A TUTORIAL ON LTSPICE:

DC SWEEP (.DC), TRANSIENT ANALYSIS (.TRAN), AND PARAMETRIC SWEEP (.STEP PARAM)

Description: LTspice is a Simulation Program with Integrated Circuit Emphasis (SPICE)−based high performance analogue electronic circuit simulator computer software, produced by semiconductor manufacturer Analog Devices (originally by Linear Technology). It is the most widely distributed and used SPICE software in the industry. Though it is freeware, LTspice is not artificially restricted to limit its capabilities (no feature limits, no node limits, no component limits, no subcircuit limits). It ships with a library of SPICE models from Analog Devices, Linear Technology, Maxim Integrated, and 3rd party sources too. LTspice provides schematic capture to enter an electronic schematic for an electronic circuit, an enhanced SPICE type analogue electronic circuit simulator, and a waveform viewer to show the results of the simulation. Circuit simulation analysis based on transient, noise, AC, DC, DC transfer function, DC operating point can be performed and plotted as well as Fourier analysis. Heat dissipation of components can be calculated, and efficiency reports can also be generated. It has enhancements and specialized models to speed the simulation of switched-mode power supplies (SMPS) in DC-to-DC converters. LTspice does not generate printed circuit board (PCB) layouts, but netlists can be exported to PCB layout software. While LTspice does support simple logic gate simulation, it is not designed specifically for simulating logic circuits. It is used by many users in fields including radio frequency electronics, power electronics, audio electronics, digital electronics, and other disciplines.

Note**: This is not a software developer's official instruction. The tutorial is intended for the students enrolled in the laboratory course CSE251 conducted by BRAC University's Department of Computer Science and Engineering (CSE). The author of this tutorial is neither an expert nor a representative of the developer company. The methods/procedures demonstrated in this tutorial may not be the most efficient; rather, it is intended mostly for newcomers to this tool. Since its author lacks expertise, it can have inaccuracies. If a discrepancy is found, make the necessary corrections on your own.**

☞ The tutorial was created using the Windows operating system. However, the author made an effort to keep the instructions as generic as possible so that Mac users might also benefit.

☞ See also: [Tutorial for CSE250](#)

1. Installing LTspice:

    Visit the following site and download the latest version of LTspice compatible to your operating system. Official site link: LTspice Simulator | Analog Devices

    Windows:

    ➢ Open the downloaded **.exe** file.

    ➢ ***Click Accept →*** *[Optional] Modify the installation directory if necessary* ***→ Install Now***

    ➢ Upon successful installation, a window stating **'LTspice XVII has been successfully installed'** will show up. ***Click OK.*** LTspice will start automatically after a while.

    Mac:

    ➢ Open the downloaded **.pkg** file.

    ➢ ***Click Continue → Continue → Agree → Continue →*** *[Optional] Modify the installation directory if necessary* ***→ Install → Insert Admin Username & Password → Install Software → Close***

2. Opening LTSPICE:

    Windows: Select ***Start*** ⊞ ***→ All apps → Scroll and find LTspice XVII → click to open***. A window like this will open.

    Mac: ***Command + Space → Search for LTspice → Find LTspice → Click to open → Start a new, blank Schematic.***
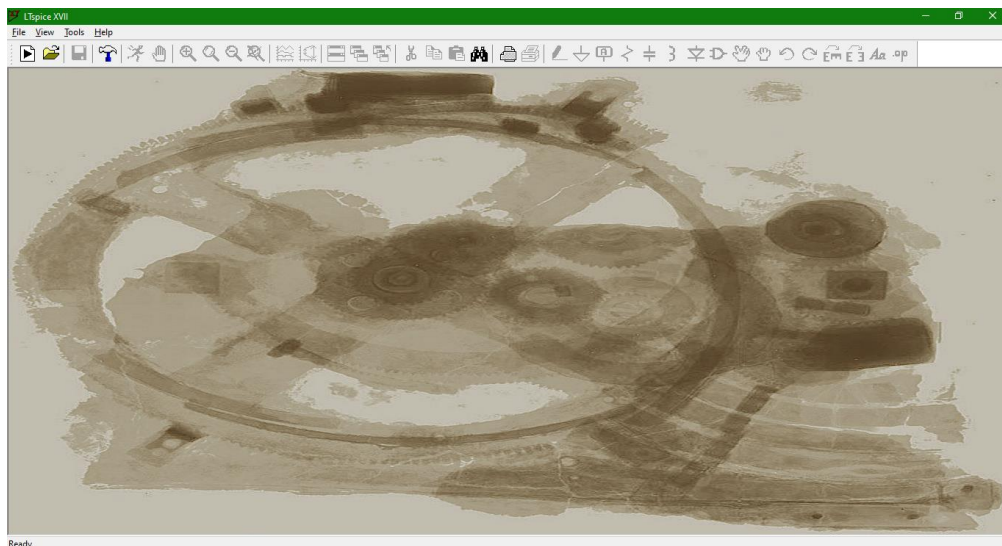
3. Get acquainted with the user interface.
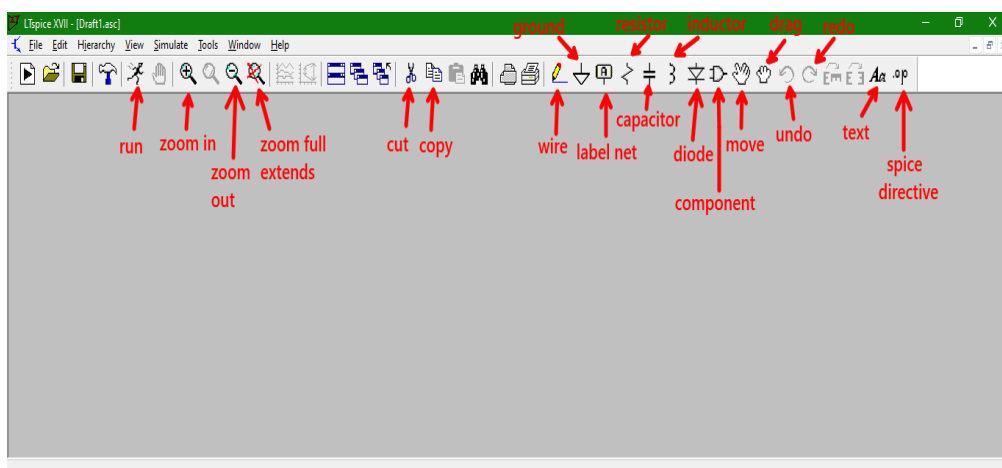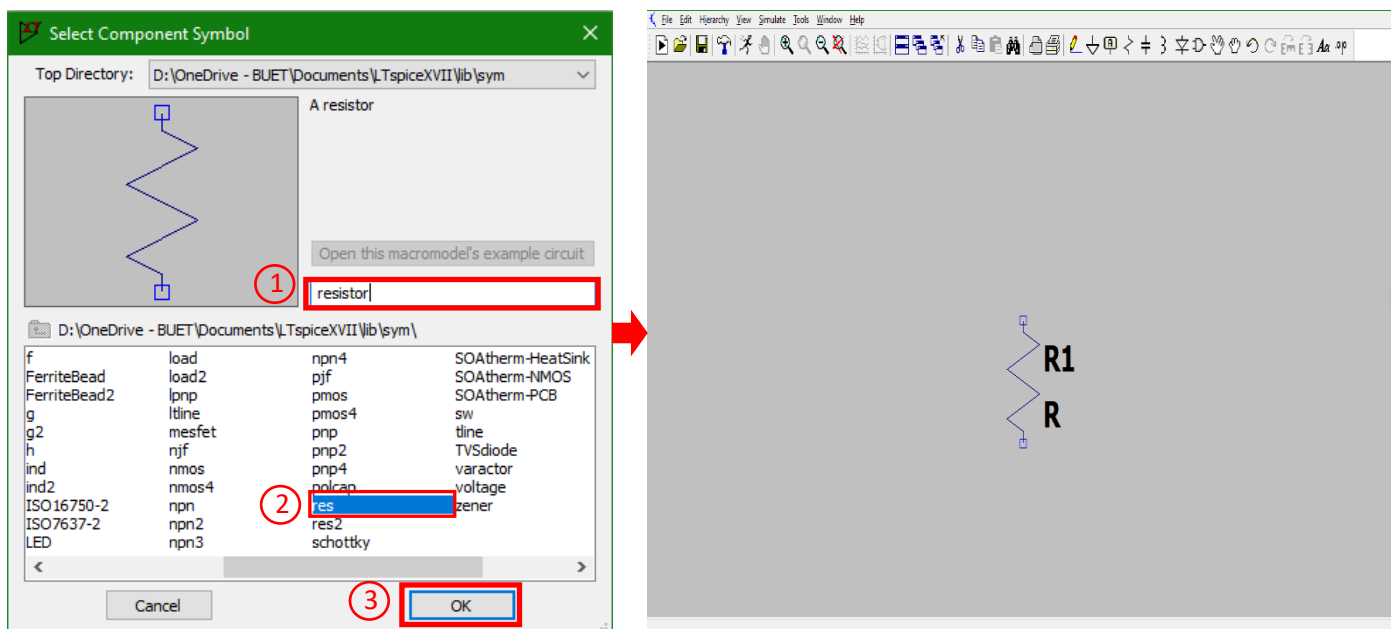

4. Go to **File → New Schematic.** Save the file by **File → Save As → diodeIV.acs.**
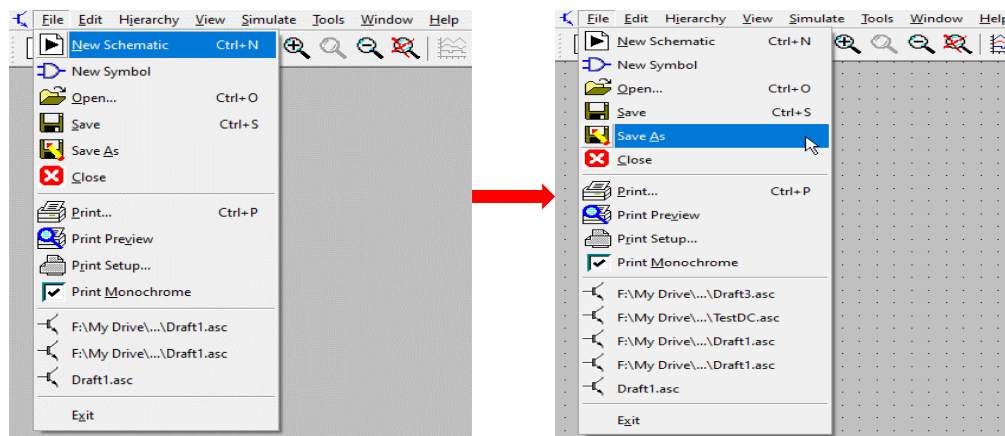
5. <u>Inserting a component</u>: **Right-click on blank space → Draft → Component** | Or, **Press F2** | Or, **click this icon on the toolbar** . A **Select Component Symbol** window will open like this.
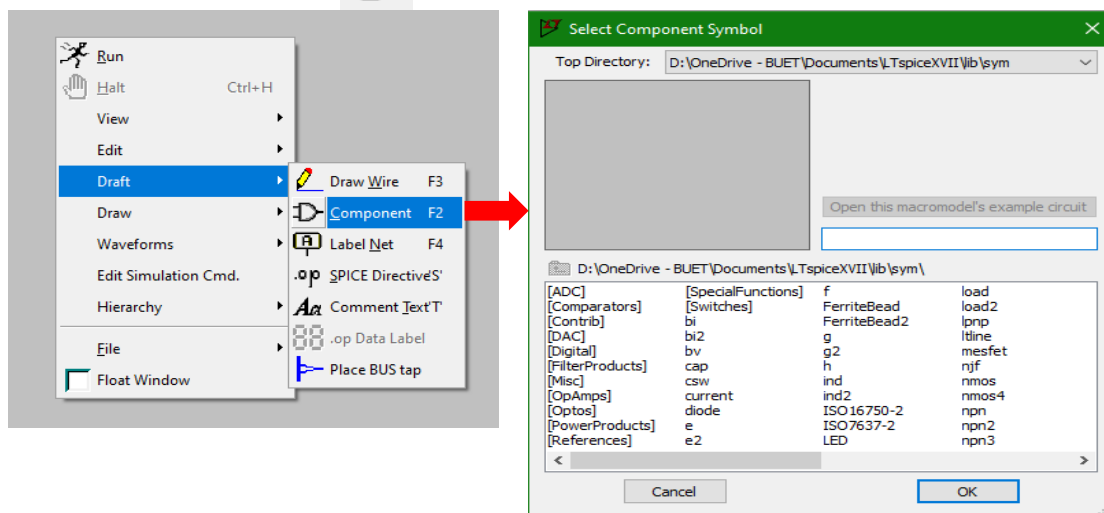

I. Type the keyword for a component to be inserted. For example, **type 'resistor' → select the component → click OK** | Or, **click this icon on the toolbar** The component (resistor in this case) will be selected and will follow the cursor's movement. Press **CTRL + R** to rotate the component by 90^0 if necessary. **Left click** on any suitable place in the grey interface to place it. **Press ESC** to deselect the component.

Component Name	LTspice Keyword
Independent Voltage Source	voltage
Independent Voltage Source	current
Resistor	resistor
Capacitor	cap
Inductor	ind
Diode	diode
Zener diode	zener
N-channel MOSFET	nmos
P-channel MOSFET	pmos
NPN BJT	npn
PNP BJT	pnp
VCVS	e or e2
CCCS	F
VCCS	g or g2
CCVS	h

6. Moving a component: *Right-click on blank space → Edit → Move* | Or, *Press F7* | Or, *click this icon on the toolbar* . The cursor will be change into . Then *click on the component you want to move* (to move multiple components together, select them by left clicking and dragging). The component will be attached to the cursor. *Left click* to place on a different position. The component can be rotated by pressing *CTRL + R* while moving as well.

7. Duplicating a component: *Right-click on blank space → Edit → Duplicate* | Or, *Press F6* | Or, *click this icon on the toolbar* . The cursor will be change to . Then *click on the component you want to copy* (to copy multiple components together, select them by left clicking and dragging)*.* A copy of the component will be attached to the cursor. *Left click* to place on a different position. The component can be rotated by pressing *CTRL + R* while moving as well.

8. Deleting a component: *Right-click on blank space → Edit → Delete* | Or, *Press F5* | Or, *click this icon on the toolbar* . The cursor will be change to . Then *click on the component you want to remove* (to delete multiple components select them by left clicking and dragging).

9. Renaming a component: *Hover the cursor on the name of a component.* The cursor will be change to . *Right-click on it → Type* the new name *→ Click OK.*
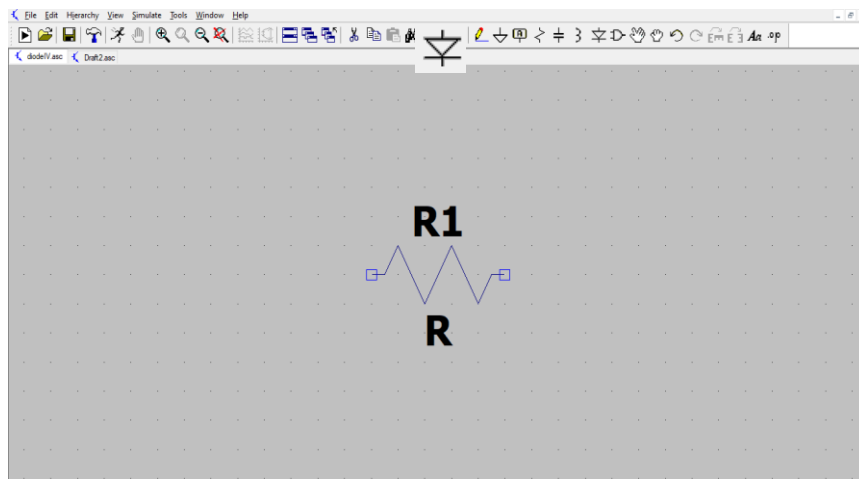
10. I-V characteristics of diode (DC Sweep [.dc]):

I. Create a new schematic file as instructed in step 4. Go to *File → Save as*. Rename the file as *diodeIV.asc* and save it at a suitable place.
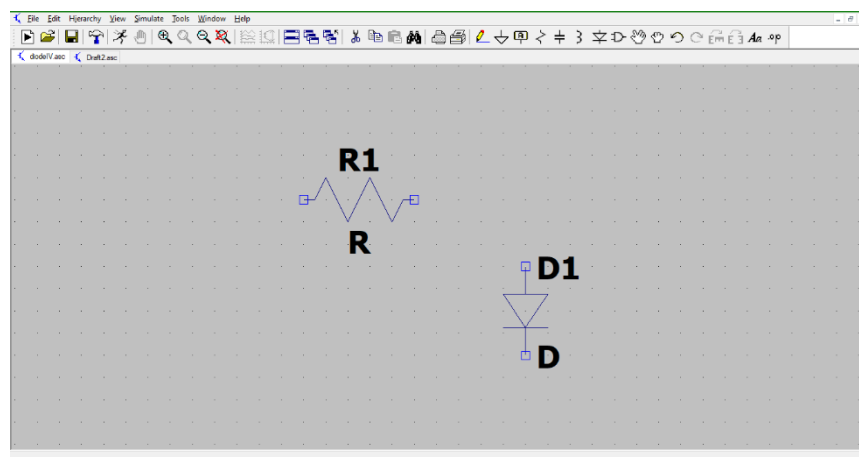
II. Select *View → check the Show Grid* option for better visibility of the interface.
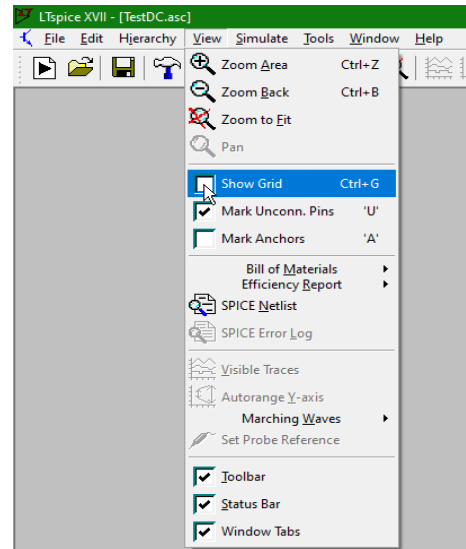
III. Insert a resistor from component library as instructed in the step 5.

IV. Type or find '*diode*' in the **Select Component Symbol** window in step 5 to insert a diode. Alternatively, *click this icon on the toolbar* .

V. Type or find '*voltage*' in the **Select Component Symbol** window in step 5 to insert a voltage source.
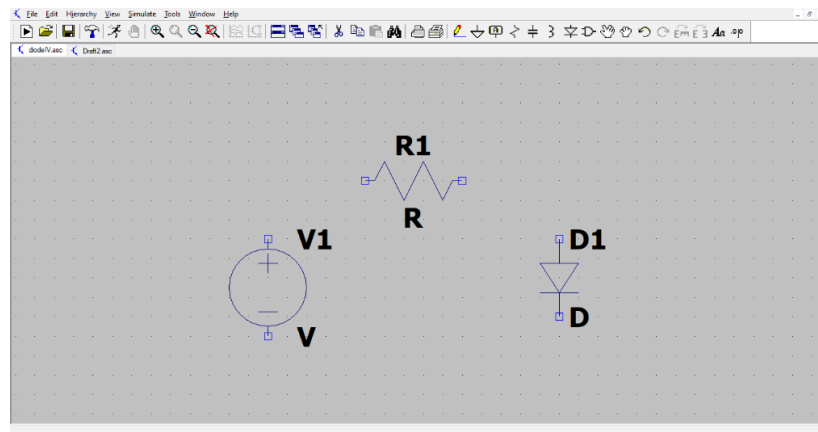
VI. Now we need to connect the components using wire. ***Right-click on blank space → Draft → Draw Wire*** | Or, ***Press F3*** | Or, ***click this icon on the toolbar*** ✏. The cursor will change into ⊹
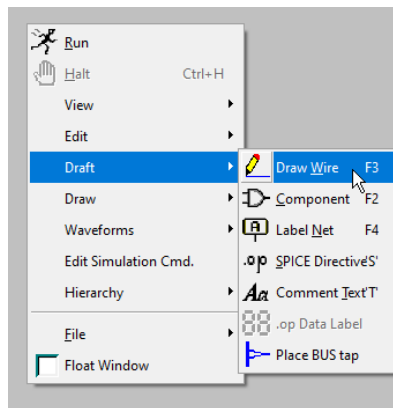

VII. ***Click on one of the terminals*** (small squares attached to a component) to start connecting. For example, shown in the figure below the steps in connecting the voltage source to the resistor R1.
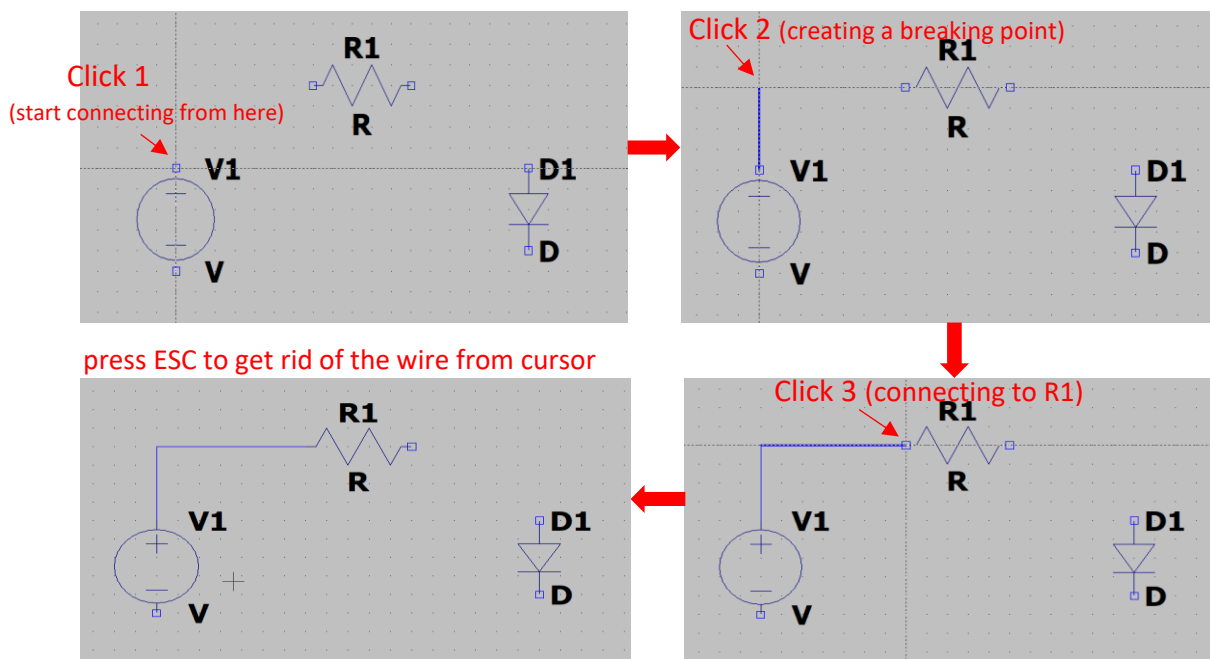
VIII. Similarly, wire all of the components together. The circuit will be as follows:
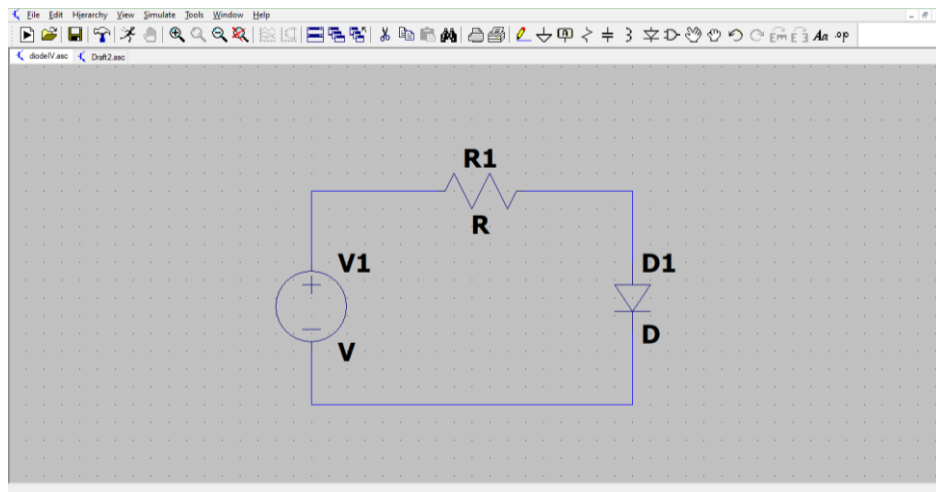

IX. Placing a ground: **Right-click on blank space → Draft → Label Net** | Or, **Press F4**. A **Net Name** window will open. Select **GND (global node 0) → click OK**. Alternatively, **click this icon on the toolbar** .The cursor will change into . Place the ground symbol as shown in the following figure.
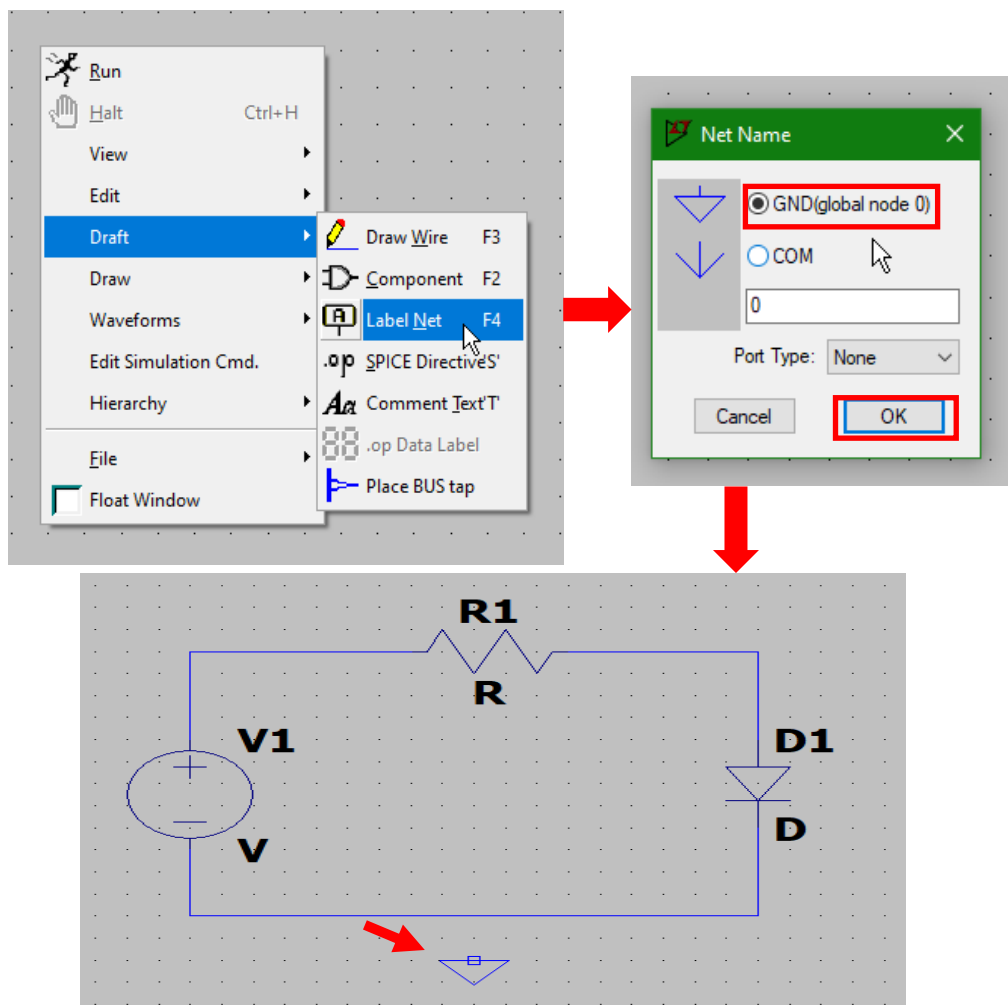
X.   Connect the ground to the circuit using wire as described in the steps IV and V.
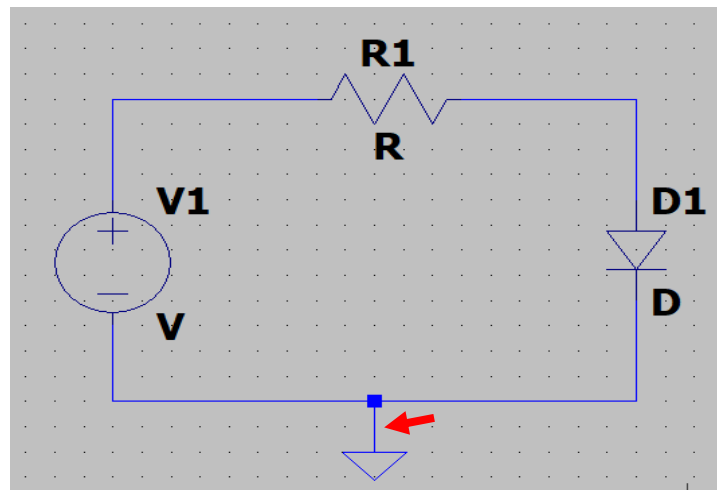

XI.   Now we need to set the value of the components. To set the resistance of a particular resistor (for example R1), ***hover the cursor on the resistor***. The cursor will change into a hand. ***Right click on it***. It will open up a new window for different specs of the resistor R1. Type **1k** for **Resistance [Ω]**. Leave the other fields as it is and ***Click OK***. This will set R1 as 1 kΩ.
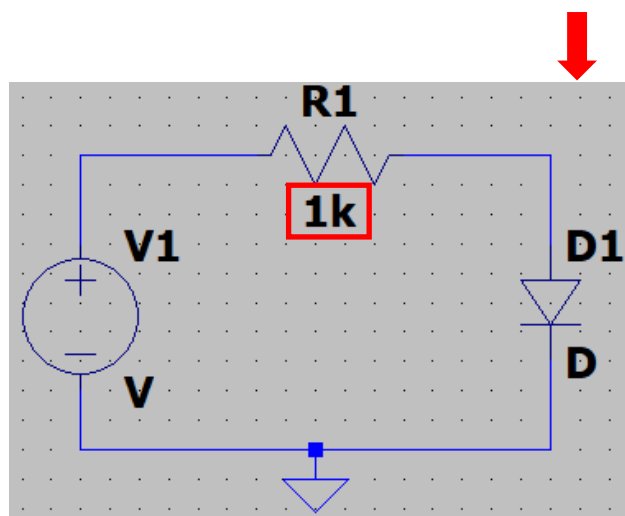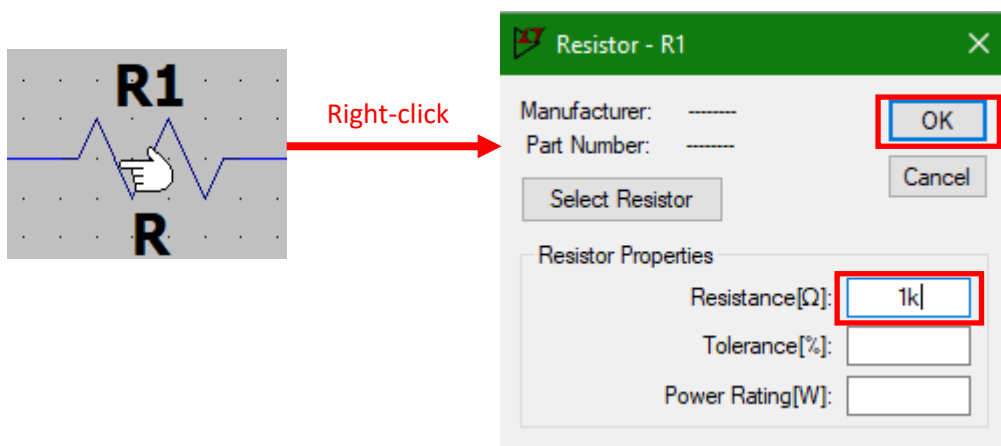
XII.   Now **Right-click on the diode** to open the property window. **Pick New Diode → Select 1N4148 by looking in the Part No. → Click OK.**
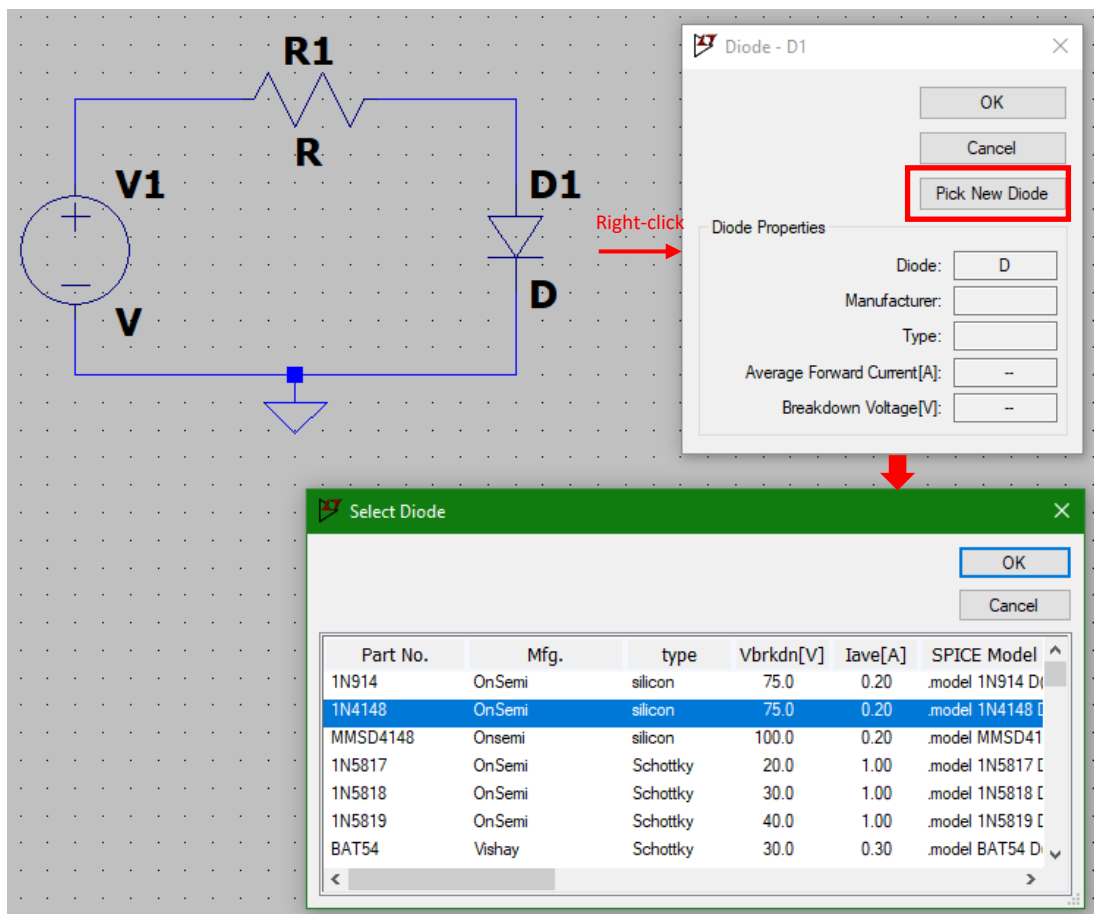

XIII.   To simulate the I-V characteristics of the diode, we need to vary the voltage applied to the circuit. Before that, we need to set an arbitrary value for the voltage. To do this, **hover the cursor over it → right click**. This will open a setting window for the voltage source. Type **'5'** in the DC Value [V] field and Click OK. This will set the voltage source as 5 V DC. There are some advanced options for the voltage source. We will explore those later.
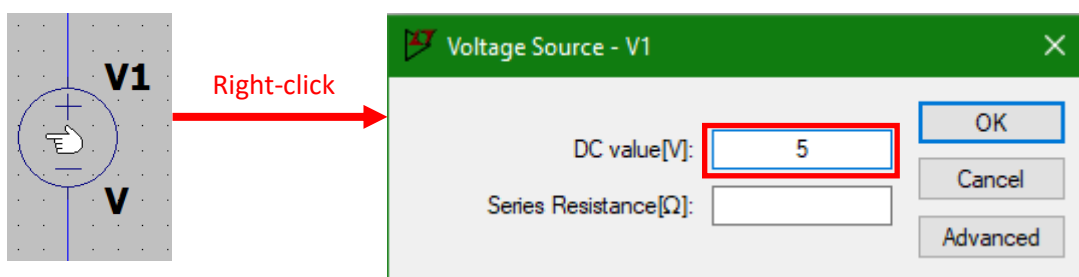
XIV. <u>Labelling nodes</u>: We will now label the circuit nodes. This helps in keeping track of voltages. To label a node, ***Right-click on blank space → Draft → Label Net*** | Or, ***Press F4*** | Or, ***click this icon on the toolbar*** 🅐 . A **Net Name** window will open. Type 'Vd' in the box and click OK. Place the label Vd as shown in the following figure. Then Vd denotes the potential difference between across the diode. The node variables are case-insensitive.
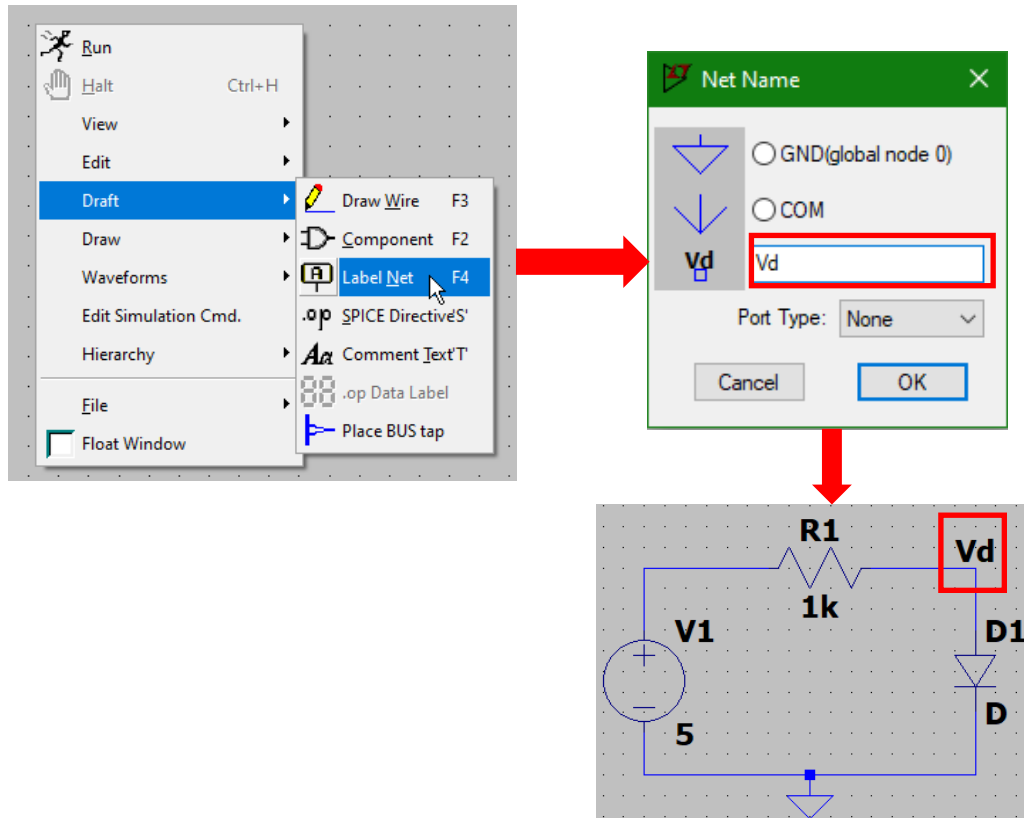

XV. <u>Running the simulation</u>: Now we are all set to run the circuit to observe the I-V characteristics of the diode. ***Right-click on blank space → Draft → SPICE Directive'S'*** | Or, ***click this icon on the toolbar*** .op This opens a netlist window. ***Right click on the black text box → Help me Edit → Analysis Command.*** This opens the '**Edit Simulation Command**' window.
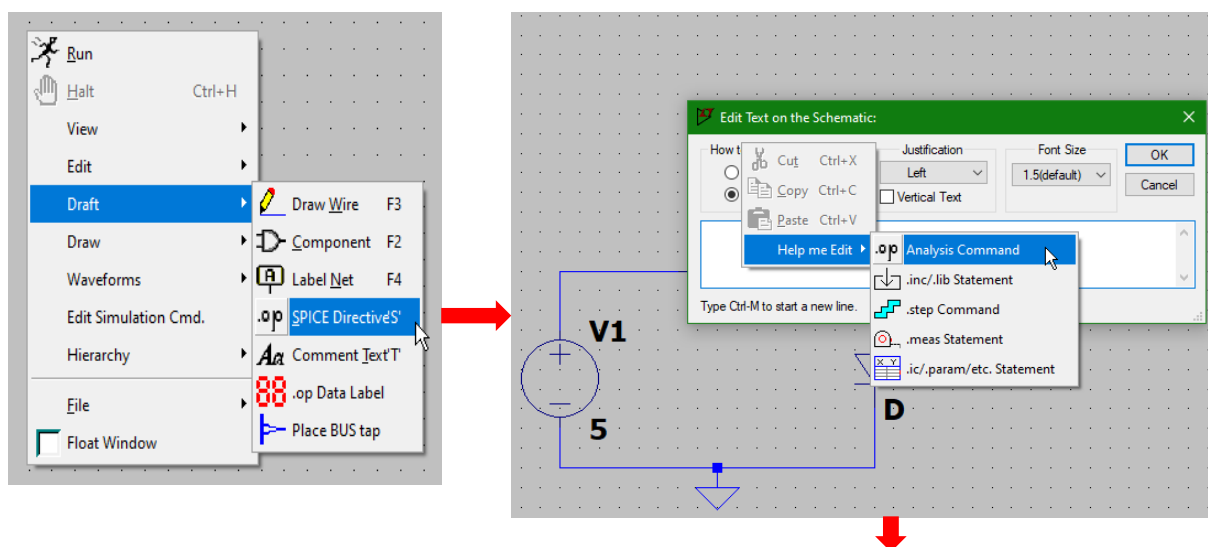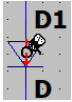
XVI.   In the **Edit Simulation Command** window, go to the '**DC sweep**' tab. In the '**1st source**' tab write **'V1'** in the 'Name of 1st source to sweep' field. Select the 'Type of sweep' as **'Linear'.** Fill up the following fields as **'**Start value' = 0; 'Stop value' = 10; 'Increment' = 0.001 and Click OK. Place the command anywhere within the grey interface. The input voltage (V1) is linearly varied from 0 V to 10 V with an increment of 0.001 V.
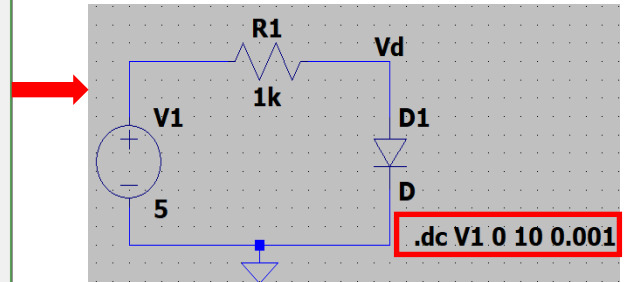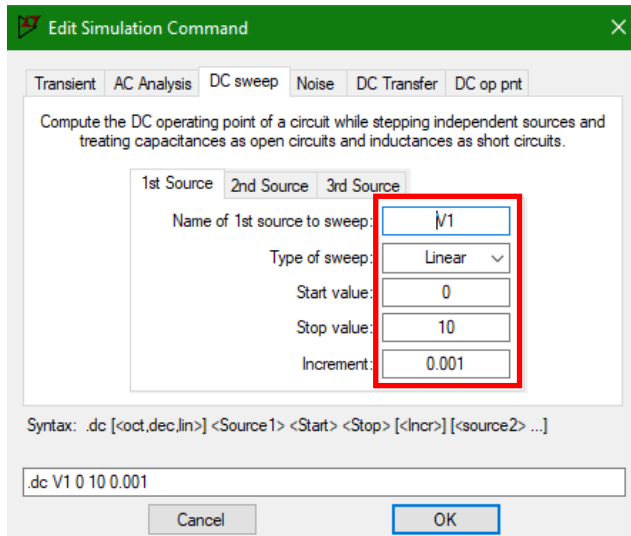
XVII. ***Right-click on blank space → Run*** | Or, ***click this icon on the toolbar*** to run the simulation. This opens a plot window (diodeIV.raw).
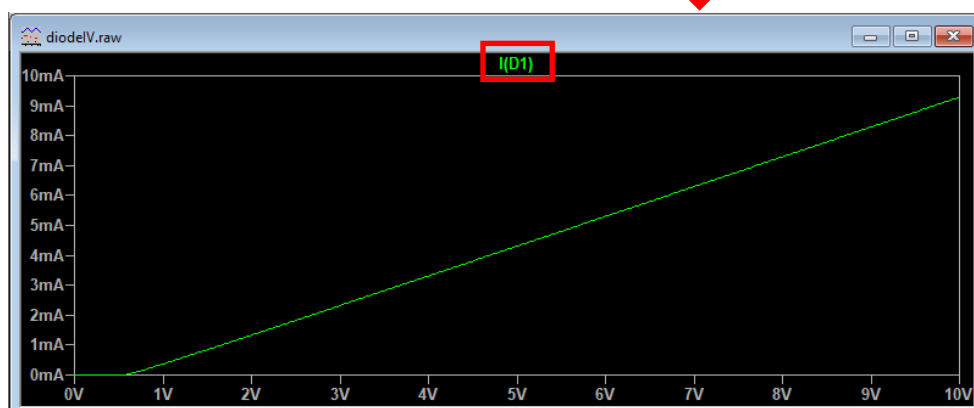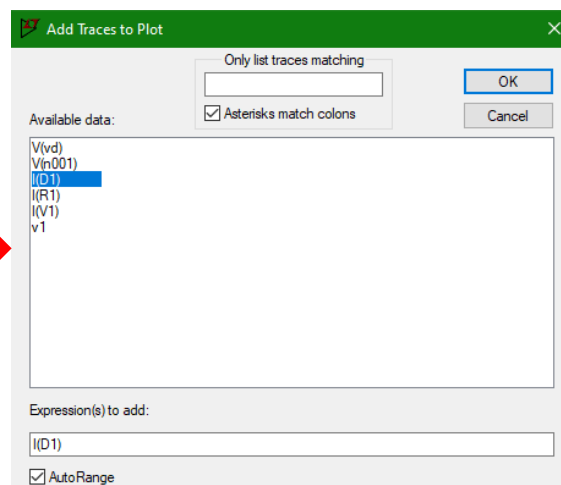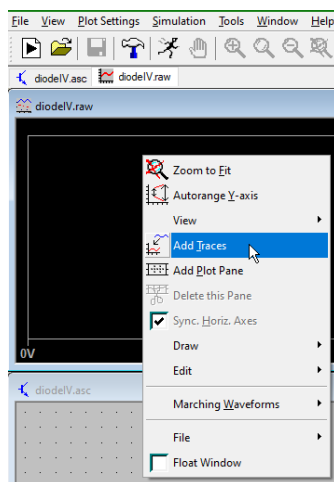
XVIII. To plot the current through the diode, ***Right click on the black area → Add Traces***. This will open a window where all the node voltages and the elemental currents are listed. ***Select I(D1) or I(R1) or I(V1) → Click OK***. You will get a plot like the above. Note that this is not the I-V curve for the diode. The varying source voltage is plotted in the horizontal axis. We have to plot I(D1) vs Vd.

XIX. Alternately, a voltage or current can be plotted by selecting first the circuit window and hovering the cursor over any node to view the voltage at that node or over any element to view the current flowing through that element. The cursor will change into **Vd** and **D1** respectively for node voltage and branch current. Left clicking will plot the particular current/voltage.

XX. To plot the diode voltage along the horizontal axis, hover the cursor over the horizontal axis. The cursor changes into a ruler ⬜ . ***Right click***. It will open a property window for the axis. Write '**V(Vd)**' in the 'Quantity Plotted' field and **Click OK**. ***Right click on the plot → View → Check the Grid.*** This is the I-V characteristics of diode.
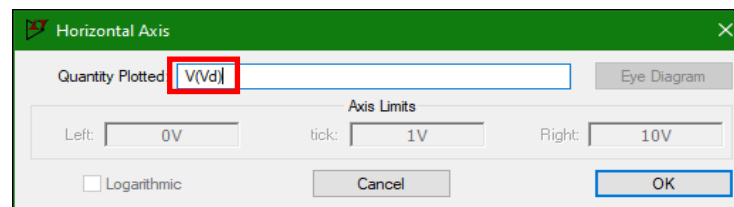
XXI. To see values at different positions in a plot, ***Left-click on the label of the trace.*** This enables the data cursor. Click and drag the cursor. The attached window shows the cursor coordinates.
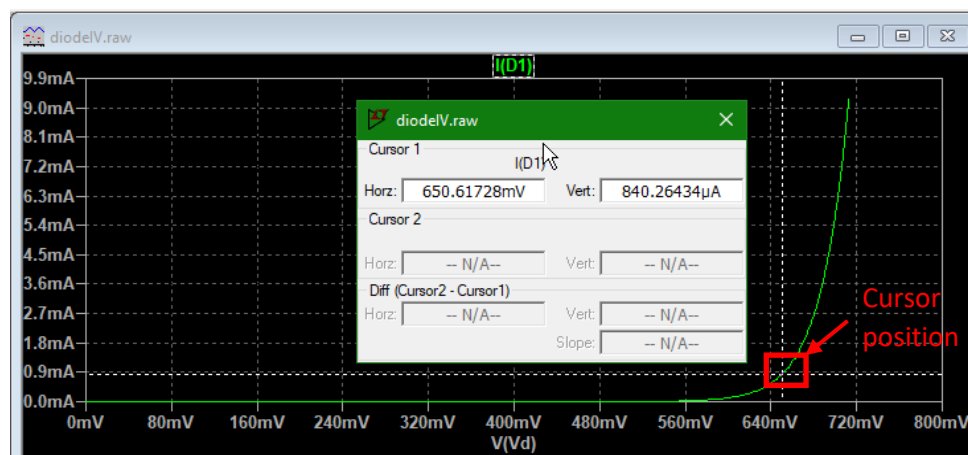
XXII.   To mark a data point, ***Right click on the plot → Draw → Cursor Position.*** The colour of the data point indicator can be changed by right-clicking on it.
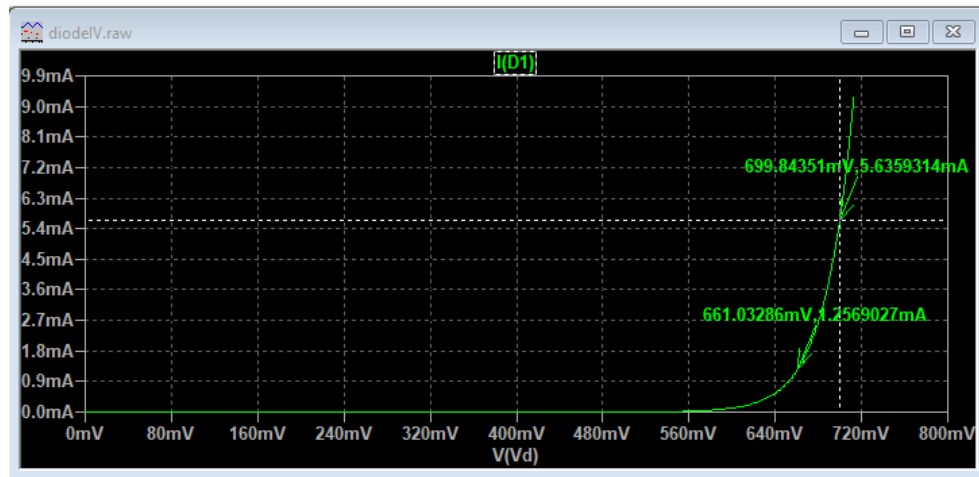
XXIII.   The plots can be saved for future use and analysis by ***selecting the plot window → File → Save Plot Settings <u>As</u> → Name.plt.***

11. <u>Simulation of Half-wave and Full-wave rectifiers (Transient analysis [.tran])</u>: We shall now use transient analysis to simulate a HW and a FW rectifier circuit and study the waveforms.

<u>Half wave rectifier (HW)</u>

I.   ***Open a new schematic*** and ***build the circuit*** below using the processes outlined in steps <u>10(I)</u> through <u>10(XIV)</u>. ***Select the diode model*** as '1N4148' and ***label the nodes*** as shown in the figure.
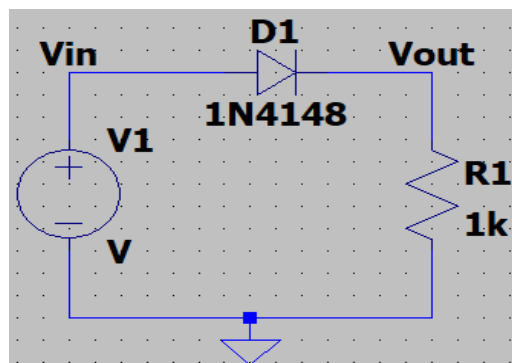

II.   A sinusoidal ac voltage will be applied to the rectifier circuit to study its behaviour. Go to the properties of the voltage source by ***Right clicking on it → Advanced***. This will launch the property editor, as illustrated below. ***Configure*** the properties as seen in the next illustration.
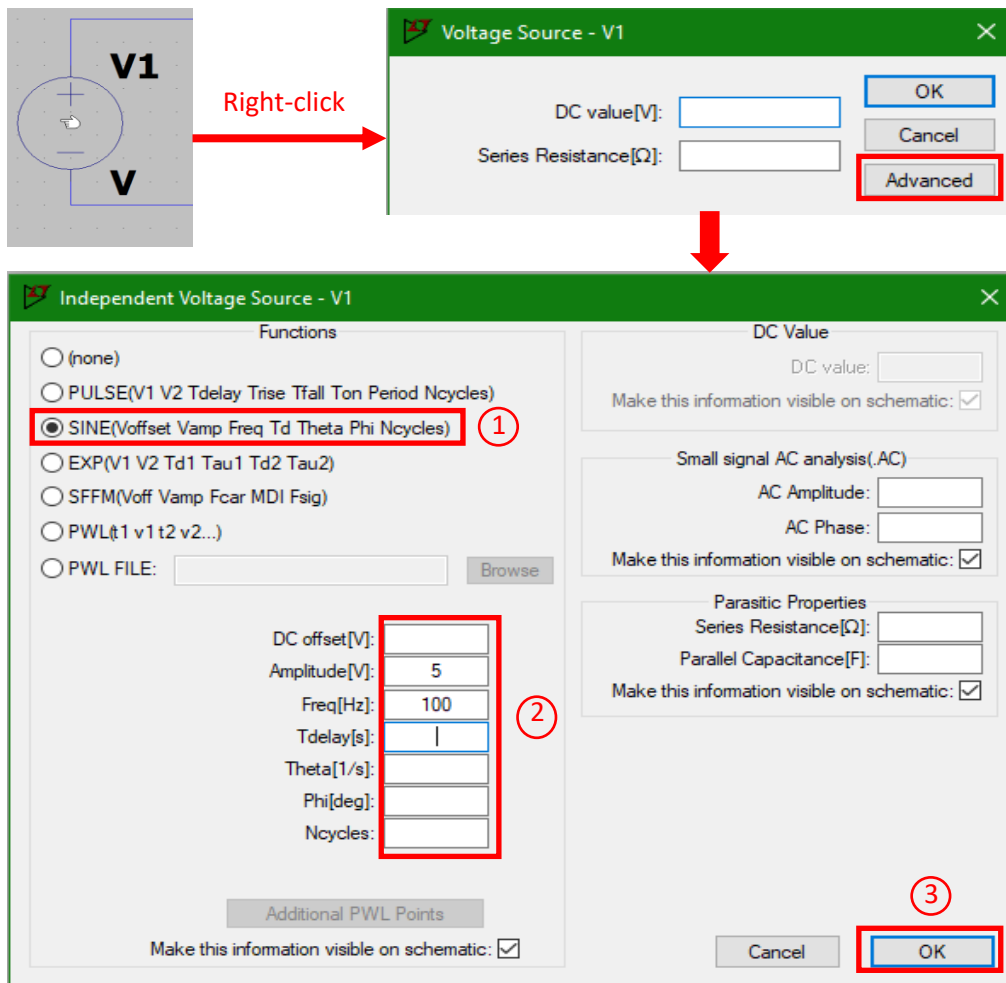
III. Go to **'Edit Simulation Command'** window following the instructions in step 10(XV). In the transient tab, set the 'Stop time' as **50 ms** with a 'Maximum Timestep' **1 μs**. Leave the remaining fields unchanged.
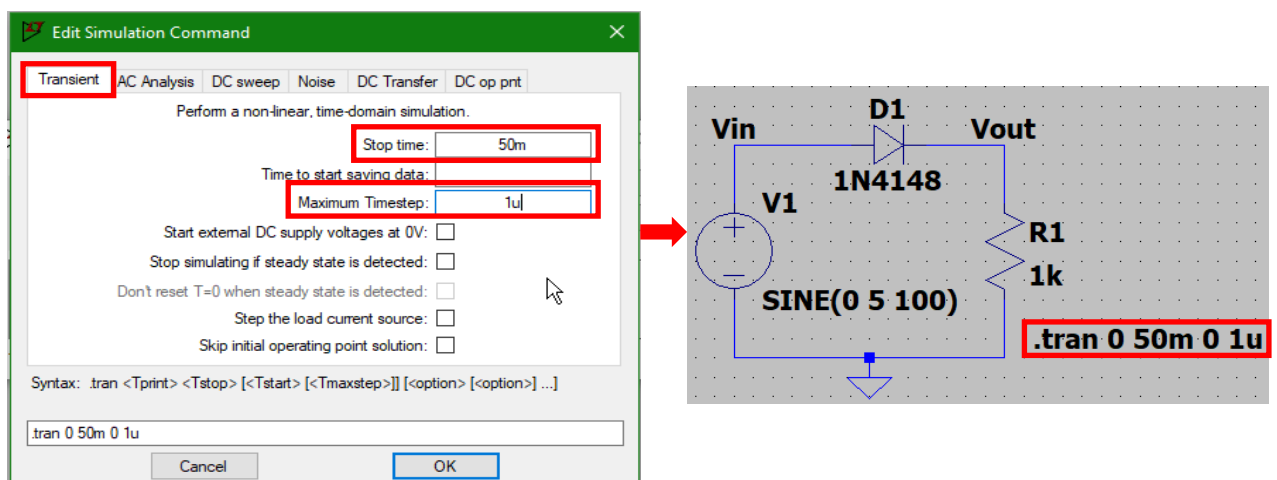
IV.   *Run the simulation* [See if necessary].

V.   Plot **Vout** following one of the approaches outlined in step or . Repeat the same process to plot **Vin** in the same plot window.
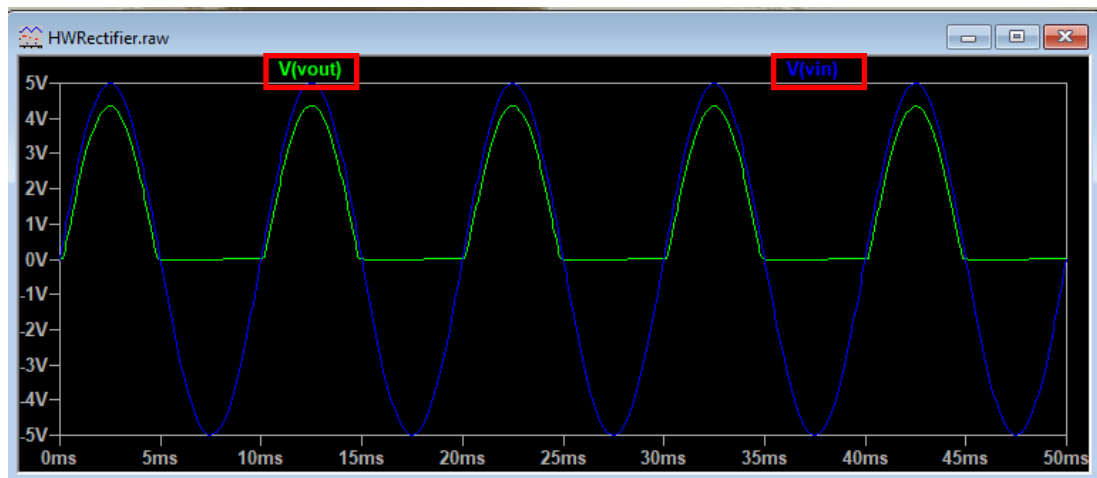

VI.   To delete a particular trace, *hover the cursor on the parameter → Right click → Delete this Trace*.
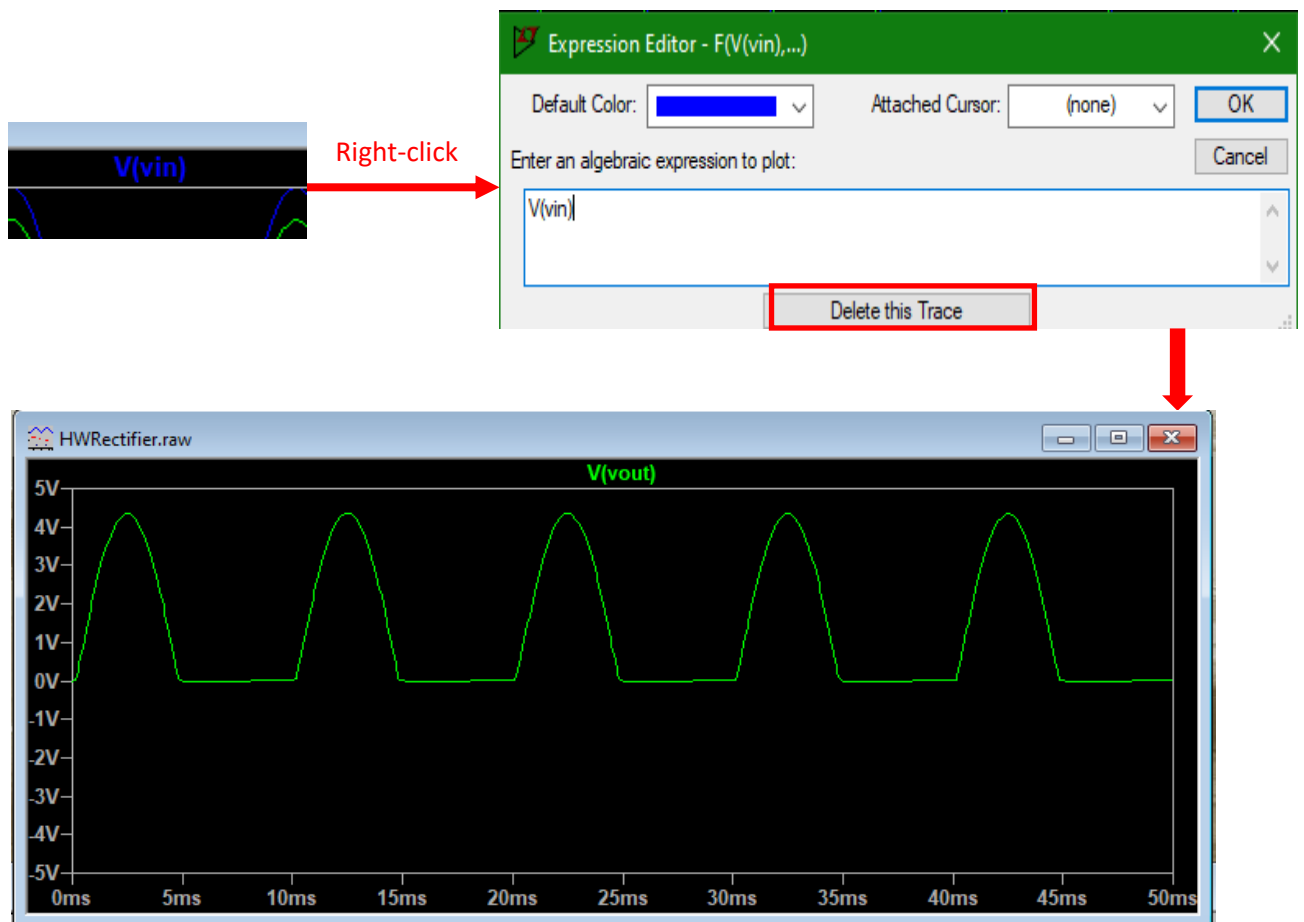
VII.   Multiple plot panes can be added to separately plot different circuit parameters. ***Right click on the black area → Add Plot Pane.*** The plot window will now have a new pane. If necessary, adjust the size of the plot window by extending it from the boundary or maximizing it. Currents/voltages can be plotted to each pane using the same approach as in stages 10(XVIII) or 10(XIX). To delete a specific pane, ***Right-click it → Delete this Pane***.
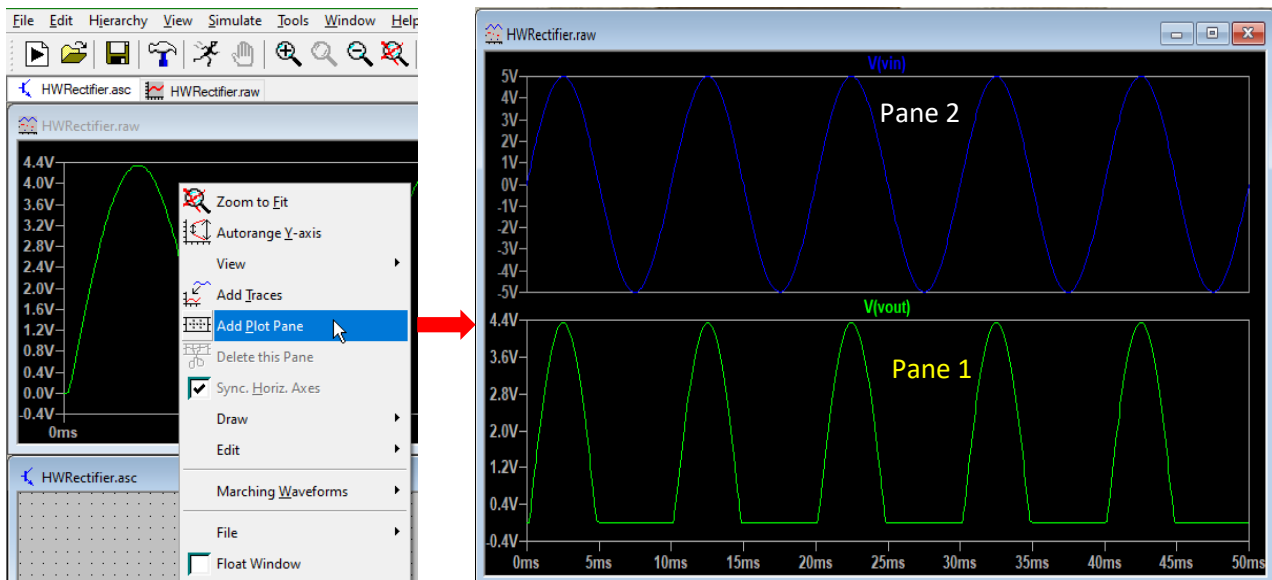

VIII.   <u>Voltage across an element</u>: To see the potential across the diode for example, ***Right click on the black area → Add trace →*** write ***V(b) – V(c)*** in the '**Expression(s) to add field**' of the Trace adding window and ***hit OK.*** Note that the maximum reverse bias voltage (Peak Inverse Voltage) across the diode is 5 V.
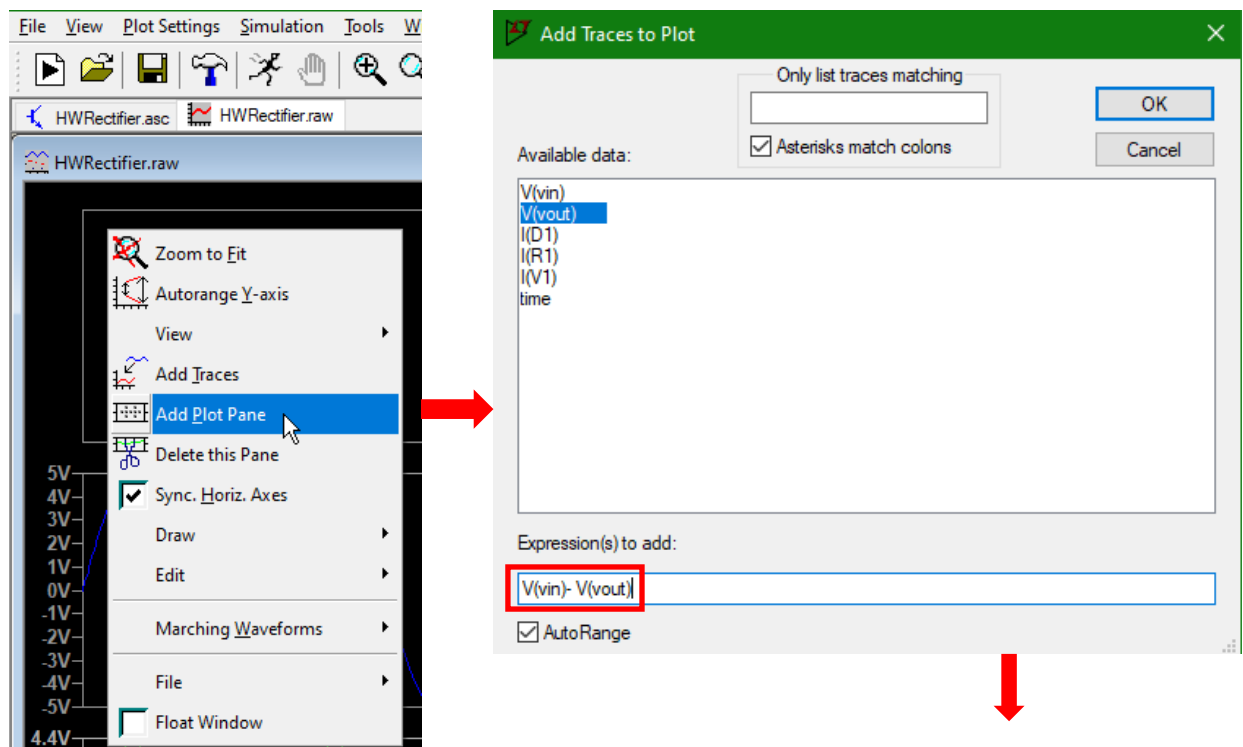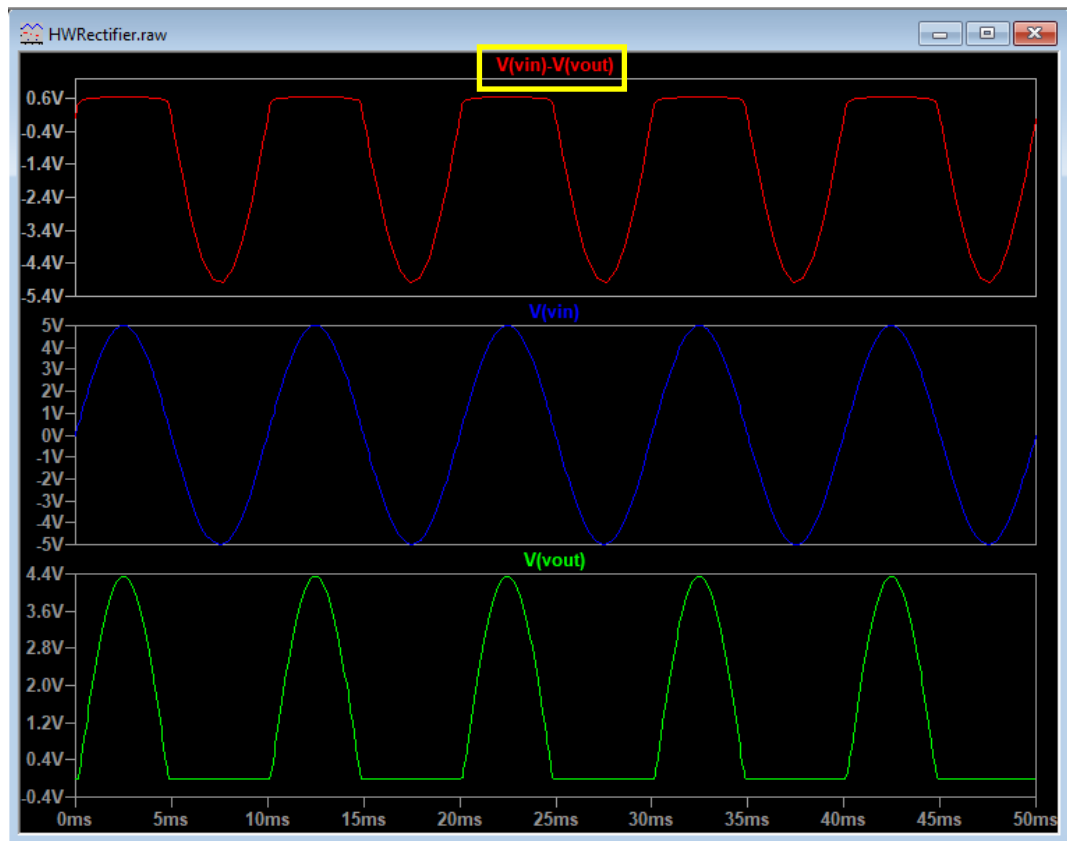
IX. Now go to the circuit window and add a capacitor in parallel to the resistor. To insert a capacitor, type or find **'cap'** in the select component symbol window in step 5. Alternatively, select this icon on the toolbar ⊥ Change the value of the resistance to 10 kΩ. This will increase the time constant.
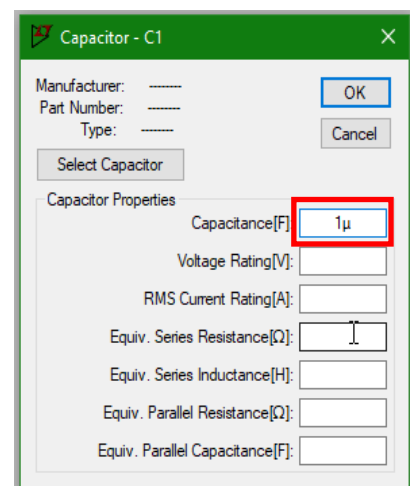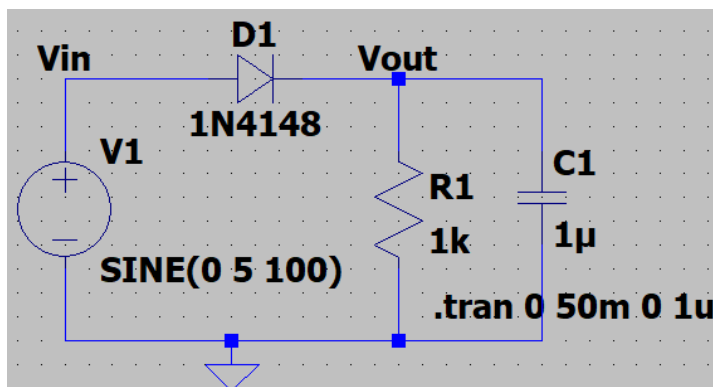
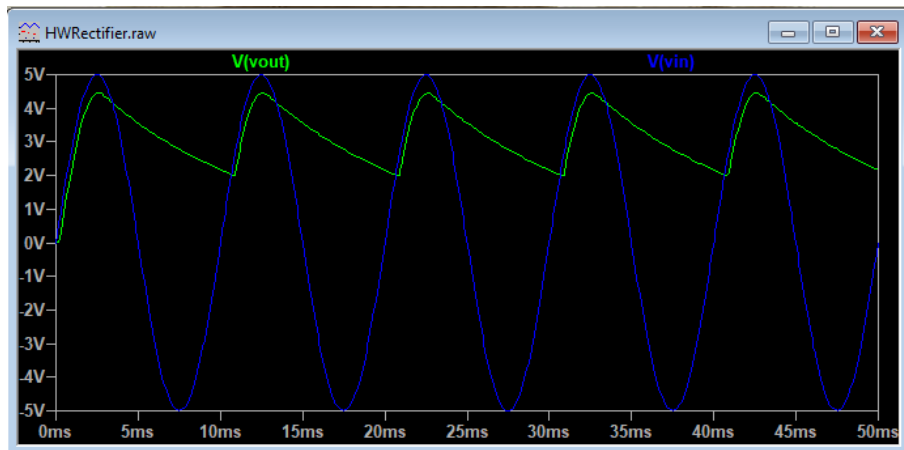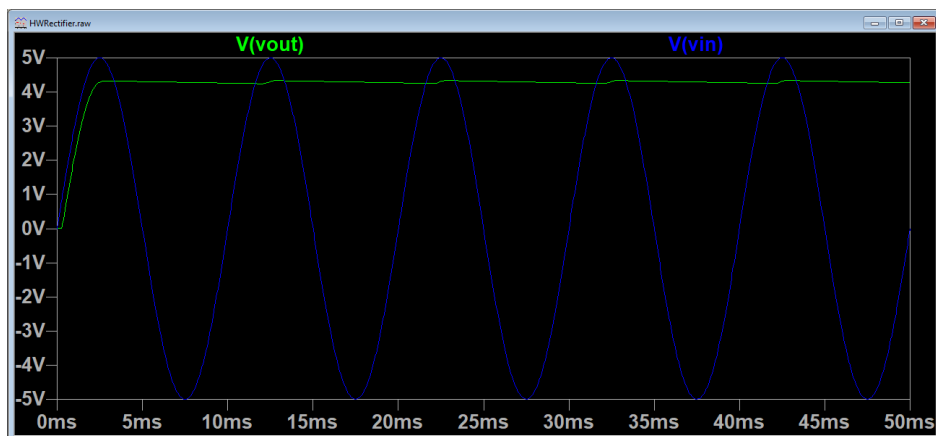X.   ***Run again*** and observe the waveforms. Further increase in resistance or capacitance will make the output voltage more flatten.

XI.   Now increase the capacitance to 47µF and simulate again. You will get a plot like this.

XII.   Explore more plot window features, such as dragging a rectangle to zoom into a specific area, changing trace colours by right clicking on the parameter label, and left clicking on the parameter label to enable the data pointer to see the coordinates, right clicking on axis to change the axis parameter or range or scale.

Full-wave rectifier (FW)

I.   ***Open a new schematic*** and ***build the circuit*** below using the processes outlined in steps 10(I) through 10(XIV).
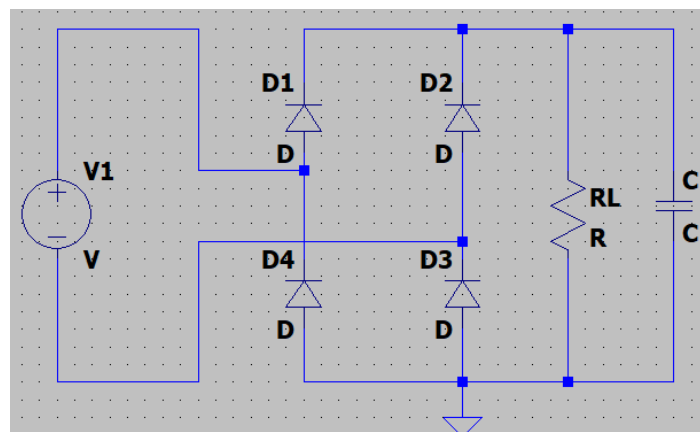
II.   ***Select the diodes model*** as '1N4148' and ***label the nodes*** as shown in the following figure. Use RL = 10 kΩ and C = 1 μF. Set V1 as 5 V (amp) 100 Hz sinusoidal voltage. *[Go to 10(XI), 10(XII), 10(XIV), and 11(II) if necessary]*. Note that the negative terminal of the voltage source (V1) is not connected to ground. So, to observe the input waveform, two nodes 'Vin+' and 'Vin-' are defined. Then *V1 = Vin+ - Vin-.*
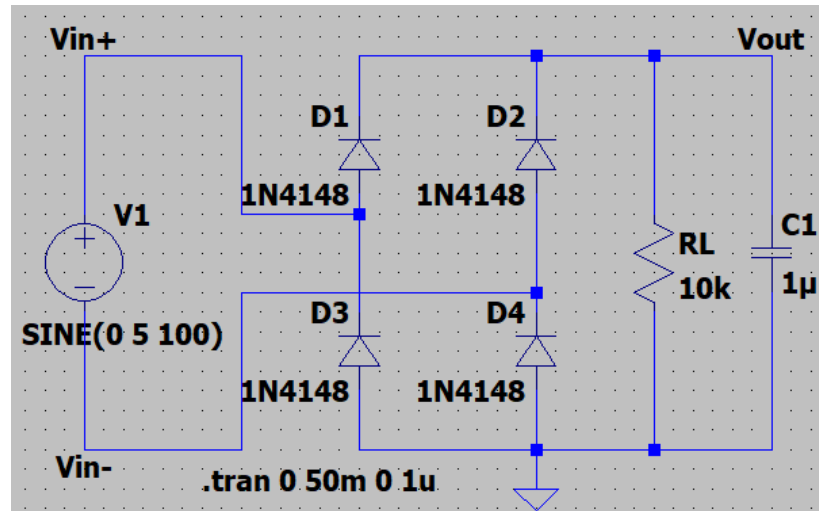
III.   Set the simulation mode as 'Transient' with a 'Stop time' equal to **50 ms** and **1 μs** 'Maximum step size'. [See step 11(III) if necessary]

IV.   ***Run the simulation***

V.   Plot *'Vout'* following the process outlined in 10(XVIII) or 10(XIX). Plot **V1 = Vin+ - Vin-** following 11(VIII). You should get a plot like this. The voltage across the diode D1 can be plotted using *Vin+ - Vout*. The peak output voltage is 2Vd less than the peak input voltage, which is consistent with theory.

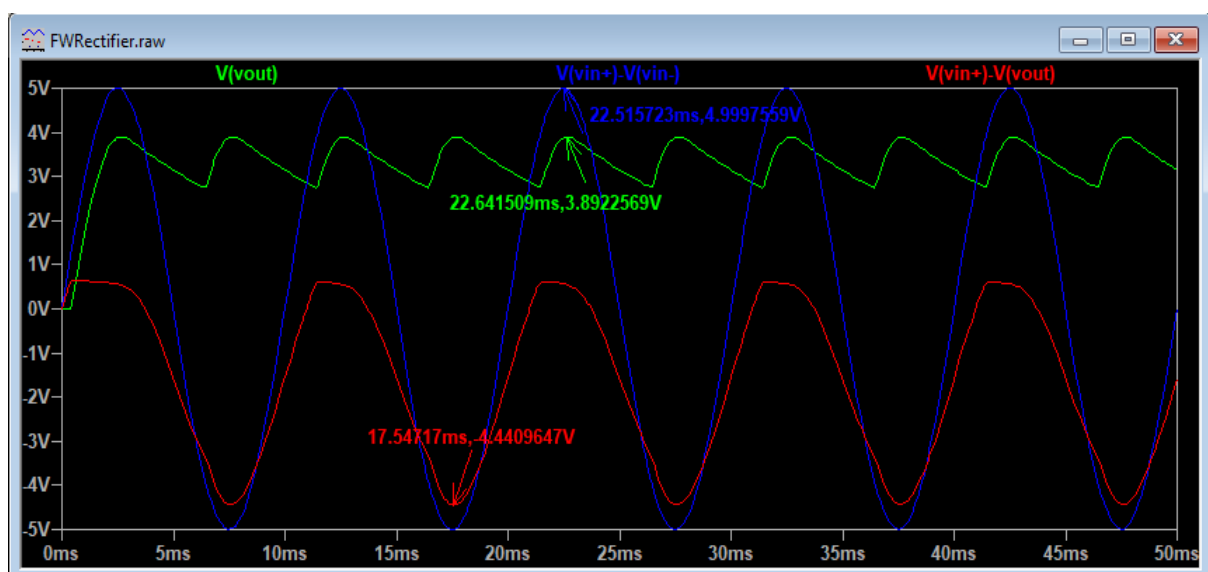VI.   Now increase the capacitance to 47μF and simulate again. You will get a plot like this.

12. I-V characteristics of Zener diode and its application in voltage regulation ([.dc], [.tran], [.step])

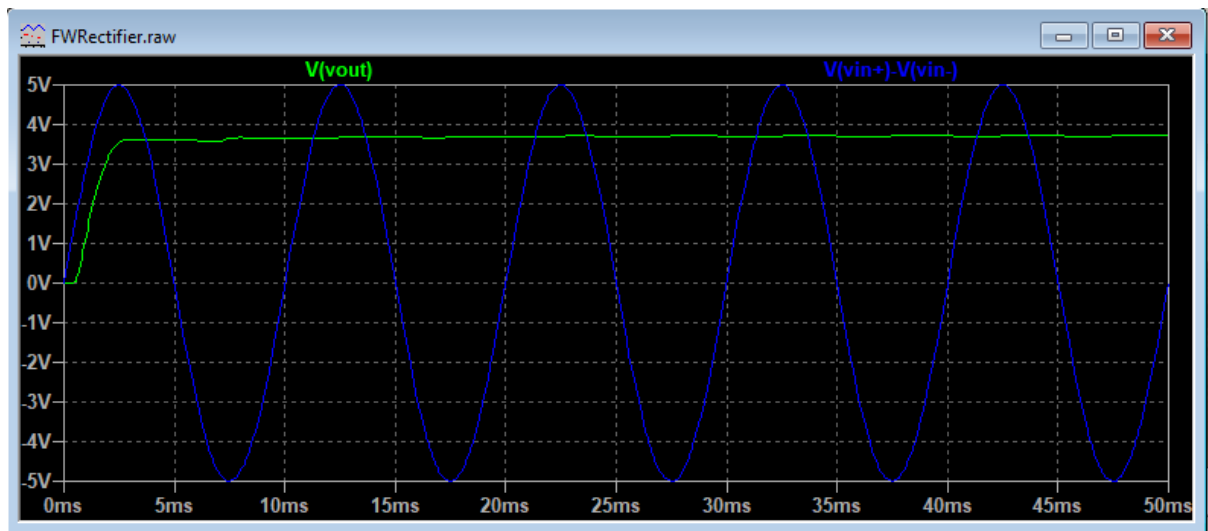I.   **Open a new schematic** and **build the circuit** below using the processes outlined in steps 10(I) through 10(XIV). To insert a Zener diode, type or find **'zener'** in the select component symbol window shown in step 5.

II.   **Select the diode model** as '*BZX84C6V2L*' and **label the nodes** as shown in the following figure. Set R = 470 kΩ.
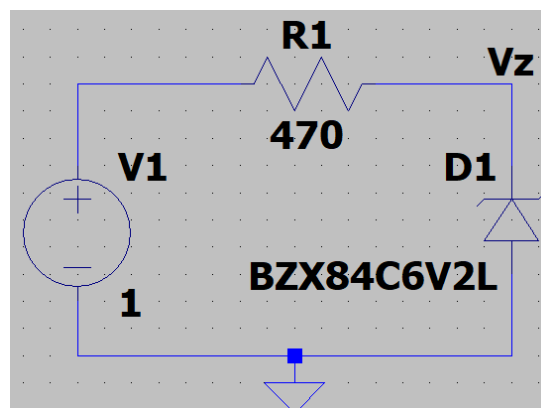
III.   Go to the **Edit Simulation Command** window following the procedure outlined in 10(III). In the 'DC sweep' tab, set the properties to sweep V1 as shown in the figure below.
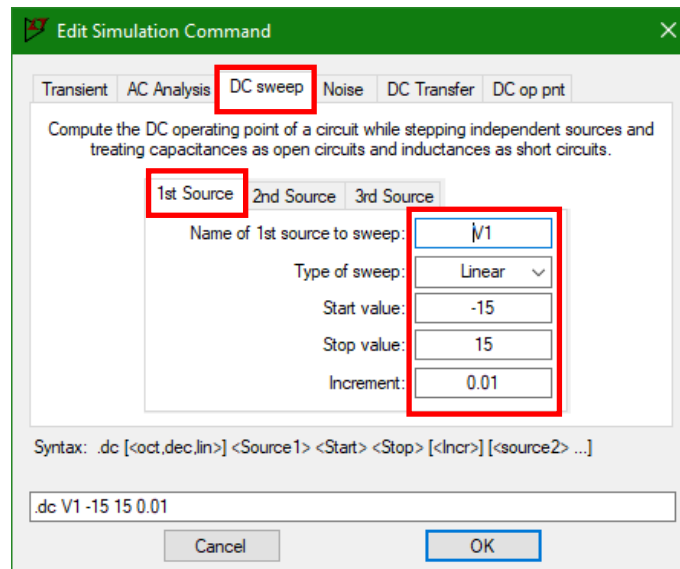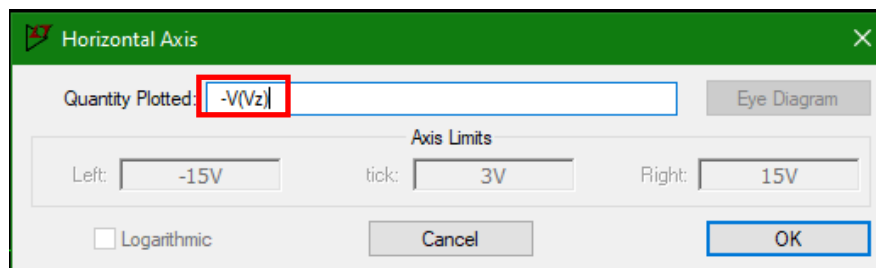

IV.   ***Run the simulation** [See 10(XVII) if necessary].*

V.   Note that **'V1'** is plotted along the horizontal axis. Change the horizontal attribute to **'-Vz'**

VI.   Plot the current through the resistor. *[See steps 10(XVIII) or 10(XIX) if necessary].* The plot will look like the following figure. Notice that, in the forward bias condition, the Zener diode acts like an ordinary diose. However, for a particular reverse bias voltage (6.2 V in this case) the Zener diode goes into breakdown.

VII.   Now we will observe the voltage regulative behaviour of a Zener diode and calculate the **line regulation**. ***Add a current source in parallel to the Zener diode.*** We assume that the load draws a constant current from the circuit, which we model with a current source in place of a load. ***Set the current as 1 mA***.

VIII.   To observe the line regulation (i.e. change in the output voltage for a change in the input voltage, measured in mV/V), we vary the input voltage linearly.

IX.   Go to the **Edit Simulation Command** window following the procedure outlined in 10(III). In the 'DC sweep' tab, set the properties to sweep V1 as shown in the figure below.

X.   ***Run the simulation*** [See 10(XVII) if necessary].
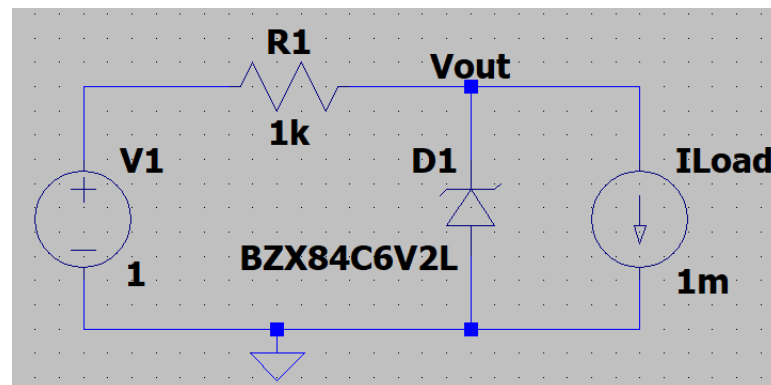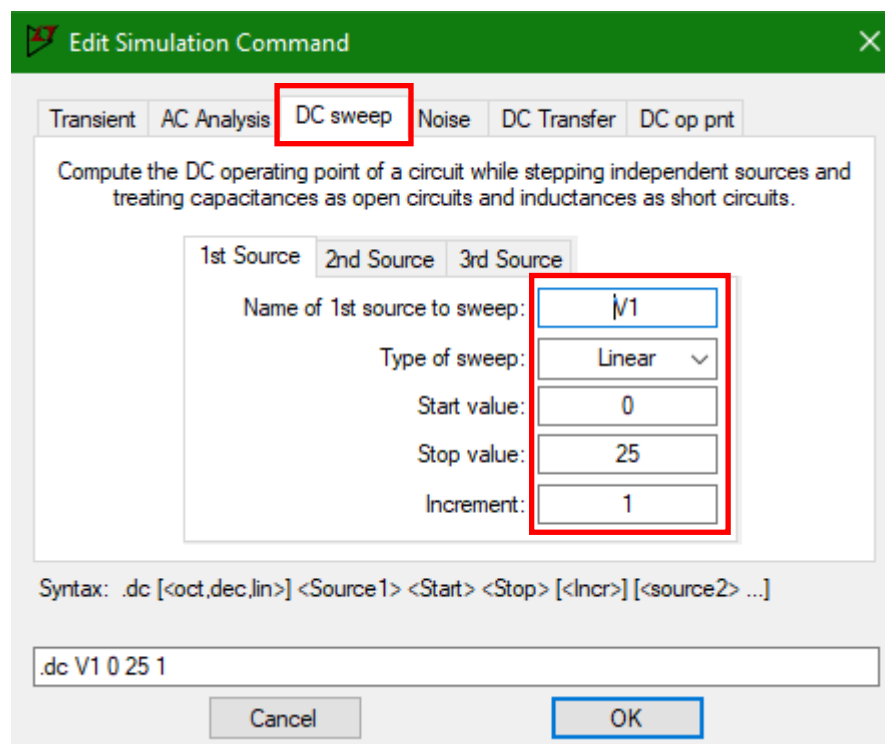
XI.   Plot the the output voltage (Vout). The plot should look like the one shown below. As expected, the output voltage is constant (almost) from the point where the input voltage exceeds the breakdown voltage of the zener diode (6.2 V in this case). A close inspection to the plot reveals that after the breakdown voltage, the output voltage has a non-zero increasing slope. This is the line regulation.
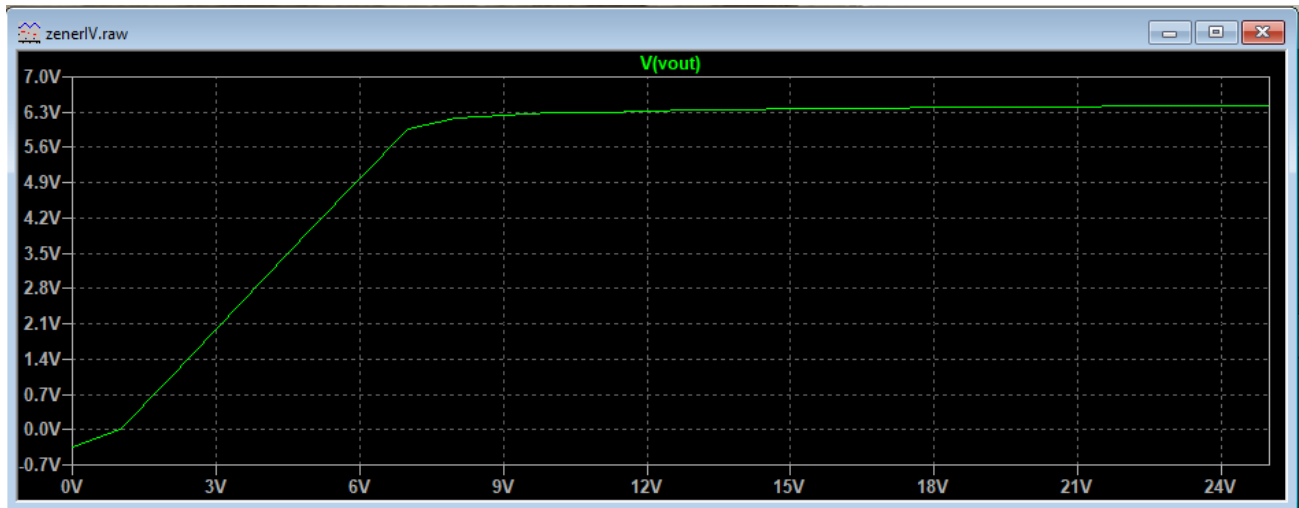
XII.   To measure the slope of the output voltage for the portion where the input voltage is above the breakdown voltage take two cursors and place them to a few volts apart as shown in the figure below.
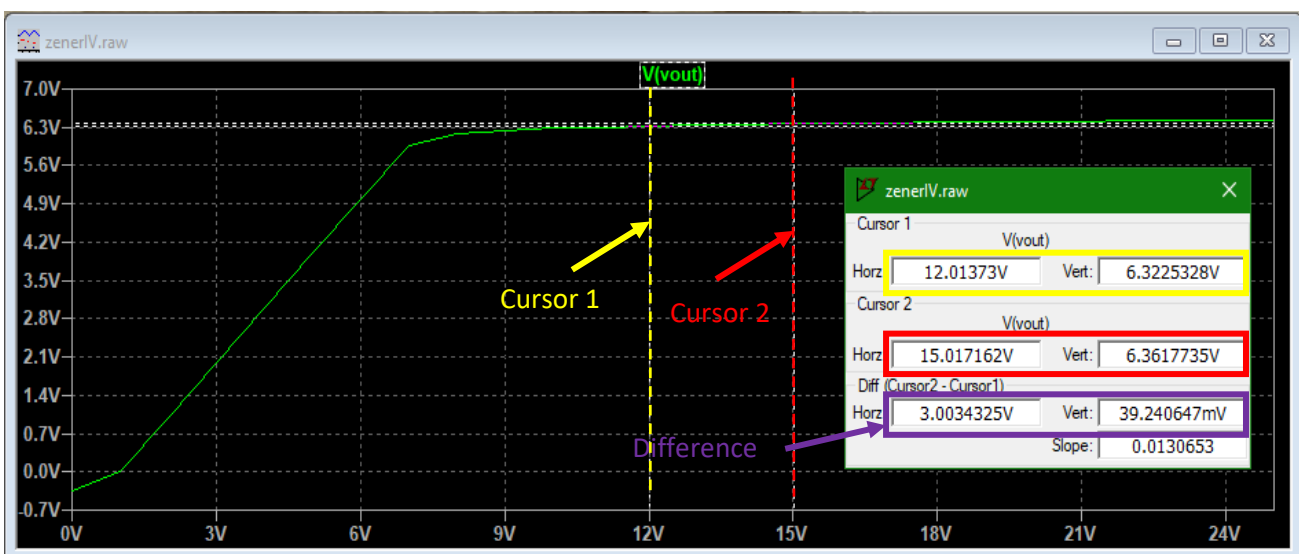

XIII.   The horizontal and vertical difference between the two cursors is shown in the cursor window. So, the line regulation is,

$$\frac{(39.24067)}{3.0034325}\left(\frac{mV}{V}\right) \approx 13 \left(\frac{mV}{V}\right)$$

XIV.   We will now measure the load regulation. Load regulation, measured in mV/mA, is the change in output voltage with respect to load current. Delete the dc simulation command placed in the grey interface. Set the input dc voltage source to a voltage above the breakdown voltage (previously set to 13 V).

XV.   Remove the current source and connect a resistor in parallel to the Zener diode. This is the load in this case.
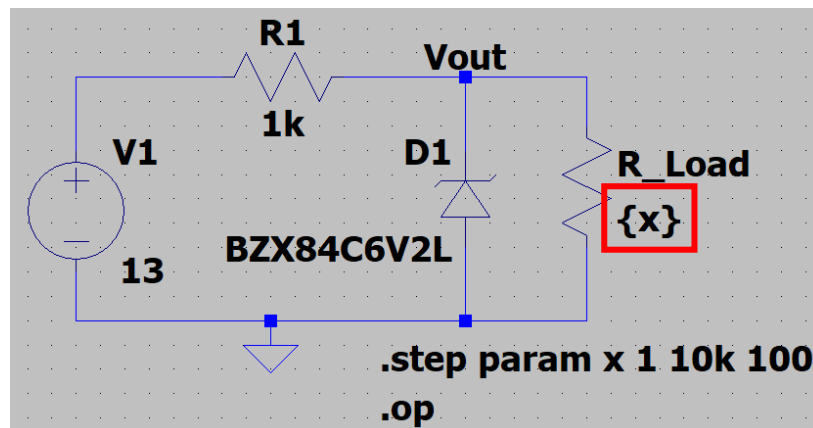
Parametric sweep ([.step]

XVI.   We will now vary the load resistance to represent it as a load drawing variable current. Set the resistance of the resistor to a variable inside the curly braces (for example **{x}**).

XVII.   *Right-click on blank space → Draft → SPICE Directive'S'* | Or, *click this icon on the toolbar* ⋅ᵒ|ᵖ . This opens a netlist window. *Right click on the black text box → Help me Edit → .step Command.* This opens the '**.step Statement Editor'** window.
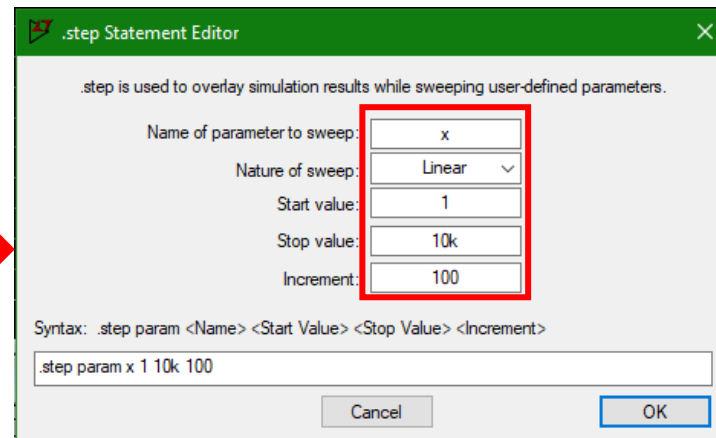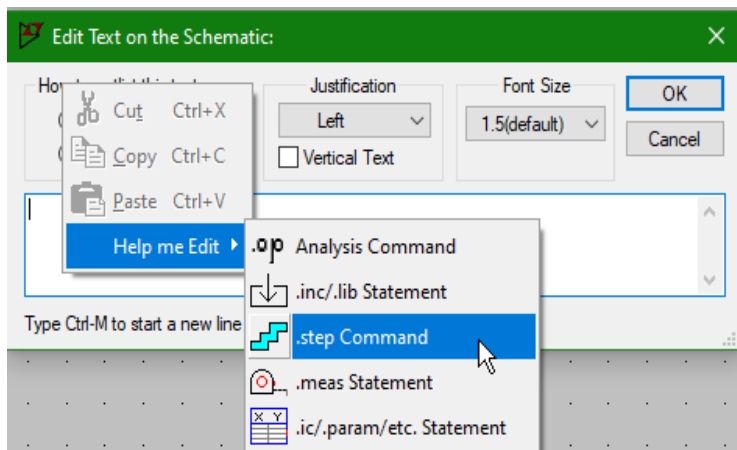

XVIII. In the **.step Statement Editor** window set '**x**' as the name of the parameters to sweep. Select '**Linear**' as the sweep type. Set a start value of **1** and an end value of **100k**. Set the step size as **100** Ohm. *Click OK*. Place the simulation command anywhere in the grey interface.

XIX.   *Right-click on blank space → Draft → SPICE Directive'S'* | Or, *click this icon on the toolbar* . This opens a netlist window. *Type '.op'* and *Click OK*. Place the dc operating point (.op) anywhere in the grey interface.

XX.   *Run the simulation* [See <u>10(XVII)</u> if necessary]. Note that the load resistance is plotted along the horizontal axis. Plot the output voltage, V(vout). Notice that, the output voltage

drops with the decrease in the load resistance below 1 kΩ. The zener fails to regulate the voltage when the load current gets high (lower load resistance).


XXI.   We are interested to calculate the load regulation from the portion where the load voltage remains constant (apparently). To do this, take two cursors and place them to a few kΩs apart as shown in the figure below.
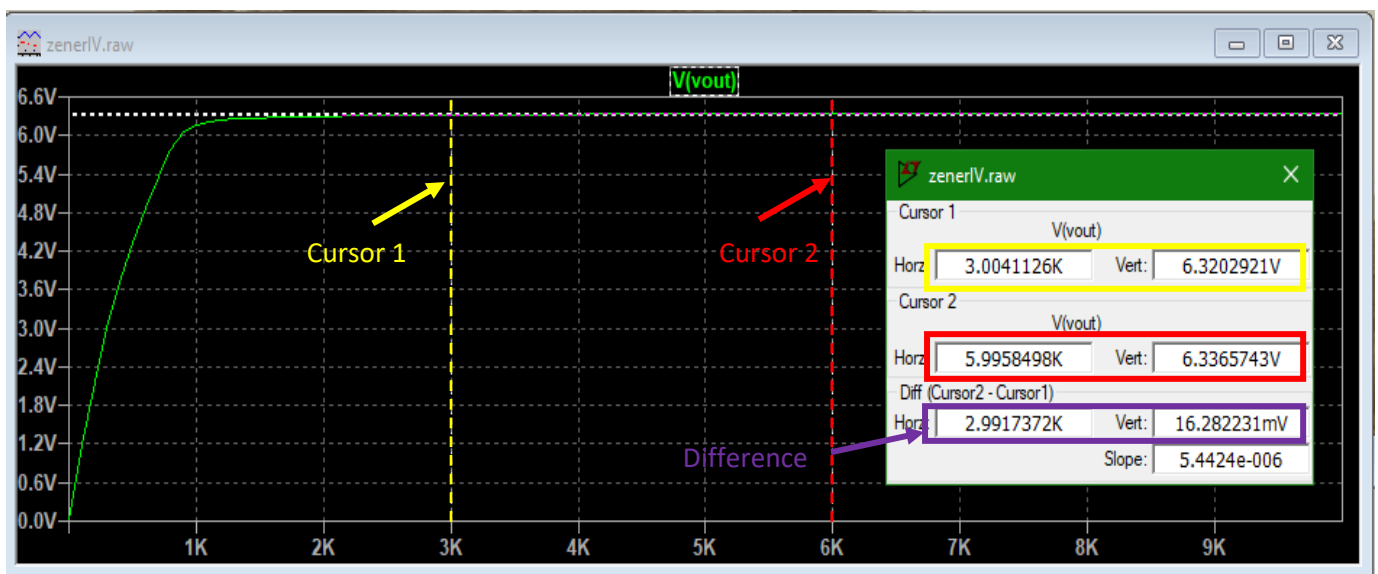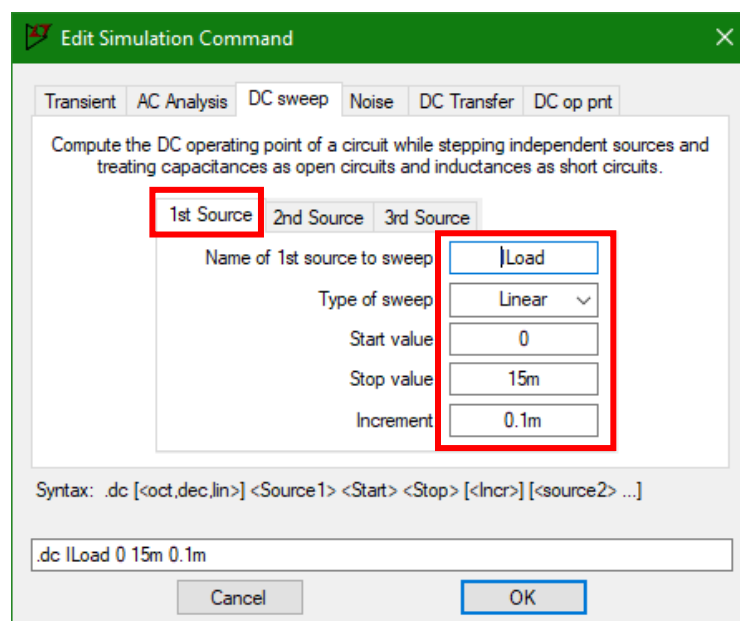

XXII.   The horizontal and vertical difference between the two cursors is shown in the cursor window. So, the line regulation is,

$$\left(\frac{16.282231}{\frac{6.2}{3} - \frac{6.2}{6}} \right) \left(\frac{mV}{mA} \right) \approx 17 \left(\frac{mV}{mA} \right)$$

XXIII. Alternatively, we can calculate the load resistance by modelling the load as a varying current source. Replace the load resistor with a current source and set a dummy value (let's say 1 mA).
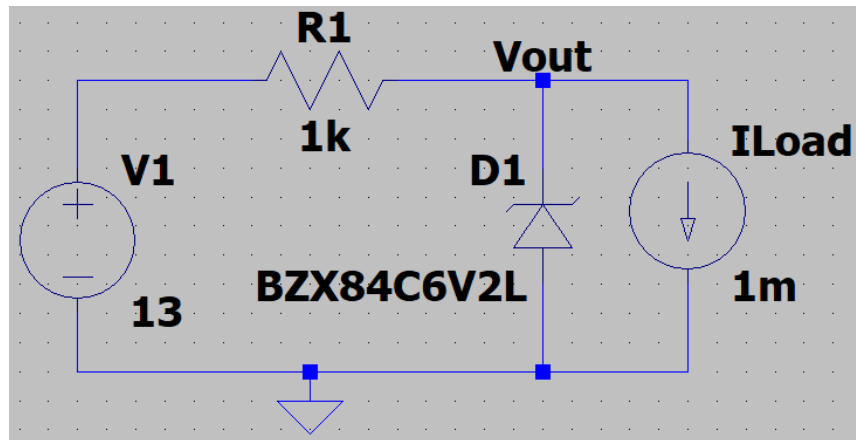

XXIV. Go to the **Edit Simulation Command** window following the procedure outlined in 10(III). In the 'DC sweep' tab, set the properties to sweep ILoad as shown in the figure below.

XXV. ***Run the simulation** [See 10(XVII) if necessary]*.

XXVI. Plot the the output voltage (Vout). The plot should look like the one shown below. As expected, the Zener can regulate the voltage within a range of the load current. If the load current is higher than that, the load voltage drops. A close inspection to the plot reveals the output voltage has a non-zero decreasing slope where it is apparently constant. This is the load regulation.

XXVII.   TO measure the load regulation, take two cursors and place them to a few mAs apart as shown in the figure below.
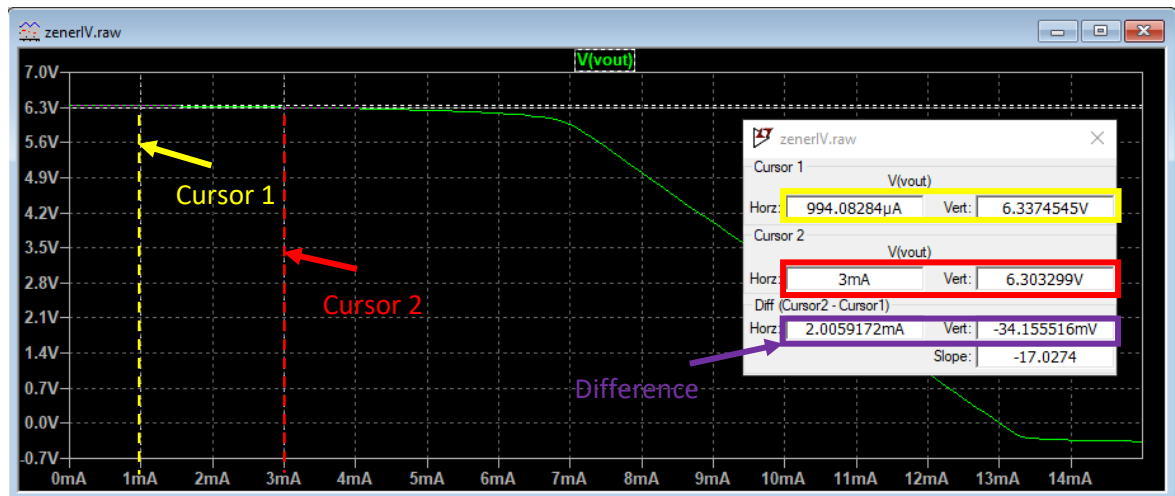

XXVIII.   The horizontal and vertical difference between the two cursors is shown in the cursor window. So, the line regulation is,

$$\left(\frac{34.155516}{2.0059172}\right)\left(\frac{mV}{mA}\right) \approx 17 \left(\frac{mV}{mA}\right)$$

13. I-V characteristics of a MOSFET (DC sweep [.dc])

I.   **Open a new schematic** and **build the circuit** below using the processes outlined in steps 10(I) through 10(XIV). To insert an n-channel MOSFER, type or find **'nmos'** in the select component symbol window shown in step 5.
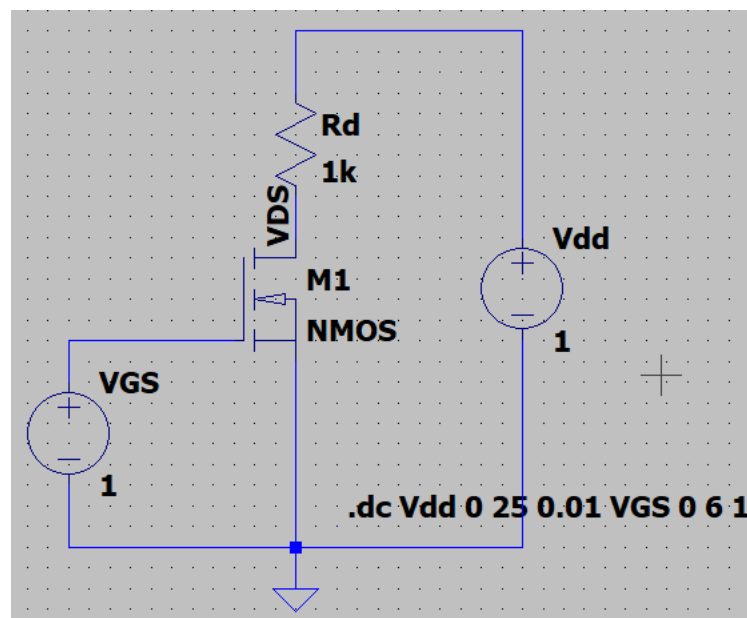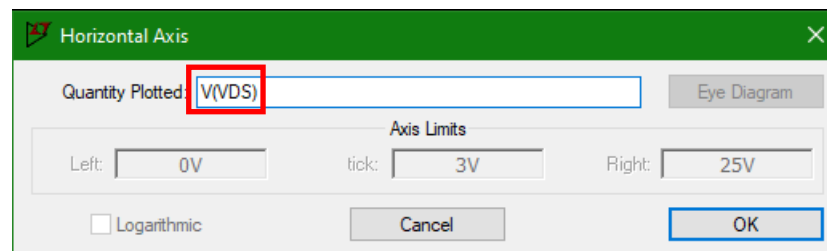

II.   Rename (9) the components as shown in the figure. Set 'Rd' as **1 kΩ**. Set the two voltages (VGS and Vdd) to any dc value for now. We will sweep the two sources in a while. Label the node '**VDS**'.
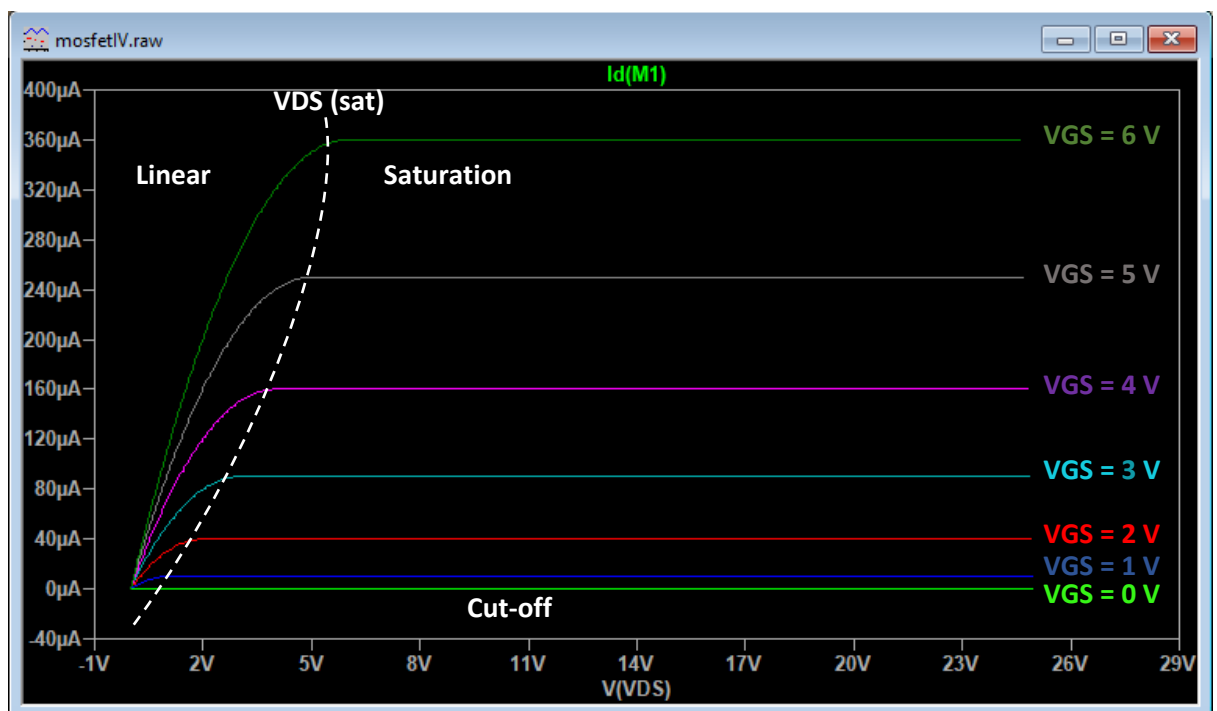
III.   Go to the **Edit Simulation Command** window following the procedure outlined in 10(III). In the 'DC sweep' tab, set the properties to sweep VGS and Vdd as shown in the figure below.
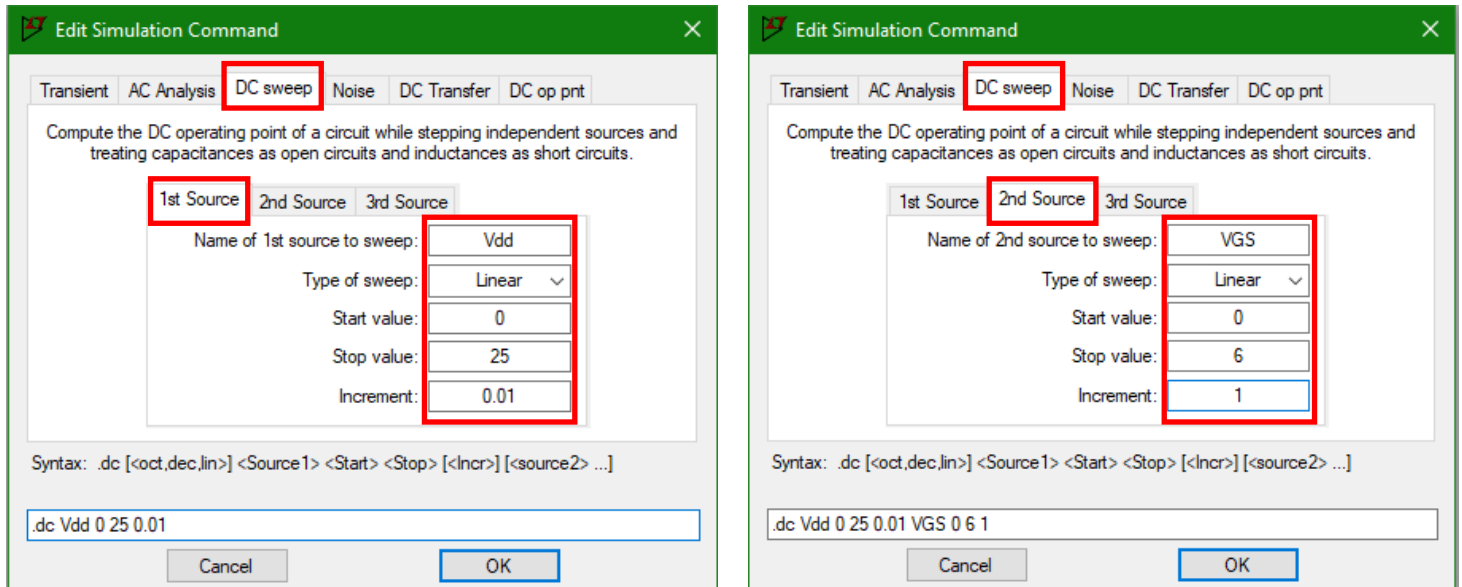
IV.   ***Run the simulation*** *[See 10(XVII) if necessary].*

V.   Note that *'Vdd'* is plotted along the horizontal axis. Change the horizontal attribute to ***'VDS'***.


VI.   Plot the current through the resistor (or MOSFET). *[See steps 10(XVIII) or 10(XIX) if necessary].* You should get a plot like this. Connect this to your theoretical understanding.
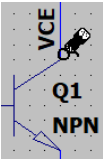
14. I-V characteristics of Bipolar Junction Transistor (BJT) (DC sweep [.dc]):

I. **Open a new schematic** and **build the circuit** below using the processes outlined in steps 10(I) through 10(XIV). To insert an n-channel MOSFER, type or find **'nmos'** in the select component symbol window shown in step 5.
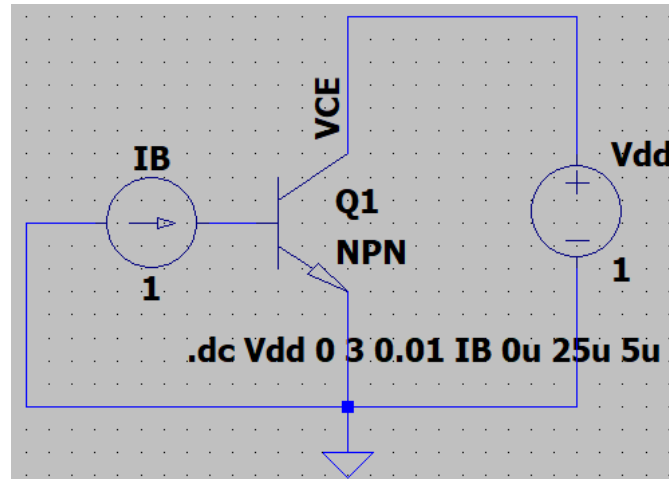

II. Rename (9) the components as shown in the figure. Set the two voltages (VGS and Vdd) to any dc value for the time being. We will sweep the two sources in a while. Label the node '**VCE**'.

III. Go to the **Edit Simulation Command** window following the procedure outlined in 10(III). In the 'DC sweep' tab, set the properties to sweep 'VCE' and 'IB' as shown in the figure below.
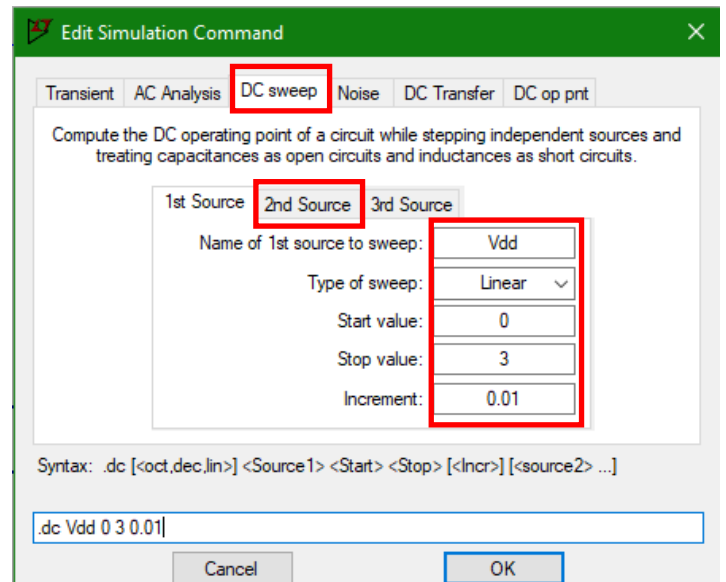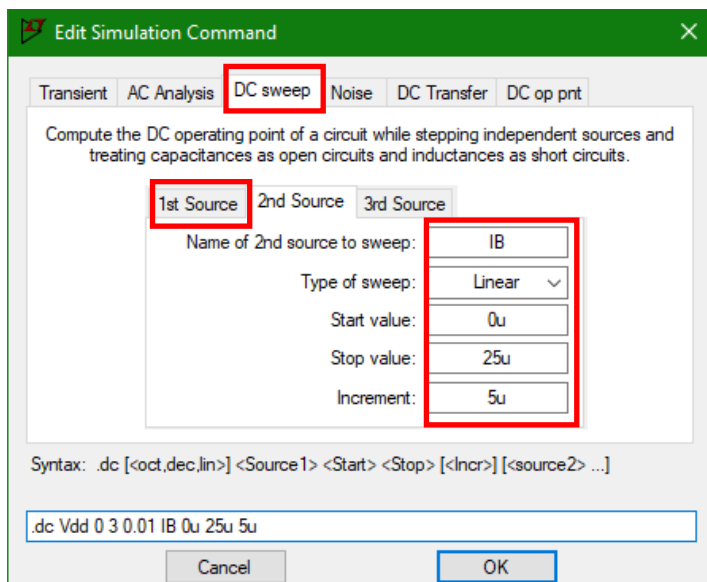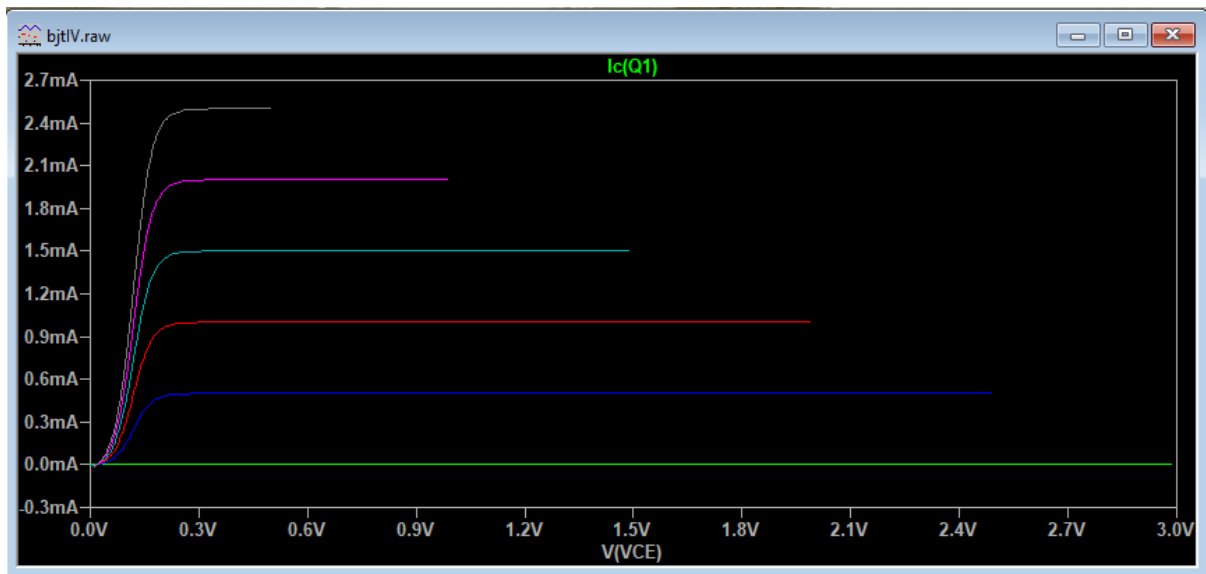
IV.   ***Run the simulation*** *[See 10(XVII) if necessary].*

V.   Note that **'Vdd'** is plotted along the horizontal axis. For this circuit diagram *Vdd = VCE.*

VI.   Plot the current through the collector to emitter of the BJT. To do this, **hover the cursor close to the collector terminal**. The cursor will change into You should get a plot like this. Connect this to your theoretical understanding.
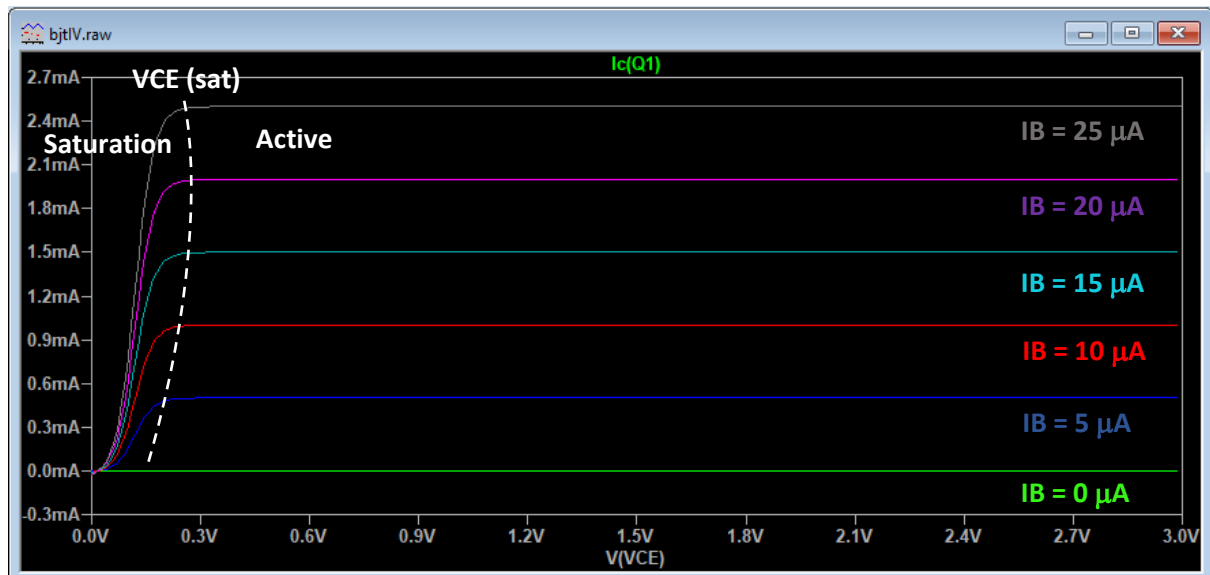
VII.   Now add a resistor of 1 kΩ in series with the collector terminal and simulate. Plot the collector to emitter current vs VCE. Don't forget to change the horizontal axis from 'Vdd' to 'VCE'.
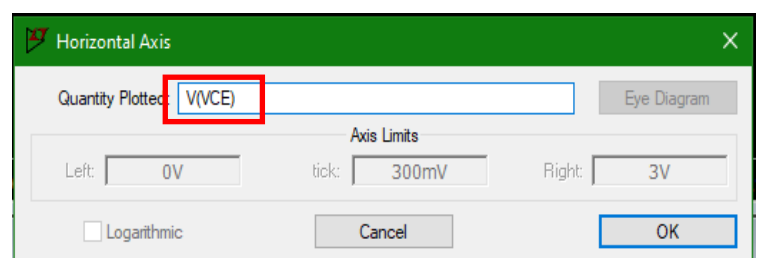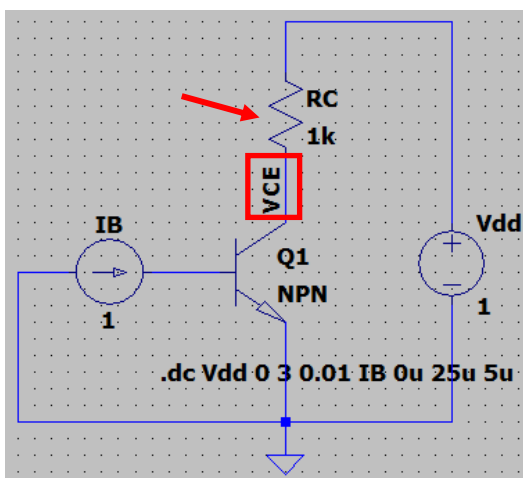
VIII.   A plot like the following one will be generated. Why does the collector current not reach the end of VCE? Think!

Prepared by,

PURBAYAN DAS