Faculty Of Engineering and
Information Sciences
UOWD

ENGG104 – PART 1/2

Laboratory Workbook

Student Name:

Student Number:

Lab Partners Name:

Introduction

This is the ENGG104 Laboratory Logbook. It is to be used for answering all questions requested in the laboratory notes.

The laboratory notes for the first TWO experiments are online and can be accessed via Moodle.

The laboratory notes for experiments 43 to 6 are available in this workbook. However you should continue to use the resources available online where suitable.

All requirements to the running of the laboratory are outlined in the online laboratory notes.

Be aware that preparation is mandatory and that MOST experiments have tasks that must be completed before you arrive for your scheduled laboratory. Penalties apply for not completing the activities.

This logbook will be inspected by the laboratory demonstrator after demonstrations and a grade will be given for each task recorded in the logbook. It is your responsibility to ensure that the lab demonstrator views and signs your logbook.

For the first two experiments, tasks in your laboratory notes are completed in this workbook using the following approach:

- Aim: What is the task trying to teach you?
- Results: What are the results/answers to the task?
- Summary: What did you learn by completing the task?

An example question and response is shown below:

Task 1:
The digital multimeter used in the laboratory has 4 sockets that the leads can be plugged into. What sockets would you use for measuring current?

Answer:
Aim: To learn how to correctly setup the leads for the measurement of current
Results: The COM socket is common for all measurements and is used for the black lead. The A and mA sockets can both be used for measuring current where the A socket is used for measuring large currents and the mA socket is used for measuring small currents in the milliamp range. The red lead is connected to one of these sockets.
Summary: Connecting up the leads on the digital multimeter is dependent on the size of the current to be measured.

Experiment 1

SECTION TWO

Task 1

Aim:

Results:

i)

ii)

iii)

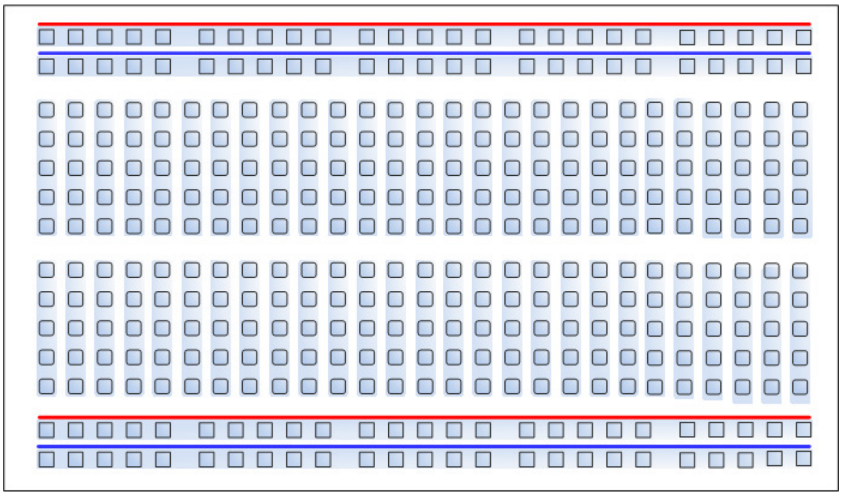
iv)

Summary:

Task 2

Aim:

Results:



Summary:

SECTION THREE

Task 1 – page 3 of lab book

Task 2

Aim:

Results:

i)

Resistor	Nominal value	Measured value using analogue meter	Measured value using digital meter
Resistor 1			
Resistor 2			
Resistor 3			

ii)

Summary:

SECTION FOUR

Task 1

Aim:

Results:

Summary:

Task 2

Aim:

Results:

Summary:

SECTION FIVE

Task 1:

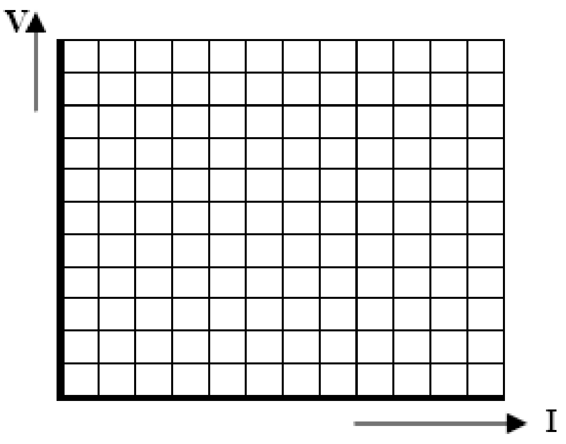
Aim:

Results:

i)

Voltage (V)	0	1	2	3	4	5	6	7	8	9	10	11	12
Current (mA)													

ii)



iii)

iv)

Summary: What learning objectives (see Getting Started page) did you experience in this experiment?

End of Experiment 1

This table is completed by your supervisor.

Supervisor	
Date	
Comments	

Note: Supervisors to check that lab kit resistor table at start of this workbook has been complete.

Experiment 2

SECTION ONE

Task 1

Aim:

Results:

i)

ii)

iii)

iv)

v)

vi)

vii)

Task 2

Aim:

Results:

i)

ii)

iii)

iv)

Task 3

Results:

SECTION TWO

Task 1

Aim:

Results:

i)

ii)

iii)

Source Voltage E	Voltage Across R_1 V_1	Voltage Across R_2 V_2	$V_1 + V_2$	$V_1 : V_2$	R_1:R_2
6V					
9V					
12V					

iv)

Summary:

SECTION 3
Task 1

Aim:

Results:

i)

Source Voltage E	R_1	R_2	Total Current I_s	Current through R_1 I_1	Current through R_2 I_2	$I_1 + I_2$
8V	1.2kΩ	1.8kΩ				
8V	1.8kΩ	470Ω				
8V	1.2kΩ	1.2kΩ				

ii)

Summary: What learning objectives (see Getting Started page) did you experience in this experiment?

End of Experiment 2

This table is completed by your supervisor.

Supervisor	
Date	
Comments	

Experiment 3

SECTION ONE

In this section you will be introduced to Multisim, a powerful simulation program used in electronics.

To open Multisim, click the shortcut icon or find the executable in location: Start -> All Programs -> National Instruments -> Circuit Design Suite 12.0 -> Multisim 12

Multisim is a simulation tool that allows us to easily learn about electronics and prototype circuits.

The most important toolbars that you must be aware of are the 'Component Toolbar' and the 'Instrument Toolbar'. You can right click in the toolbar area to show a whole range of other toolbars available at your disposal.

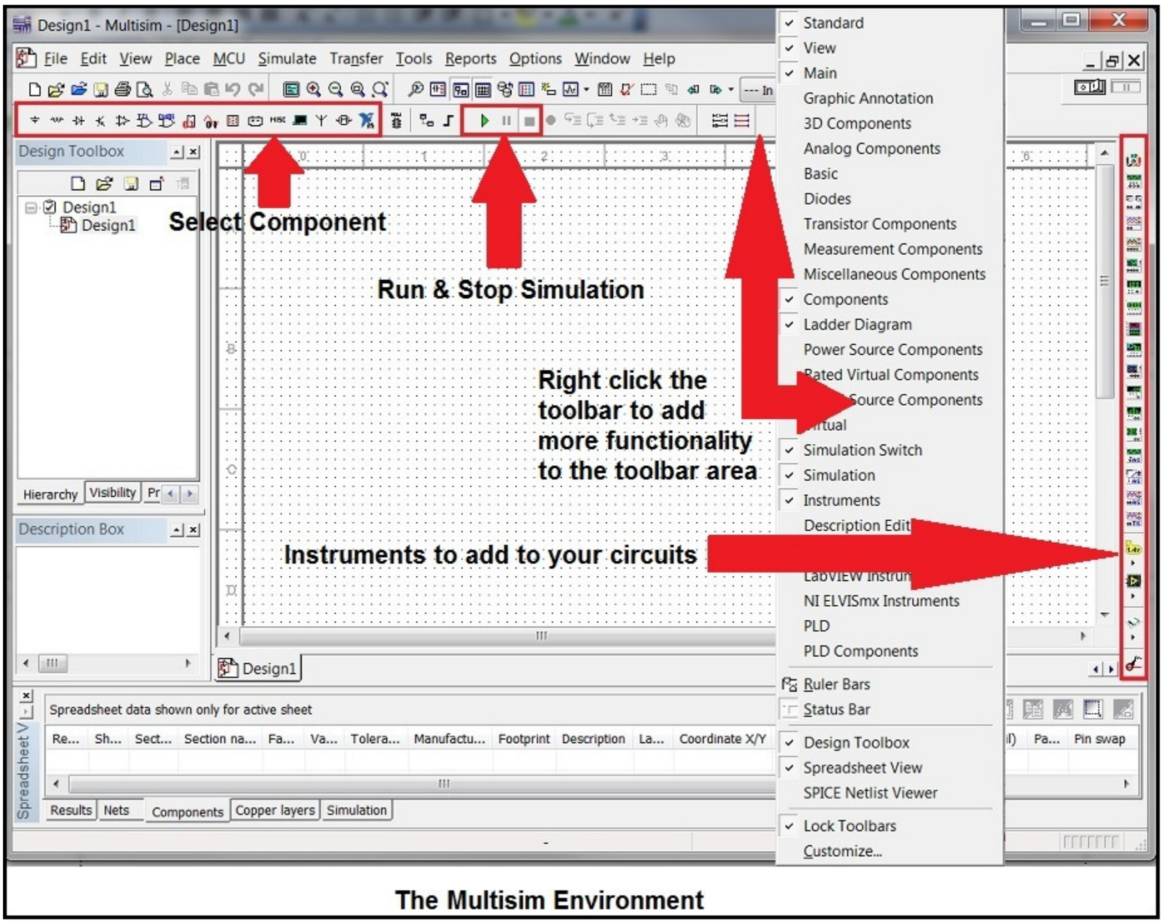


Figure 3.1: Multisim Environment

The Component Toolbar allows you to place components onto the work area. Multisim provides a selection of thousands of components that you can choose from. Components include power sources, resistors, capacitors, light bulbs and so forth as shown in Fig. 3.2.

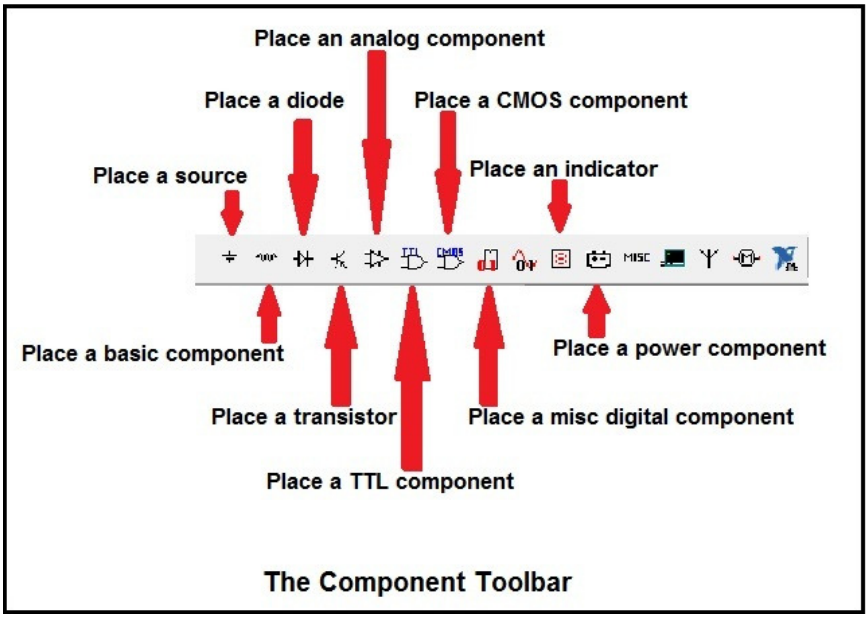


Figure 3.2: Component Toolbar

From the Component Toolbar click on the Place a Source icon. The Component Browser will now appear. From the component browser you can select the component that you would like to add to your circuit as shown in Fig. 3.3.

The "Group" drop down list contains all the component groups as shown on the Component Toolbar. You can change your grouping if you selected the wrong group icon. Each group has families associated with it. Treat families like subgroups of components. The components associated with the families are shown in the Component list.

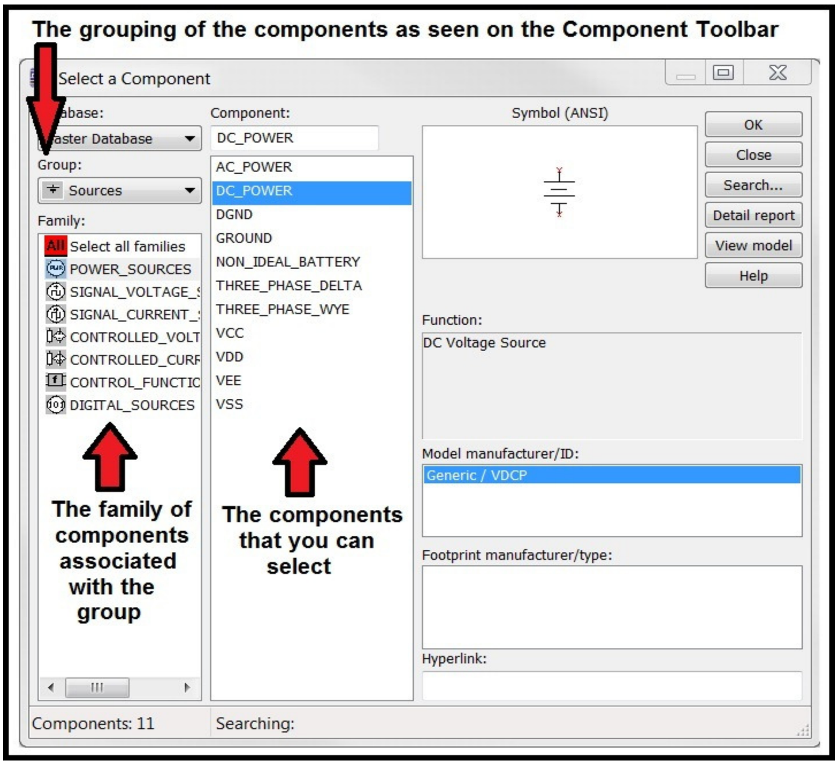


Figure 3.3: Component Browser

The Instruments Toolbar allows you to place instruments onto the work area. Instruments are used to analyse and take measurements from your circuit. Instruments used within ENGG104 experiments include multimeters, function generators and oscilloscopes.

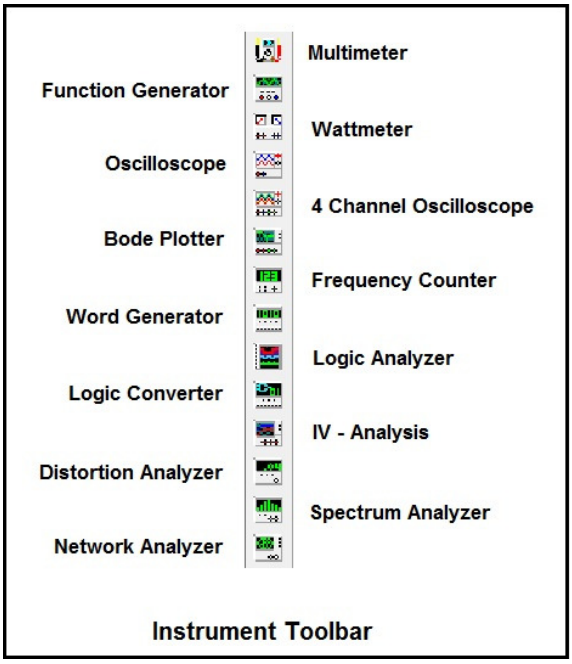


Figure 3.4: Instrument Toolbar

Some important information to be aware of:

- Without a power source and a ground, your simulation cannot run.
- You cannot make changes to your circuit while the simulation is running.
- By default, the component selection box keeps returning as a pop-up until you have completed placing your components. Close the window to return to the schematic entry window. You can change this option in the global preferences dialog box.
- Multisim is a modeless wiring environment. This means that Multisim determines the functionality of the mouse tool by the position of the mouse. You do not have to return to the menu to choose between placement, wiring, and editing tools.
- Multimeters need to be set to behave as ammeters, voltmeter, or ohmmeters. The setting can be changed by double clicking on the multimeter and selecting the desired behaviour.
- A search can be made for a desired component if the component's name is known. When in the Component Browser, ensure that the 'database' is set to 'Master Database' and change the 'Group' to 'Select all groups' and change 'family' to 'Select all Families'. Then under component you can type the name of the component you are looking for and Multisim will search for a match from the entire database.

Build the circuit shown in Fig. 3.5 in the Multisim environment. Ensure that all multimeters are set to ammeters, except for XMM6 which should be set to a voltmeter. Run the simulation and record the results of each meter.

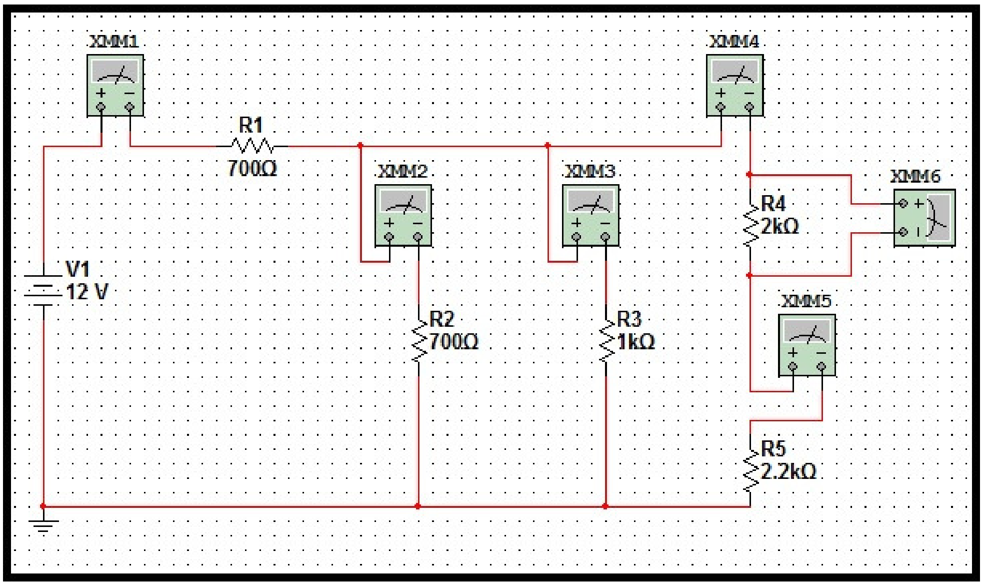


Figure 3.5: Multisim Circuit

Meter	Measurement (mA) or (V)
XMM1	
XMM2	
XMM3	
XMM4	
XMM5	
XMM6	

SECTION TWO

In this section, the voltage divider and current divider rules shall be verified both on a breadboard and in the Multisim environment.

Part 1:

The voltage divider rule, when applied to two series connected resistors, can be expressed as:

$$V1 = \frac{E.R1}{R1 + R2}$$

Or:

$$V2 = \frac{E.R2}{R1 + R2}$$

Note that this equation only applies to the circuit shown in Fig. 3.6. The value E is the combined voltage across both elements R1 and R2. V1 and V2 are the voltages across elements R1 and R2, respectively.

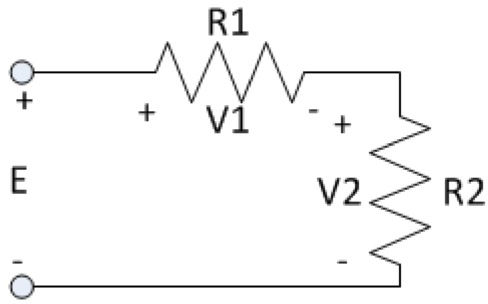


Figure 3.6: Voltage Divider Rule Circuit

It is important to recognise that, in cases where more than two resistors are used, the circuit will often be reducible and the same method can then be implemented. Consider the circuit shown in Fig. 3.7.

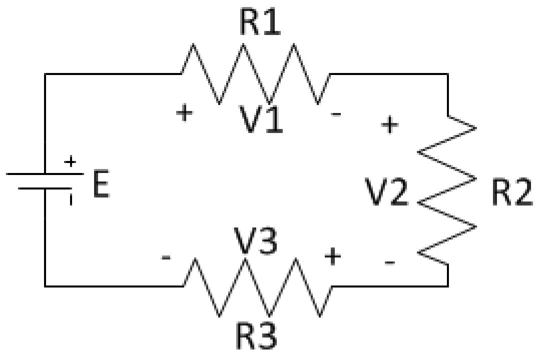


Figure 3.7: Three Element Voltage Divider

If the desired voltage is V1, resistors R2 and R3 can be added to form one resistor. The resultant equation would be:

$$V1 = \frac{E.R1}{\sum R}$$

Where ΣR is the sum of all series connected resistances. If V2 and V3 are desired, the respective equations become:

$$V2 = \frac{E.R2}{\sum R}$$

$$V3 = \frac{E.R3}{\sum R}$$

Calculate the voltages V1, V2 and V3 for the circuit shown in Fig. 3.7 with the circuit parameters:
E=12 V, R1 = 3.3 kΩ, R2 = 10 kΩ and R3 = 2.2 kΩ.
Enter the calculated voltages into the table below. Next, build the same circuit on a breadboard and in Multisim. Use multimeters to obtain the measured and simulated voltages, filling out each in the table below.

	Calculated voltage	Measured Voltage	Simulated Voltage
V1			
V2			
V3			

How do your results compare? Explain any discrepancies.

Part 2:

The current divider rule, when applied to two parallel connected resistors, can be expressed as:

$$I1 = \frac{Is.R2}{R1 + R2}$$

Or:

$$I2 = \frac{Is.R1}{R1 + R2}$$

Notice that for the voltage divider rule, the numerator contains the resistor of the desired voltage, whereas the current divider rule's numerator contains the other resistor. These equations can only be applied to the circuit shown in Fig. 3.8.

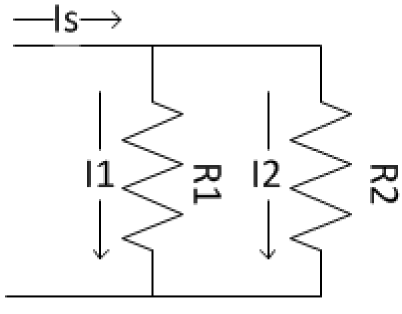


Figure 3.8: Current Divider Rule

Calculate the total resistance R_{total} of the circuit shown in Fig. 3.9. Use R_{total} to calculate the total current Is.

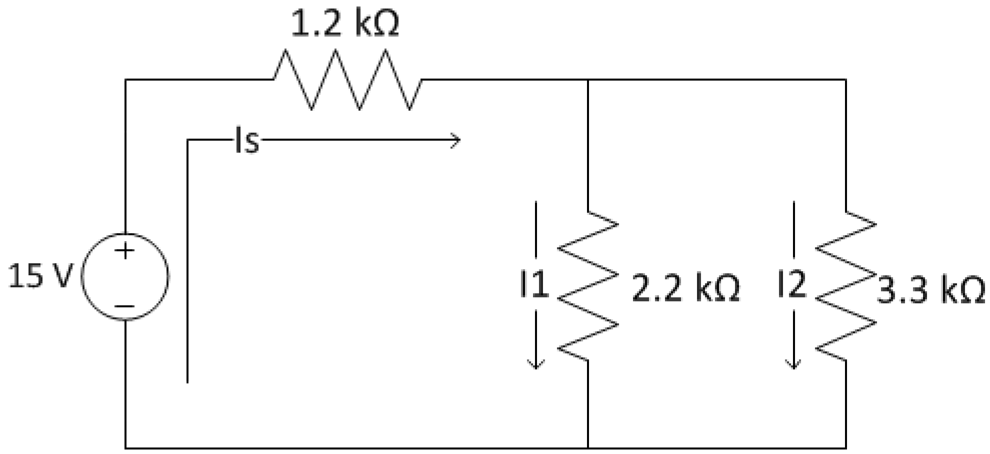


Figure 3.9: Current Divider Rule Example

R_{total}=________________

Is=___________________

Next, calculate the currents I1 and I2 using the current divider rule and enter the values into the calculated currents in the table below. Build the circuit on a breadboard and in Multisim. Measure the currents using multimeters. Enter these currents into the table below under measured and simulated current.

	Calculated Current	Measured Current	Simulated Current
I1			
I2			

How do the calculated, measured and simulated currents compare? Explain any discrepancies?

End of Experiment 3

This table is completed by your supervisor.

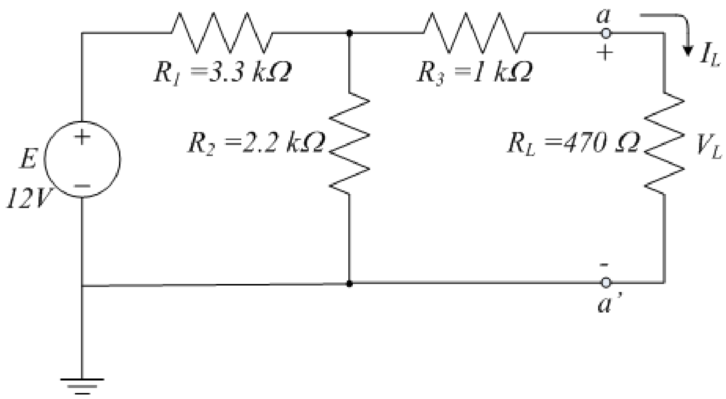
Supervisor	
Date	
Comments	

Experiment 4

SECTION 1

The objective of this section is to implement and experimentally validate Nodal Analysis.

Build the circuit shown in Fig. 4.1 on your breadboard and in the Multisim environment. This circuit shall be needed in Section 2, so do not deconstruct until the end of the laboratory. Measure and record each resistor individually. The measured values should be used in Multisim and in all calculations, instead of using the nominal values.



Measured R_1 =__________

Measured R_2 =__________

Measured R_3 =__________

Measured R_L =__________

Figure 4.1: Nodal Analysis Example Circuit

The steps for Nodal Analysis are as follows:

1. Identify all nodes and label all known and unknown voltages. Series connected resistors can be combined to reduce the number of unknown nodes.

2. Assert current directions for each branch (it doesn't matter which direction you pick, but be consistent!)

3. Use Ohm's Law ($I = \frac{Vfrom-Vto}{R}$) and KCL to derive an equation for each node with an unknown voltage.

4. Solve the equations. Cramer's rule should be used for more complicated cases.

If the current directions you assert are incorrect, the current will be negative but the answer is still valid. There is an example of Nodal Analysis on Moodle if you require more help.

To find the current I_L, reduce the 1 kΩ resistor and 470 Ω resistor into a single 1.47 kΩ resistor, thus eliminating an unnecessary node. The unknown voltage across the 2.2 kΩ resistor and 1.47 kΩ resistor may then be found using nodal analysis. Once this unknown voltage is determined, the current I_L may be calculated by Ohm's Law.

Calculate the current I_L using nodal analysis and Ohm's Law. Remember to use the measured values in your calculations.

Using ammeters, measure the current I_L on the physical and simulated circuits you have built. Enter the calculated, measured and simulated values for I_L in the table below.

	Calculated Current	Measured Current	Simulated Current
I_L			

How do the results compare? Explain any discrepancies.

SECTION 2

Thevenin equivalent circuits are an excellent method used to reduce the complexity of circuits. A Thevenin equivalent replaces a segment of a circuit with a series connected voltage source and resistor as shown in Fig. 4.2.

The terminals are used to separate the segment of the circuit that is to be converted from the remainder of the circuit. In Fig. 4.2, all the circuitry to the left of the terminals is replaced with a voltage source of magnitude E_{TH} connected in series with a resistor of magnitude R_{TH}. Note that the circuitry to the right of the terminals can have any complexity and does not necessarily have to be a single resistor.

The Thevenin equivalent can be achieved for any combination of voltage sources, current sources and resistors, irrespective of the complexity of the original circuit.

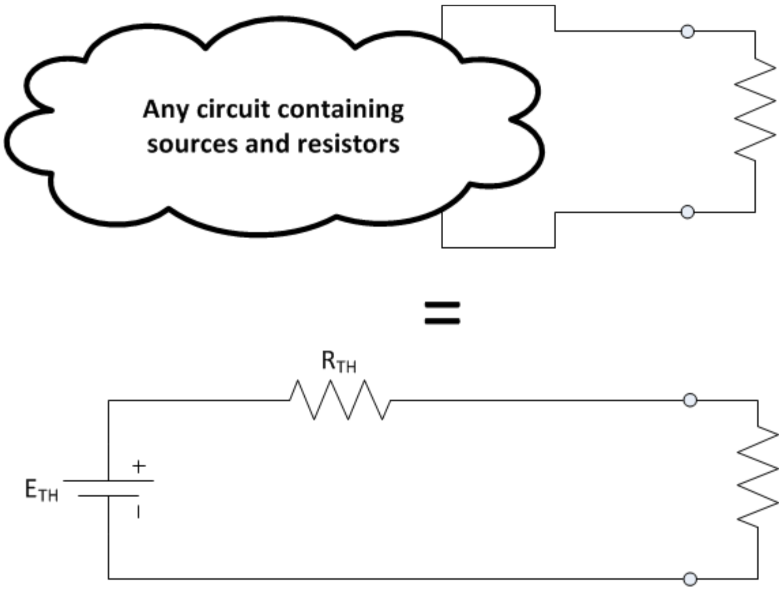


Figure 4.2: Thevenin Equivalent

To find R_{TH}:

1. Remove all circuitry that is not part of the conversion. In Fig. 4.2, the resistor to the right of the terminals is removed and replaced with an open circuit.
2. Replace all voltage sources with short circuits.
3. Replace all current sources with open circuits.
4. Calculate the total resistance of the resultant circuit from the perspective of the terminals. (This is R_{TH}).

To find V_{TH}:

1. Reset the circuit to its original form.
2. Again, remove all circuitry that is not part of the conversion.
3. Calculate the voltage across the terminals. (This is E_{TH}).

The voltage may be calculated using any desired method. Generally, the voltage divider rule may be used for very simple cases; for more complex cases, Nodal analysis may be used.

Calculate the Thévenin voltage and resistance to the left of points a-a' in Fig. 4.3 using measured resistor values (found in Section 1). Show all work.

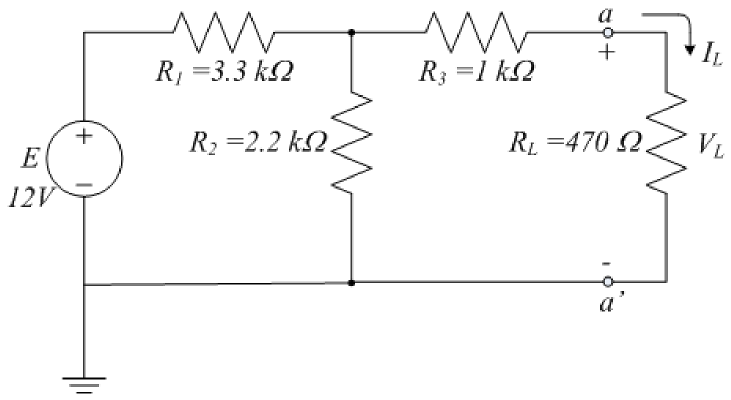


Figure 4.3: Thevenin Equivalent Example

Measured R_1 =__________

Measured R_2 =__________

Measured R_3 =__________

Measured R_L =__________

Insert the values of E_{Th} and R_{Th} in Fig. 4.4 **Error! Reference source not found.** and calculate I_L via Ohm's Law. Record the calculated E_{Th} and R_{Th} in Table 4.1 (see next page). How does the I_L compare with the results of Section 1?

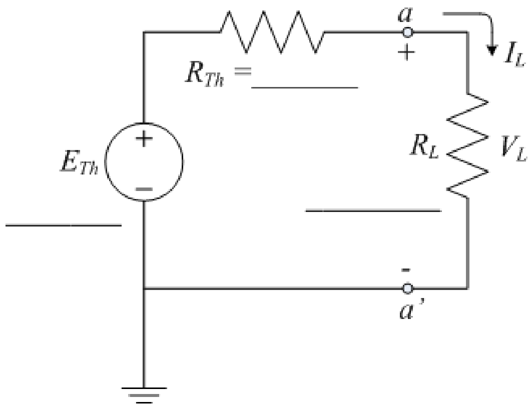


Figure 4.4: Measured values of Thevenin Equivalent Circuit

Determine R_{TH} by constructing the network of Fig. 4.5 and measuring the resistance between points a-a' with R_L removed. Note that the external circuit has been removed and the voltage source is replaced by a short circuit as is required in the calculation of R_{TH}. Record in Table 4.1 (see next page).

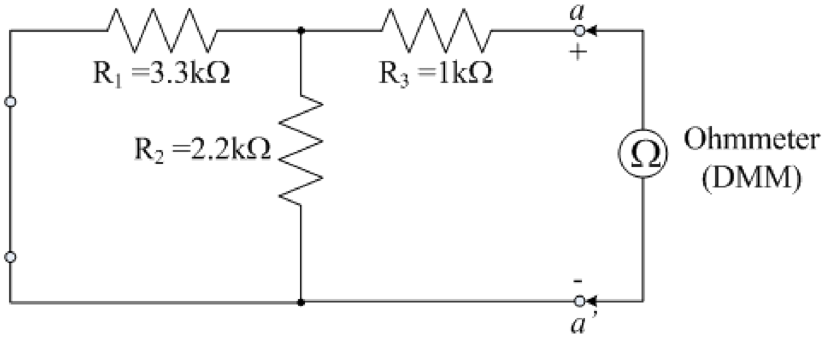


Figure 4.5: Determine R_{TH} of Thevenin Equivalent Circuit

Determine E_{TH} by constructing the network of Fig. 4.6 and measuring the open-circuit voltage between points a-a'. Record in Table 4.1.

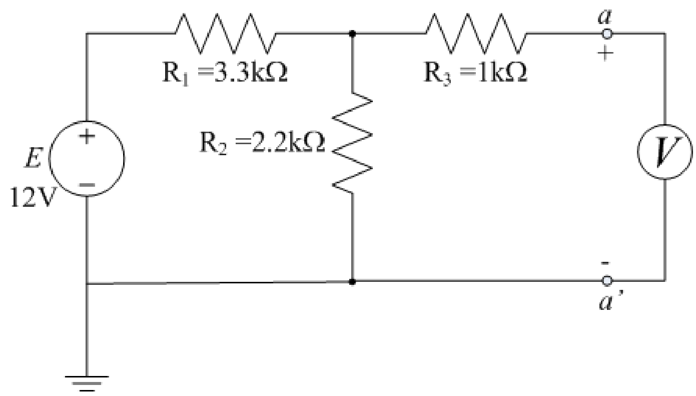


Figure 4.6: Determine E_{TH} of Thevenin Equivalent Circuit

Using the same circuits above, determine R_{Th} and E_{Th} in Multisim and record in Table 1.

Table 4.1

	Calculated values	Measured values	Multisim values
E_{Th}			
R_{Th}			

How do the results compare? Explain any discrepancies.

End of Experiment 4

This table is completed by your supervisor.

Supervisor	
Date	
Comments	

Experiment 5

SECTION ONE
This section shall cover a difficult nodal analysis question and the implementation of Cramer's Rule.

Construct the circuit shown in Fig. 5.1 on a breadboard and in Multisim. The voltage source on the left hand side shall be constructed using your variable DC power supply. The voltage source on the right hand side shall be constructed using the fixed 5 V voltage source available on your bench. The voltage is often calibrated poorly and may be as high as 6 V. Use a multimeter to measure the source voltage and enter the measured value in Fig. 5.1. The measured voltage should be used in calculations and in Multisim. Pay attention to the polarities of the voltage sources.

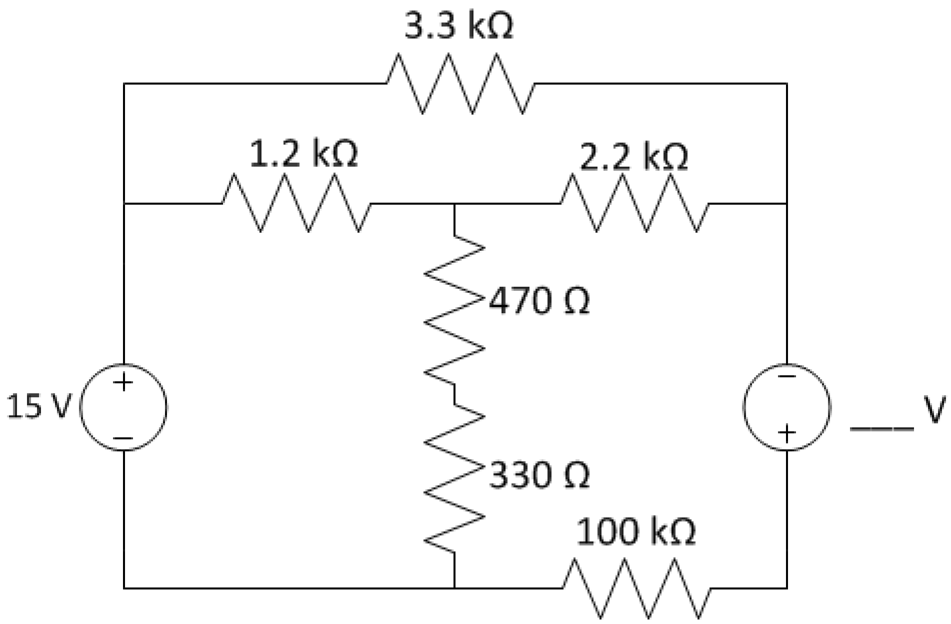


Figure 5.1: Advanced Nodal Circuit

Individually measure the resistance of each resistor and record the data in the table below. Measured resistance should be used in all calculations.

Resistor	Measured Resistance
330 Ω	
470 Ω	
1.2 kΩ	
2.2 kΩ	
3.3 kΩ	
100 kΩ	

Once the circuits are built, use a multimeter to measure the voltages across each resistor for both the breadboard and in Multisim simulation. Enter the voltages into Table 5.1 (two pages ahead).

Redraw the circuit shown in Fig. 5.1. Identify all nodes, denoting known and unknown node voltages. Assert the current directions. Derive the two equations needed to find the unknown node voltages.

First, the set of equations must be expressed in matrix form, for example:

$$\begin{bmatrix} a & b \\ c & d \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} k_1 \\ k_2 \end{bmatrix}$$

Where $\begin{bmatrix} a & b \\ c & d \end{bmatrix}$ is the coefficient matrix and $\begin{bmatrix} k_1 \\ k_2 \end{bmatrix}$ is the constant matrix. The values V_1 and V_2 may then be solved using determinates:

$$V_1 = \frac{\begin{vmatrix} k_1 & b \\ k_2 & d \end{vmatrix}}{\begin{vmatrix} a & b \\ c & d \end{vmatrix}}$$

$$V_2 = \frac{\begin{vmatrix} a & k_1 \\ c & k_2 \end{vmatrix}}{\begin{vmatrix} a & b \\ c & d \end{vmatrix}}$$

Notice that for the first variable, the numerator is the determinate of the coefficient matrix with the first column replaced with the constant matrix. For the second variable, the second column of the numerator is replaced with the constant matrix. The denominator is always the determinate of the coefficient matrix. To calculate a 2x2 determinate, use $\begin{vmatrix} w & x \\ y & z \end{vmatrix} = wz - xy.$

Write the equations you derived for the circuit in Fig. 5.1 in matrix form. Solve these equations using Cramer's Rule.

Once you have found the unknown node voltages, calculate the voltages across each resistor, entering your answers into Table 2. How do your answers compare with the measured and simulated values?

Table 5.1

Resistor	Measured Voltage	Simulated Voltage	Calculated Voltage
330 Ω			
470 Ω			
1.2 kΩ			
2.2 kΩ			
3.3 kΩ			
100 kΩ			

SECTION TWO

This section covers the steady state behaviour of series RC circuits.

The resistor dissipates electrical energy in the form of heat. In contrast, the capacitor is a component that stores electrical energy in the form of an electrical field. The capacitance of a capacitor is a function of its geometry and the dielectric used. Dielectric materials are rated by their ability to support an electric field in terms of a figure called the dielectric constant (*k*). A vacuum is the standard dielectric for purposes of reference and is assigned the value of unity. The capacitance (in farads) is determined by

$$C = 8.85 \times 10^{-12} \varepsilon_r \frac{A}{d}$$

where ε_r is the relative permittivity (or dielectric constant), A is the area of the plates (m^2), and d is the distance between the plates (m). By changing anyone of the three parameters, one can easily change the capacitance.

The dielectric constant is not to be confused with the dielectric strength of a material, which is given in volts per unit length and is a measure of the maximum stress that a dielectric can withstand before it breaks down and loses its insulator characteristics.

In a DC circuit, the volt-ampere characteristics of a capacitor in the steady-state mode are such that the capacitor prevents the flow of dc current but will charge up to a DC voltage. Essentially, therefore, the characteristics of a capacitor in the steady-state mode are those of an open circuit. The charge Q stored by a capacitor is given by

$$Q = CV$$

Where C is the capacitance and V is the voltage impressed across the capacitor.

The total capacitance of series connected capacitors may be calculated similarly to resistors in parallel:

$$\frac{1}{C_T} = \frac{1}{C_1} + \frac{1}{C_2} + \frac{1}{C_3} + \cdots + \frac{1}{C_N}$$

The total capacitance of parallel connected capacitors may be calculated similarly to resistors in series:

$$C_T = C_1 + C_2 + C_3 + \cdots + C_N$$

The energy (W) stored by a capacitor (in joules) is determined by

$$W = \frac{1}{2} C V^2$$

For the network shown in Fig. 5.2, the capacitor will, for all practical purposes, charge up to *E* volts in five time constants, where a time constant (τ) is defined by

$$\tau = RC$$

In one time constant, the voltage v_c will charge up to 63.2% of its final value, in 2τ up to 86.5%, in 3τ up to 95.1%, in 4τ up to 98.1%, and in 5τ up to 99.3%, as defined by

$$v_c = E(1 - e^{-t/RC})$$

The current i_c is defined by

$$i_c = \frac{E}{R}(e^{-t/RC})$$

At steady state (after 5 time constants), a capacitor may be assumed to behave as an open circuit.

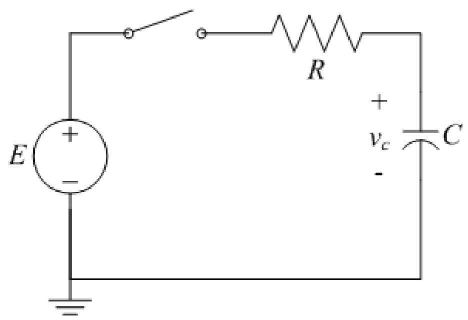
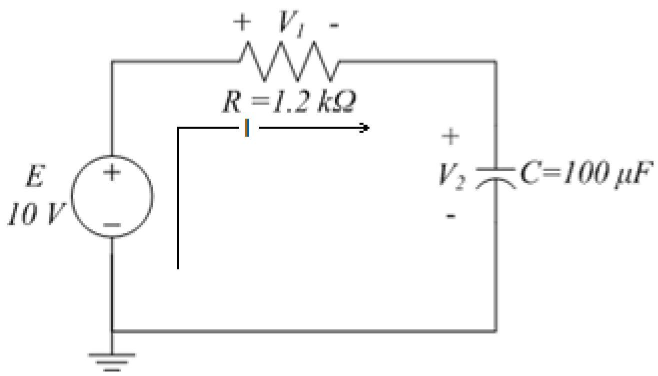


Figure 5.2: Series RC Circuit

Construct the network of Fig. 5.3. Insert the measured resistor value. Be sure to note polarity on electrolytic capacitors as shown in the Fig. 5.3.



Measured R =________

Figure 5.3: Series RC Circuit Example

1) Calculate the steady-state value (defined by a period of time greater than (and NOT equal to)) five time constants) of the current *I* and the voltages V_1 and V_2.

2) Measure the voltages V_1 and V_2 and calculate the current *I* from Ohm's law. Compare these values with the results obtained in Part (1).

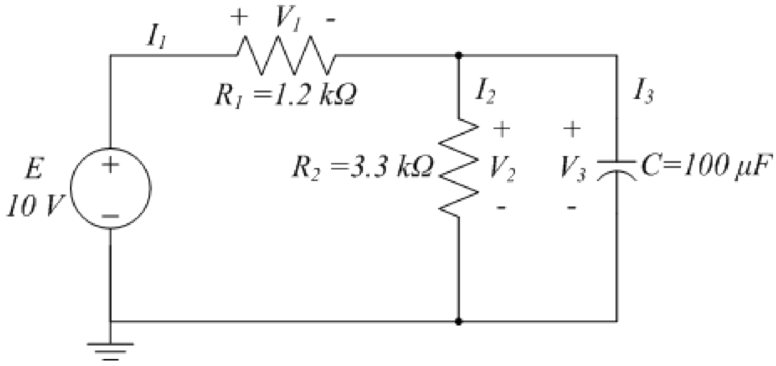
3) Calculate the energy stored by the capacitor.

4) Carefully disconnect the supply and quickly measure the voltage across the disconnected capacitor. Is there a reading? Why?

5) Short the capacitor terminals with a lead and then measure again. Why was it necessary to perform this step to repeat the experiment?

Section 3: Parallel *R-C* dc network

1) Construct the network as shown in Fig. 5.4 Insert the measured resistor values.



Measured R_1 =_______

Measured R_2 =_______

Figure 5.4: Parallel RC DC Circuit

2) Using the measured values, calculate the theoretical steady-state levels (time **greater than** five time constants) of all the voltages and currents for the network of Fig. 5.4 and record in Table 3 in the calculated row.

Table 5.2

	I_1	I_2	I_3	V_1	V_2	V_3
Calculated						
Measured						

3) Energise the system and measure the voltages V_1, V_2, and V_3. Calculate the currents I_1 and I_2 from Ohm's law and the current I_3 from Kirchhoff's current law. Record all the results in the Table 5.2. Compare the results with those of part (2).

End of Experiment 5

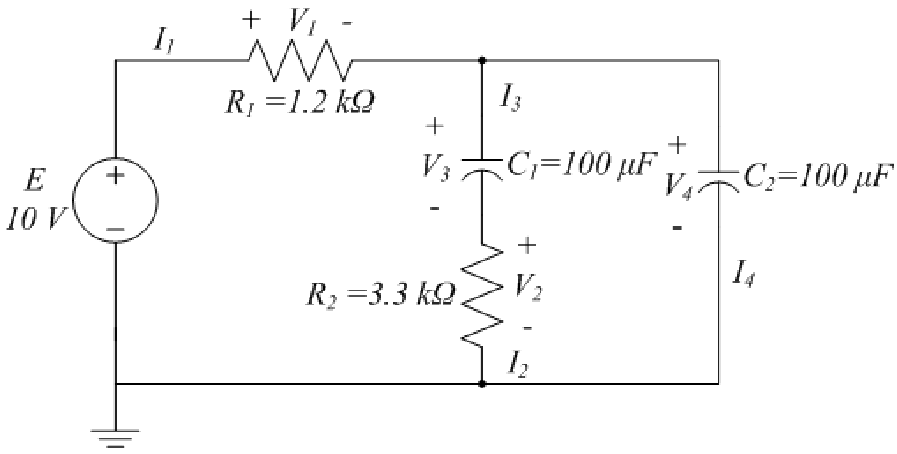
This table is completed by your supervisor.

Supervisor	
Date	
Comments	

Experiment 6 *R-C* circuits

SECTION ONE

1) Construct the network of Fig. 6.1. Insert the measured resistor values.



Measured R_1 =_______

Measured R_2 =_______

Figure 6.1: Series-parallel RC Circuit

2) Assuming ideal capacitors and using measured resistor values, calculate the theoretical steady-state levels of the currents in Table 6.1 and voltages in Table 6.2.

Table 6.1

	I_1	I_2	I_3	I_4
Calculated				

Table 6.2

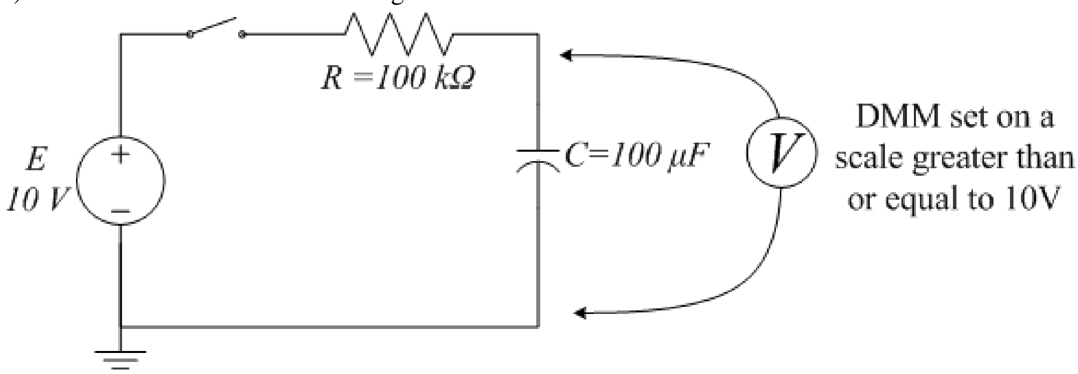
	V_1	V_2	V_3	V_4
Calculated				
Measured				

3) Energise the system and measure the voltages V_1, V_2, V_3 and V_4. Record the results in the bottom row of Table 6.2. Compare the results with those of part (2).

SECTION TWO

This part of the experiment will determine the actual capacitance of the capacitor. In most cases, the actual value will be more than the nameplate value.

1) Construct the network of Fig. 6.2. Record the measured resistor value.



Measured R_1 =_______

Figure 6.2: Circuit used to determine C

Calculate the theoretical time constant determined by the measured resistance value and the nameplate capacitance value. Record in Table 6.3.

Table 6.3

	Theoretical	Measured
τ		

2) Before turning on the power supply or closing the switch, be sure to discharge the capacitor by placing a lead across its terminals. Then energise the source, close the switch and use a timer to note how many seconds pass before the voltage v_c reaches 63.2% of its final value or (0.632)(10 V) = 6.32 V. Recall from the theory that the voltage v_c should reach 63.2% of its final steady-state value in one time constant. Record the measured time constant in Table 6.3.

3) The actual capacitance (measured value) is then defined by

$$C_{\text{measured}} = \frac{\tau(\text{measured})}{R(\text{measured})}$$

Determine C_{measured} for the capacitor of Fig. 6.2, recording the result in Table 6.4.

Table 6.4

	100μF	220μF
C_{meas}		

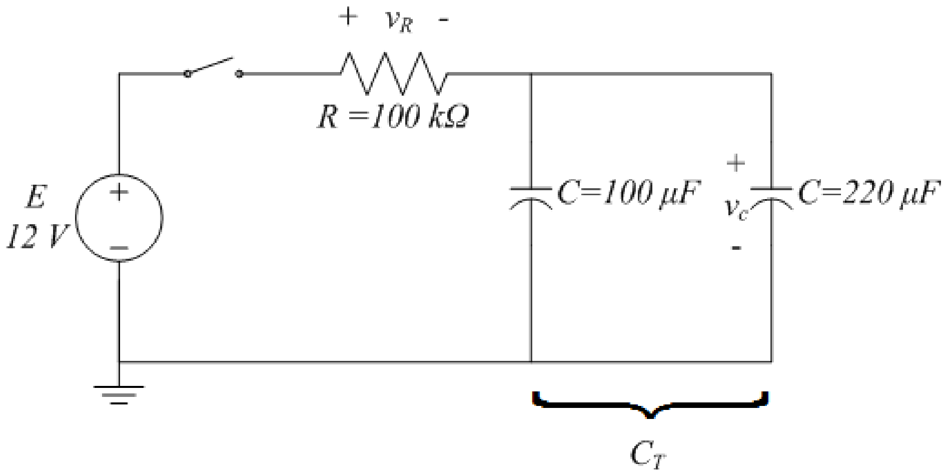
4) Repeat this process and determine C_{measured} for the 220 μF capacitor.

For the rest of this experiment, use the measured value in your calculations for each capacitance.

How do the measured and nameplate values of C compare? What does the difference suggest about the actual versus nameplate levels of capacitance?

SECTION 3: Charging network (parallel capacitors)

1) Construct the network of Fig. 6.3. Note the change in voltage! Insert the measured resistance and capacitance values.



Measured R =_______

Measured C_1=_______

Measured C_2=_______

Figure 6.3: Parallel Capacitors

2) Calculate the total capacitance for the network using the measured capacitance levels. Record in Table 6.5.

Table 6.5

C_T	τ	5τ

3) Determine the time constant for the network and record in Table 6.5.

4) Calculate the charging time (5τ) for the voltage across the capacitor C_T and record in Table 6.6.

5) Using a timer, record (to the best of your ability) the voltage across the capacitor at the time intervals appearing in the table below after the switch is closed. You may want to make a test run before recording the actual levels in the table. Complete the table using the fact that $v_R = E - v_C$. Be sure to discharge the capacitor between each run.

Table 6.6

t(s)	0	10	20	30	40	50	60	70	80	90
v_C (V)										
v_R (V)										

t(s)	100	110	120	130	140	150	160	170	180	200
v_C (V)										
v_R (V)										

6) Plot the curves of v_C and v_R versus time on Fig. 6.4. Label each curve and indicate the intervals 1τ through 5τ on the horizontal axis.

7) Record the level of v_C after one and five time constants from Fig. 6.4. Is v_C approximately 63.2% of nominal after one time constant? Does the level of v_C suggest that the major portion of the transient phase has passed after five time constants?

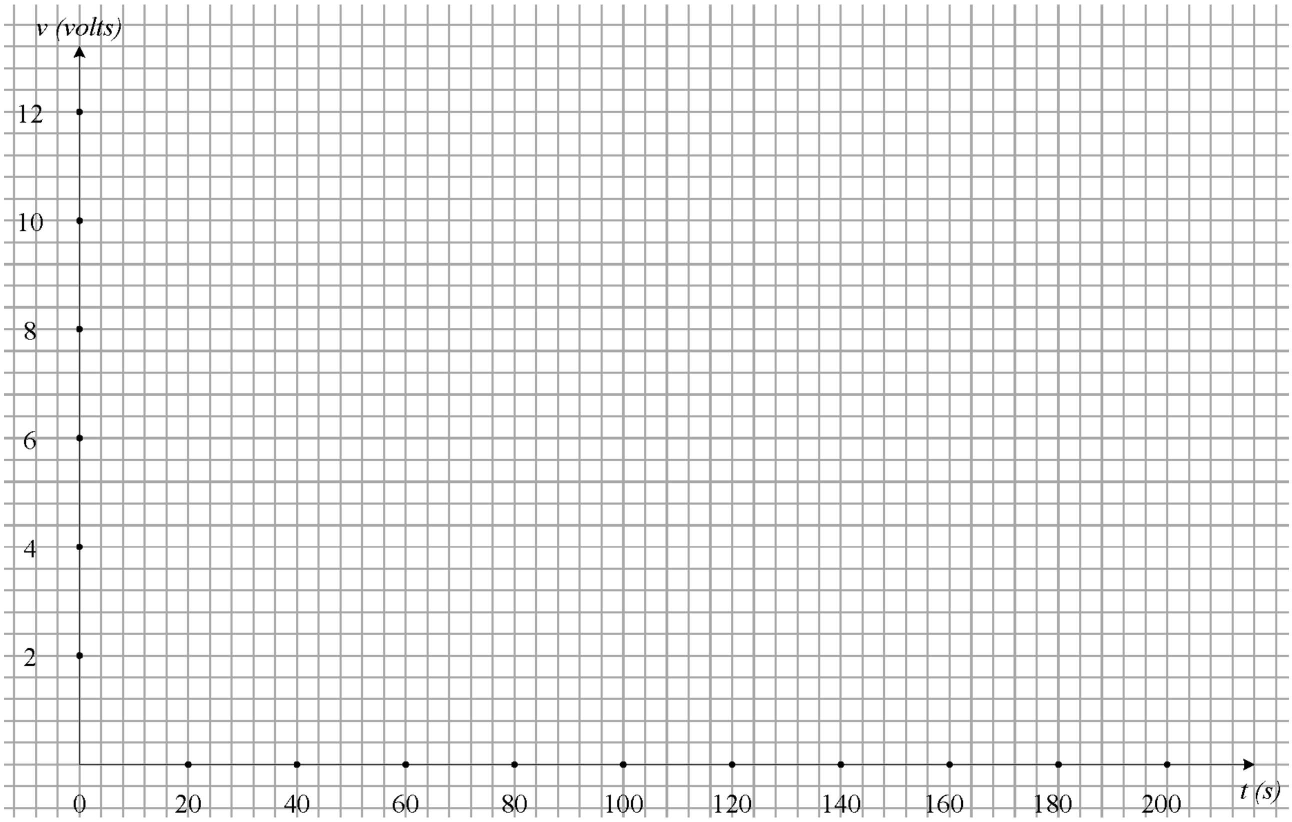


Figure 6.4: Vc and V_R versus time

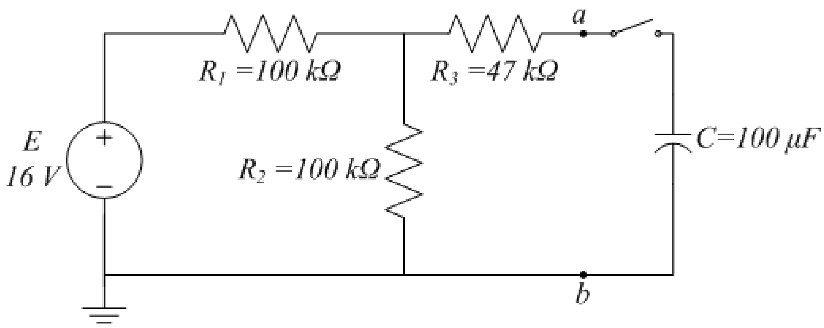
8) Write the mathematical expression for the voltage v_C during the charging phase.

9) Determine the voltage at t = 25 s by substituting the time into the preceding mathematical expression and performing the required mathematical computations.

10) Determine v_C at t = 25 s from the curve of Fig. 6.4 How do the calculated (Part (9)) and measured levels of v_C at t = 25 s compare?

SECTION 4: Applying Thévenin's Theorem

1) Construct the network of Fig. 6.5. Insert the measured resistance and capacitance values.



Measured R_1 =_______

Measured R_2=_______

Measured R_3=_______

Measured C=_______

Figure 6.5: Applying Thevenin

2) Using Thévenin's theorem, calculate the open-circuit Thévenin voltage and Thévenin resistance between terminals a and b for the network to the left of the capacitor.

3) In the space below, redraw the network of Fig. 6.5 with the equivalent Thévenin circuit in place of the circuitry to the left of the switch and capacitor.

4) Calculate the resulting time constant 1τ and charging time 5τ for the voltage across the capacitor *C* after the switch is closed.

5) Write the mathematical expression for the charging voltage v_C and determine the voltage v_C after one time constant.

6) Close the switch for the network of Fig. 6.5 and record the level of v_C after one time constant. How does the value of v_C after one time constant compare with the calculated value of part (5)? Does the result of part (4) validate the Thévenin equivalent circuit?

End of Experiment 6

This table is completed by your supervisor.

Supervisor	
Date	
Comments	