

A TUTORIAL ON LTSPICE:

DC OPERATING POINT (.OP), 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 CSE250 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 CSE251](#)

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](https://www.analog.com/en/design-center/evaluation-hardware-and-software/evaluation-embedded-software/analog-ltspice.html)

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.

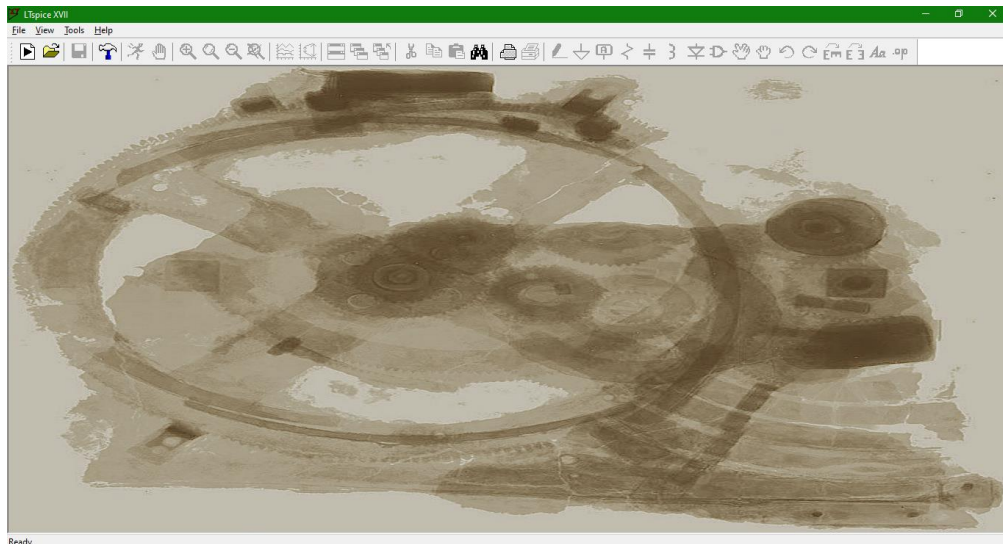
Mac:

- 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**

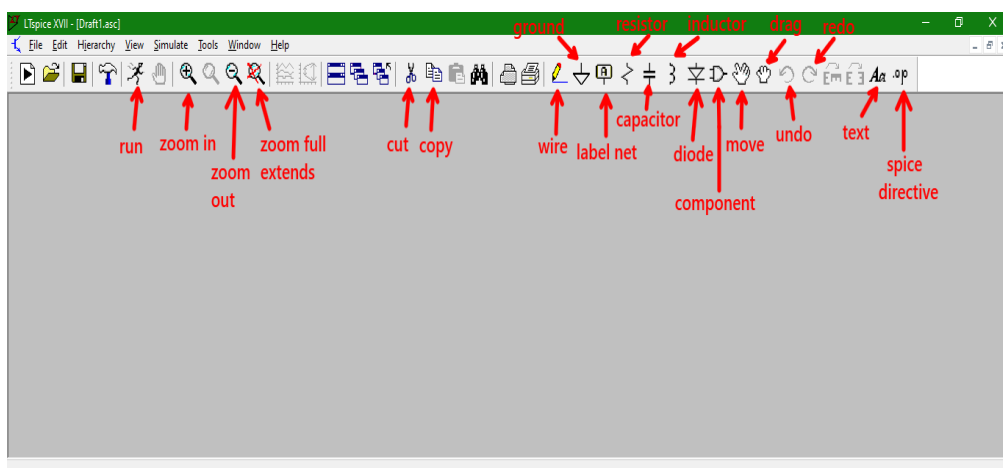
2. Opening LTSPICE:

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

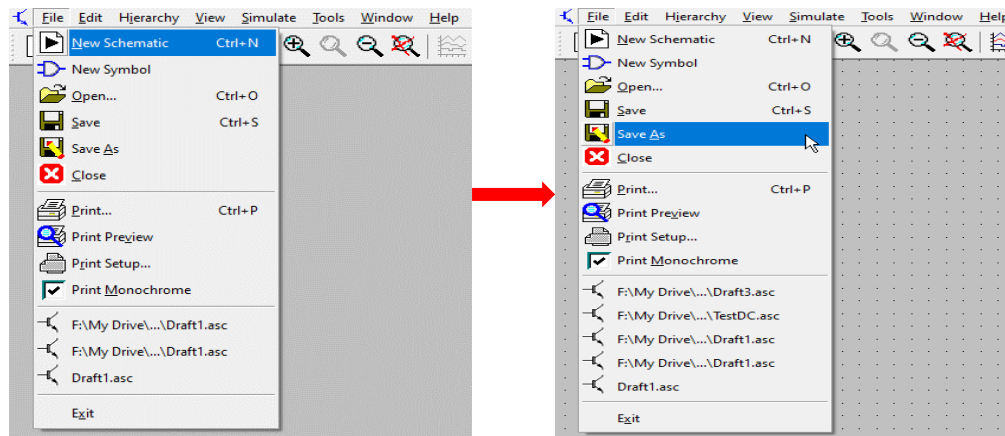
Mac: **Command + Space** → **Search for LTspice** → **Find LTspice** → **Click to open** → **Start a new, blank Schematic**.



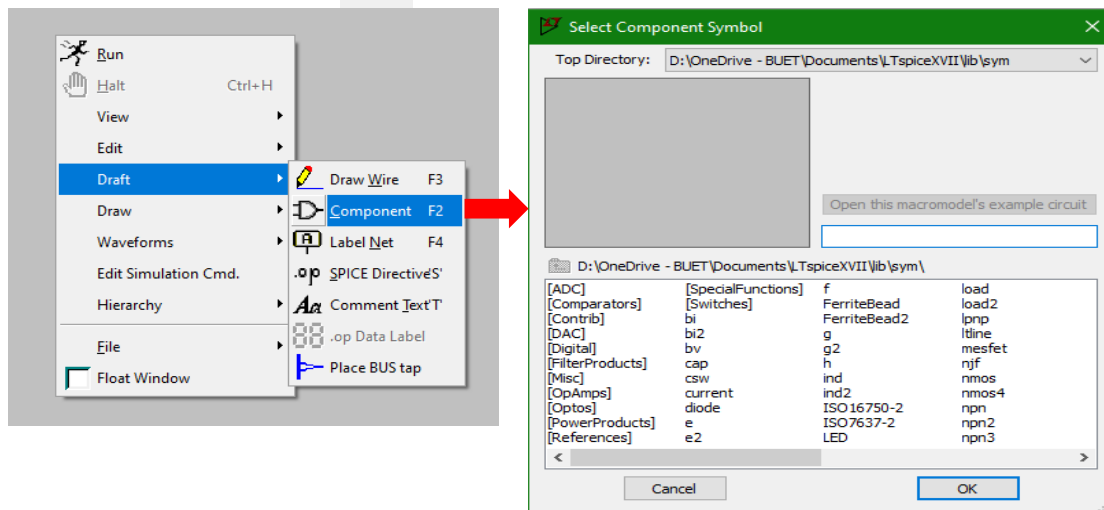
3. Get acquainted with the user interface.




4. Go to **File** → **New Schematic**. Save the file by **File** → **Save As** → **Name.acs**.

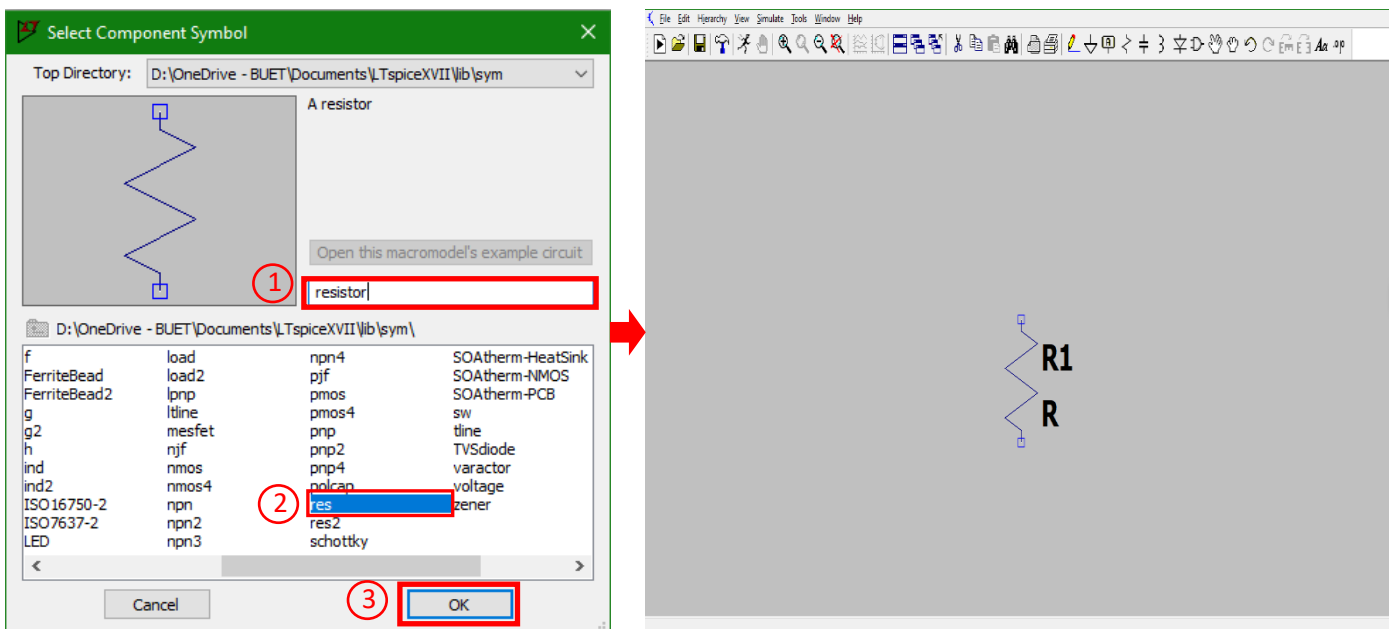


5. **Inserting a component:** Right-click on blank space → **Draft** → **Component** | Or, Press **F2** | Or, click this icon on the toolbar . A Select Component Symbol window will open like this.










[Click here](#) to see how to model dependent sources

- 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° if necessary. **Left click** on any suitable place in the grey interface to place it. **Press ESC** to deselect the component.



II. Keywords for common equipment:

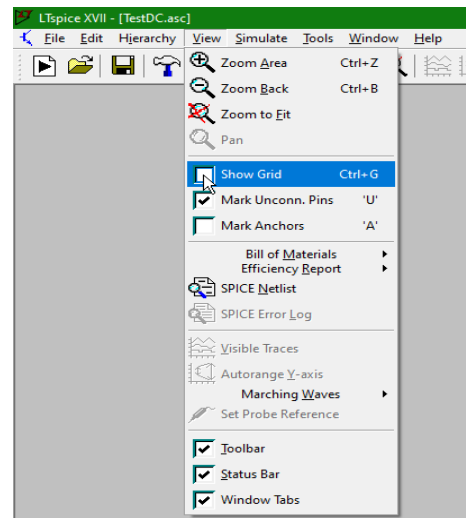
Component Name	LTspice Keyword
Independent Voltage Source	voltage
Independent Voltage Source	current
Resistor	resistor
Capacitor	cap
Inductor	ind
Diode	diode
Zener diode	zener
N-channel MOSFET	nmos
P-channel MOSFET	pmos
NPN BJT	npn
PNP BJT	pnp
VCVS	e or e2
CCCS	F
VCCS	g or g2
CCVS	h

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(s) will be attached to the cursor. *Left click* to place on a different position. The component/block 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(s) will be attached to the cursor. *Left click* to place on a different position. The component/block 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.*

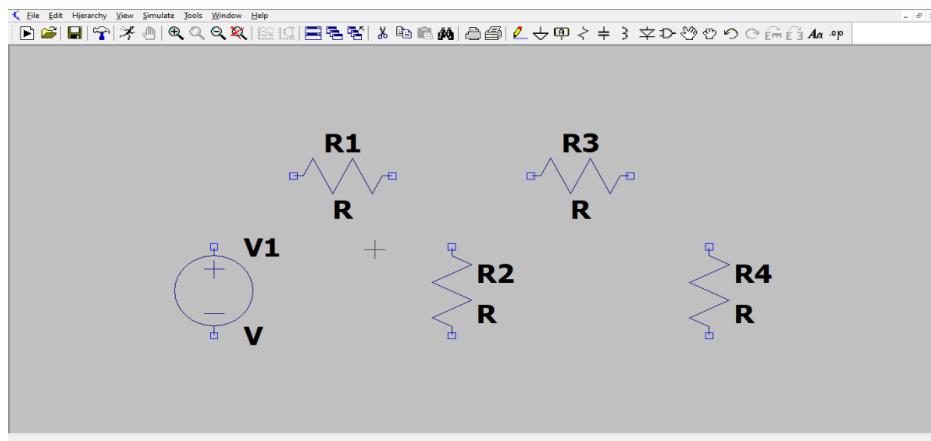
10. Simulating a DC circuit (.op and .tran):



- I. Create a new schematic file as instructed in step 4. Go to **File** → **Save as**. Rename the file as **TestDC.asc** and save it at a suitable place.

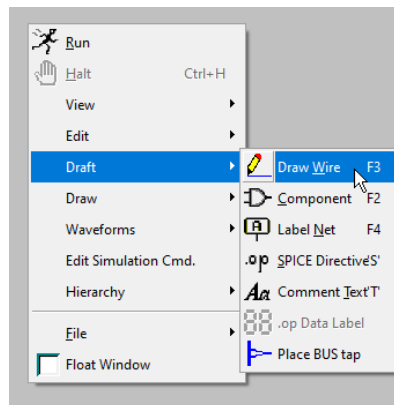
- II. Select **View** → **check the Show Grid** option for better visibility of the interface.



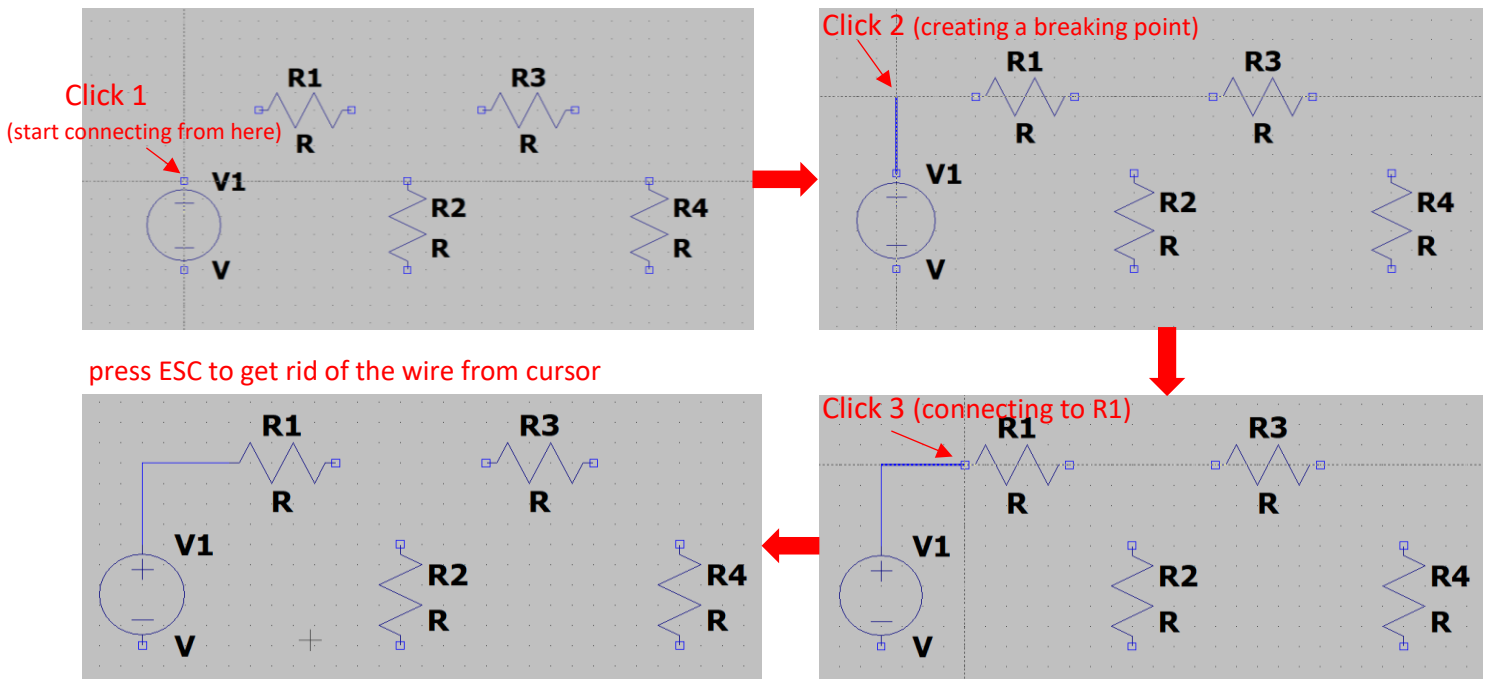
- III. Take resistors from component library as instructed in the step 5 and place them as shown in the following figure. Type or find '**voltage**' in the **Select Component Symbol** window in step 5 to insert a voltage source. Use **CTRL + R** to rotate a component and see step 6 and step 7 respectively to move or copy an element if necessary.



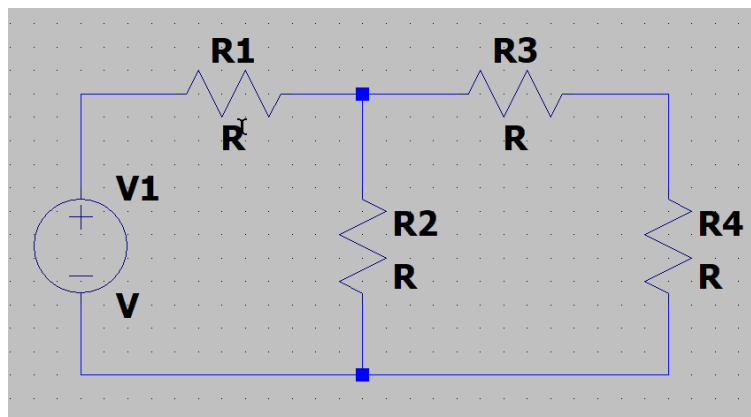
- IV. Now we need to connect the components using wire. **Right-click on blank space** → **Draft** → **Draw Wire** | Or, **Press F3** | Or, **click this icon on the toolbar** . The cursor will change into .





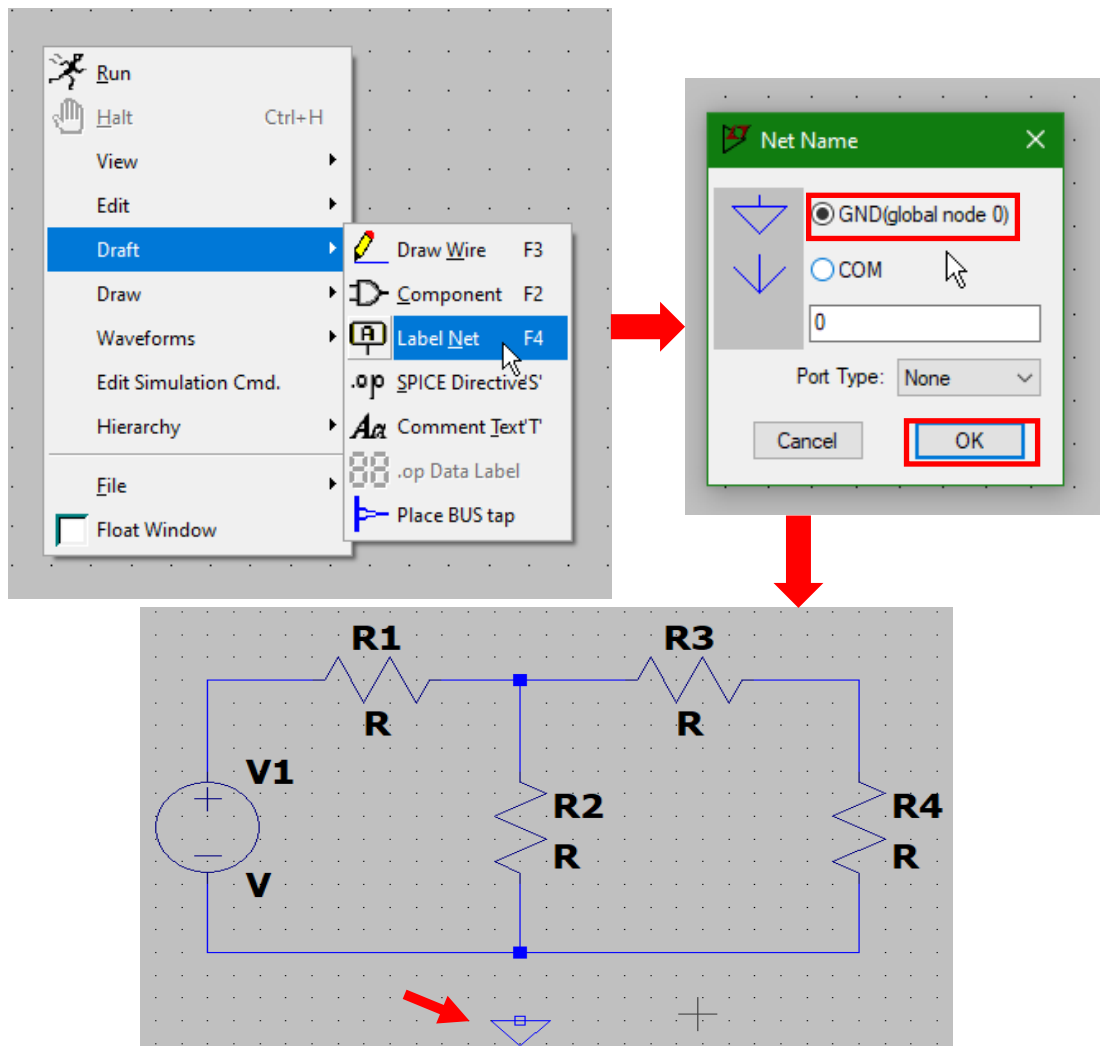
- V. **Click on one of the terminals** (small squares attached to a component) to start connecting. For example, shown in the figure below the steps in connecting the voltage source to the resistor R1.



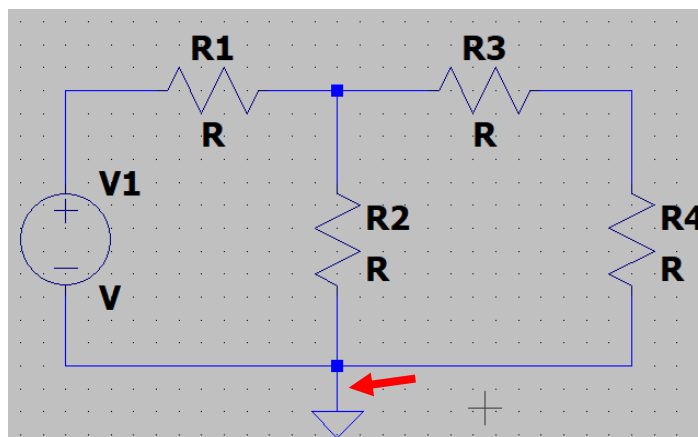
- VI. Similarly, wire all of the components together. The circuit will be as follows:



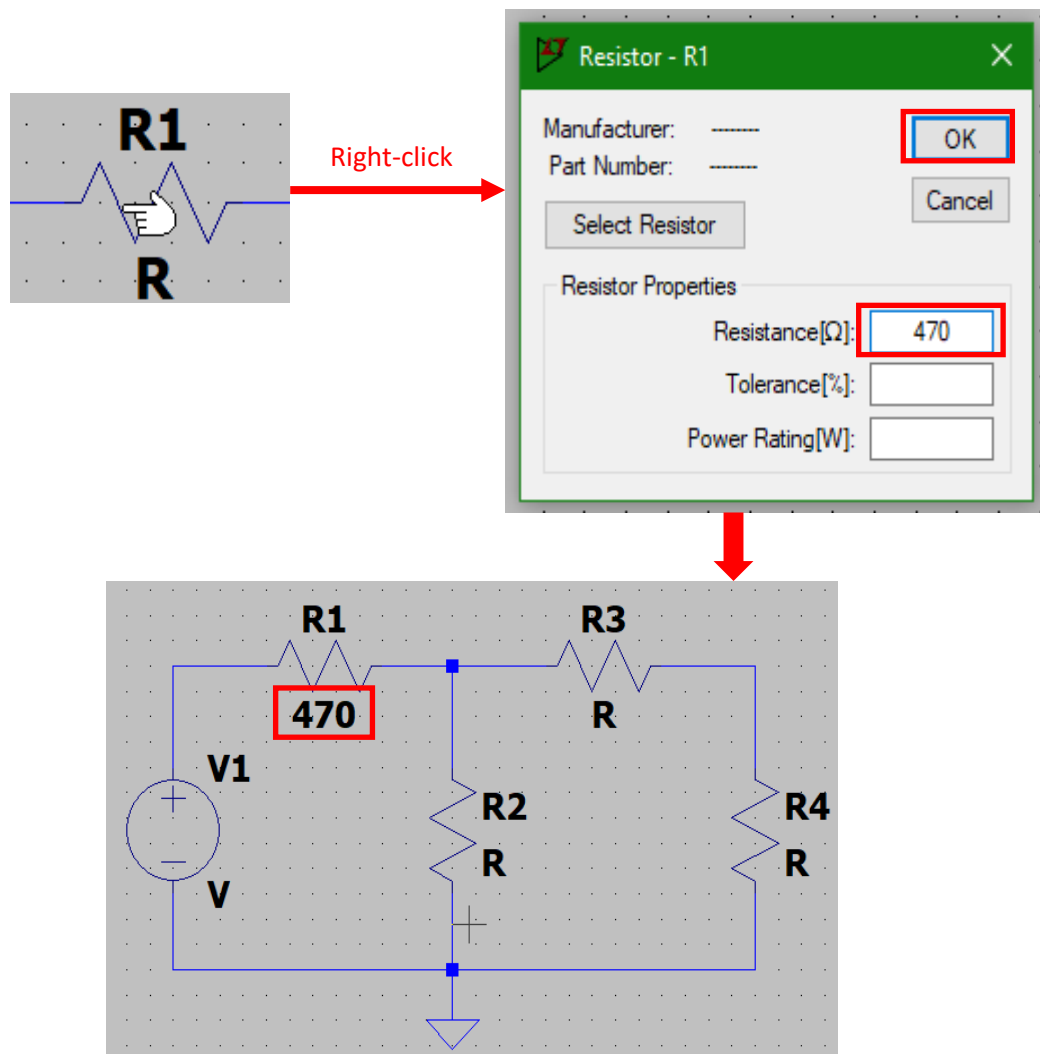
- VII. **Placing a ground:** Right-click on blank space → Draft → Label Net | Or, Press F4. A Net Name window will open. Select **GND (global node 0)** → click OK. Alternatively, click this icon on the toolbar . The cursor will change into . Place the ground symbol as shown in the following figure.



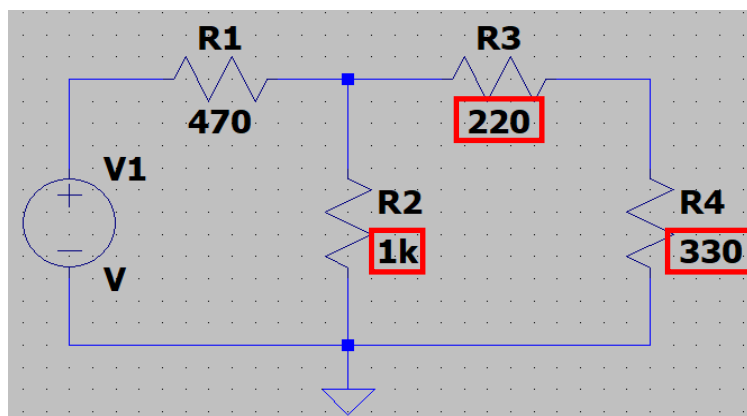
- VIII. Connect the ground to the circuit using wire as described in the steps [IV](#) and [V](#).



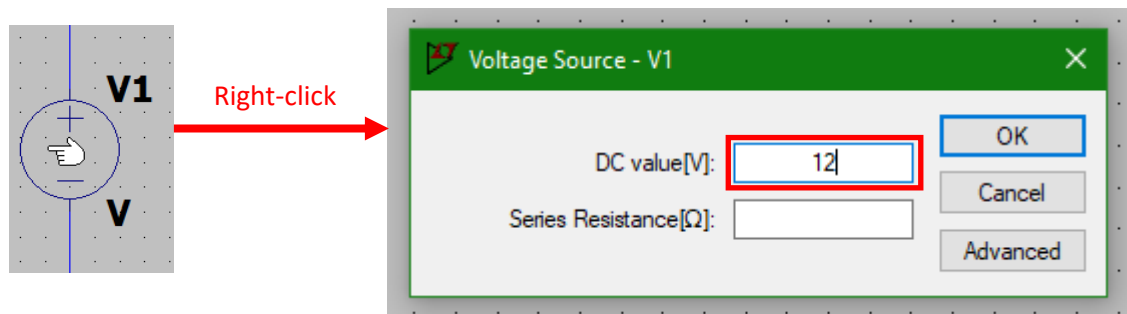
- IX. Now we need to set the value of the components. To set the resistance of a particular resistor (for example R1), **hover the cursor on R1**. The cursor will change into a hand. **Right click on it**. It will open up a new window for different specs of the resistor R1. Type 470 for **Resistance [Ω]**. Leave the other fields as it is and click OK. This will set R1 as 470 Ω.




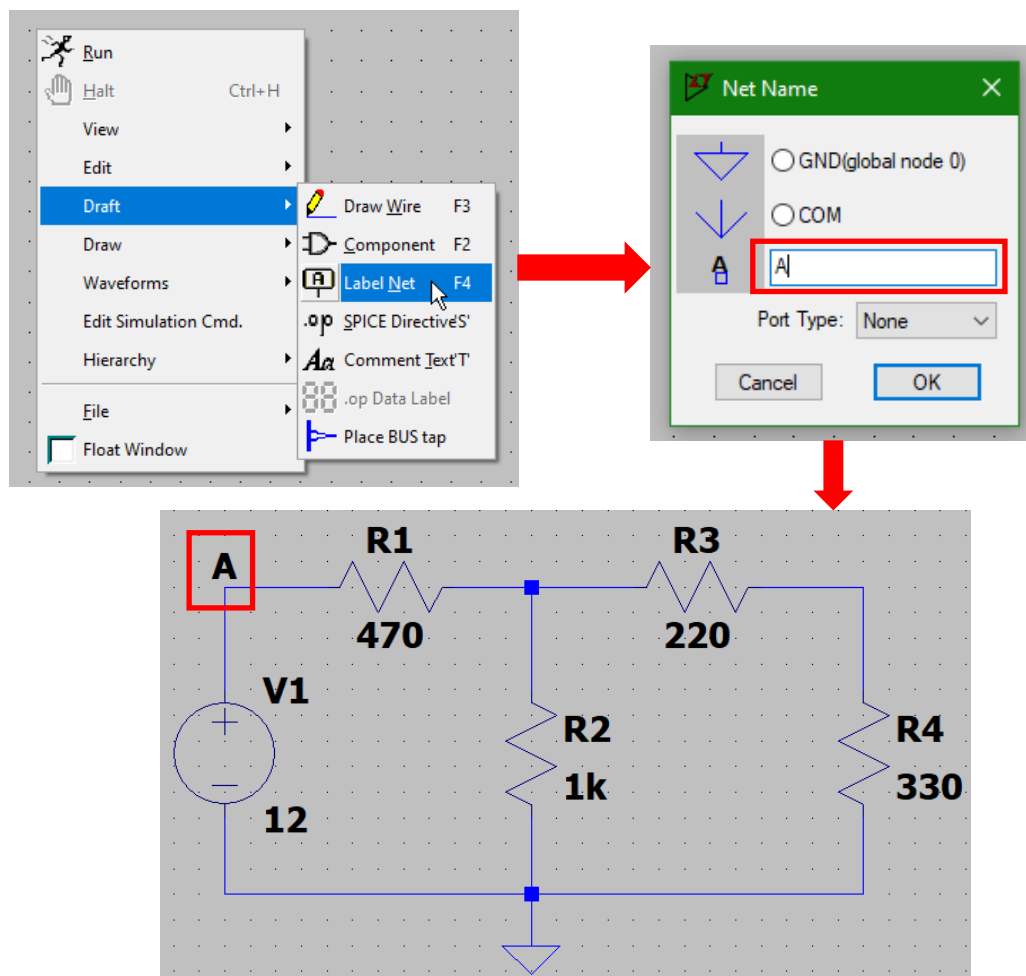
- X. In a similar way, set R2, R3, and R4 as 1 kΩ, 220 Ω, and 330 Ω respectively. For kilo ohm and mega ohm resistances, add 'k' and 'M' after the value. For example, '1k' for 1 kΩ resistance and '10M' for 10 MΩ resistance.



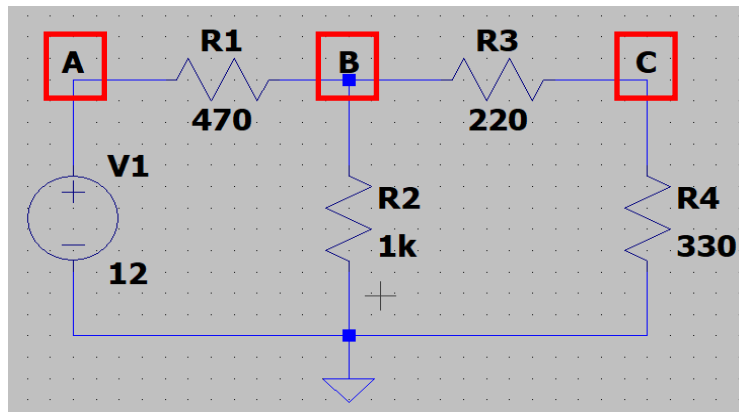
- XI. To set up the voltage source as dc, **hover the cursor over it → right click**. This will open a setting window for the voltage source. Type '12' in the DC Value [V] field and click ok. This will set the voltage source as 12 V DC. There are some advanced options for the voltage source. We will explore those later.





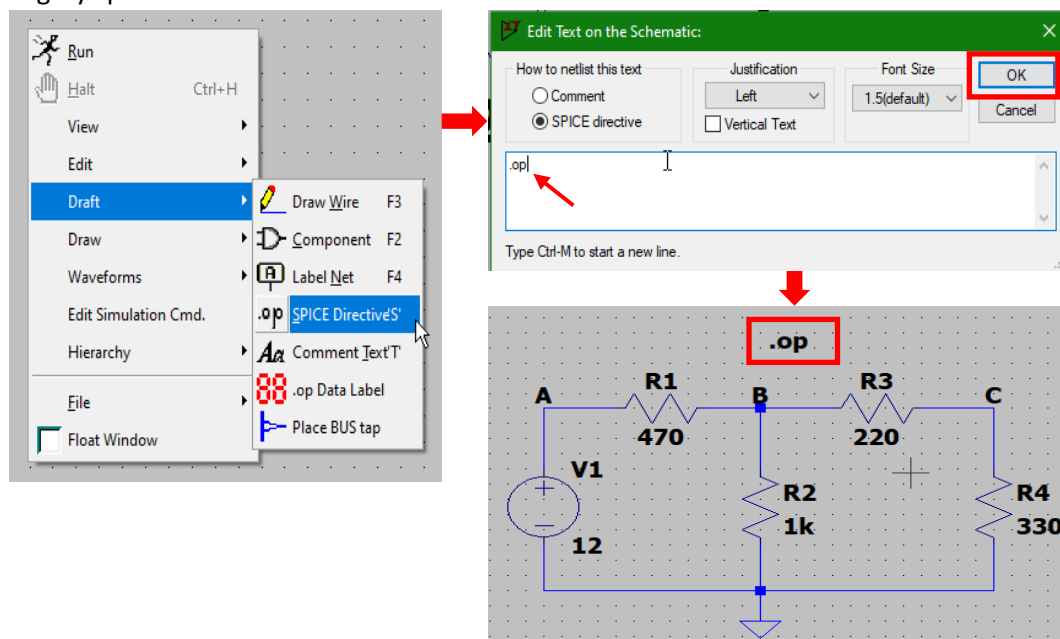
- XII. **Labelling nodes:** We will now label the circuit nodes. This helps in keeping track of voltages. To label a node, **Right-click on blank space → Draft → Label Net** | Or, **Press F4** | Or, **click this icon on the toolbar** . A **Net Name** window will open. Type 'A' in the box and click OK. Place the label A as shown in the following figure. Then V_A denotes the potential difference between node A and ground, i.e., $V_A = V_{\text{Input}} = 12 \text{ V}$.




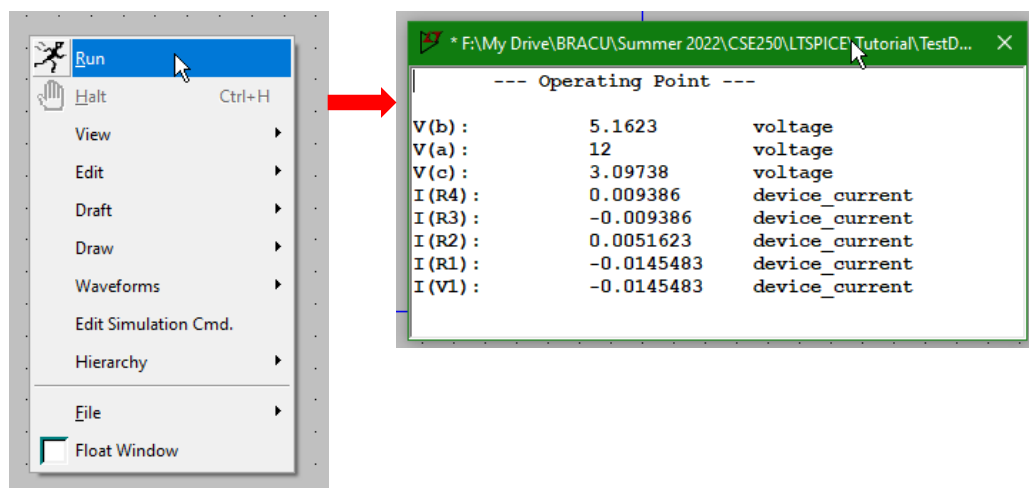
- XIII. In a similar way, label the remaining nodes as shown in the following figure. **The node variables are case-insensitive.**



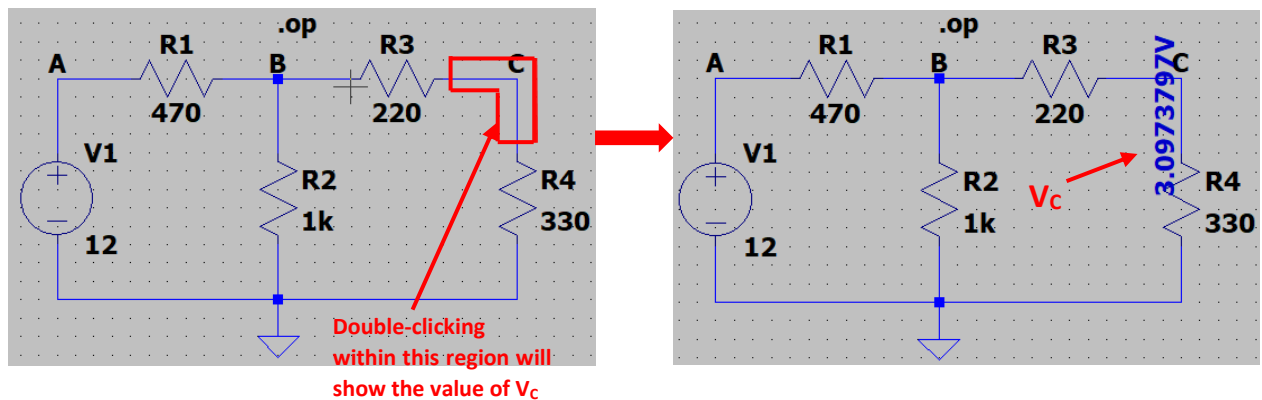
- XIV. **Running the simulation:** Now we are all set to run the circuit to observe the dc voltages and currents. **Right-click on blank space** → **Draft** → **SPICE Directive's** | Or, **click this icon on the toolbar** . This opens a netlist window. Make sure **SPICE Directive** is selected. Write **'.op'** in the text box and click OK. The cursor will change into . Place the **'.op'** text anywhere in the grey space.



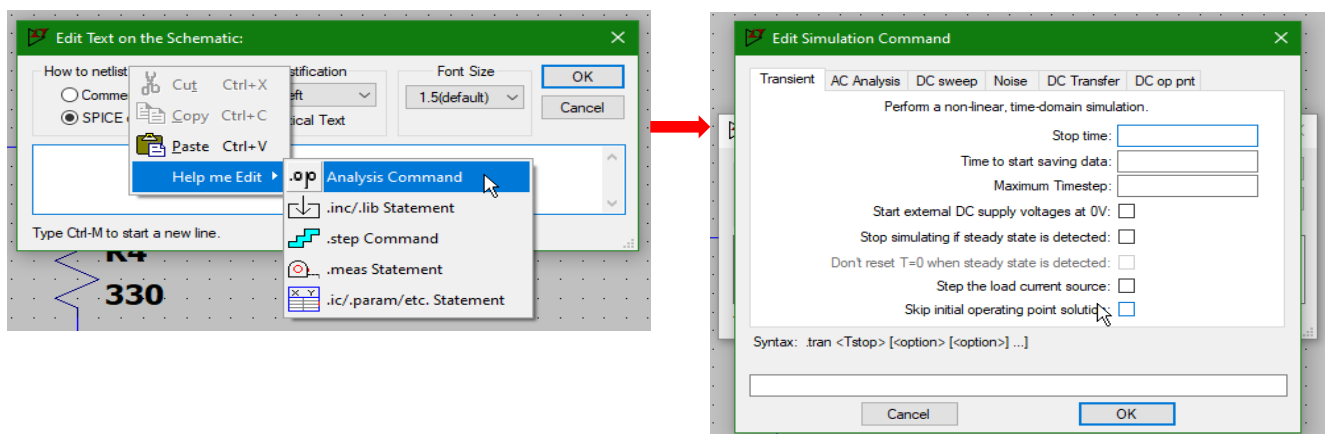
- XV. **Right-click on blank space** → **Run** | Or, **click this icon on the toolbar**  to run the simulation. This opens a new window, where the values of the node voltages and currents through the circuit elements are shown.



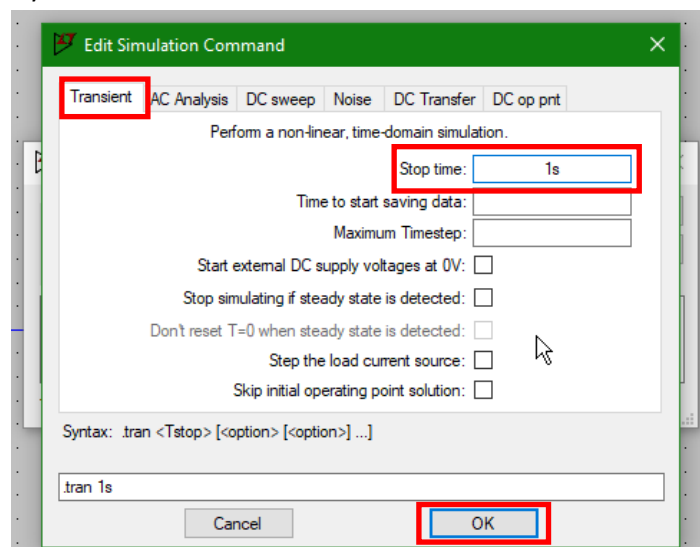
- XVI. Alternatively, the node voltages can also be seen by **double clicking** on any node (on the wires connected to a particular node) as shown in the following figure.



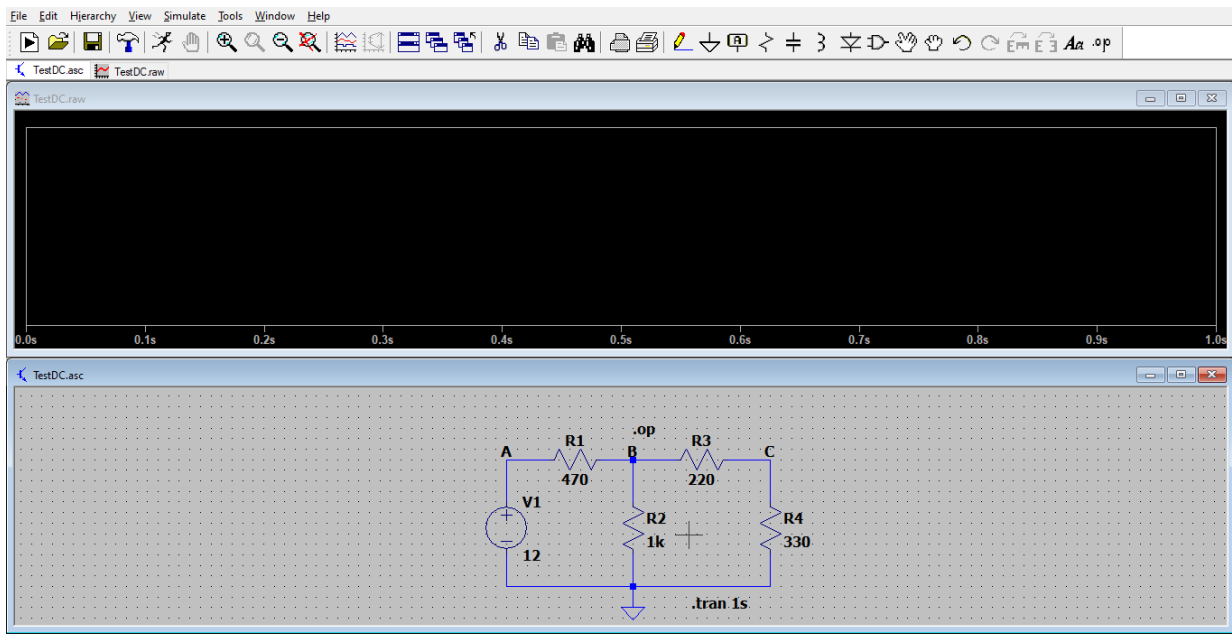
- XVII. However, to see the voltage across an element (for example the voltage across the R3 resistor) transient analysis must be performed. Open the netlist window again following the procedure in the step XIV. **Right click on the black text box → Help me Edit → Analysis Command.** This opens the 'Edit Simulation Command' window.



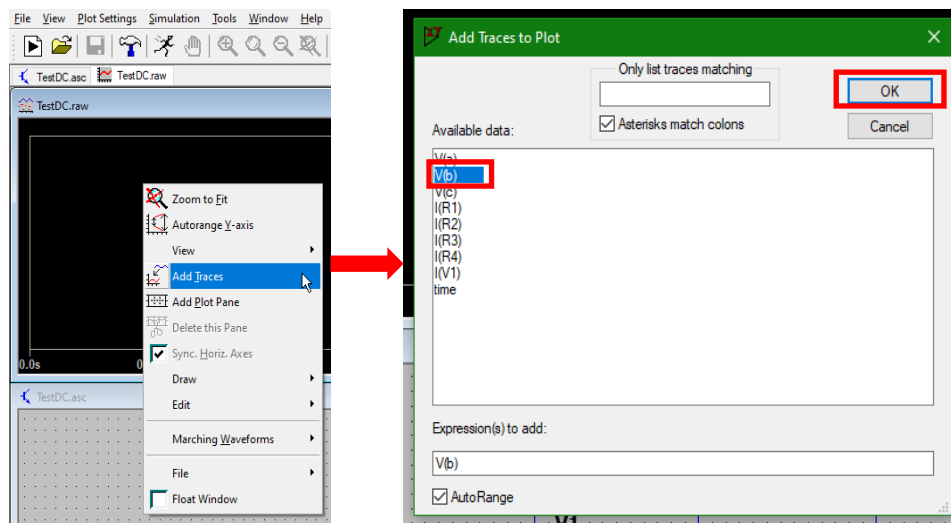
- XVIII. In the transient tab, write '**1s**' in the **Stop time** field. The simulation will run for 1s. Leave the other fields untouched for now. Hit OK. The cursor will change into **tran 1s** place the '**tran 1s**' text anywhere.





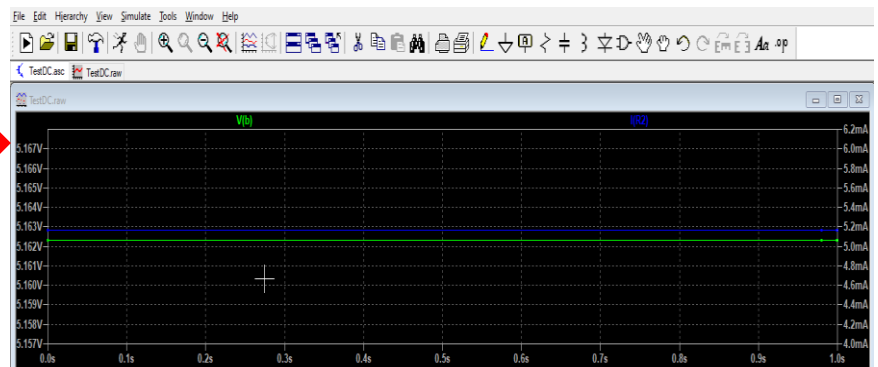
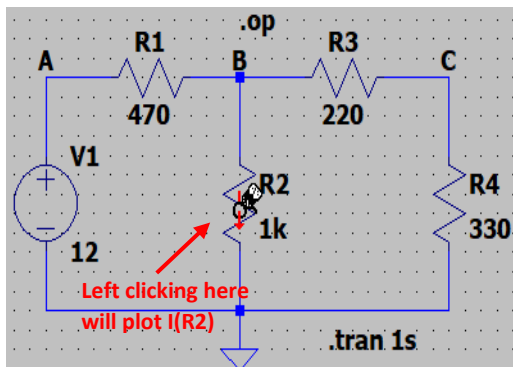
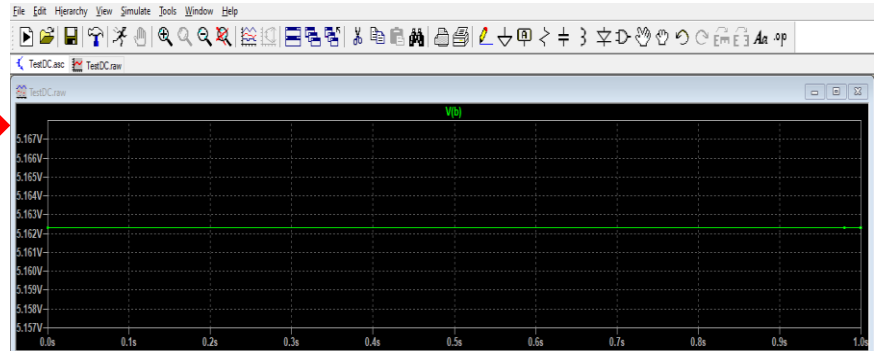
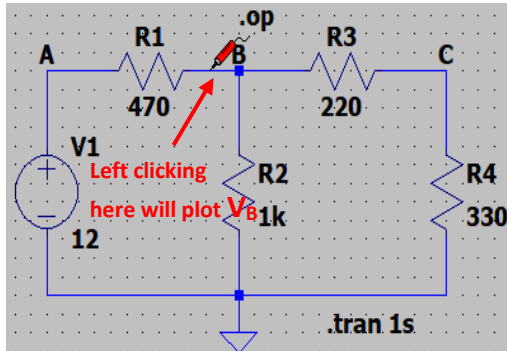
- XIX.** *Run the simulation* again following step [XV](#). This will lead to a new window as shown below. Note that the horizontal axis of the plot window has a duration up to the stop time we set.



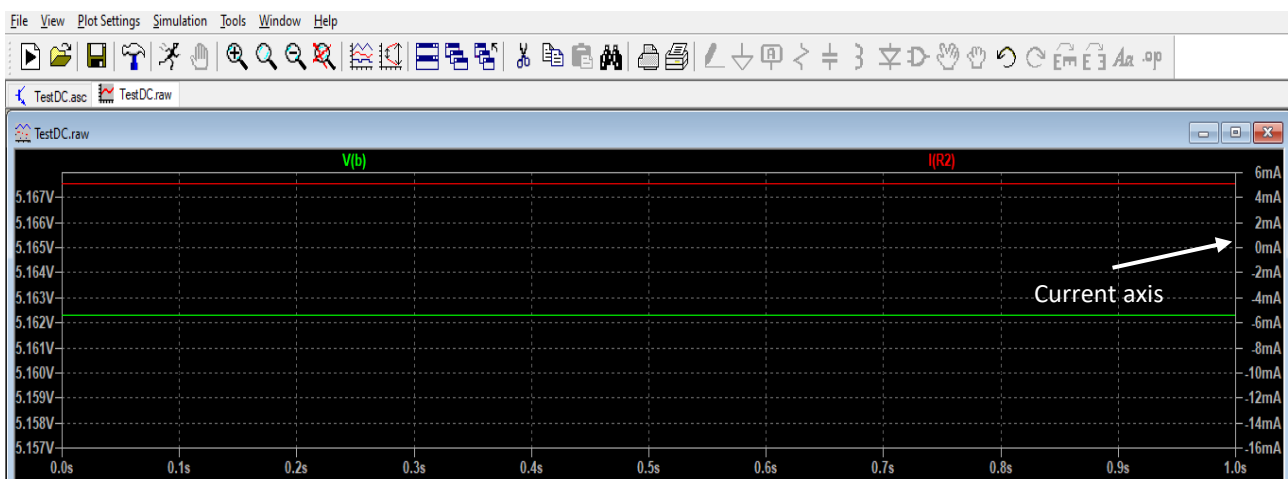
- XX.** To plot a voltage or current, **Right click on the black area** → **Add Traces**. This will open a window where all the node voltages and the elemental currents are listed. **Select the one you want to plot** → **click OK**. The following figure shows the plot of V_B . **Right click on the plot window** → **View** → **Check the Grid**.



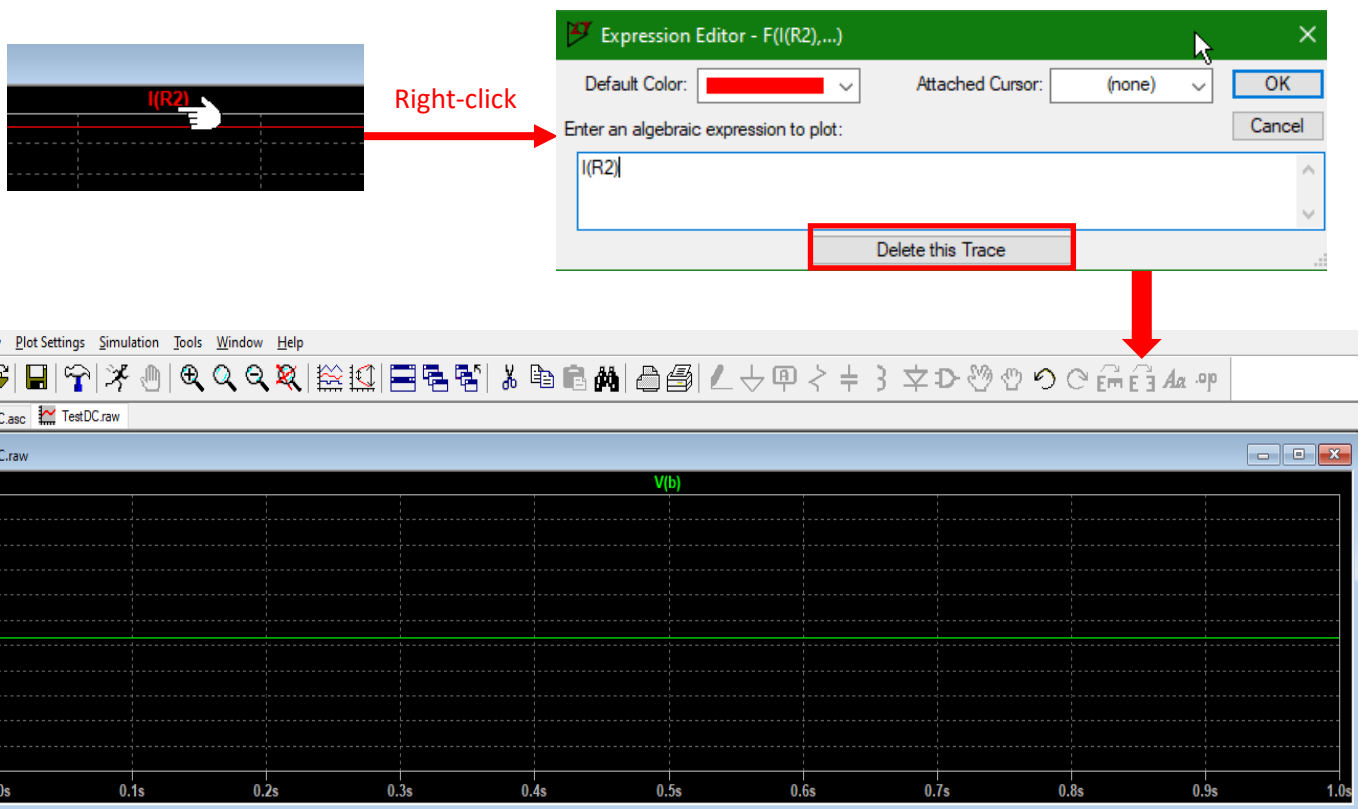
XXI. Alternately, you can plot voltage or current by choosing the circuit window and moving the cursor over any node to view the voltage at that node or over any element to view the current flowing through that element. The cursor will change into and respectively  for  node voltage and branch current. Left clicking will plot the particular current/voltage.



XXII. Multiple voltages or currents can be plotted in the same plot. For example, using the same approach as in steps [XX](#) or [XXI](#), we can plot the current through the resistor R2 together with V_B . Note that the current axis is on the right.



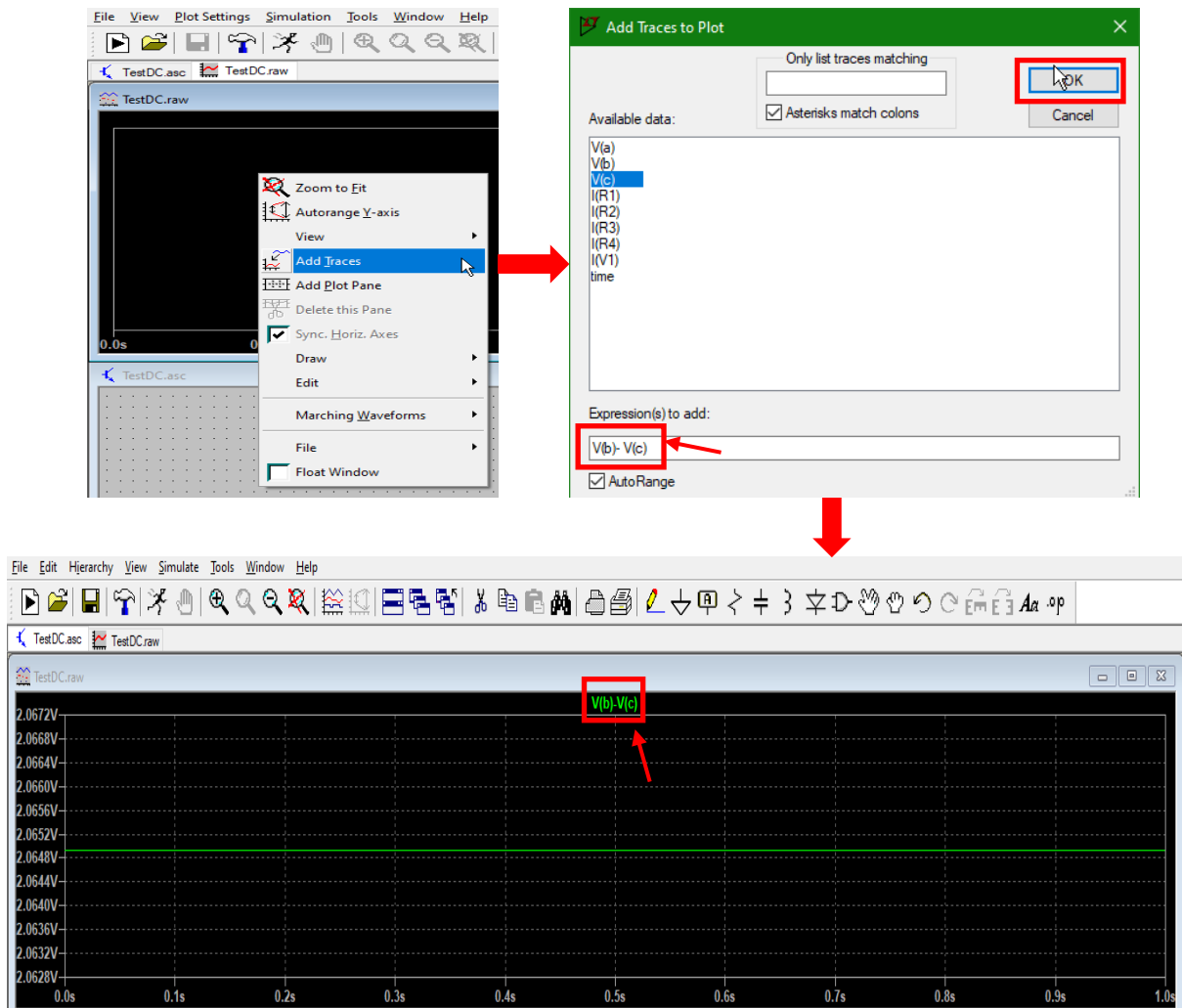
XXIII. To delete a particular trace, **hover the cursor on the parameter** → **Right click** → **Delete this Trace**.



XXIV. 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 [XX](#) or [XXI](#). To delete a specific pane, **Right-click it** → **Delete this Pane**.




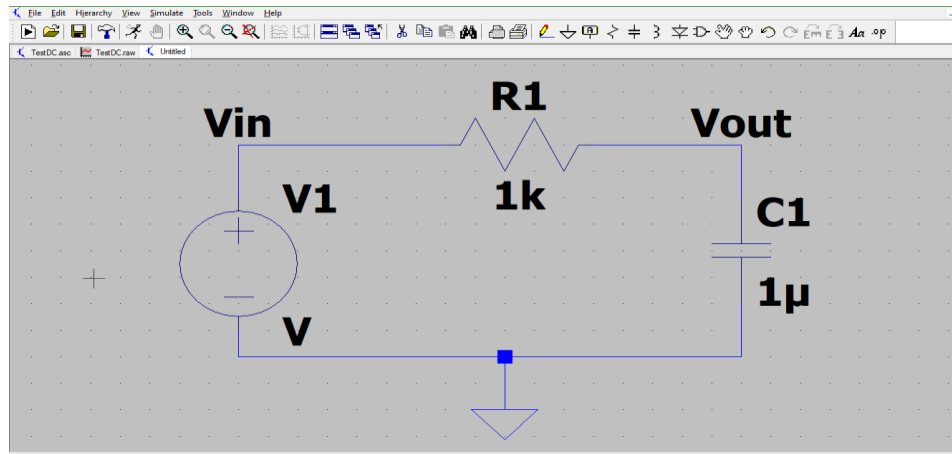
- XXV. The plots can be saved for future use and analysis by **selecting the plot window** → **File** → **Save Plot Settings As** → **Name.plt**.
- XXVI. **Voltage across an element**: To see the potential across R3, **Right click on the black area** → **Add trace** → write $V(b) - V(c)$ in the 'Expression(s) to add field' of the Trace adding window and **hit OK**.



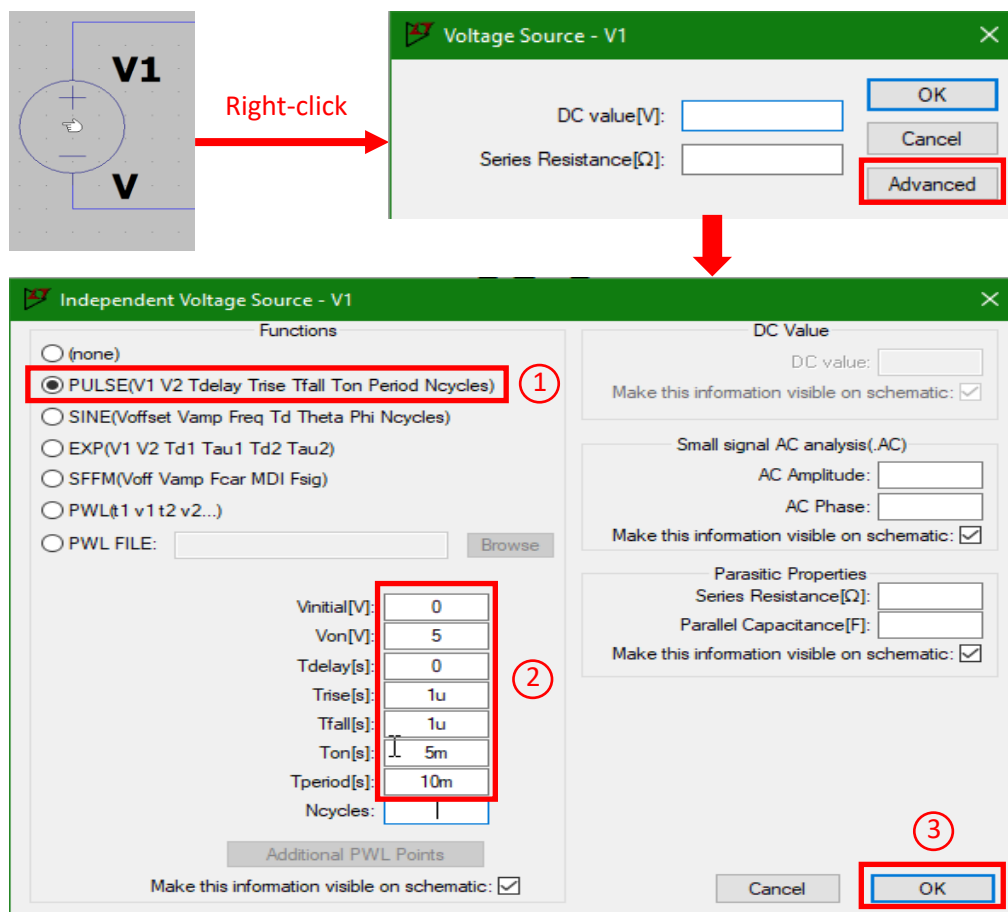
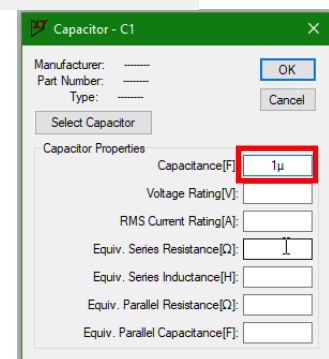
- XXVII. Explore more plot window features, such as dragging a rectangle to zoom into a specific area, changing trace colours by right clicking on the parameter label, and left clicking on the parameter label to enable the data pointer to see the coordinates, right clicking on axis to change the axis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

11. Transient analysis of a first order RC circuit [.tran]: We shall now study the transient response of a RC circuit with step input.

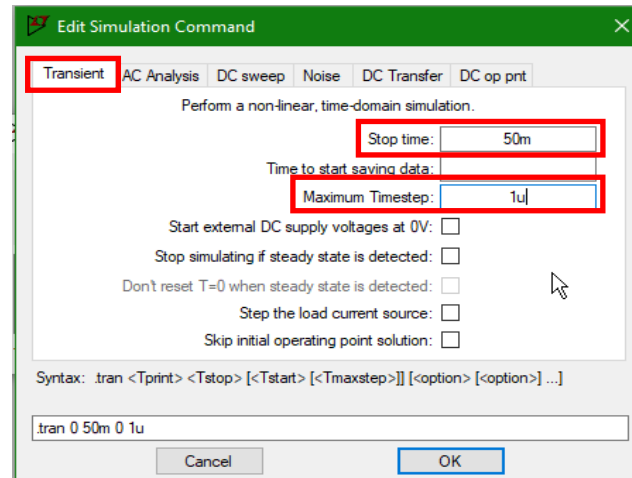
- I. **Open a new schematic** and **build the circuit** below using the processes outlined in steps [10\(I\)](#) through [10\(X\)](#). **Label the nodes** as shown in the figure. To insert a capacitor, type or find 'cap' in the select component symbol window in step [5](#). Alternatively, select this icon on the toolbar .



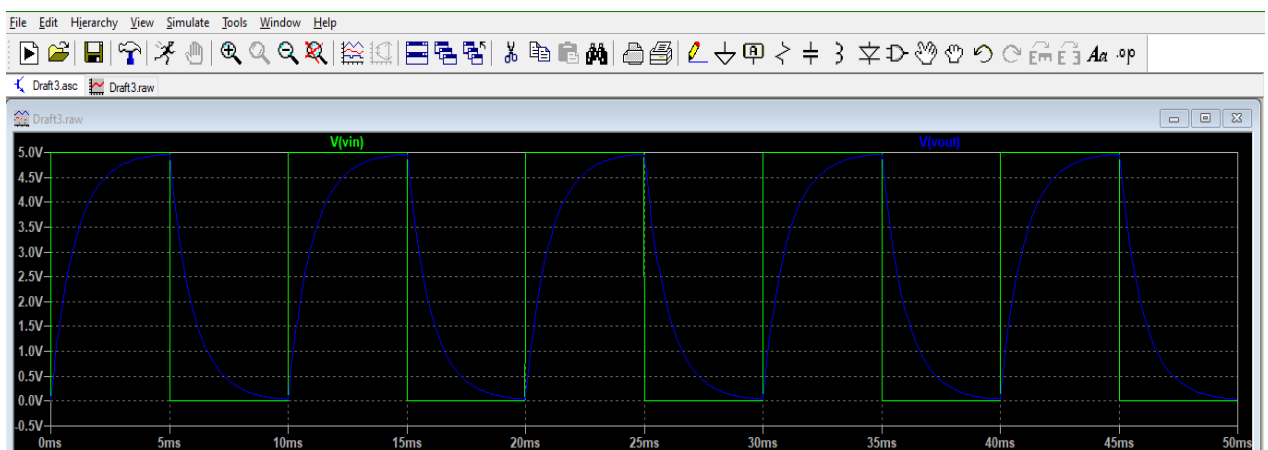
- II. Set the resistance to 1 kΩ and the capacitance to 1 μF. Leave the remaining fields unchanged.
- III. A pulsating dc will be applied to the RC circuit to study its transient behaviour. Go to the properties of the voltage source by **Right clicking on it** → **Advanced**. This will launch the property editor, as illustrated below. **Configure** the properties as seen in the next illustration.



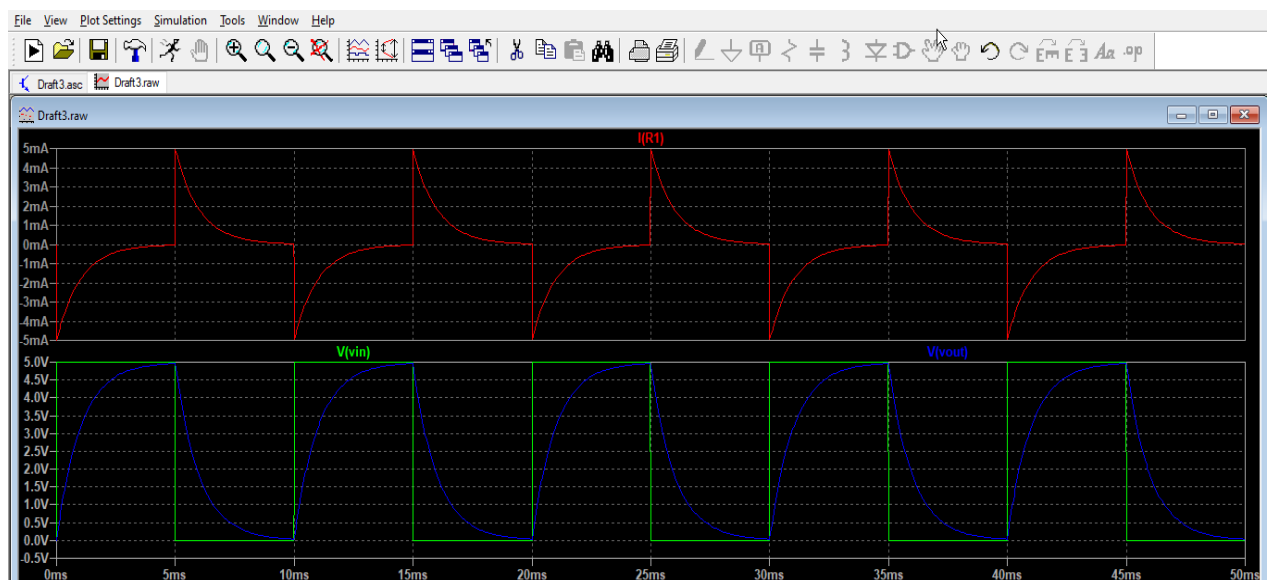
- IV. Go to 'Edit Simulation Command' window following the instructions in step [10\(XVII\)](#). In the transient tab, set the **Stop time** as 50 ms with a **Maximum Timestep** 1 μ s. Leave the remaining fields unchanged.




- V. **Run the simulation** [See the step [10\(XV\)](#) if necessary].
- VI. **Add Vin and Vout traces together in the same plot pane** [See the steps [10\(XX\)](#) through [10\(XXII\)](#) to add traces to a pane]. You should get a plot like this. With the applied voltage on and off, respectively, observe how the capacitor is storing and supplying energy.

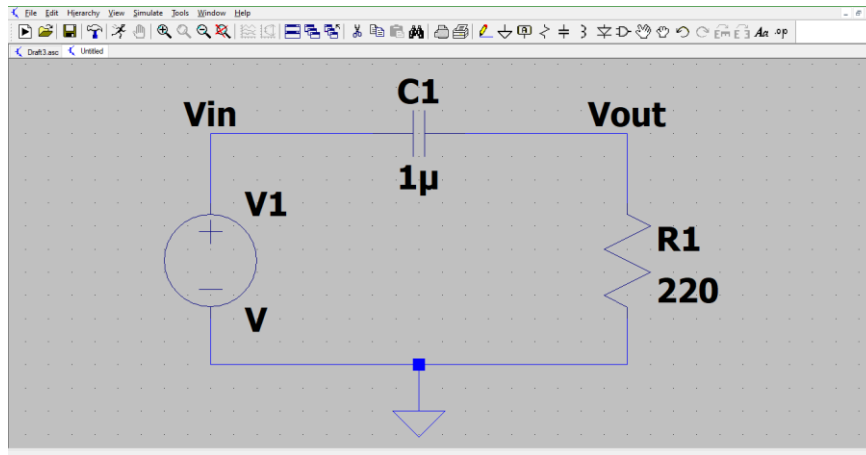


- VII. **Add a separate pane to the plot window.** Plot the current of the series circuit [See the step [10\(XXIV\)](#) if necessary]. Observe the current through the resistor/capacitor. Relate with the theoretical knowledge you've gained.

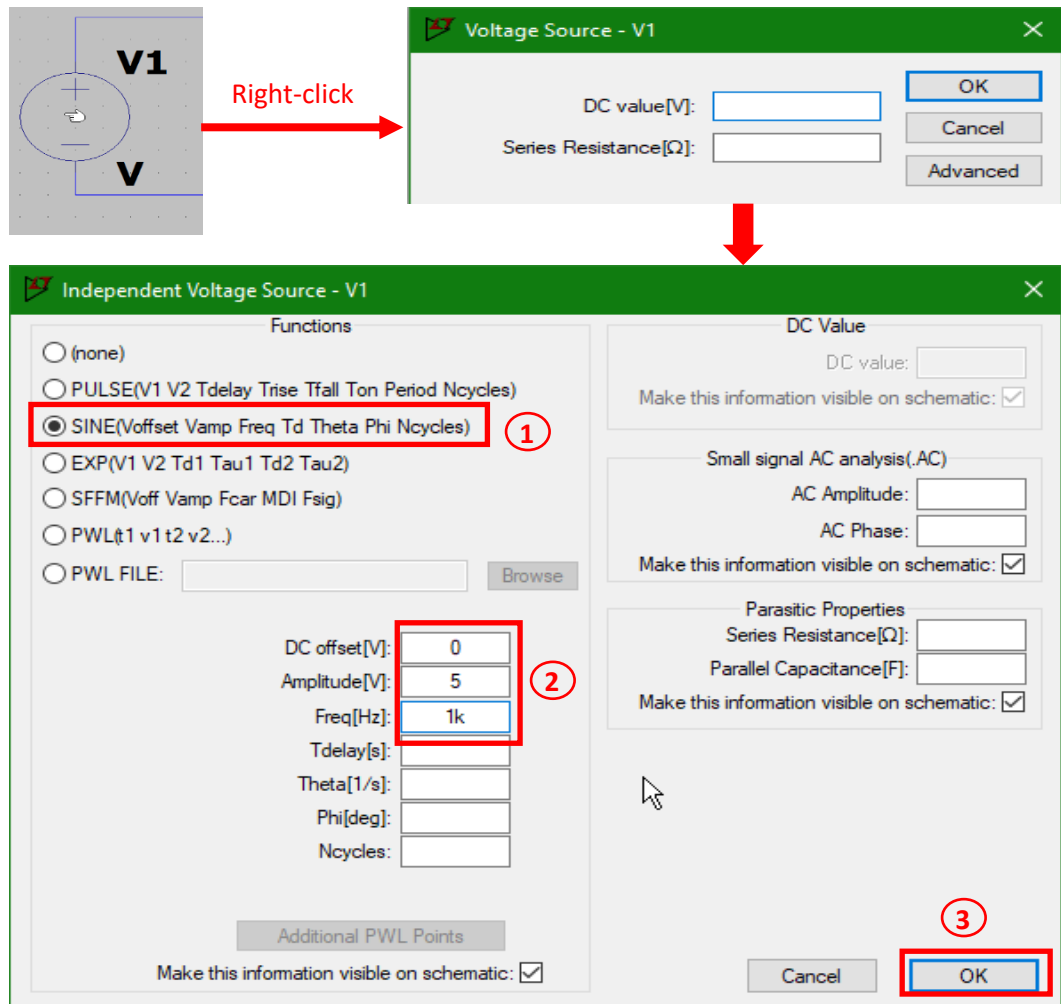


12. Simulating AC circuit [.tran]: We shall now study alternating current circuits and the aspects of sinusoidal waveforms.

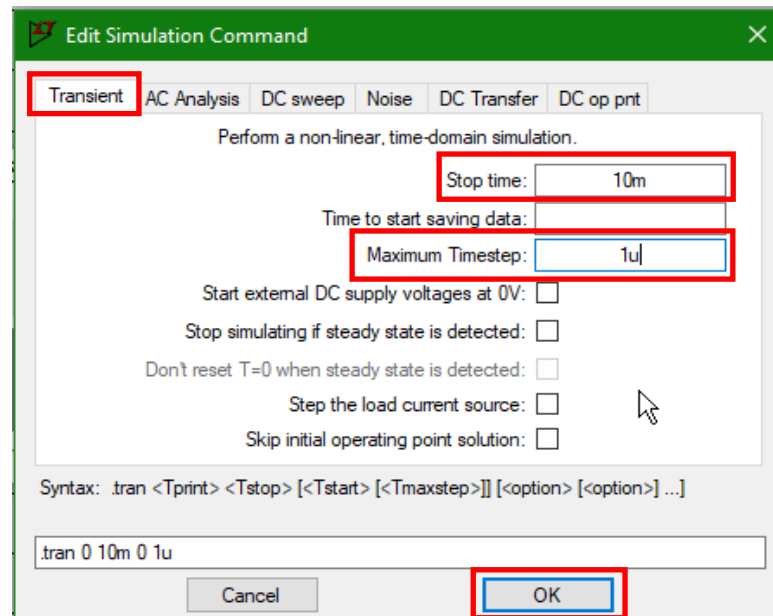
- I. **Open a new schematic** and **build the circuit** below using the processes outlined in the steps [10\(I\)](#) through [10\(X\)](#). **Label the nodes** as shown in the figure. To insert a capacitor, type or find 'cap' in the select component symbol window in step 5. Alternatively, select this icon on the toolbar . **Set** the resistance to 1 k Ω , and the capacitance to 1 μ F.



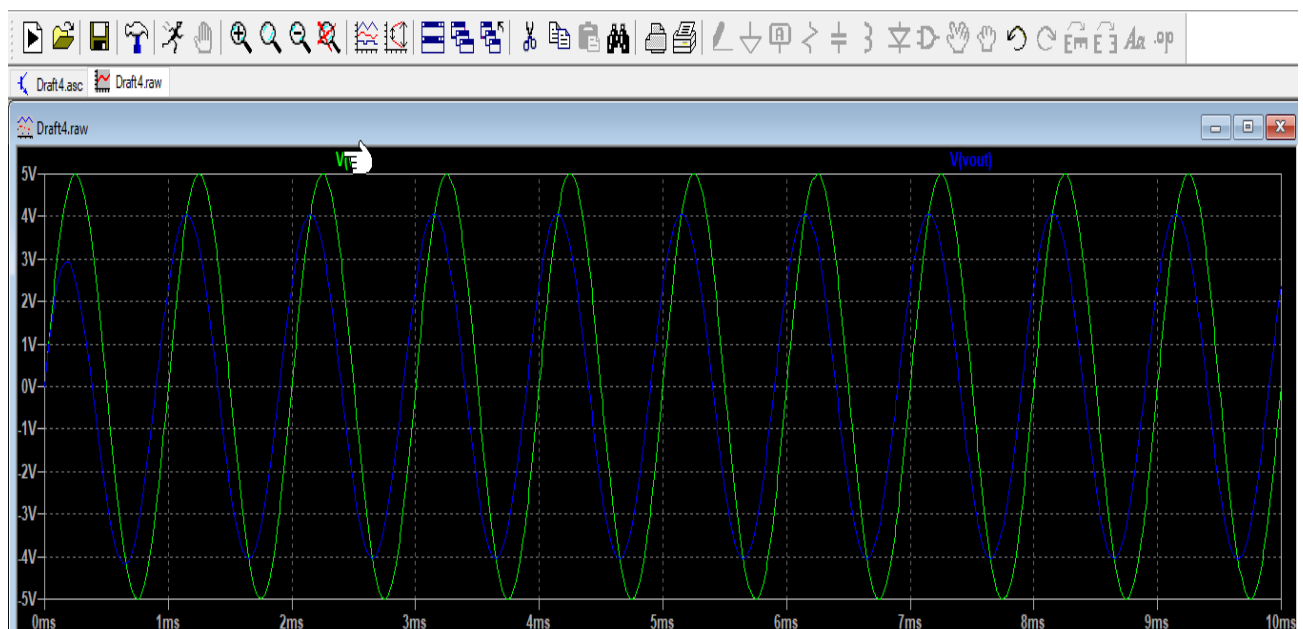
- II. The circuit will be subjected to a sinusoidal voltage. Go to the properties of the voltage source by **Right clicking on it** \rightarrow **Advanced**. This will open the property editor window. **Set** the voltage source to be a **10 V p-p 1 kHz** sinusoidal wave with as shown in the following figure.



- III. Open the **'Edit Simulation Command'** window following the instructions in the step [10\(XVII\)](#). In the transient tab, set the **Stop time** as 10 ms with a **Maximum Timestep** 1 μ s. Leave the remaining fields unchanged.

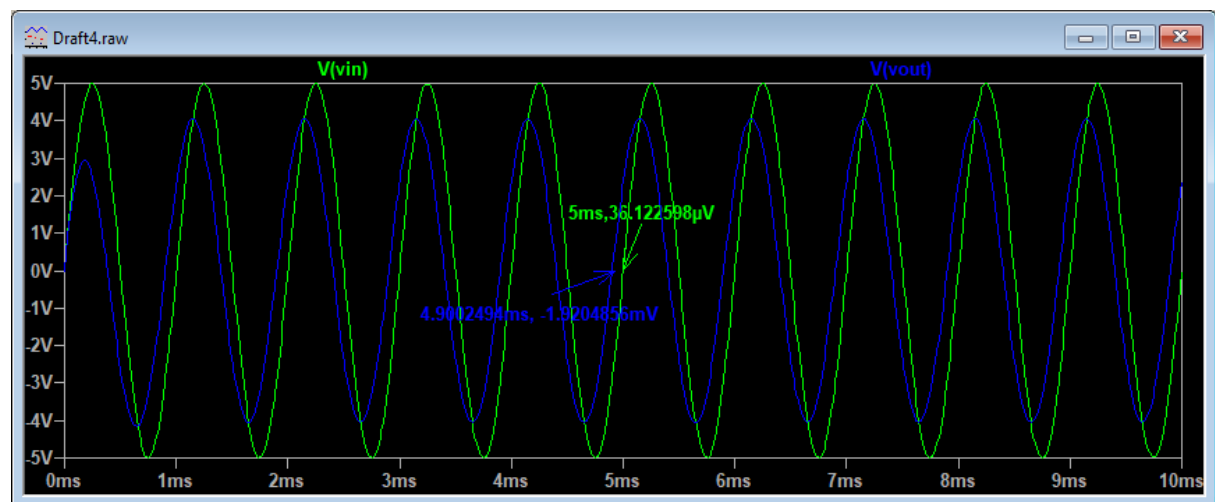


- IV. **Run the simulation** [See step [10\(XV\)](#) if necessary].
- V. **Add 'Vin' and 'Vout' traces together in the same plot pane** [See steps [10\(XX\)](#) through [10\(XXII\)](#) to add traces to a pane]. You should get a plot like this. Observe the phase difference between the input and the output waveforms.



VI. **Measuring phase difference:** Using data cursor, we will now calculate the phase difference from the graphic. The phase difference is the angle between any two points on the two waveforms that are in the same phase. A cursor for a particular trace will appear by clicking on the label of that trace. The arrow keys on the keyboard can be used to move the cursor's data point.

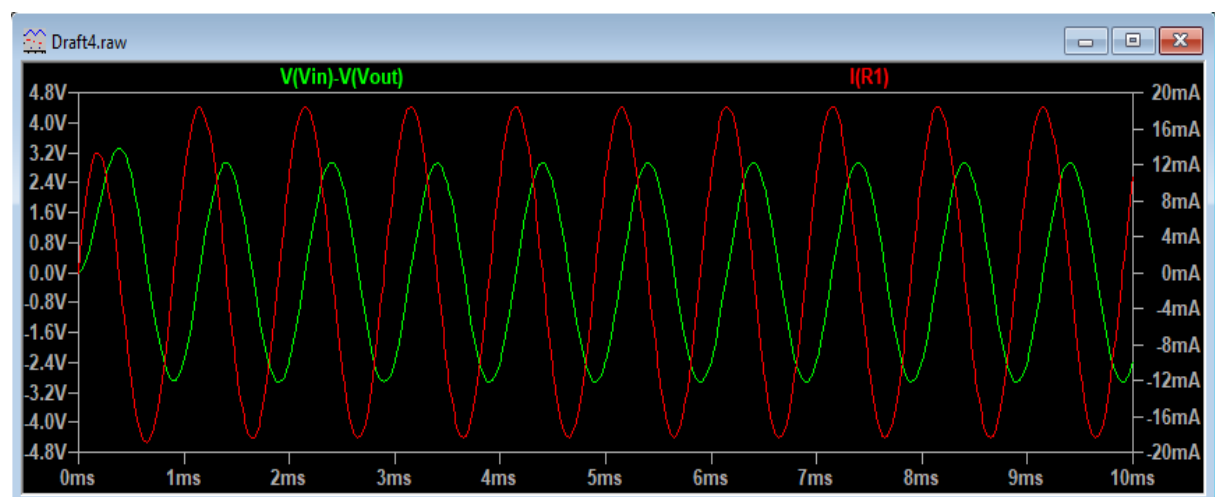
- **Left click on the trace label 'V(vin)'** to enable its cursor. Use keyboard arrows or left click and drag the cursor to one of the zero crossings. **Right click on the plot → Draw → Cursor Position.** Notice that the data point is pinned on the plot. Right click on the 'pinned data' to modify its properties.
- Similarly, **Left click on the trace label 'V(vout)'** to enable its cursor. Position the cursor to the immediately adjacent zero crossing point of 'Vout'. These are the points on same phase. **Right click on the plot → Draw → Cursor Position.** Notice that the data point is pinned on the plot. Right click on the 'pinned data' to modify its properties.



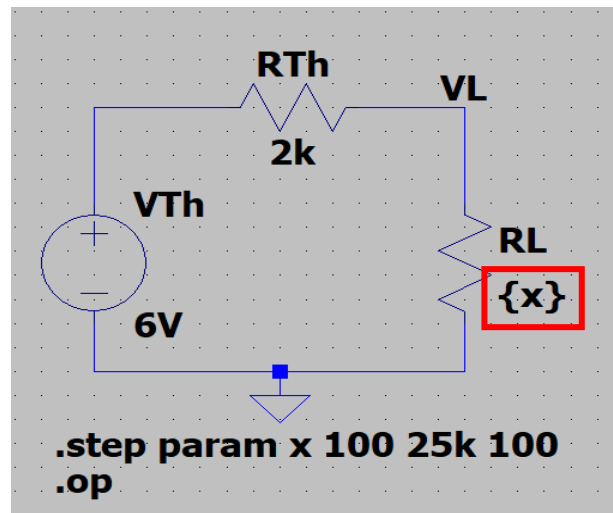
- The phase difference can be calculated using the relation,


$$\begin{aligned}\theta &= 360 \times f \times t_{diff} \\ &\approx 360 \times 1 \text{ (kHz)} \times (5 - 4.9002494) \text{ (ms)} \\ &\approx 36^\circ\end{aligned}$$

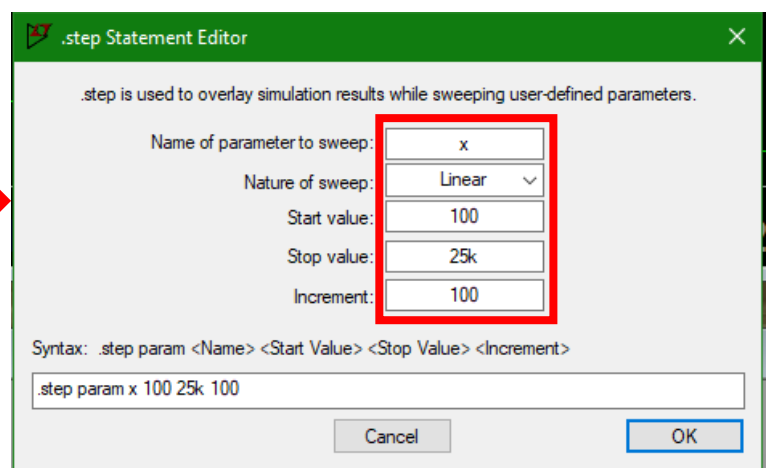
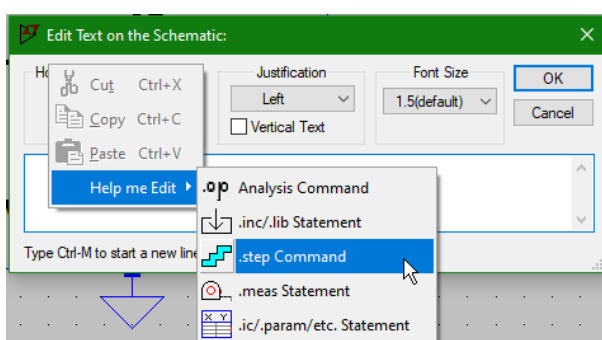
- **Plot**, in a separate pane, the voltage across the capacitor and current through the series circuit.




13. **Parametric sweep (.step param):** We shall now study how to sweep a parameter using Step Command in LTspice.



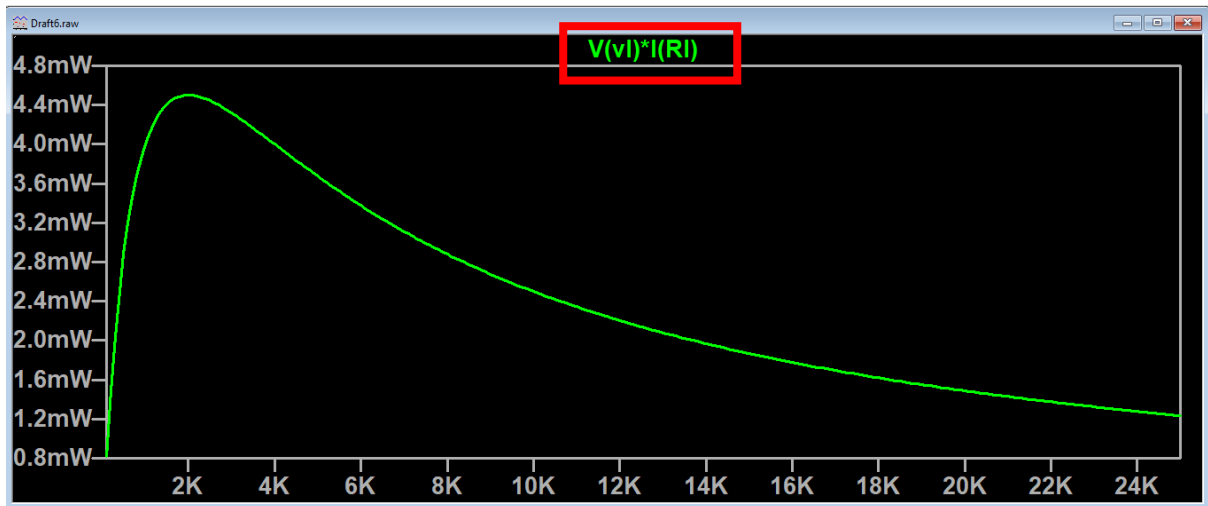
- I. We will sweep the load resistance of a Thevenin equivalent circuit to observe how the load power varies with respect to the load resistance. **Open a new schematic** and **build the circuit** below using the processes outlined in the steps [10\(I\)](#) through [10\(X\)](#).
- II. Let's assume the Thevenin voltage is found to be 6V and the Thevenin resistance is 2 kilo Ohm. **Label the node** of the load as 'VL'. We will vary the load resistance, 'RL'. For this, the value of the sweeping component is to be set any variable. Here, the value of the load RL has been set to 'x'.
- III. **Right-click on blank space** → **Draft** → **SPICE Directive's** | Or, **click this icon on the toolbar** . This opens the netlist window. **Right click on the black text box** → **Help me Edit** → **.step Command**. This opens the '.step Statement Editor' window.



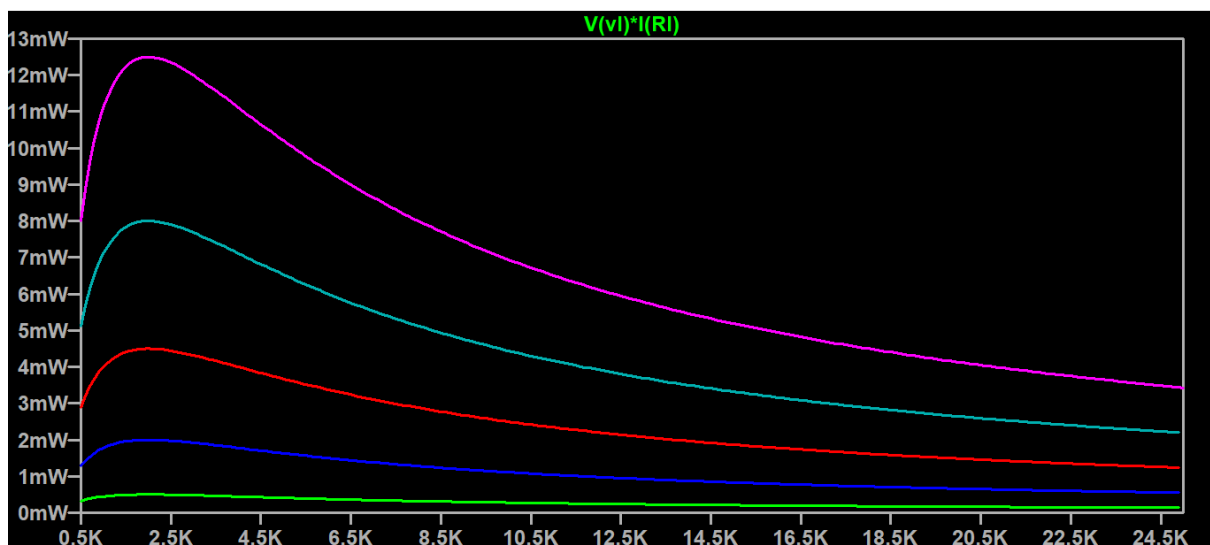
- IV. Set 'Name of parameter to sweep' as the variable name ('x'). Linearly vary the load resistance from 100 Ω to 20 k Ω with an increment of 100 Ω . **Click OK** → **Place the command anywhere in the grey interface**.
- V. As we shall observe the dc voltage/current/power, we must do a dc analysis. Again open the netlist window by **Right-clicking on blank space** → **Draft** → **SPICE Directive's** | Or, by

clicking this icon on the toolbar . Write '.op' in the text box and **Click OK**. Place the command anywhere in the grey interface.

- VI.** **Run the simulation** [See step [10\(XV\)](#) if necessary]. This opens up a plot window with RL in the horizontal axis.
- VII.** We shall now plot the power of the load. **Right click on the black area** → **Add trace** → write **V(vl)*I(Rl)** in the 'Expression(s) to add field' of the Trace adding window and **hit OK**. You should get a plot look like this.

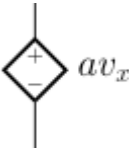
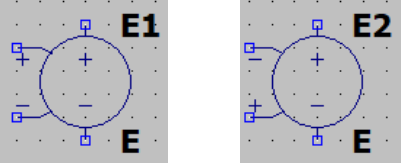

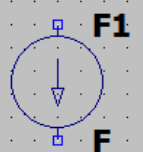
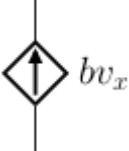
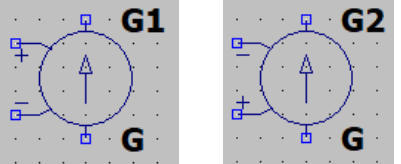

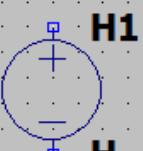
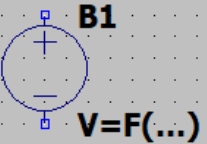
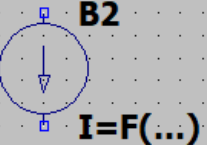


- VIII.** Let's validate the fact that the maximum power transfer occurs when $R_L = R_{Th}$ ($= 2 \text{ k}\Omega$ here). Take a cursor by clicking on the plot label $[V(vl)*I(Rl)]$. Move the cursor to the maximum position of the curve. From the cursor window, it can be seen that the maximum power occurs at $R_L \approx 2 \text{ k}\Omega$ and the maximum power is $\approx 4.45 \text{ mW}$ which match with theoretical values ($R_L = R_{Th} = 2 \text{ k}\Omega$, $P_{max} = \frac{V_{Th}^2}{R_{Th}}$).
- IX.** Multiple parametric sweeps can also be done. For example, if we vary the Thevenin voltage along with RL, we will get power curves corresponding to each V_{Th} . The V_{Th} has been varied *linearly* from 2 to 10 V with 2 V increment (**.step param y 2 10 2**).



14. Modelling Dependent sources:

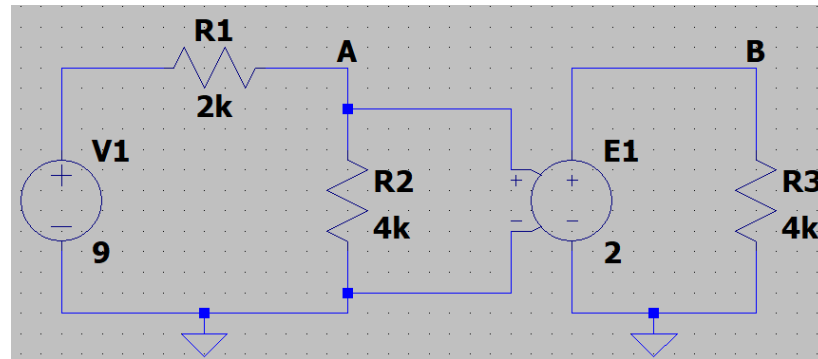
- I. The LTspice keywords for different dependent sources are given below along with the component symbols.

Component	Keyword	Circuit Symbol	LTspice Symbol
Voltage controlled voltage source (VCVS)	e or e2		
Current controlled current source (CCCS)	f [*]		
Voltage controlled current source (VCCS)	g or g2		
Current controlled voltage source (CCVS)	h [*]		
Arbitrary value voltage source	bV	—	
Arbitrary value current source	bI	—	

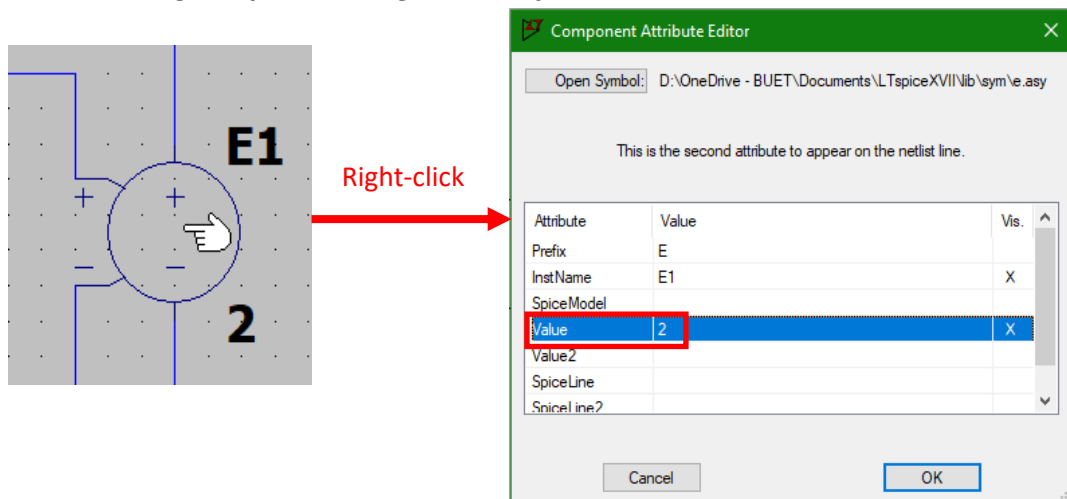
* The controlling current for the sources f and h can only be that supplied by an independent voltage source. We will use the *arbitrary value current source (bI)* as the CCVS or CCCS to control the dependent sources with any current through any given branch.

II. VCVS:

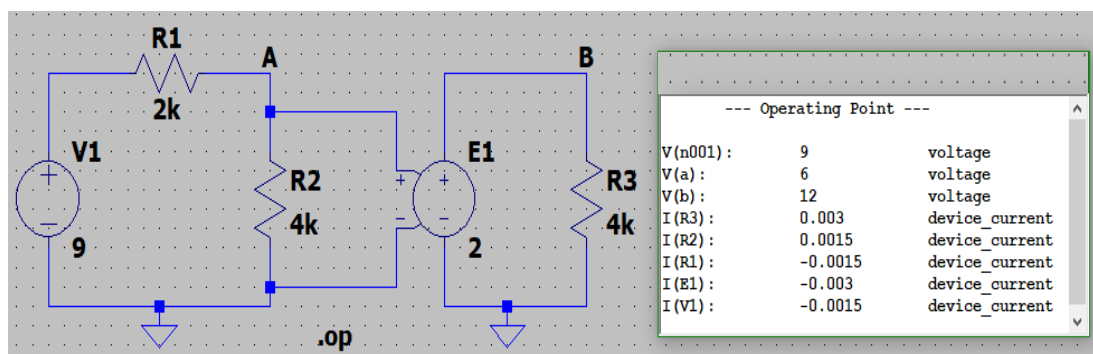
- **Open a new schematic** and **build the circuit** below using the processes outlined in steps [10\(I\)](#) through [10\(X\)](#).
- Note in this circuit that the dependent voltage source is controlled by the voltage across the resistor R2

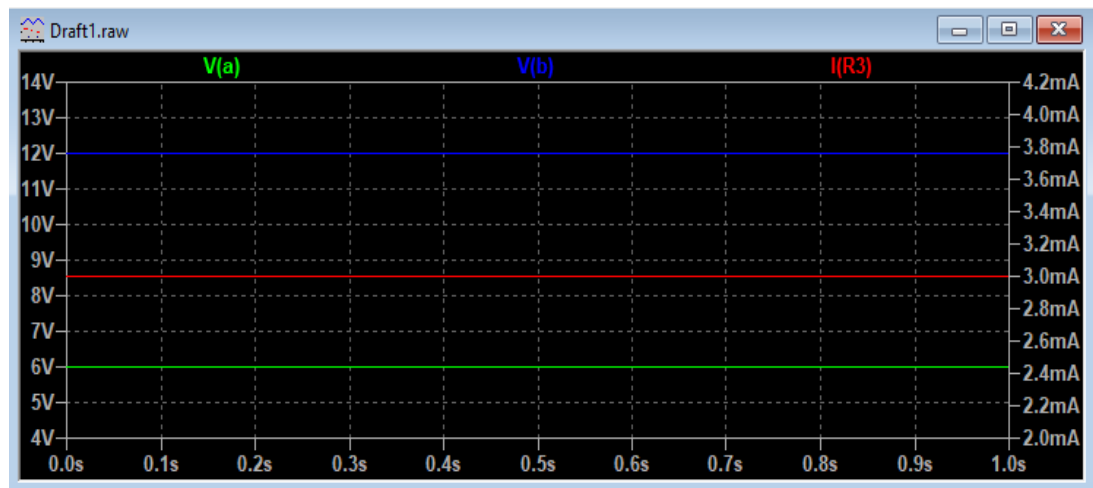


- To set the gain (= 2 in this example) of the dependent source, **hover the cursor on the source** → **Right click**. This will open the **Component Attribute Editor** window. **Set the gain by overwriting the value field** → **Click OK**.



- **Run** the dc operating point simulation (.op) and/or the transient analysis (.tran) to observe the voltages/currents.

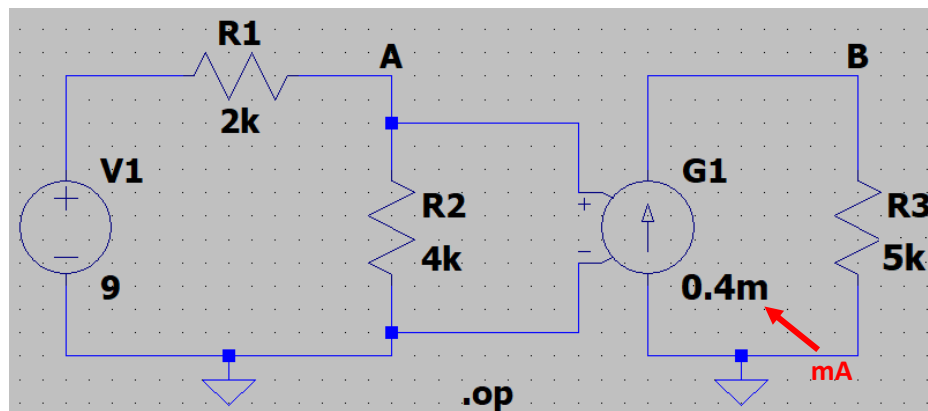




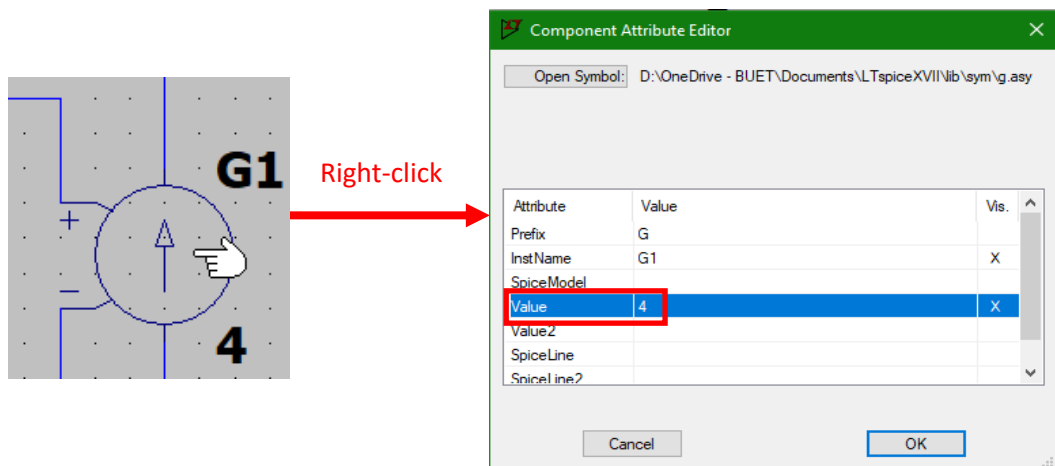
- Simple circuit analysis can confirm that the observed voltages/currents match those predicted by theory.

III. VCCS:

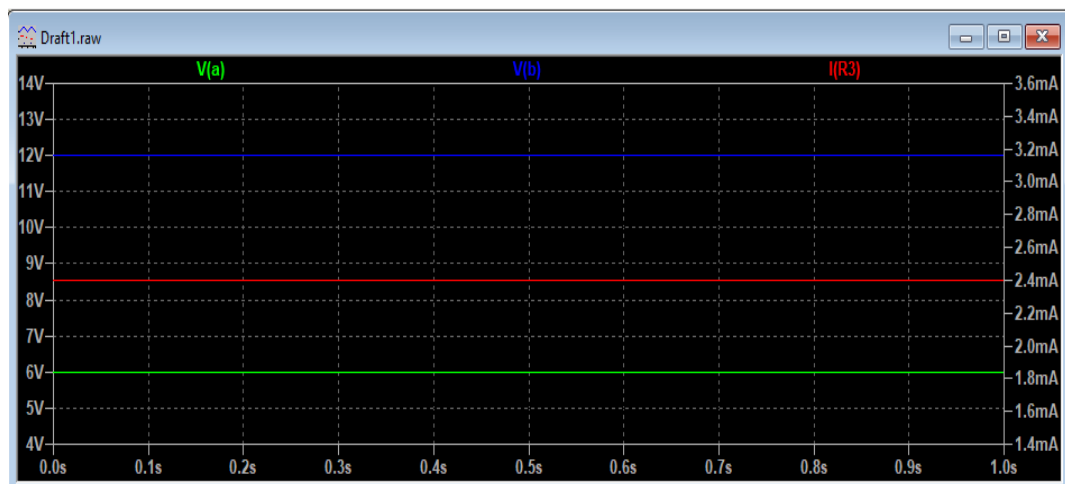
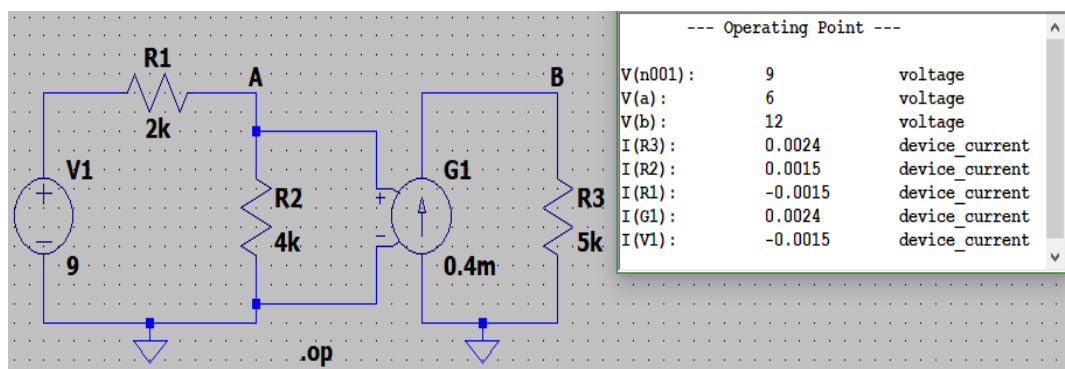
- **Open a new schematic** and **build the circuit** below using the processes outlined in steps [10\(I\)](#) through [10\(X\)](#).
- Note in this circuit that the dependent voltage source is controlled by the voltage across the resistor R2



- To set the gain (= 0.4m in this example) of the dependent source, **hover the cursor on the source** → **Right click**. This will open the **Component Attribute Editor** window. **Set the gain by overwriting the value field** → **Click OK**.



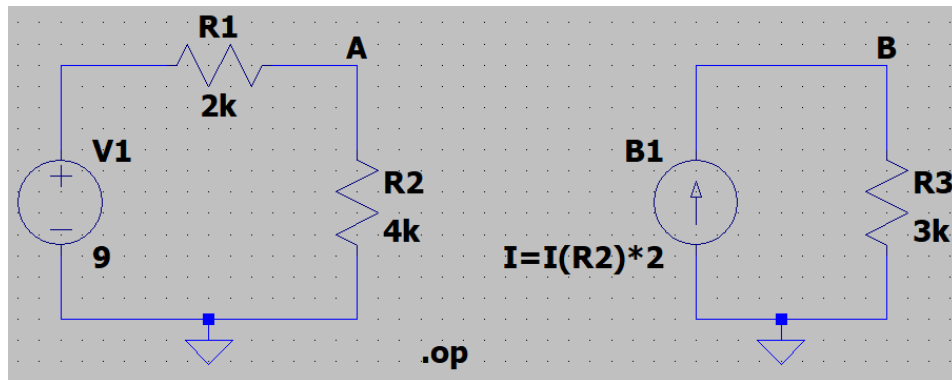
- **Run** the dc operating point simulation (.op) and/or the transient analysis (.tran) to observe the voltages/currents.



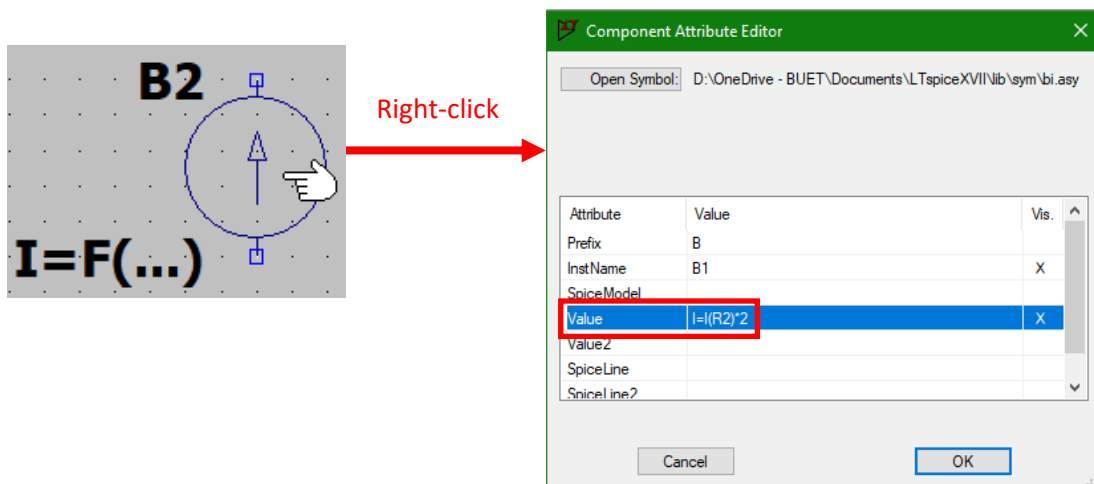
- Simple circuit analysis can confirm that the observed voltages/currents match those predicted by theory.

IV. CCCS:

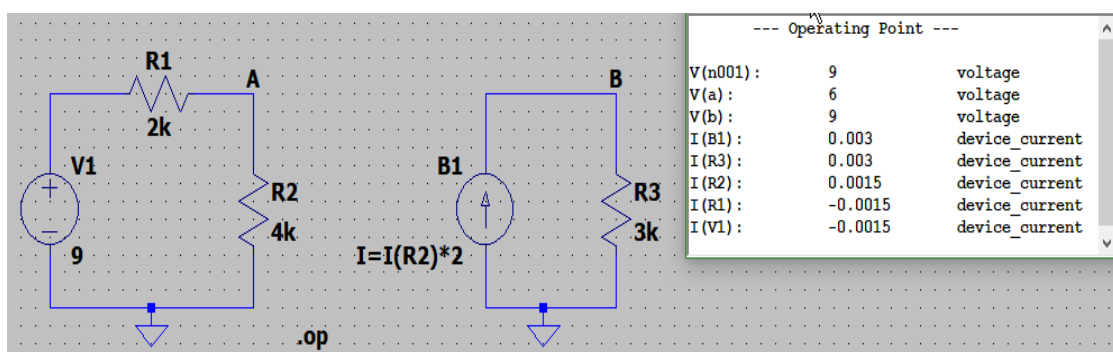
- Read the [note \(*\)](#) first.
- **Open a new schematic** and **build the circuit** below using the processes outlined in steps [10\(I\)](#) through [10\(X\)](#).
- We will set the current through R2 as the controlling current for the dependent current source (B1).

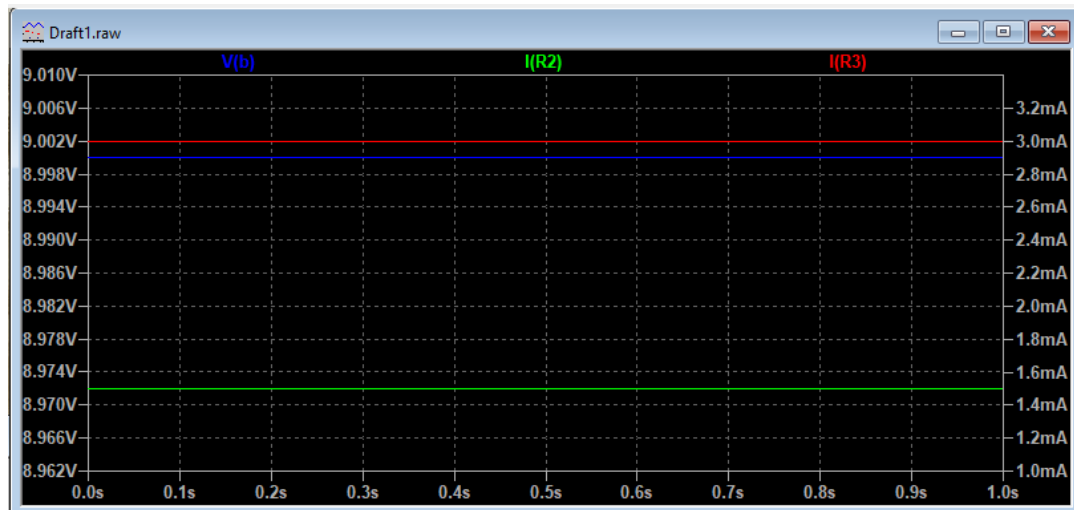


- To set the source B1 as stated in the previous step, **hover the cursor on the source** → **Right click**. This will open the **Component Attribute Editor** window. **Overwrite the value field by "I(R2)*2"** → **Click OK**. "I(R2)*2" means the controlling current is the current through the resistor R2 and the gain is 2.



- **Run** the dc operating point simulation (.op) and/or the transient analysis (.tran) to observe the voltages/currents.

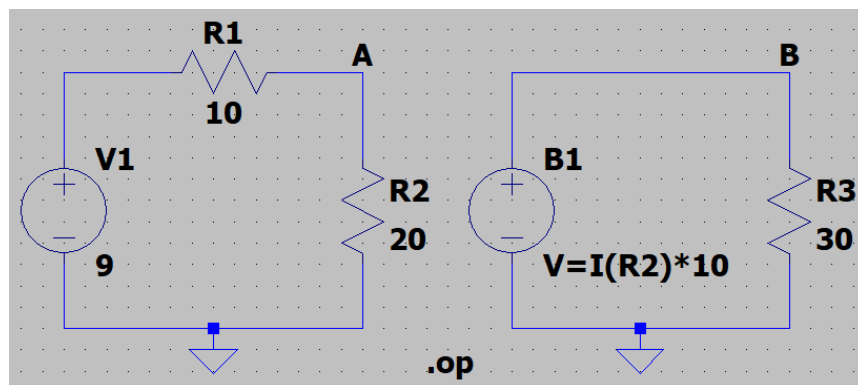




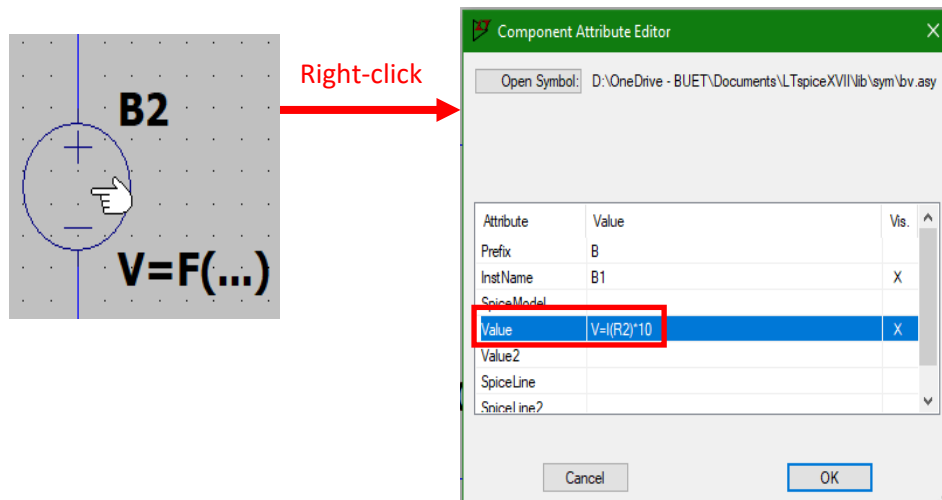
- Simple circuit analysis can confirm that the observed voltages/currents match those predicted by theory.

V. CCVS:

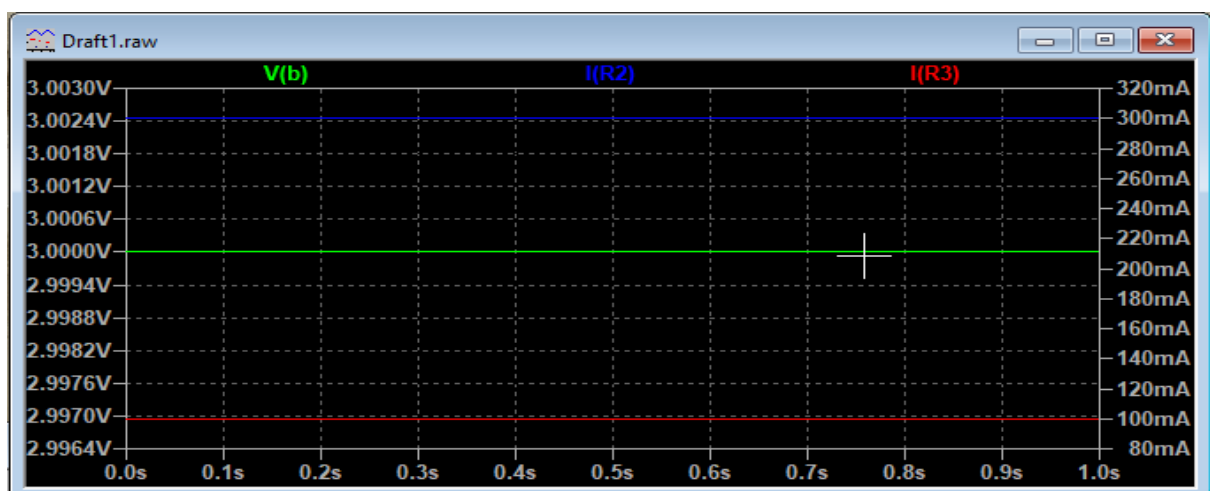
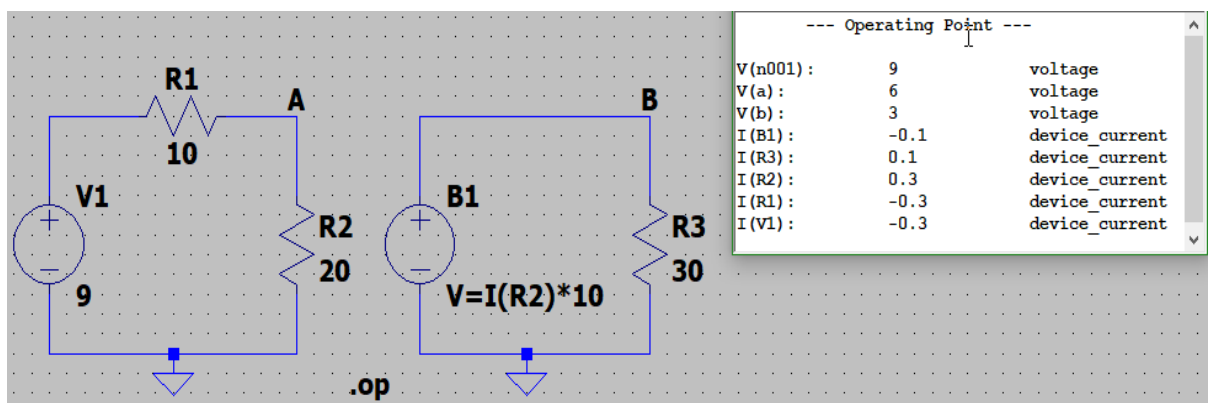
- Read the [note \(*\)](#) first.
- **Open a new schematic** and **build the circuit** below using the processes outlined in steps [10\(I\)](#) through [10\(X\)](#).
- We will set the current through R2 as the controlling current for the dependent current source (B1).



- To set the source B1 as stated in the previous step, **hover the cursor on the source** → **Right click**. This will open the **Component Attribute Editor** window. **Overwrite the value field by "I(R2)*10"** → **Click OK**. "I(R2)*10" means the controlling current is the current through the resistor R2 and the gain is 10.



- **Run** the dc operating point simulation (.op) and/or the transient analysis (.tran) to observe the voltages/currents.



- Simple circuit analysis can confirm that the observed voltages/currents match those predicted by theory.

Prepared by,
[PDS] PURBAYAN DAS