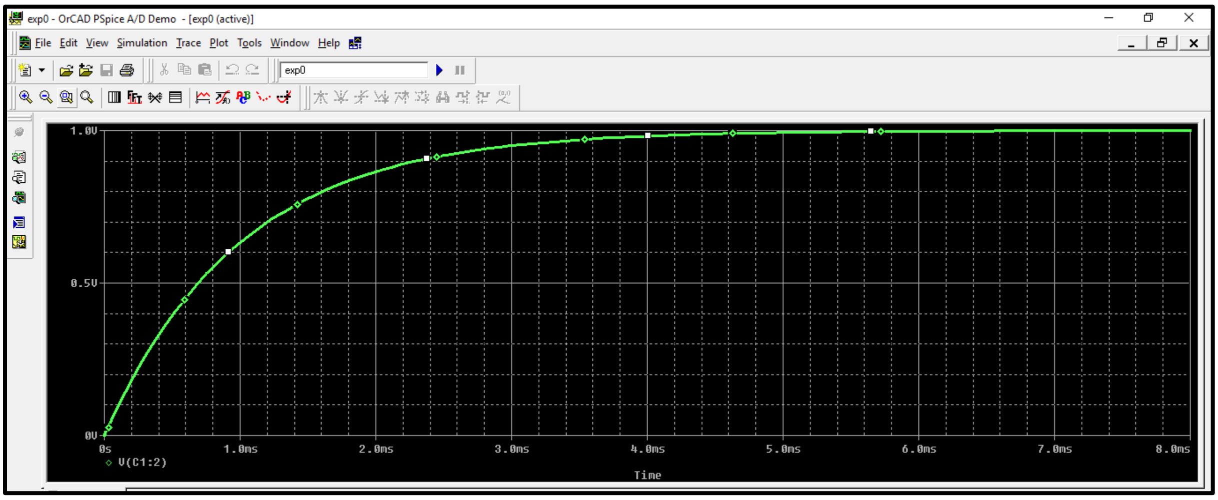

Figure.15

- **To change any trace properties such as width and color, right click on the trace and select properties as shown in figure.16 (the trace in figure 14 is different and not the same trace of figure 15).**

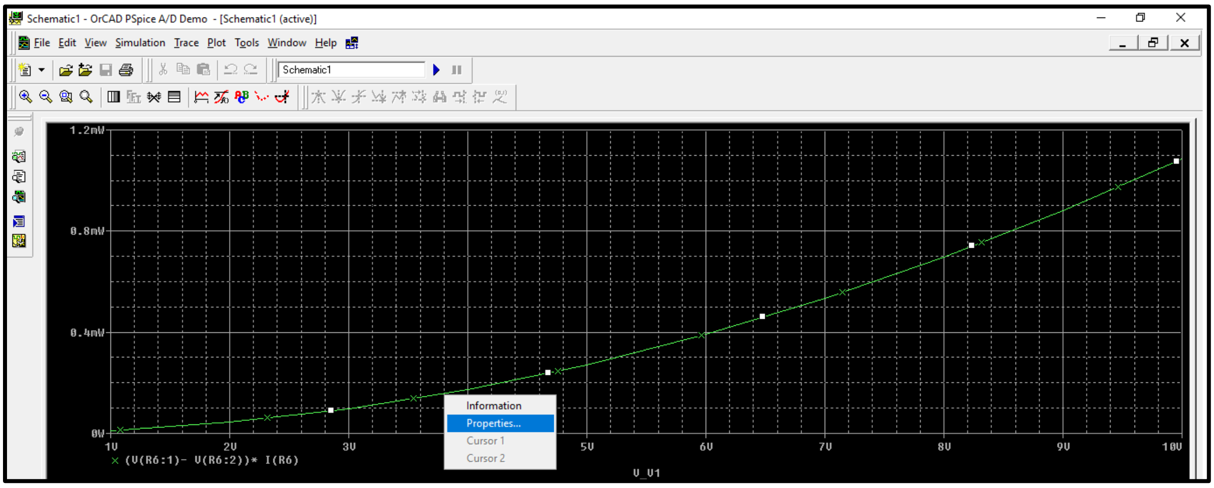

Figure.16

- The window shown in figure.17 will show up, you can change trace properties.

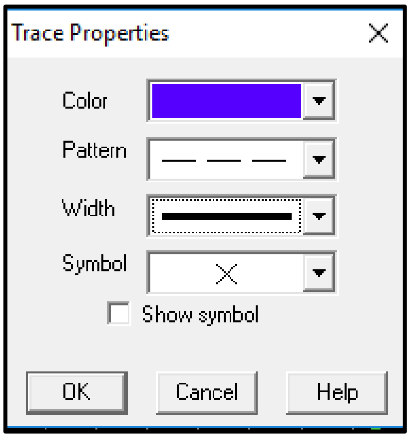

Figure.17

- After changing trace properties, the simulation output window will be as shown in figure.18, **always make sure to increase the trace width in your report.**

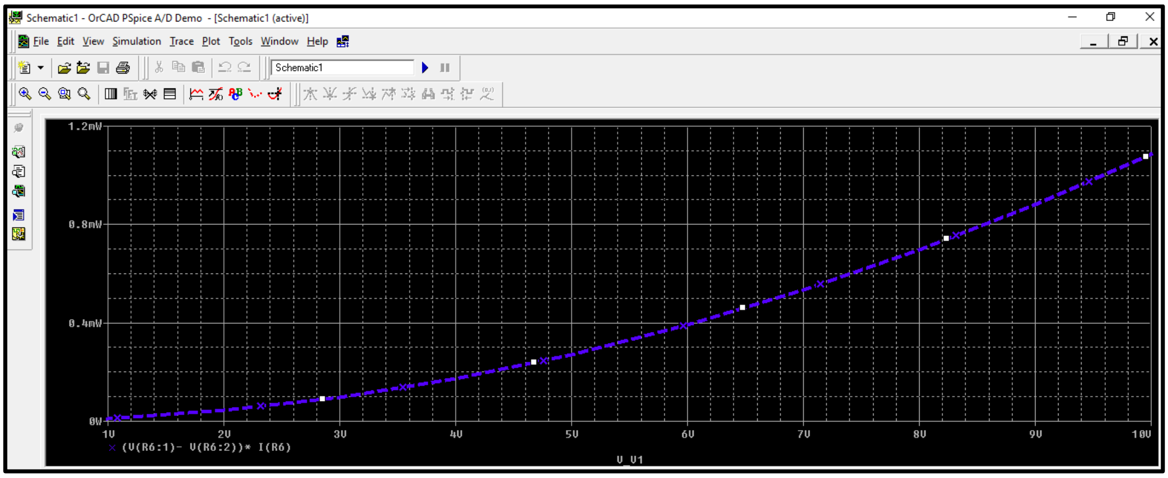

Figure.18

- **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.19.

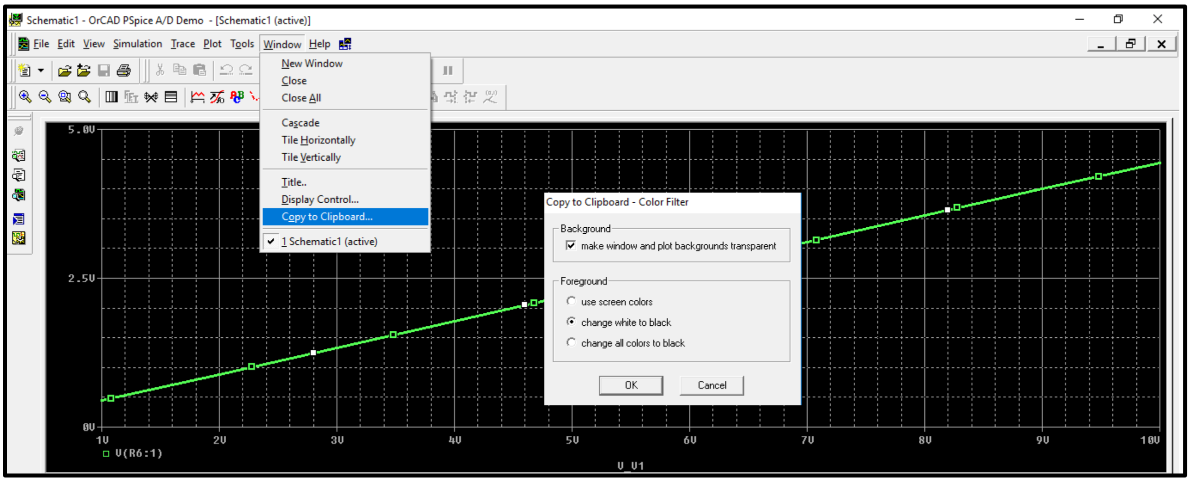

Figure.19

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