COMSATS University Islamabad (CUI), Lahore Campus
Department of Electrical & Computer Engineering

CPE121 –Electric Circuit Analysis-I

Lab Manual for Semester Fall 2022

Lab Resource Person

Muhammad Usman

Theory Resource Person(s)

Dr. Arsla Khan

Mr. Ahmad Mdassir

Supervised By

Dr. Khurram Zaidi

Name: ___________________________ **Registration #:** *CUI/ - - /LHR*

Program: ________________________ **Batch:** ___________________________________

Semester ____________________________

Revision History

Sr. #	Activity	Date	Performed by
1	Lab Manual Review	Aug-2014	Mr. Mohsin Ali, Ms Sabeen Amin, Mr. Muhammad Usman
2	Lab Manual Review	Mar-2015	Mr. Hasnain Kashif, Mr. Abdul Moeed Amjad, Ms Amna Arif, Mr. Muhammad Usman
3	Lab Manual Modified And Updated	Sep-2015	Inam Ullah Khan
4	Lab Manual Review (again)	May-2016	Mr. Abdul moeed Amjad, Mr. Waheeb Ahmad Butt, Mr. Inam Ullah Khan, Ms. Ghazala Mushtaq
5	Lab Manual Modification and Updated	July-2016	Arfa Tariq
6	Lab Manual Modification According To OBE	Nov-2017	Arfa Tariq, Sara Sajid
7	Lab Manual Review	Nov-2017	Dr. Muhammad Jawad
8	Layout Modifications	Sep-2018	Sarmad Hassan
9	Lab Manual Modification and Updation	Feb-2020	Sara Sajid
10	Lab Manual Review	Feb-2020	Dr. Farooq-i-Azam
11	Lab Manual Modification and Updation	Mar-2022	Ghazala Mushtaq
12	Lab Manual Review	Mar-2022	Dr. Khurram Zaidi

Preface

Acquiring knowledge and skills in any discipline is hard work and perhaps more so in the field of circuit analysis. The material presented in this manual is intended to establish a clear relationship between the basic principles of electric circuit analysis, problem solving procedures and their implementation on laboratory trainers for analyzing electric circuits. Clarity, conciseness, completeness and above all readability are intended as the major characteristics of this of this manual. Electric circuit analysis is one of the first courses of study required in electrical engineering degree programs. The laboratory manual of Electric Circuit Analysis-I provides a comprehensive practical coverage of basic Electrical Engineering concepts. It provides introduction to the use of the Multi-meter along with detailed experiments on circuit analysis techniques. Do not except every concept to become clear immediately. Most of the practical are not overtly difficult but some may require several readings, implementation on trainers and in some cases help from your lab instructor before you can really understand them.

Books

Text Books

1. "Fundamentals of Electric Circuits", 4th International edition, by Alexander published 2013.
2. "Engineering Circuit Analysis", 7th International Edition, McGraw-Hill, written by William Hayt, Kammerly and Durbin Published 2014.

Reference Books

1. Schaum outline series on Electric Circuits
2. Network Analysis by William Val Valkanberg

Learning Outcomes

Theory CLOs:

1. Apply the basic concepts of current, voltage, power, energy and manipulation of basic voltage, current and series-parallel combination laws on various circuit elements (PLO1-C3)
2. Analyze the steady state response and transient responses of R, L and C circuits using principle of linearity and superposition, source transformation, and Thevenin's and Norton's equivalent circuits for electric circuit analysis. (PLO2-C4)

Lab CLOs:

1. To reproduce the electric circuits and show the electrical measurements to derive the valid conclusion using hardware and software tools. (PLO5-P3)
2. To demonstrate the output/result of circuit individually and in teamwork affectively through simulation and hardware during the laboratory sessions. (PLO9-A3)*

* Lab CLO 2 will be evaluated in Mid-Term and Terminal Lab Exam

PLO / CLOs	PLO1	PLO2	PLO3	PLO4	PLO5	PLO6	PLO7	PLO8	PLO9	PLO10	PLO11	PLO12	Cognitive level	Psychomotor Level	Affective Level
CLO1	X												C3		
CLO2		X											C4		
Lab CLO1					X									P3	
LabCLO2									X						A3

CLO-PLO Mapping

Lab CLOs – Lab Experiment Mapping

Lab / CLO	Lab 1	Lab 2	Lab 3	Lab 4	Lab 5	Lab 6	Lab 7	Lab 8	Lab 9	Lab 10	Lab 11	Lab 12
Lab CLO1	P1	P2	P2	P2	P3	P3	P3	P3	P3	P3	P3	P3

Grading Policy

The final marks for lab would comprise of Lab Assessment (25%), Lab Midterm (25 %) and Lab Terminal (50%).

Lab Assignments: i. Lab Assignment 1 Marks (Lab marks from Labs 1-3) ii. Lab Assignment 2 Marks (Lab marks from Labs 4-6) iii. Lab Assignment 3 Marks (Lab marks from Labs 7-9) iv. Lab Assignment 4 Marks (Lab marks from Labs 10-12)	25%
Lab Mid Term = 0.5*(Lab Mid Term) + 0.5*(average of lab evaluation of Lab 1-7)	25%
Lab Terminal = 0.5*(Lab Terminal Exam) +0.375*(average of lab evaluation of Lab 8-12) + 0.125*(average of lab evaluation of Lab 1-7)	50%
Total (lab)	**100%**

The minimum passing marks for both lab and theory shall be 50%. Students obtaining less than 50% marks (in either theory or lab, or both) shall be deemed to have failed in the course. The final marks would be computed with 75% weight to theory and 25% to lab final marks.

List of Equipment

- Bread Board Panel
- Digital Multi Meter (DMM)
- Power Supply
- Jumper Wire
- Resistor
- Capacitor
- Inductor

Software Resources

- PSpice
- Proteus

Lab Instructions

- This lab activity comprises of three parts: Pre-lab, In-Lab Exercises, and Post-Lab Viva session
- The students should perform and demonstrate each lab task separately for step-wise evaluation (please ensure that course instructor/lab engineer has signed each step after ascertaining its functional verification)
- Only those tasks that completed during the allocated lab time will be credited to the students. Students are however encouraged to practice on their own in spare time for enhancing their skills

Safety Instructions

The following general rules and precautions are to be observed at all times in the laboratory. These rules are for the benefit of the experimenter as well as those around him/her. Additional rules and precautions may apply to a particular laboratory.

1. There must be at least two (2) people in the laboratory while working on live circuits.
2. Remove all loose conductive jewelry and trinkets, including rings, which may come in contact with exposed circuits.
3. When making measurements, form the habit of using only one hand at a time. No part of a live circuit should be touched by the bare hand.
4. Keep the body, or any part of it, out of the circuit. Where interconnecting wires and cables are involved, they should be arranged so people will not trip over them.
5. Be as neat a possible. Keep the work area and workbench clear of items not used in the experiment.
6. Always check to see that the power switch is OFF before plugging into the outlet. Also, turn instrument or equipment OFF before unplugging from the outlet.
7. When unplugging a power cord, pull on the plug, not on the cable.
8. When disassembling a circuit, first remove the source of power.
9. "Cheater" cords and 3-to-2 prong adapters are prohibited unless an adequate separate ground lead is provided, the equipment or device is double insulated, or the laboratory ground return is known to be floating.

10. No ungrounded electrical or electronic apparatus is to be used in the laboratory unless it is double insulated or battery operated.
11. Keep fluids, chemicals, and beat away from instruments and circuits.
12. Report any damages to equipment, hazards, and potential hazards to the laboratory instructor.
13. If in doubt about electrical safety, see the laboratory instructor. Regarding specific equipment, consult the instruction manual provided by the manufacturer of the equipment. Information regarding safe use and possible- hazards should be studied carefully.

Table of Contents

LAB # 1

To identify the values of resistances and describe how to build electric circuit using hardware and Software tool

Objectives

- To identify values of resistances using color codes.
- To Identify the values of resistance using Digital Multimeter (DMM)
- To describe how to build the electrical circuits using Bread Board, Jumper wire, and DC power supply
- To describe how to build the electrical circuits using Pspice Software

Pre Lab Section

Part 1 -Familiarize yourself with equipment frequently used in lab

Breadboard

This is used to construct circuits using components and jumper wires. All the components are mounted on to the breadboard with jumper wires used to connect them to each other. The breadboard also has a DC power supply ranging from 0-15 VDC. Also a constant 5 VDC output is available. The circuit ground connection is also available.

Jumper wires

These are used to connect circuit components to each other and to the supply available on the breadboard or the function generator. These can be simple wires as well as "Alligator Clip end" wires.

Function generator

This is a device which produces different types of supply voltages and currents, some of which are DC, sinusoidal, square and saw-tooth waveforms.

Multimeter

It is used for measurement of circuit variables (like voltage and current), circuit continuity, resistance, diode terminal identification etc. some multi-meters also have an option of measurement of temperature. These are mainly of two types, digital (with LCD display of readings) and analogue (with moving needle on a graduated scale). There are two probes, one colored red and the other colored black. The black probe is plugged into a socket marked **"COM"**. There are three sockets for the red probe to be plugged into. One is marked **"V, Ω, diode, temp"** indicating the position where this probe should be while measuring/ identifying voltage, resistance, diode, temperature etc. The second socket is marked **"mA"** indicating the position of the red probe for measuring current in the milli-Ampere range. The third socket is marked **"A"** indicating the position of the red probe for measuring current in Ampere range. There is a selector available on the meter, which has to be moved to appropriate variable and range while measuring any variable / element.

Resistors

Essential in construction of any circuit, these have two terminals for connection with other circuit elements or plugging into breadboard. These are color coded according to their values and permissible tolerances.

Part 2 – Calculation of Resistance with color codes and verification through measurement with Multi-meter

Introduction

The calculation from color codes is done according to the following rules:

Table 1.1

Color	Digit	Multiplier	Tolerance (%)
Black	0	10^0 (1)	
Brown	1	10^1	
Red	2	10^2	
Orange	3	10^3	
Yellow	4	10^4	
Green	5	10^5	
Blue	6	10^6	
Violet	7	10^7	
Grey	8	10^8	
White	9	10^9	
Gold	0.1	10^{-1}	5
Silver	0.01	10^{-2}	10
None			20

The digit column gives the first two digits in the 4-band code and first 3 digits in the 5-band code. Gold and silver appear in the 3rd digit place in the 5-band code. The multiplier column gives the values, which have to be multiplied to the digit value attained above. The colors gold and silver give the tolerances in the 4 band codes. The colors brown, red, green, blue, and violet are used as tolerance codes on 5-band resistors only. All 5-band resistors use a colored tolerance band. The blank (20%) Band is only used with the "4-band" code (3 colored bands + a blank "band").

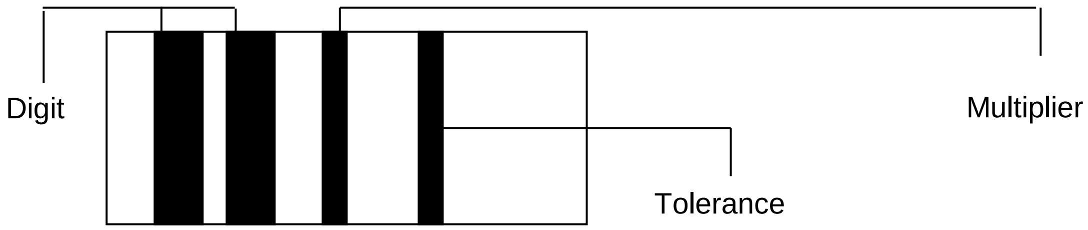


Figure 1.1: 4-Band Code

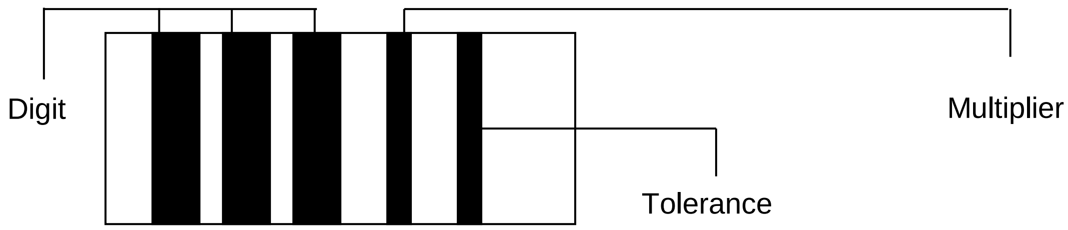


Figure 1.2: 5-Band Code

Part 3- Introduction to Simulation Tool

In Pspice the program we run in order to draw circuit schematics is called CAPTURE. The program that will let us run simulations and see graphic results is called PSPICE. You can run simulation from the program where your schematic is. There are a lot of things we can do with Pspice, but the most important things for you to learn are:

1. Design and draw circuits
2. Simulate circuits
3. Analyze simulation results (Probe for older versions)

For this course you will not need the full capacity of CAPTURE. The devices that we will use are resistors, inductors, capacitors and various independent/dependent sources. It is good to know that CAPTURE has extensive symbol libraries and includes a fully integrated symbol editor for creating your own symbols or modifying existing symbols.

The main tasks in CAPTURE are:

1. Creating and editing designs
2. Creating and editing symbols
3. Creating and editing hierarchical designs
4. Preparing your design for simulation

In this part we will create a simple DC circuit just to let you know how to start working with Pspice. Your goal is to find the current value in the resistor labeled R1.

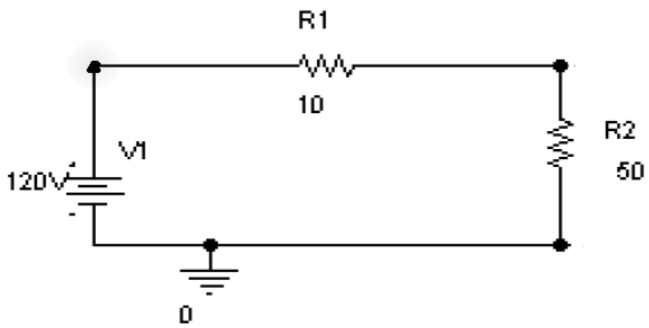


Figure 1.3: Basic DC circuit

In Lab Section

Lab Task 1: Identification of resistors using color code and DMM

1. Calculate the resistance of the resistors available by using color codes.
2. Switch the selector of multi-meter to resistance measurement.
3. Select the minimum range and measure resistance by connecting both probes of meter to the ends of resistor. Be sure, not to touch both ends of the resistor while the probes of meter are connected to the resistor. This will make a resistive path through your body in parallel with the resistor, thus affecting the correct reading.
4. If the reading of the multi-meter is pegged, i.e. displays **"1"** on the left hand side of LCD display, this means the resistance is greater than the selected range.
5. Switch to the next available range and measure the resistance again.
6. Repeat step 5 till a reading is available on the display.
7. Check whether the calculated and measured values are comparable.
8. Repeat the procedure for different resistors available.
9. Fill out the table 1.2.

Table 1.2

S. No.	**Resistance Value (Ω)**		**Difference (Ω)**
	Calculated (min-max) Range	**Measured**	
1.			
2.			
3.			

Lab Task 2: Circuit implementation using hardware and software

Hardware Task:

1. Select the any two resistors from In-lab task 1 and place it on bread board to make the circuit shown in Fig 1.3.
2. Energies the circuit with a fix +12 DC voltage and ground.
3. Now connect a DMM to measure the voltage across R_1 and R_2. Also measure the value of fix +12 DC power supply.

4. Note the values in the table given below:

Table 1.3

Measured Values	V_1	V_{R_1}	V_{R_2}
Hardware			
Software			

Software Task:

Proceed as follows to obtain the answer using Pspice.

1. Run the CAPTURE program.
2. Select File/New/Project from the File menu.
3. On the New Project window select Analog or Mixed A/D, and give a name to your project then click OK.
4. The Create Pspice Project window will pop up, select Create a blank project, and then click OK.
5. Now you will be in the schematic environment where you are to build your circuit.
6. Select Place/Part from the Place menu.
7. Click ANALOG from the box called Libraries, then look for the part called R. You can do it either by scrolling down on the Part List: box or by typing R on the Part box. Then click OK.
8. Use the mouse to place the resistor where you want and then click to leave the resistor there. You can continue placing as many resistors as you need and once you have finished placing the resistors right-click your mouse and select end mode.
9. To rotate the components there are two options:
10. Rotate a component once it is placed: Select the component by clicking on it then Ctrl-R
11. Rotate the component before it is placed: Just Ctrl-R.
12. Select Place/Part from the Place menu.
13. Click SOURCE from the box called Libraries, then look for the part called V_{DC}. You can do it either by scrolling down on the Part List: box or by typing V_{DC} on the Part box, and then click OK. Place the Source.
14. Repeat steps 10 - 12 to get and place a current source named I_{DC}.
15. Select Place/Wire and start wiring the circuit. To start a wire, click on the component terminal where you want it to begin, and then click on the component terminal where you want it to finish. You can continue placing wires until all components are wired. Then right-click and select end wire.
16. Select Place/Ground from the Place menu, click on GND/CAPSYM. Now you will see the ground symbol.
17. Type 0 on the Name: box and then click OK. Then place the ground. Wire it if necessary.

18. Now change the component values to the required ones. To do this you just need to double-click on the parameter you want to change. A window will pop up where you will be able to set a new value for that parameter.
19. Once you have finished building your circuit, you can move on to the next step – prepare it for simulation.
20. Select Pspice/New Simulation Profile and type a name, this can be the same name as your project, and click Create.
21. The Simulations Settings window will now appear. You can set up the type of analysis you want Pspice to perform. In this case it will be Bias Point. Click Apply then OK.
22. Now you are ready to simulate the circuit. Select Pspice/Run and wait until the Pspice finishes. Go back to Capture and see the voltages and currents on all the nodes.

If you are not seeing any readout of the voltages and currents, then select Pspice/Bias Point/Enable Bias Voltage Display and Pspice/Bias Point/Enable Bias Current Display. Make sure that Pspice/ Bias Point/Enable is checked

Post Lab Section

- Perform all the following circuit of Figure 1.4 on software.

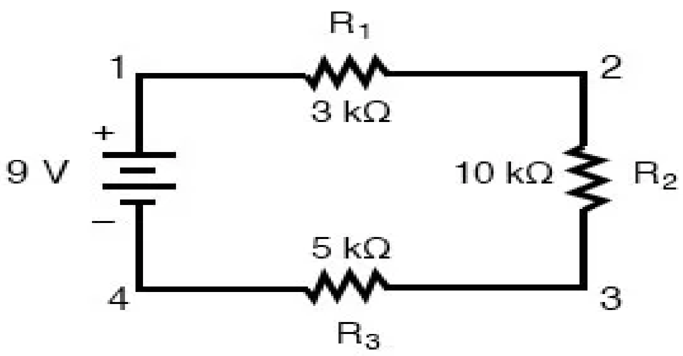


Figure 1.4

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ________________________________ Date: ______________________

LAB # 2

To explain the concept of equivalent resistance of series and parallel and combination of series-parallel circuits using hardware and software tool

Objectives

- To explain the concept of the equivalent resistances of series, parallel and combination of series-parallel circuits using hardware tool
- To explain the concept of the equivalent resistances of series, parallel and combination of series-parallel circuits using software tool

Pre Lab Section

Calculation of Equivalent Resistance Introduction

Series connection

Two resistors are said to be in series if joining of the two forms a node. The total (often referred to as "Equivalent Resistance", abbreviated as R_{eq}) in this case is the simple sum of the individual resistance. Current in series-connected elements is the same (a direct derivation from KCL). Also, equivalent resistance is equal to the voltage applied across the combination divided by the current flowing through it (R = V / I).

Parallel connection

Two or more resistors are said to be in parallel, if joining them forms a node pair. The reciprocal of equivalent resistance is the sum of reciprocals of the resistors connected in parallel. Voltage across parallel-connected elements is the same. Also, equivalent resistance is equal to the voltage applied across the combination divided by the current flowing through it (R = V / I).For two resistors in parallel, the equivalent resistance is equal to the product of the two resistor values divided by their sum.

Viewpoint

Viewpoint is the pair of access points to the circuit, where the multi-meter probes are connected to the circuit. All theoretical calculations have to be made keeping in view the selected set of points.

Pre Lab Exercise Question:

Question 1: Determine i and Vo in the circuit of Figure 2.1.

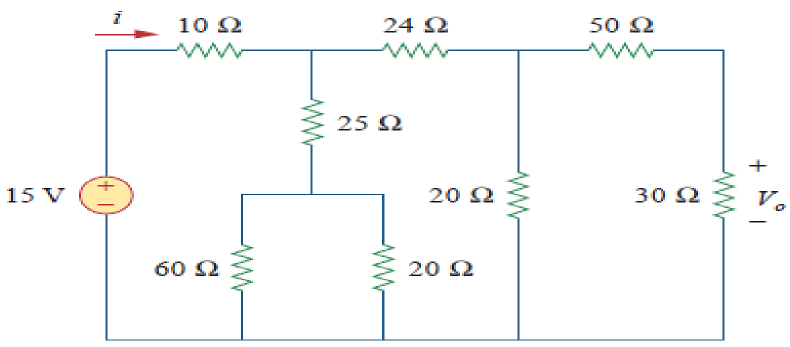


Figure 2.1

In Lab Section

Lab Task 1: Series Circuit

Procedure

1. Take three resistors and note their values after confirmation by color code and multimeter measurements.

Table 2.1

Resistor symbol	Value (Ω)		Value to be used in calculation (Ω)
	Color Code	**Multimeter**	
$\mathbf{R_1}$			
$\mathbf{R_2}$			
$\mathbf{R_3}$			

2. Construct a series circuit of these resistors as shown below.

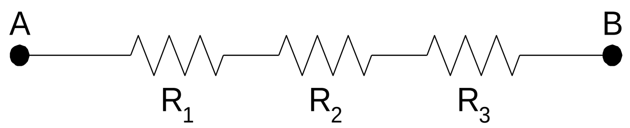


Figure 2.2: Resistors connected in series

3. Calculate the equivalent resistance using formula
4. Check the resistance of the series combination by connecting the multimeter probes at points A and B.
5. Connect a measured voltage between points A and B and note the current flowing through the circuit (For this the multimeter has to be connected in series with the circuit, with selector pointed at mA and

black probe plugged into mA socket of the meter). The total resistance is calculated using the relationship **R = V / I**. The circuit diagram is depicted below:

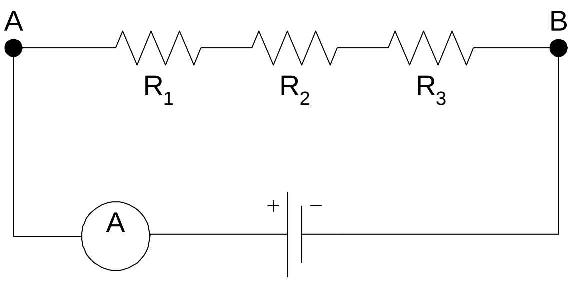


Figure 2.3: Series circuit

Table 2.2

Sr. No.	Total Resistance Value (Ω)		Applied Voltage (V)	Current (A)	Resistance value (V/I) (Ω)
	Calculated using formula	Measured with multimeter			
1.					

Lab task 2: Parallel Circuit

Procedure

1. Repeat step 1 of Lab Task 1.
2. Construct a parallel circuit of these resistors as shown below:

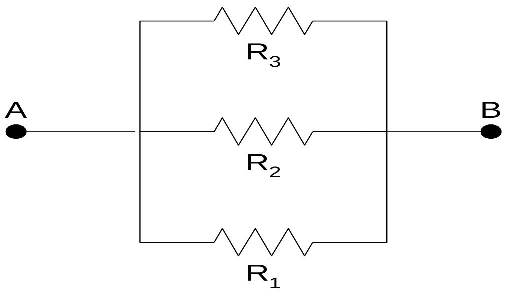


Figure 2.4: Resistors connected in parallel

3. Calculate the equivalent resistance using formula
4. Check the resistance of the parallel combination by connecting the multimeter probes at A and B.
5. Connect a measured voltage between points A and B and note the current flowing through the circuit (For this the multimeter has to be connected in series with this parallel circuit, with selector pointed at mA and black probe plugged into mA socket of the meter). The total resistance is calculated using the relationship **R = V / I**. The circuit diagram is depicted below:

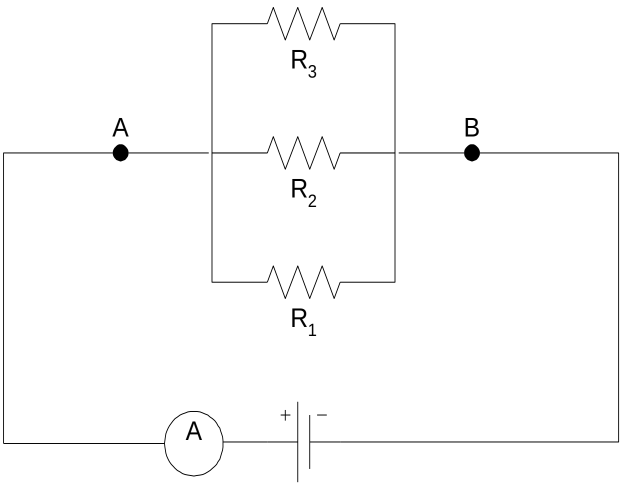


Figure 2.5: Parallel circuit

6. Fill out the following table.

Table 2.3

S. No.	Total Resistance Value (Ω)		Applied Voltage (V)	Current (A)	Resistance value (V/I) (Ω)
	Calculated using formula	Measured with multimeter			
1.					

Lab task 3: Series Parallel Circuit

Procedure

1. Repeat step 1 of Lab Task 1.
2. Construct a series parallel circuit of these resistors as shown below:

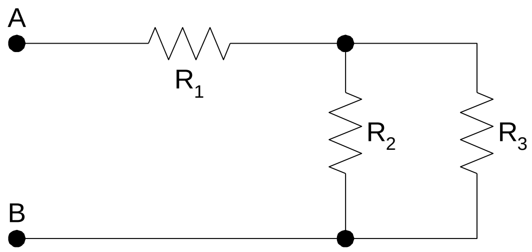


Figure 2.6: Combination of series and parallel resistances

3. Calculate the equivalent resistance using formula.

4. Check the resistance of the series parallel combination by connecting the multimeter probes at A and B.
5. Connect a measured voltage between points A and B and note the current flowing through the circuit (For this the multimeter has to be connected in series with this series-parallel circuit, with selector pointed at mA and black probe plugged into mA socket of the meter).
6. The total resistance is calculated using the relationship **R = V / I**. The circuit diagram is depicted below:

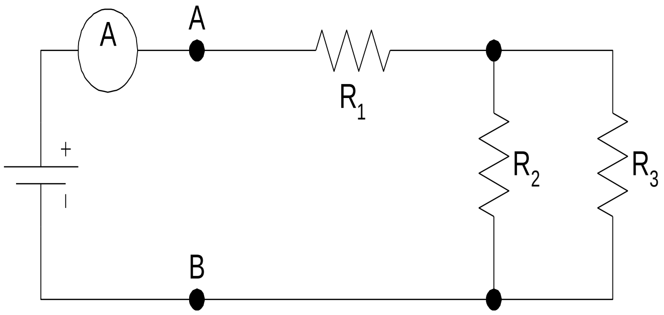


Figure 2.7: Series and parallel circuit

7. Fill out the following table.

Table 2.4

S. No.	Total Resistance Value (Ω)		Applied Voltage (V)	Current (A)	Resistance value (V/I) (Ω)
	Calculated using formula	Measured with multimeter			
1.					

Lab task 4: View point

Procedure

1. Repeat step 1 of Lab Task 1.
2. Construct a circuit of these resistors as shown below in Figure 2.8.

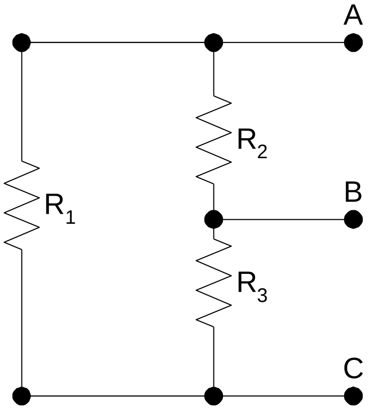
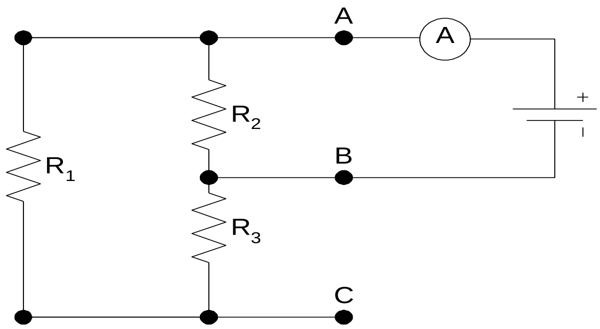
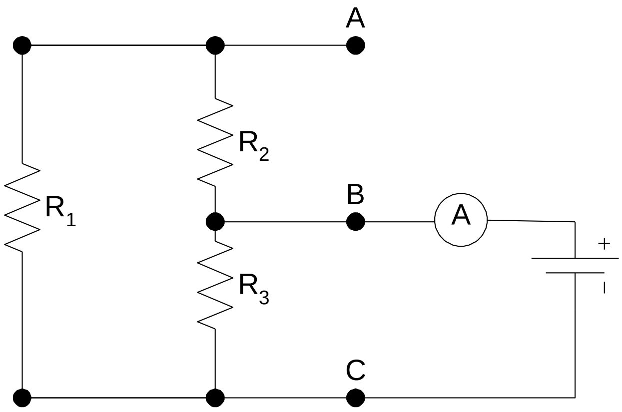


Figure 2.8: Illustration of view points

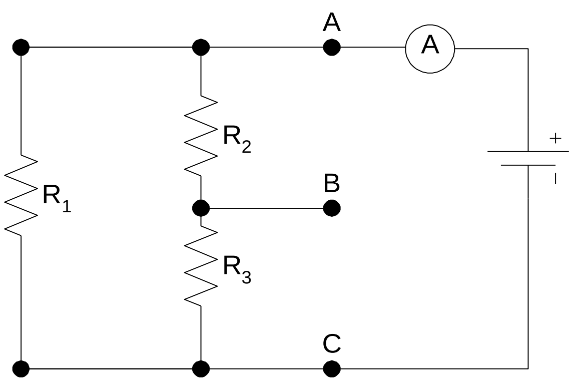
3. Calculate the resistance between points AB, BC and AC using theoretical concepts.
4. Check the resistance of the circuit by connecting the multimeter probes at A and B, then at B and C and then at A and C.
5. Connect a measured voltage between points A and B and note the current flowing through the circuit (For this the multimeter has to be connected in series with the circuit, with selector pointed at mA and black probe plugged into mA socket of the meter). The total resistance is calculated using the relationship **R = V / I**. The different circuit configurations are depicted below:



(a)



(b)



(c)

Figure 2.9: Circuit configurations for measuring the values of (a) AB, (b) BC and (c) AC resistances

6. Fill out the following table

Table 2.5

Sr. No.	**Total Resistance Value (Ω)**		**Applied Voltage (V)**	**Current (A)**	**Resistance value (V/I) (Ω)**
	Calculated using formula	**Measured with multimeter**			
AB					
BC					
AC					

Post Lab Section

- Perform all the Pre-lab and In-lab tasks on software and compare your results.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ________________________________ Date: ____________________

LAB # 3

To explain the Kirchhoff's Current Law (KCL) and display the values in KCL based resistive circuit using hardware and software tool

Objectives

- To show series and parallel resistive circuits based on KCL and display the circuit parameters using hardware tool.
- To show the KCL based resistive circuit and display currents and voltages using software tool.

Pre Lab Section

Familiarize yourself with Kirchhoff's Current Law

Introduction

Kirchhoff's Current Law (abbreviated as KCL) states that: **"The Algebraic Sum of currents at any node is zero".** Equations developed using KCL can help find out an unknown current at any node, provided all other currents associated with that node are known. In applying KCL, the referenced directions of currents have to be taken into consideration, for correct assignment of algebraic signs to the currents. This is done giving an arbitrary sign (either positive or negative) to currents coming into a node and assigning the currents leaving the same node with an opposite sign. The above means that we have to give reference directions to currents through all elements connected at any node. This is done by noting the current by connecting the multimeter as an ammeter in series with each element. The direction of current through each element is understood to be from red probe to black probe. If the reading is positive, the actual and assumed directions are the same. If the reading is negative, this means that the actual direction of flow of current is opposite to that assumed by us.

Pre Lab Exercise Question:

Question 1. For the circuit in Figure. 3.1 use KCL to find the branch currents $I1$ to $I4$.

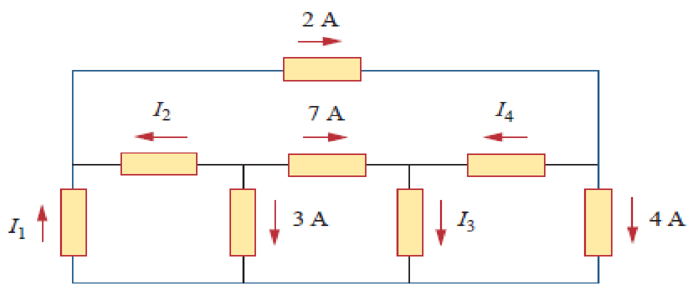


Figure 3.1

In Lab Section

Lab Task 1: Simple series circuit

Procedure

1. Take three resistors and construct a series circuit as shown below.

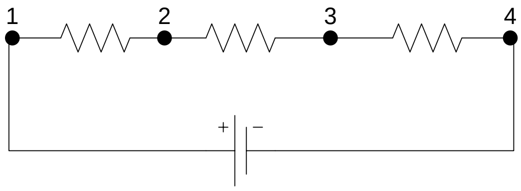


Figure 3.2: Single mesh multiple nodes circuit

2. Identify nodes and mark them. These are 1, 2, 3 and 4.
3. The currents are I_{41}, I_{12}, I_{23} and I_{34}.
4. The first digit in the subscript of each current signifies the terminal from where the current is flowing towards the terminal (signified by the second digit in subscript) through one specific circuit element.
5. The current in each element will be measured by connecting the multimeter as an ammeter in series with that particular element, with red probe nearest to the first point in the subscript and black probe nearest to the second point.
6. There are a number of KCL equations at each node, these are:

   at node 1: $-I_{41} + I_{12} = 0$

   at node 2: $-I_{12} + I_{23} = 0$

   at node 3: $-I_{23} + I_{34} = 0$

   at node 4: $-I_{34} + I_{41} = 0$

Measured Currents:

I_{41}= I_{12}= I_{23}= I_{34}=

Verification of equations:

$-I_{41} + I_{12} =$ + =

$-I_{12} + I_{23} =$ + =

$-I_{23} + I_{34} =$ + =

$-I_{34} + I_{41} =$ + =

Lab task 2: Multiple loops circuit

Procedure

1. Take four resistors and construct a multiple loops circuit as shown below:

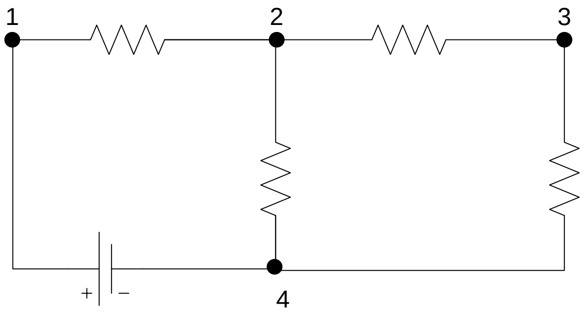


Figure 3.3: Multiple loops multiple nodes circuit

2. Identify nodes. In the given circuit, there are 4 nodes, namely 1, 2, 3 and 4.
3. The current in different branches are I_{41}, I_{12}, I_{24}, I_{23} and I_{34}.
4. Measure and note these currents. Take the sum and verify KCL for each node.

Measured Currents:

I_{41}= I_{12}= I_{24}= I_{23}= I_{34}=

Verification of equations:

$-I_{41} + I_{12} =$ + =

$-I_{12} + I_{24} + I_{23} =$ + + =

$-I_{23} + I_{34} =$ + =

$-I_{24} - I_{34} + I_{41} = -$ - + =

Lab task 3: Unknown current calculation

Procedure

1. Take four resistors and construct a circuit as shown below:

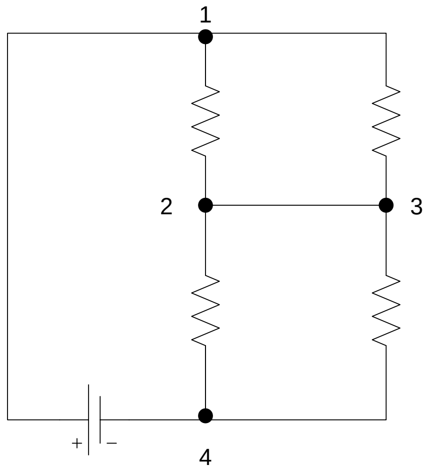


Figure 3.4: Circuit for calculating the value of I_{34}

2. The purpose is to calculate the current I_{34}.
3. The currents associated with node 2 (or 3) are I_{12}, I_{13}, I_{24} and I_{34}.
4. The application of KCL at node 2 (or 3) will give us the equation: $-I_{12} - I_{13} + I_{24} + I_{34} = 0 \Rightarrow I_{34} = I_{12} + I_{13} - I_{24}$.
5. Take measurements and verify the results.

I_{12}= I_{13}= I_{24}=

$I_{34} = I_{12} + I_{13} - I_{24} =$ + - =

Measured I_{34} =

Post Lab Section

- Perform all the Pre-lab and In-lab tasks on software and compare your results.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ________________________________ Date: ______________________

LAB # 4

To explain the Kirchhoff's Voltage Law (KVL) and display the values in KVL based resistive circuit using hardware and software tool

Objectives

- To show series and parallel resistive circuits based on KCL and display the circuit parameters using hardware tool
- To show the KCL based resistive circuit and display currents and voltages using software tool

Pre Lab Section

Familiarize yourself with Kirchhoff's Voltage Law (KVL)

Introduction

Kirchhoff's Voltage Law (commonly abbreviated as KVL) states that: **"The Algebraic Sum of voltages in a closed loop is zero".** Equations developed using KVL can help find out an unknown voltage between any two points if all other voltages in a loop are known. In applying KVL, the referenced polarities of the voltages have to be taken into consideration, for correct assignment of algebraic signs to these voltages. This is done giving an arbitrary direction to the current in the loop and then indicating these voltage drops. The terminal, through which the current enters a source, is marked negative and the terminal, through which it leaves, is marked positive. For resistors, the terminal, through which current enters the resistor, is marked positive and the terminal, through which the current leaves, is marked negative. This gives a theoretical form of the equation for KVL. In lab, the terminal at which the red probe is connected is termed positive and the terminal, at which the black probe is connected, is termed negative.

Pre Lab Exercise Question:

Question 1: Given the circuit in Figure. 4.1, use KVL to find the branch voltages V1 to V4.

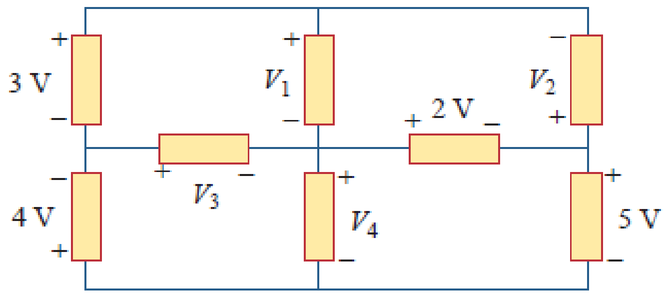


Figure 4.1

In Lab Section:

Lab Task 1: Simple Series Circuit

Procedure

1. Take three resistors and construct a series circuit as shown below.

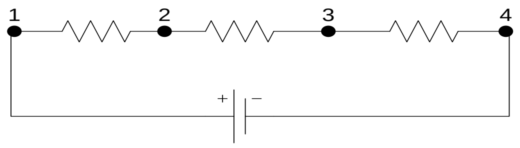


Figure 4.2: Single loop circuit

2. Identify points between which voltage can be measured.
3. In the given drawing, these are 1-2, 1-3, 1-4, 2-3, 2-4 and 3-4.
4. The measured voltages are indicated as V_{12}, V_{13}, V_{14}, V_{23}, V_{24} and V_{34}.
5. The subscript of each voltage signifies that the first digit represents the point which is assumed to be positive and the second point represents the point which is assumed negative with respect to the first point.
6. The voltages between all of these points will be measured with red probe connected to first point in the subscript and black probe connected to second point.
7. The voltages may also be measured by interchanging the probe position between any two points.
8. The voltages in this case are V_{21}, V_{31}, V_{41}, V_{32}, V_{42} and V_{43}. It will be interesting to note that $V_{21} = -V_{12}$, $V_{31} = -V_{13}$ and so on.
9. For measurement of above voltages, follow step 6.
10. Measure and note all these voltages.
11. There can be many equations for KVL within this loop, these are:

$V_{41} + V_{12} + V_{23} + V_{34} = 0$

$V_{41} + V_{12} + V_{24} = 0$

$V_{41} + V_{13} + V_{34} = 0$

$V_{41} + V_{14} = 0$

$V_{42} + V_{23} + V_{34} = 0$

$V_{42} + V_{24} = 0$

$V_{43} + V_{34} = 0$

$V_{14} + V_{43} + V_{32} + V_{21} = 0$

$V_{14} + V_{43} + V_{31} = 0$

$V_{14} + V_{42} + V_{21} = 0$

$V_{14} + V_{41} = 0$

$V_{13} + V_{32} + V_{21} = 0$

$V_{13} + V_{31} = 0$

$V_{12} + V_{21} = 0$

Verify these equations.

Measured Voltages:

V_{12}=	V_{13}=	V_{14}=	V_{23}=	V_{24}=	V_{34}=
V_{21}=	V_{31}=	V_{41}=	V_{32}=	V_{42}=	V_{43}=

Verification of equations:

$V_{41} + V_{12} + V_{23} + V_{34} =$ + + + =

$V_{41} + V_{12} + V_{24} =$ + + =

$V_{41} + V_{13} + V_{34} =$ + + =

$V_{41} + V_{14} =$ + =

$V_{42} + V_{23} + V_{34} =$ + + =

$V_{42} + V_{24} =$ + =

$V_{43} + V_{34} =$ + =

$V_{14} + V_{43} + V_{32} + V_{21} =$ + + + =

$V_{14} + V_{43} + V_{31} =$ + + =

$V_{14} + V_{42} + V_{21} =$ + + =

$V_{14} + V_{41} =$ + =

$V_{13} + V_{32} + V_{21} =$ + + =

$V_{13} + V_{31} =$ + =

$V_{12} + V_{21} =$ + =

Lab task 2: Multiple Loops Circuit

Procedure

1. Take four resistors and construct a multiple mesh loops circuit as shown below:

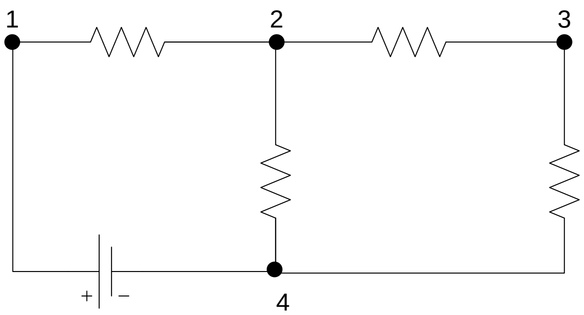


Figure 4.3: Multiple loops circuit

2. Identify closed loops. In the given circuit, there are three closed loops namely 4-1-2-4, 4-2-3-4 and 4-1-2-3-4.

3. The voltages in the first loop (while traversing the loop from point 4 in the clockwise direction, containing drops across all elements) are V_{41}, V_{12}, and V_{24}. Measure and note these values. Take the sum and verify KVL.
4. The voltages in the second loop (while traversing the loop from point 4 in the clockwise direction, containing drops across all elements) are V_{42}, V_{23}, and V_{34}. Measure and note these values. Take the sum and verify KVL.
5. The voltages in the third loop (while traversing the loop from point 4 in the clockwise direction, containing drops across all elements) are V_{41}, V_{12}, V_{23} and V_{34}. Measure and note these values. Take the sum and verify KVL.

Loop 1 (4-1-2-4)

V_{41}= V_{12}= V_{24}=

$V_{41} + V_{12} + V_{24}$=   +   +   =

Loop 2 (4-2-3-4)

V_{42}= V_{23}= V_{34}=

$V_{42} + V_{23} + V_{34}$=   +   +   =

Loop 3 (4-1-2-3-4)

V_{41}= V_{12}= V_{23}= V_{34}=

$V_{41} + V_{12} + V_{23} + V_{34}$=   +   +   +   =

Lab task 3: Unknown Voltage Calculation

Procedure

1. Take four resistors and construct a circuit as shown below:

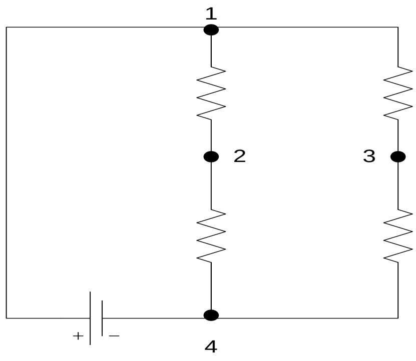


Figure 4.4: Circuit for calculating the value of V_{23}

2. The purpose is to calculate the voltage V_{23} (= $-V_{32}$).

3. If we assume that an element of infinite resistance is connected between node 2 and 3, then two loops will be formed namely, 1-3-2-1 (or 1-2-3-1) and 2-3-4-2 (or 2-4-3-2).
4. The voltages in the first loop shall be V_{13}, V_{32}, V_{21} and KVL application will give us the equation $V_{13} + V_{32} + V_{21} = 0$ from which $V_{32} = -V_{13} - V_{21}$. Measure these voltages and calculate V_{32} (= - V_{32}). Measure V_{32} and verify your calculation.
5. The voltages in the second loop shall be V_{23}, V_{34}, V_{42} and KVL application will give us the equation $V_{23} + V_{34} + V_{42} = 0$ from which $V_{23} = -V_{34} - V_{42}$. Measure these voltages and calculate V_{23} (= - V_{32}). Measure V_{23} and verify your calculation.

Loop 1 (1-3-2-1)

V_{13}= V_{21}=

$V_{32} = -V_{13} -V_{21} =$ + =

Measured V_{32} =

Loop 2 (2-3-4-2)

V_{34}= V_{42}=

$V_{23} = -V_{34} - V_{42} =$ + =

Measured V_{23} =

Post Lab Section

- Perform all the Pre-lab and In-lab tasks on software and compare your results.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ______________________________ Date: ____________________

LAB # 5

To reproduce the Wye-Delta transformation circuit using hardware and software tool

Objectives

- To reproduce the Wye-Delta transformation circuits using hardware tool
- To reproduce the Wye-Delta transformation circuits using software tool

Pre Lab Section

Familiarize with Wye-Delta connection

Introduction:

In series or parallel combination of resistors, we define connection by focusing at two resistors at a time, and then declaring the nature of connection that either it is series or parallel.

However, when we have to analyze three resistances at a time then instead of series and parallel, we define the connectivity of resistors as “star” connection or “delta” connection. Such type of connections becomes more important when we study three phase power systems. Also, these connections are applicable not only for resistor, they are defined either for individual R(resistance), XL (inductive reactance), Xc (capacitive reactance) or Z (impedance) as whole according to the nature of system under consideration.

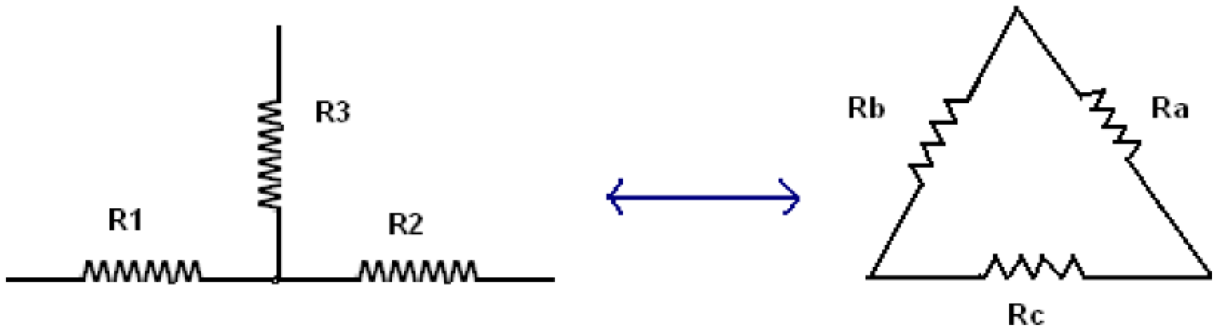


Figure 5.1:

To convert a delta in to star or vice versa we use the following conversion equations:

Delta to Star:

$$R1 = \frac{RB \times RC}{RA+RB+RC}$$

$$R2 = \frac{RA \times RC}{RA+RB+RC}$$

$$R3 = \frac{RA \times RB}{RA+RB+RC}$$

Star to Delta:

RA= R2 + R3 + $\frac{\text{R2 x R3}}{R1}$

RB=R1+R3+ $\frac{\text{R1 x R3}}{R2}$

RC=R1+R2+ $\frac{\text{R1 x R2}}{R3}$

If RA=RB=RC, then

RY=RΔ / 3

If R1= R2= R3, then

RΔ = 3 x R Y

Pre Lab Exercise Question:

Question 1: Convert the Δ network in Figure 5.2 to an equivalent Y network.

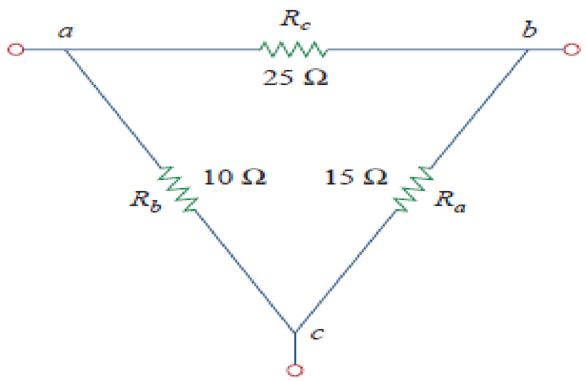


Figure 5.2

In Lab Section

Lab Task 1:

Procedure:

1. Construct the network of Figure 5.3.
2. Measure the current 'I' and voltage 'V**ab**' and record in the observation table.
3. Calculate the equivalent Y for the Δ formed by three 390 ohm resistors (using R**Y**=RΔ / 3)
4. Insert the values of resistors in the Y as shown in Figure 5.4.
5. Measure the current 'I' and voltage 'V**ab**' and record in the observation table.
6. Construct the network of Figure 5.5.
7. Measure the current 'I' and voltage 'V**ab**' and record in the observation table.
8. Calculate the power absorbed by using the formula P= I x Vs and record it in Table 5.1

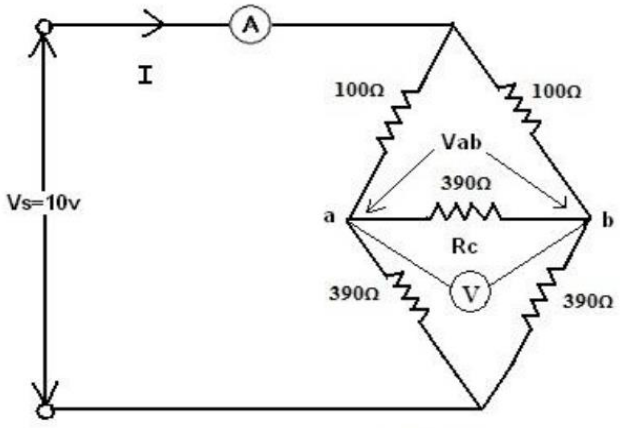


Figure 5.3

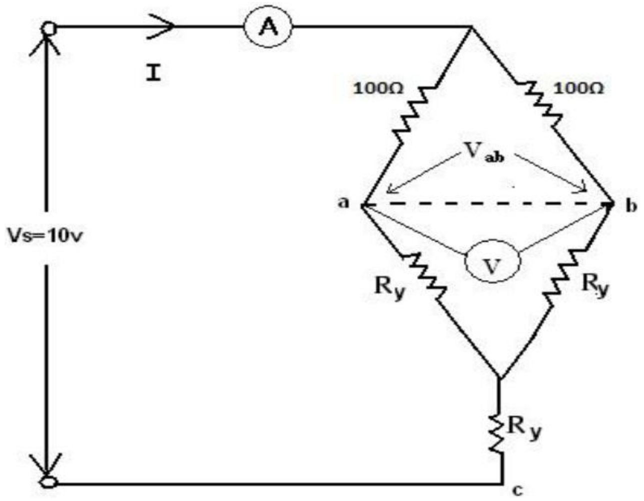


Figure 5.4

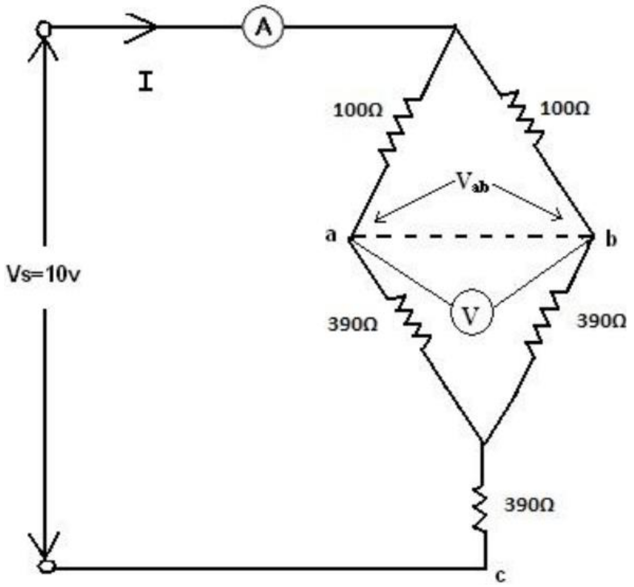


Figure 5.5

Table 5.1

	I	V_{ab}	$P = IV_S$
Figure 5.3			
Figure 5.5			

Post Lab Section

- Perform all the Pre-lab and In-lab tasks on software and compare your results.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ________________________________ Date: _____________________

LAB # 6

To reproduce the circuit based on Nodal Analysis using hardware and software tool

Objectives

- To reproduce the series and parallel resistive circuits based on Nodal Analysis and display the circuit parameters using hardware tool
- To show the Nodal based circuit and display currents and voltages using software tool

Pre Lab Section

Familiarize with Nodal Analysis as an Extension of Kirchhoff's Current Law (KCL)

Introduction

Node Voltage Analysis method is a technique of circuit analysis which helps us calculate current, voltage and power in any branch of a circuit. Circuits with lesser number of nodes with unknown voltages than the number of meshes are easier to solve with the Node Voltage Analysis as compared to the Mesh Current Analysis Technique. The first step is the identification of total nodes. Out of these nodes, principal or essential nodes are identified. Out of these, the ones with unknown voltages are designated with names. One of these nodes is designated as a reference node. The voltages of other essential nodes with unknown voltages are assigned relative to reference node. After this, node voltage equations are developed using KCL and expressing the currents at each node in terms of voltages and resistances. The process is elaborated in the figure shown below:

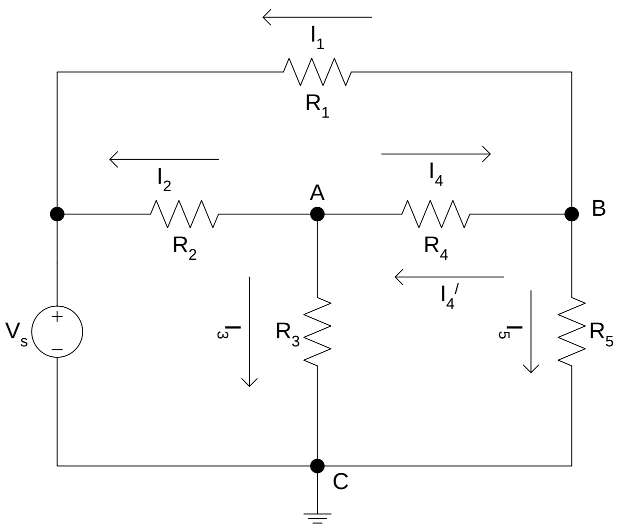


Figure 6.1: Illustration of nodal analysis

Normally, directions of currents are assumed to be away from the node. The solution of simultaneous equations developed using KCL gives the value of node voltage and these are then used to calculate the currents and voltage drops in all other branches.

Pre Lab Exercise Question:

Question 1: Using nodal analysis, find V_0in the circuit of Figure. 6.2

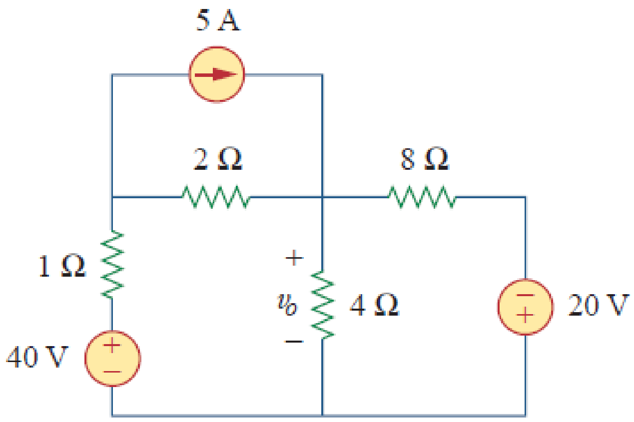


Figure 6.2

In Lab Section

Lab Task 1:

1. Measure five resistors and construct a circuit as shown in pre-lab. These are R_1, R_2, R_3, R_4 and R_5.
2. Note down the values of the resistors and the applied voltage V_s.

R_1 =	kΩ	R_2 =	kΩ
R_3 =	kΩ	R_4 =	kΩ
R_5 =	kΩ	V_S =	V

3. The identified essential nodes with unknown voltages are A, B and C.
4. Node C is taken as a reference. This means node voltage equation is to be written for only A and B.
5. Measure voltages at A and B and note them down.
6. Measure currents as indicated in the circuit diagram and note them down.

   Node A: $I_2 + I_3 + I_4 = 0 \Rightarrow$

   Node B: $I_1 + I_4^{/} + I_5 = 0 \Rightarrow$

7. Develop node voltage equations and expressing currents in the form of node voltages and resistance.
8. Solve these for unknown node voltages and compare with your results.
9. Using the node voltages, calculate the currents in each resistor and compare with current measurements.
10. Fill out the following table.

Table 6.1

Node	Measured				Calculated			
	(V)	(mA)	(mA)	(mA)	(V)	(mA)	(mA)	(mA)
A	V_A	I_2	I_3	I_4	V_A	I_2	I_3	I_4
B.	V_B	I_1	I_5	$I_4^{/}$	V_B	I_1	I_5	$I_4^{/}$

Post Lab Section

- Perform all the Pre-lab and In-lab tasks on software and compare your results.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ______________________________ Date: ____________________

LAB # 7

To reproduce the circuit based on Mesh Analysis using hardware and software tool

Objectives

- To reproduce the mesh analysis based circuit and display currents and voltages using hardware tool
- To show the mesh analysis based circuit and display currents and voltages using software tool

Pre Lab Section

Extension of Kirchhoff's Voltage Law (KVL) as Mesh Analysis

Introduction

Mesh current method is a technique of circuit analysis which helps us calculate current, voltage and power in any branch of a circuit. The first step is the identification of meshes and assigning circulating currents to these meshes. After this, mesh equations are developed using KVL and expressing voltage drops in terms of Mesh currents. The process is elaborated in the figure shown below. Normally, directions of mesh currents are assumed to be clockwise. Notice that the mesh currents are the currents flowing in those branches of each mesh which are not common to other meshes. The solution of simultaneous equations developed using mesh current analysis gives the value of mesh currents and these are then used to calculate the currents and voltage drops in all other branches.

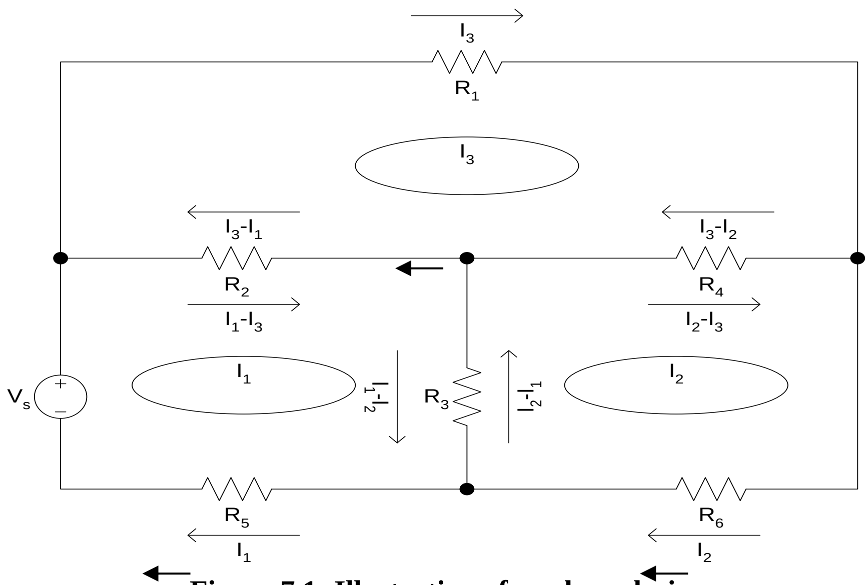


Figure 7.1: Illustration of mesh analysis

Pre Lab Exercise Question:

Question 1: For the bridge network in Figure 7.2, find i_0 using mesh analysis.

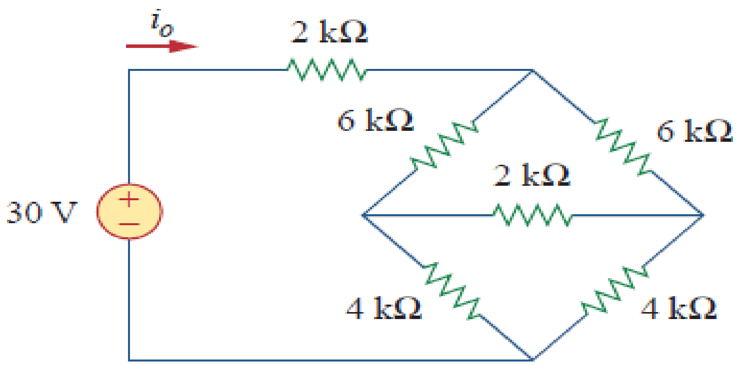


Figure 7.2

In Lab Section

Lab Task 1:

1. Measure six resistors and construct a circuit as shown in Fig. 7.1.
2. Note down the values of the resistors and the applied voltage V_s.

R_1 =	kΩ	R_2 =	kΩ
R_3 =	kΩ	R_4 =	kΩ
R_5 =	kΩ	R_6 =	kΩ
V_s =	V		

3. Measure the currents in those branches of each mesh which are not common to any other mesh. These are the mesh currents I_1 (which is the source current as well), I_2 and I_3.
4. In the given circuit, these elements are R_5, R_6 and R_1 respectively.
5. As per rules of mesh current analysis, the currents in all other branches should be equal to the difference of mesh currents depending upon commonality of those branches between respective meshes.
6. Measure all these currents with specific regard to the assumed direction.
7. Calculate the values which should be present within these common branches.
8. Complete the Table 7.1.

The measured and calculated values should tally which will verify the mesh current analysis technique.

Table 7.1

Mesh No.	**Measured Values (mA)**			**Calculations (mA)**	
1.	I_1	$I_1 - I_2$	$I_1 - I_3$	$I_1 - I_2$	$I_1 - I_3$
2.	I_2	$I_2 - I_1$	$I_2 - I_3$	$I_2 - I_1$	$I_2 - I_3$
3.	I_3	$I_3 - I_1$	$I_3 - I_2$	$I_3 - I_1$	$I_3 - I_2$

Post Lab Section

- Perform all the Pre-lab and In-lab tasks on software and compare your results.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ______________________________ Date: ____________________

LAB # 8

To reproduce the resistive circuit based on superposition theorem and display the circuit parameters using hardware and software tool

Objectives

- To reproduce series and parallel resistive circuits based on superposition theorem and display the circuit parameters using hardware tool
- To show the superposition based circuit and display currents and voltages using software tool

Pre Lab Section

Familiarize yourself with Superposition Theorem

Introduction

Superposition theorem is related to circuits with multiple sources. It states that: **"The combined effect (voltage or current) in any circuit element in a multiple source circuit is equal to the algebraic sum of individual effects of each source while others replaced with their internal impedances".** Considering the circuit shown below:

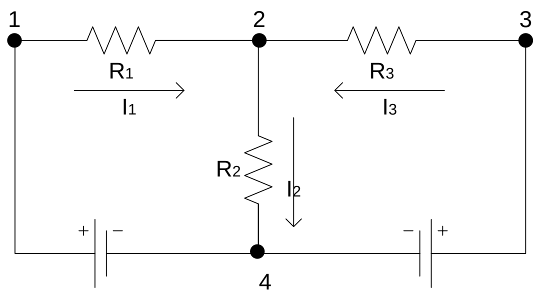
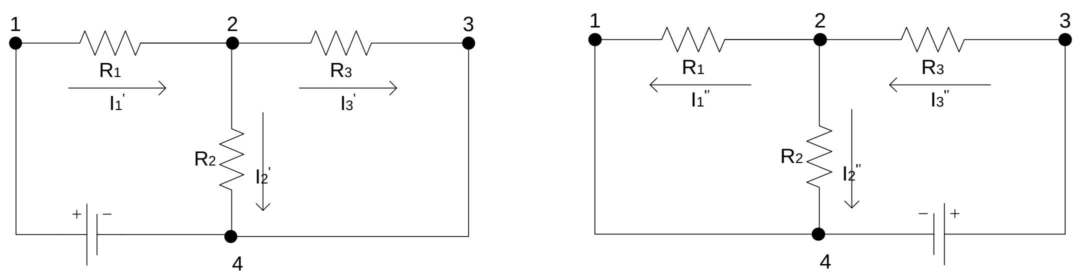


Figure 8.1: Multisource circuit

The circuit has two sources and the total current through R_1, R_2& R_3 can be considered to be I_1, I_2& I_3. Then, superposition theorem, instructs us to take the effect of each source independently and sum them up algebraically in order to get the overall effect. The two circuits with assumed directions of current in each case are shown below:



(a) **(b)**

Figure 8.2: Application of superposition principle

It is worth noting that internal resistances of both sources have been considered to be zero, which is practically not the case. However, this is quite an accurate approximation, as the electronic circuit within the breadboard (providing DC voltage) normally has negligible output impedance. As per assumed directions of currents through each element, the total currents can be calculated using the following relationships:

$$I_1 = I_1' - I_1'' \qquad I_2 = I_2' + I_2'' \qquad I_3 = I_3'' - I_3'$$

Similarly, voltage drops can be calculated using the following relationships:

$$V_{12} = V_{12}' + V_{12}'' \qquad V_{24} = V_{24}' + V_{24}'' \qquad V_{32} = V_{32}' + V_{32}''$$

Pre Lab Exercise Question:

Question 1: Use superposition to find V_0in the circuit of Figure 8.3

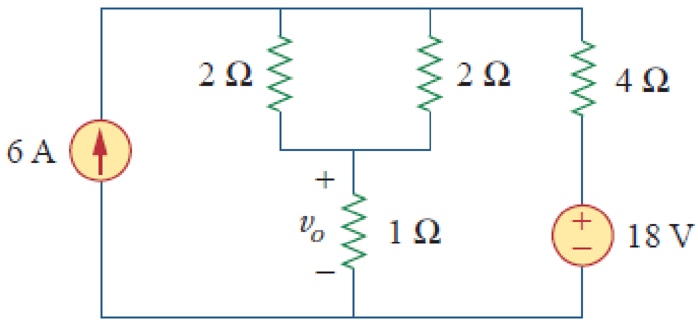


Figure 8.3

In Lab Section

Lab Task 1:

1. Construct a circuit as shown in Figure 8.1 above. Take one source voltage from variable supply and the other voltage from fixed supply of breadboard. Note these two, so that in case of these being disturbed, the same can be readjusted to the original value.
2. Measure and note the values of currents and voltages in each branch by connecting the multimeter in an appropriate manner across each element, with specific reference to assumed polarity and direction of flow of current.
3. Turn by turn; take the effect of each source by replacing the other with a short circuit and disconnecting it as shown in Figure 8.2. Repeat step 2 for each source until individual effects of all the sources have been taken into account.
4. Take algebraic sum of voltages and currents at each branch and verify superposition theorem.

Observations:

Measured values

I_1'= I_2'= I_3'=

V_{12}'= V_{24}'= V_{32}'=

I_1''= I_2''= I_3''=

V_{12}''= V_{24}''= V_{32}''=

Verification of equations

$I_1 = I_1' - I_1''$ = - =

$I_2 = I_2' + I_2''$ = + =

$I_3 = I_3'' - I_3'$ = - =

$V_{12} = V_{12}' + V_{12}''$= + =

$V_{24} = V_{24}' + V_{24}''$= + =

$V_{32} = V_{32}' + V_{32}''$= + =

Measured values

I_1 = I_2 = I_3 =

V_{12} = V_{24} = V_{32}

Post Lab Section

- Perform all the Pre-lab and In-lab tasks on software and compare your results.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ______________________________ Date: ____________________

LAB # 9

To reproduce the resistive circuit based on Thevenin's and Norton Theorems and display the circuit parameters using hardware and software tool

Objectives

- To reproduce the Thevenin, Norton equivalent circuits to display the output voltage and current with change in load using hardware tool
- To show the Thevenin, Norton equivalent circuits to display the output voltage and current with change in load using software tool

Pre Lab Section

Familiarize yourself with Thevenin's Theorem

Introduction

According to Thevenin theorem, any circuit while viewed from a specific pair of points can be reduced to a simple series circuit with a voltage source V_{Th} and a series resistance R_{Th} connected across that specific pair of points. Both the original circuit and Thevenin's Equivalent Circuit are equivalent in terms of voltage and current at the specific pair of points. The voltage source value is the open circuit voltage across the pair of points and the series resistance value is the resistance seen looking from the open circuited set of points and all voltage and current sources in the original circuit replaced with their internal resistances. The internal resistance of a voltage source is connected in series with it and that of a current source is connected in parallel with it. This scheme is adopted to indicate the loss of voltage and current with change in loading. Ideally, the internal resistance of a voltage source is 0, i.e. it can maintain the same voltage across its terminals regardless of the load being fed by it and that of a current source is∞, i.e. it can maintain a constant current through its terminals regardless of the load being fed by Lab equipment's have these source values approaching ideal, therefore, we shall be using the ideal approach while replacing sources with their internal impedances.

Familiarize yourself with Norton's Theorem

Introduction

According to Norton theorem, any circuit while viewed from a specific pair of points can be reduced to a simple circuit with a current source with current I_n and a parallel resistance R_n connected across that specific pair of points. Both the original circuit and Norton Equivalent Circuit are equivalent in terms of terminal voltage and current at the specific pair of points. The I_n value is the short circuit current through the pair of points and R_n value is the resistance seen looking from the open circuited set of points and all voltage and current sources in the original circuit replaced with their internal resistances. The internal resistance of a voltage source is connected in series with it and that of a current source is connected in parallel with it. This scheme is adopted to indicate the loss of voltage and current with change in loading. Ideally, the internal resistance of a voltage source is 0, i.e. it can maintain the same voltage across its terminals regardless of the load being fed by it and that of a current source is∞, i.e. it can maintain a constant current through its terminals regardless of the load being fed by it. Lab equipment's have these source values approaching ideal, therefore, we shall be using the ideal approach while replacing sources with their internal impedances.

Pre Lab Exercise Questions:

Question 1: Determine V_{TH} and R_{TH} at terminals 1-2 of each of the circuits in Figure 9.1.

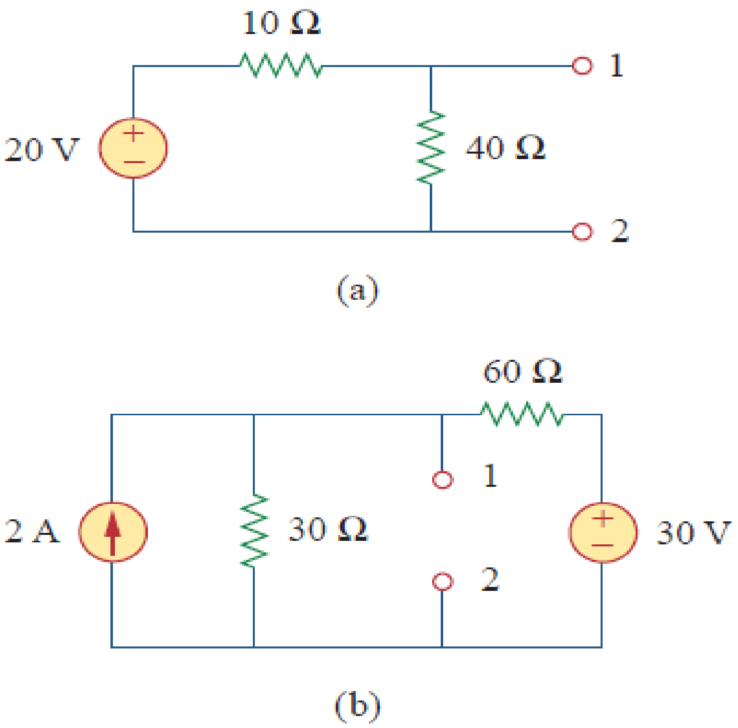


Figure 9.1

Question 2: Find the Norton equivalent of the circuit in Figure 9.2.

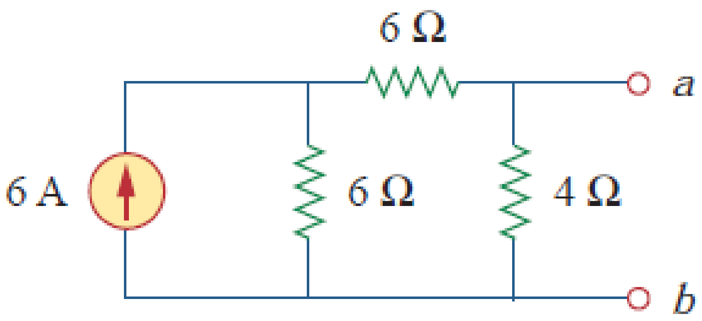


Figure 9.2

In Lab Section

Lab Task 1: Verification of Thevenin Theorem

1. Measure four resistors and construct a series circuit as shown below. A precaution for minimizing the source resistance effect is to use a high value resistor which is connected to positive battery terminal.

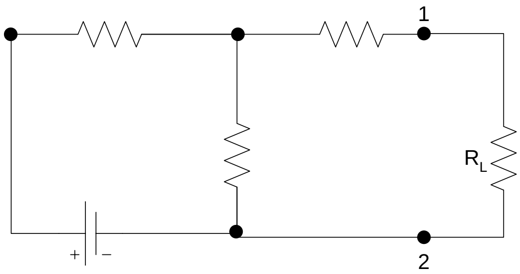


Figure 9.3: Circuit for the application of Thevenin's theorem

2. The specific set of points is 1 & 2, with respect to which, the circuit has to be Thevenized.
3. Measure and note V_{12} and I_{12} for different values of source voltage.
4. Remove resistance R_L from the circuit and measure V_{12} for different values of source voltage (used in point 3 above) as shown below. This voltage is the Thevenin voltage V_{Th}.

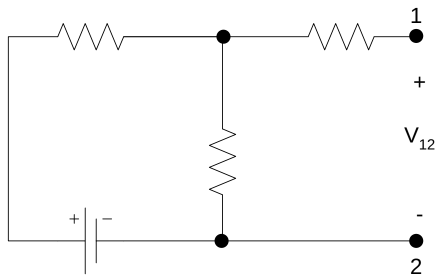


Figure 9.4: Circuit for finding the value of V_{Th}

5. Turn of source supply and place a short circuit across voltage source as shown below. Measure resistance R_{12} which is the Thevenin resistance R_{Th}.

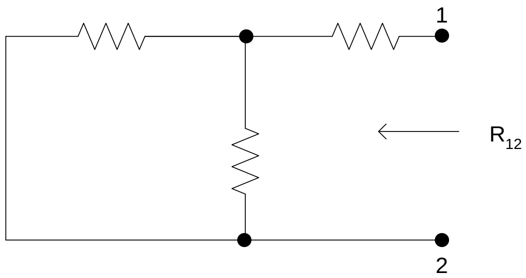


Figure 9.5: Circuit for finding the value of R_{Th}

6. Construct the Thevenin equivalent circuit and connect R_L between points 1 & 2 as shown below with source voltage adjusted to values of V_{Th} (obtained in point 4 above). R_{Th} value will be obtained using a variable resistance.

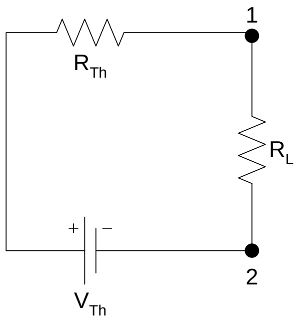


Figure 9.6: Thevenin's equivalent circuit

7. Measure V_{12} and I_{12} and compare with the results obtained in 3 above.
8. Fill out the following Table 9.1.

Table 9.1

Sr. No.	Source voltage (Volt)	Measurements in Original Circuit		Thevnin Voltage V_{Th} (Volt)	Thevenin Resistance R_{Th} (Ω)	Measurements in Thevenin Equivalent Circuit	
		V_{12} (Volt)	I_{12} (Amp.)			V_{12} (Volt)	I_{12} (Amp.)
1.							
2.							
3.							

Lab Task 2: Verification of Norton Theorem

1. Measure four resistors and construct a circuit as shown below.
2. The specific set of points is 1 & 2, with respect to which, the Norton equivalent circuit has to be drawn.

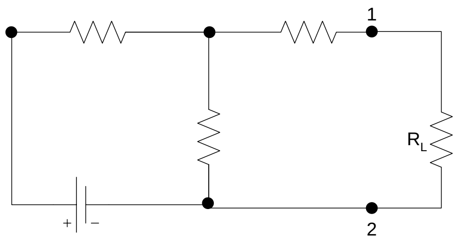


Figure 9.7: Circuit for the application of Norton's theorem

3. Measure and note source voltage, V_{12} and I_{12} for different source voltages.
4. Remove resistance R_L and replace it with a short circuit and measure current through terminals 1 and 2 for different values of source voltage (used in point 2 above) as shown below. This current is the Norton current I_n.

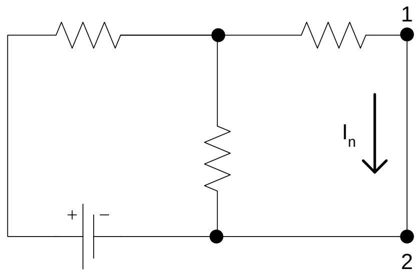


Figure 9.8: Circuit for finding the value of I_{SC}

5. Turn off supply and place a short circuit across voltage source as shown below. Measure the resistance which is Norton equivalent resistance R_n.

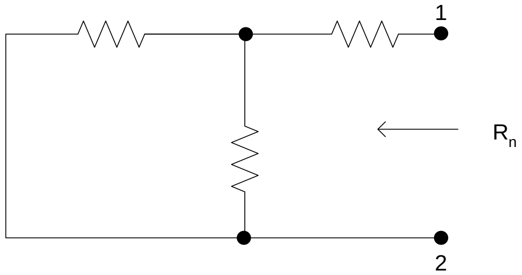


Figure 9.9: Circuit for finding the value of R_n

6. The Norton equivalent circuit is shown below:

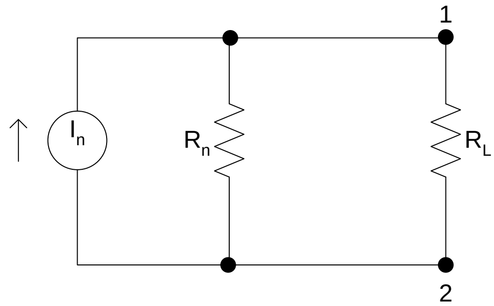


Figure 9.10: Norton's equivalent circuit

7. As a current source with a parallel resistance can be converted to an equivalent voltage source in series with a source resistance, the same circuit can be redrawn as shown below:

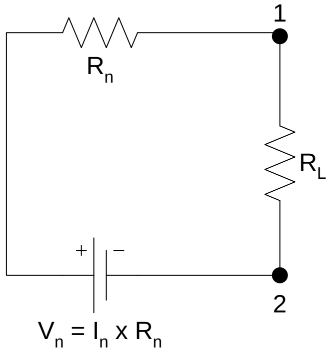


Figure 9.11: Norton's equivalent circuit (obtained through Source transformation theorem)

8. Measure V_{12} and I_{12} in the circuit drawn in point 6 above and compare with the results obtained in 2 above.
9. Fill out the following Table 9.2.

Table 9.2

Sr. No.	Source voltage (Volt)	Measurements in Original Circuit		Norton Equivalent Values			Measurements in Norton Equivalent Circuit	
		V_{12} (V)	I_{12} (mA)	I_n (mA)	R_n (kΩ)	$V_n = I_n \times R_n$ (V)	V_{12} (V)	I_{12} (mA)
1.								
2.								
3.								

Post Lab Section

- Perform all the Pre-lab and In-lab tasks on software and compare your results.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ______________________________ Date: ____________________

LAB # 10

To reproduce the resistive circuit based on Maximum Power Transfer Theorem and Source Transformation to display the circuit parameters using hardware and software tool

Objectives

- To reproduce the resistive circuit which will provide maximum power to the given load resistance using hardware tool
- To show the resistive circuit which will provide the maximum power of the given load using software tool
- To reproduce the source transformation equivalent circuit to display the circuit parameters using hardware tool
- To show the source transformation equivalent circuit to display the circuit parameters using software tool

Pre Lab Section

Maximum Power Transfer Theorem

Introduction

It has been found out that any network can be reduced to a Thevenin or a Norton Equivalent Circuit with respect to any two points of interest. The Thevenin Equivalent Circuit consists of a voltage source in series with the Thevenin resistance. The Norton Equivalent Circuit consists of a current source in parallel with a Norton resistance. According to Maximum Power Transfer Theorem, any circuit will be able to provide maximum power to a load resistance provided the value of the load resistance is equal to the Thevenin or Norton resistance. For all other values of load resistance, the power dissipation will be less than the maximum power dissipated.

Source Transformation Theorem

Introduction

Source transformation is another tool for simplifying circuits. Basic to these tools is the concept of equivalence. It is expedient in circuit analysis to be able to substitute a voltage source in series with a resistor for a current source in parallel with a resistor, or vice versa, as shown in Figure 10 1. Either substitution is known as a source transformation.

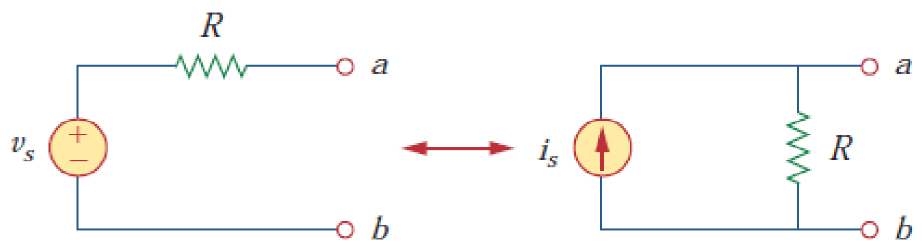


Figure 10.1

The two circuits in Figure 1 are equivalent—provided they have the same voltage-current relation at terminals a-b. It is easy to show that they are indeed equivalent. If the sources are turned off, the equivalent resistance at terminals *a-b* in both circuits is *R*. Also, when terminals a-b are short-circuited, the short circuit current flowing from a to b is $I_{SC} = \frac{V_S}{R}$ in the circuit on the left-hand side and $I_{SC} = I_S$ for the circuit on the right hand side. Thus $\frac{V_S}{R} = i_S$, in order for the two circuits to be equivalent. Hence, source transformation requires that

$$V_S = I_S \times R \qquad \text{Or} \qquad I_S = \frac{V_S}{R}$$

Source transformation also applies to dependent sources, provided we carefully handle the dependent variable. As shown in Figure 10.2, a dependent voltage source in series with a resistor can be transformed to a dependent current source in parallel with the resistor or vice versa where we make sure that Eq. 1 is satisfied.

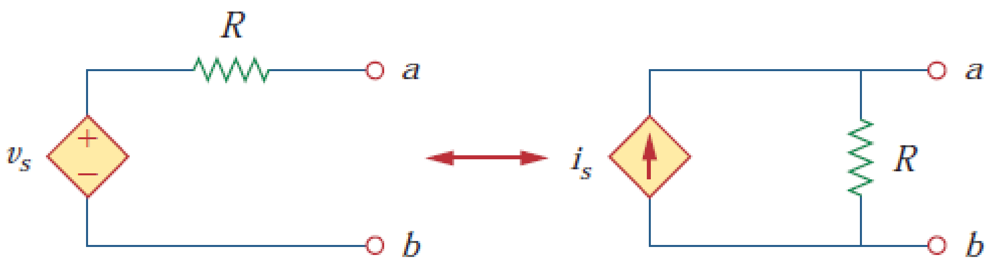


Figure 10.2

Like the wye-delta transformation, a source transformation does not affect the remaining part of the circuit. When applicable, source transformation is a powerful tool that allows circuit manipulations to ease circuit analysis.

Pre Lab Exercise Question:

Question 1: Find the maximum power that can be delivered to the resistor *R* in the circuit of Figure 10.3.

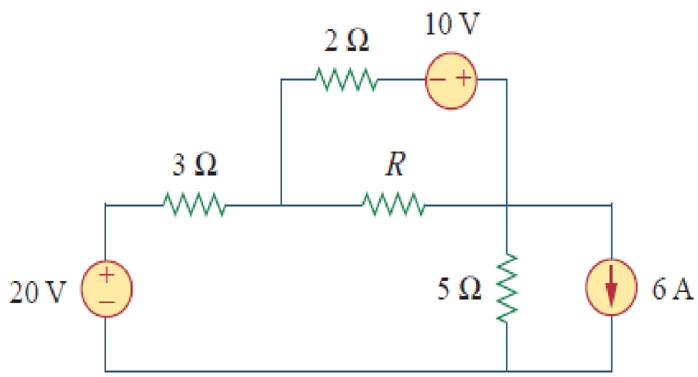


Figure 10.3

Question 2: Use Source transformation to find V_0 in the circuit of Figure 10.4.

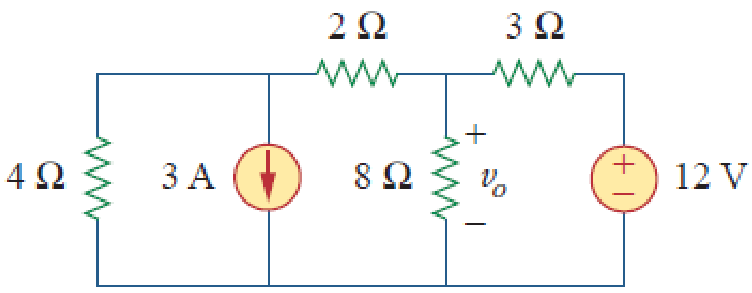


Figure: 10.4

In Lab Section

Maximum Power Transfer

Lab Task 1: Verification of maximum power transfer theorem

1. Measure the resistors and construct a circuit as shown below.

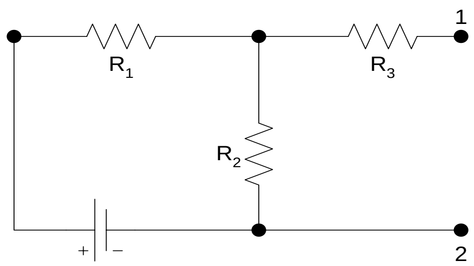


Figure 10.5: Circuit for Maximum power transfer theorem

R_1 =   **kΩ**   **R_2 =**   **kΩ**   **R_3 =**   **kΩ**

2. Maximum Power Transfer Theorem tells us that if we connect a resistance R_L between points 1 & 2, maximum power will be transferred from the network to the load resistance if R_L. = Thevenin or Norton resistance.
3. We shall use Thevenin equivalent circuit during this practical.
4. Set the source voltage at the breadboard to a maximum value (i.e. 15 volts)
5. Measure voltage between points 1 & 2 and note it down. This is the Thevenin voltage V_{Th}.

V_{source} =   **V**   **V_{Th} =**   **V**

6. Measure the Thevenin equivalent resistance R_{Th} and note it down.

R_{Th} =   **kΩ**

7. Construct the Thevenin equivalent circuit by setting the supply voltage to V_{Th} and the variable resistance at the breadboard equal to the Thevenin resistance R_{Th} and connect a variable resistance between points 1 & 2. The circuit configuration is shown below:

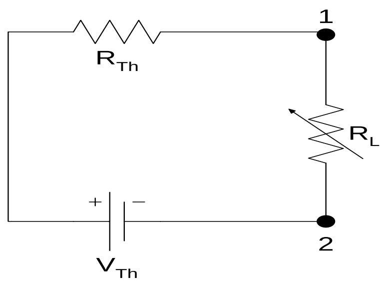


Figure 10.5: Thevenin's equivalent circuit

8. Measure V_{12} in the Thevenin equivalent circuit for different values of R_L (i.e. 0.8 R_{Th}, 0.9 R_{Th}, R_{Th}, 1.1 R_{Th}, and 1.2 R_{Th}).
9. Calculate the power delivered using formula $P_L = V_{12}^2 / R_L$.
10. Fill out the following table.

Table 10.1

Sr. No.	**Load Resistance (R_L)**		**V_{12} (V)**	**$P_L = V_{12}^2 / R_L$ (mW)**
	x R_{Th}	**Value (kΩ)**		
1.	**0.8**			
2.	**0.9**			
3.	**1.0**			
4.	**1.1**			
5.	**1.2**			

Source Transformation

1. Source transformation is a tool for simplifying circuits with multiple sources.
2. It makes use of the fact that a voltage source can be converted to a current source and vice versa.
3. Consider the circuit shown below.

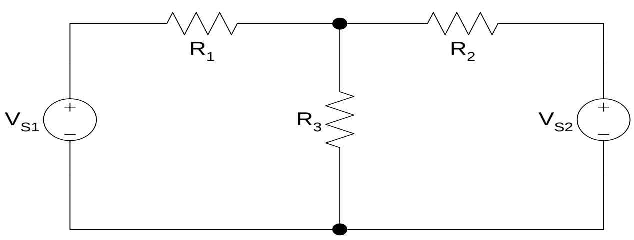


Figure 10.6: Circuit diagram for Source transformation theorem

4. If we are asked to calculate the voltage and current through R3, we can do it in many ways, but the application of Source Transformation will make the circuit very simple to solve.
5. If we convert, the voltage sources to equivalent current sources, then through conversion, the circuit configuration will become as shown below.

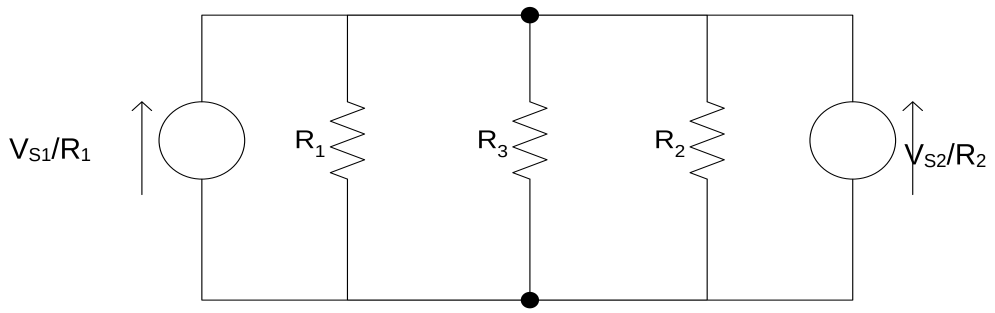


Figure 10.7: Source transformed circuit

6. Now, the two current sources can be combined to make one current source in parallel with a single element. The resulting circuit is shown below:

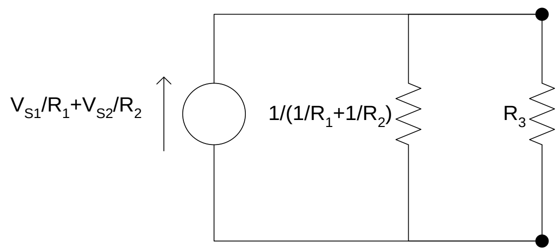


Figure 10.8: Equivalent circuit with current source

7. This single current source, when converted to a voltage source will have the configuration as shown below:

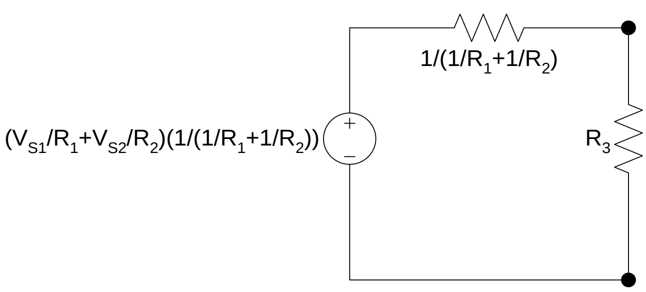


Figure 10.9: Equivalent circuit with voltage source

8. Now the circuit is a simple series circuit and current through and voltage across R_3 can be calculated easily.

Lab Task 2: Verification of Source Transformation Theorem

Procedure:

1. Measure three resistors and construct a circuit as shown in Figure 1.01. These are R_1, R_2 and R_3.
2. Note down the values of the resistors and the applied voltages $V_{S1}$and V_{S2}.
3. Measure the current though and voltage across R_3 and note them down.
4. Now construct a circuit as shown in Figure 10.6, with the values of the source voltage and resistance adjusted to the calculated values of the single source and resistance.
5. Measure and note down the values of the current through and voltage across R_3 in this circuit.
6. The results should verify the legitimacy of the Millman's Theorem.
7. Repeat the procedure for different values of R_3.

R_1 = ______________________kΩ **R_2 = _______________kΩ**

V_{S1} =______________________V **V_{S2} =_______________V**

Table 12.2

Sr. No.	**R_3**	**Original Circuit**		**Equivalent Circuit**			
		Measurements across R_3		**Equivalent Source**		**Measurements across R_3**	
		Voltage	**Current**	**Voltage**	**Resistance**	**Voltage**	**Current**
	(kΩ)	**(V)**	**(mA)**	**(V)**	**(kΩ)**	**(V)**	**(mA)**
1							
2							
3							

Post Lab Section

- Perform all the Pre-lab and In-lab tasks on software and compare your results.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ______________________________ Date: ____________________

LAB # 11

To Reproduce the RC circuit and trace the complete response of circuits using software and hardware tool

Objectives

- To reproduce the RC circuit to trace the natural and step response using hardware tool
- To reproduce the RC circuit to trace the natural and step response using software tool

Pre Lab Section

Introduction

The complete time response of a circuit comprises of two parts

a) Natural response

b) Step response

Measurement of the Natural Response of First Order RC Circuit:

Any circuit configuration that consists only of resistors and capacitors is known as an RC circuit. An example RC circuit is shown below in figure 11.1. For circuits containing more than one resistor or capacitor, the series- parallel- simplification rules outlined in the textbook can be applied to reduce the combinations to a single equivalent resistor and a single equivalent capacitor.

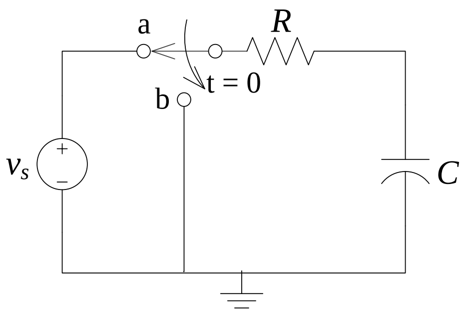


Figure 11.1: RC circuit (t < 0)

The switch is positioned at ‘a’ for a long time. The current flows through the circuit causing the capacitor to gradually charge. As the capacitor charges, it opposes the flow of current causing the current to decrease. The build-up of charge causes the voltage across the capacitor to increase while the voltage across the resistor decreases and the current decreases. All the source voltage (V_S) appears at the capacitor terminals, since a capacitor is an open-circuit to dc. Now if the switch is moved to position ‘b’ at time $t = 0$, the capacitor discharges causing current to flow in the circuit. The energy stored in the capacitor is dissipated by the heating of the resistor. The voltage\e, current, and charge dissipate exponentially in time. For $t \geq 0$, the above circuit of Figure 11.1 is reduced to:

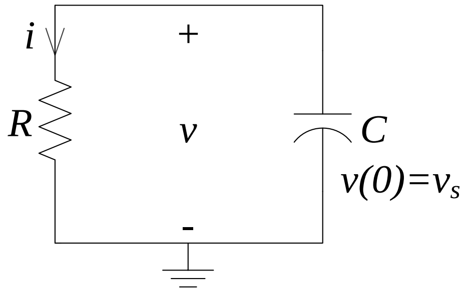


Figure 11.2: Reduced RC circuit (t ≥ 0)

Apply the node-voltage method with ground as reference node, we obtain:

$$C\frac{dv}{dt}+\frac{v}{R}=0 \tag{11.1}$$

Using elementary calculus, the expression for voltage V(t)can be derived:

$$v(t)=v(0)e^{-t/RC} \quad , \quad t\geq 0 \tag{11.2}$$

Where v (0) denotes the initial voltage on the capacitor i.e. the voltage on the capacitor before the switch is moved to position 'b'. It has been already noted, the initial voltage on the capacitor equals the voltage source voltage V_s. Equation 11.2 is termed as the natural response of an RC circuit.

In this experiment, the time constant τ for a discharging RC circuit will also be measured using an oscilloscope. The time constant τ is defined as the time required for a physical quantity to fall to $^1/_e$ i.e. 36.8% of its initial value. Time constant for an RC circuit is the product of R and C. Numerically, $^1/_e$ can be approximated, to within a 2% difference, by the fraction $^3/_8$.

$$v_C(t)=v(0)e^{-t/\tau} \rightarrow \frac{v_C(t=\tau)}{v(0)}=e^{-1}\approx\frac{3}{8} \tag{11.3}$$

$$\tau=RC \tag{11.4}$$

In other words, when a time interval equaling the time constant has passed, the voltage across the capacitor is $^3/_8$ of the initial voltage. The oscilloscope will be used to measure how long it takes for the voltage to fall to this fraction of the initial voltage. A comparison between theoretical and experimental values of the time constant will be determined after recording appropriate measurements of the analyzed circuits.

Measurement of the Step Response of First Order RC Circuit:

In an RC circuit (see Fig. 11.3) the initial voltage across the capacitor is assumed to be $V_o = 0$. The expressions for the current and voltage in the capacitor after the voltage source is applied are:

$$v(t) = V_s(1 - e^{-t/\tau}) \qquad (11.5)$$

$$i(t) = I_s\,(e^{-t/\tau}) \qquad (11.6)$$

Notice that Eq. 11.5 indicates that the voltage increases from zero to a final value of V_{sat} a rate determined by the time constant τ= RC.

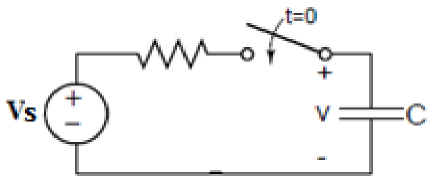


Fig. 11.3

Pre Lab Exercise Question:

Question 1: The switch in the circuit in Figure 11.4 has been closed for a long time, and it is opened at t=0. Find V (t) for t ≥ 0. Calculate the initial energy stored in the capacitor.

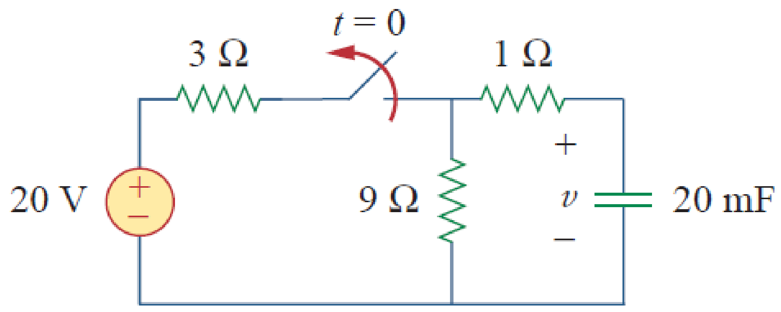


Figure 11.4

In Lab Section:

Lab Task 1:

1. Build the circuit shown in the figure 11.5.

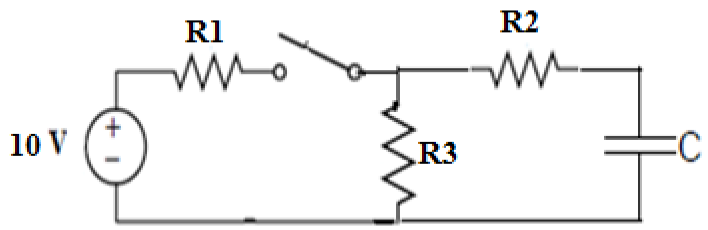


Fig. 11.5 RC Circuit

2. For t<0, the switch is closed, the capacitor is an open circuit dc. Find the capacitor voltage $V_C(0)$ by connecting the voltmeter probes across its terminals.

   Vc (0) = ------------------

3. For $t > 0$, the switch is opened and we have the source free RC circuit. Find

R_{TH} = --------------------------------

Time constant (RC) = ------------

4. Find the initial energy stored in the capacitor ($W_C(0) = \frac{1}{2} C V_{C(0)}^2$).

   $W_C(0)$ = -------------

5. Determine the voltage response V(t).

   V(t) = -------------------------------------

Calculations for Natural response of RC circuit:

Measured voltage amplitude for eight time constants in RC circuit**:**

Table 11.1

Time	**Voltage**	**Time**	**Voltage**
τ		5τ	
2τ		6τ	
3τ		8τ	
4τ		10τ	

Calculated voltage amplitude for eight time constants in RC circuit**:**

Table 11.2

Time	**Voltage**	**Time**	**Voltage**
τ		5τ	
2τ		6τ	
3τ		8τ	
4τ		10τ	

Lab Task 2:

1. Build the circuit shown in the figure 11.5.
2. For t>0 the switch is again closed and the capacitor will start charging and reaches to steady state. After that, it remains at this state unless time reaches to infinity. After a good time passed, Find

   $V(\infty)/Vs$ =------------, V(0) = ------------, τ = ---------------

3. Calculate the complete response:

   $V(t) = V(\infty) + [V(0) - V(\infty)]e^{-t/\tau}$

   When capacitor is assumed to be initially uncharged and V0 = 0 then:

   $V(t) = V(\infty)(1 - e^{-t/\tau})$

Calculations for Step response of RC circuit:

Measured voltage amplitude for eight time constants in RC circuit**:**

Table 11.3

Time	**Voltage**	**Time**	**Voltage**
τ		5τ	
2τ		6τ	
3τ		8τ	
4τ		10τ	

Calculated voltage amplitude for eight time constants in RC circuit**:**

Table 11.4

Time	**Voltage**	**Time**	**Voltage**
τ		5τ	
2τ		6τ	
3τ		8τ	
4τ		10τ	

- Trace the output waveforms for Voltage, using the Tables 11.1 & 11.3 for Natural and Step response of RC Circuit.

Post Lab Section

- Perform all the Pre-lab and In-lab tasks on software and compare your results.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ______________________________ Date: ____________________

LAB # 12

To reproduce the RL circuit and trace the complete response using hardware and software tool

Objectives

- To reproduce the RL circuit to sketch the natural and step response using hardware tool
- To reproduce the RL circuit to sketch the natural and step response using software tool

Pre Lab Section

Introduction

The complete response of a circuit comprises of two parts

a) Natural response

b) Step response

Measurement of the Natural Response of First Order RL circuit:

Any circuit configuration that consists only of resistors and inductors is known as an RL circuit. An example RL circuit is shown below in figure 12.1. For circuits containing more than one resistor or inductor, the series- parallel- simplification rules outlined in the textbook can be applied to reduce the combinations to a single equivalent resistor and a single equivalent inductor.

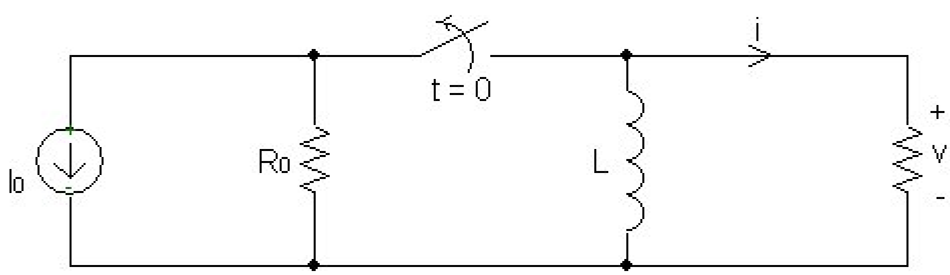


Figure 12.1 RL Circuit (t<0)

It is assumed that the switch has been closed long enough so that the inductor is fully charged. This means that all voltages and currents have reached constant values. Thus only constant (or dc.) currents can appear just prior to the switch opening and the inductor appears as a short circuit.

As the inductor appears as a short circuit there can be no current in either R_0 or R. Hence all of the source current, I_0, appears in the inductive branch and the voltage across this branch is zero. To find the natural response we need to see what happens when the source is disconnected. Hence we say when $t = 0$ the switch is opened. This then reduces the above circuit to:

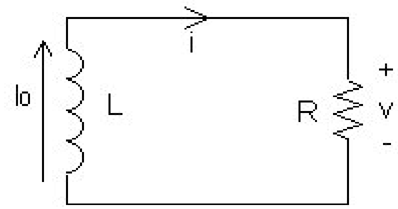


Figure 12.2 reduced RL circuit (t>0)

To find i(t) we use Kirchhoff's voltage law to obtain an expression involving i, R, and L. Summing the voltages around the closed loop gives:

$$L\frac{di}{dt} + Ri = 0 \tag{12.1}$$

This is known as a first order differential equation and can be solved by rearranging and then 'separating the variables'. This gives us:

$$\frac{di}{i} = -\frac{R}{L}dt \tag{12.2}$$

Then by integrating both the right hand side and left hand side and including a constant of integration i(0) gives:

$$\ln\frac{i(t)}{i(0)} = -\frac{R}{L}t \tag{12.3}$$

hence taking inverse logs:

if we use 0^- to represent the time just prior to switching and 0^+ just after switching. Due to the characteristics of an inductor an instantaneous change of current in an inductor is not possible, therefore the current just after switching is equal to the current just prior to switching, then this then gives us the final value for the current of:

$$i(t) = i(0)e^{-t/\tau} \tag{12.4}$$

This leads us to define the time constant for a RL circuit:

$$\tau = L/R \tag{12.5}$$

We can then derive the voltage across the resistor from a direct application of Ohm's Law:

$$V = iR = I_0 e^{-t/\tau} \quad (12.6)$$

Measurement of the Step Response of First Order RL circuit:

In an RL circuit the initial conditions to determine the step response are assumed to be *Io*=0. The expressions for the current in the circuit and the voltage across the inductor after the voltage source is applied are:

$$i\,(t) = I_S\,(1-e^{-t/\tau}) \quad (12.7)$$

$$V(t) = Vse^{-t/\tau} \quad (12.8)$$

Notice that Eq. 14.7 indicates that the current increases from zero to a final value of $V_s = R$ at a rate determined by the time constant. $^{L}/_{R}$

In Lab Section:

Lab Task 1:

1. Build the circuit shown in the figure 12.3:

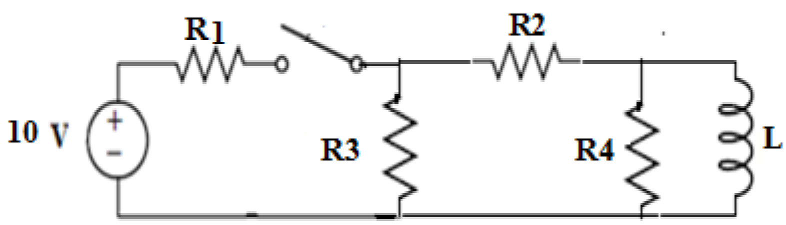


Fig. 12. 3 RL Circuit

2. For t<0, the switch is closed, the inductor is a short circuit to dc. Find the inductor current I_L (0) by connecting the ammeter probes across terminals.

   I_L (0) = -------------------

3. For t>0, the switch is opened and we have the source free RL circuit. Find

   R_{TH} = --------------------------------

   Time constant (L/R_{eq}) = ------------

4. Find the initial energy stored in the inductor ($W_L\,(0) = ½LI_L{}^2{}_{(0)}$).

   $W_L(0)$ = -------------

5. Determine the current response I(t).

   I(t) = ------------------------------

Calculations for Natural response of RL circuit:

Measured Current amplitude for eight time constants in RL circuit**:**

Table 12.1

Time	**Current**	**Time**	**Current**
τ		5τ	
2τ		6τ	
3τ		8τ	
4τ		10τ	

Calculated Current amplitude for time constants in RL circuit**:**

Table 12.2

Time	**Current**	**Time**	**Current**
τ		5τ	
2τ		6τ	
3τ		8τ	
4τ		10τ	

Lab Task 2:

1. Build the circuit shown in the figure 14.5.

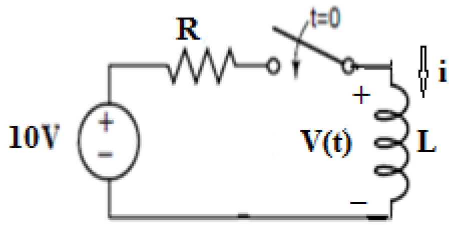


Fig. 12.5

2. For t>0 the switch is again closed and the inductor will again behave as a short circuit and reaches to steady state. After that, it remains at this state unless time reaches to infinity. After a good time passed, Find:

   $\mathrm{I}(\infty)/Is$ = ------------, I(0) = ------------, τ = ---------------

3. Calculate the complete response:

   I(t) = I(∞) + [I(0) – I(∞)$e^{-t/\tau}$]

   When I (0) = 0 initially, then:

   I(t) = I(∞) (1$-e^{-t/\tau}$)

Calculations for Step response of RL circuit:

Measured current amplitude for eight time constants in RL circuit**:**

Table 12.3

Time	Current	Time	Current
τ		5τ	
2τ		6τ	
3τ		8τ	
4τ		10τ	

Calculated current amplitude for eight time constants in RL circuit**:**

Table 12.4

Time	**Current**	**Time**	**Current**
τ		5τ	
2τ		6τ	
3τ		8τ	
4τ		10τ	

- Sketch the output waveforms for Current, using the Tables 12.1 & 12.3 for Natural and Step response of RL Circuit.

Post Lab Section

- Perform all the Pre-lab and In-lab tasks on software and compare your results.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ______________________________ Date: ____________________