Student ID:	22301689	Lab Section:	28
Name:	TASNIM RAHMAN MOUMITA	Lab Group:	03

Experiment No. 5

Study of I-V Characteristics of Linear Circuits & Verification of Thevenin's Theorem and Maximum Power Transfer Theorem Using Software (LTSpice) Simulation.

Part 1: Introduction to Simulation using LTSpice & Study of I-V Characteristics of Linear Circuits

Objective

The aim of this experiment is to introduce a circuit simulation software (LTSpice) and to acquaint students with the concept of I-V characteristics. They will simulate I-V characteristics of some circuits consisting of linear components using LTSpice.

Theory

I-V characteristics, also known as current-voltage characteristics, describe the relationship between the current flowing through a device/circuitry and the corresponding potential difference (voltage) across it. This concept is commonly used in the field of electronics and electrical engineering to analyze the behavior of various components such as resistors, diodes, transistors, and in general, circuits.

I-V characteristics provide a way to understand how current and voltage interact in electrical and electronic components and circuits. By analyzing these characteristics, circuits/devices can be designed and optimized, appropriate components can be selected, and the behavior of devices under different operating conditions can be predicted.

A circuit is linear if its I-V characteristic is linear, represented by a straight line in an I versus V plot. A circuit composed of linear components (resistors, voltage sources, current sources) has a straight line I-V characteristic.

For a simple resistor, the I-V characteristics follow Ohm's Law, which states that the current passing through a resistor is directly proportional to the voltage applied across it.

Mathematically, this relationship can be expressed as $I = \frac{V}{R}$, where I is the current, V is the voltage, and R is the resistance. The characteristic line passes through the origin.

The I-V characteristic of an ideal voltage source (an ideal voltage source is a theoretical concept that maintains a constant voltage across its terminals, regardless of the current flowing through it.) is a vertical line on an I-V graph, indicating that the voltage remains constant (V) regardless of the current (I). Mathematically, it can be represented as

exactly $V = constant$ behave this way, but should closely resemble an ideal voltage source. Similarly, an . Real life voltage sources (for example, DC power supply in our labs) do not

ideal current source has an I-V line parallel to the voltage axis since it supplies a constant current with theoretically any voltages across.

Procedures

Simulation using LTspice

Let's get introduced to LTspice by simulating the simple circuit shown below (**Circuit 1**).
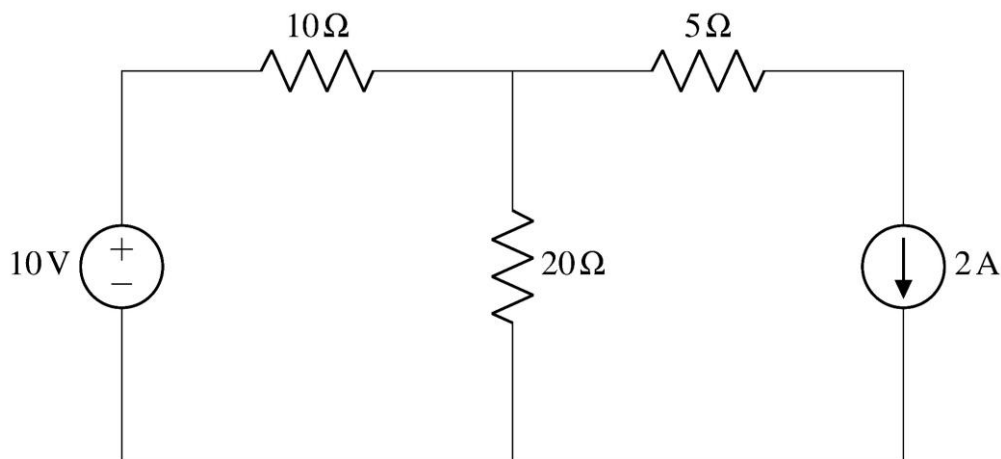
Circuit 1

➤ Open a new schematic window by clicking **File → New Schematic**. To find the required circuit components, **Right-click on blank space → Draft → Component** or

click on the **'Component'** icon from the toolbar right above the schematic window. A window titled **'Select Component Symbol'** will appear. Type the keyword for a component to be inserted. For example, for a resistor type **'resistor' → select the component → click OK**. 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. To assign the required value (Resistance, Voltage, Current etc.) to each component **Right click on the component symbol placed in the**

schematic → Fill up the corresponding box.

➢ To connect the circuit components *Right-click on blank space → Draft → Draw Wire* or click on the *'Wire'* icon from the toolbar.

➢ Find the ground symbol from the toolbar ▽ , place it on the schematic and connect it to the circuit.

➢ Label the nodes of the circuit using *'Label Net'*. Find *'Label Net'* from the toolbar by clicking this [A] icon or *Right-click on blank space → Draft → Label Net*. After these, your circuit should look like this,
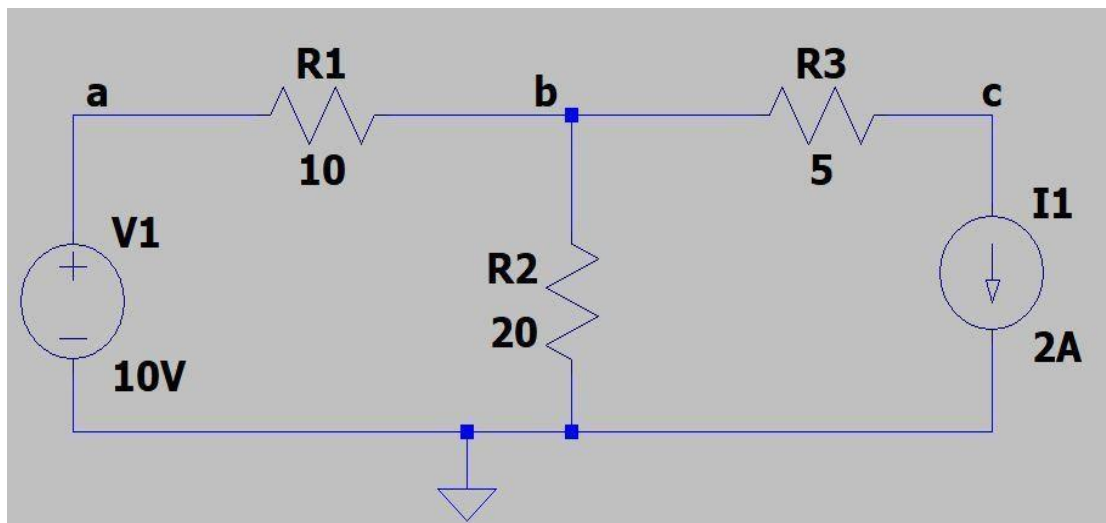

➢ To analyze the circuit (to get the node voltages and branch currents), we have to do **'DC operating point'** analysis.

To run the 'DC operating point analysis, we have to write the analysis command. First, find the *'Spice Directives'* option by *Right-clicking on the schematic*

Draft Spice Directives or clicking on the "SPICE Directive" .op icon from the toolbar.

➢ After clicking the *'Spice Directives'*, the *'Edit Text on the Schematic'* window will appear. Now *Right-click on the blank space on this window Select 'Help me Edit' Analysis Command.* A window titled *'Edit Simulation Command'* will appear. Select **'DC op pnt' OK.** It will generate a '.op command,which is the command for DC operating point analysis. Place this command somewhere in the schematic.

3
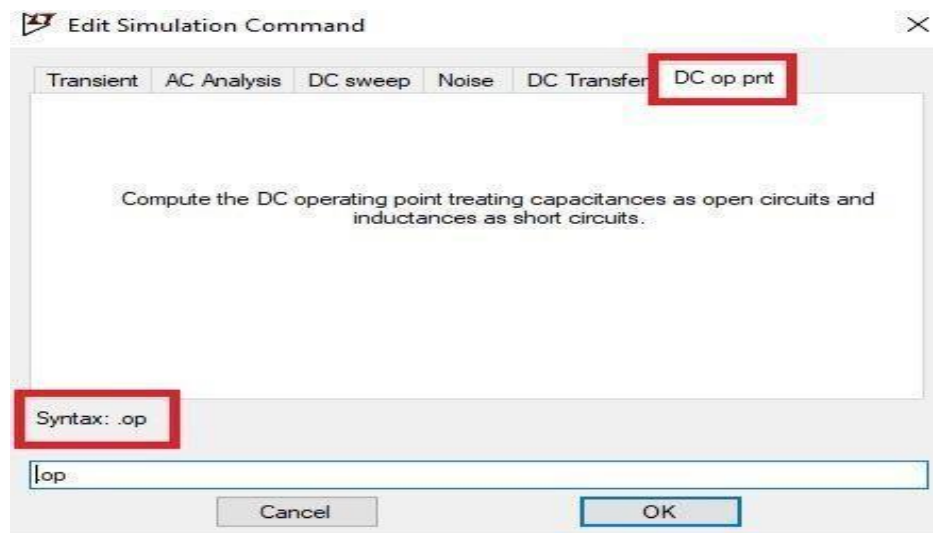
[Notice the '.op' syntax for DC operating point analysis. You could just write the command in the command editor box and place that command in the schematic.]

➢ To run the simulation, click **'Run'**. Find the **'Run'** button ![icon] from the above toolbar or **Right-click on the schematic →Run**.

➢ After clicking 'Run' a new window titled **'Operating Point'** will appear, containing all node voltages and branch currents.
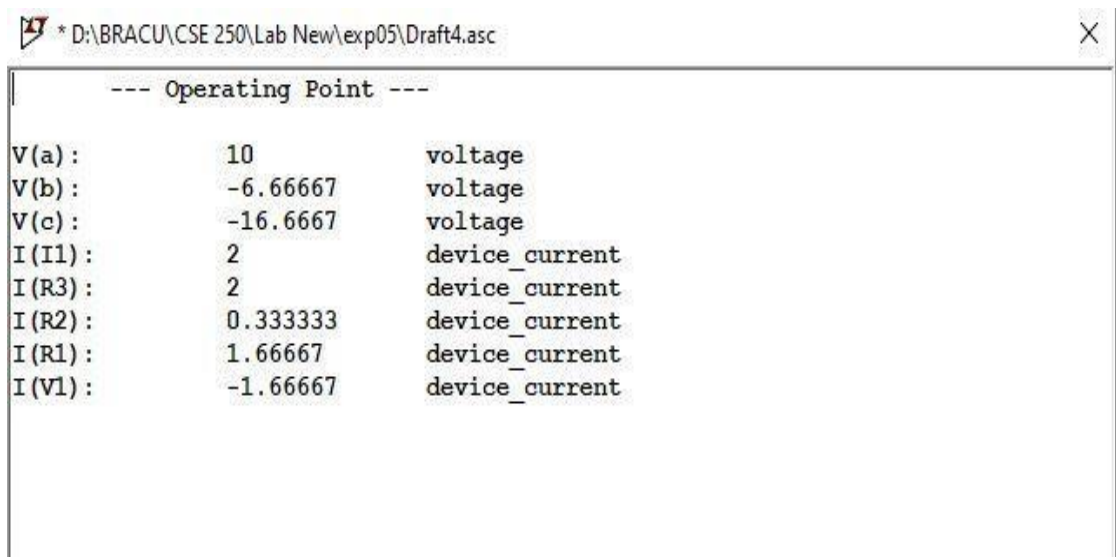

I-V Characteristic plotting using LTspice

In **Experiment 4,** we assessed the IV characteristics graph of a given circuit and that of its equivalent circuit too and found that the IV graphs are exactly the same for both circuits, which confirms their equivalency. We can easily get the IV characteristics of those circuits using LTspice. Here are the circuits we studied in Experiment -04.
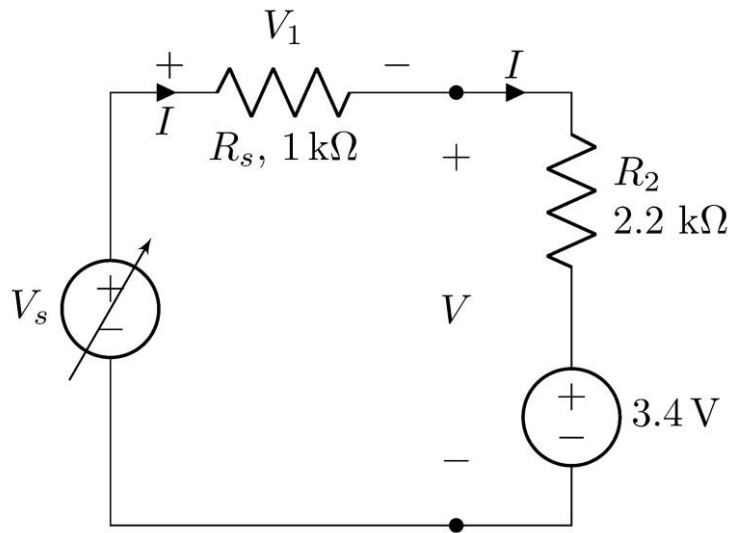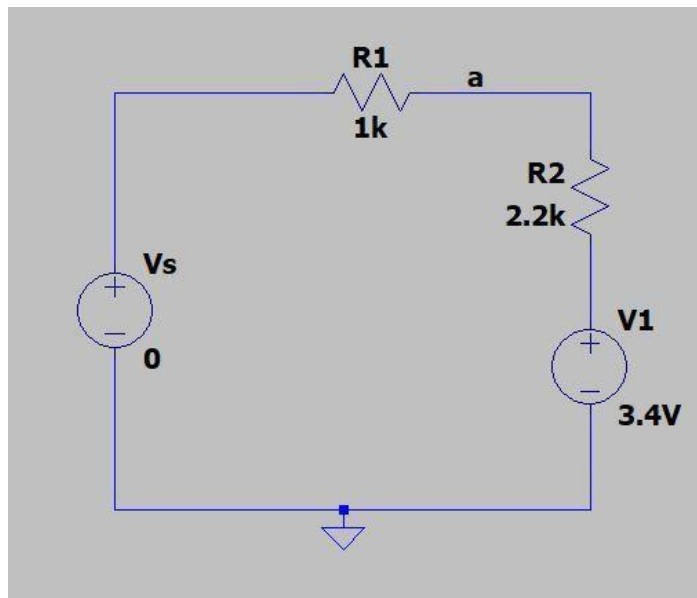
Circuit 2
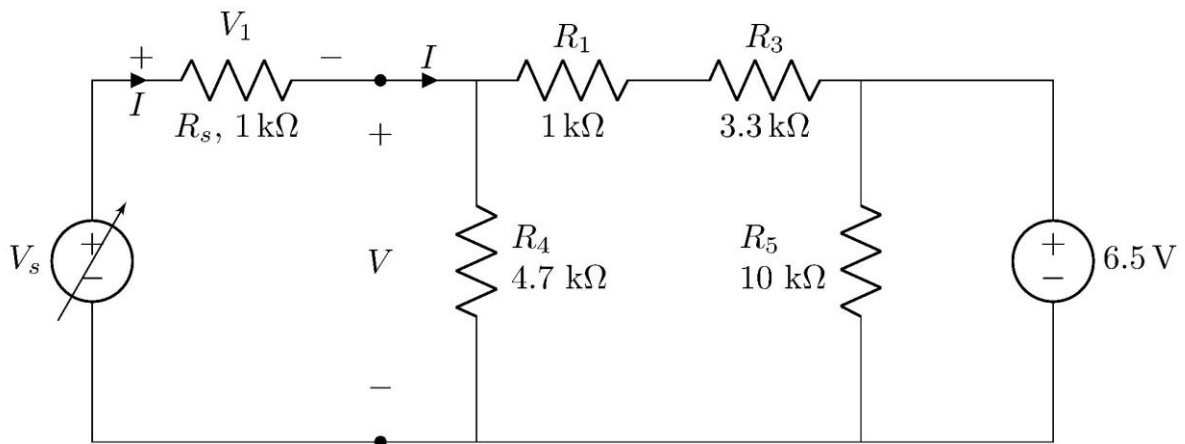
Circuit 3

➤ Draw **Circuit 2** shown in a new schematic in LTspice. Modify the components with their values and name the nodes. Label the nodes. For the variable voltage source 'Vs' shown in the circuit , we will use a normal voltage source and vary its value during simulation.

Note that, here the variable voltage source has been renamed to 'Vs' and assigned with a value of **0 Volt**.

➢ To plot the IV characteristics graph, we need to vary the voltage of the source 'Vs'. And to do so, we have to do 'DC Sweep Analysis'. Now Find the *'Spice Directives'* option by ***Right-clicking on the schematic Draft Spice Directives*** or find it from the toolbar above.

➢ After clicking the *'Spice Directives'*, the *'Edit Text on the Schematic'* window will appear. Now ***Right-click on the blank space on this window Select 'Help me Edit' Analysis Command.*** A window titled *'Edit Simulation Command'* will appear. Insert values in the boxes as below and click OK. It will generate a transient analysis command. Place the command somewhere on the schematic.
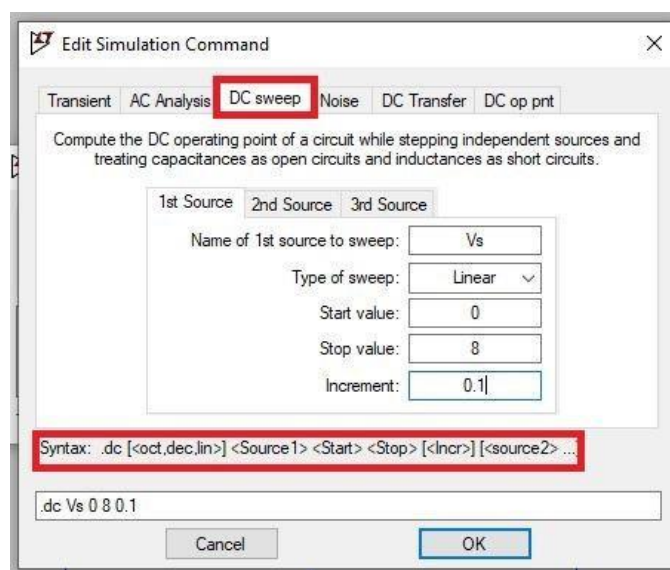
[Notice the '.dc' syntax for DC Sweep Analysis. You could just write the command in the command editor box and place that command in the schematic.]

➢ To run the simulation, click *'Run'*. Find the *'Run'* button from the above toolbar or *Right-click on the schematic →Run*..

➢ After the simulation 'run' is complete, a blank plot window will appear. Notice that in the X axis it shows the voltage of source 'Vs'. In the I-V characteristics graph, there should be voltage of the node **'a'** (according to the circuit) in the X axis. To show the voltage of node 'a' in the X axis, *Right click on the X axis →Type V(a) in the 'Quantity plotted' box →OK.*
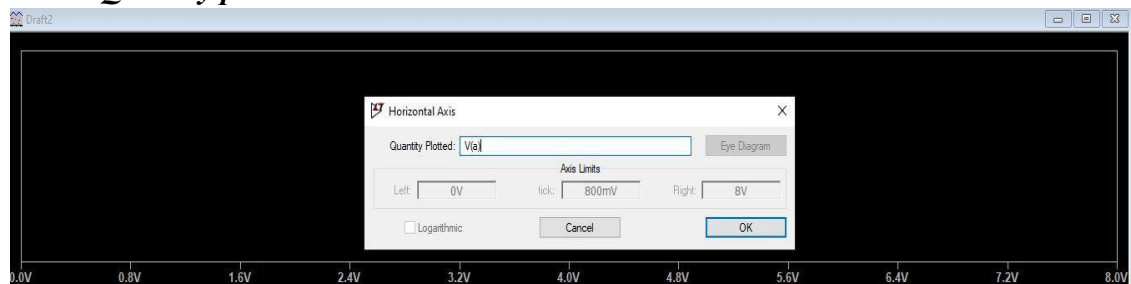
➢ To plot the current, *Right click on the blank space in the plot window → Add Traces.* A new window will appear. Select *I(R1)* (according to the circuit) and click *OK*.

➢ To extract data from a plot/response, use the *data cursor*. A cursor for a particular trace will appear by clicking on the name of that trace. The data point of the cursor can be moved by the arrow keys (navigation keys) on the keyboard.
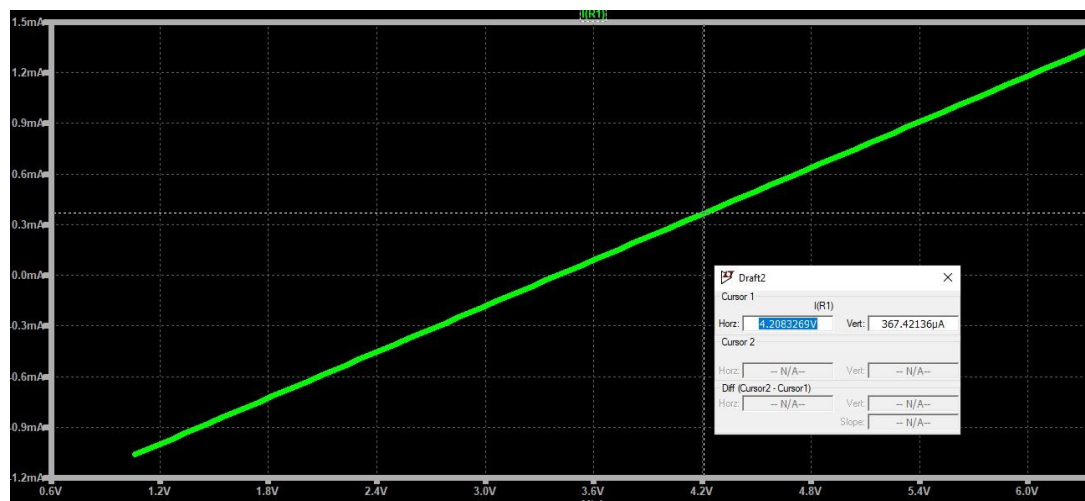
Lab Work: Simulate and plot the I-V characteristics of **Circuit 3** following the steps mentioned above. You should get the same result as you did in the hardware lab.

Part 2.1: Verification of Thevenin's Theorem using Software Simulation (LTSpice)

Objective

The aim of this experiment is to verify Thevenin's theorem with reference to a given circuit using LTSpice. They will find out the Thevenin's equivalent circuit parameters ($\mathbf{V_{OC}}$, $\mathbf{I_{SC}}$, $\mathbf{R_{th}}$) of a given circuit using simulation and determine the Thevenin's equivalent circuit. They will also simulate I-V characteristics of the given circuit and its Thevenin's equivalent circuit using LTSpice to verify Thevenin's theorem.

Theory

It is often desirable in circuit analysis to study the effect of changing a particular branch element while all other branches and all the sources in the circuit remain unchanged. Thevenin's theorem is a technique to this end, and it greatly reduces the number of computations that we have to do each time a change is made. Using Thevenin's theorem the given circuit except the particular branch to be studied is reduced to the simplest equivalent circuit possible and then the branch to be changed is connected across the equivalent circuit.

Thevenin's theorem states that any two-terminal linear bilateral networks containing sources and passive elements can be replaced by an equivalent circuit consisting of a voltage source ($\mathbf{V_{th}}$) in series with a resistor ($\mathbf{R_{th}}$), where,

$\mathbf{V_{th}}$ = The open circuit voltage ($\mathbf{V_{OC}}$) at the two terminals A & B.
$\mathbf{R_{th}}$ = The resistance looking into terminals A and B of the network with all sources removed.
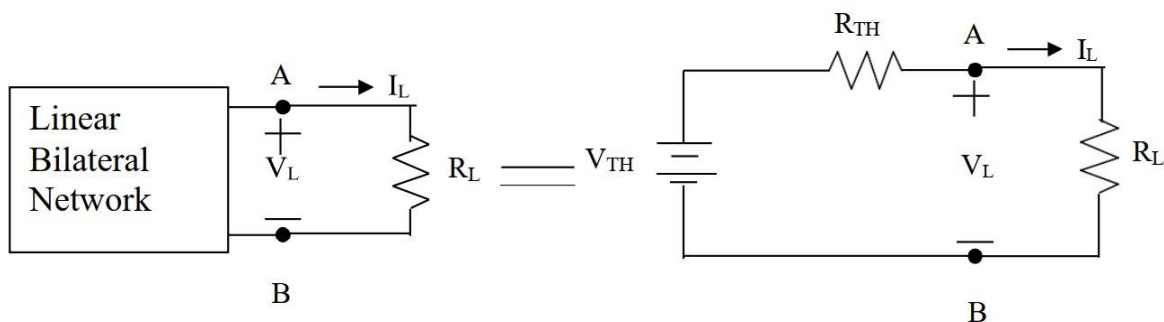
There are several methods for determining Thevenin resistance $\mathbf{R_{th}}$. An attractive method for determining $\mathbf{R_{th}}$ is: (1) determine the open circuit voltage, and (2) determine the short circuit current $\mathbf{I_{SC}}$ as shown in the figure; then

$$R_{th} = \frac{V_{OC}}{I_{SC}}$$
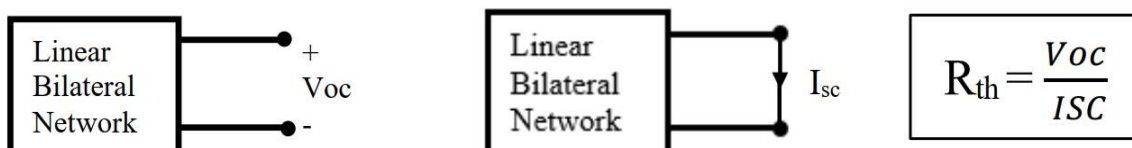
# Methodology of Determining Thevenin's Circuit Parameters (V_{OC}, I_{SC}, R_{th})

In this experiment, We will determine the Thevenin's Circuit Parameters (V_{OC}, I_{SC}, R_{th}) and plot the I-V characteristics of the following Circuit with respect to terminals **a, b**.
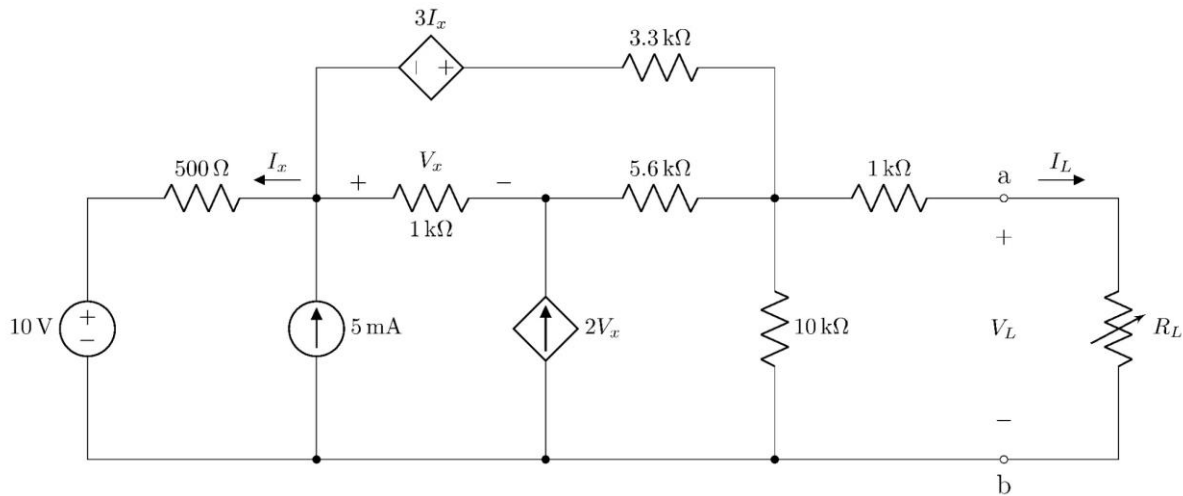
Circuit 4: Original Circuit

Step 1: Determining V_{OC}

Remove the load resistance R_L and find the open circuit voltage between terminals A & B. This voltage is called Thevenin's voltage, i.e., $\mathbf{V_{th}=V_{OC}}$. It is also known as the Open Circuit Voltage.
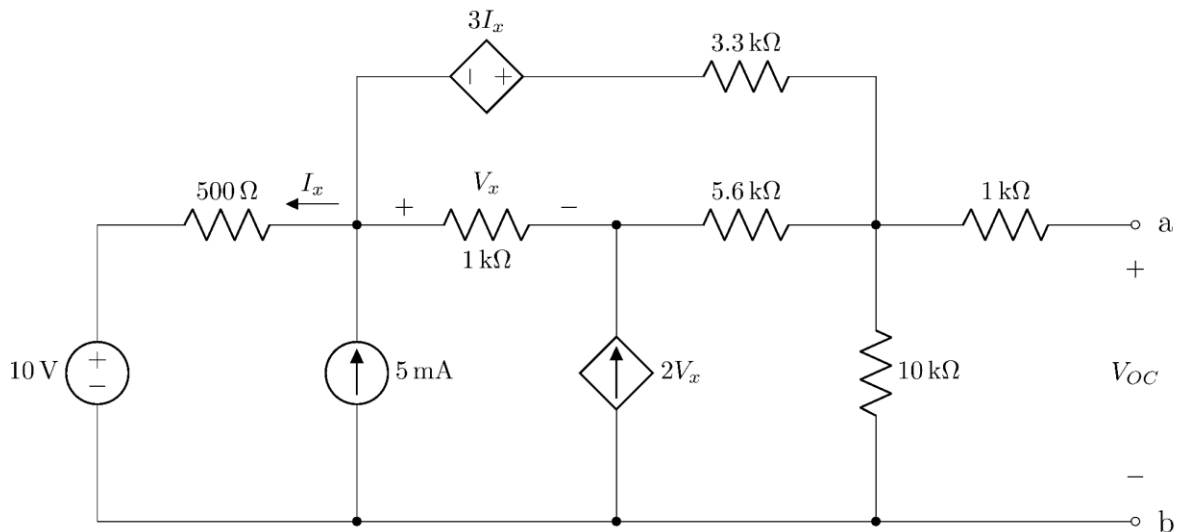
Circuit 5: Circuit for finding $\mathbf{V_{OC}}$

Step 2: Determining I_{SC}

Place a short circuit between terminals A & B (simply connect them through a wire). The current through the short circuit is called Norton's Current, i.e., $\mathbf{I_N=I_{SC}}$. It is also known as the Short Circuit Current.
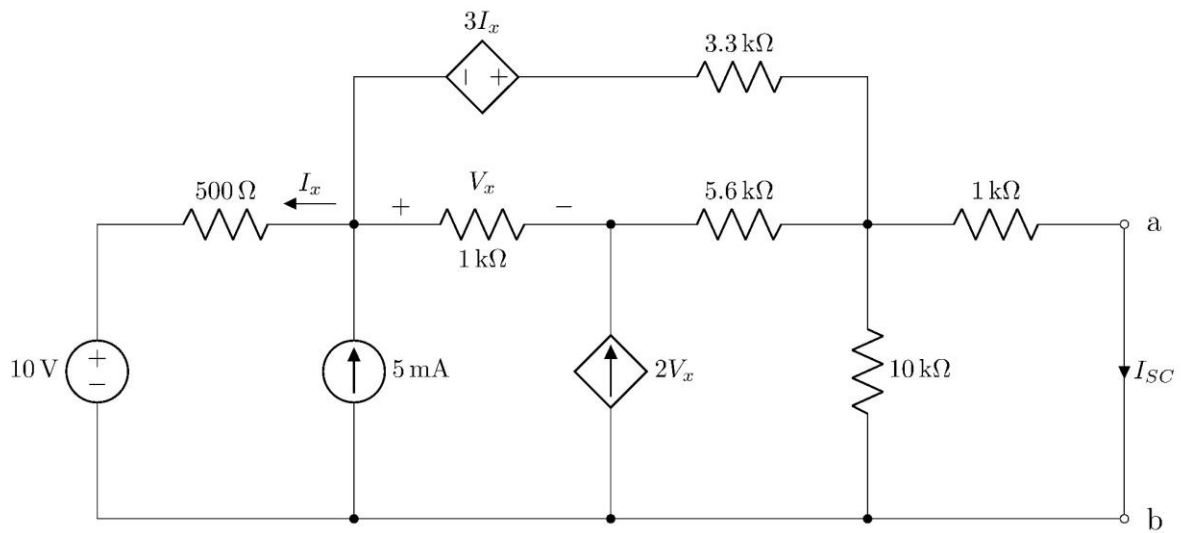
Circuit 6: Circuit for finding I_{SC}

Step 3: Determining R$_{th}$

Divide the Open Circuit Voltage by the Short Circuit Current to determine the Thevenin's

Resistance. $\mathbf{R_{th}} = \dfrac{V_{OC}}{I_{SC}}$

Step 4: Constructing Thevenin's Equivalent Circuit

Construct Thevenin's equivalent circuit as shown in the following figure setting the voltage source at **V$_{th}$** volts and the series resistance at **R$_{th}$** ohms. The values shown here should closely match the corresponding values you determined from the earlier steps.

Circuit 7: Thevenin's Equivalent Circuit

10
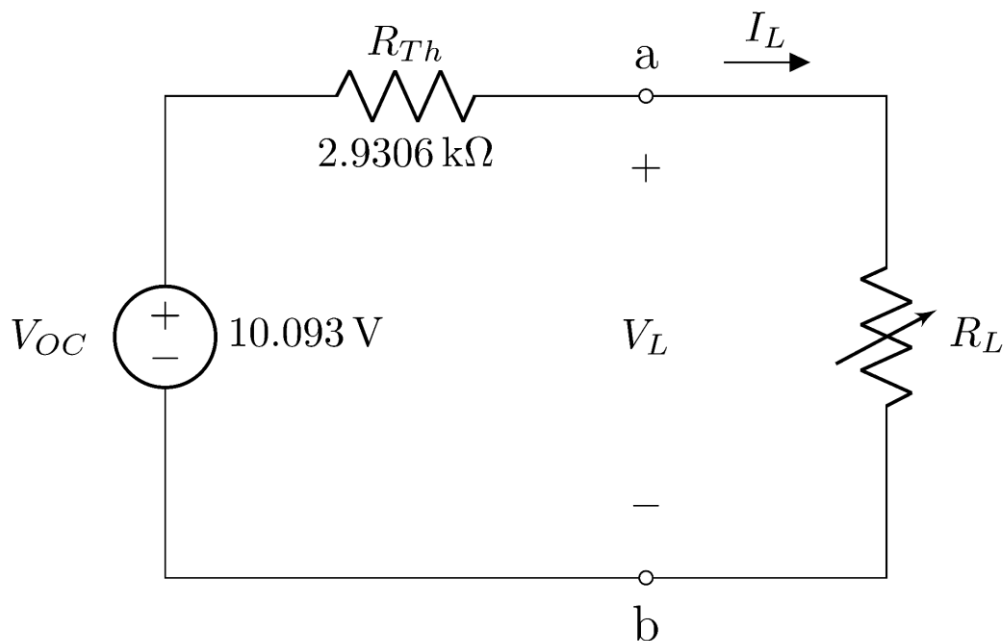
Procedures

Finding V_{OC} using LTspice

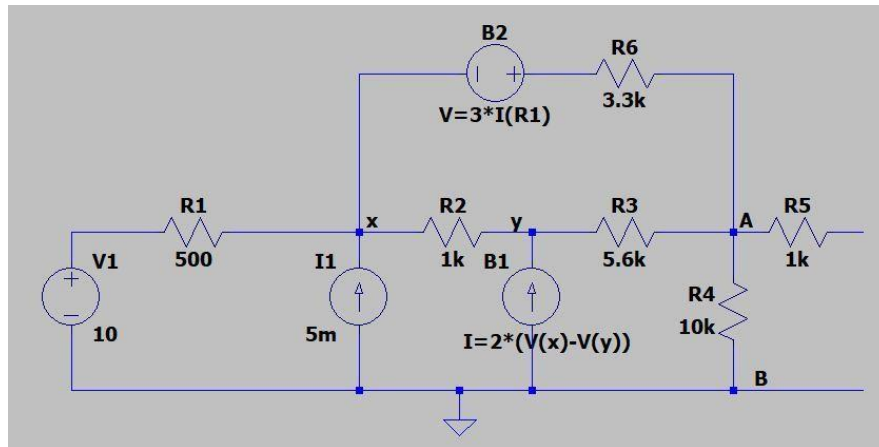➢ Draw **Circuit 5** in a new schematic. Modify the components with appropriate values and label the nodes.

➢ To measure open circuit voltage, **Voc**, we have to run *'DC Operating Point'* analysis. To run the *'DC Operating Point'* analysis we have to write the analysis command. Find the *'Spice Directives'* option by ***Right-clicking on the schematic Draft Spice Directives*** or find it from the toolbar above*.*

➢ After clicking the *'Spice Directives'*, the *'Edit Text on the Schematic'* window will appear. Now ***Right-click on the blank space on this window Select 'Help me Edit' Analysis Command.*** A window titled *'Edit Simulation Command'* will appear. Select *'DC op pnt' OK.* It will generate a *'.op'* command which is the command for DC operating point analysis. Place this command somewhere in the schematic.

➢ To run the simulation, click *'Run'*. Find the *'Run'* button from the above toolbar or ***Right-click on the schematic →Run***.

➢ After clicking *'Run'* a new window titled **'Operating Point'** will appear, containing all node voltages and branch currents. From this window, note the voltage at node **'A'** (according to the circuit). This is the open circuit voltage, **Voc**.
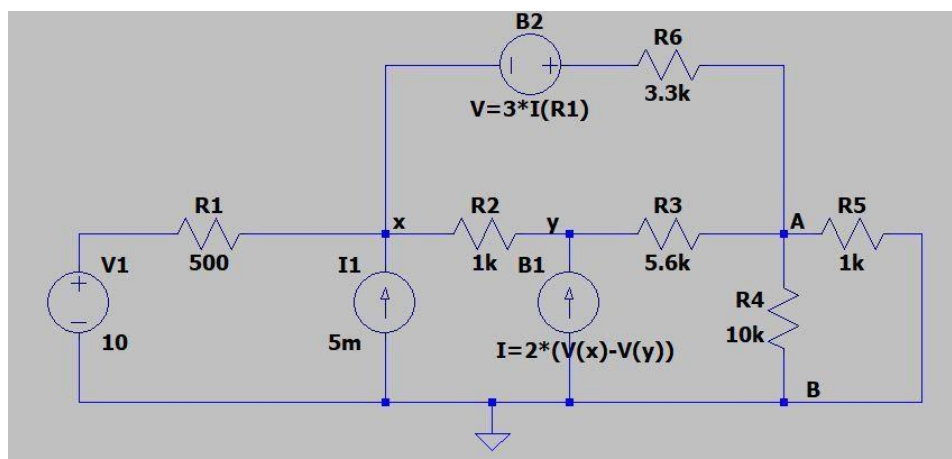
11

```
* D:\BRACU\CSE 250\Lab New\exp05\Draft1.asc

        --- Operating Point ---

V(n002):      10              voltage
V(x):         12.1816         voltage
V(y):         12.1814         voltage
V(a):         10.0939         voltage
V(n001):      12.1947         voltage
V(n003):      10.0939         voltage
I(B2):        -0.000636612    device_current
I(B1):        0.000372589     device_current
I(I1):        0.005           device_current
I(R5):        0               device_current
I(R6):        0.000636612     device_current
I(R4):        -0.00100939     device_current
I(R3):        -0.000372775    device_current
I(R2):        1.86295e-007    device_current
I(R1):        0.0043632       device_current
I(V1):        0.0043632       device_current
```

Finding I$_{SC}$ using LTspice

➢ Draw **Circuit 6** in a new schematic. Modify the components with appropriate values and label the nodes.

➢ To measure the short circuit current **Isc**, we have to run *'DC Operating Point'* analysis. Run the simulation in *'DC Operating Point'* analysis mode by following the steps mentioned earlier.

➢ After clicking *'Run'* a new window titled **'Operating Point'** will appear, containing all node voltages and branch currents. From this window, note the current through resistor **R5** (according to the circuit). This is the short circuit current, **Isc**.

```
*  * D:\BRACU\CSE 250\Lab New\exp05\Draft1.asc

          --- Operating Point ---

V(n002):          10               voltage
V(x):             11.3071          voltage
V(y):             11.3064          voltage
V(a):             3.44448          voltage
V(n001):          11.315           voltage
I(B2):            -0.002385        device_current
I(B1):            0.00140322       device_current
I(I1):            0.005            device_current
I(R5):            0.00344448       device_current
I(R6):            0.002385         device_current
I(R4):            -0.000344448     device_current
I(R3):            -0.00140392      device_current
I(R2):            7.01611e-007     device_current
I(R1):            0.00261429       device_current
I(V1):            0.00261429       device_current
```
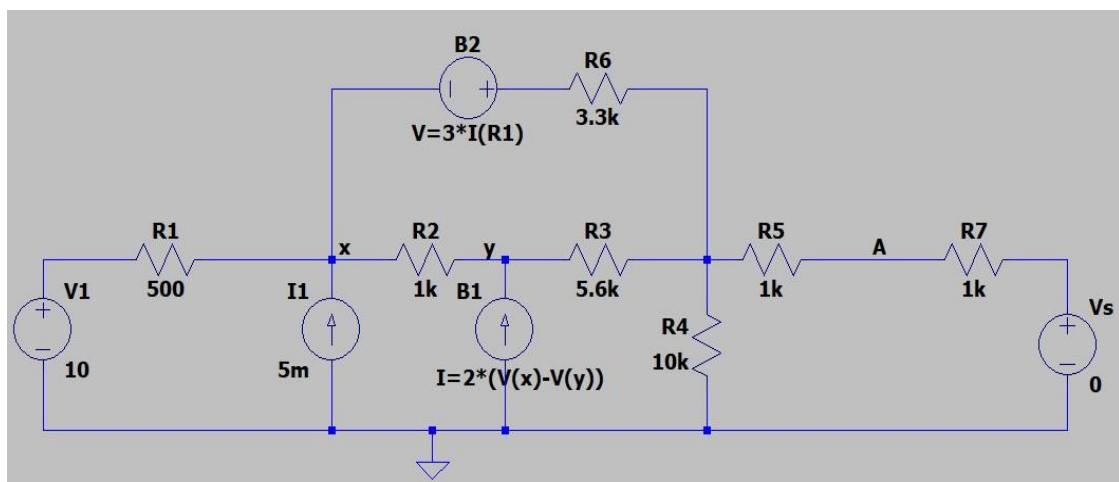
To confirm the equivalency of the given circuit and its assessed Thevenin's equivalent circuit, we will compare their **I-V characteristics graphs**.
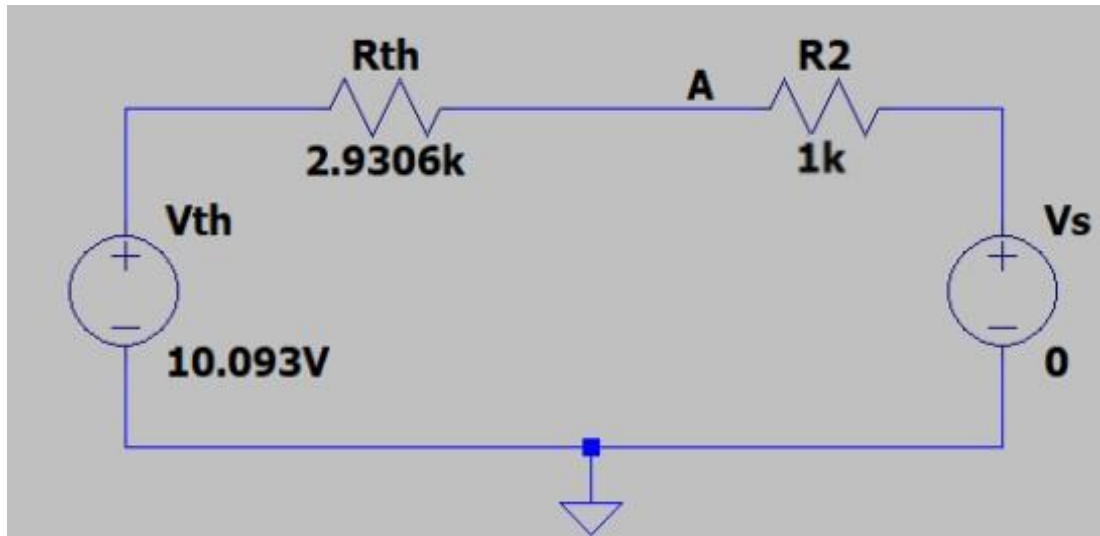
I-V Characteristics of the Given Circuit

➢ Draw **Circuit 4** in a new schematic. Modify the components with appropriate values and label the nodes.

➢ Get the I-V characteristics graph by following the steps mentioned earlier. *[Hint: Run* ***'DC Sweep Analysis'*** *simulation. In the graph, plot the voltage at node 'A',* ***'V(a)'*** *(according to the circuit shown above, can be different in your case) in the X axis and the current through R7,* ***'I(R7)'*** *(according to the circuit shown above).]*

I-V Characteristics of the Thevenin's Equivalent Circuit

➢ Draw **Circuit 7** in a new schematic. Modify the components with appropriate values and label the nodes. Notice that the value of the voltage source is equal to **V_{th}** and that of the resistance is equal to **R_{th}**.

➢ Get the I-V characteristics graph by following the steps mentioned earlier. *[Hint: Run 'DC Sweep Analysis' simulation. In the graph, plot the voltage at node 'A', 'V(a)' (according to the circuit shown above) in the X axis and the current through R2, 'I(R2)' (according to the circuit shown above).]*

Lab Work: Compare the I-V characteristics of **the Given Circuit & the Thevenin's Equivalent Circuit** and state your opinion on whether Thevenin's Theorem has been verified not.
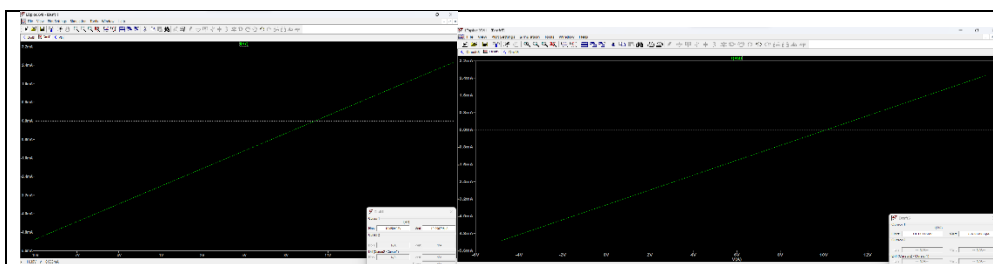
As we can see, both the graphs' IV characteristics plot r giving the same amount of value that is basically the motive of Thevenin's theorem ,which says that -it allows us to simplify complex circuits into a single voltage source and a single series resistor, making analysis much easier. So, Thevenin's Theorem has been verified .

Part 2.2: Verification of Maximum Power Transfer Theorem using Software Simulation (LTSpice)

Objective

The aim of this experiment is to verify the Maximum Power Transfer theorem with reference to a given circuit using LTSpice. They will plot the Load Power of a given circuit against different values of Load Resistance using simulation and determine the Load Resistance Value for which the Maximum Load Power is achieved. They will also be able to verify the Maximum Power Transfer theorem using this information.

Theory

The Maximum Power Transfer Theorem is a fundamental concept in electrical engineering that relates to the transfer of maximum power from a source to a load. The Maximum Power Transfer theorem states that *A resistive load will receive maximum power when its total resistive value is exactly equal to Thevenin's resistance of the network as "seen" by the load.*
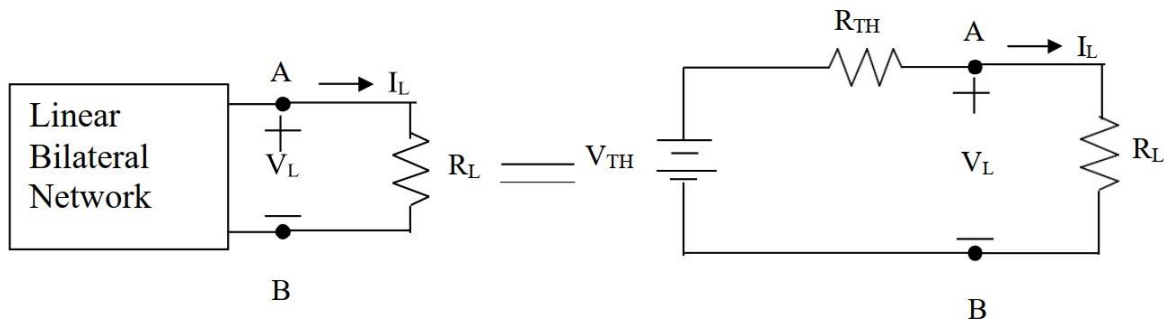
We know that any circuit A terminated with a load R_L can be reduced to its Thevenin's equivalent. Now according to this theorem, the load R_L will receive maximum power when

$R_L = R_{th}$. We can calculate the Maximum Power theoretically using the formula,

$$\mathbf{P_{max}} = \overline{V_{th2}\,4}R_{th}$$

The theorem focuses on the transfer of power between a source and a load. In electrical circuits, power is transferred from a source (such as a generator) to a load (such as a resistor) through a transmission medium (such as wires or conductors).

It's worth noting that the Maximum Power Transfer Theorem is a theoretical concept and is not always practical or desirable in real-world scenarios. In many practical applications, impedance matching is employed to achieve efficient power transfer, but it may not always result in maximum power transfer. Design considerations, system constraints, and other factors often influence the choice of impedance matching in electrical circuits.

Procedures

We will use Thevenin's Equivalent Circuit we determined earlier to verify Maximum Power Transfer Theorem.
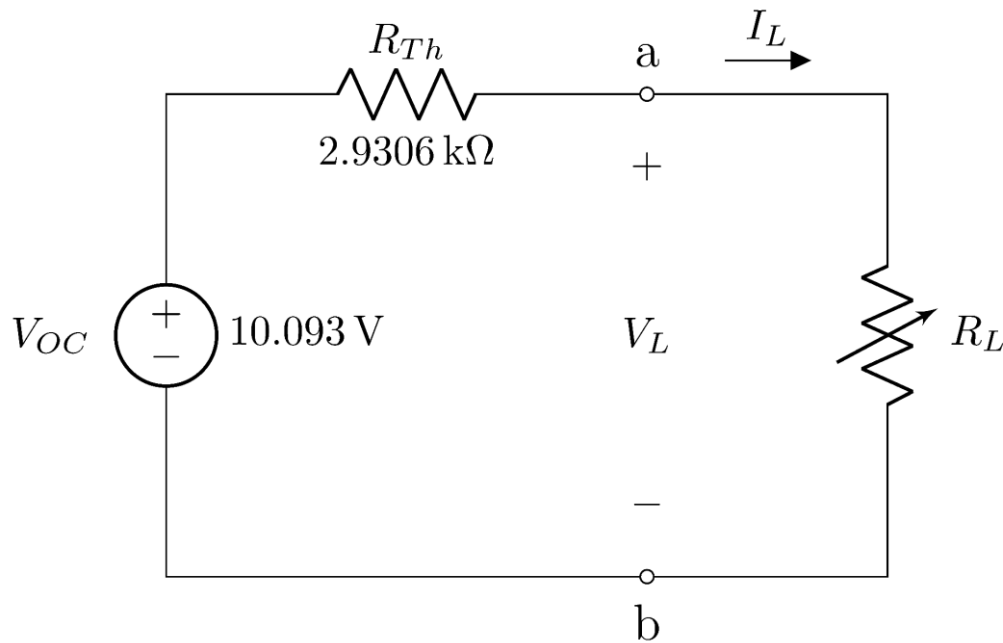
Circuit 7: Thevenin's Equivalent Circuit

**Verifying Maximum Power Transfer Theorem using LTSpice**

➢ Draw the **Circuit 7** in a new schematic in LTspice. Modify the components with their values and name the nodes. Notice that the value of the voltage source is equal to $\mathbf{V_{th}}$ and that of the resistance is equal to $\mathbf{R_{th}}$.
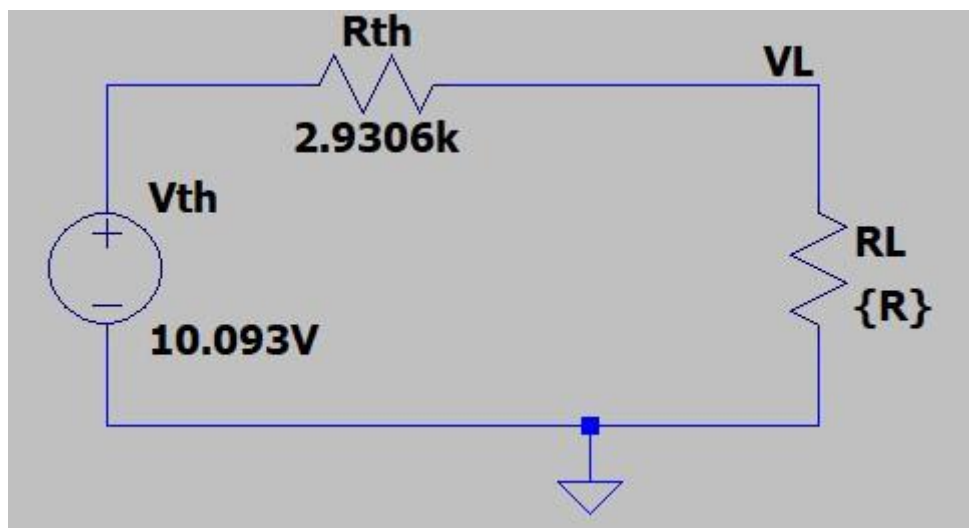
➢ As we have to calculate the power at load resistance $\mathbf{R_L}$ for different values of $\mathbf{R_L}$, we will vary the value of $\mathbf{R_L}$. For this, set the value of $\mathbf{R_L}$ with $\mathbf{\{R\}}$ so that $\mathbf{R}$ can be used as a variable parameter.

➢ To vary the value of R_L we have to do *'Parametric Sweep'* using the variable parameter R which we have set as the value of R_L. To do so, find the *'Spice Directives'* option by *Right-clicking on the schematic Draft Spice Directives* or

find it from the toolbar above.

➢ After clicking the *'Spice Directives'*, the *'Edit Text on the Schematic'* window will appear. Type *'.step param R 200 20k 100'* in the text box and click **OK**. It will generate a command. Place it somewhere in the schematic.
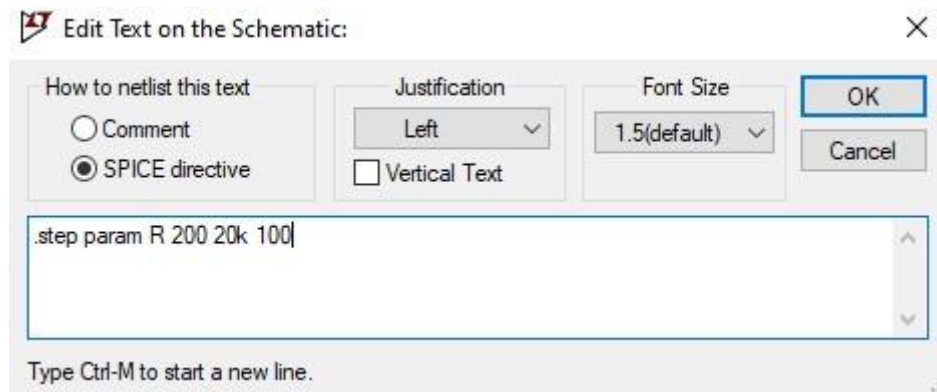

Notice the **'.step'** syntax here. This command causes an analysis to be repeatedly performed while stepping the temperature, a model parameter, a global parameter, or an independent source. Steps may be linear, logarithmic, or specified as a list of values. Here, *'.step param R 200 20k 100'* this command causes the parameter R to increase from **200 Ω** to **20 Ω** with a step size **100** i.e. it will perform simulation for **RL = 200 Ω, (200+100) Ω, (200 +2*100) Ω ….up to 20 K Ω.**

➢ To calculate the power at the load, we will measure the voltage and current at the load (P_L = V_L*I_L). To do so we have to do **'DC operating point'** analysis. Run the simulation in *'DC Operating Point'* analysis mode by following the steps mentioned earlier.

➢ To run the simulation, click *'Run'*. Find the *'Run'* button from the above toolbar or by just Right-clicking on the schematic.

➢ After the simulation 'run' is complete, a blank plot window will appear. *Now Right-click on the blank plot space Add trace Select V_L Type '*' Select I(R_L) OK.*
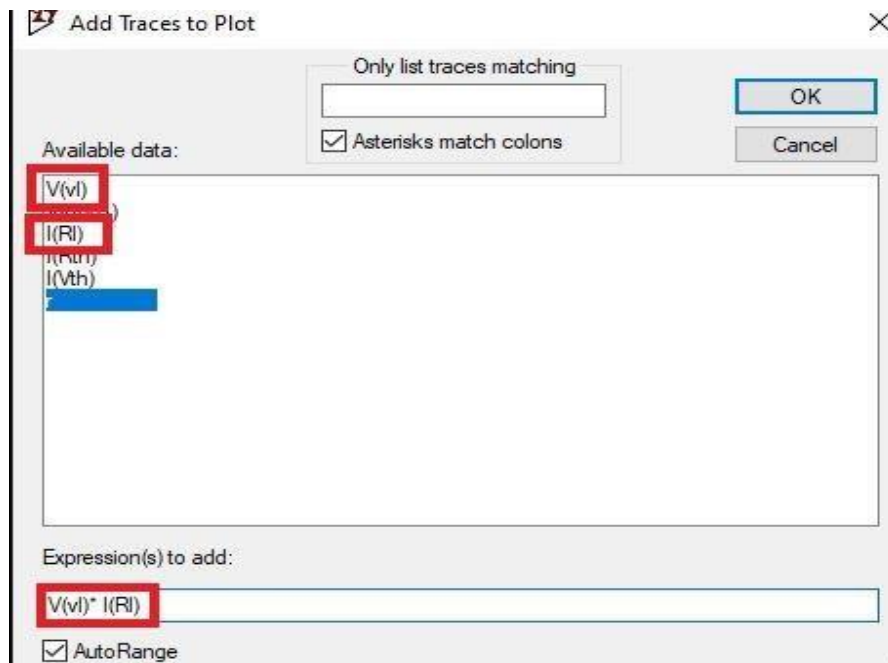
➢ A power vs resistance plot will be generated. On the Y axis, there is the power associated with the load resistance, and on the X axis, there are the values of load resistance R_L. Click on the title of the plot; a cursor will appear. Using this cursor, find the maximum power (peak of the graph) point from the plot and note down the corresponding value of the load resistance R_L from the X axis. The value should be equal to that of the Thevenin resistance.
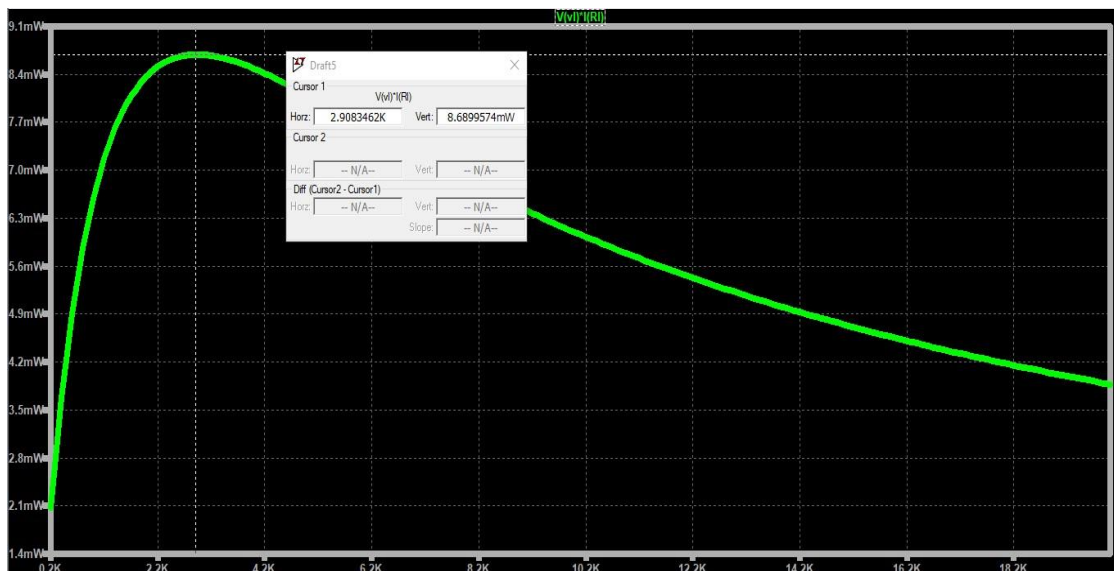
Lab Work: From the plot, Find out the value of R_L for which Maximum Power was achieved and state your opinion on whether Maximum Power Transfer Theorem has been verified or not.
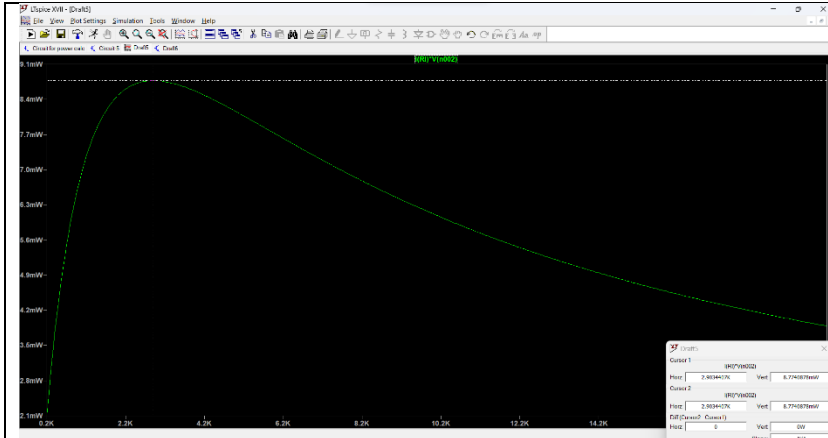
from this plot, we can see that, the value of R_L for which Maximum Power was achieved of R_L is 2.90344 k (ohm).

The Maximum Power Transfer Theorem is a principle used in electrical engineering that states the maximum power from a source circuit to a load occurs when the load resistance is equal to the source resistance.This is achieved by adjusting the resistance of the load to match the internal resistance of the power source. It's a fundamental concept that helps in understanding how to effectively distribute power in complex circuits.

And from the plot attached above, it is clear to see that the highest point of the graph is showing us the maximum power.

Therefore, Maximum Power Transfer Theorem has been verified

Report

1. Answer to questions and Complete the Lab work sections.
2. Save all your .asc and .plt files and make a zip file. You need to submit it with the report.
3. Discussion *[comment on the obtained results and discrepancies]*. Start writing from below the line.

Answer to the question No-3

DISCUSSION:

In this lab, we studied and applied the I-V characteristics of linear circuits and verified
Thevenin's Theorem and the Maximum Power Transfer Theorem using LTSpice software
simulation.

The I-V characteristics of the linear circuits were plotted and analyzed carefully. The results
showed a linear relationship between current and voltage, which supplied a fundamental
understanding of how current and voltage acts in a linear circuit.

Thevenin's Theorem was then verified through simulation. By replacing a complex circuit
with its Thevenin equivalent (a single voltage source and a single series resistor), we were
able to simplify the analysis of the circuit. The I-V characteristics of the original circuit and
its Thevenin equivalent were found to match, confirming the validity of Thevenin's Theorem.
Finally, the Maximum Power Transfer Theorem was verified. This theorem states that
maximum power transfer from a source to a load occurs when the load resistance equals the
source resistance. By adjusting the load resistance in the simulation, we were able to achieve
a condition of maximum power transfer, thus verifying the theorem.
Overall, the lab provided valuable hands-on experience in analyzing linear circuits and
applying key electrical theorems. The use of LTSpice software simulation was instrumental
but lengthy in visualizing circuit behaviors and verifying theoretical concepts. These findings
have significant implications for the design and optimization of electrical circuits.