Preface

The **EE 213-Electric Circuits** Lab is an essential component of the Electrical Engineering program. The lab is designed to help students understand the basic principles of electric circuits and to give them an insight into the design, simulation and hardware implementation of electric circuits. The main objective is to provide hands-on experience for the students so they are able to put theoretical concepts into practice.

The experiments include both software circuit simulations and hardware-based measurements with actual electrical components. Computer simulation is stressed as a key analysis tool in engineering design. ***Multisim Electronics Workbench*** software is used for simulation of electric circuits. Hardware experimentation with components and laboratory equipment is always important for acquiring the necessary hands-on practical skills in engineering education.

The present manual includes eleven experiments that cover topics taught in the two Electric Circuits courses EE211 and EE213. The manual also includes appendices on components color codes and general laboratory regulations and safety rules.

List of Experiments

Experiment 1

Introduction to Electrical Circuits Simulation and Testing

Simulation is a mathematical way of emulating the behavior of a real world system. With simulation modeling, we can determine a circuit's performance without physically constructing the circuit or using actual test instruments. **Multisim Workbench** 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 user interface for all design needs.

Introduction

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

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

Most of the analysis can be performed by turning on/off the simulate switch. If the simulate switch shown in Fig. 1 does not appear on your screen then select **View→Show Simulate Switch** in the Main Menu. This will open the Simulation Switch. **<u>Always, remember to save your file to keep the changes you made in the circuit by</u>** selecting **File→Save.**

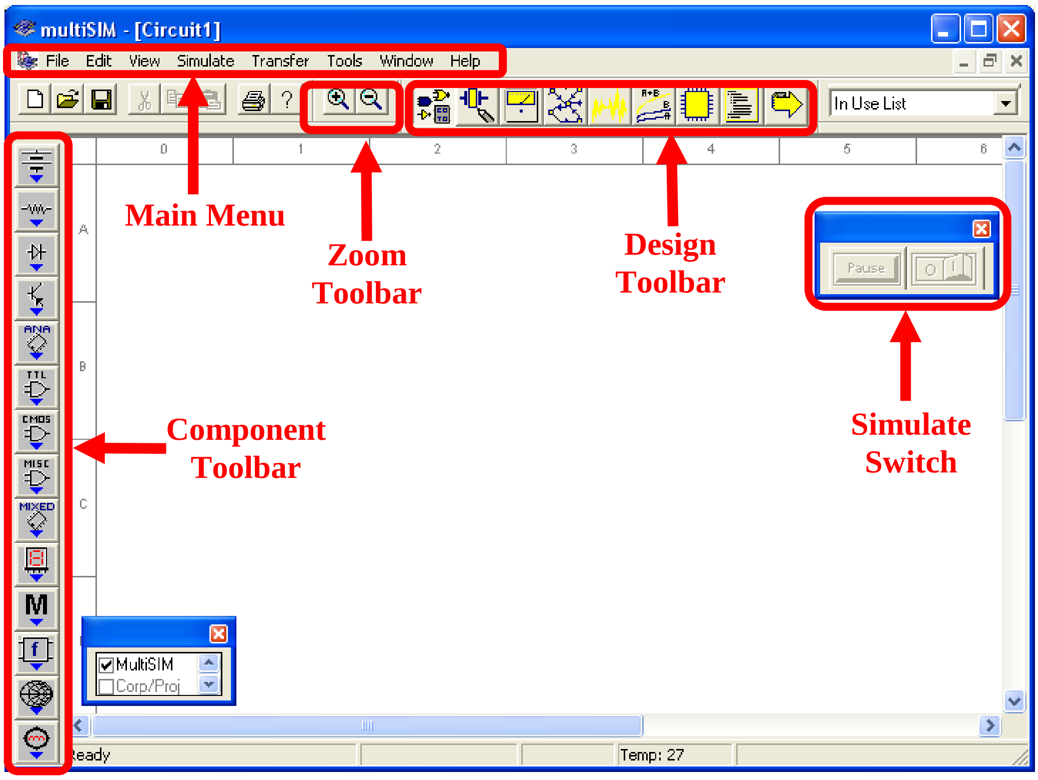


Figure 1: Main Window of **Multisim** Simulation Software

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

Example

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

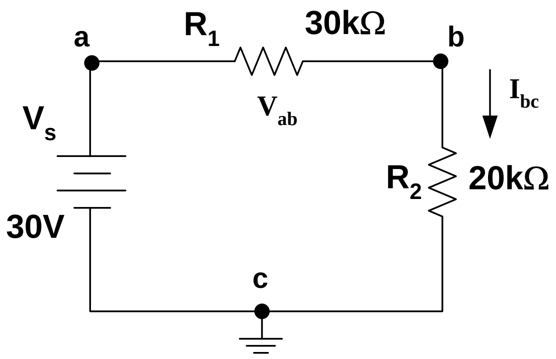


Figure 2: Circuit for **Multisim** Simulation

STEP A: Placing the Components

1. Place a Battery (DC Source)

a. Bring a DC source into the Multisim workspace:

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

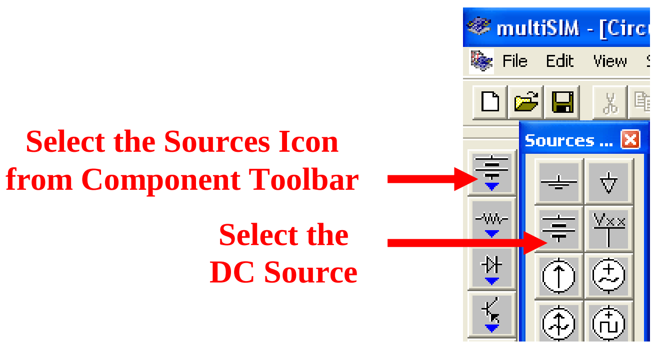
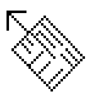
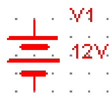


Figure 3: Selection of DC Voltage Source in **Multisim**

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



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

b. Change the value and name of the voltage source:

Double-click on the voltage source that you just placed in the workspace, and a new window with the name **Battery** will appear, as shown in Fig. 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**.

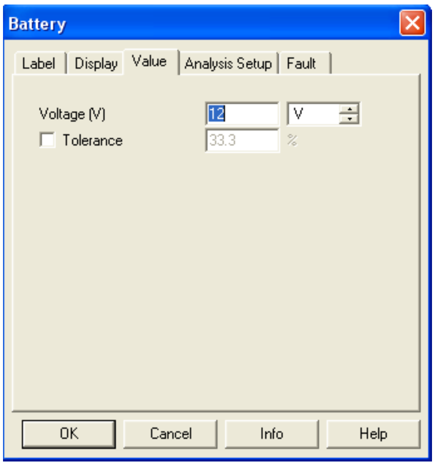


Figure 4: Battery Window for setup of DC voltage source

2. Place a Resistor:

a. Bring a resistor into the Multisim workspace:

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

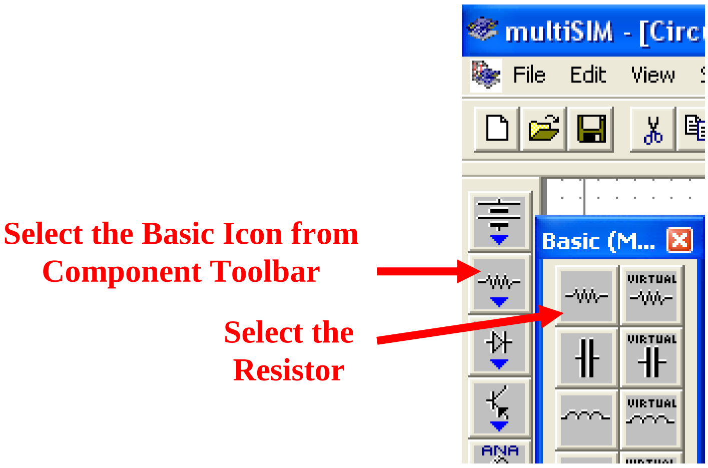
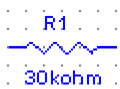


Figure 5: Selection of Resistor in **Multisim**

Click on "Resistor." This will open the **Browser-Basic** window, as shown in Fig. 6. Scroll through the **Component List,** select 30 kOhm, and click OK. The cursor shape will change again. Click in the workspace and this will show the resistor as:



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

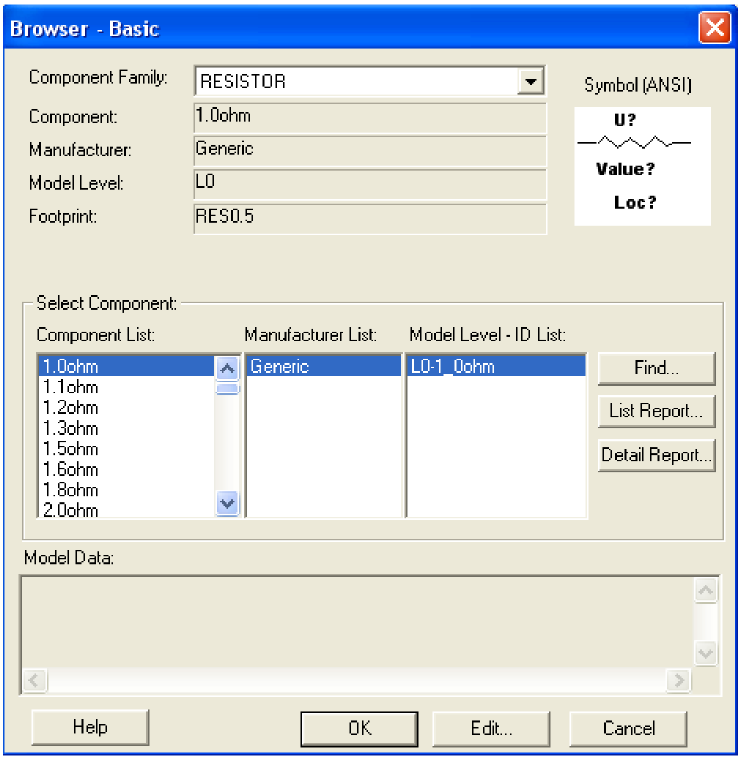


Figure 6: Setup window for Resistor values

b. Change the name of the resistor:

Double-click on the resistor, and a new window with the name **Resistor** will open as shown in Fig. 7. Select **Label** from the menu of this window. Change the **Reference ID** to R_1 (if it is not already) and press **OK.** This will change the name of the resistor to R_1.

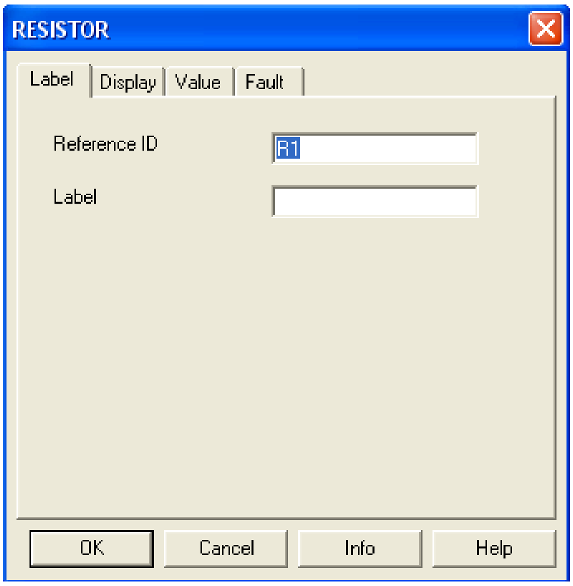


Figure 7: Battery Window for Label of Resistor

d. ***Add another resistor R_2***

Place resistor R_2 of value 20 kΩ 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 '20 kOhm' individually to put them in a suitable place.

3. Place Ground:

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

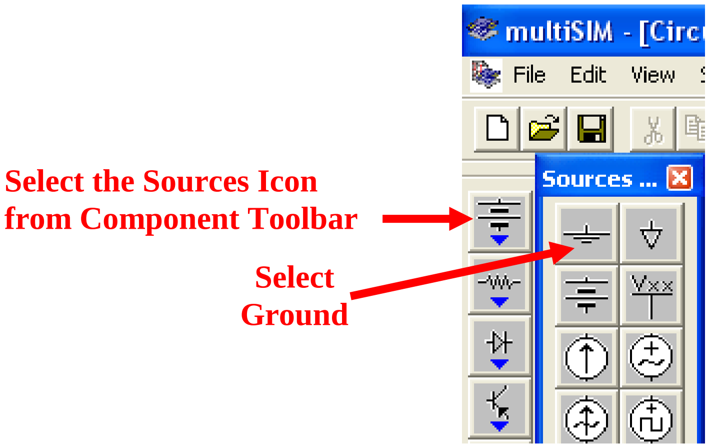


Figure 8: Selection of Ground in **Multisim**

STEP B: Connecting the Components

1. Arrange the components properly:

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

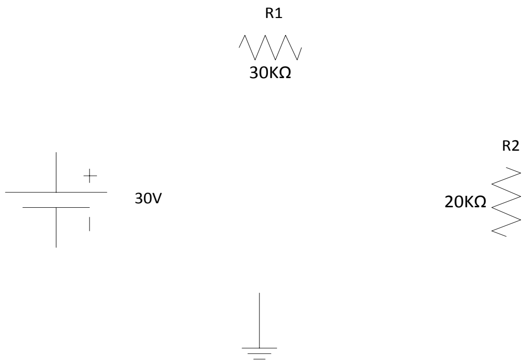


Figure 9: Arranging the Components in proper order

2. **Show Grid in the workspace:**

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

3. **Connect DC Voltage Source "Vs" to "R_1":**

   Bring the cursor close to the upper pin of "Vs"; the 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 Fig. 10. In this way, when the cursor reaches the pin of R_1 click again and this will connect "Vs" to "R_1". Notice that a node number is automatically given.

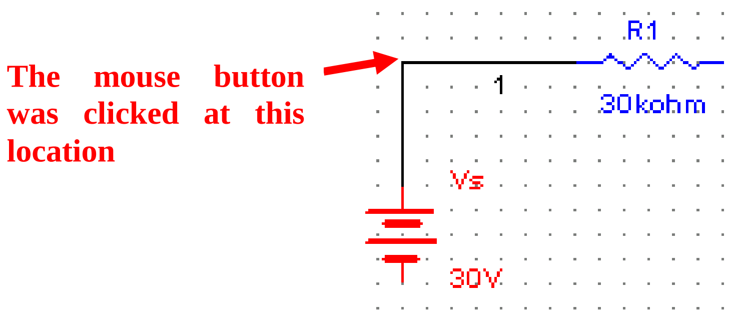


Figure 10: Manual connection of components

4. **Connect "R_1" to "R_2":**

   Connect R_1 to R_2 using the same procedure.

5. **Making use of the Junction to connect the Ground:**

   In a similar manner, connect the ground with V_s and R_2. Notice that a small black circle appears just above the ground, this is called the junction. When two 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 that the ground node is automatically given node number 0. Do not alter it.

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

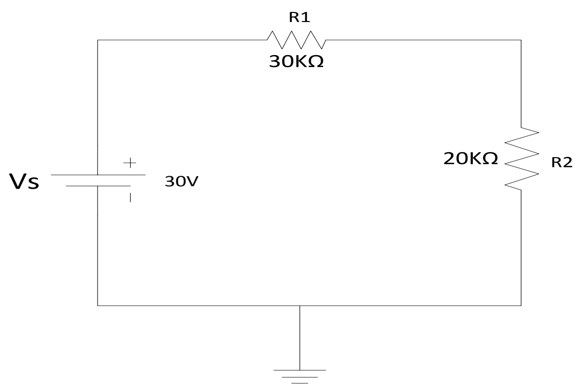


Figure 11: Complete Circuit in **Multisim**

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 Fig. 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.

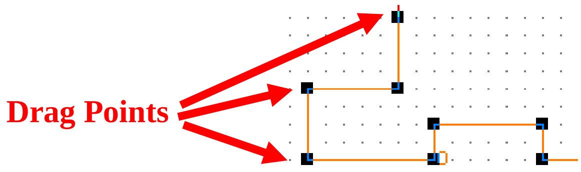


Figure 12: Drag points for connecting wire

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 Fig. 13.

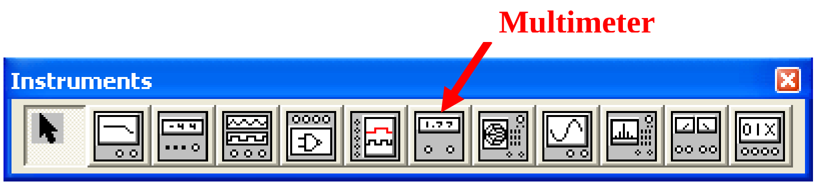


Figure 13: Instruments Toolbar

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

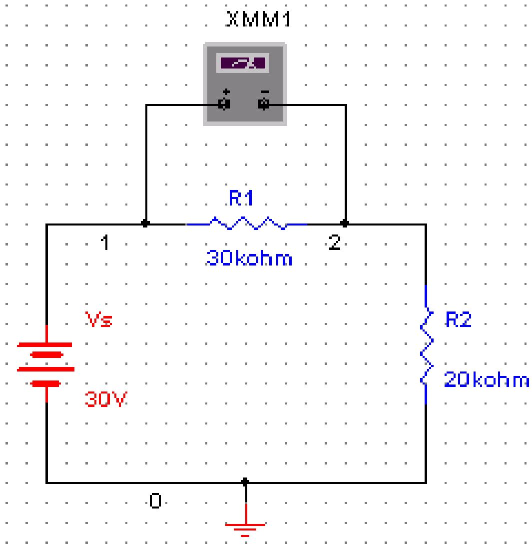


Figure 14: **Multimeter** connection for voltage measurement

c. Set the Multimeter to measure DC voltage:

Double-click on **Multimeter** to open the properties window shown in Fig. 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.

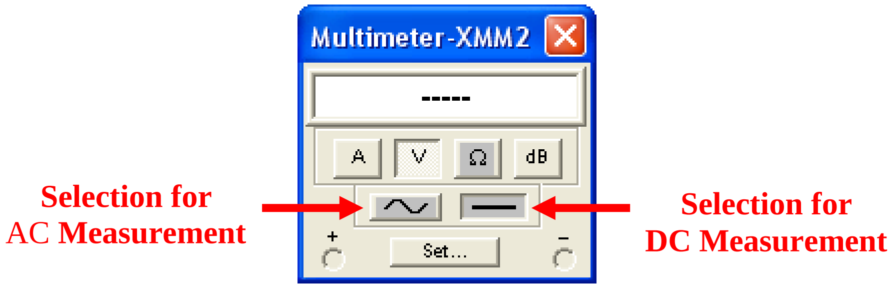


Figure 15: **Multimeter** properties window

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 Fig. 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 the current flows from 3 to zero, the meter will read positive.

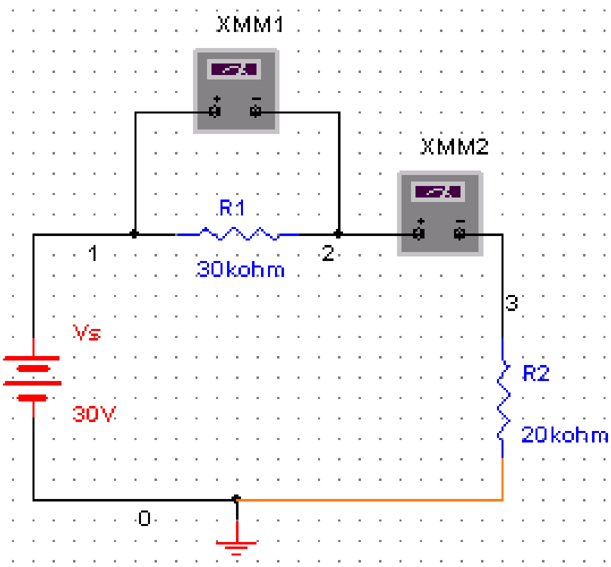


Figure 16: **Multimeter** connection for current measurement

STEP E: Circuit Simulation

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 **Multimeter**. Click to the '1' position (ON) of the simulation switch to start simulation. Results will appear in the properties window of the **Multimeter**. Compare your result with those in Fig. 17 and show them to your instructor.

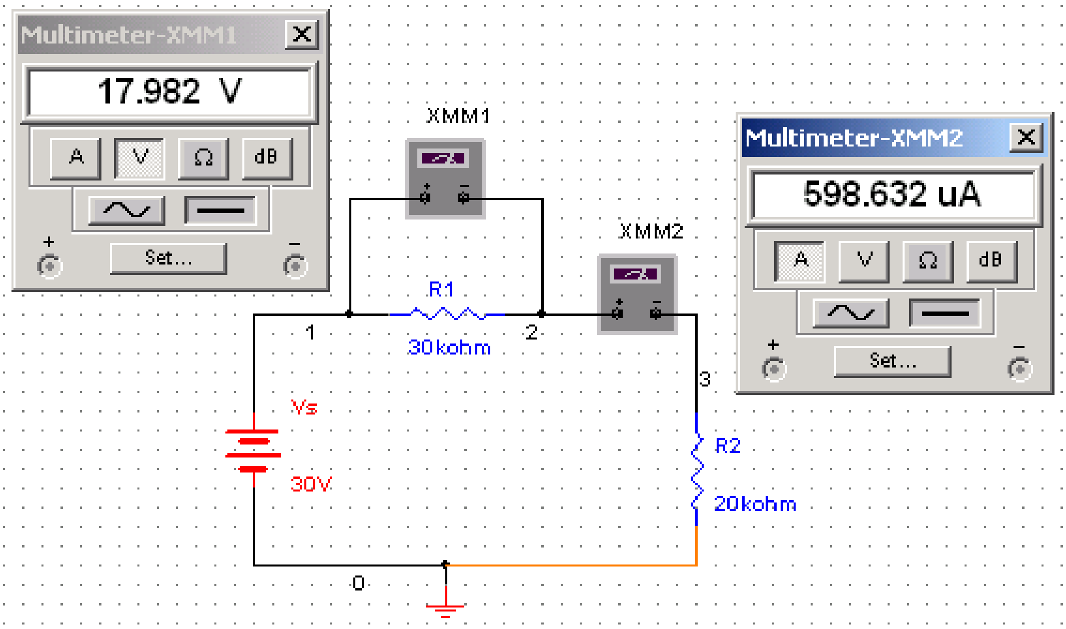


Figure 17: Simulation result

Exercise 1

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

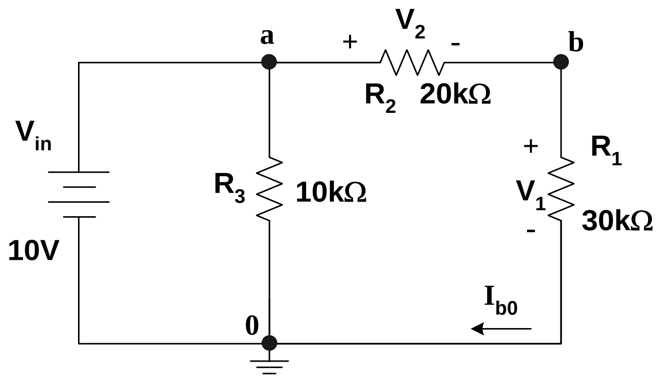


Figure 18

Exercise 2

1. Build the circuit of Fig. 19 in ***Multisim*** *Electronics Workbench*.
2. Simulate the circuit and find i_x and i_y (magnitude and phase)

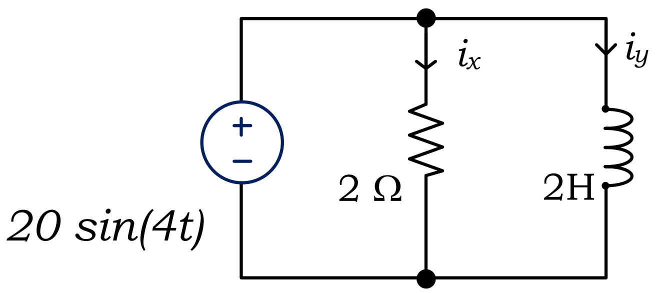


Figure 19

Experiment 2

Electric Circuits Fundamental Laws

Introduction

This experiment covers electrical circuit's fundamental laws, including Ohm's law, and Kirchkoff's voltage and current laws. It consists of three parts which have to be first carried out using the ***Multisim*** *Electronics Workbench* software, and then repeated with hardware components and laboratory equipment. The experiments involve the measurement of resistance, voltage and current in DC circuits.

PART I: Resistance, Voltage & Current in DC Circuits

Objectives

1. Use of voltmeter & ammeter to measure voltage and current through a resistor
2. Verification of component power rating

Materials

DC power supply
DC 0-20V Voltmeter
DC 0-100mA Ammeter
Multimeter
5V/1W lamp
Assorted carbon resistors (100 Ω, 150 Ω, 220 Ω, 330Ω)

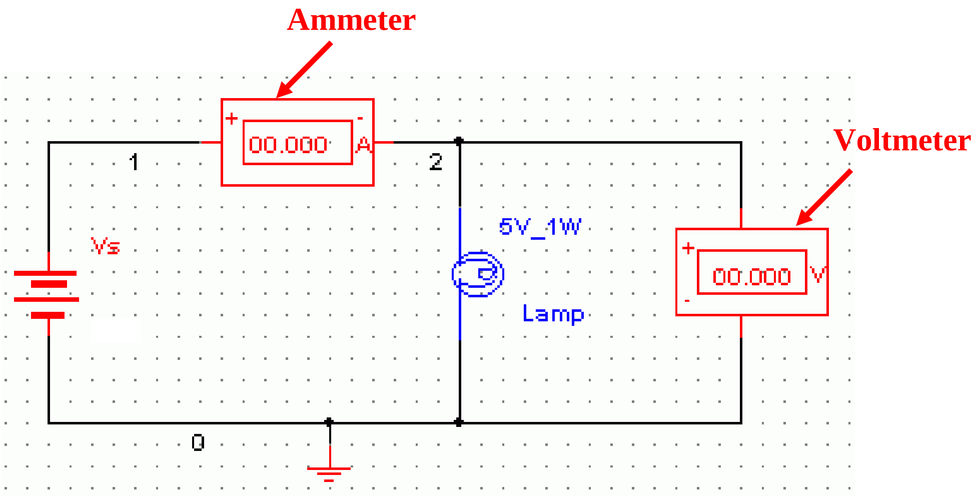


Figure 1: Measuring current and voltage: a lamp connected across a battery

Procedure

Workbench Simulation

1. Construct the circuit shown in Fig. 1 on the ***Multisim*** *Electronics Workbench*. Use a 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, 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 and current in Table 1 and calculate power dissipation.
5. Change the DC supply voltage to 8V. Run the analysis and observe the value of the 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 would the lamp be damaged when the voltage goes to 8V? Explain by comparing power dissipation with its rated value.

PART II: Resistance Measurement and Ohm's Law

Objectives

1. Usage of the Multimeter to measure resistance.
2. Verification of Ohm's law

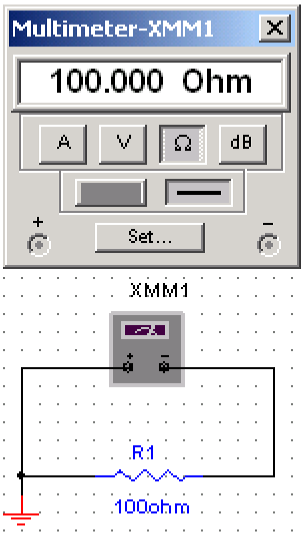


Figure 2: Measuring resistance

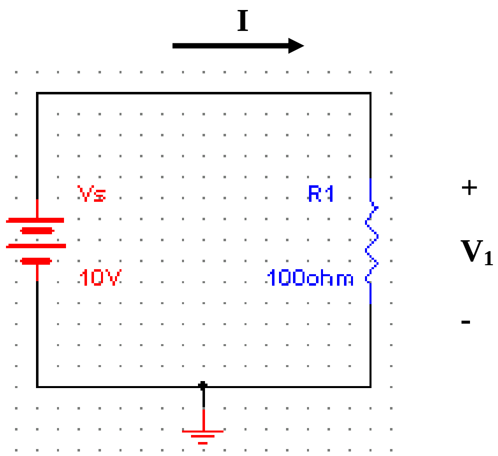


Figure 3: Verifying Ohm's Law

Procedure

Workbench Simulation

1. Build the circuit of Fig. 2 using ***Multisim*** *Electronics Workbench*. Connect a **Multimeter** between the terminals of the resistor and set it to read resistance. Select R_1=100Ω. Click the **Simulation Switch** to run the analysis. Record the value of resistance R_1 in Table 3's Ohmmeter Reading.
2. Build the circuit given in Fig. 3. Set V_s = 10V and R_1=100Ω. Click **Simulation Switch** to run the 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 the Multimeter reading as a reference, calculate the % error.
3. Vary the DC supply voltage V_s in steps of 2V and record the current in each case. Enter your results in Table 4.
4. Plot '**V**' vs. '**I**' in the graph of Table 5.
5. Calculate the resistor value based on the slope of the V-I curve plotted in step 4.

Hardware Experiment

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

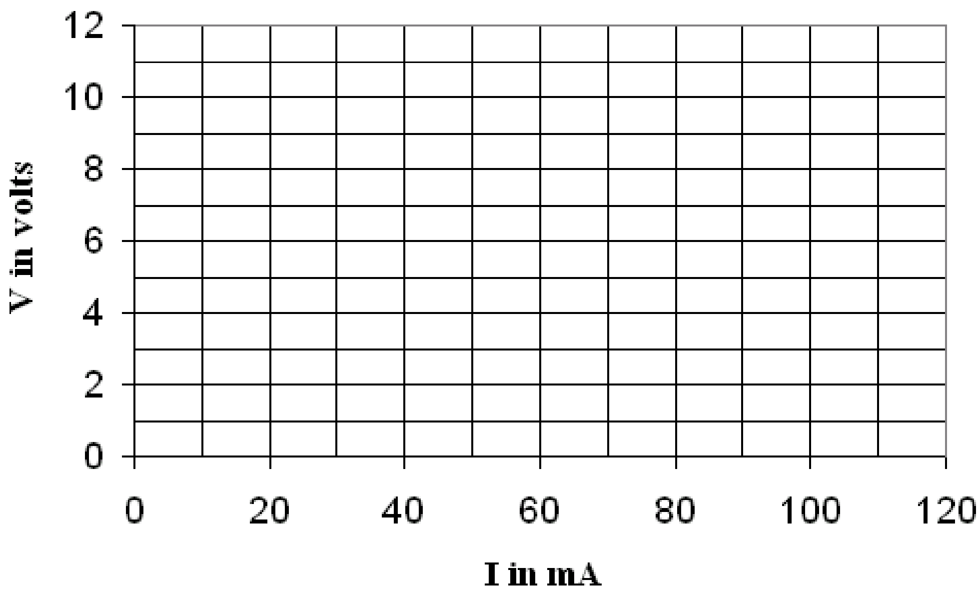
Table 3: Resistance Measurement

	Ohmmeter Reading	Ohm's Law		
		V_1	I	$R=V_1/I$
Workbench Simulation	R=			
Hardware Experiment	R=			

Table 4: V-I measurements

Workbench		**Hardware**	
V (Volts)	I (mA)	V (volts)	I (mA)
0		0	
2		2	
4		4	
6		6	
8		8	

V-I Plot



Resistance measurement from V-I slope

- Electronics Workbench experiment R=
- Hardware experiment R=

Questions

1. Compare the values of resistance obtained with the Ohmmeter and with Ohm's law (V/I) in the hardware experiment. Find the percentage error between them with the Ohmmeter reading as the reference.

2. Comment on your results and observations.

PART III: Kirkchoff's Voltage and Current Laws

Kirkchoff's Voltage Law (KVL) states that the algebraic sum of all voltages around any closed path equals zero. Kirchhoff's Current Law (KCL) states that the algebraic sum of all the currents at a node is zero (the current entering a node has the opposite sign to the current leaving the node).

Objectives

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

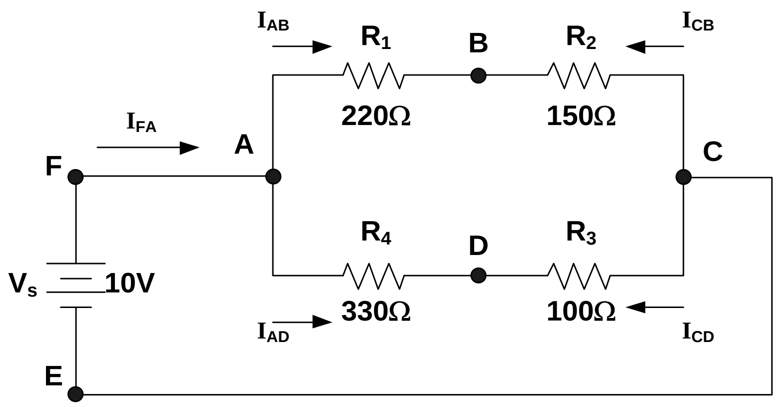


Figure 4: Resistive circuit diagram

Procedure

1. Construct the circuit shown in Fig. 4 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 5.
3. Measure the currents I_{AB}, I_{CB}, I_{AD}, I_{CD} and I_{FA}. Enter the values in Table 6. Note the polarity (sign) of the currents.
4. Calculate the voltages around the following loops ABCEFA, ABDA, BDCB, ABCDA, and record them in Table 8.
5. Verify KCL by adding currents at nodes A, B, C and D. Enter your results in Table 7.
6. Construct the circuit in Fig. 4 with hardware components. Repeat steps 2-5. Enter your results in Tables5-8. Compute the percentage errors. Comment on any discrepancy in the voltage/current experimental and theoretical values

Table 5: Voltage measurements

	V_{AB}	V_{BC}	V_{AD}	V_{DC}	V_{BD}	V_{AC}
Simulation						
Hardware						
% Error						

Table 6: Current measurements

	I_{AB}	I_{CB}	I_{AD}	I_{CD}	I_{FA}
Simulation					
Hardware					
% Error					

Table 7: Sum of currents at nodes

	A	**B**	**C**	**D**
Simulation				
Hardware				

Table 8: Voltages around loops

	ABCEFA	ABDA	BDCB	ABCDA
Simulation				
Hardware				

Experiment 3

Voltage & Current Dividers and Superposition Principle

Introduction

This lab covers two parts:

1. the voltage divider and current divider rules
2. the superposition principle

The lab work is based on the use of Multisim Workbench simulation and hardware-based circuits to verify the voltage current divider and voltage divider rules.

PART I: Voltage & Current Dividers

For a **series circuit** the voltage divider rule (VDR), refer to Fig. 1:

$$V_1 = \frac{R_1}{R_1 + R_2 + R_3} V_s$$
$$V_2 = \frac{R_2}{R_1 + R_2 + R_3} V_s \qquad (1)$$
$$V_3 = \frac{R_3}{R_1 + R_2 + R_3} V_s$$

For a **parallel circuit** the current divider rule (CDR), refer to Fig. 2:

$$I_1 = \frac{R_2}{R_1 + R_2} I_s \qquad (2)$$
$$I_2 = \frac{R_1}{R_1 + R_2} I_s$$

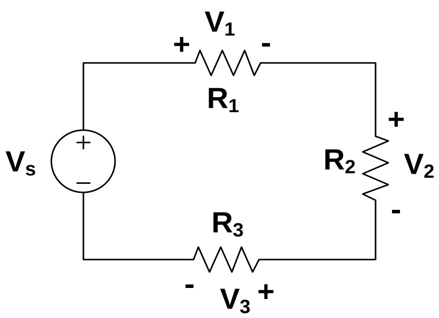


Figure 1: Series circuit

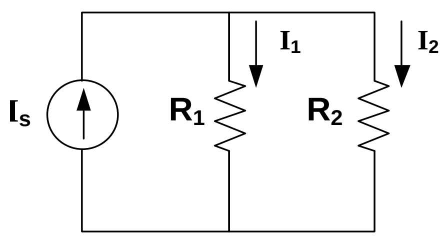


Figure 2: Parallel circuit

Objectives

1. To verify the voltage current divider and voltage divider rules.
2. To verify the superposition principle using ***Multisim*** *Electronics Workbench*.

Materials

DC power supply
Multimeter
Assorted carbon resistors (100 Ω, 150 Ω, 220 Ω, 330Ω)

Procedure

Simulation Experiment

1. Build the circuit given in Fig. 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, 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 of Fig. 4 and record the values in Table 2.

Hardware Experiment

5. Build the circuit of Fig. 3 with the hardware components. Take the voltage current measurements and R_{eq} and record them in Table 1. Considering the Workbench results as the base, compute the percentage errors.
6. Build the circuit of Fig. 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.

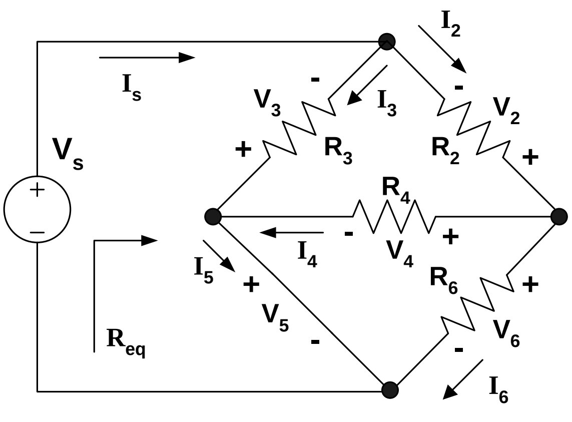


Figure 3: Series-parallel circuit I

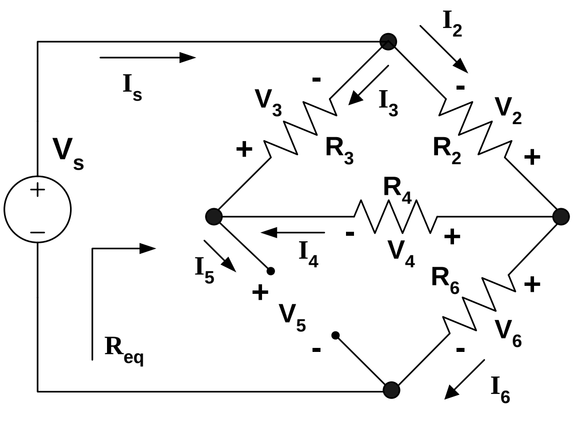


Figure 4: Series-parallel circuit II

$V_S = 10V, R_2 = 100\Omega, R_3 = 150\Omega, R_4 = 220\Omega, R_6 = 330\Omega$

Table 1: Simulation and experimental results for Fig. 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}
Simulation												
Hardware												

Table 2: Simulation and experimental results for Fig. 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}
Simulation												
Hardware												

Questions

Refer to Fig. 3 and the results of Table 1:

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

2. Are R_3 and R_4 in parallel or in series? Why?

3. Are Vs and R_3 in parallel or in series? Why?

4. Is VDR applicable for R_3 and R_4? Why?

5. Is CDR applicable for R_4 and R_6? Why?

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

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

7. Are R_4 and R_6 in parallel or in series? Why?

8. Are Vs and R_3 in parallel or in series? Why?

9. Is VDR applicable for R_3 and R_4? Why?

10. Is CDR applicable for R_4 and R_6? Why?

PART II: Superposition Principle

If there is more than one source in an electric network, the response (voltage or current) can be obtained 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 are shorted and current sources are opened).

Objectives

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

Materials

Two DC power sources
Multimeter
Assorted carbon resistors (100Ω, 220Ω, 330Ω, 1kΩ, 1.5kΩ)

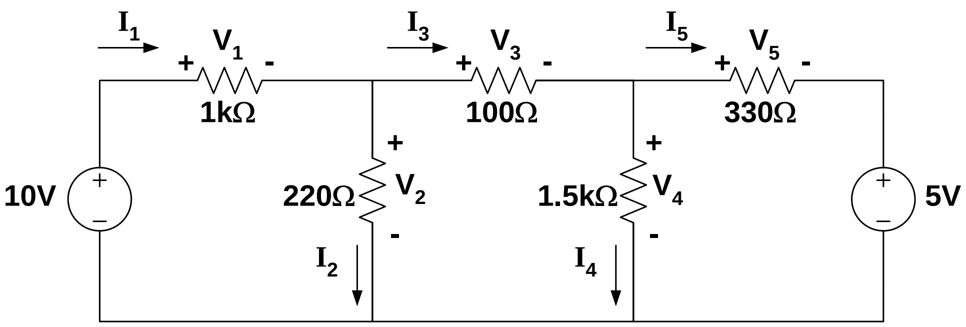


Figure 5: Resistive circuit with two sources

Procedure

Simulation

1. Construct the circuit in Fig. 5 on *Multisim Electronics Workbench.* Put the meters in the appropriate places to read the voltages and currents.
2. Run the simulation. Record all the voltages and currents in Table 3. 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 Fig. 5.
4. Remove the 5-V source from the circuit. Replace it with a short circuit.
5. Run the simulation. Measure the voltages and currents and fill in Table 4.
6. Put the 5-V source back in the circuit. Remove the 10-V source and replace it with a short circuit.
7. Run the simulation. Record all the voltages and currents in the circuit. Enter them in Table 5

Table 3: Simulation results for voltage and current with both sources

	1 kΩ	220 Ω	100 Ω	1.5 kΩ	330 Ω
Voltage					
Current					

Table 4: Simulation results with the 10-V source only

	1 kΩ	220 Ω	100 Ω	1.5 kΩ	330 Ω
Voltage					
Current					

Table 5: Simulation results with the 5-V source only

	1 kΩ	220 Ω	100 Ω	1.5 kΩ	330 Ω
Voltage					
Current					

Hardware Experiment

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

Table 6: Experimental results with both sources

	1 kΩ	220 Ω	100 Ω	1.5 kΩ	330 Ω
Voltage					
Current					

Table 7: Experimental results with 10-V source only

	1 kΩ	220 Ω	100 Ω	1.5 kΩ	330 Ω
Voltage					
Current					

Table 8: Experimental results with 5-V source only

	1 kΩ	220 Ω	100 Ω	1.5 kΩ	330 Ω
Voltage					
Current					

Questions

1. Check for the superposition principle. Write your observations.

2. Compare the simulation and experimental results, and comment on any error causes.

3. Superposition only applies to current and voltage, but not to power. Explain why.

Experiment 4

Thevenin's Theorem and Maximum Power Transfer

Introduction

A two-terminal resistive network can be replaced by a voltage source in series with an equivalent resistor. The value of the voltage source 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 independent sources of the circuit are deactivated (voltage source shorted and current source opened). This is known as Thevenin's theorem. The voltage source is called Thevenin's voltage (V_{TH}) and the equivalent resistor, the Thevenin's resistance (R_{TH}).

Instead of representing two-terminal resistive network with a voltage source in series with a resistor, we can represent it with a current source in parallel with a resistor. This is known as Norton Equivalent Circuit. The value of the current source (I_{SC}) equals the current passing through a short circuit placed between the two terminals under consideration. In general, the two equivalent circuits are related by source transformation. Then:

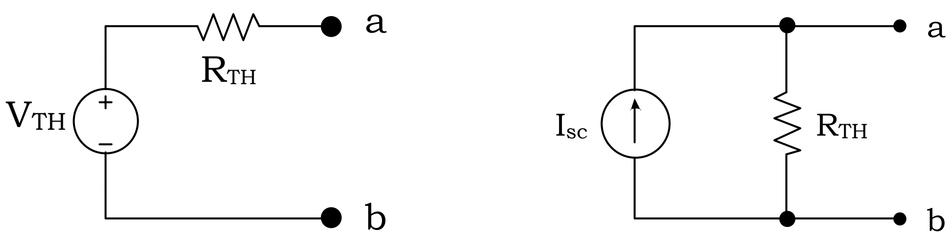


Figure 1: Thevenin's and Norton Equivalent Circuits

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

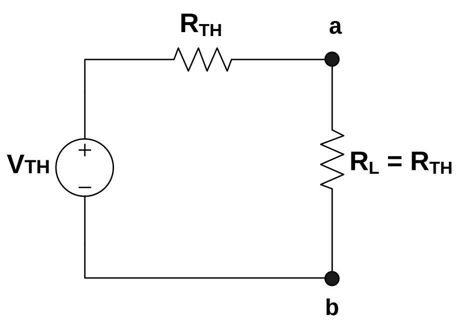


Figure 2: Maximum power output condition

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

$$P_{max} = \frac{V_{TH}^2}{4R_{TH}}$$

Objectives

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

Materials

Two DC power sources
One **Multimeter**
Assorted carbon resistors (1 kΩ, 10 kΩ, 33 kΩ, 47 kΩ)
One decade resistor

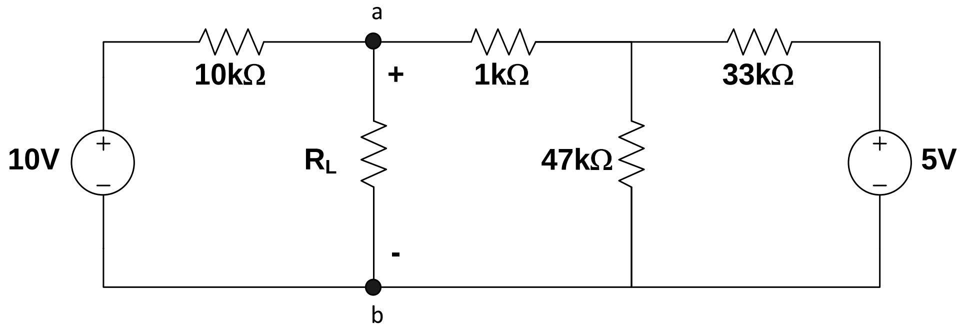


Figure 3: Circuit for Thevenin's and Norton's equivalent

Procedure

Simulation

1. Construct the circuit given in Fig. 3 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 V_{TH} for this circuit between *a* and *b*. Record the value in Table 1.
3. Place an **ammeter** (represents a short circuit) between terminals *a* and *b* to read the short circuit current between *a* and *b*. Simulate and record the current. This is I_{SC} for this circuit between *a* and *b*. Record the value in Table 1.
4. Remove the 10-V source. Replace it with a short circuit.
5. Remove the 5-V source. Replace it with a short circuit.
6. 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 Fig. 1. Record the value in Table 1.

Hardwired Experiment

7. Build the circuit of Fig. 3 with hardwired components in the laboratory.
8. Repeat steps 2 to 6 and find the values of V_{TH}, I_{SC} and R_{TH} experimentally. Considering the *Workbench* results as the base, compute the percentage errors.
9. In the circuit of Fig. 3 connect a variable resistor (R_L) between *a* and *b*.

10. Vary R_L between 2.5 kΩ and 10.5 kΩ in steps of 1 kΩ. Measure the voltage between *a* and *b* (V_L) in each case. Enter your results in Table 2.

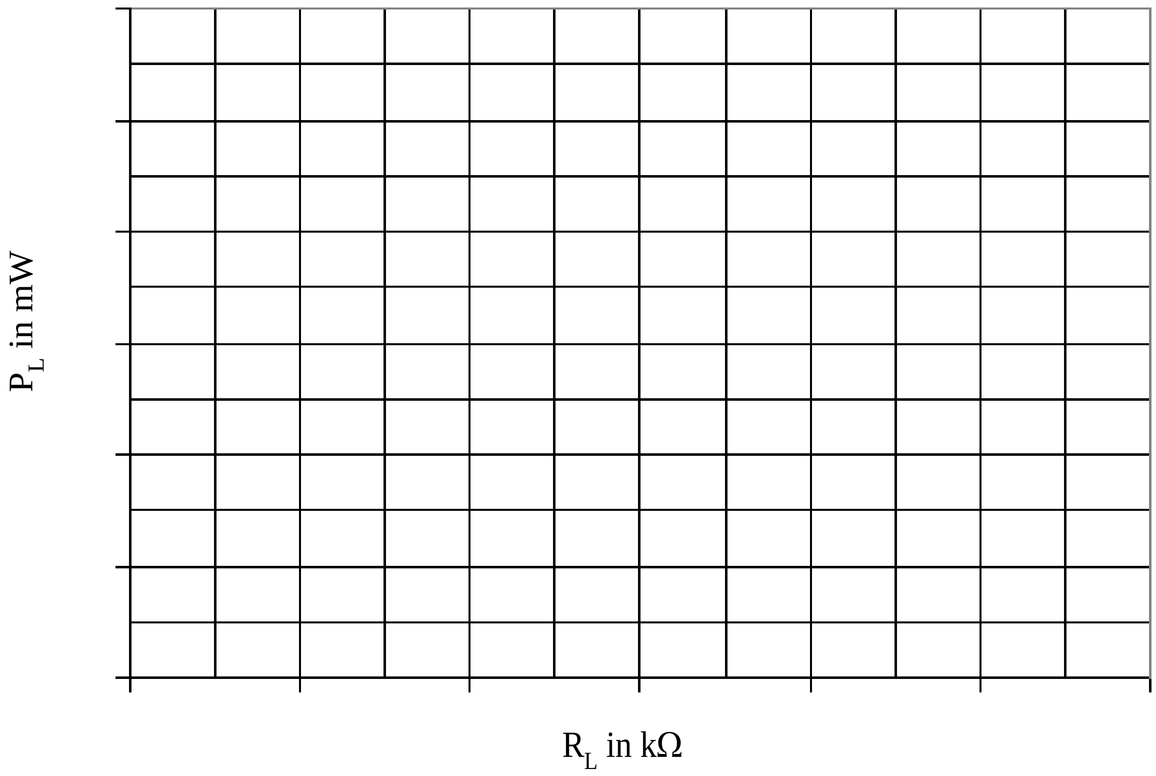
Table 1: Simulation and experimental results for Thevenin voltage and resistance

	V_{TH}	I_{SC}	R_{TH}
Workbench			
Hardwired			
% Error			

Table 2: Experimental results for maximum power transfer

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

11. Plot R_L vs. P_L



Questions

1. At what value of R_L does the maximum value of P_L occur in the graph?

2. Does this value of R_L compare with R_{TH} you obtained experimentally or through *Workbench*?

3. Using your results show that the two equivalent circuits are related by source transformation.

4. Can you give examples for cases were Thevenin's and Norton's equivalents are not related by source transformation?

Experiment 5

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 the 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 wave shape or you want the curve plot displayed as soon as possible, select AUTO. The display can be set to start on a 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' or '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. *Multisim Electronics Workbench* can supply *Sine Wave*, *Square Wave*, and *Triangular Wave* voltage functions. The wave shape, 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 proves 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 wave shape by clicking the appropriate wave shape on the top of the **Function Generator**.
2. The frequency control allows you to 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 Fig. 1. You can select different colored 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~~ for Ch. B, if you wish.

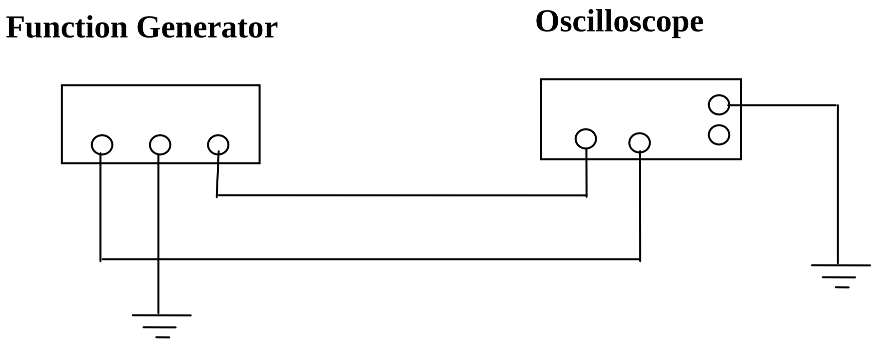


Fig. 1: The **Function Generator** and **Oscilloscope** connection

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 5 V/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 2 kHz 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 wave shape.

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 out the steps.

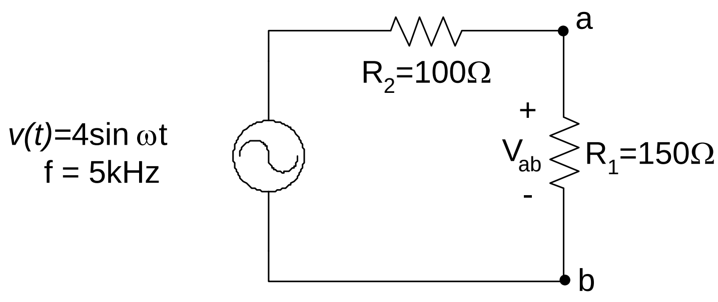


Figure 2: Measurement of RMS voltage

19. Connect the hardwired circuit shown in Fig 2. Measure the amplitude (peak value) of voltage v_1 using the **Oscilloscope** and record it.

$v_{ab\ (peak)}$ (using **Oscilloscope**) =

Connect an AC **voltmeter** between terminals ‘a’ and ‘b’ and record the value

v_{ab} (using AC **voltmeter**) =

This value is called the RMS value of the voltage wave.

Questions

1. Find the ratio of the two measured values.

$$\frac{v_{ab}\ \text{(using oscilloscope)}}{v_{ab}\ \text{(using voltmeter)}} =$$

2. Compare the above value with the theoretical (ideal) value and find the percentage error.

Experiment 6

Sinusoidal AC Analysis

Introduction

The students learned µhow 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 compared with the simulation results. Consider the circuit given in Fig. 1.

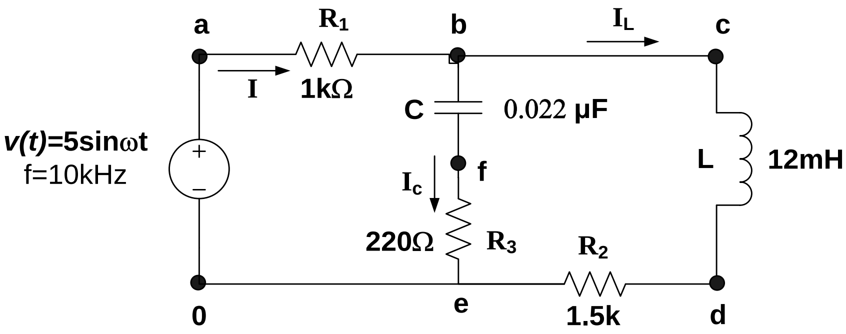
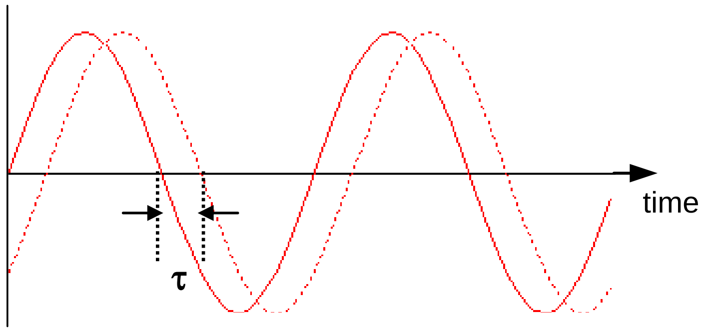


Fig. 1: RLC circuit

The equation for sinusoidal current is written in the form,

$$i(t) = i_m \sin(\omega t+\theta) \tag{1}$$

where, $i(t)$ is the instantaneous value of current, i_m is the peak value, ω is the angular frequency and θ 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 Fig. 2.



Time shift = τ
Phase shift = $\theta = \omega\tau$
Phase shift = $\theta = 360\ (\tau/T)$
Where T = time period

Fig. 2: Phase angle measurement

The RMS (Root Mean Square) or effective value of the sinusoid is given as $i_m/\sqrt{2}$. The real power dissipated can be computed by using,

$$P = V_{rms} I_{rms} \cos\theta \tag{2}$$

Objectives

1. Measure currents in each branch of the circuit of Fig. 1 using the **Oscilloscope**
2. Verify KCL for AC circuit
3. Calculate the power dissipated in the resistor.

Materials

Dual trace **Oscilloscope**
Signal generator
Resistors (1kΩ, 1.5kΩ, 220Ω)
Inductor, 12mH (iron core)
Capacitor, 0.022 μF

Procedure

1. Construct the circuit of Fig. 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 θ_2 of voltage v_2 with respect to generator voltage 'v'. Calculate the current $I_2(t)= {v_2(t)}/{R_2}$ and enter time domain expression for i_L below.

Table 1: Sinusoidal expressions for voltage (v_2) and current ($i(t)$)

	$v_2(t)$ (amplitude)	θ_2 (phase, deg)	I_L(amplitude) = ${v_2(t)}/{R_2}$	Expression i_L(t)
Workbench				
Hardwired				

3. 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 θ_3 with respect to '$v(t)$'. Calculate the current i_C ($v_3(t)/R_3$) and enter time domain expression for $i_C(t)$ below.

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

	$v_{3(t)}$(amplitude)	θ_3 (phase, deg)	$i_C(t)$(amplitude)=$v_3(t)/R_3$	Expression $i_C(t)$
Workbench				
Hardwired				

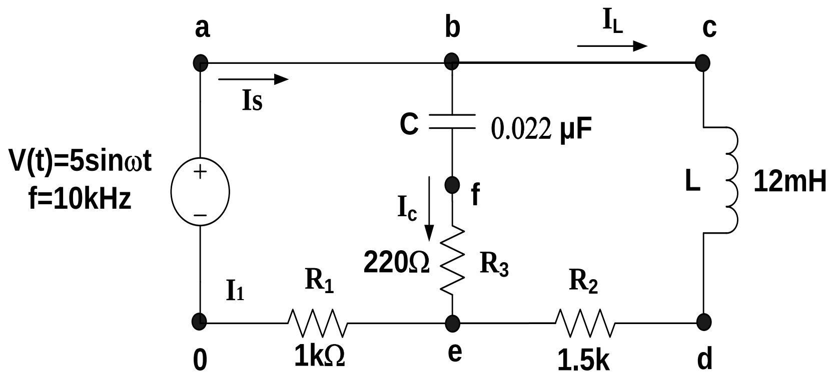


Figure 3:

5. Change the position of resistor R_1 to that shown in Fig. 3. This change will not affect the circuit. Connect Ch A to point 'a' and Ch B to point 'e'. Measure amplitude of the voltage across R_1 (V_1) from Ch B. Determine phase angle θ_1 of voltage $v_1(t)$with respect to generator voltage '$v_1(t)$'. Calculate the current $I_s = I_1 = {}^{V_1}/_{R_1}$ and enter time domain expression for i(t) below.

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

	$v_1(t)$(amplitude)	θ_1 (phase, deg)	$i_L(t)$(amplitude)=$v_1(t)/R_1$	Expression $i(t)$
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, $I = 5\angle 30^0 A$). Compare these values with the ones obtained through simulation.

<u>Hardwired Experimental Values</u> <u>Simulation Values</u>

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

Experiment 7
Three Phase Circuits

Introduction

Three phase circuits or system are made of three phase source, three phase feeder and three phase load. The three phase loads are connected in why (Y) or delta Δ shape. The line (L) and phase (P) quantities of a balanced three phase circuits are related to each other as follows.
In a Y-connection, the line and the phase quantities are related by:

$$V_P = V_L/\sqrt{3} \tag{1}$$

$$I_P = I_L \tag{2}$$

Whereas the relationships for a Δ-connection are

$$I_P = I_L/\sqrt{3} \tag{3}$$

$$V_P = V_L \tag{4}$$

The real and reactive powers for a 3 phase circuit (either Y or Δ connection) are given as

$$P=\sqrt{3}\, V_L\, I_L \cos(\theta) \tag{5}$$

$$Q=\sqrt{3}\, V_L\, I_L \sin(\theta) \tag{6}$$

Where θ is the power factor angle of the load

$$\theta = \tan^{-1}(Q/P) \tag{7}$$

$$PF=\cos(\theta) \tag{8}$$

Equation (7) and (8) can be used also to calculate the per factor for every phase

Objectives

1. Voltages and currents measurement in the three-phase circuits.
2. Power measurement and power factor determination in three-phase circuits.

Materials

1 Variable AC power supply 0-400V.

1 Resistive load bank.

1 Inductive load bank.

1 Capacitive load.

1 Set of 30 safety connectors, blue, black, red

1 Set of 10 safety connectors, green/yellow.

1 Digital multi meter.

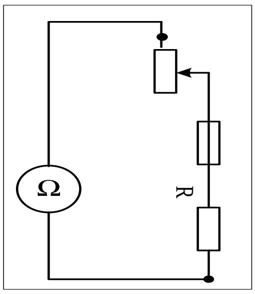


Figure 1 Phase resistance measurement

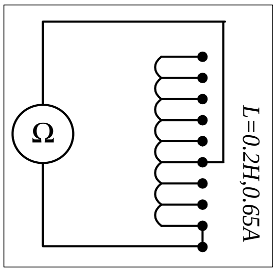


Figure 2 Phase internal series resistance measurement

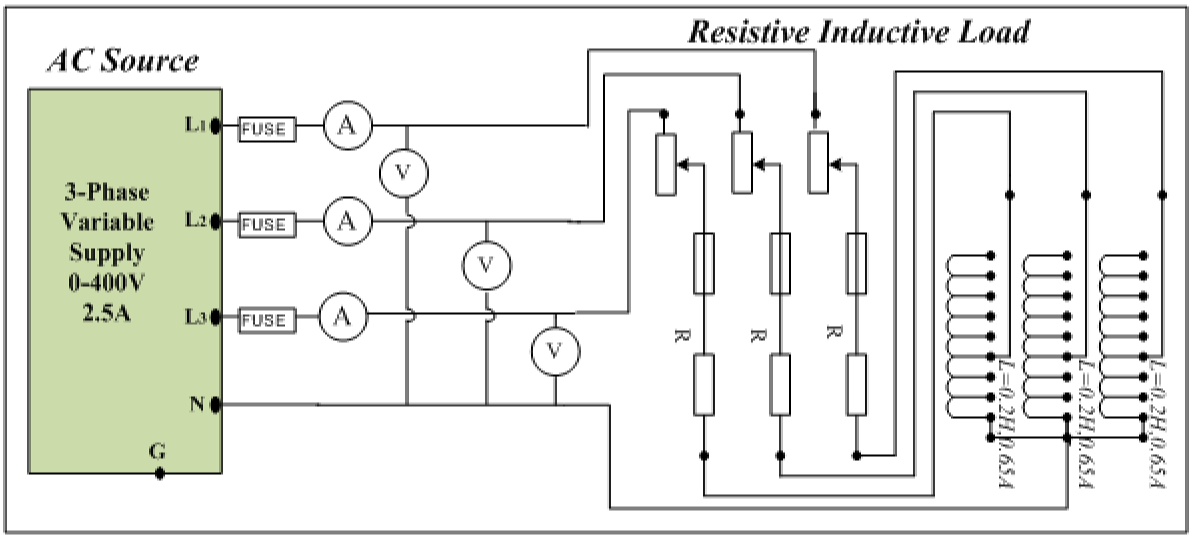


Figure 3 Y-Connected Resistive Inductive Load

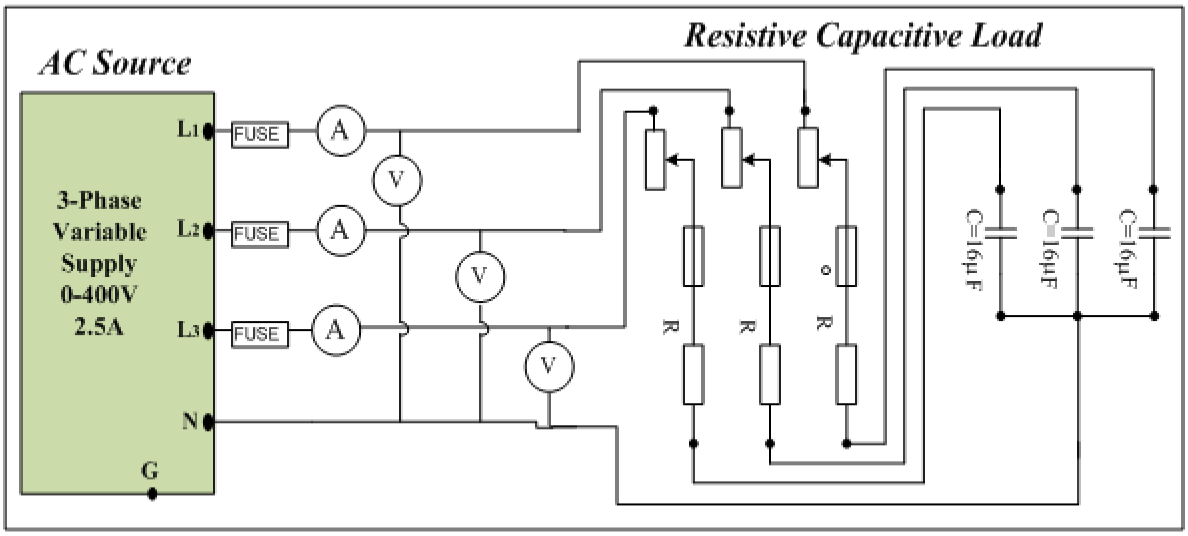


Figure 4 Y-Connected Resistive Capacitive Load

Procedure

1. For preliminary measurements use the digital Multimeter to measure each phase resistance of the resistive load bank as per figure 1 while the resistance control position is at (40%) then enter each phase resistance value in Table 1. **Do not change the position of the resistance knob through the rest of the experiment.**

2. Repeat the phase measurement for the inductive load bank (L=0.2H, 0.65A) internal series resistance for each phase as per figure 2 and enter the values in Table 2.

Table 1: Load Resistance measurements

Resistance control Position (%)	Load1(Ω)	Load2(Ω)	Load2(Ω)
(40%)			

Table 2: Inductance internal series resistances measurements

Inductance (H), Current Rating (A)	Load1(Ω)	Load2(Ω)	Load2(Ω)
0.2 H, 0.65 A			

Simulation

3. Build circuit shown in Fig. 3 using ***Multisim** Electronics Workbench.*
4. Simulate the three phase Y connected voltage source and apply 200 V line to neutral.
5. Use the resistance values recorded in table 1 and table 2 and indicated value to estimate the power factor for each phase.
6. Connect the **ammeters** and **voltmeters** at appropriate positions to measure voltages and currents, real and reactive powers then enter the simulation data shown in Table 3 and table 4.
7. Repeat steps 3 to 6 for the circuit given in Fig. 4 then enter the simulation data values in Table 5 and table 6.

Hardwired Experiment

8. Build the circuit of Fig. 3 with the hardwired components. Take the measurements of voltages, currents, power factor (from step 5) real power, reactive power and frequency then record them in Table 3 and table 4. Considering the Workbench results as the base, compute the percentage errors.
9. Build the circuit of Fig. 4 with the hardwired components. Take the measurements of voltages, currents, power factor (from step 7) real power, reactive power and frequency measurements then record them in Table 5 and table 6. Considering the Workbench results as the base, compute the percentage errors.

Note: (PF: Power Factor and FR: Power Frequency)

Table 3: Simulation and experimental results for Fig. 3

Variable	I_{L1}	I_{L2}	I_{L3}	V_{L1N}	V_{L2N}	V_{L3N}	V_{L1L2}	V_{L2L3}	V_{L3L1}
Workbench									
Hardwired									
% Error									

Table 4: Simulation and experimental results for Fig. 3

Variable	P_{L1}	P_{L2}	P_{L3}	P_{TOT}	Q_{L1}	Q_{L2}	Q_{L3}	Q_{TOT}	PF	FR
Workbench										
Hardwired										
% Error										

Table 5: Simulation and experimental results for Fig. 4

Variable	I_{L1}	I_{L2}	I_{L3}	V_{L1N}	V_{L2N}	V_{L3N}	V_{L1L2}	V_{L2L3}	V_{L3L1}
Workbench									
Hardwired									
% Error									

Table 6: Simulation and experimental results for Fig. 4

Variable	P_{L1}	P_{L2}	P_{L3}	P_{TOT}	Q_{L1}	Q_{L2}	Q_{L3}	Q_{TOT}	PF	FR
Workbench										
Hardwired										
% Error										

Questions

1. Compare the results obtained with Workbench with those from the hardware circuit, and comment on the error obtained between hardwired and Workbench results of each case.

2. What is the relation between the line voltage and the corresponding phase voltage based on the Multisim workbench results? Is it confirmed by the hardwired results**?**

3. Do we get similar relation between phase and line current if the load is Δ connected?

4. In a balanced system, all phase voltages have the same magnitude, what makes the difference between two consecutive voltages higher 170% of the phase voltage?

5. Do reactive powers in Figure 3 and Figure 4 circuit have the same direction? Why?

6. Does the value of the capacitance have any impact on the load voltage magnitude and phase? Explain.

Experiment 8

Transient Circuit Analysis

Introduction

For electrical circuits such as the RC circuit of Fig. 1, the total step response (sum of forced "steady-state" response and "transient" natural response) can be obtained by solving the 1st order differential equation describing the circuit. When there is no initial charge across the capacitor, the voltage $V_C\ (t)$ for $t \geq 0$ is shown to be given by:

$$V_C(t) = V(1 - e^{-\frac{t}{RC}}) \qquad (1)$$

For the 2nd order series RLC circuit shown in Fig.2, the total step response is also given by the sum of the forced and natural response, where the latter can be over-damped, under-damped or critically damped depending on whether the characteristic roots $S_{1,2}$ are real, complex conjugates, or double roots. With $\alpha = R/2L$ and $\omega_0^2 = 1/LC$, we have:

$$s_{1,2} = -\alpha \pm \sqrt{\alpha^2 - w_0^2} \qquad (2)$$

The characteristic roots are real (negative) for $\alpha > \omega_0$, complex conjugates for $\alpha < \omega_0$, or double for $\alpha = \omega_0$. The different forms of $V_C\ (t)$ (for $t \geq 0$) are respectively given by

$$v_c(t) = V_f + A'_1 e^{s_1 t} + A'_2 e^{s_2 t}$$

$$v_c(t) = V_f + B'_1 e^{-\alpha t} Cos\omega_d t + B'_2 e^{-\alpha t} S\text{in}\,\omega_d t$$

$$v_c(t) = V_f + D'_1 t e^{-\alpha t} + D'_2 e^{-\alpha t}$$

where V_f is the final forced response of $V_C(t)$

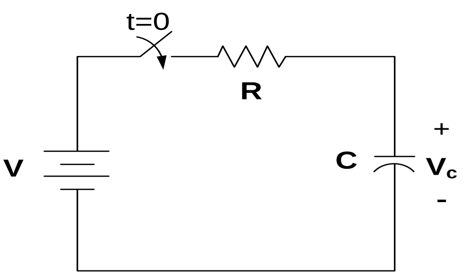


Figure 1: RC series circuit

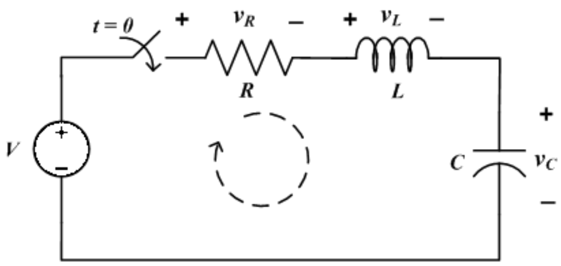


Figure 2: Series RLC circuit

Objectives

The experiment covers the analysis of the transient step response of 1st and 2nd order circuits. The experiment is based on Multisim Workbench simulation and verification with hardware measurements and real components. The main objectives are:

1. To perform transient analysis of a 1st order RC circuit
2. To perform transient analysis of a 2nd order RLC circuit

Part I: Transient Analysis of RC Circuit

Materials

Function Generator
Oscilloscope
22nF Capacitor
3kΩ Resistor

Procedure

Workbench Simulation

1. Build the circuit of Fig. 1 in *Multisim Electronics Workbench*. Note node numbers
2. Run the Transient Analysis. The procedure is as follows:
   a. Select **Simulate** from the **Main Menu**
   b. Select **Analysis**
   c. Select **Transient Analysis**
   d. Select **Analysis Parameters**
      - Initial Conditions: Set to zero
      - Start time: 0 s End time: 0.5 ms
   e. Select **Output Variables**

      Select node **2** (left box), this is the node for the capacitor

      Select **plot during simulation** (right box)
   f. Select **Simulate**
3. Observe the wave shape. Using the cursor tool, record the results and enter the data in Table 1. Plot V_c vs. time on the graph provided.

Hardware Experiment

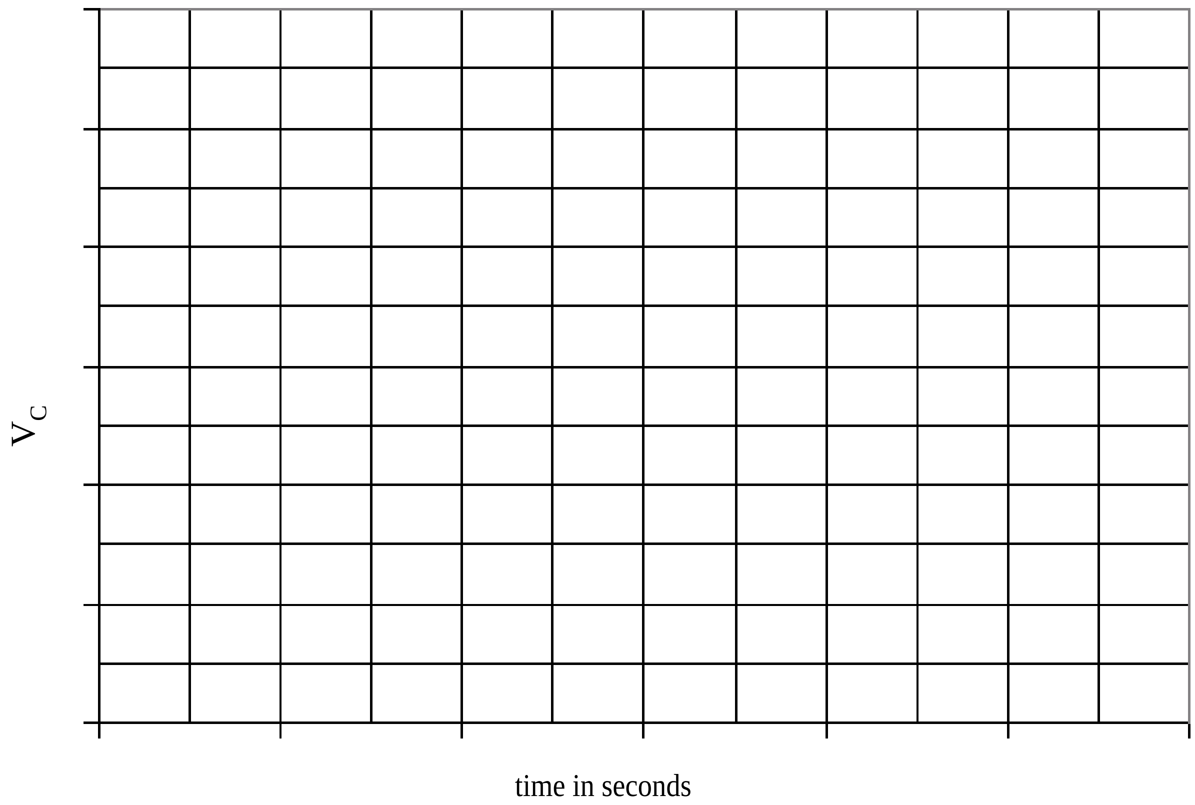
4. Build the circuit given in Fig. 1 with the hardware components. The voltage source should be drawn from the Function Generator.
5. Set the Function Generator to produce a square wave signal with amplitude 5V and frequency 1 KHz. This represents a repetitive (periodic) step voltage input.

6. Apply the input voltage to Channel A of the Oscilloscope, and the capacitor voltage to Channel B. Adjust the amplitude scale to 5V/div and time base 0.1ms/div to get a full-screen display (showing only 1 period of the square wave).
7. Read the values of the output voltage V_C and record the data in Table 1.
8. Plot the voltage response on the same graph paper as the simulation results.

Table 1: V_C Transient response of RC circuit

Time (ms)	0	0.05	0.1	0.15	0.2	0.25	0.3	0.35	0.4	0.5
V_c (Simulation)										
V_c (Measurement)										

Figure 3: Simulated and Measured Transient Voltage Responses



Questions

1. What is the theoretical value of the time constant τ of the circuit in Fig. 1?

2. Explain how to calculate the time constant from the Vc vs. time curves (simulation and hardware). Comment on any difference between them.

3. If you apply a square wave input with frequency 1MHz, explain whether this is suitable for measuring the time constant of the circuit and why?

Part II: Transient Analysis of RLC Circuit

Materials

Function Generator
Oscilloscope
22nF Capacitor
12mH Inductor
220Ω Resistor, 10kΩ Resistor

Procedure

Workbench Simulation

1. Build the circuit of Fig. 2 in *Multisim Electronics Workbench*. Use 220Ω resistor.
2. Run the Transient Analysis following the same procedure as in Part I.
3. Observe and record the V_C waveform. Enter the data in Table 2. Plot V_C vs. time on the graph provided in Fig. 4.

Hardware Experiment

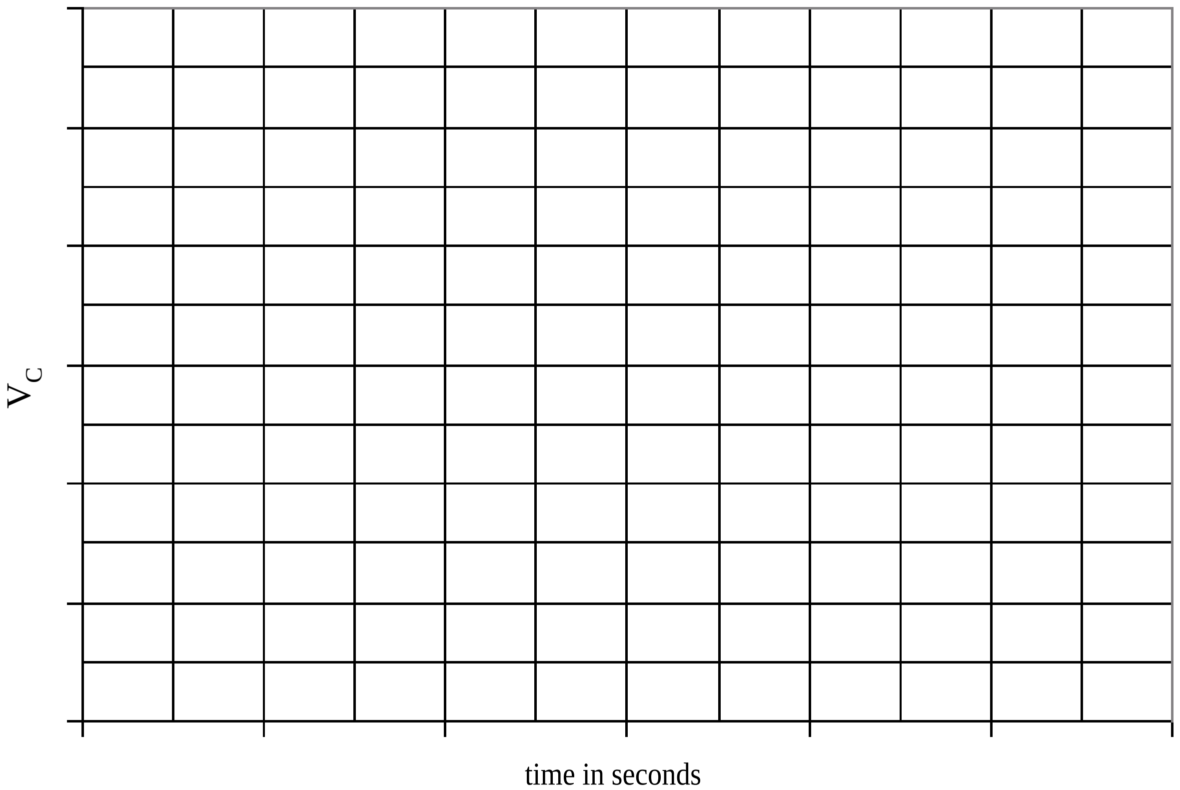
4. Build the circuit of Fig. 2 with the C & L components and the 220Ω resistor. The voltage source is applied from the Function Generator.
5. Set the Function Generator to produce a square wave signal with amplitude 5V and frequency 1 kHz.

6. Apply the input voltage to Channel A of the Oscilloscope, and the capacitor voltage to Channel B. Adjust the scale and time base to get a full-screen display
7. Read the values of the output voltage V_C and record the data in Table 2.
8. Plot the voltage response on the same graph paper as the simulation results.
9. Change the resistor to 10KΩ value. Notice the impact on the transient response.

Table 2: V_C Transient response of RLC circuit

Time (ms)	0	0.05	0.1	0.15	0.2	0.25	0.3	0.35	0.4	0.5
V_c (Simulation)										
V_c (Measurement)										

Figure 4: Simulated and Measured Transient Voltage Responses



Questions

1. What is the type of the transient response? What are the theoretical values of the damping factor α and the oscillation frequency ω_d?

2. Using the measured V_c values, estimate these factors directly from the waveform plots. Comment on any difference between the values.

3. When the resistor is set to 10kΩ, what is the type of the transient response? Explain why?

Experiment 9
Transformer Circuit

Introduction

The principal purpose of a transformer is to convert AC power at one voltage level to AC power of the same frequency at another voltage level. It is made of two winding and a magnetic core. Both windings are linked by magnetic flux through a magnetic core. The transformer has N_P turns at the input or primary side and N_S turns at its output or secondary winding. The relationship between the voltage V_P applied to the primary side of the transformer and the voltage V_P produced on the secondary side is

$$V_P / V_S = N_P / N_S \quad (1)$$

$$N_P / N_S = a \quad (2)$$

Where **a** is defined as the turns ratio of the transformer. The relationship of the current I_P flowing in the primary side of the transformer and the current I_S flowing out of the secondary side of the transformer is

$$I_P / I_S = 1/a \quad (3)$$

The real power PP supplied to the transformer by the primary circuit is given by

$$P_P = V_P I_P \cos(\theta_P) \quad (4)$$

Where θ_p is the angle between the primary voltage and the primary current. The real power P_S supplied by the transformer secondary circuit to its load is given by

$$P_S = V_S I_S \cos(\theta_S) \quad (5)$$

Where θ_S is the angle between the secondary voltage and the secondary current.

Objectives

1. To verify the voltages and currents transformer ratio.
2. To evaluate the Power behavior for loaded single phase transformer.

Materials

1 Variable AC power supply 0-400V.

1 Single-phase transformer. (**N_P = 847, N_S = 456, N_T = 456**)

1 Resistive load bank.

1 Inductive load bank.

1 Capacitive load .

1 Set of 30 safety connectors, blue, black, red

1 Set of 10 safety connectors, green/yellow.

2 Digital Multi meters.

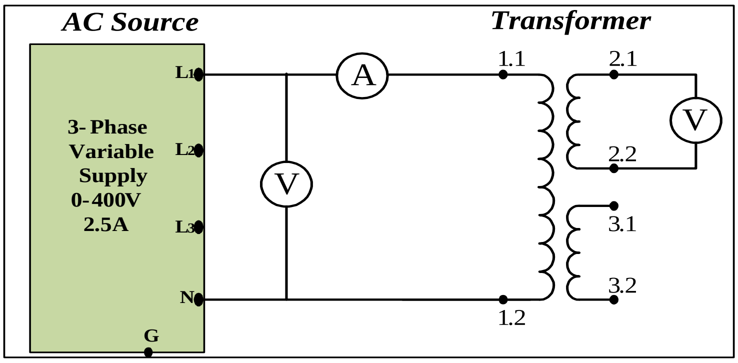


Figure 1 No load test voltage measurements

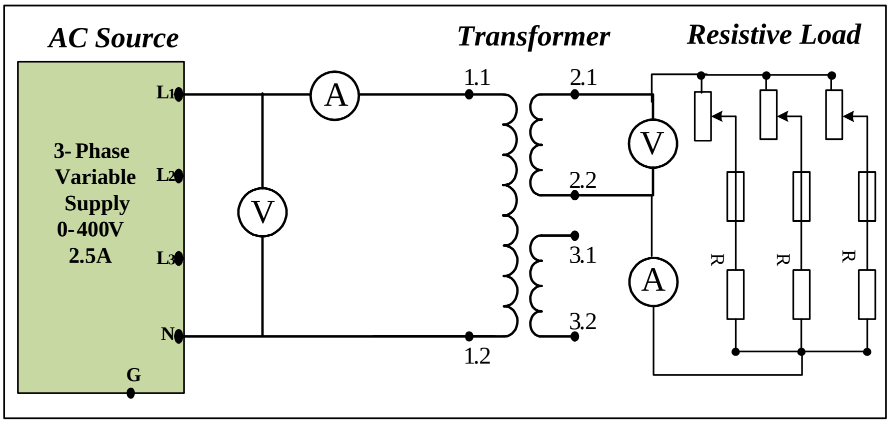


Figure 2 Resistive load voltage, current and power measurements

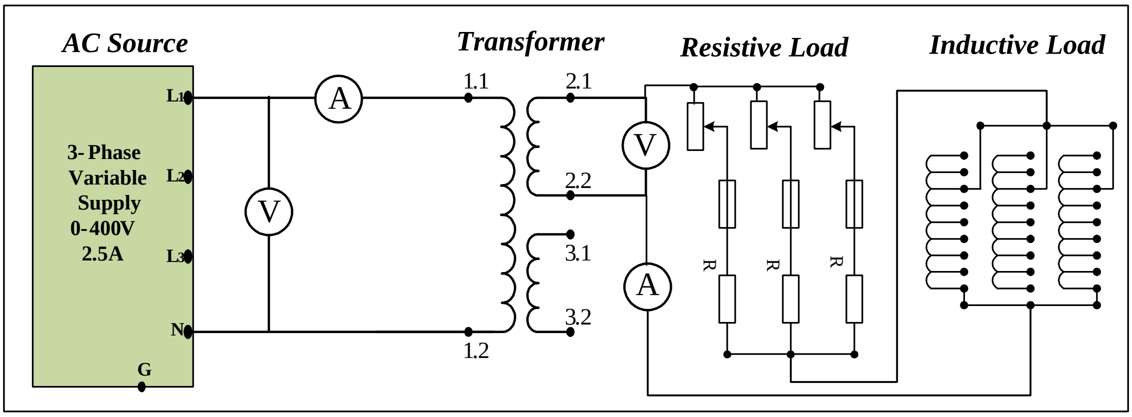


Figure 3 Inductive load voltage, current and power measurement

Procedure

Calculation and Initial Measurements

1. Consider the circuit shown in Fig. 1 then fixe the source voltage at 224 V line to neutral.
2. Place the ammeter and voltmeters at appropriate position in Fig. 1 circuit. Calculate the voltages, current and the apparent power then enter the calculated data in Table 1.
3. While the resistive load bank is separated from Fig. 2 circuit, **fixe its control knob at 40%** then measure the per phase load resistance using the Multimeter.

4. Calculate the voltages, currents and powers for the connected Fig. 2 circuit then enter the data in Table 2.
5. While the inductive part of the load is separated from Fig. 3 circuit, measure the per phase internal series resistance of the inductive load using the Multimeter. Choose the per phase inductance 1.2 Henry.
6. Calculate the voltages, currents and powers for the connected Fig. 3 circuit and enter the obtained data in Table 3

Hardwired Experiment

7. Build the circuit of Fig. 1 with the hardwired components. Take the measurements of voltages, currents, and record them in Table 1. Considering the calculated results as the base, compute the percentage errors.
8. Build the circuit of Fig. 2 with the hardwired components **control knob at 40%**). Take the measurements of voltages, currents and powers then record them in Table 2. Considering the calculated results as the base, compute the percentage errors.
9. Build the circuit of Fig. 3 with the hardwired components (**control knob at 40%**). Take the measurements of voltages, currents and powers then record them in Table 3. Considering the calculated results as the base, compute the percentage errors

Table 1: Simulation and experimental results for Fig. 1

Quantity	I_S	V_P	S_P	V_S
Calculated				
Hardwired				
% Error				

Table 2: Simulation and experimental results for Fig. 2

Quantity	V_P	I_P	P_P	Q_P	V_S	I_S	P_S	Q_S
Calculated								
Hardwired								
% Error								

Table 3: Simulation and experimental results for Fig. 3

Quantity	V_P	I_P	P_P	Q_P	V_S	I_S	P_S	Q_S
Calculated								
Hardwired								
% Error								

Questions

1. Compare the calculated results with those obtained from the hardware circuit, and comment on the error obtained between hardwired and calculated results of each case.

2. Is relation (2) verified with relation (1) in figure 1 circuit?

3. Is $S_P = S_S$? Why? ($S_P=V_P*I_P$ and $S_S=V_S*I_S$)

4. Why do we have primary current in the circuit of figure 1 although there is no load at the secondary side of the transformer?

5. What is the reason behind V_s of circuit in figure 2 being lower then V_s of circuit in figure 1?

6. What is the nature of the current I_S in the circuit of figure 3? Is it in favor of the voltage support?

7. Physically! what causes the consumption of Q_P and Q_S in the circuit in figure 3? Explain?

Experiment 10
Frequency Selective Circuits

Introduction

Frequency selective circuits exhibit a behavior that depends on the frequency of the applied signals. Filters (and other circuits such as resonant circuits) are typical examples of frequency selective circuits, and have many numerous applications in electrical engineering. In this experiment, we study examples of low-pass and band-pass filters.

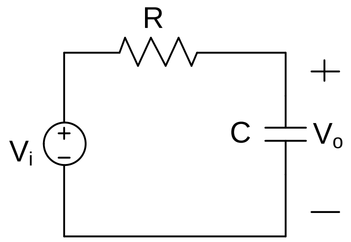


Figure 1: RC Low-Pass filter circuit

For the RC circuit example shown in Fig. 1, the transfer function $H(s)=V_o(s)/V_i(s)$ is easily obtained by VDR. The evaluation of its magnitude and phase (for s=jω) gives:

$$|H(j\omega)| = \frac{\frac{1}{RC}}{\sqrt{\omega^2 + \left(\frac{1}{RC}\right)^2}} \qquad \theta(j\omega) = -\tan^{-1}\omega RC \qquad (1)$$

This is a low-pass filter with half-power (3dB) cutoff frequency $\omega_c=1/RC$.

On the other hand, for the RLC circuit of Fig.2 (where the output is taken across R), the resulting transfer function will have the following magnitude and phase:

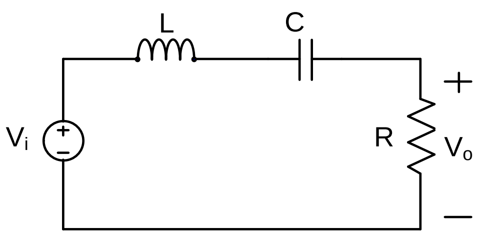


Figure 2: Band-pass RLC filter circuit

$$|H(j\omega)| = \frac{\omega\frac{R}{L}}{\sqrt{\left(\frac{1}{LC}-\omega^2\right)^2 + \left(\omega\frac{R}{L}\right)^2}} \qquad \theta(j\omega) = 90^\circ - \tan^{-1}\left(\frac{\omega\frac{R}{L}}{\frac{1}{LC}-\omega^2}\right) \qquad (2)$$

The circuit frequency response is that of a band-pass filter with center frequency (in rd/s):

$$\omega_0 = \sqrt{\frac{1}{LC}} \qquad (3)$$

and the 3dB (half-power) cutoff frequencies are given (in rd/s) by:

$$\omega_{C1} = -\frac{R}{2L} + \sqrt{\left(\frac{R}{2L}\right)^2 + \left(\frac{1}{LC}\right)} \qquad (4)$$

$$\omega_{C2} = \frac{R}{2L} + \sqrt{\left(\frac{R}{2L}\right)^2 + \left(\frac{1}{LC}\right)} \qquad (5)$$

Objectives

The experiment deals with the steady-state analysis of frequency selective filter circuits. The experiment is based on Multisim Workbench simulation and hardware measurements. The main objectives are:

1. To perform AC frequency response analysis of a low-pass RC filter circuit
2. To perform AC frequency response analysis of a band-pass RLC filter circuit

Part I: Frequency Response of Low-pass RC Filter Circuit

Materials

Function Generator
Oscilloscope
22nF Capacitor
1kΩ Resistor

Procedure

Workbench Simulation

1. Build the circuit of Fig. 1 in *Multisim Electronics Workbench*.
2. Select **Options/Preferences**, to show node numbers
3. Select **Simulate** from the **Main Menu**
4. Select **Analysis**/**AC Analysis**
5. Select **Analysis Parameters**
   a. Start frequency: 1Hz
   b. Stop frequency: 100kHz
   c. Sweep type: decade
   d. Number of pts/dec: 5
   e. Vertical scale: linear
6. Select **Output Variables**
   a. Select node **2** (left box) for the capacitor voltage
   b. Select **plot during simulation** (right box)

7. Select **Simulate**
8. Observe the filter frequency response. Using the cursor tool, read the coordinates of the points on the plot. In particular, read the values at the half-power cutoff frequency
9. Fill Table 1, and plot the filter magnitude and phase response on the graph provided.

Hardware Experiment

10. Build the circuit given in Fig. 1 with the hardware components. The voltage source is applied from the Function Generator.
11. Set the Function Generator to produce a sine wave signal with fixed 1V amplitude (the frequency will be changed from 1 to 100 kHz).
12. Apply the input voltage to Channel A of the Oscilloscope, and the capacitor voltage to Channel B. The display should be set to facilitate reading the output voltage gain and phase shift with respect to the input.
13. Change the input frequency in small steps, taking a few points per decade. Pay attention to take a measurement at the cutoff frequency.
14. Read the values of the output voltage magnitude and phase (w.r.t the input voltage) and record the data in Table 1. Plot the circuit magnitude response and phase response on the same graph paper as the simulation results.
15. Change the input waveform shape to square wave type, with a 5 kHz frequency. Observe and record the output signal.

Table 1: Frequency response of RC LPF

Freq (Hz)											
Mag(V_o)											
Phase(V_o)											

Figure 3A: RC LPF Frequency Response

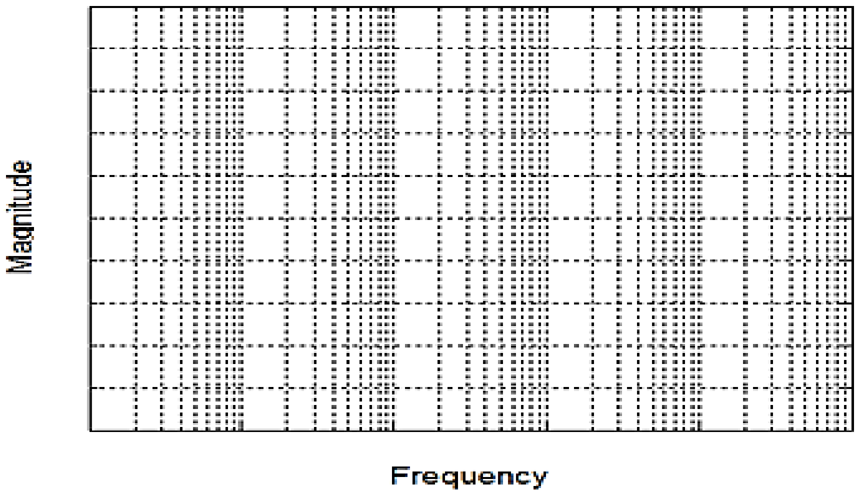
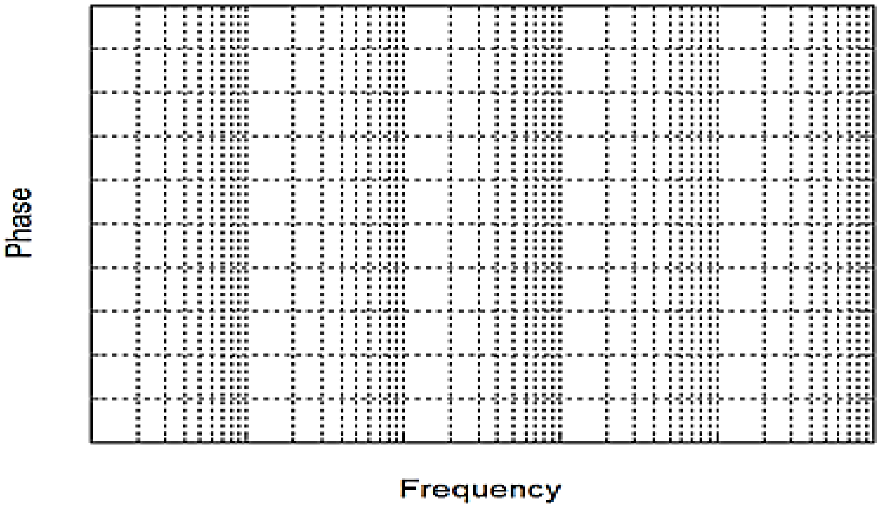


Figure 3B: RC LPF Frequency Response



Questions

1. From the plotted graphs, what is the filter type? What it the measured 3dB cutoff frequency (in Hz)? Compare it to the theoretical value.

2. Explain the reason for your observations with the square wave input in Step 9.

3. If the capacitor is replaced with a 10mH inductor, explain qualitatively the impact on the filter type and its characteristics

Part II: Frequency Response of Band-pass RLC Filter Circuit

Materials

Function Generator
Oscilloscope
5nF Capacitor
12mH Inductor
1kΩ Resistor, 220Ω Resistor

Procedure

Workbench Simulation

1. Build the circuit of Fig. 2 in *Multisim Workbench* (use the 1kΩ resistor first).
2. Run the frequency sweep analysis following the same procedure as in Part I. Use the cursor to mark the values for cutoff frequencies.
3. Observe and record the circuit magnitude and phase frequency response. Enter the data in Table 2. Plot the magnitude and phase on the graph provided in Fig. 4.
4. Change the resistor to 220Ω, observe and record the impact on the response.

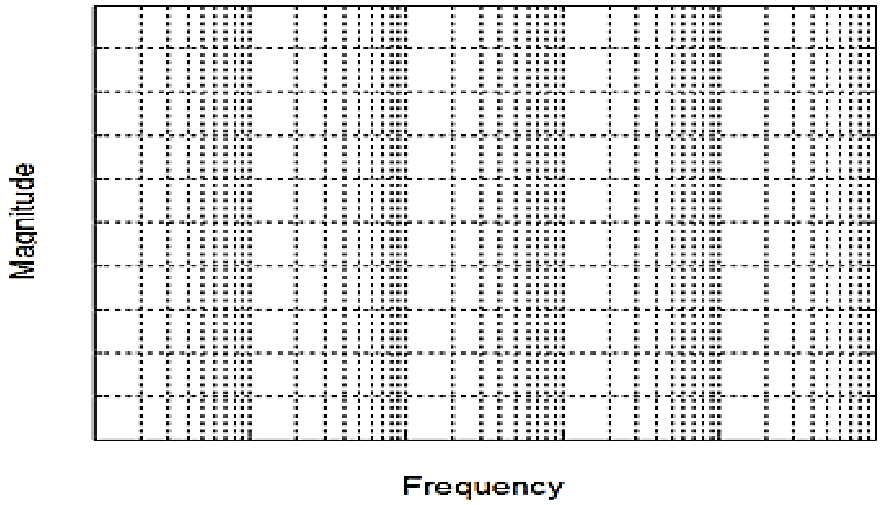
Hardware Experiment

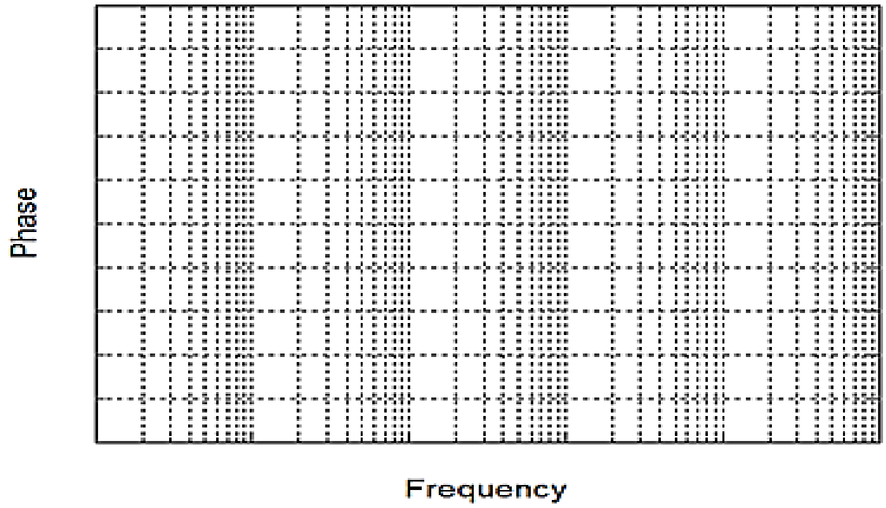
5. Build the circuit of Fig. 2 with the hardwired R, L & C components. The voltage source is applied from the Function Generator.
6. Set the Function Generator to produce a sine wave signal with 1V fixed amplitude (the frequency will be varied from 10Hz to100kHz).
7. Apply the input voltage to Channel A of the Oscilloscope, and the resistor voltage to Channel B.
8. Change the frequency in small steps, taking a few points per decade. Make sure to take measurements at the cutoff frequencies. Record the values of the output voltage magnitude and phase (with respect to the input voltage) in Table 2.
9. Plot the circuit magnitude and phase responses on the same graph paper as the simulation results.

Table 2: Frequency response of RLC BPF

Freq (kHz)											
Mag(V_o)											
Phase(V_o)											

Figure 4: RLC BPF Frequency Response





Questions

1. Comment on the frequency response type?

2. For each of the two resistors, specify the values of the half-power (3dB) cutoff edge frequencies (in Hz)? Discuss and compare with theoretical values

3. What are the bandwidth and quality factor (for both resistors)?

4. Comment of the resonance behavior of this circuit (for both resistors)

5. If the output is taken across L&C (together), explain qualitatively the type of filter in this case, and why?

Experiment 11
Two-Port Networks

Introduction

For many electrical systems (or networks), focusing on the input/output terminals can provide convenient models for circuit analysis without worrying about the internal details. Often, a signal is fed into input terminals, processed then extracted at the other output terminals. There are four variables of interest in this case, as shown in Fig.1.

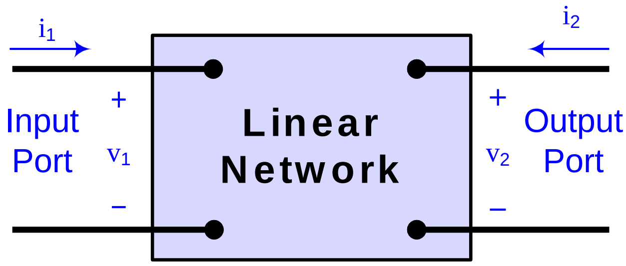


Figure 1: Two-port circuit model.

There are six groups of two-port circuit parameters (e.g., the impedance z-parameters, admittance y-parameters, hybrid h-parameters, etc) that can be used to relate the input/output voltage & current variables. Any set of parameters can be derived from the other ones. In this lab, we consider the case of (impedance) z-parameters.
The z-parameters are defined by:

$$\begin{aligned} V_1 &= z_{11} I_1 + z_{12} I_2 \\ V_2 &= z_{21} I_1 + z_{22} I_2 \end{aligned} \qquad (1)$$

These parameters can be measured by the following tests (based on open-circuit input port condition, and open-circuit output port condition):

$$z_{11} = \left.\frac{V_1}{I_1}\right|_{I_2=0}, \quad z_{12} = \left.\frac{V_1}{I_2}\right|_{I_1=0}, \quad z_{21} = \left.\frac{V_2}{I_1}\right|_{I_2=0}, \quad z_{22} = \left.\frac{V_2}{I_2}\right|_{I_1=0} \qquad (2)$$

Objectives:

The experiment is based on Workbench simulation and hardware measurements. The main objectives are:

1. To perform z-parameters characterization of a given two-port circuit
2. To use the z-parameters model for analysis of a loaded two-port circuit

Part I: Z-Parameters Characterization

Materials

Voltage Source
Multimeter
220Ω, 1kΩ Resistors

Procedure

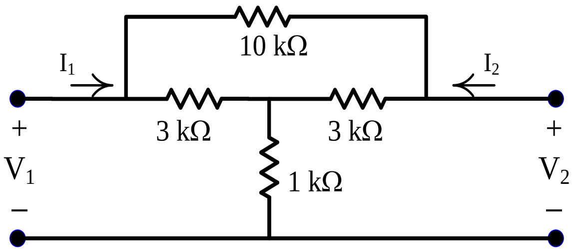


Figure 2: Two-Port Circuit Diagram

Workbench Simulation

1. Build the circuit of Fig. 2 in *Multisim Electronics Workbench*.
2. Apply a DC voltage test source to the input port with the output port open. Use a voltmeter and ammeter to read the values of the voltages and currents necessary to compute Z_{11} and Z_{21}. Observe the proper polarity of the currents.
3. Repeat a similar procedure with a DC voltage test source applied to the output port with the input port open. Use a voltmeter and ammeter to read the voltage and current values, and derive the parameters Z_{22} and Z_{12}.
4. Record the values of the Z-parameters in Table 1.

Hardware Experiment

5. Build the circuit given in Fig. 2 with the hardware components.
6. Follow identical steps to (2) and (3) above by applying a DC test voltage to the input and output ports alternatively, and measuring the relevant voltages and currents. Again, pay attention to the direction (and sign) of the currents
7. Compute and record the Z-parameters based on hardware measurements in Table 1.

Table 1: Z-Parameters of Two-Port Circuit

Z-parameters	Z_{11}	Z_{12}	Z_{21}	Z_{22}
Simulation				
Measurement				

Questions

1. Compare the simulation and hardware measurement results. What would be the sources of any discrepancies?

2. Explain whether this two-port circuit is reciprocal, and why?

3. Explain whether this two-port circuit is symmetric, and why?

4. Deduce the values of the Y-parameters directly from your results

Part II: Analysis of Loaded Two-Port Circuit

Materials

Voltage Source
Voltmeter, Ammeter
Assorted Resistors: 100Ω, 1kΩ, 3kΩ, 10kΩ, 1kΩ

Procedure

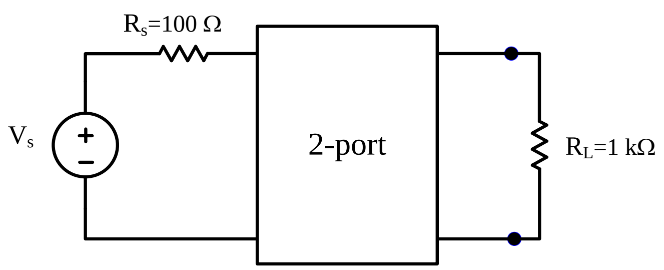


Figure 3: Loaded Two-Port Circuit

Workbench Simulation

1. Build the circuit of Fig. 3 in *Multisim Workbench*.
2. Run the simulation with a 10V DC voltage source. Insert Multimeters to read the voltage and current of the load resistance.
3. Evaluate the voltage gain $G_v = V_2/V_s$ and current gain $G_i = I_2/I_1$
4. Record your results in Table 2.

Hardware Experiment

5. Build the circuit of Fig. 3 with the hardware components.
6. Follow identical steps to (2) and (3) above by applying a 10V DC voltage to the input side. Use voltmeter and ammeter to read the relevant voltage and current.
7. Evaluate the voltage gain $G_v = V_2/V_s$ and current gain $G_i = I_2/I_1$
8. Record the results in Table 2.

Table 2: Voltage and Current Gains

Load Variables	G_v	G_i
Simulation		
Measurement		

Questions

1. Based on the Z-parameters obtained in Part I, derive the expressions for the voltage and current gains G_v and G_i
(Hint: write KVL equations for the input and output loops, in addition to the Z-parameters equations)

2. Derive the equivalent Thevenin source model as seen by the load resistance

3. Comment on the usefulness of the Z-parameters (and other types of parameters) for analyzing circuits such as that of Fig. 2

Appendix I

EE213 Design Experiment Sample

1. Analysis and Tuning of a Band-Reject Filter

Part 1: Theoretical Analysis

Consider the circuit given in the figure.

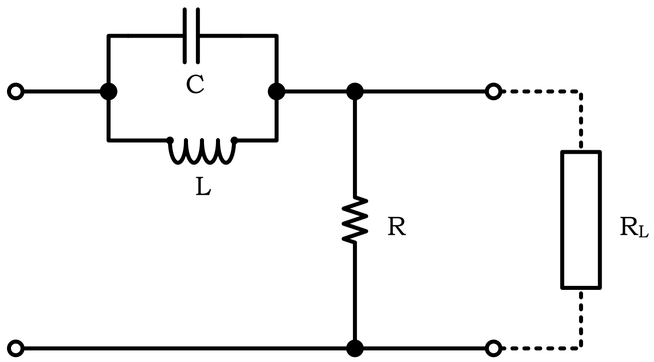


Figure 1: BRF Circuit

a) Show by qualitative analysis that the circuit is a band-reject filter. What is the rejected frequency?
b) Derive the voltage transfer function and validate your answer to (a)
c) Derive the expressions of the cutoff corner frequencies
d) Derive the expressions of the bandwidth and quality factor

Part 2: Multisim Workbench Simulation

a) Build the circuit in Multisim Workbench
b) Choose the components such that the reject frequency band is centered at 10kHz and the quality factor is at least 5
c) Verify the proper operation of your design by plotting the filter frequency response
d) Investigate the impact of connecting a load resistor to the filter. Discuss and explain your results

Part 3: Experimental Testing

a) Build the circuit with hardware components
b) Use the function generator and oscilloscope to test the filter design you obtained in Part 2
c) Investigate experimentally the impact of a load resistor (test different values) on the filter reject frequency and selectivity
d) Discuss your results and draw relevant conclusions

EE213 Design Experiment Sample

2. Analyzing a Factory and Designing an Expansion

A factory located in the industrial area is planning for an expansion. As an engineer you have been assigned to perform all of the required analysis and design calculations for this expansion. The circuit given below represents a circuit model of the devices in the factory before the expansion.

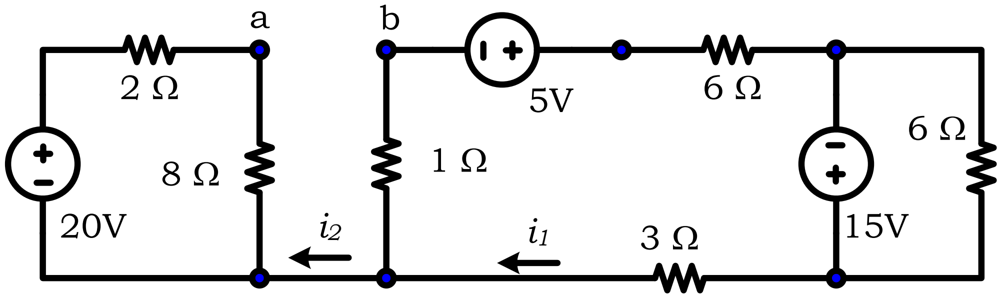


Figure 1: Circuit Model of the old factory

Part 1: Required analysis of the existing factory

a. Calculate the currents ***i_1* and *i_2*.**

b. Calculate **the power of each source in the circuit**.

c. Calculate the **voltage V_{ab}** of the factory before the expansion.

d. Build the circuit in Multisim Workbench verify your calculations in a, b and c above

Part 2: Required design calculations for the new expansion

Knowing that the devices of the new expansion will be connected between terminals a and b, design the elements of a model representing the new expansion to the factory if you are given that after the expansion the **voltage V_{ab}** is to be 20V**.**

a. Build the complete circuit in Multisim Workbench and verify the proper operation of your design by measuring the **voltage V_{ab}**.

b. Build the circuit with hardware components and verify the proper operation of your design by measuring the **voltage V_{ab}**.

c. What will be the new values of the currents ***i_1* and *i_2*.**

d. Discuss your results and draw relevant conclusions

EE213 Design Experiment Sample

3. Design a Scheme to Improving Load Power Factor

The utility company is supplying a static three phase Y connected load at 200 V line to neutral. Each phase is composed an impedance of 350 Ω in series with 1.2 H inductor.

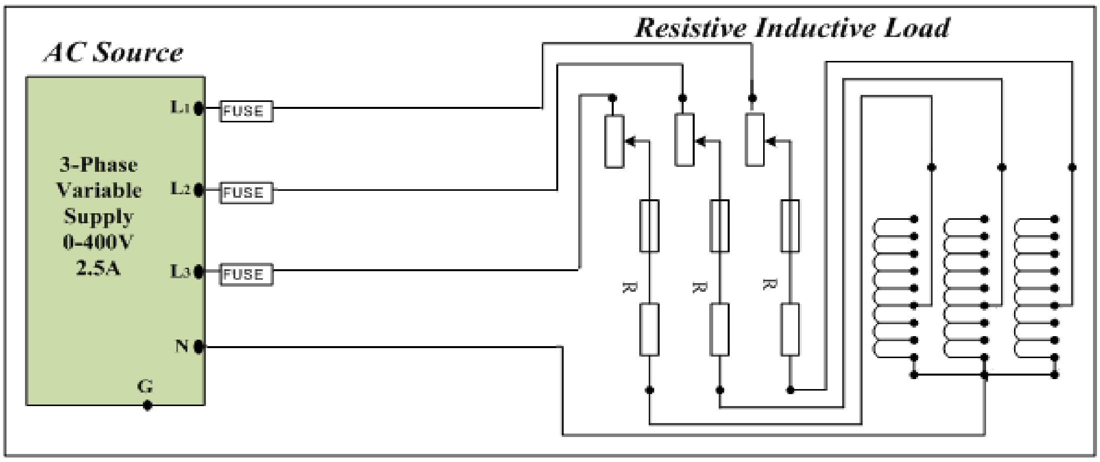
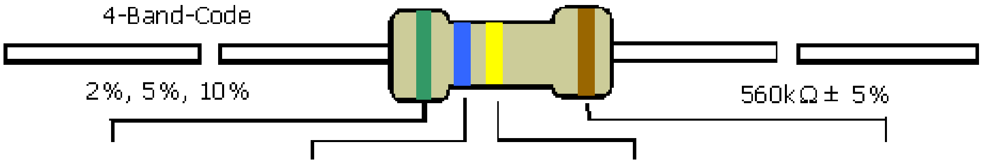


Figure 1 The Load Circuit Connection to Utility

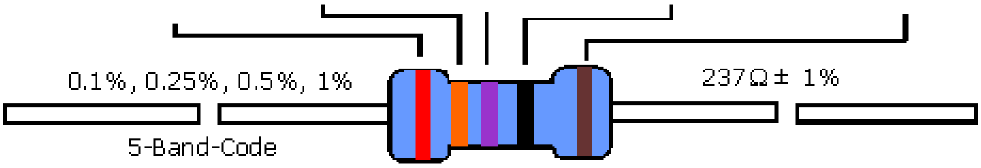
The load nature presents a costly and unacceptable low lagging power factor to the power company. The load power factor has to be improved to 0.8 lagging or higher. It is required to evaluate the actual power factor and then to design a scheme to be installed for improving the load power factor. The required design steps has to conducted then the hardwire design has to be installed and tested. Practical results are required.

Appendix II

Resistor Conversion Table



COLOR	1st BAND	2nd BAND	3rd BAND	MULTIPLIER	TOLERANCE
Black	0	0	0	1Ω	
Brown	1	1	1	10Ω	± 1% (F)
Red	2	2	2	100Ω	± 2% (G)
Orange	3	3	3	1KΩ	
Yellow	4	4	4	10KΩ	
Green	5	5	5	100KΩ	±0.5% (D)
Blue	6	6	6	1MΩ	±0.25% (C)
Violet	7	7	7	10MΩ	±0.10% (B)
Grey	8	8	8		±0.05%
White	9	9	9		
Gold				0.1	± 5% (J)
Silver				0.01	± 10% (K)



Capacitor Conversion Table

uf,microFarads $=10^{6}$ Farads	nf,nanoFarads $=10^{9}$ Farads	pf,picoFarads $=10^{12}$ Farads	Multiplier used Exemple = .01uF = 103
0.000001uf=	0.001nf=	1pf	(1)
0.00001uf=	0.01nf=	10pf	(10)
0.0001uf=	0.1nf=	100pf	(101)
0.001uf=	1nf=	1000pf	(102)
0.01uf=	10nf=	10000pf	(103)
0.1uf=	100nf=	100000pf	(104)
1.uf=	1000nf=	1000000pf	(105)
100uf	100000nf=	10000000pf	———

Appendix III

LABORATORY REGULATIONS AND SAFETY RULES

The following regulations and safety rules must be observed in all laboratory locations.

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 equipments are properly working before using 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 in the presence of the instructor or lab staff.**
5. **Students should keep a safe distance from the circuit breakers, electric circuits or any moving parts during the experiment.**
6. **Avoid any body contact between energized circuits and ground.**
7. **Switch off equipments and disconnect power supplies from the circuit before leaving the laboratory.**
8. **Observe cleanliness and proper laboratory housekeeping of the equipment and other related accessories.**
9. **Wear the proper clothes and safety gloves or goggles required in working areas involving fabrications of printed circuit boards, chemical process control systems, 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 or 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, eating and drinking in the laboratory are prohibited.**

The above rules and regulations are necessary precautions in the electrical laboratory to safeguard the students, laboratory staff, the equipments and other laboratory users.