Ryerson University

Department of Electrical, Computer, & Biomedical Engineering
Faculty of Engineering & Architectural Science

ELE 202
Electric Circuit Analysis
LAB COVER PAGE for **Part I** submission.

Lab #:	4	**Lab Title:**	Circuit theorems and the superposition Principle

Last Name:	Malik
First Name:	Hamza

Student #:	501112545
Signature:	Hamza

Section #:	22	
Submission date and time:	wed march 9	11:15pm
Due date and time:	Thurs march 10	2pm

Document submission for Part I:

- A completed and signed "COVER PAGE – **Part I**" has to be included with your submission. The report will not be graded if the signed cover page is not included.
- Your completed handwritten pages of **Section 4.0** should be scanned (via a scanner or phone images), together with the required MultiSIM images. **Note:** *MultiSIM results must be generated using the Department's licensed version of MultiSIM, and the captured screenshots should show your name (at the center-top) and the timestamp (at the bottom-right corner of your screen).*
- Collate and create a *.pdf* or *.docx* file of the above, and upload it via D2L **any time prior to the start of your scheduled lab**. Upload instructions are provided on D2L.

Zero marks will be assigned for the entire lab if this Part I is not submitted prior to your scheduled lab.

****By signing above, you attest that you have contributed to this submission and confirm that all work you have contributed to this submission is your own work. Any suspicion of copying or plagiarism in this work will result in an investigation of Academic Misconduct and may result in a "0" on the work, an "F" in the course, or possibly more severe penalties, as well as a Disciplinary Notice on your academic record under the Student Code of Academic Conduct, which can be found online at: www.ryerson.ca/senate/current/pol60.pdf.***

ELE 202
Laboratory #4

Circuit Theorems and the Superposition Principle

1.0 INTRODUCTION

A complex linear circuit network often consists of combinations of linear elements, linear dependent sources and linear independent sources. Circuit theorems (and related procedures) can be used to reduce such complex circuits to simpler ones, thereby making the circuit analysis that much simpler using *nodal* voltage or *mesh* current techniques.

- **Thevenin's** and **Norton's** theorems allow the isolation of a portion of a circuit network while the remaining portion of the network is replaced by an equivalent network. Related to these theorems is the ***Source Transformation*** technique for simplifying circuits, which is the process of replacing a voltage source in series with a resistor, by a current source in parallel with a resistor, and *vice versa*.

- **Maximum Power Transfer** theorem states that the maximum power gets delivered (by the source) to the load when the load resistance ($\mathbf{R_L}$) equals the Thevenin's resistance ($\mathbf{R_{Th}}$) as seen at the terminal of the load.

The ***Principle of Superposition*** is a conceptual aid that can be very useful in visualizing the behavior of a circuit or network system containing multiple sources. This principle states that when a number of independent input sources are applied simultaneously to a linear network, the response is the sum of the responses due to *each* input acting alone. Hence, the Superposition principle helps simplify the analysis of circuits with multiple independent sources by allowing the voltage across (or current through) any element to be obtained by adding algebraically all the individual voltages (or currents) caused by *each* independent source acting *alone*, with all other independent voltage sources replaced by *short-circuits* and all other independent current sources replaced by *open-circuits*.

Note: A ***two-terminal network*** is defined when interconnections of circuit elements *inside a box* have only two accessible terminals for connection to other networks.

References: (i) Course Textbook: "*Fundamentals of Electric Circuits*" by C. K. Alexander and M. N. O. Sadiku; and (ii) "*Linear Circuit Analysis*" by R. A. DeCarlo and P-M. Lin.

2.0 OBJECTIVES

- To analyze and examine the *Thevenin*'s equivalent circuit of a *two-terminal* source network, and verify the *maximum power transfer* theorem; and
- To analyze and experimentally verify the *Superposition Principle*.

3.0 REQUIRED LAB EQUIPMENT & PARTS

- Digital Multimeter (DMM) and Power Supply
- ELE202 Lab Kit:- various components, breadboard, wires and jumpers.

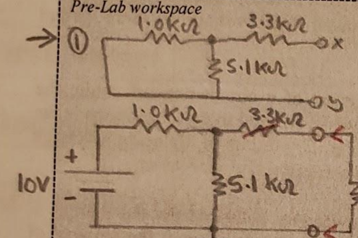
4.0 PRE-LAB: ASSIGNMENT

(a) *Thevenin Equivalent Circuit*

(i) For the circuit shown in **Figures 2.0a** and **2.0b**, consider the input D.C. source voltage, **E = 10 V**. **(1)** Determine the Thevenin's equivalent circuit of the two-terminal source network in **Figure 2.0a** by finding the Thevenin voltage, $\mathbf{V_{Th}}$ and the Thevenin resistor, $\mathbf{R_{Th}}$ as shown in **Figure 2.0c**; **(2)** referring to circuits in **Figure 2.0b**, use either nodal-voltage or mesh-current technique to find the respective open-circuit voltage, $\mathbf{V_{OC}}$ and short-circuit current, $\mathbf{I_{SC}}$; and **(3)** use the Thevenin equivalent circuit model in **Figure 2.0c** to show that $\mathbf{V_{Th} = V_{OC}}$, and $\mathbf{R_{Th} = V_{OC}/I_{SC}}$. Show all your analysis below, and record your results in **Table 2.0**.

Pre-Lab workspace

①



$R_{Th} = (5.1k\Omega \parallel 1.0k\Omega) + 3.3k\Omega$

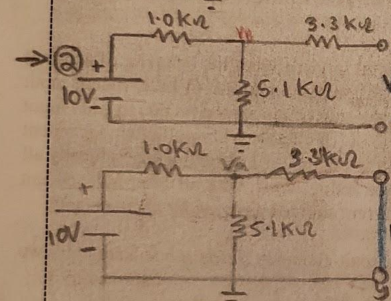
$= \left(\frac{5.1 \times 1.0}{5.1 + 1.0}\right) + 3.3$

$R_{Th} = 4.136\,k\Omega$

$V_{Th} = 10 \times \frac{5.1}{6.1}$

$V_{Th} = 8.361V$

②



$V_a = V_{Th} = V_{OC}$

KCL @ node Va:

$\left(\frac{V_a - 10}{1}\right) + \left(\frac{V_a}{5.1}\right) + 0 = 0$

$V_a\left(\frac{1}{1} + \frac{1}{5.1}\right) = \frac{10}{1}$

$\left(\frac{6.1}{5.1}\right) V_a = \frac{1}{100}$

$V_a = 8.3607$

$V_{oc} = 8.361V$

KCL @ node Va

$\left(\frac{V_a - 10}{1}\right) + \left(\frac{V_a}{5.1}\right) + \left(\frac{V_a}{3.3}\right) = 0$

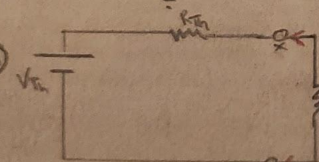
$V_a\left(\frac{1}{1} + \frac{1}{5.1} + \frac{1}{3.3}\right) = \frac{10}{1}$

$\left(\frac{8.41}{5.61}\right) V_a = \frac{1}{100}$

$V_a = 6.67V$

$I_{sc} = \frac{6.67}{3.3} = 2.021\,mA$

③



$V_{oc} = V_{Th}$

$V = IR \rightarrow R_{Th} = \frac{V_{Th}}{I_{sc}} = \frac{V_{oc}}{I_{sc}}$

$R_{Th} = \frac{V_{oc}}{I_{sc}} = \frac{8.361V}{2.021mA}$

$R_{Th} = 4.137\,k\Omega$

$V_{Th} = 8.361V$

$V_{oc} = 8.361V$

$I_{sc} = 2.201mA$

(ii) Construct the circuits of **Figure 2.0b** in MultiSIM, and measure corresponding open-circuit voltage, V_{OC} and short-circuit current, I_{SC}; and then determine V_{Th} and R_{Th} from these values. Record your results in **Table 2.0**.

- Copy and paste a screenshot showing one MultiSIM readings on the circuit. Include the MultiSIM circuit file (**.ms14**) in your Pre-Lab submission.
- All screenshots should show your name printed on the center-top of the MultiSIM screen and the timestamp at the bottom-lower corner.

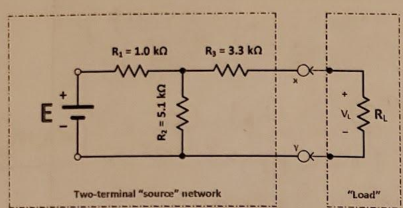


Figure 2.0a: Original Circuit

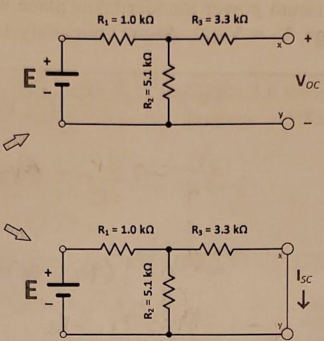


Figure 2.0b: In-circuit measurements for V_{Th} and R_{Th}

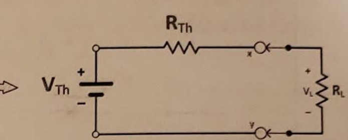


Figure 2.0c: Thevenin Equivalent Circuit

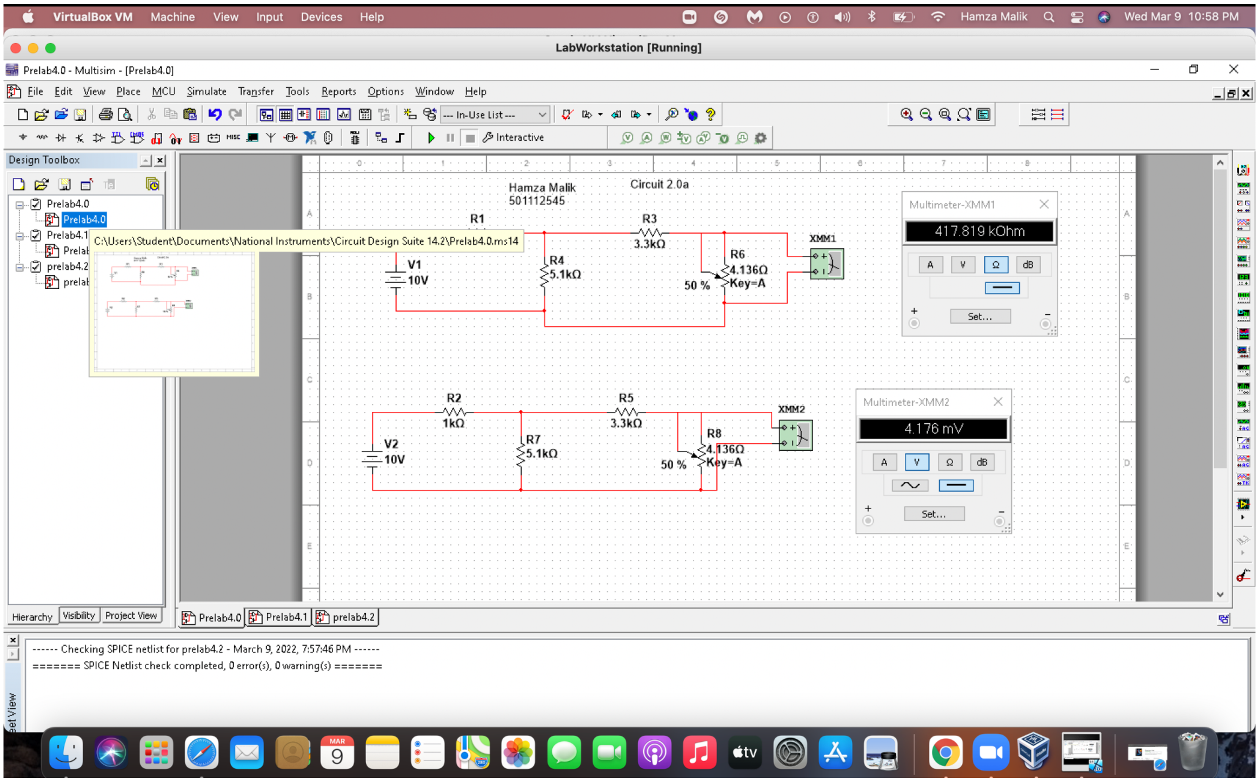
	V_{Th}	R_{Th}	V_{OC}		I_{SC}		$V_{Th} = V_{OC}$		$R_{Th} = V_{OC}/I_{SC}$	
E	Theory (volts)	Theory (kΩ)	Theory (volts)	MultiSIM (volts)	Theory (mA)	MultiSIM (mA)	Theory (volts)	MultiSIM (volts)	Theory (kΩ)	MultiSIM (kΩ)
10 (volts)	8.361	4.137	8.361	8.361	2.021	2.021	8.361	8.361	4.137	4.137

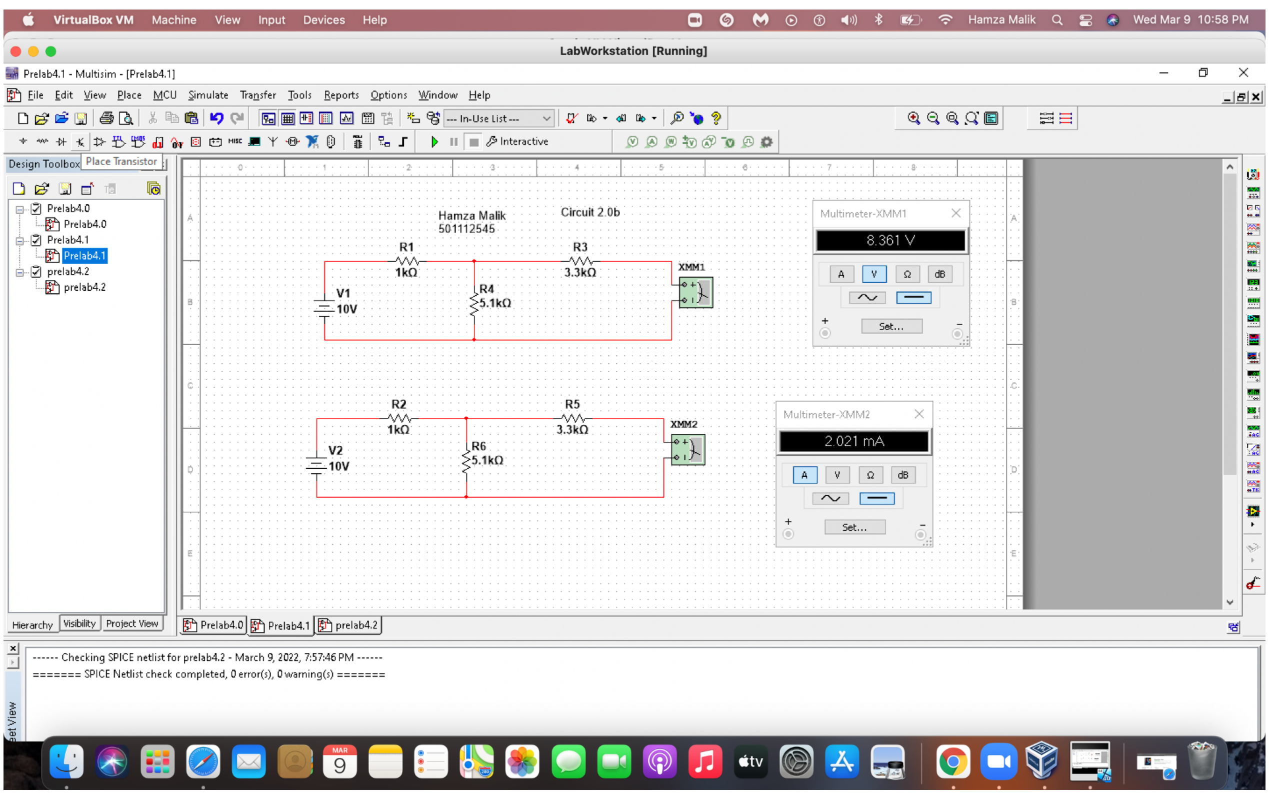
Table 2.0: Theoretical and MultiSIM results of the Figure 2.0 circuits

Compare the MutliSIM simulation results with the calculated theoretical values in **Table 2.0**. Comment on the results, and on any discrepancies.

Pre-Lab workspace

Comparing the multiSIM simulation results with the calculated theoretical values in Table 2.0, they are the same and there are no discrepancies.


Hamza Malik
501112545
Circuit 2.0a
R1
R3
3.3kΩ
V1
10V
R4
5.1kΩ
R6
4.136Ω
Key=A
50 %
XMM1
417.819 kOhm
R2
1kΩ
R5
3.3kΩ
V2
10V
R7
5.1kΩ
R8
4.136Ω
Key=A
50 %
XMM2
4.176 mV
------ Checking SPICE netlist for prelab4.2 - March 9, 2022, 7:57:46 PM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======


Hamza Malik
501112545
Circuit 2.0b
R1
1kΩ
R3
3.3kΩ
V1
10V
R4
5.1kΩ
XMM1
8.361 V
R2
1kΩ
R5
3.3kΩ
V2
10V
R6
5.1kΩ
XMM2
2.021 mA
------ Checking SPICE netlist for prelab4.2 - March 9, 2022, 7:57:46 PM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======

(b) *Maximum Power Transfer*

Consider the circuit in **Figure 2.0c**.

(i) Show that the power, P_L delivered to the load, R_L can be expressed as: -

$$P_L = \left[\frac{V_{Th}}{R_{Th} + R_L}\right]^2 R_L$$ Show your analysis below.

(ii) Show that the maximum power transfer takes place when the load resistance, $R_L = R_{Th}$, with the load voltage, $V_L = V_{Th}/2$. Show your analysis below.

Pre-Lab workspace

(i) $P = VI \rightarrow P = (IR)I$

$P = I^2 R$

$P = \frac{V^2}{R}$

Total Resistance

$R_{Th} + R_L$

$I = \frac{V_{Th}}{R_{Th} + R_L}$

$R_{Th} \rightarrow$ Fixed

$P_L = I^2 R_L$

$\therefore P_L = \left(\frac{V_{Th}}{R_{Th}+R_L}\right)^2 R_L$

(ii) $P = \left(\frac{V_{Th}}{R_{Th}+R_L}\right)^2 R_L = \frac{V_{Th}}{(R_{Th}+R_L)^2}$, $R_L = \frac{(V_{Th}^2 R_L)}{(R_{Th}+R_L)}$

$\frac{dP}{dR_L} = 0$ $\frac{dP}{dR_L} = \frac{(V_{Th}^2 1)(R_{Th}+R_L)^2 - (V_{Th}^2 R_L)2(R_{Th}+R_L)}{(R_{Th}+R_L)^4}$

$(V_{Th})^2\left[(R_{Th}+R_L)^2 - 2(R_{Th}+R_L)R_L\right] = 0$

$(R_{Th}+R_L)^2 - 2(\cancel{R_{Th}+R_L})R_L = 0$

$R_{Th} = 2R_L - R_L$

$\therefore R_{Th} = R_L$

Also,

$V_L = IR_L$

$V_L = \left(\frac{V_{Th}}{R_{Th}+R_L}\right)R_L \Rightarrow V_L = \frac{V_{Th}}{2R_L} \cdot \cancel{R_L} \Rightarrow V_L = \frac{V_{Th}}{2}$

(iii) Construct the circuit of **Figure 2.0a** in MultiSIM but use the **5 kΩ** potentiometer, R_P as the variable load resistance, R_L illustrated below in **Figure 2.0d**. Set the input D.C. source voltage, **E** to **10V**. Monitor the load voltage, V_L across R_P with the DMM Voltmeter. Adjust the potentiometer until this output voltage, $V_L = V_{Th}/2$, at which point theoretically the maximum power transfer should occur per the theorem (note: refer your earlier Pre-Lab analysis for the V_{Th} value). Record your measured values of V_L and R_L in **Table 3.0**.

- Copy and paste a screenshot showing one MultiSIM readings on the circuit. Include the MultiSIM circuit file (.ms14) in your Pre-Lab submission.
- All screenshots should show your name printed on the center-top of the MultiSIM screen

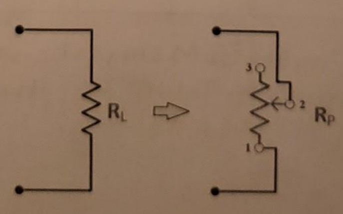


Figure 2.0d: Load, R_L replaced with potentiometer, R_P

Input Source, E	Thevenin voltage, V_{TH} (volts)	Measured load voltage, V_L (volts)	Load resistance R_L per resultant potentiometer, R_P reading. (kΩ)
	From **Table 2.0**	MultiSIM	MultiSIM
10 (volts)	8.361	4.181	4.137

Table 3.0: MultiSIM results at the Maximum Power Transfer for the Figure 2.0 circuit.

Compare the <u>above</u> MutliSIM simulation results in **Table 3.0** with your calculated theoretical values in **Table 2.0**. Was the maximum power transfer theorem verified? Explain, and comment on any discrepancies.

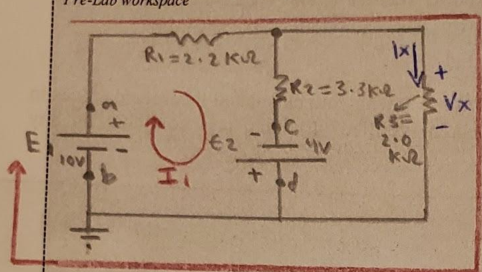
Pre-Lab workspace

Comparing the moltisim simulation results and the calculated theoretical values, both of the results match. The maximom power theorem was verified because when the circuit was made in the moltisim $V_L = V_{Th}/2$ and the value for V_L was as said half of V_{th}, the values of R_L was the same as R_{Th}. Also there were no discrepancies.

(c) *Superposition Principle*

(i) For the circuit shown in **Figures 3.0**, consider the input D.C. source voltages, E_1 = **10V** and E_2 = **4V** (*note the polarities of each source!*). Determine the values of voltage, V_X and current, I_X using either the ***nodal-voltage*** or the ***mesh-current*** analysis technique. Show your analysis below, and record your results in **Table 4.0**.

Pre-Lab workspace



$$\begin{bmatrix} 5.5 & 2.2 \\ 2.2 & 4.2 \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} 14 \\ 10 \end{bmatrix}$$

$$\Delta = (5.5 \times 4.2) - (2.2 \times 2.2)$$
$$= 23.1 - 4.84$$
$$\Delta = 18.26$$

$$\Delta 1 = \begin{bmatrix} 14 & 2.2 \\ 10 & 4.2 \end{bmatrix}$$
$$= (14 \times 4.2) - (2.2 \times 10)$$
$$= 58.8 - 22.0$$
$$\Delta 1 = 36.8$$

Mesh 1

$$-10 + 2.2(I_1 + I_2) + 3.3 I_1 - 4 = 0$$
$$I_1(2.2 + 3.3) + I_2(2.2) = 14 \rightarrow ①$$
$$I_1(5.5) + I_2(2.2) = 14$$

$$\Delta 2 = \begin{bmatrix} 5.5 & 14 \\ 2.2 & 10 \end{bmatrix}$$
$$= (5.5 \times 10) - (14 \times 2.2)$$
$$= 55.0 - 30.8$$
$$\Delta 2 = 24.2$$

Mesh 2

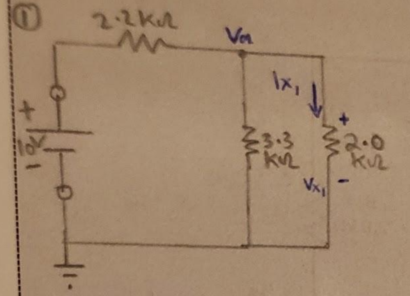
$$-10 + 2.2(I_2 + I_1) + 2.0 I_2 = 0$$
$$I_1(2.2) + I_2(2.2 + 2.0) = 10$$
$$I_1(2.2) + I_2(4.2) = 10 \rightarrow ②$$

(ii) For the same circuit shown in **Figures 3.0** with the input D.C. source voltages, E_1 = **10V** and E_2 = **4V**, apply the Superposition Principle technique as follows:-

1. Remove input source, E_2 and replace it with a short-circuit by connecting a wire between "c" and "d". Determine the resultant voltage, V_{X1} across R_3 and current, I_{X1} through R_3. Show your analysis below, and record the results in **Table 4.0**.

2. Connect input source, E_2 in its original place between "c" and "d". Be mindful of the polarity connections. Then remove input source, E_1 and replace it with a short-circuit by connecting a wire between "a" and "b". Determine the resultant voltage, V_{X2} across R_3 and current, I_{X2} through R_3. Show your analysis below, and record the results in **Table 4.0**.

Pre-Lab workspace



$$\left(\frac{V_a-10}{2.2}\right)+\left(\frac{V_a}{3.3}\right)+\left(\frac{V_a}{2.0}\right)=0$$

$$V_a\left(\frac{1}{2.2}+\frac{1}{3.3}+\frac{1}{2.0}\right)=\frac{10}{2.2}$$

$$\frac{8.3}{6.6}V_a=\frac{1}{220}$$

$$V_a=3.614V$$

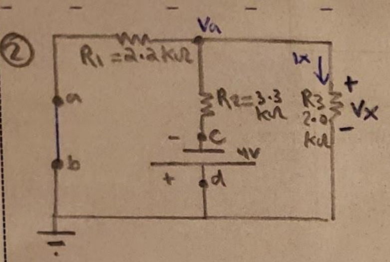
$V_{x1}=V_a$

$V_{x1}=3.614V$

$I_{x1}=\frac{V_a}{2.0}$

$=\frac{3.614}{2.0}$

$I_{x1}=1.807mA$



$$\left(\frac{V_a}{2.2}\right)+\left(\frac{V_a-(-4)}{3.3}\right)+\frac{V_a}{2.0}=0$$

$$V_a\left(\frac{1}{2.2}+\frac{1}{3.3}+\frac{1}{2.0}\right)=-\frac{4}{3.3}$$

$$\frac{8.3}{6.6}V_a=\frac{-4}{3.3}$$

$$V_a=-0.9639V$$

$V_{x2}=V_a$

$V_{x2}=-0.9639V$

$I_{x2}=\frac{V_a}{2.0}$

$=\frac{-0.9639}{2.0}$

$I_{x2}=-0.4819mA$

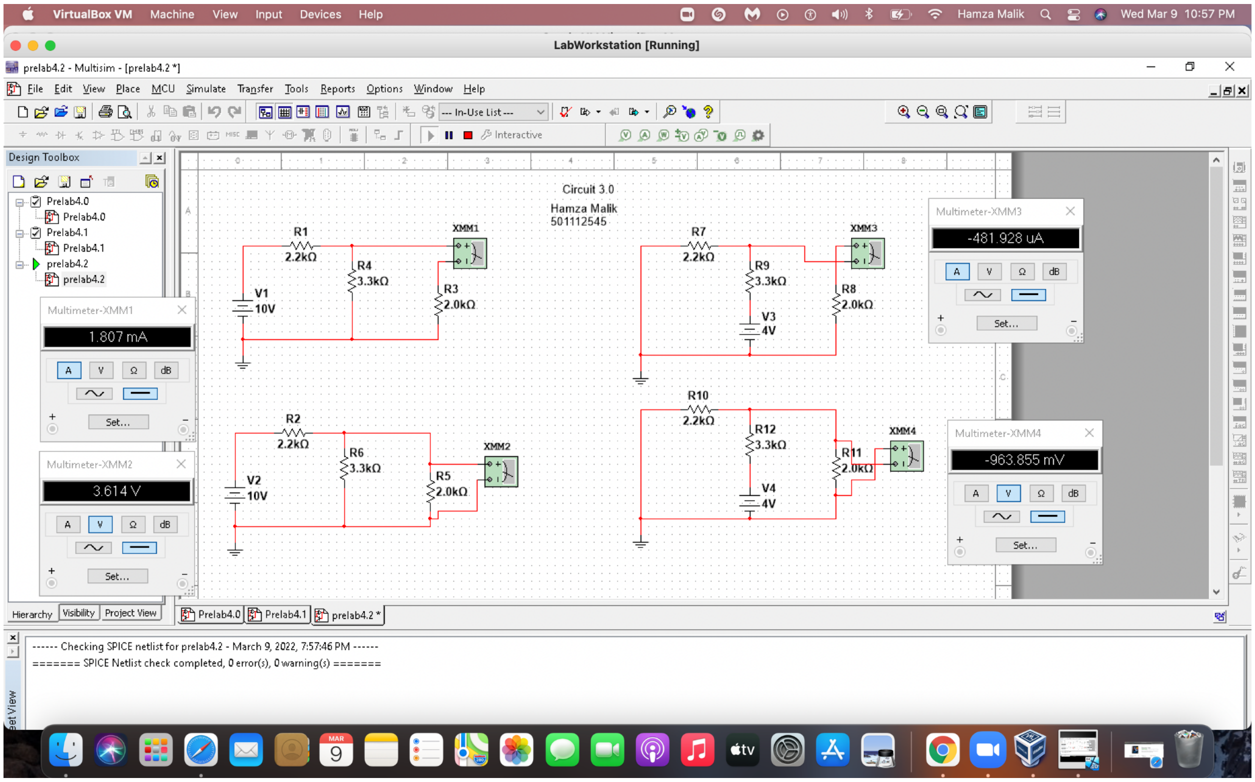
(iii) Construct the circuit of **Figure 3.0** in MultiSIM, and repeat the above circuit-procedures (i) and (ii) to measure V_X, I_X, V_{X1}, I_{X1}, V_{X2} and I_{X2}. Record the results in **Table 4.0.**

- Copy and paste a screenshot showing one MultiSIM readings on the circuit. Include the MultiSIM circuit file (**.ms14**) in your Pre-Lab submission.
- All screenshots should show your name printed on the center-top of the MultiSIM screen

From your results in **Table 4.0**, was the Superposition Principle technique verified using the expressions $V_X = V_{X1} + V_{X2}$ and $I_X = I_{X1} + I_{X2}$? Explain why, and comment on any discrepancies. Does the Superposition Principle make it easier to analyze circuits with multiple power sources? Why?

Pre-Lab workspace

From the results the superposition principle was verified using the expressions above. This is due to the fact that V_{x1} and I_{x1} ~~can~~ show the values in first half the circuit where as V_{x2} and I_{x2} show in the other half of the circuit. When both are added together we get the total V_x across the resistor and I_x through R_3. There were no discrepancies and for the superposition principle, makes it easier to visualize and work with the circuits because when we turn off the power, it makes it alot more easier to calculate any sort of value.



LabWorkstation [Running]
prelab4.2 - Multisim - [prelab4.2 *]
Circuit 3.0
Hamza Malik
501112545
R1
2.2kΩ
R4
3.3kΩ
R3
2.0kΩ
V1
10V
XMM1
Multimeter-XMM1
1.807 mA
R7
2.2kΩ
R9
3.3kΩ
R8
2.0kΩ
V3
4V
XMM3
Multimeter-XMM3
-481.928 uA
R2
2.2kΩ
R6
3.3kΩ
R5
2.0kΩ
V2
10V
XMM2
Multimeter-XMM2
3.614 V
R10
2.2kΩ
R12
3.3kΩ
R11
2.0kΩ
V4
4V
XMM4
Multimeter-XMM4
-963.855 mV
------ Checking SPICE netlist for prelab4.2 - March 9, 2022, 7:57:46 PM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======

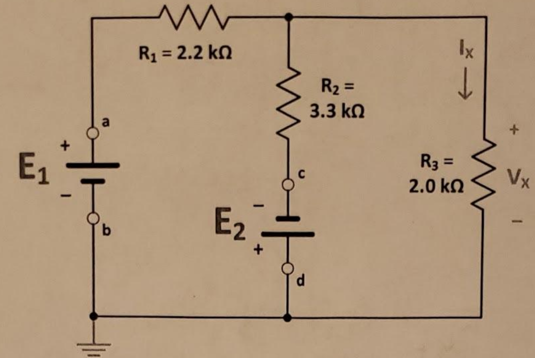


Figure 3.0: Circuit with multiple voltage sources

V_X (volts)		I_X (mA)		V_{X1} (volts)		I_{X1} (mA)		V_{X2} (volts)		I_{X2} (mA)		$V_X = V_{X1} + V_{X2}$		$I_X = I_{X1} + I_{X2}$	
Th.	MS.	Th.	MS.	Th.	MS.	Th.	MS.	Th.	MS.	Th.	MS.	Th.	MS.	Th.	MS.
2.651	2.651	1.325	1.325	3.614	3.164	1.807	1.807	-0.964	-0.964	-0.482	-0.482	2.651	2.65	1.325	1.325

Table 4.0: Theoretical (Th.) and MultiSIM (MS.) results of the Figure 3.0 circuits