

Preface

The **EE 202 : Fundamentals of Electric Circuits** Lab is designed to help students understand the basic principles of electric circuits as well as giving them the insight on design, simulation and hardware implementation of circuits. The main aim is to provide hands-on experience to the students so that they are able to put theoretical concepts to practice.

Each experiment consists of two parts, ‘simulation’ and ‘hardwired’. Computer simulation is stressed upon as it is a key analysis tool of engineering design. “Multisim Electronics Workbench” software is used for simulation of electric circuits.

Students will also carry out a design experiment in addition to the ten experiments provided in this lab manual. Students will be given a specific design problem, which after completion they will verify using the simulation software.

List of Experiments

Exp No.	Title	Page
1	Electrical Circuits Simulation using Multisim Electronics Workbench: An Introduction	1
2	Electric Circuit Fundamentals	10
3	Resistors in Series, Color Codes & Power Rating	15
4	Kirchhoff's Laws	21
5	Series & Parallel Circuits, Voltage Divider & Current Divider Rules	24
6	Superposition Theorem	29
7	Thevenin's Theorem and Maximum Power Transfer	33
8	Transients of a First Order RC Circuit	37
9	The Oscilloscope and Function Generator	41
10	Sinusoidal AC Analysis	48
Appendix	Laboratory Regulations and Safety Rules	53

Experiment 1

Electrical Circuits Simulation using Multisim Electronics Workbench: An Introduction

Simulation is a mathematical way of emulating the behavior of a circuit. With simulation, you can determine a circuit's performance without physically constructing the circuit or using actual test instruments. Multisim is a complete system design tool that offers a very large component database, schematic entry, full analog/digital SPICE simulation, etc. It also offers a single easy to use graphical interface for all design needs.

Introduction

Go to **Start**→**Programs**→**Multisim** and click on **Multisim**. This will open the main window as shown in Figure 1. In Figure 1 important toolbars and menu are labeled. In addition to toolbars shown in Figure 1, there maybe other toolbars appearing on your screen concentrate on the labeled items in Figure 1 at this time.

You can always open and close a toolbar from Main Menu. For example if you want to open or close (select/unselect) the Design Toolbar, select **View**→**Toolbars**→**Design**. If any toolbar is not appearing on your screen then use the above procedure to bring the toolbar.

Most of the analysis can be performed turning on-off the simulate switch. If the Simulation Switch shown in Figure 1 is not appearing on your screen then select **View**→**Show Simulate Switch** in the Main Menu. This will open the Simulation Switch.

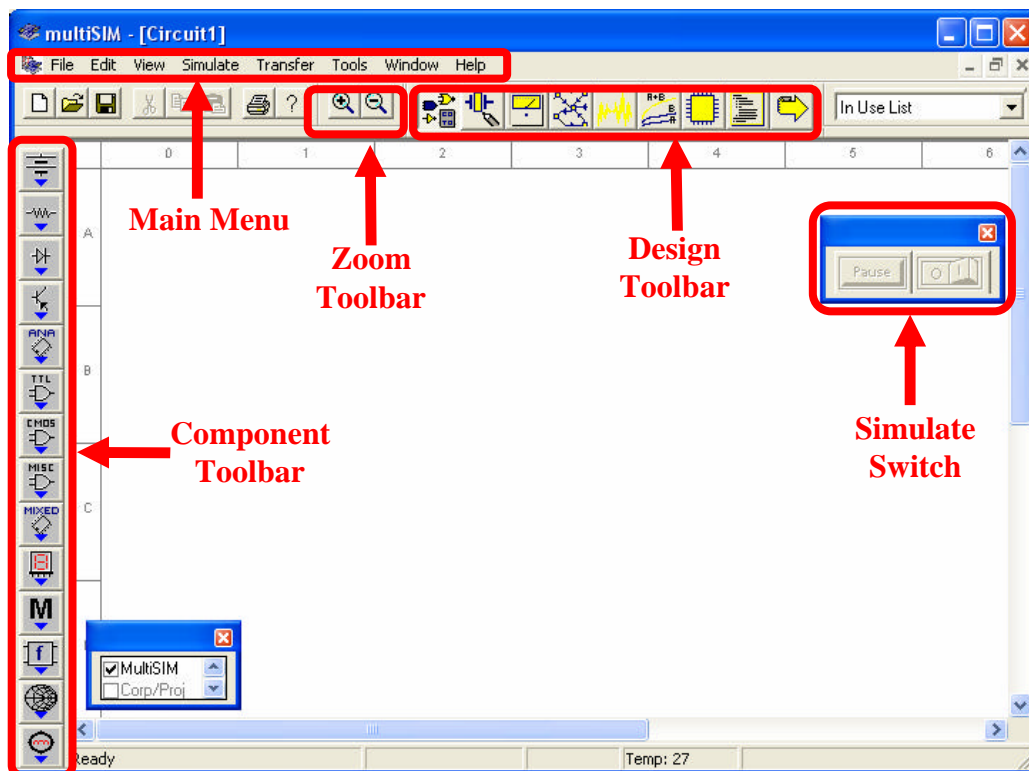


Figure 1: Main Window of Multisim Simulation Software

We will now try to learn about Multisim simulation techniques by solving a simple example.

Example

Build the circuit shown in Figure 2 using *Multisim Electronics Workbench*.

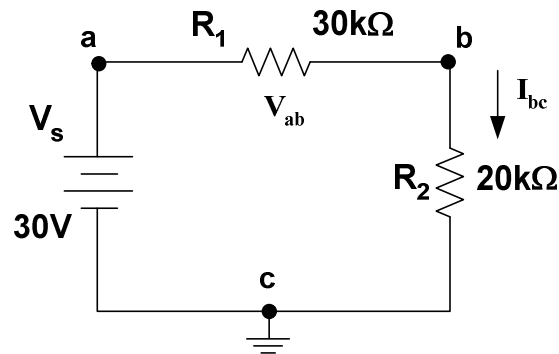


Figure 2: Circuit for Multisim Simulation

STEP A: Placing the Components

1. Place a Battery (DC Source)

a. Bring a dc source in the Multisim workspace:

Open the Multisim program if it is not open. In the Component Toolbar, select **Sources** icon (refer to Figure 1 to find the Component Toolbar). This will open another window with several types of dc sources and other components as shown below in Figure 3. Click on “DC Voltage Source” in this new window.

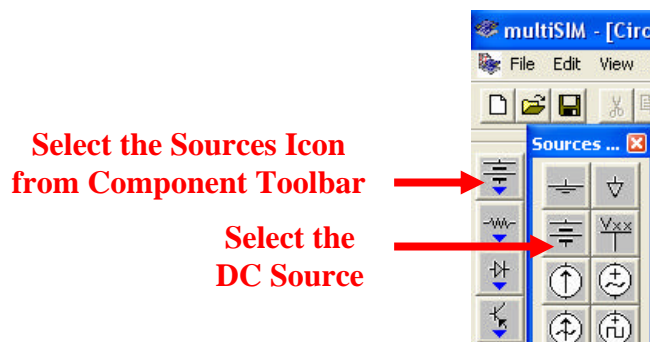


Figure 3: Selection of DC Voltage Source in Multisim

Now bring your cursor in the workspace area and notice the change in the shape of cursor to



Click at any point in the workspace. This will put the voltage source as



b. Change the value and name of voltage source:

Double click on the voltage source that you just placed in the workspace, a new window with the name **Battery** will appear, as shown in Figure 4. Select **Value** in the Battery menu, if it is not already selected. Change the value from 12 to 30. Keep the unit as Volts in this menu. Now select **Label** in this menu and change the **Reference ID** to Vs. Click on **OK**.

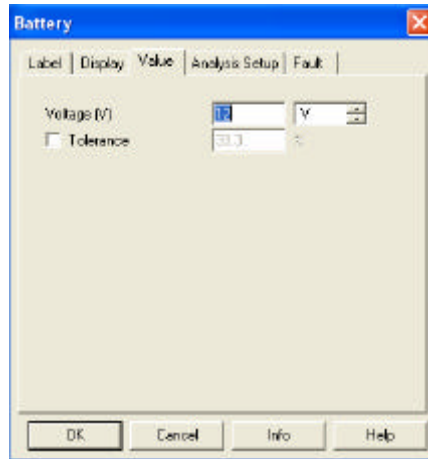


Figure 4: Battery Window for setup of DC voltage source

2. Place a Resistor:

a. Bring a resistor in the Multisim workspace:

In the Component toolbar, select **Basic** icon as shown. This will open another window with several basic components as shown below in Figure 5.

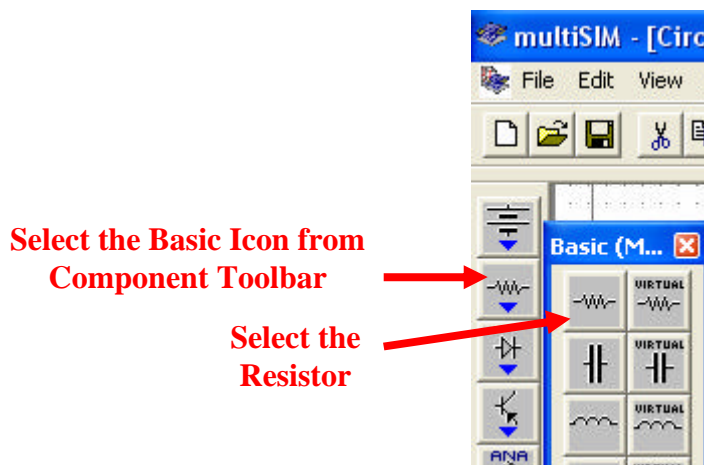
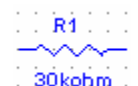


Figure 5: Selection of Resistor in Multisim

Click on “Resistor”, this will open the **Browser-Basic** window, as shown in Figure 6. Scroll through the **Component List**, select 30kohm, and click OK. The cursor shape will change again. Click in the workspace and this will put the resistor as,



Tip To make your scroll through the Browser's **Component List** faster, simply type the first few characters of the component's name. For example, type 30k to move directly to the area of 30kohm list.

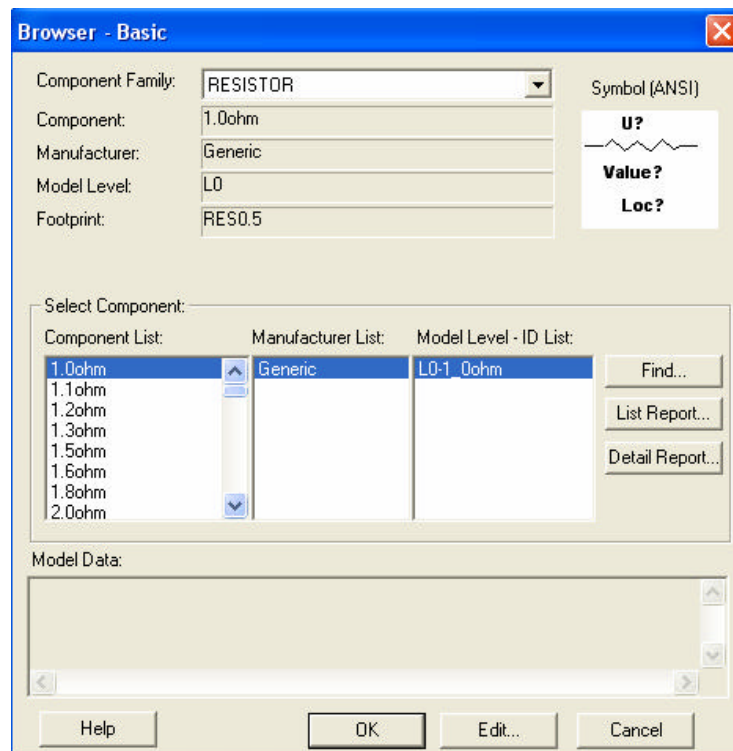


Figure 6: Setup window for Resistor values

b. Change the name of resistor:

Double click on the resistor, a new window with the name **Resistor** will open as shown in Figure 7. Select Label from the menu of this window. Change the **Reference ID** to R₁ (if it is not) and press OK. This will change the name of the resistor to R₁.



Figure 7: Battery Window for Label of Resistor

d. Add other resistor R_2

Place resistor R_2 of value $20\text{k}\Omega$ in the workspace through the same procedure.

e. Rotate the resistor:

Select resistor ' R_2 ' and press **Ctrl-R** to rotate the resistor or select **Edit→90 Clockwise** from the Main Menu. This will make the resistor vertical. Labels and values of all the components can be dragged individually. Drag the label ' R_2 ' and value ' 20kohm ' individually to put them at a proper place.

3. Place Ground:

In the Component Toolbar, select **Sources** icon. Now click on **Ground** icon in the new window as shown in Figure 8. Click in the workspace to put the Ground symbol as

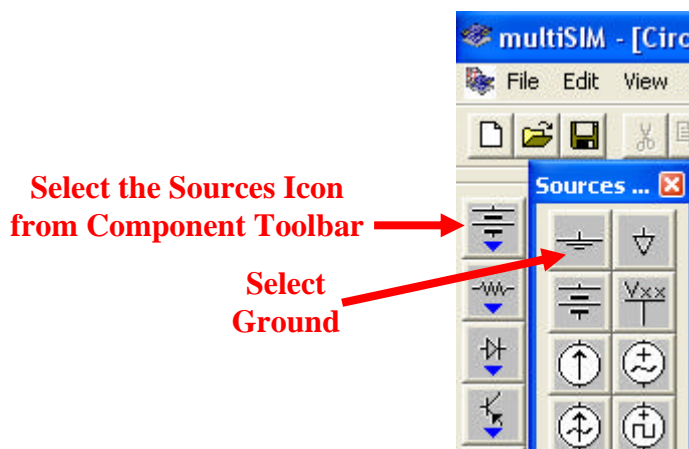


Figure 8: Selection of Ground in Multisim

STEP B: Connecting the Components

1. Arrange the components properly:

Arrange the components according to the circuit given in Figure 2. You can select and drag the component to any place in the workspace. Select the components and drag them one by one to proper places as shown in Figure 9.

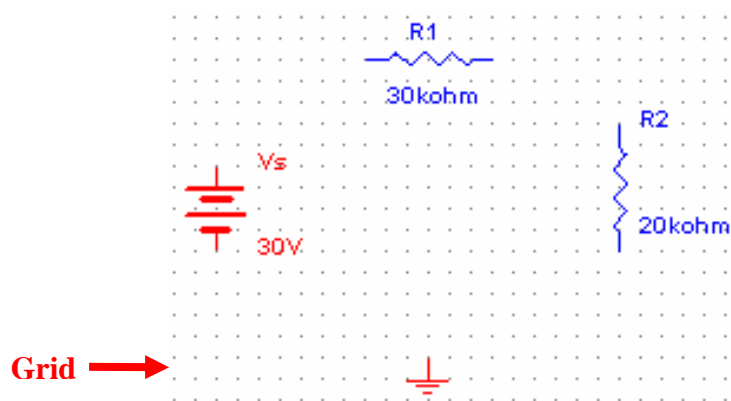


Figure 9: Arranging the Components in proper order

2. Show Grid in the workspace:

You may show grid for ease of drawing the connections. Select **View**→**Grid Visible** in the Main Menu if it is not visible.

3. Connect DC Voltage Source “Vs” to “R₁”:

Bring the cursor close to upper pin of “Vs”; cursor shape will change to a plus sign. Click and move a little upward. A wire appears, attached to the cursor. Click again at a small distance above the “Vs” source. Notice that the line will change direction. Control the flow of the wire by clicking on points as you drag. Each click fixes the wire to that point as shown in Figure 10. In this way, when the cursor reaches the pin of R₁ click again, this will connect “Vs” to “R₁” in a nice manner. Notice that a node number is automatically given.

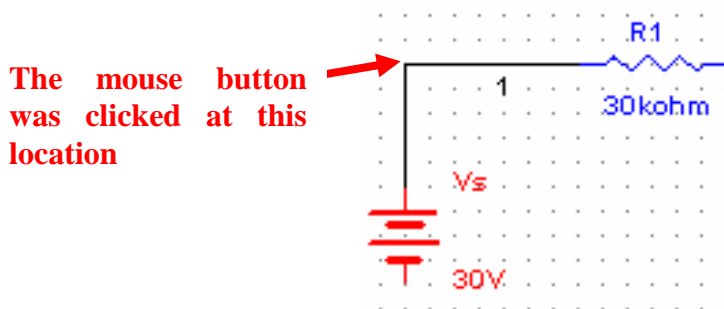


Figure 10: Manual connection of components

4. Connect “R₁” to “R₂”:

In the same way connect R₁ to R₂ through the same procedure.

5. Making use of Junction to connect Ground:

In the similar manner connect ground with V_s and R₂. Notice that a small black circle appears just above the ground, this is called junction. When 2 or more components are connected at one point, a junction is created. A junction can also be placed manually by pressing Ctrl+J or selecting **Edit**→**Place Junction**. This can be used to control the connection points manually. Also notice the ground node is automatically given node number 0. Do not alter it.

This completes the connection and the complete circuit is shown in Figure 11.

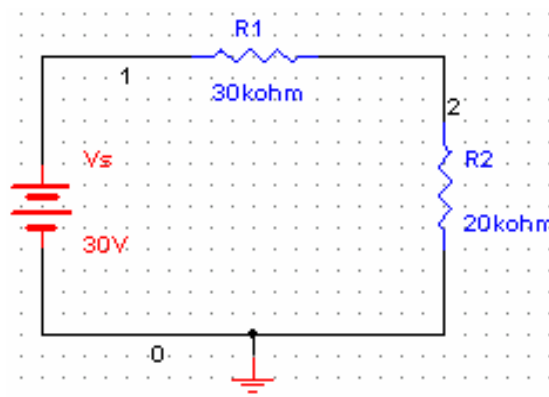


Figure 11: Complete Circuit in Multisim

6. Wire paths can be modified using drag points. Click on a wire. A number of drag points will appear on the wire as shown in Figure 12. Click any of these and drag to modify the shape. You can also add or remove drag points to give you even more control over the wire shape. To add or remove drag points, press CTRL and click on the location where you want the drag point added or removed.

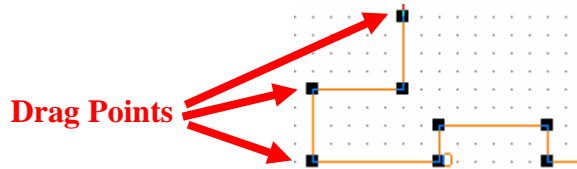


Figure 12: Drag points for connecting wire

STEP C: Placing Multimeter or Voltmeter in parallel to measure voltage.

1. To connect a Multimeter:

- a. Select **View** → **Toolbar** → **Instruments**. The Instruments toolbar will open as shown in Figure 13.

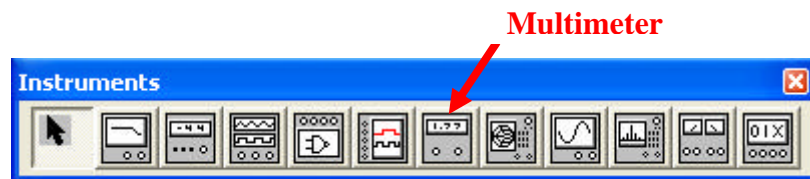


Figure 13: Instruments Toolbar

- b. Click on **Multimeter** icon. Now click in the workspace to place the Multimeter. Drag it and place it near resistor R_1 as shown in Figure 14. Make a connection from '+' terminal of Multimeter to the left pin of R_1 and from '-' terminal to right pin of R_1 . Note that reversal of + and - terminals will give opposite readings.

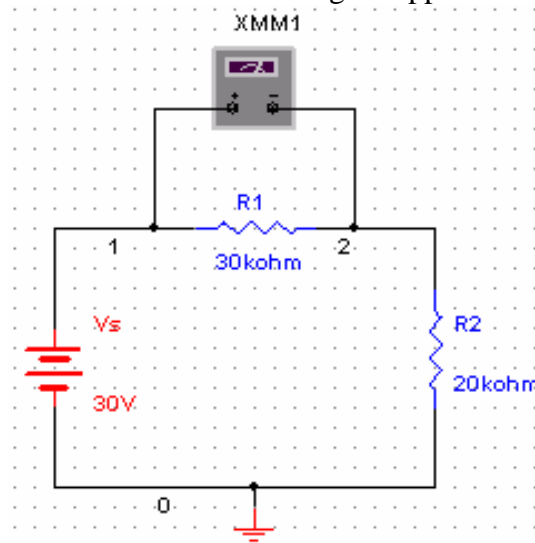


Figure 14: Multimeter connection for voltage measurement

c. Set the Multimeter to measure DC voltage:

Double click on Multimeter to open the properties window shown in Figure 15. Select 'V' to measure voltage. Select the DC wave shape. (Notice that the meter can also measure current 'A' and resistance ' Ω '. It can measure AC as well as DC values. Leave the window open for viewing the measurements.



Figure 15: Multimeter properties window

STEP D: Placing a Multimeter or Ammeter in series to measure current.

1. Place a second Multimeter in the workspace as we did in Step C. Remove the connection between R_1 and R_2 . Connect the '+' terminal of the Multimeter towards R_1 and the '-' terminal towards R_2 as shown in Figure 16.
2. **Set the Multimeter to measure current:**
Double click on this multimeter and select 'A' in the multimeter properties window. Set the wave shape to DC. If current flows from 3 to zero, the meter will read positive.

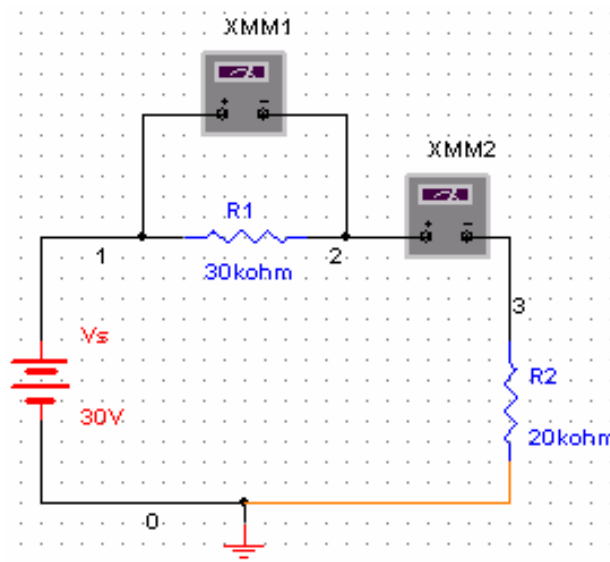


Figure 16: Multimeter connection for current measurement

STEP E: Simulate the circuit.

1. Save the file.

Select **File**→**Save**

2. Show the Simulate Switch, on the workspace.

Select **View**→**Simulate Switch**.

3. If the properties window is not open, double click the multimeters. Click to '1' position (ON) of the simulation switch to start simulation. Results will appear in the properties window of Multimeter. Compare your result with those in Figure 17 and show them to your instructor.

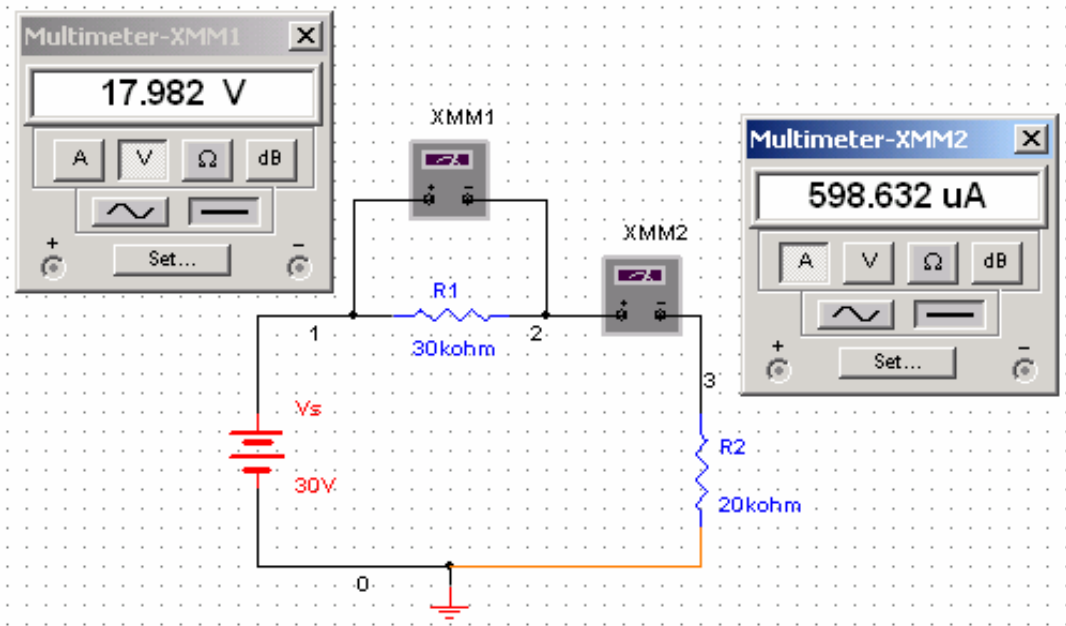
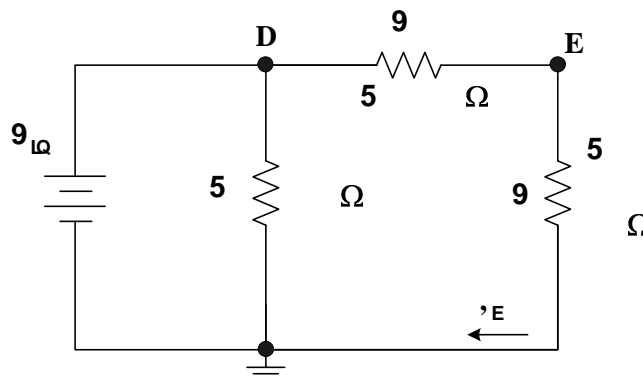


Figure 17: Simulation result

Exercise

1. Build the circuit of Figure 18 in *Multisim Electronics Workbench*.
2. Connect a voltmeter between nodes 'a' and 'b'.
3. Connect an ammeter for the measurement of I_{b0} .



Experiment 2

Electric Circuit Fundamentals

Introduction

This experiment has two parts. Each part will have to be carried out using the *Multisim Electronics Workbench* software. The experiment will then be repeated with hardwired laboratory equipment. The experiments involve measurement of voltage and current in a dc circuit and verification of ohm's law.

PART I: VOLTAGE AND CURRENT IN A DC CIRCUIT

Objectives

1. Investigate how to use a voltmeter to measure voltage across a circuit component
2. Investigate how to use an ammeter to measure current flow in a circuit component
3. Investigate what happens if the rated value of a component is exceeded (only simulation will be used to investigate this step)

Materials

- One dc power supply
- One dc 0-20V Voltmeter
- One dc 0-100mA Ammeter
- One Multimeter
- One 5V, 1W lamp

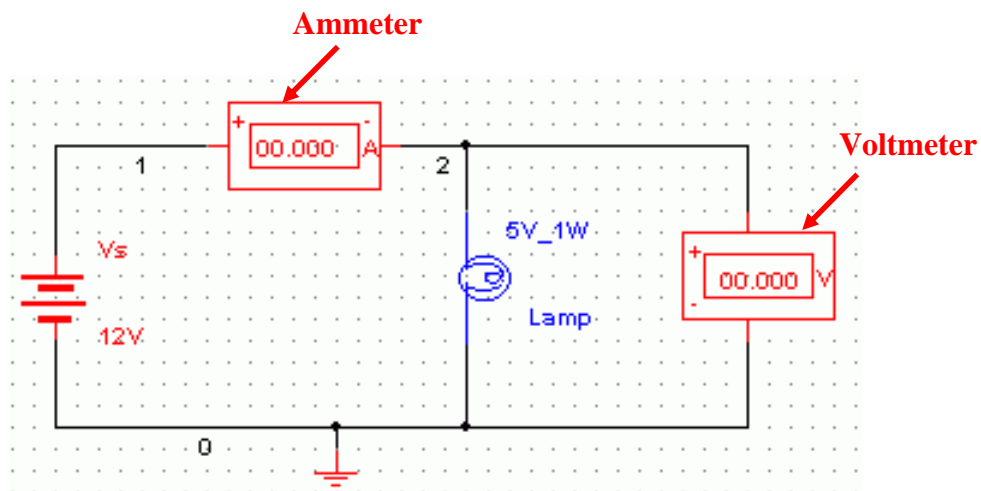


Figure 1: Measuring current and voltage: a lamp connected across a battery

Procedure

Simulation

1. Construct the circuit shown in Figure 1 on the *Multisim Electronics Workbench*. Use 5V, 1W lamp for simulation. (In the **Components Toolbar** select the **Indicator** icon; now find **Voltmeter**, **Ammeter** and **Lamp** in this window).
2. Set the dc supply voltage to 3V and click the **Simulate Switch** and verify that the battery voltage is 3V as measured by the voltmeter. Record the voltage across the lamp terminals 'V' and the current 'I' flowing through it in Table 1. Calculate power dissipation in the lamp using the relationship $P=VI$ and note it down.
3. Change the dc supply voltage to 5V. Run the analysis again. Record voltage and current in Table 1 and calculate power dissipation in the lamp.
4. Change the dc supply voltage to 7V. Run the analysis and see the effect on the intensity of light. Record voltage, current in Table 1 and calculate power dissipation.
5. Change the dc supply voltage to 8V. Run the analysis and observe the value of current, also observe the glow of the lamp. What happened? Explain

Table 1: Simulation results using *Multisim Electronics Workbench*

Source Voltage (V)	Lamp Voltage (V)	Current (A)	Power , VI (W)
3			
5			
7			

Note: The lamp is rated at 5V, 1W

Question: Why the lamp is damaged when the voltage across it goes to 7V? Explain by comparing power dissipation with rated value.

Hardwired Experiment

6. Build the circuit of Fig. 1 with hardwired laboratory components. Before switching on the DC power supply set its voltage to 0 volts.
7. Turn the power supply on. Increase the supply voltage to 3V. Enter the voltage and current in Table 2.
8. Increase the supply voltage to 5V and repeat step 7.

Note: Do not raise the battery voltage beyond 5V as it will burn the lamp.

Table 2: Hardwired experimental results

Source Voltage (V)	Lamp Voltage (V)	Current (A)	Power , VI (W)
3			
5			

Question: Compare the *Multisim Electronics Workbench* results with the hardwired laboratory ones. Comment on the results.

PART II: OHM'S LAW - RESISTANCE

Objectives

1. Learn how to use multimeter to measure resistance
2. Verify Ohm's law
3. Determine the relationship for voltage and current for constant resistance
4. Determine value of resistance from slope of I-V characteristic curve.

Materials

- One dc variable voltage power supply
- One multimeter
- One 0-100mA Ammeter
- Resistor (100Ω)

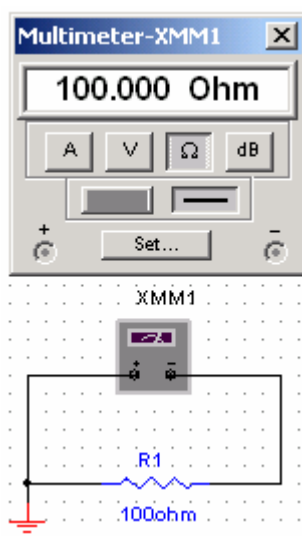


Figure 2: Measuring resistance

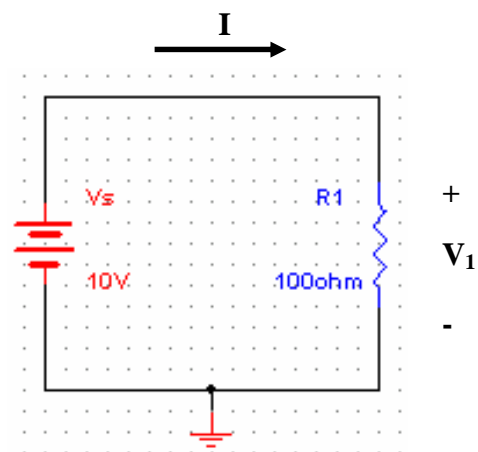


Figure 3: Verifying Ohm's Law

Procedure

Simulation

1. Build the circuit of Figure 2 using *Multisim Electronics Workbench*. Connect a multimeter between the terminals of resistor and set it to read resistance. Select $R_1=100\Omega$. Click the **Simulation Switch** to run analysis. Record the value of resistance R_1 in Table 3.
2. Build the circuit given in Figure 3. Set $V_s = 10V$ and $R_1=100\Omega$. Click **Simulation Switch** to run analysis. Record voltage ' V_1 ' across resistor R_1 by connecting a multimeter in parallel to it. Record the value of current ' I ' flowing through R_1 by connecting another multimeter in series to R_1 . Note down the values in Table 3. From the voltage current readings verify Ohm's law $V_1=R_1 I$. Considering multimeter reading as reference, calculate % error.
3. Vary the dc supply voltage V_s in steps of 2V and record current in each case. Enter your results in Table 4.
4. Plot ' I ' vs. ' V ' in the graph of Table 5.
5. Calculate the value of resistor based on the slope of the V-I characteristic curve plotted in step 4.

Hardwired Experiment

6. Repeat steps 1-5 with the laboratory hardwired components.

Note: Use different color pen to plot the experimental results in Table 5.

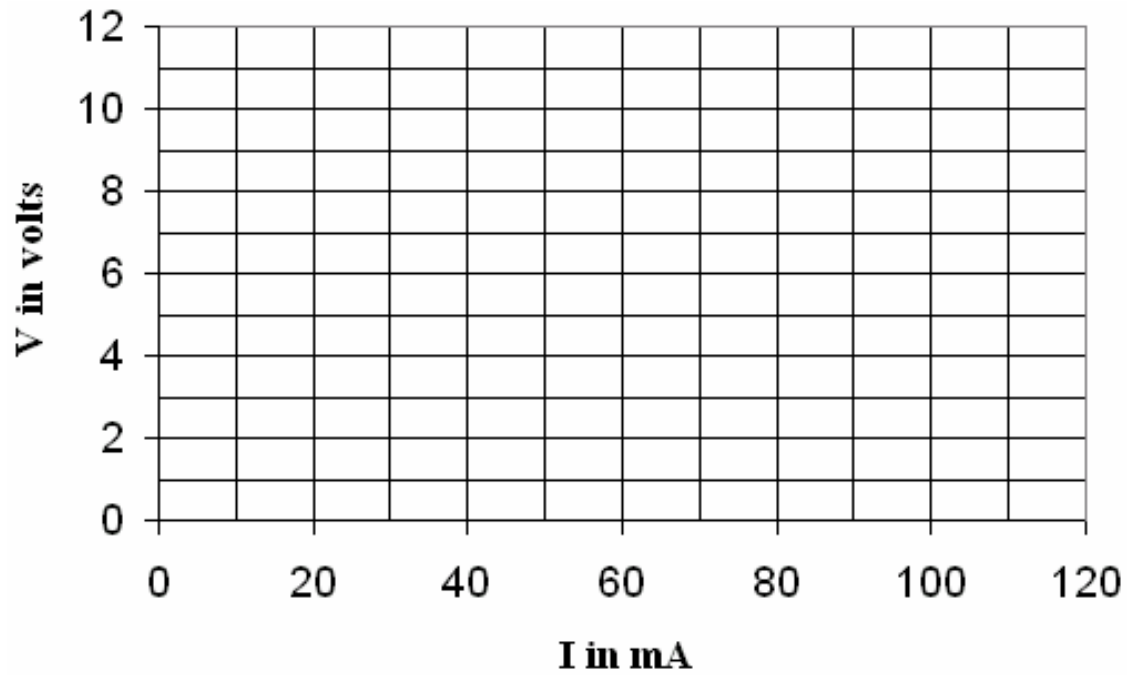
Table 3: Resistance Measurement

	Ohmmeter Reading	Ohm's Law (V_1/I)
Workbench	$R_1=$	$R_1=$
Hardwired	$R_1=$	$R_1=$
% Error		

Table 4: V-I measurements

Workbench		Hardwired	
V (volt)	I (mA)	V (volt)	I (mA)
0		0	
2		2	
4		4	
6		6	
8		8	
10		10	

Table 5: V-I plot



Resistance Measurement from V-I slope

- Workbench experiment R=
- Hardware experiment R=

Questions

1. Compare the values of resistance obtained with ohmmeter, Ohm's law (V/I) and slope of the V-I plot. Comment on your results.

2. How could you tell if the resistances are linear or not?

Any other observations or comments

Experiment 3

Resistors in Series, Color Codes & Power Rating

Introduction

This experiment is about measurement of resistance directly and also through voltage and current measurement. The experiment has two parts – the first part using *Multisim Electronics Workbench*, and the second part through hardware components.

PART I: RESISTORS IN SERIES (USING MULTISIM)

Objectives

1. Measure the equivalent resistance of a series circuit
2. Determine current in each resistor
3. Determine voltage across each resistor
4. Determine equivalent resistance
5. Demonstrate Kirchhoff's voltage law

Materials

One dc voltage supply
One multimeter
Five dc 0-10 V voltmeters
Five dc 0-5mA ammeters
Assorted resistors

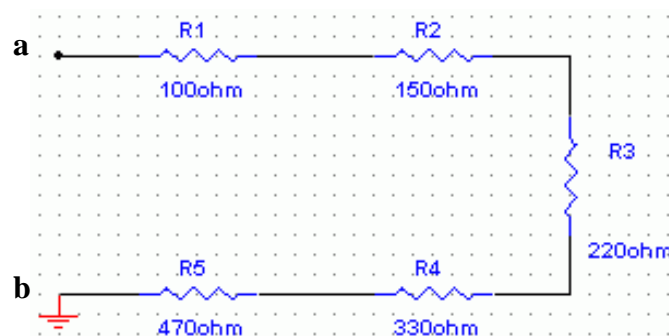


Figure1. Measurement with multimeter

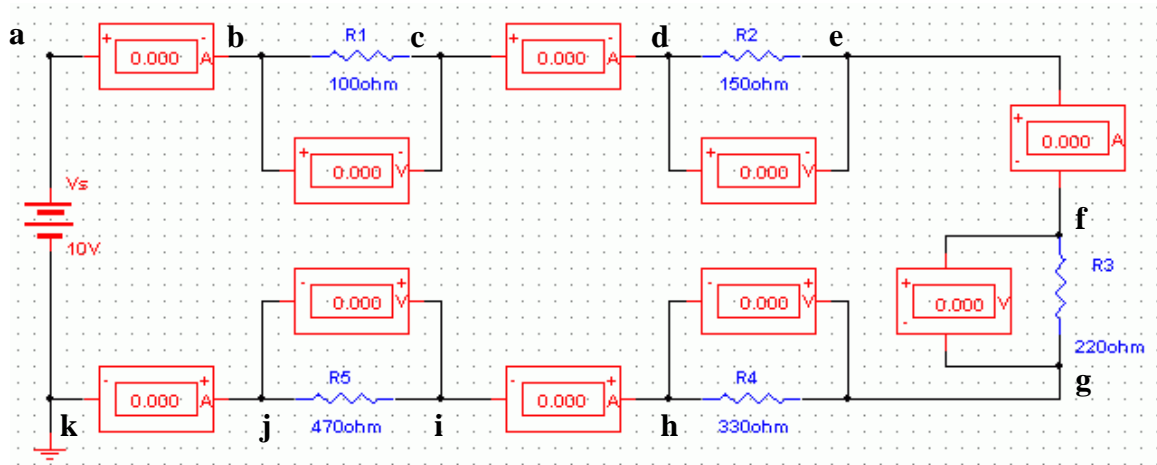


Figure 2: Resistors in series

Procedure

1. Draw Figure 1 using the Multisim simulation software. Connect the multimeter as ohmmeter between 'a' and 'b'. Click the simulate switch to run analysis. Record the measured resistance. Calculate the total value just by addition and record it.

$$R_{eq} \text{ (measured)} =$$

$$R_{eq} \text{ (calculated)} =$$

Question: How did the measured value compare with the calculated value? Comment.

2. Draw Figure 2. Run the simulation and record the currents and voltages.

I_{ab}	I_{cd}	I_{ef}	I_{hi}	I_{jk}	V_{bc}	V_{de}	V_{fg}	V_{gh}	V_{ij}

3. Based on the equivalent resistance R_{eq} calculated in step 1 and value of the voltage source (V), calculate the source current (I_s).

$$I_s =$$

Question: How did the measured current I_{ab} compare with the calculated current (I)?

4. From the measured value of voltages across the resistances, calculate the current. Compare these calculated values of currents with the measured values of currents.

Measured	I_{ab}	I_{cd}	I_{ef}	I_{hi}	I_{jk}
Calculated	$I_{ab}=V_{bc}/R_1$	$I_{cd}=V_{de}/R_2$	$I_{ef}=V_{fg}/R_3$	$I_{hi}=V_{gh}/R_4$	$I_{jk}=V_{ij}/R_5$

Question: How do they compare?

5. Calculate the sum of the voltages V_{bc} , V_{de} , V_{fg} , V_{gh} , V_{ij}

$$V_{bc} + V_{de} + V_{fg} + V_{gh} + V_{ij} =$$

Question: What is the relationship between the supply voltage (V_S) and the sum obtained? Does it confirm Kirchhoff's law? Explain.

PART II: RESISTORS, MEASUREMENTS AND COLOR CODES (HARDWIRED EXPERIMENT)

Objectives

1. Determine the resistance of a selection of carbon resistors by color codes.
2. Compare values obtained with voltage current readings.
3. Observe power dissipation property of carbon resistors.

Materials

DC power supply
Digital multimeter
Assorted resistors

Color Codes

The approximate value of a carbon resistor can be found by 4 color bands on it. The 9 colors in the sequence are black, brown, red, orange, yellow, green, blue, violet, grey and white. Carbon resistors may have a fifth band which indicates reliability of the resistor. Figure 3 shows the color code structure.

Example: Brown Red Orange Gold
 1 2 10^3 5% = 12 K Ω , $\pm 5\%$

Procedure

1. Get 5 carbon resistors of the same value as you used in Figure 1. Find the nominal value and the tolerance of each resistance using the color codes and record in Table 1. Note the relation between the power rating and the physical size of the resistance.
2. Using the digital multimeter as an ohmmeter, measure and record the resistance of each resistor.
3. Connect the circuit as shown in Figure 2. Set the source voltage to 12 V. Measure all the voltages and currents and find the values of resistance using ohm's law i.e. $R=V/I$. Record your results in Table 1.

Table 1: Resistor values

Resistor	R1	R2	R3	R4	R5
Color codes					
Nominal value					
Tolerance					
Ohmmeter reading					
V / I					
% deviation from nominal value					
Actual value within tolerance?					

Answer the Following Questions

1. An electric heater takes 1.48 kW from a voltage source of 220 V. Find the resistance of the heater.
2. If the current in a resistor doubles, what happens to the dissipated power? (Assume the resistor operates in the linear region).

3. A 4Ω resistor is needed to be used in circuit where the voltage across the resistor is 3V. If two 4Ω resistor with 1W and 3 W power rating are available, which one will you use and why?
4. Consider a resistor with a positive temperature coefficient. The resistor operates at higher power level than its rating when current of 1A passes through it. The resulting voltage across the resistor in this case is 20V. If the current is increased to 1.1A, do you expect the voltage to be:
- Less than 22V.
 - Equal to 22 V.
 - Higher than 22 V.

Explain the reason for your expectation.

Any other observations or comments

Resistor Color Code Chart

Color	1st Band (1st Figure)	2nd Band (2nd Figure)	3rd Band (Multiplier)	4th Band (Tolerance)
Black	0	0	10^0	
Brown	1	1	10^1	
Red	2	2	10^2	$\pm 2\%$
Orange	3	3	10^3	
Yellow	4	4	10^4	
Green	5	5	10^5	
Blue	6	6	10^6	
Violet	7	7	10^7	
Gray	8	8	10^8	
White	9	9	10^9	
Gold			10^{-1}	$\pm 5\%$
Silver			10^{-2}	$\pm 10\%$

Figure 3: Resistor color codes

Experiment 4

Kirchhoff's Laws

Introduction

Kirchhoff's voltage law (KVL) states that the algebraic sum of all voltages around any closed path equals zero. The Kirchhoff's current law (KCL) states that the algebraic sum of all the currents at a node is zero (current entering a node has opposite sign to the current leaving the node). This experiment studies these two laws using *Multisim Electronics Workbench*, then with actual hardwired circuit.

Objectives

1. Voltage current measurement in a dc circuit
2. Verification of Kirchhoff's voltage and current laws.

Materials

One dc power supply
One multimeter
Assorted carbon resistors

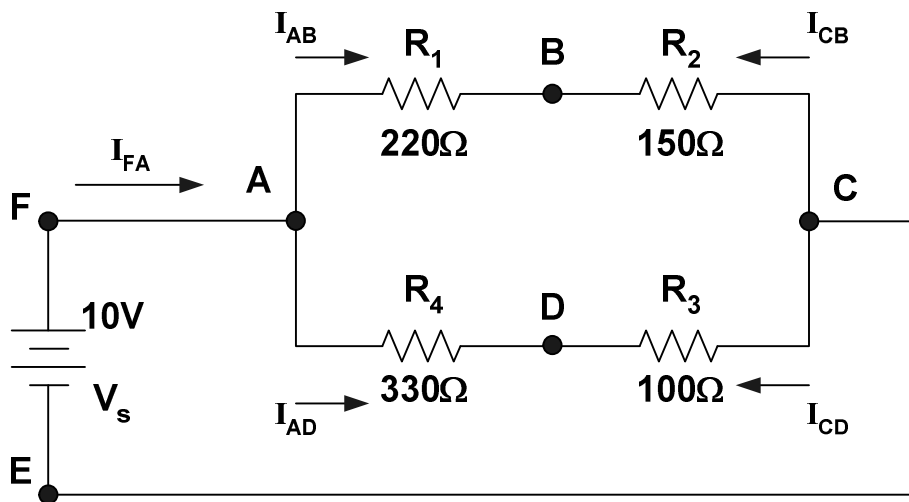


Figure 1: Resistive circuit

Procedure

1. Construct the circuit shown in Figure 1 using *Multisim Electronics Workbench*.
2. Measure the voltages V_{AB} , V_{BC} , V_{AD} , V_{DC} , V_{BD} , and V_{AC} . Enter the values in Table 1. Note the polarities of the voltages.
3. Measure the currents I_{AB} , I_{CB} , I_{AD} , I_{CD} and I_{FA} and enter the values in Table 2. Note the polarity (sign) of the currents.

4. Calculate the voltages around the following loops and record them in Table 4.

ABCEFA, ABDA, BDCB, ABCDA

5. Verify KCL by adding the currents at nodes A, B, C, D. Enter your results in Table 3.

6. Construct the circuit in Figure 1 with hardware components. Repeat steps 2-5. Enter your results in Tables 1-4. Considering the Workbench results as the base compute the percentage errors.

Table 1: Voltage measurement

	V_{AB}	V_{BC}	V_{AD}	V_{DC}	V_{BD}	V_{AC}
Workbench						
Hardwired						
% Error						

Table 2: Current measurement

	I_{AB}	I_{CB}	I_{AD}	I_{CD}	I_{FA}
Workbench					
Hardwired					
% Error					

Table 3: Sum of currents at nodes

	A	B	C	D
Workbench				
Experiment				
% Error				

Table 4: Voltages around loop

	ABCEFA	ABDA	BDCB	ABCDA
Workbench				
Experiment				
% Error				

Answer the following Questions:

1. Do the experimental and theoretical values of voltages and currents agree?
2. Give possible reasons for any discrepancies.
3. Are KVL and KCL verified?
4. Give reasons for any discrepancies.

Any other observations or comments

Experiment 5

Series & Parallel Circuits

Voltage Divider & Current Divider Rules

Introduction

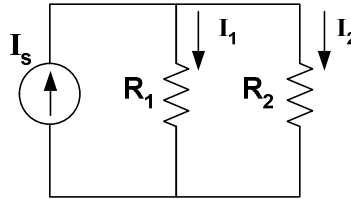
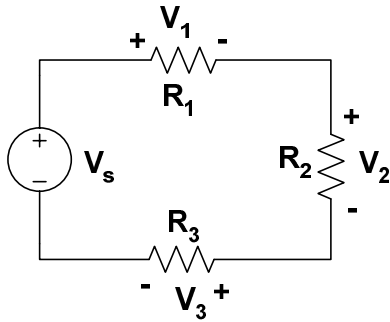


Figure 2: Parallel circuit

For a series circuit shown in Figure 1, the voltages across resistors R_1 , R_2 and R_3 can be written as,

$$\begin{aligned}V_1 &= \frac{R_1}{R_1 + R_2 + R_3} V_s \\V_2 &= \frac{R_2}{R_1 + R_2 + R_3} V_s \\V_3 &= \frac{R_3}{R_1 + R_2 + R_3} V_s\end{aligned}\tag{1}$$

This is the voltage divider rule (VDR).

For a parallel circuit given in Fig. 5.2, the branch currents can be written in terms of the total current as,

$$\begin{aligned}I_1 &= \frac{R_2}{R_1 + R_2} I_s \\I_2 &= \frac{R_1}{R_1 + R_2} I_s\end{aligned}\tag{2}$$

This is termed as the current divider rule (CDR).

Objectives

1. To study the voltage current relationships of series and parallel circuits
2. To verify the voltage current divider and voltage divider rules.

Materials

One dc power supply
One multimeter
Assorted resistors

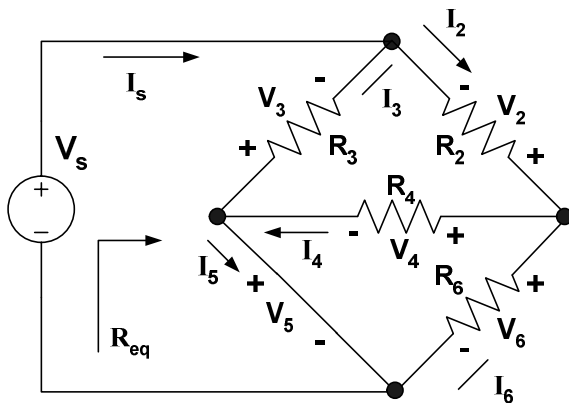


Figure 3: Series-parallel circuit I

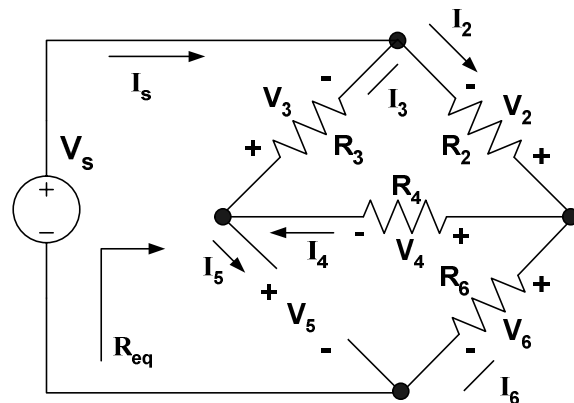


Figure 4: Series-parallel circuit II

$$R_2 = 100\Omega, R_3 = 150\Omega, R_4 = 220\Omega, R_6 = 330\Omega$$

Procedure

Simulation

1. Build the circuit given in Figure 3 on *Multisim Electronics Workbench*.
2. Connect voltmeters, ammeters (or multimeters) at appropriate positions to measure voltages and currents shown in Table 1.
3. Disconnect the voltage source. Connect a multimeter and measure the total resistance and record the value in Table 1. (Remember resistance is always measured without any source connected to the circuit)
4. Repeat steps 2 and 3 for the circuit given in Figure 4 and record the values in Table 2.

Hardwired Experiment

5. Build the circuit of Figure 3 with the hardwired components. Take the voltage current measurements and R_{eq} and record in Table 1. Considering the Workbench results as the base compute the percentage errors.
6. Build the circuit of Figure 4 with the hardwired components. Take the voltage current measurements and R_{eq} and record in Table 2. Considering the Workbench results as the base compute the percentage errors.

Table 1: Simulation and experimental results for Figure 3

	I_s	I_2	I_3	I_4	I_5	I_6	V_2	V_3	V_4	V_5	V_6	R_{eq}
Workbench												
Hardwired												
% Error												

Table 2: Simulation and experimental results for Figure 4

	I_s	I_2	I_3	I_4	I_5	I_6	V_2	V_3	V_4	V_5	V_6	R_{eq}
Workbench												
Hardwired												
% Error												

Questions

Refer to Figure 3 and the results obtained in Table 1 and answer the following questions:

1. Are R_4 and R_6 in parallel or in series? Why? Refer to voltage current measurements for your answer to justify.
2. Are R_3 and R_4 in parallel or in series? Why? Justify
3. Are V_s and R_3 in parallel or in series? Why? Justify
4. Are V_s and R_6 in series or in parallel? Why? Justify.
5. Are V_s and R_{eq} in parallel or in series? Why? Justify

6. Is VDR applicable for applicable R_3 and R_4 ? Why? Justify your answer on the basis of theory given in the introduction.

7. Is CDR applicable for R_4 and R_6 ? Why? Justify your answer on the basis of theory given in the introduction.

8. Is the parallel combination of R_4 and R_6 in series or in parallel with R_2 ? Why? Justify.

Refer to Figure 4 and the results obtained in Table 2 and answer the following questions:

9. Are R_4 and R_6 in parallel or in series? Why? Refer to voltage current measurements for your answer to justify.

10. Are R_3 and R_4 in parallel or in series? Why? Justify

11. Are V_s and R_3 in parallel or in series? Why? Justify

12. Are V_s and R_6 in series or in parallel? Why? Justify.

13. Are V_s and R_{eq} in parallel or in series? Why? Justify

14. Is VDR applicable for applicable R_3 and R_4 ? Why? Justify your answer on the basis of theory given in the introduction.

15. Is CDR applicable for R_4 and R_6 ? Why? Justify your answer on the basis of theory given in the introduction.

16. Is the parallel combination of R_4 and R_6 in series or in parallel with R_2 ? Why? Justify.

Any other observations or comments

Experiment 6

Superposition Theorem

Introduction

If there is more than one source in an electric network, the response (voltage or current) can be determined by considering one source at a time. The total response is the algebraic sum of the individual responses. This is known as the superposition principle. While determining the responses with a particular source, all other sources have to be deactivated (voltage sources replaced by short circuits and current sources by open circuit).

Objectives

1. To verify superposition principle using *Multisim Electronics Workbench*.
2. To verify superposition with hardwired components.

Materials

Two dc power sources
One multimeter
Assorted resistors

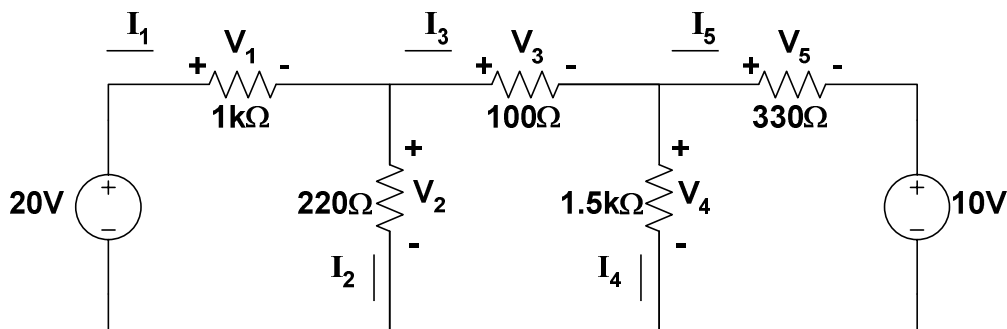


Figure 1: Resistive circuit with two sources

Procedure

Simulation

1. Construct the circuit in Figure 1 on *Multisim Electronics Workbench*. Put the meters in the appropriate places to read voltages across and currents through each resistor.
2. Run the simulation. Record all the voltages and currents in the circuit and enter them in Table 1. You can use ammeters for current measurements, voltmeter for voltage measurements or multimeter for both.
3. Note the current directions and voltage polarities shown in Figure 1.

Table 1: Simulation results for voltage and current with both sources

	10 K Ω	22 K Ω	1 K Ω	47 K Ω	33 K Ω
Voltage					
Current					

- Remove the 5-V source from the circuit. Replace it by a short circuit.
- Run the simulation. Measure the voltages and currents and record in Table 2.

Table 2: Simulation results for voltage and current with the 10-V source only

	10 K Ω	22 K Ω	1 K Ω	47 K Ω	33 K Ω
Voltage					
Current					

- Put the 5-V source back to the circuit. Remove the 10-V source and replace it by a short circuit.
- Run the simulation. Record all the voltages and currents in the circuit. Enter them in Table 3.

Table 3: Simulation results for voltage and current with the 5-V source only

	10 K Ω	22 K Ω	1 K Ω	47 K Ω	33 K Ω
Voltage					
Current					

Question: Check for superposition principle. Enter your observations here.

Hardwired Experiment

8. Repeat the procedure with hardwired circuit elements. Enter your results in below.

Table 4: Experimental results with both the sources

	10 K Ω	22 K Ω	1 K Ω	47 K Ω	33 K Ω
Voltage					
Current					

Table 5: Experimental results with 10-V source only

	10 K Ω	22 K Ω	1 K Ω	47 K Ω	33 K Ω
Voltage					
Current					

Table 6: Experimental results with 5-V source only

	10 K Ω	22 K Ω	1 K Ω	47 K Ω	33 K Ω
Voltage					
Current					

Questions:

1. Compare the results obtained with Workbench with those from hardware circuit, and comment.

2. Superposition theorem applies for only certain types of circuit. State what is the type?

3. Superposition applies to only some variables or quantities like current and voltage. It does not apply to, for example, power. State why not.

Any other observations or comments

Experiment 7

Thevenin's Theorem and Maximum Power Transfer

Introduction

A two terminal network resistive network can be replaced by a voltage source in series with an equivalent resistor. The value of the source voltage equals the open circuit voltage of the two terminals under consideration. The value of the equivalent resistor equals the resistance measured between the open terminals when all the sources of the circuit are deactivated (voltage source shorted and current source opened). This is termed as the Thevenin's theorem. The voltage source is called Thevenin's voltage (E_{TH}) and the equivalent resistor, the Thevenin's resistance (R_{TH}).

The maximum power output to a variable output resistance occurs when the value of the output resistance equals the Thevenin's resistance.

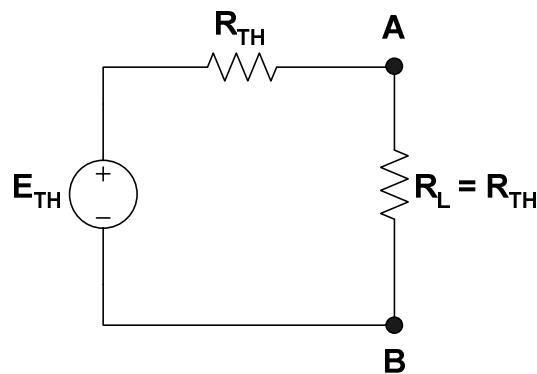


Figure 1: Maximum power output condition

The value of the maximum output power or transferred power is given as,

$$P_o = \frac{E_{TH}^2}{4R_{TH}}$$

Objectives

1. To construct Thevenin's equivalent using Workbench.
2. To verify the equivalent obtained in step 1 using hardwired components.
3. To determine maximum power transfer condition experimentally.

Materials

- Two dc power sources
- One multimeter
- Assorted resistors
- One decade resistor

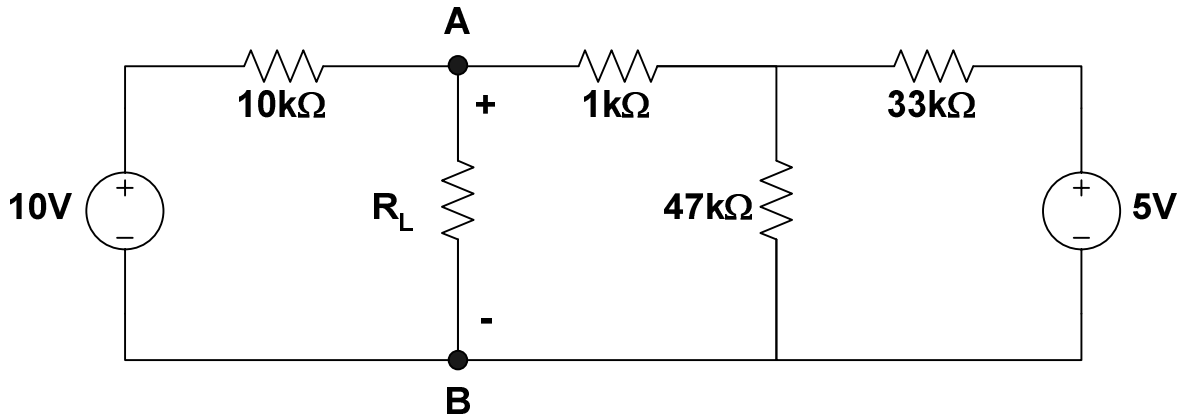


Figure 2: Circuit for Thevenin's equivalent

Procedure

Simulation

1. Construct the circuit given in Figure 2 on *Multisim Electronics Workbench*.
2. Remove the load resistor R_L and connect a multimeter (or voltmeter) to read the open circuit voltage between A and B. Simulate and record the voltage. This is E_{TH} for this circuit between A and B.
3. Remove the 10-V source. Replace it by a short circuit.
4. Remove the 5-V source. Replace it by a short circuit.
5. Connect a multimeter in the resistance measurement mode (ohmmeter) between A and B. Run the simulation and record the value of the resistor. This is R_{TH} in Figure 1.

Hardwired Experiment

6. Build circuit of Figure 2 with hardwired components in the laboratory.
7. Repeat step 2-5 and find the values of E_{TH} and R_{TH} experimentally. Considering the Workbench results as the base compute the percentage errors.

Question: Compare the values obtained with Workbench with that carried out using hardwired components. Comment on the accuracy of the methods.

8. In the circuit of Figure 2 connect a variable resistor (R_L) between A and B.
9. Vary R_L between 2.5 K Ω to 10.5 K Ω in steps of 1 K Ω . Measure voltage between A and B (V_L) in each case. Enter your results in Table 2.

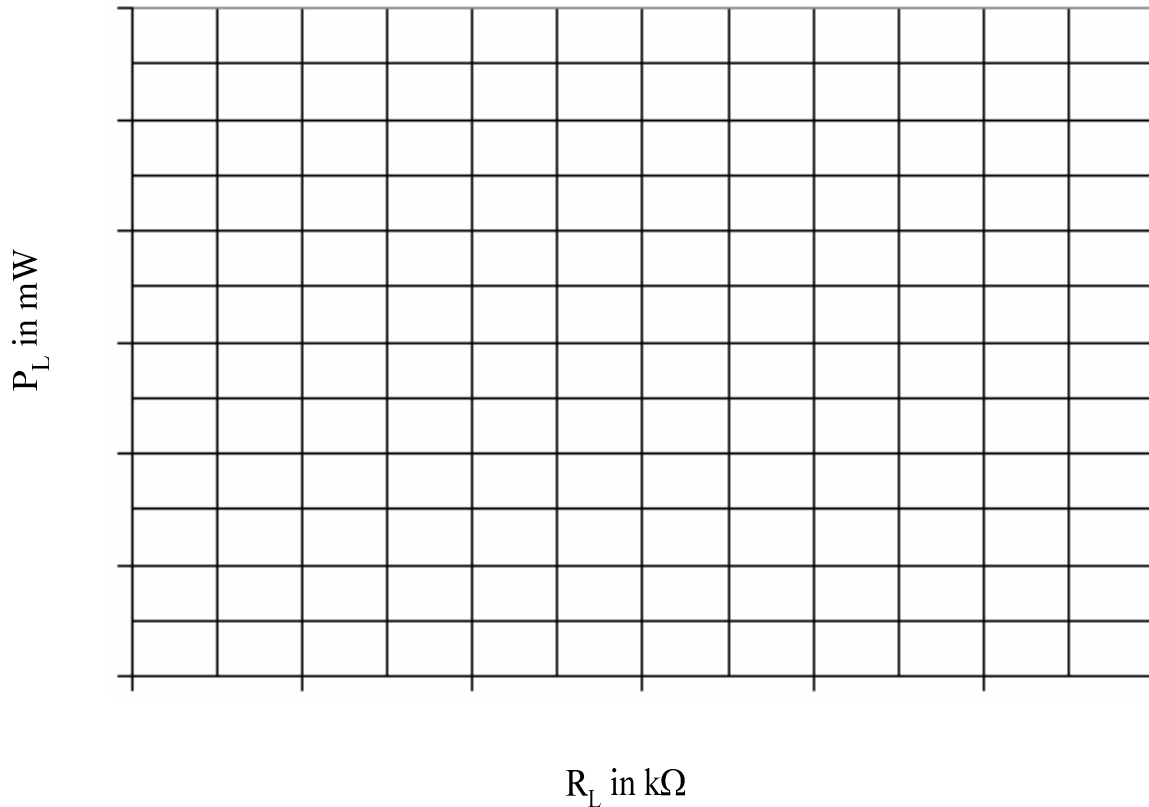
Table 1: Simulation and experimental results for Thevenin voltage and resistance

	E_{TH}	R_{TH}
Workbench		
Hardwired		
% Error		

Table 2: Experimental results for maximum power transfer

$R_L(KW)$	2.5	3.5	4.5	5.5	6.5	7.5	8.5	9.5	10.5
V_L									
P_L									

10. Plot R_L vs. P_L



Questions

1. At what value of R_L the maximum value of P_L occurs in the graph?
2. Does this value of R_L compare with R_{TH} you obtained experimentally or through Workbench?
3. If not, how much is the error in either case (take workbench result as the base)?
4. Draw the Norton's equivalent of the circuit in Figure 1.
5. Suppose you did not know the Thevenin's equivalent circuit, what procedure would you follow in the laboratory to get the Norton's equivalent?

Any other observations or comments

Experiment 8

Transients of a First Order RC Circuit

Introduction

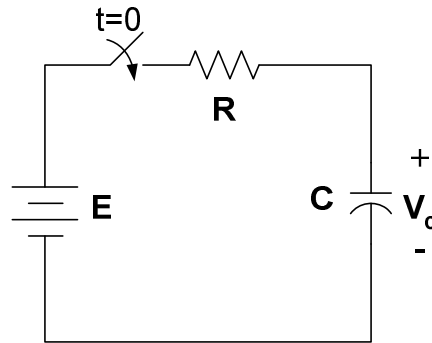


Figure 1: RC series circuit

The capacitor in the circuit of Figure 1 is assumed to have no initial voltage. When the switch is closed, the capacitor begins to charge. The voltage $V_c(t)$ across the capacitor for $t=0$ is given as,

$$V_c(t) = E(1 - e^{-t/RC}) \quad (1)$$

The current through the capacitor is expressed as,

$$I_c = \frac{E}{R} e^{-\frac{t}{RC}} \quad (2)$$

Objectives

1. To perform a transient analysis of the circuit using *Multisim Electronic Workbench*
2. To verify the simulation results with laboratory hardwired components.

Materials

One dc power source
One multimeter
Capacitor, 3.3mF, 80V dc
Resistor, 6.8K Ω
One switch & one stopwatch (for hardwired experiment).

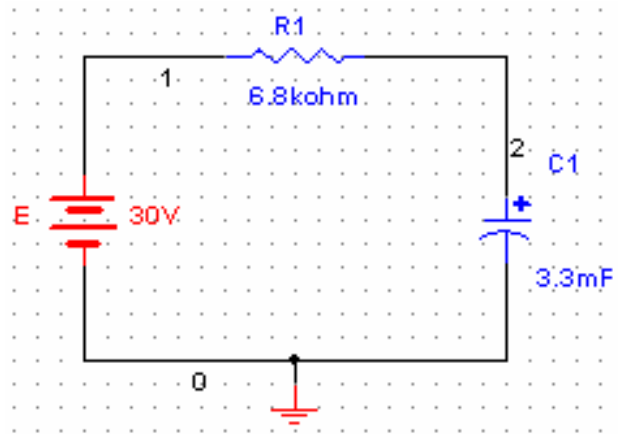


Figure 2: RC circuit for simulation on Multisim Electronics Workbench

Procedure

Simulation

1. Build the circuit on *Multisim Electronics Workbench*. Note the node numbers.
2. Run the Transient Analysis. The procedure is given as:
 - a. Select **Simulate** on **Main Menu**
 - b. Select **Analysis**
 - c. Select **Transient Analysis**
 - d. Select **Analysis Parameters**
 - Initial Conditions → Set to zero
 - Start time 0 s
 - End time 120 s
 - e. Select **Output Variables**
 - select node **2** (left box, this is the node for capacitor voltage, you might have different node number for capacitor voltage)
 - select **plot during simulation** (right box)
 - f. Select **Simulate**
3. Observe the wave shape. If you have access to a printer, print your results. Using the cursor tool record the results and enter the results into Table 1 at the given time instants.

Table 1 Transient response of RC circuit

Time (sec)	0	5	10	15	20	30	40	50	60	70	80	90	100	110	120
V_c (Workbench)															
V_c (Hardwired)															
Average															

- Plot V_c vs. time (in the event you could not get a print).

Hardwired Experiment

- Build the circuit given in Figure 3 with laboratory hardwired components. Note that this circuit is the same as in Figure 2 except the switch.

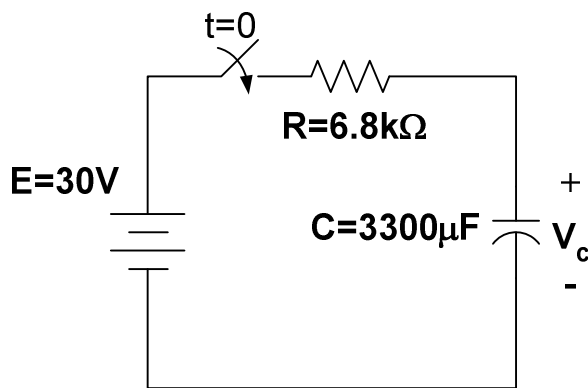


Figure 3: RC circuit for hardware experiment

- Turn switch on and measure the voltage across the capacitor using a multimeter. Keep counts of time using a stopwatch. Follow the time schedule as given in Table1.
- Repeat the measurements three times and find the average for each time steps. Enter the average value in Table 1. Before repeating the measurements, place a short circuit across the capacitor to make sure that capacitor has no voltage initially (Use a small resistance in series to discharge the capacitor)
- Plot the voltage response on the same graph paper as simulation plot.

Questions

1. Compare the results of simulation with hardwired experiment. Discuss the results.

2. Calculate the time constant from both the curves (simulation and hardwired). Compare them with the theoretical value. Comment on your results.

3. Draw the circuit diagram of a first order network that has more than one capacitor.

Any other observations or comments

Experiment 9

The Oscilloscope and Function Generator

Introduction

The oscilloscope is one of the most important electronic instruments available for making circuit measurements. It displays a curve plot of time-varying voltage on the oscilloscope screen. The oscilloscope provided with *Multisim Electronics Workbench* is a dual trace oscilloscope that looks and acts like a real oscilloscope. A dual trace oscilloscope allows the user to display and compare two time-varying voltages at one time.

The controls on the oscilloscope are as follows:

1. The TIME BASE control adjusts the time scale on the horizontal axis in time per division when Y/T is selected. When B/A is selected, the horizontal axis no longer represents time. The horizontal axis now represents the voltage on the channel A input and vertical axis represents the voltage on channel B input. When A/B is selected, the horizontal axis represents the voltage on the channel B input and the vertical axis represents the voltage on the channel A input. The X_POS control determines the horizontal position where the curve plot begins.
2. The CHANNEL A control adjusts the volts per division on the vertical axis for the channel A curve plot. The Y-POS control determines the vertical position of the channel A curve plot relative to the horizontal axis. Selecting AC places a capacitance between the channel A vertical input and the circuit testing point. Selecting "0" connects channel A vertical input to ground.
3. The CHANNEL B control adjusts the volts per division of the vertical axis for the channel B curve plot. The Y-POS determines the vertical position of the channel B curve plot relative to the horizontal axis. Selecting AC places a capacitance between the channel B vertical input and the circuit test point. Selecting "0" connects the channel B vertical input to ground.
4. The trigger settings control the conditions under which a curve plot is triggered (begins to display). Triggering can be internal (based on one of the input signals) or external (based on a signal applied to the oscilloscope external trigger input). With internal triggering AUTO, A, or B. If A is selected, the curve plot will be triggered by channel A input signal. If B is selected, the curve plot will be triggered by channel B input signal. If you expect a flat input waveshape or you want the curve plot displayed as soon as possible, select AUTO. The display can be set to start on positive or negative slope of the input by selecting the appropriate EDGE selection. The trigger LEVEL control determines the voltage level of the input signal waveform, in divisions on the vertical axis, before the waveform will begin to display.

Once a circuit simulation has been activated and a voltage curve plot has been displayed on the oscilloscope screen, the oscilloscope probes can be moved to other test points in the circuit without running the simulation again. Moving the probes automatically redraws the voltage curve plot for the new test point. You can also fine tune the settings either during or after a simulation and the display will be redrawn on the screen automatically. You can

'pause', 'resume' through the "Analysis Option". The zoom feature can be used by using "Expand". Normal size will be restored by clicking "Reduce". You can learn other features through the help menu.

The Function Generator

The function generator is a voltage source that supplies different time-varying voltage functions. The *Multisim Electronics Workbench* can supply sine wave, square wave, and triangular wave voltage functions. The waveshape, frequency, amplitude, duty cycle, and dc offset can be easily changed. It has three voltage output terminals. Connect the COM terminal to ground symbol. The +ve terminal provides output voltage that is positive with respect to the COM terminal and the -ve terminal provides an output voltage that is negative with respect to the COM terminal.

The controls on the function generator are as follows:

1. You can select a waveshape by clicking the appropriate waveshape on the top of the function generator.
2. The frequency control allows you adjust the frequency of the output voltage up to 999 MHz. Click up or down arrow to adjust the frequency, or click the frequency box and type the desired frequency.
3. The AMPLITUDE control allows you to adjust the amplitude of the output voltage measured from the reference level (common) to **peak** level. The peak to peak value is twice the amplitude setting.
4. The OFFSET control adjusts the dc level of the voltage curve generated by the function generator. An offset of 0 positions the curve plot along the x-axis with an equal positive and negative voltage setting. A positive offset raises the curve plot above the x-axis and a negative offset lowers the curve plot below the x-axis.

Consult the *Multisim Electronics Workbench* User and Reference manuals for more details on the oscilloscope and function generator.

Procedure

1. From the instrument panel pull down the Function generator and Oscilloscope and connect them as shown in Figure 1. You can select different colors of the wires by double clicking the line and choosing the colors. Select sine wave from the function generator. Set frequency to 1 kHz and amplitude to 10V. On the oscilloscope start with the Time Base of 0.2ms/div and channel A, B settings of 5V/div.
2. Click the on-off switch to run the analysis. Connect the positive output of the function generator to oscilloscope channel A input and negative output to channel B input. You may use red color for Ch. A and blue to Ch. B, if you wish.

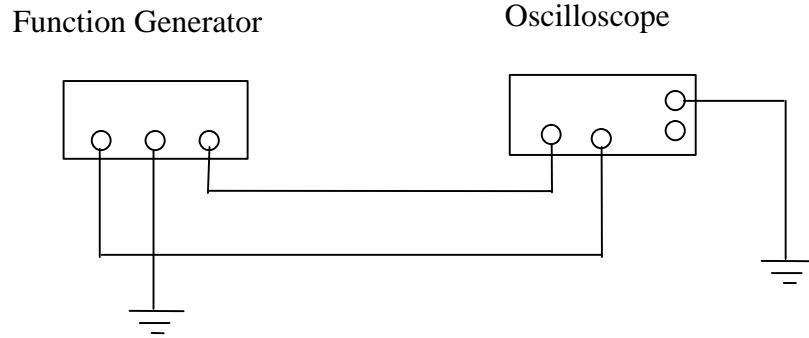


Figure 1: The function generator and oscilloscope connection

3. Select the “0” on the oscilloscope channel B input and run the analysis again.

Question: What change occurred on the oscilloscope Channel B curve plot? Explain.

4. Change the oscilloscope channel A to 10V/div.

Question: What change occurred on the oscilloscope Channel A curve plot? Explain.

5. Change the oscilloscope Time Base to 0.1ms/div and run the analysis again.

Question: What change occurred on the oscilloscope Channel A curve plot? Explain.

6. Change the oscilloscope Channel A Y-POS to 1.00 and run the analysis again.

Question: What change occurred on the oscilloscope Channel A curve plot? Explain.

7. Change the channel A Y-POS back to 0.00 and select DC on channel B input and run the analysis again.

Question: What change occurred on the oscilloscope curve plot? Explain.

8. Return the oscilloscope time base to the settings you started with and select “0” on the channel B input again. Select the triangular wave shape on the function generator and run the analysis again.

Question: What change occurred on the oscilloscope curve plot? Explain.

9. Select the square wave on the function generator and run the analysis again.

Question: What change occurred on the oscilloscope curve plot? Explain.

10. Change the AMPLITUDE on the function generator to 5V and run the analysis again.

Question: What change occurred on the oscilloscope curve plot? Explain.

11. Change the frequency on the function generator to 2KHZ and run the analysis again.

Question: What change occurred on the oscilloscope curve plot? Explain.

12. Change the offset on the function generator to 3 and run the analysis again.

Question: What change occurred on the oscilloscope curve plot? Explain.

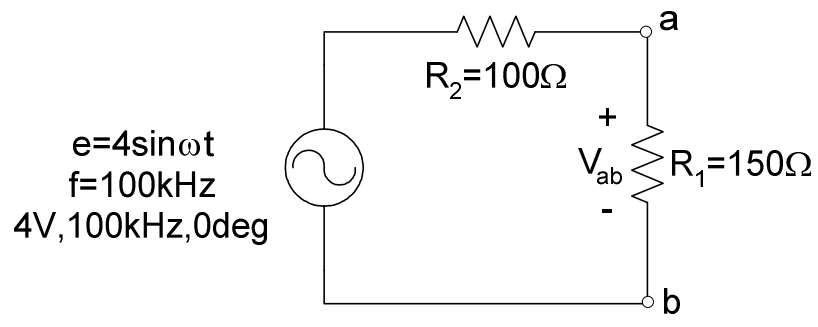
13. Change the oscilloscope channel A input to AC and run the analysis again.

Question: What change occurred on the oscilloscope curve plot? Explain.

14. Click 'expand' on the oscilloscope. Measure the time period T of one cycle on the waveshape

Question: What was the time period (T) of one cycle on the wave shape?

Repeat the steps above with the laboratory Function Generator and Oscilloscope and record your answers in the same order as the workbench. State any difficulties you encountered in carrying the steps.



Experiment 10

Sinusoidal AC Analysis

Introduction

The students learnt how to use the function generator and oscilloscope in the previous experiment. In this experiment they will learn to build and take measurements in ac circuits. The circuit will be simulated on *Multisim Electronics Workbench*. Then they will build it with hardware components and check the accuracy of the measurements comparing with the simulation results. Consider the circuit given in Figure 1.

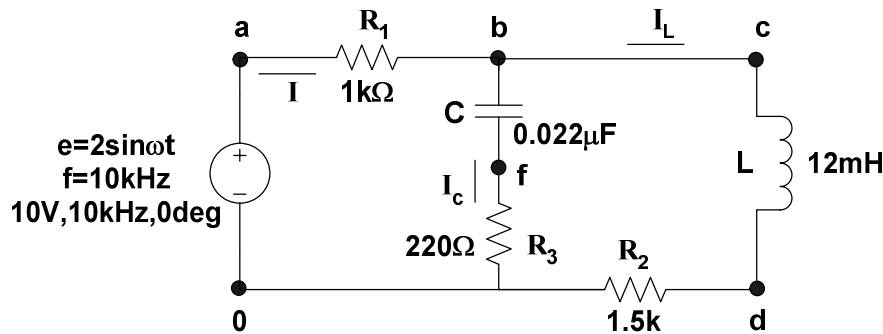


Figure 1: RLC circuit

The equations for sinusoidal current is written in the form,

$$i = I_m \sin (\omega t - \phi) \quad (1)$$

where, i is the instantaneous value of current, I_m is the peak value, ω is the angular frequency and ϕ is the phase angle of the current with respect to the source voltage. Phase angle can be measured by using the two traces of a dual trace oscilloscope and measuring the time difference between two waveforms, as shown in Figure 2.

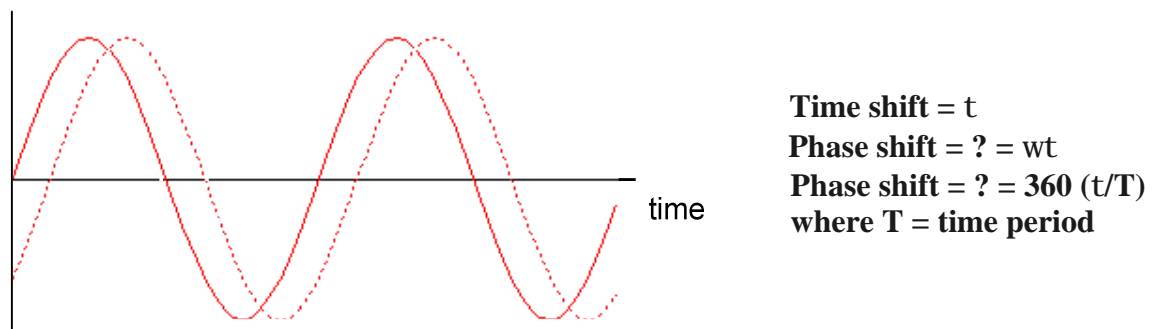


Figure 2: Phase angle measurement

The RMS or effective value of the sinusoid is given as $I_m/\sqrt{2}$. The power dissipated can be computed by using,

$$P = E_{rms} I_{rms} \cos \phi \quad (2)$$

$$P = I_{rms}^2 R = V_R^2 / R \quad (3)$$

Objectives

1. Measure currents in each branch of the circuit of Figure 1 using oscilloscope
2. Verify KCL for ac circuit
3. Calculate the power dissipated in the resistor.

Materials

Dual trace oscilloscope
 Signal generator
 Resistors (470Ω , 5Ω , 5Ω)
 Inductor, 1mH (iron core)
 Capacitor, 3300pF

Procedure

1. Construct the circuit of Figure 1. Connect Ch A of the oscilloscope to the output of signal generator at point 'a'. Oscilloscope ground and circuit ground should be the same. Set the oscilloscope to AUTO sweep, and use Ch A as the TRIGGER source. Adjust the output of the signal generator to provide a sinusoidal voltage with an amplitude (peak value) of 5V (10 Vp-p) and frequency of the function generator at $f=10$ KHz.
2. Use Ch B of the oscilloscope to observe the voltage at point d. This is the voltage across resistor R_2 . Measure the peak (half of peak to peak) voltage V_2 (across R_2). Determine phase angle ϕ_2 of voltage V_2 with respect to generator voltage 'e'. Determine the amplitude I_L (V_2/R_2) and enter time domain expression for i_L below.

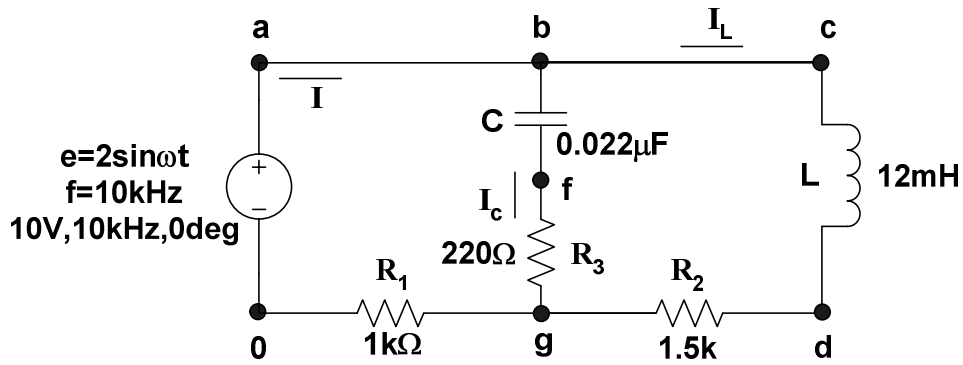
Table 1: Sinusoidal expressions for voltage (v_2) and current (i_L)

	V_2 (amplitude)	ϕ_2 (phase, deg)	Expression v_2	Expression i_L
Workbench				
Hardwired				

4. Move Ch B of the oscilloscope to observe the voltage at point f. This is the voltage across R_3 . Measure the amplitude V_3 and phase angle ϕ_3 with respect to 'e'. Determine the amplitude I_C (V_3/R_3) and enter time domain expression for i_C below.

Table 2: Sinusoidal expressions for voltage (v_3) and current (i_C)

	V_3 (amplitude)	ϕ_3 (phase, deg)	Expression v_3	Expression i_C
Workbench				
Hardwired				



5. Change the position of resistor R_1 to that shown in Figure 2. This change will not affect the circuit. Connect Ch A to point 'a' and Ch B to point 'g'. Measure amplitude of the voltage across R_1 (V_1) from Ch B. Determine phase angle ϕ_1 of voltage V_1 with respect to generator voltage 'e'. Determine the amplitude I (V_1/R_1) and enter time domain expression for i below.

Table 3: Sinusoidal expressions for voltage (v_1) and current (i)

	V_1 (amplitude)	ϕ_1 (phase, deg)	Amplitude I	Expression i
Workbench				
Hardwired				

Questions

1. Convert the measured values of currents i_R , i_L and i_C obtained in steps 3-5 into phasor form (example, $I=5\angle 30^0$ Amp). Compare these values with the ones obtained through simulation.

Hardwired Experimental Values

Simulation Values

2. Use complex algebra to show that the data in step 1 (above) satisfies Kirchhoff's current law.

3. Use the phasors E and I to calculate total power delivered to the circuit by the voltage source (equation 2).

$$P_T =$$

4. Now use the rms values of I_R and V_R to determine the power dissipated by the resistor (equation 3). How does this value compare with the total power dissipated by the source? Offer an explanation for any variation.

$P_R =$

Any other observations or comments

Appendix

(LABORATORY REGULATIONS AND SAFETY RULES)

The following Regulations and Safety Rules must be observed in all concerned laboratory location.

1. It is the duty of all concerned who use any electrical laboratory to take all reasonable steps to safeguard the HEALTH and SAFETY of themselves and all other users and visitors.
2. Be sure that all equipment is properly working before using them for laboratory exercises. Any defective equipment must be reported immediately to the Lab. Instructors or Lab. Technical Staff.
3. Students are allowed to use only the equipment provided in the experiment manual or equipment used for senior project laboratory.
4. Power supply terminals connected to any circuit are only energized with the presence of the Instructor or Lab. Staff.
5. Students should keep a safety distance from the circuit breakers, electric circuits or any moving parts during the experiment.
6. Avoid any part of your body to be connected to the energized circuit and ground.
7. Switch off the equipment and disconnect the power supplies from the circuit before leaving the laboratory.
8. Observe cleanliness and proper laboratory house keeping of the equipment and other related accessories.
9. Wear the proper clothes and safety gloves or goggles required in working areas that involves fabrications of printed circuit boards, chemicals process control system, antenna communication equipment and laser facility laboratories.
10. Double check your circuit connections specifically in handling electrical power machines, AC motors and generators before switching "ON" the power supply.
11. Make sure that the last connection to be made in your circuit is the power supply and first thing to be disconnected is also the power supply.
12. Equipment should not be removed, transferred to any location without permission from the laboratory staff.
13. Software installation in any computer laboratory is not allowed without the permission from the Laboratory Staff.
14. Computer games are strictly prohibited in the computer laboratory.
15. Students are not allowed to use any equipment without proper orientation and actual hands on equipment operation.
16. Smoking and drinking in the laboratory are not permitted.

All these rules and regulations are necessary precaution in Electrical Laboratory to safeguard the students, laboratory staff, the equipment and other laboratory users.