Brac University

Department of Electrical & Electronic Engineering

Semester Summer-24

Course Number: EEE101L

Course Title: Electrical Circuits I Laboratory

Section: 06

Lab Report

Experiment no. | 05 |
|---|

Name of the experiment: Verification of Thevenin's Theorem and Maximum Power Transfer Theorem (Software Simulation)

Name: Tanzeel Ahmed ID: 24321367

***Group Number:* 02**

Other Group members:

Sl.	*ID*	*Name*
1.	24121083	Abontika Das
2.	24121046	Lamea Akter Shimla
3.	24121238	Tahosin Tasnia Nuha

Electrical Circuits I Laboratory
EEE 101L
Department of Electrical & Electronic Engineering (EEE)
Brac University

Experiment No. 5

Verification of Thevenin's Theorem and Maximum Power Transfer Theorem

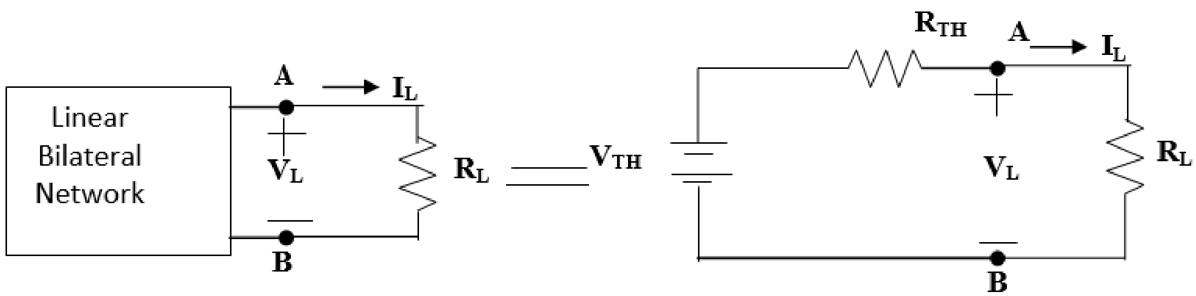
Part-A: Verification of Thevenin's Theorem

1. **Objective:** To verify Thevenin's theorem with reference to a given circuit.

2. **Theoretical Background:** 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 reduces greatly the amount of computations which we have to do each time a change is made. Using Thevenin's theorem, the given circuit, excepting 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.

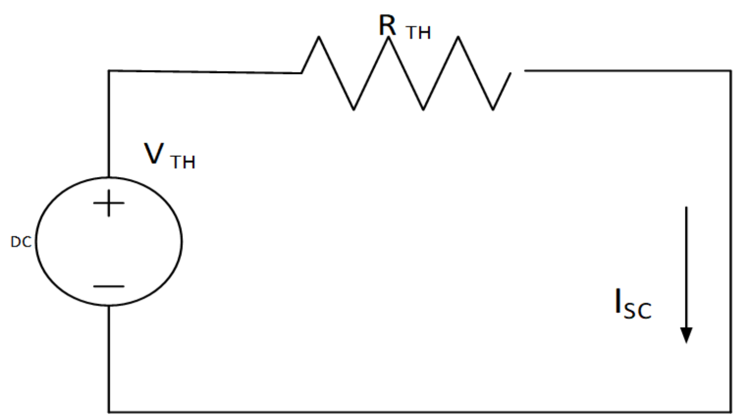
   The Thevenin's theorem states that any two terminal linear bilateral network containing sources and passive elements can be replaced by an equivalent circuit consisting of a voltage source V_{th} in series with resistor R_{th} where

   V_{th} = the open circuit voltage (V_{OC}) at the two terminals A & B.
   R_{th} = the resistance looking into the terminals A and B of the network with all sources removed.



There are several methods for determining Thevenin resistance R_{TH}. An attractive method for determining R_{TH} is: (1) determine the open circuit voltage, and (2) determine the short circuit current I_{SC} as shown in the figure; then

$$R_{TH} = \frac{V_{OC}}{I_{SC}}$$



3. Equipment:

- Resistors
- Multimeter
- DC Power Supply adjusted to 20V DC
- Breadboard

4. Circuit Diagram:

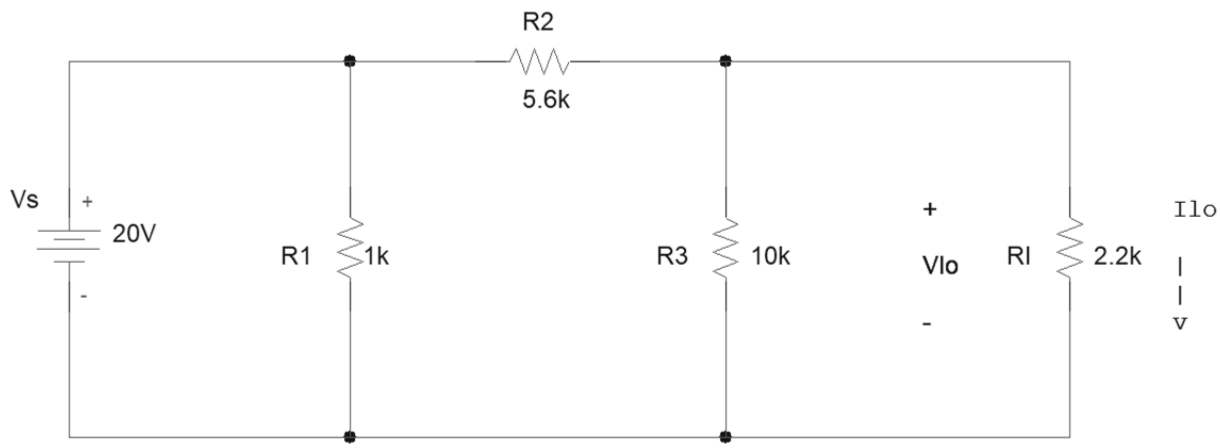


Fig. 1: Original Circuit

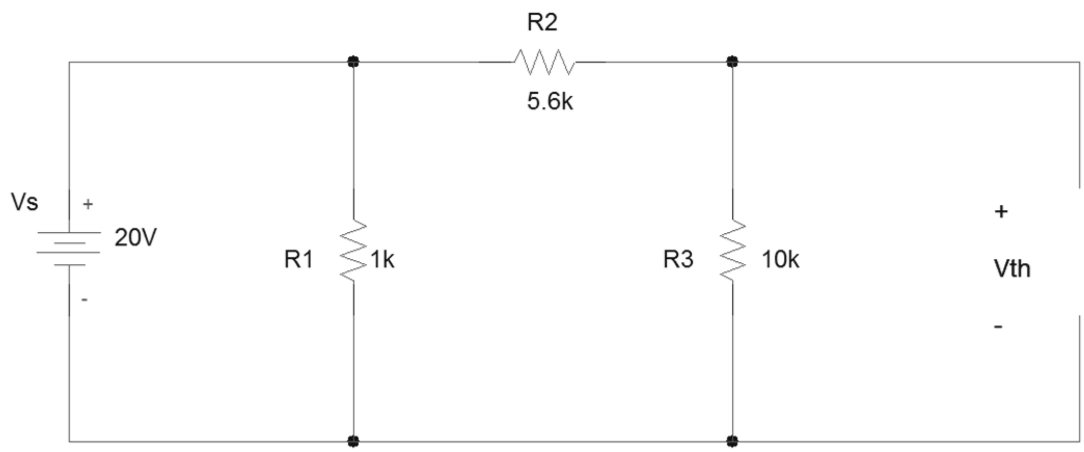


Fig. 2: Circuit for finding V_{th}

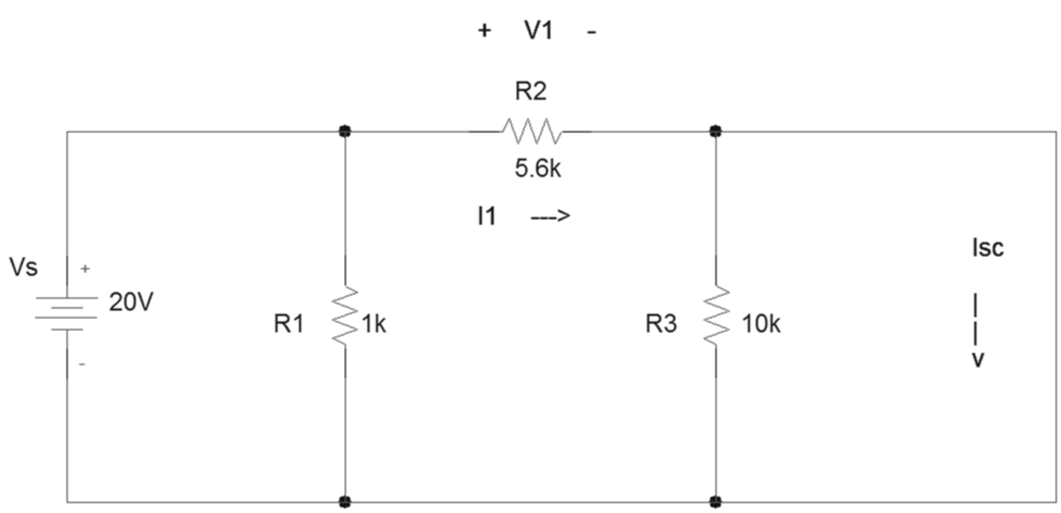


Fig. 3: Circuit for finding I_{sc}

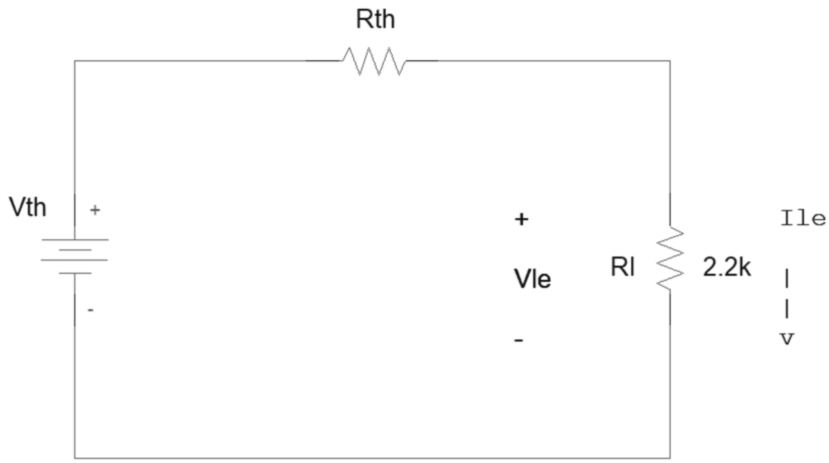


Fig. 4: Equivalent circuit

5. Procedure:

For Original Circuit:

Construct the original circuit as shown in Fig. 1. Connect source voltage V_s = 20V DC from a DC power supply. Measure R_l, V_{lo}, calculate I_{lo} and record the data in the table.

$I_{lo} = V_{lo}/R_l$

Finding V_{th} & R_{th}:

i. Remove the load resistance R_l and find the open circuit voltage. This voltage is Thevenin voltage i.e. V_{th}.
ii. Place a short circuit between the open terminals and find the short circuit current I_{sc}.

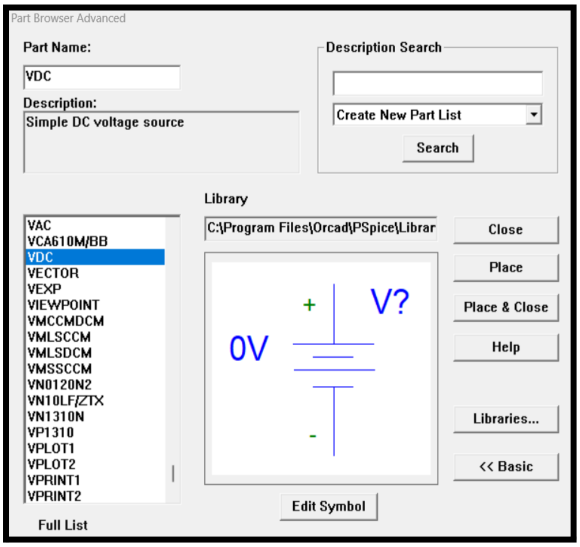
$$R_{th} = V_{th} / I_{sc}$$

$$I_{sc} = I_1 = V_1 / R_2$$

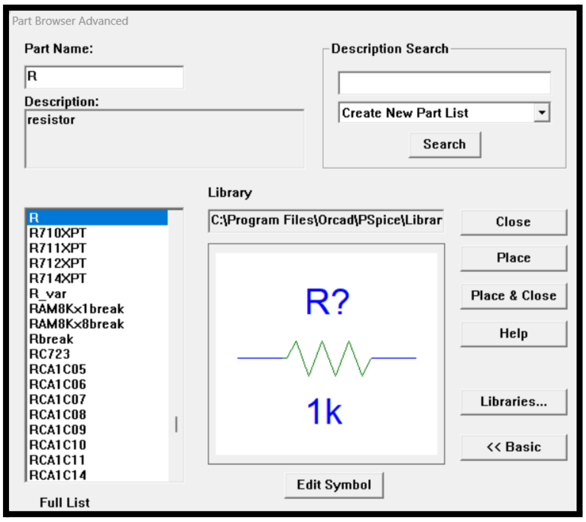
Get back to the circuit in Fig. 2. Deactivate the DC voltage source. Measure the resistance between the open circuit terminals via the multimeter. Verify the earlier value of R_{th}.

For Equivalent Circuit:

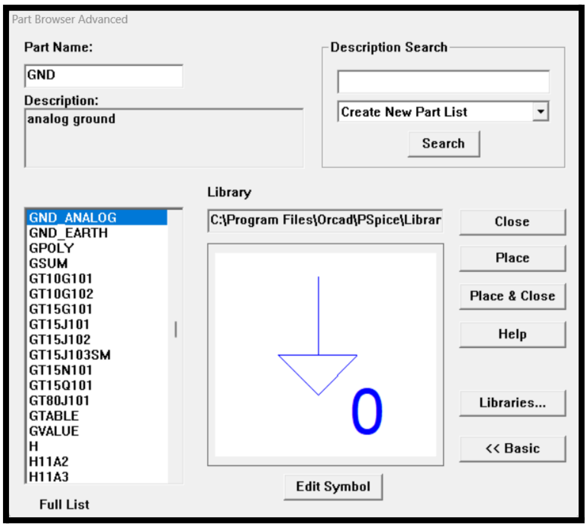
Construct the equivalent circuit as shown in Fig. 4 setting the power supply at V_{th} volts and resistor R_{th} as close as possible to the value found from above. Now measure V_{le} and calculate I_{le}. Compare these values with previous values.



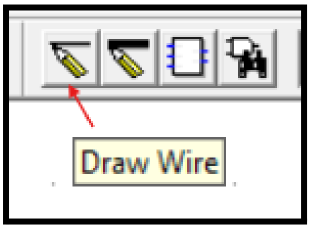
Selection of Voltage Source



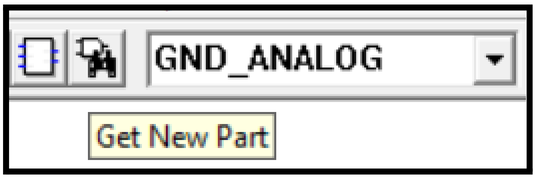
Selection of Resistor



Selection of Ground

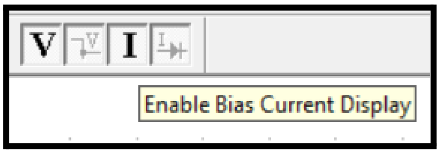


Wire tool

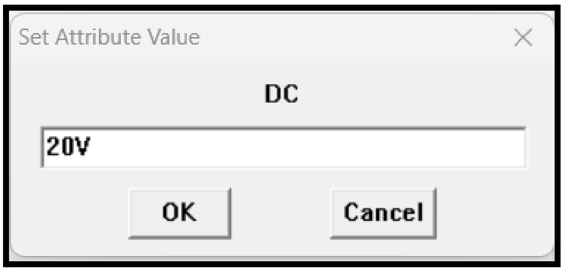


Parts menu

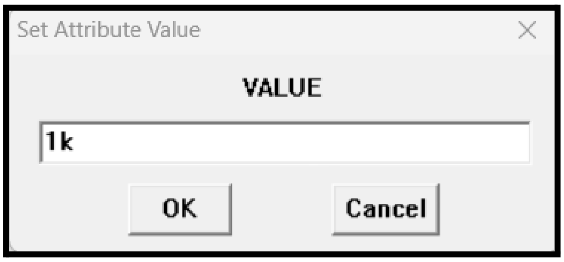
Bias Voltage



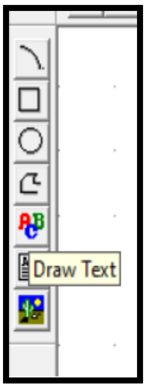
Bias Current



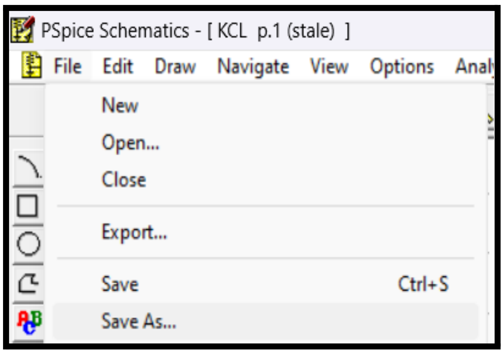
VDC Set (Voltage source value set)



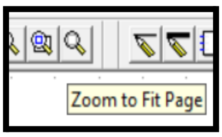
Resistor set (R value set)



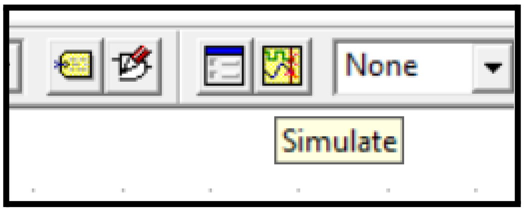
Text tools



File saving



Zoom



Begin Simulation

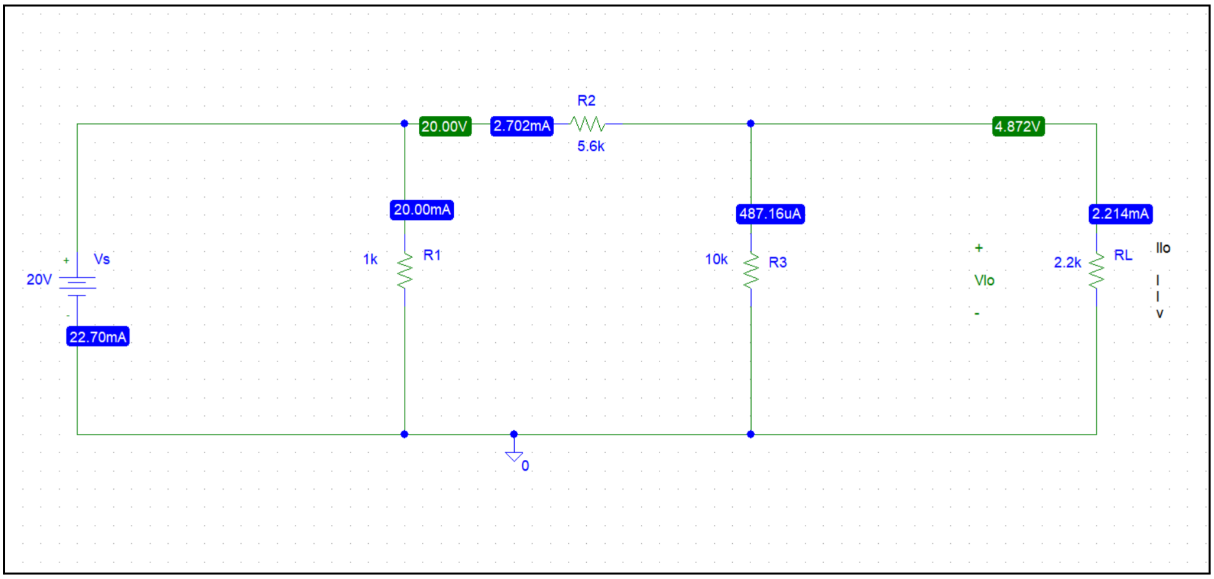


Fig.1: Original Circuit simulated in PSpice Schematics

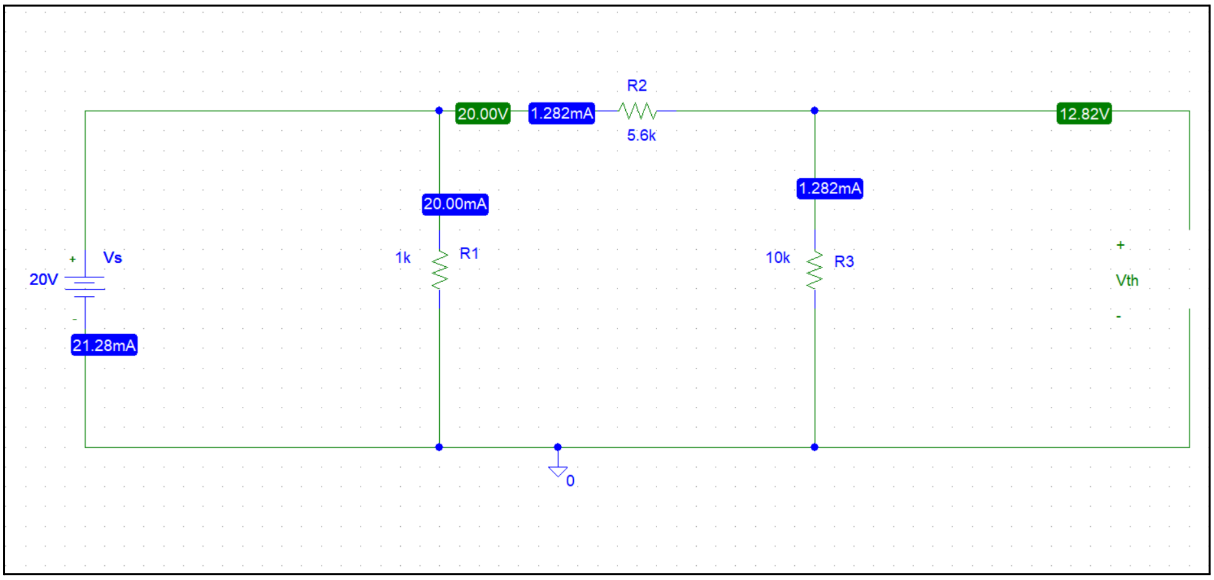


Fig.2: Circuit for finding Vth simulated in PSpice Schematics

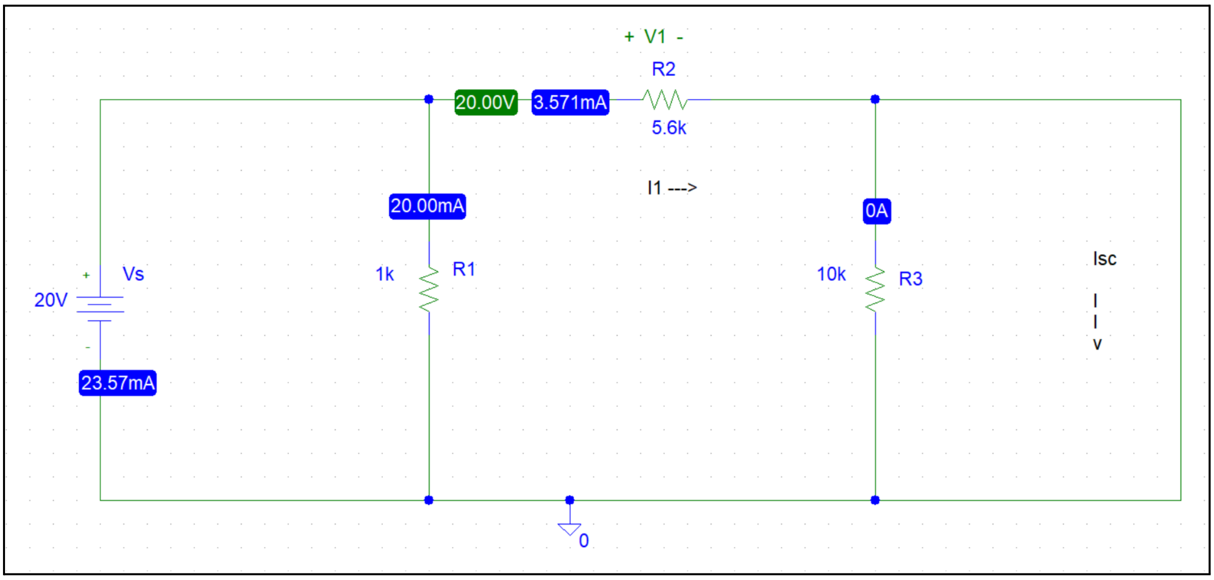


Fig.3: Circuit for finding Isc simulated in PSpice Schematics

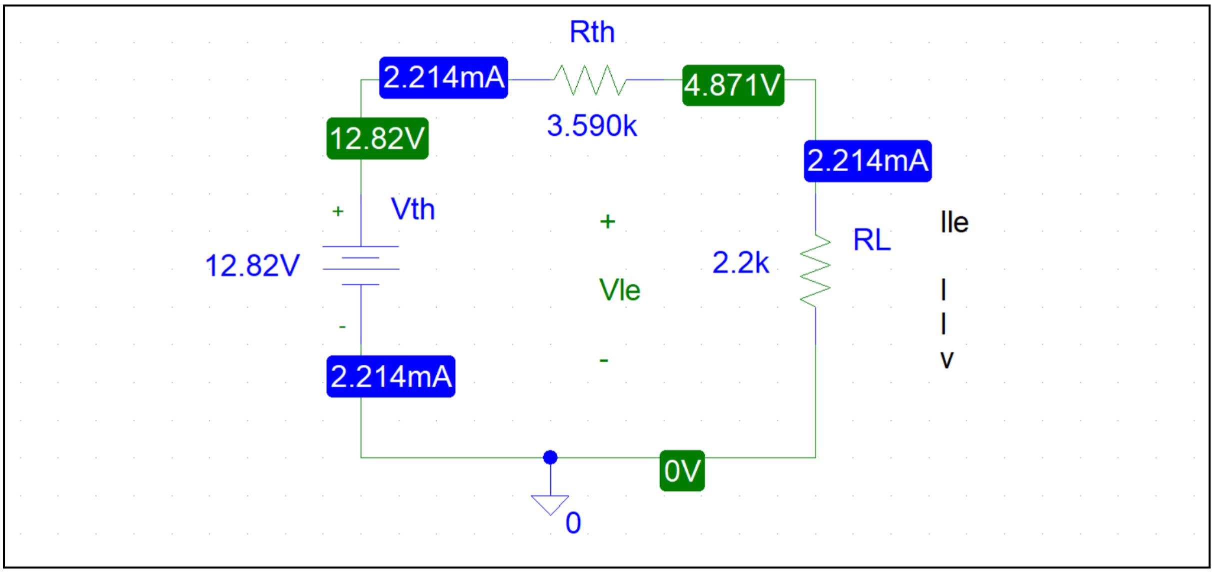


Fig.4: Equivalent circuit simulated in PSpice Schematics

6. Data Table:

Table 1: Data for original circuit

R_l (kΩ)	V_{lo} (V)	I_{lo} (mA)
1	4.872	2.214

Table 2: Data for V_{th}

V_{th} (V)
12.82

Table 3: Data for R_{th}

Indirect Method:

R_2 (kΩ)	V_1 (V)	$I_1 = V_1 / R_2$ (mA)	$I_{sc} = I_1$ (mA)	$R_{th} = V_{th} / I_{sc}$ (kΩ)
5.6k	20	3.571	3.571	3.590

Direct Method:

R_{th} (kΩ)
3.590

Table 4: Data for equivalent circuit

V_{le} (V)	I_{le} (mA)
4.871	2.214

Faculty Signature and Date

Discussion of the software simulation for Part-A

Equipments required:

1. PSpice Schematics software
2. Suitable device (PC or Laptop)

Simulation procedure:

Circuit figure.1

1. Open PSpice Schematics software.
2. Open the parts menu (click on the icon).
3. Search the necessary parts for making a series-parallel circuit (VDC, GND, R).
4. Place the parts on designated places following the provided diagram and close the parts menu.
5. Using the wire tool connect all the parts in the circuit.
6. Set the values of all the parts.
7. Rename all the parts for easier identification (VDC=Vs)
8. Use the Draw Text and Text Box tool to mark necessary information.
9. Enable Bias Voltage and Current display.
10. Use the zoom to fit page tool.
11. Save the file with a suitable name.
12. Begin circuit simulation.
13. Attach a screenshot of the circuit in the report document.
14. Fill out table 1 data boxes with values of R_l, V_{lo} and I_{lo}.

Circuit figure.2

1. Click the RL resistor and highlight it (It will be red when highlighted) and press the delete key on the keyboard to remove it.
2. Use the Draw Text and Text Box tool to mark necessary information.
3. Run the simulation again.
4. Fill out table 2 with the value of Vth.

Circuit figure.3

1. Use the wire tool to make a short-circuit connection from R2 to the end of R3.
2. Use the Draw Text and Text Box tool to mark necessary information.
3. Run the simulation again.
4. Fill out the table 3 with necessary values and calculate Rth=V_{th}/I_{sc}

Circuit figure.4

1. Open a new file in PSpice Schematics
2. Open the parts menu (click on the icon).
3. Search the necessary parts for making the Equivalent Circuit (VDC, GND, R).

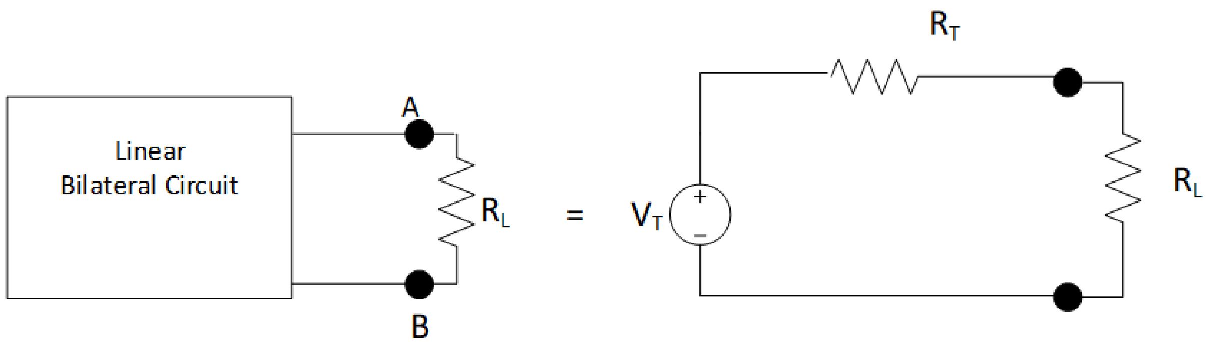
4. Place the parts on designated places following the provided diagram and close the parts menu.
5. Using the wire tool connect all the parts in the circuit.
6. Set the values of all the parts.
7. Rename all the parts for easier identification (VDC=Vth)
8. Use the Draw Text and Text Box tool to mark necessary information.
9. Enable Bias Voltage and Current display.
10. Use the zoom to fit page tool.
11. Save the file with a suitable name.
12. Begin circuit simulation.
13. Attach a screenshot of the circuit in the report document.
14. Fill out table 4 with the value of V_{le} and I_{le}

If I_{le} is equal to I_{lo} , Thevenin's Theorem has been verified for the particular circuit.

Part-B: Maximum Power Transfer Theorem

1. Objective: The objective of this experiment is to verify maximum power transfer theorem.

2. Theoretical Background: The maximum power transfer theorem states that a resistive load will receive maximum power when its total resistive value is exactly equal to the Thevenin's resistance of the network as seen by the load.

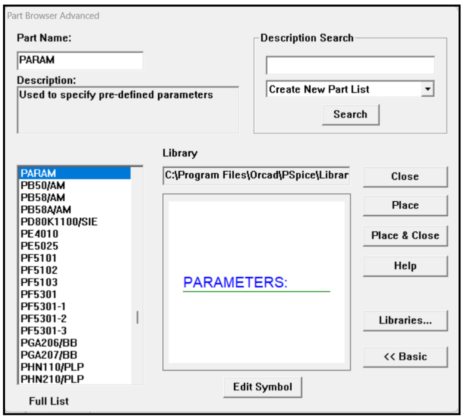


We know that any linear circuit terminated with a load R_l can be reduced to its Thevenin's equivalent. Now according to this theorem, the load R_l will receive maximum power when

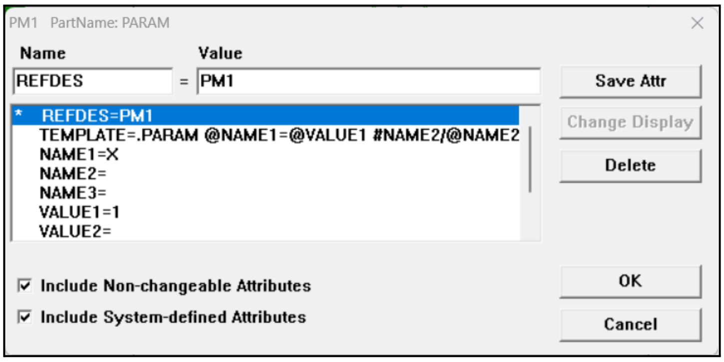
$R_l = R_{th}$

3. Equipment:

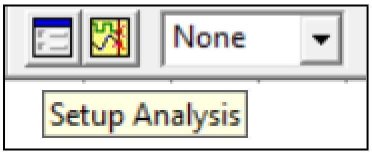
- Digital Multimeter
- DC power supply adjusted to 20V DC
- Resistor and potentiometer
- Wires



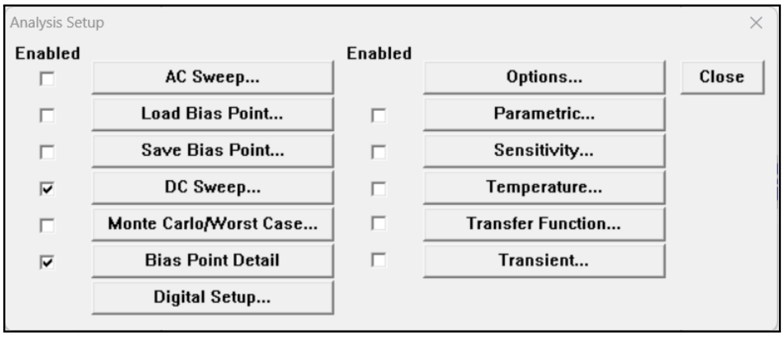
Parameter in parts menu



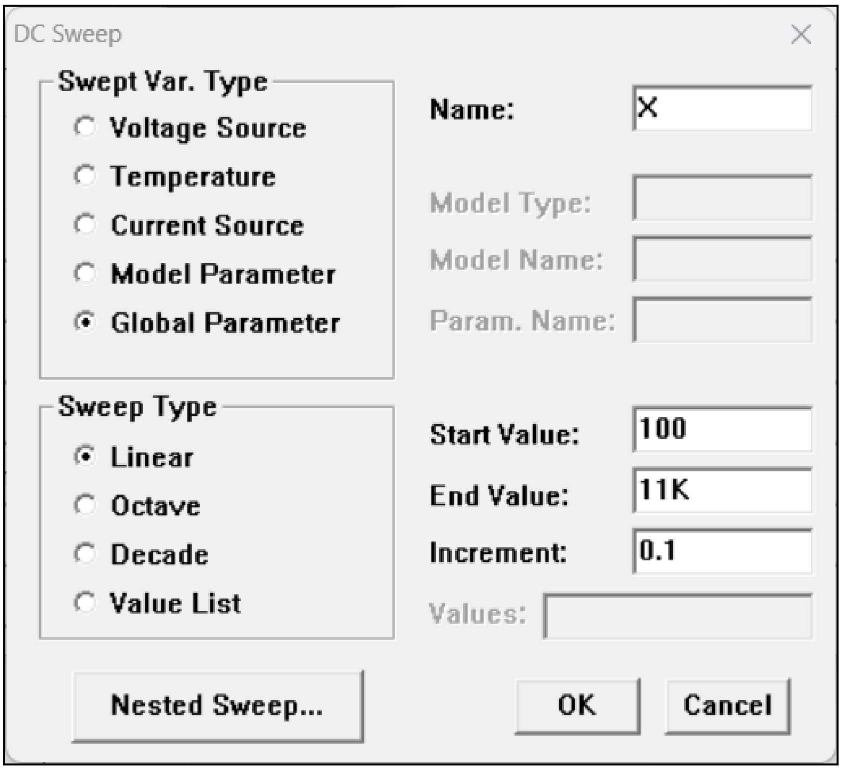
Setting the value of Parameter



Setup Analysis Icon



Setup Analysis Menu (DC Sweep ON)



Global Parameter Variable and it’s value setup with Increment

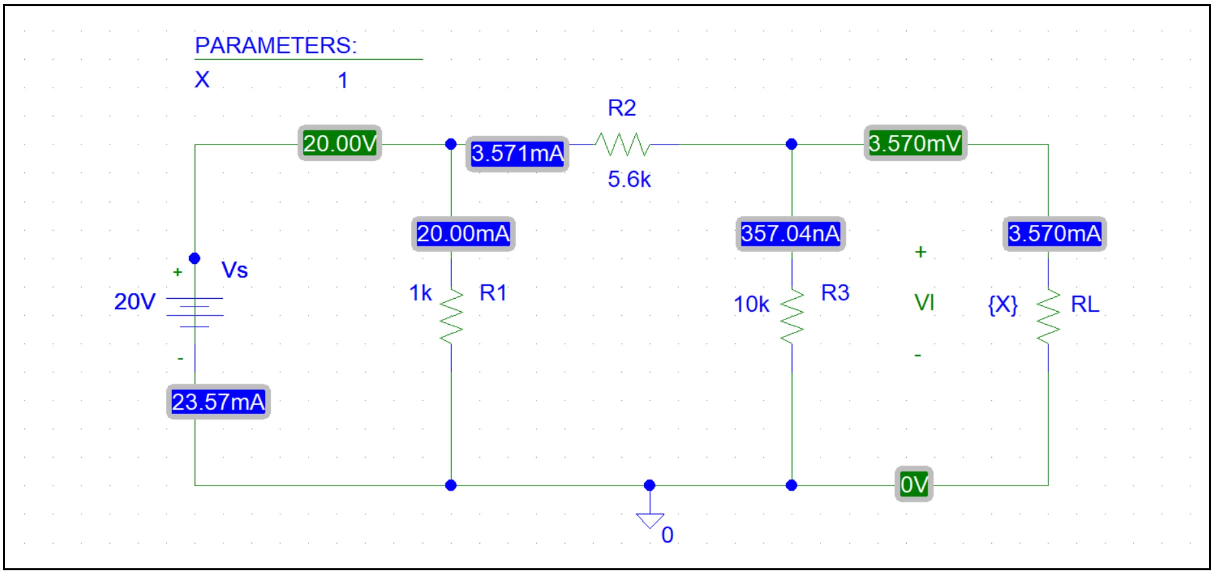
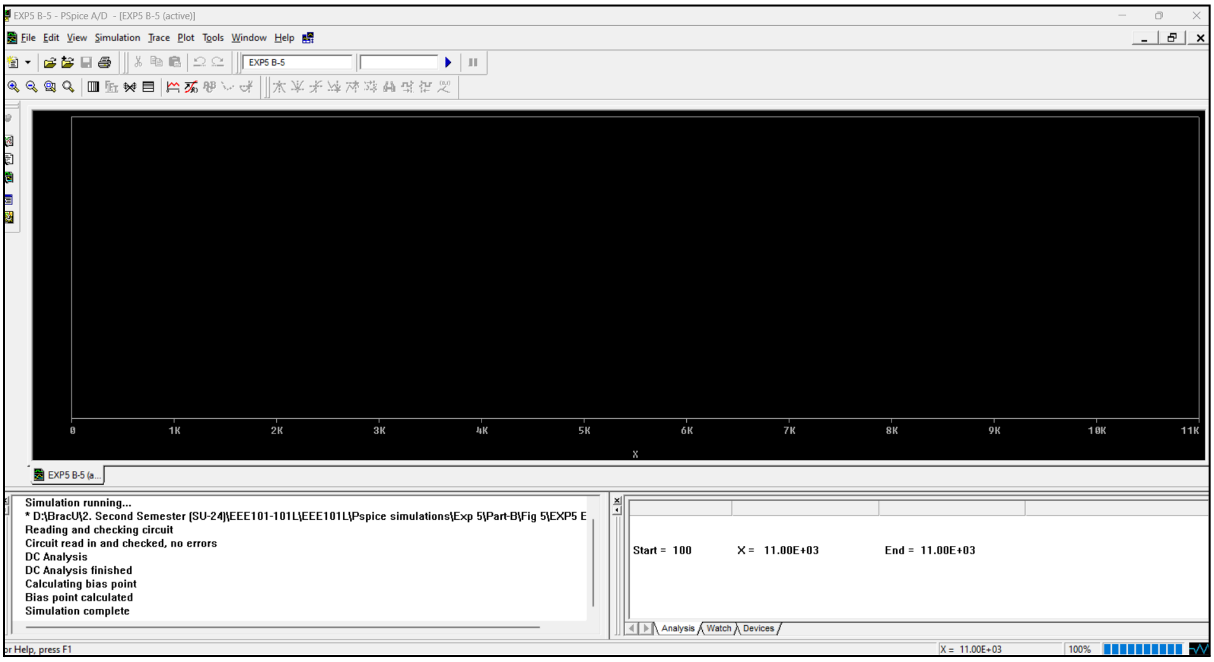
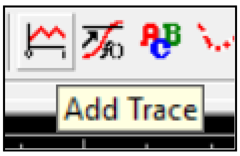


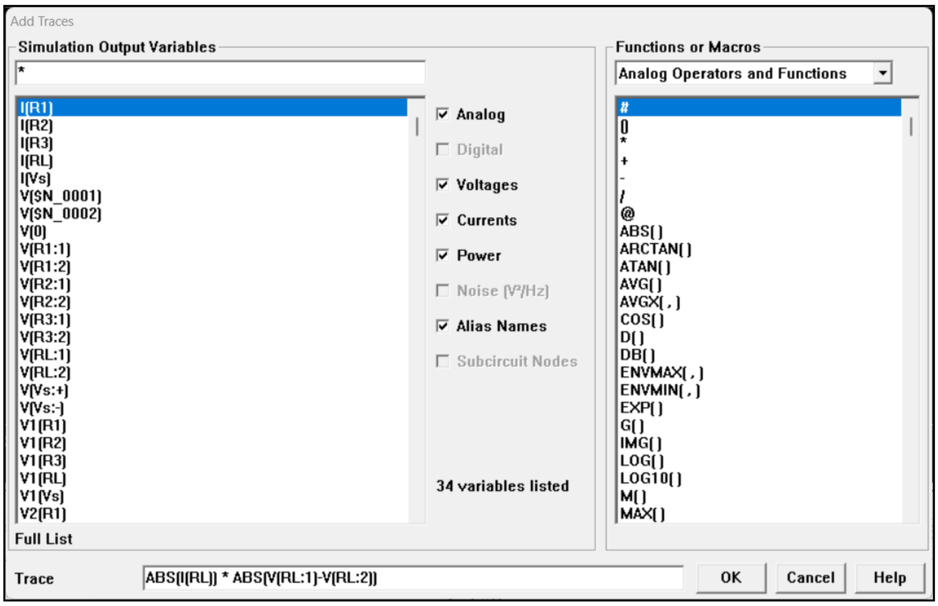
Fig.5: Circuit for verifying Maximum Power Transfer Theorem



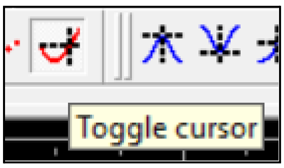
New window with graphing interface



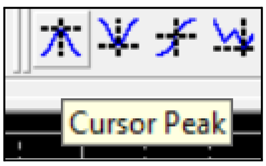
Add trace tool



Setting the trace value



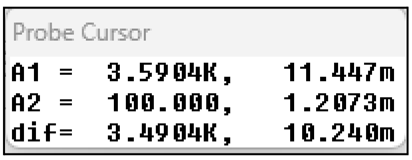
Toggle Cursor tool



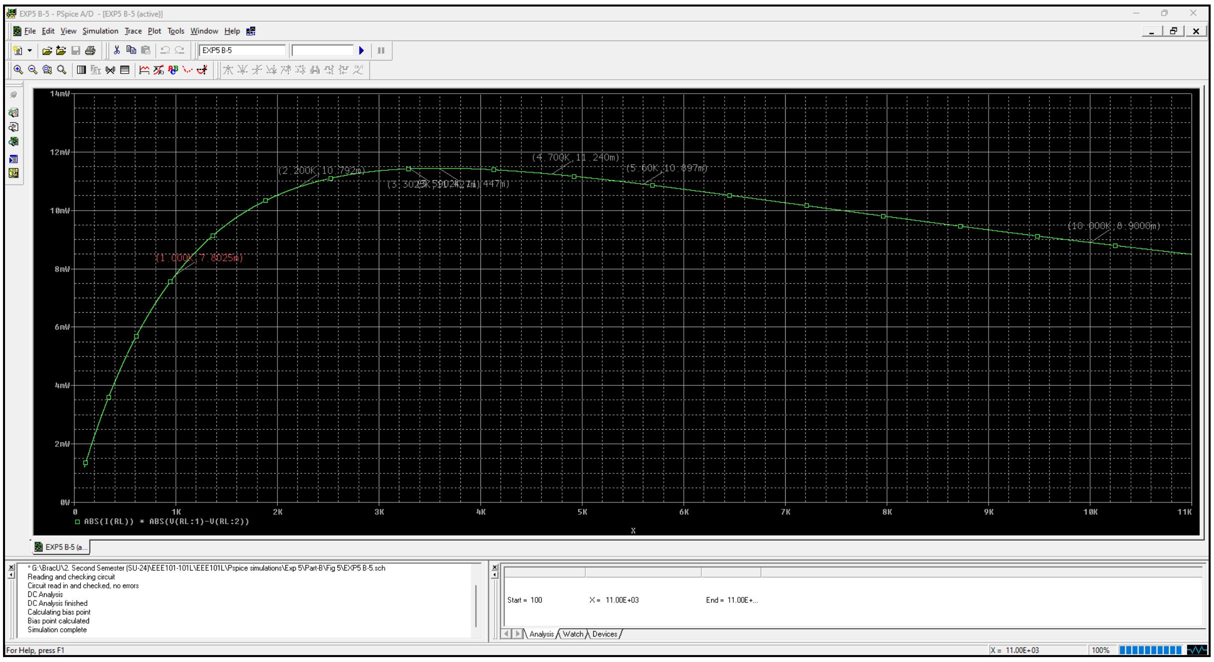
Cursor Peak Tool

Mark Label Tool

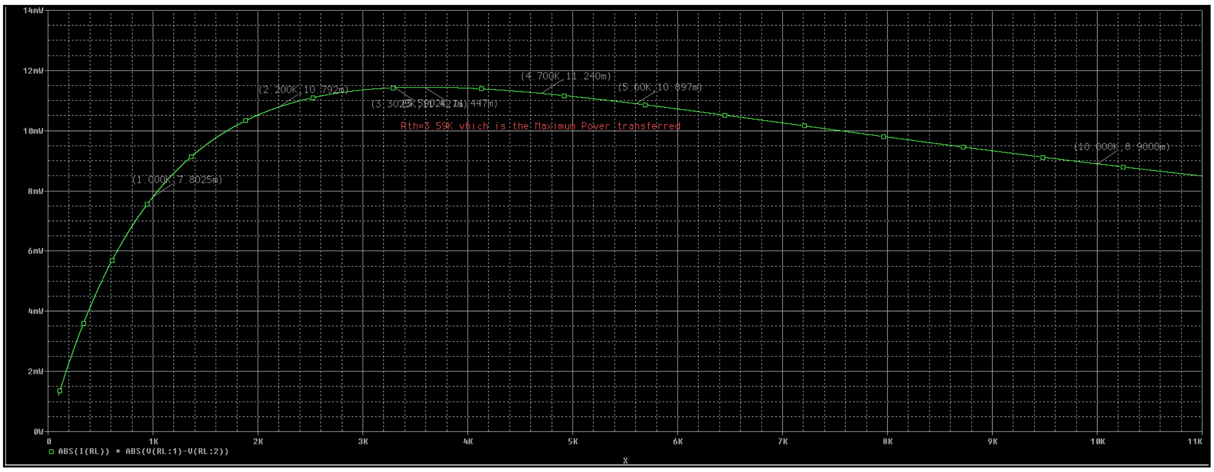
Text Label Tool



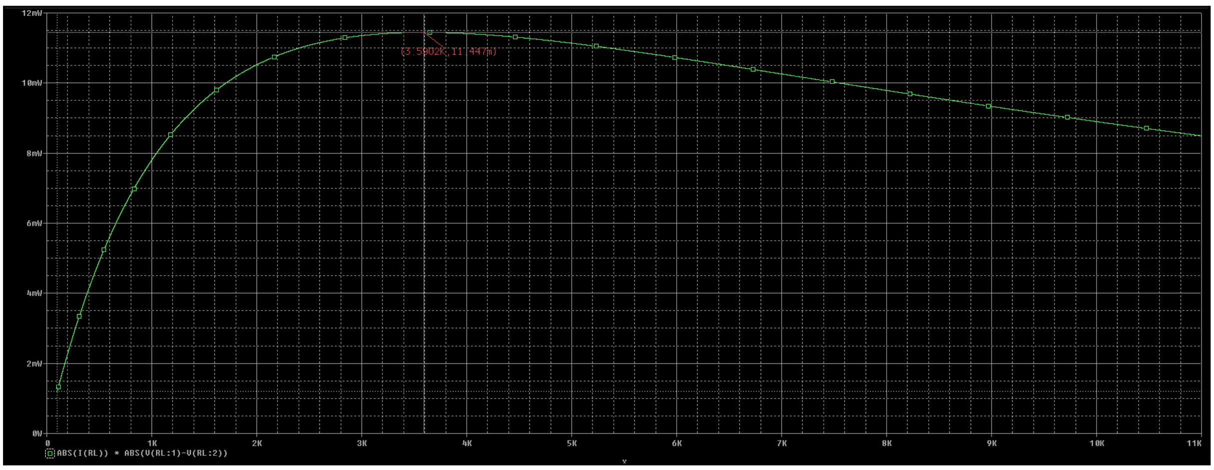
After enabling toggle cursor, Probe Cursor window will open



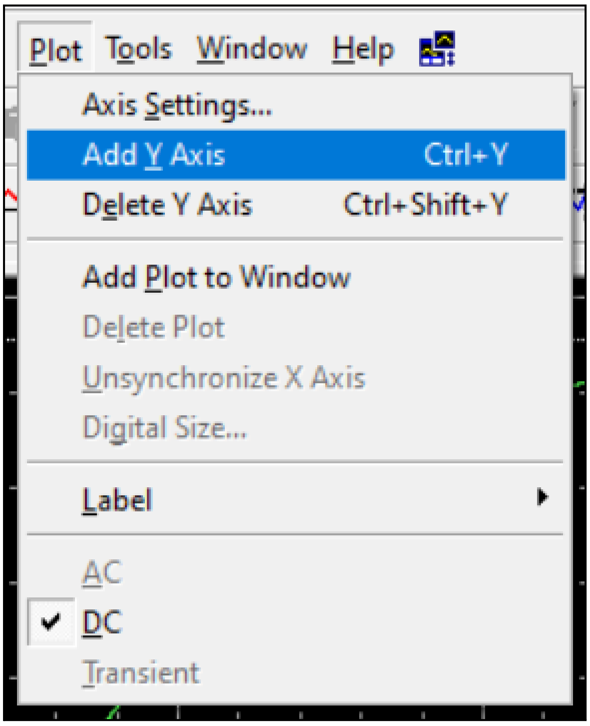
Completed graph on the interface with all the resistor points



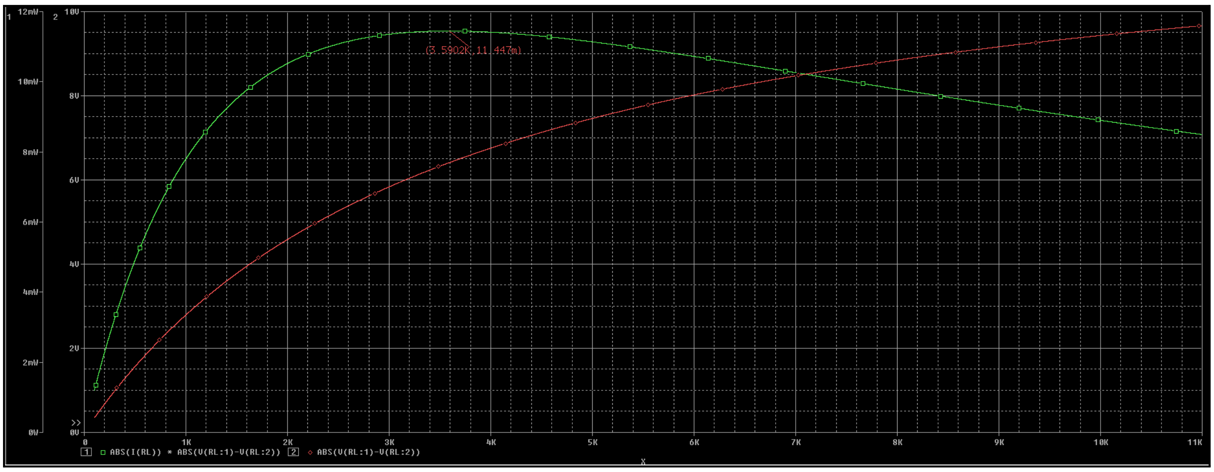
Screenshot of the graph with all the resistor points and text added



Graph with only peak value marked (Rth=3.59-ohm, Pl=11.448mW)



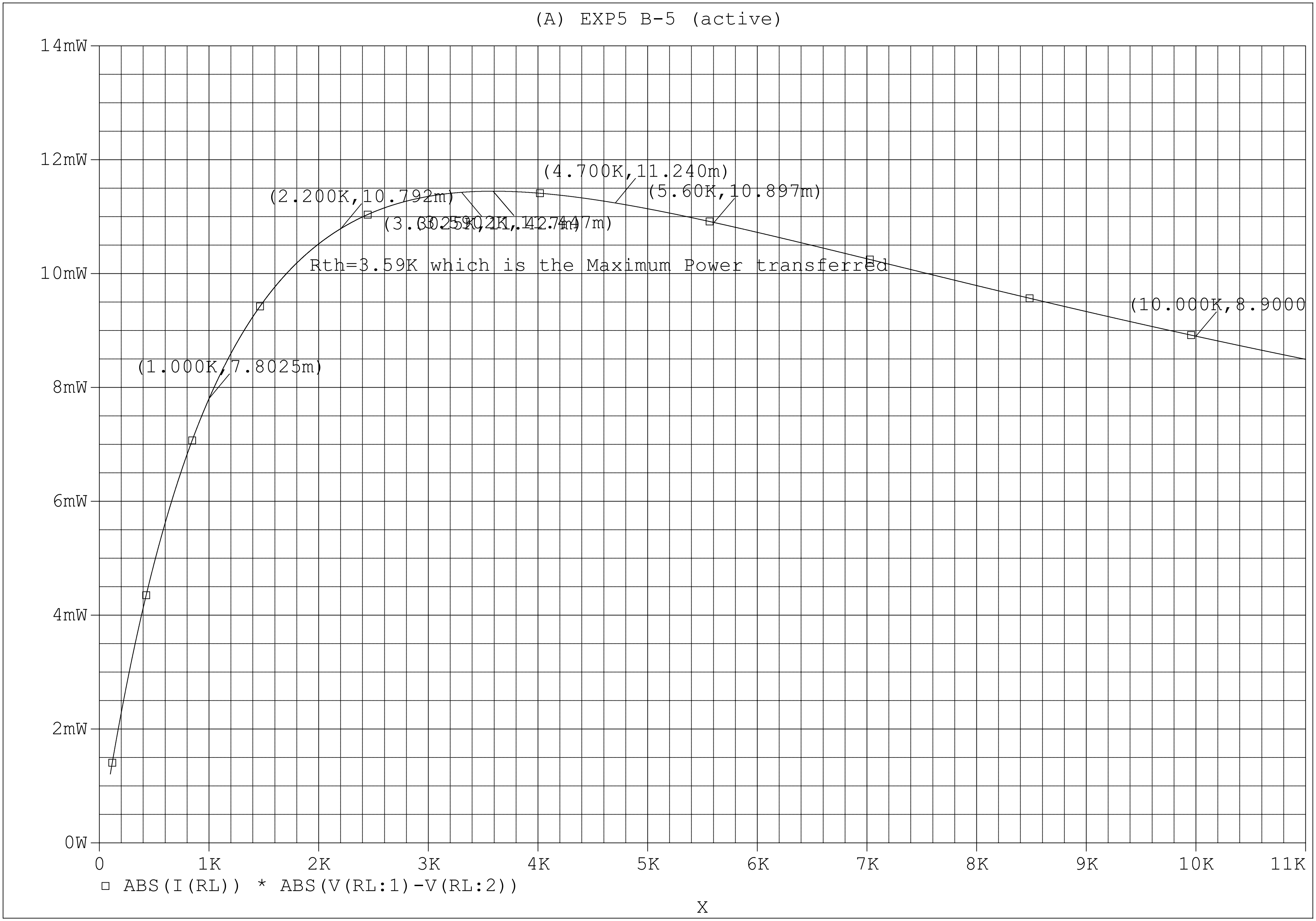
Plot Menu>Adding another Y Axis



Graph with Power and Voltage across the Y-axis

Graph with all the variable points marked

* G:\BracU\2. Second Semester (SU-24)\EEE101-101L\EEE101L\Pspice simulations\Exp 5\Part-B\Fig 5\EXP5 B-5...
Date/Time run: 09/20/24 17:25:56 Temperature: 27.0



Graph with only the peak value marked

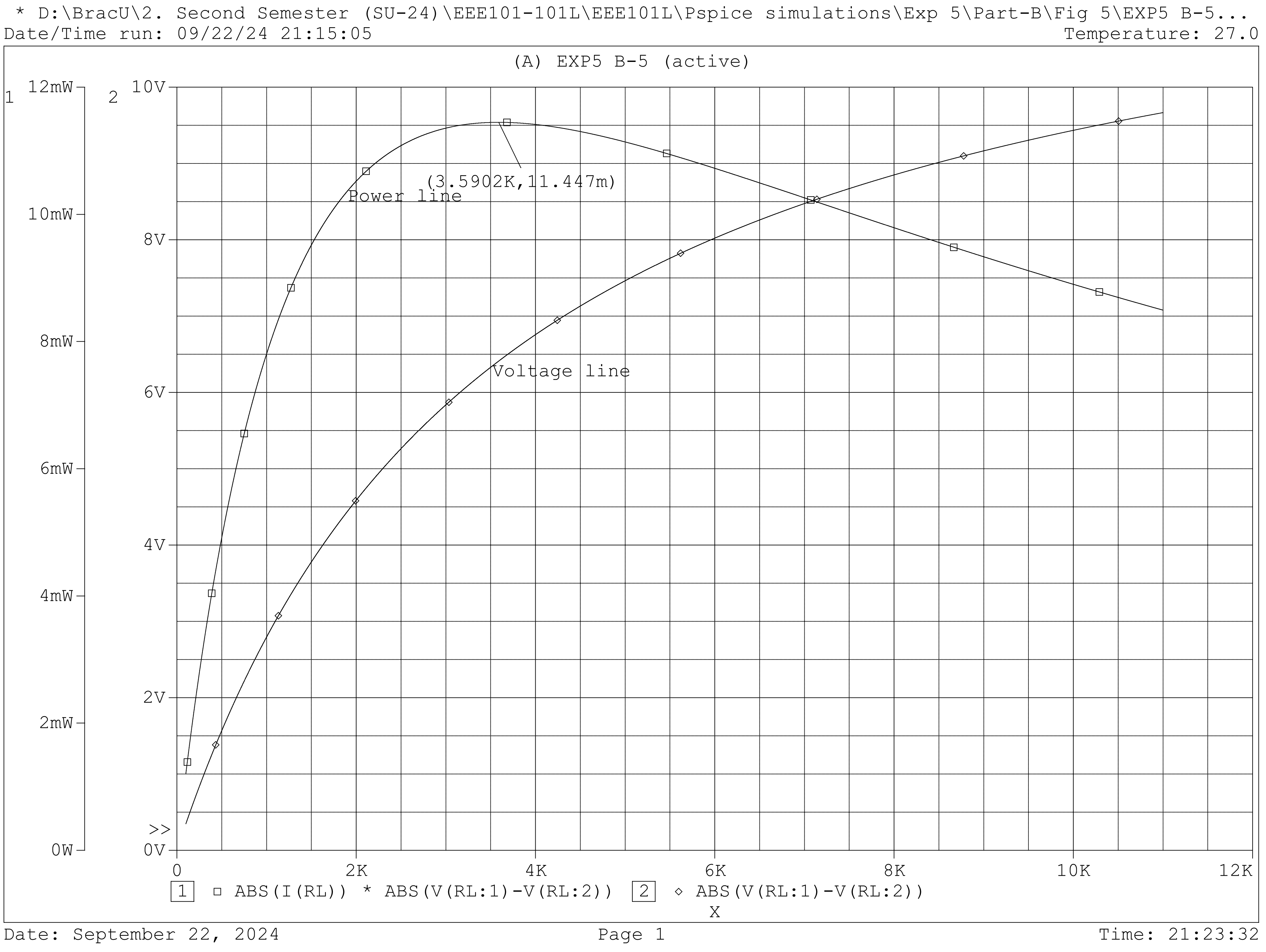
* G:\BracU\2. Second Semester (SU-24)\EEE101-101L\EEE101L\Pspice simulations\Exp 5\Part-B\Fig 5\EXP5 B-5...

Date/Time run: 09/20/24 17:57:22 Temperature: 27.0

A1:(3.5902K,11.447m) A2:(100.000,1.2073m) DIFF(A):(3.4902K,10.240m)

Date: September 20, 2024 Page 1 Time: 18:29:38

Graph with Power and Voltage values



4. Circuit Diagram:

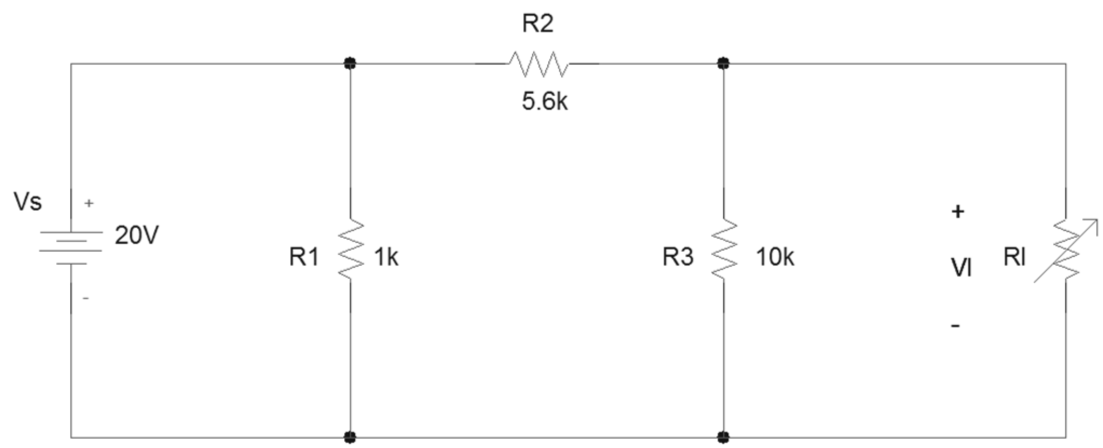


Fig. 5: Circuit for verifying maximum power transfer theorem

5. Procedure:

i. Set up the circuit as shown in Fig. 5.
ii. Use the following values of R_l.

1kΩ, 2.2kΩ, 3.3kΩ, R_{th}, 4.7kΩ, 5.6kΩ and 10kΩ

Set R_{th} up using potentiometer. Measure R_l, V_l and calculate $P_l = V_l^2 / R_l$. The power shall be maximum for condition where the $R_l = R_{th}$.

6. Data Table:

R_l (kΩ)	V_l (V)	$P_l = V_l^2 / R_l$ (mW)
1	2.793	7.80
2.2	4.872	10.789
3.3	6.141	11.427
3.59	6.411	11.448 Rth
4.7	7.269	11.242
5.6	7.813	10.90
10	9.434	8.90

Faculty Signature and Date

<u>Discussion of the software simulation for Part-B</u>

Equipments required:

1. PSpice Schematics software
2. Suitable device (PC or Laptop)

Simulation procedure:

<u>Circuit construction and simulation start:</u>

1. Open PSpice Schematics software.
2. Open the parts menu (click on the icon).
3. Search the necessary parts for making a (VDC, GND, R).
4. Place the parts on designated places following the provided diagram and close the parts menu.
5. Using the wire tool connect all the parts in the circuit.
6. Rename all the parts for easier identification (VDC=Vs, R4=RL)
7. Set the values of all the parts according to the diagram and only set the value of RL to {X}
8. Use the Draw Text and Text Box tool to mark necessary information.
9. Enable Bias Voltage and Current display.
10. Open the parts menu again, search for PARAMETER and place it above the circuit.
11. Double click PARAMETER to set its value.
12. After setting the right values, click Setup Analysis on the toolbar and to open its menu.
13. Enable DC sweep.
14. Open the DC Sweep menu, enable Global Parameter and set the necessary values.
15. Use the zoom to fit page tool.
16. Save the file with a suitable name.
17. Begin circuit simulation.
18. Attach a screenshot of the circuit in the report document.
19. Fill out the data tables with necessary information (Voltage and current through each component).

<u>Graphing window:</u>

1. After the simulation has been completed, a new window will open which is a graphing interface for the particular circuit.
2. Open the Add Trace menu from the toolbar.
3. Set the trace value to ABS(I(RL)) * ABS(V(RL:1)-V(RL:2)) and press OK
4. The graph of maximum power transfer will appear on screen.
5. Enable Toggle Cursor from the toolbar.

6. After enabling it, a small window will appear which will show the value of a certain point in the graph where the cursor is placed.
7. Use the Cursor Peak Tool to auto point the highest area (Pl=11.448mW) in the graph which will be the point where the Maximum Power Transfer theorem is verified.
8. Mark the point using the Mark Label tool.
9. Now using the cursor and Mark Label tool, point out all the other resistor values in the graph (1k, 2.2k, 3.3k, 4.7k, 5.6k and 10k)
10. Open the plot menu and add another Y-axis.
11. Use the add trace menu again and this time set the value to ABS(V(RL:1)-V(RL:2)) and press OK
12. The voltage graph will appear alongside the graph of maximum power transfer.
13. Use the Text Label tool to write necessary information on the graph.
14. Take screenshots of important steps and attach them in the lab report.
15. Use the print to PDF option in the Print menu (Files>Print>Microsoft Print to PDF) to make a PDF copy of the graph and add it to the report.

<u>Data Table:</u>

Take notes of the variable resistor points, their respective voltages and the values of Pl on each segment from the graph and fill out the data table. The power shall be maximum for the condition Rl=Rth which will verify the Maximum Power Transfer theorem.