Electric Circuit Analysis I

EEE-121

Lab Manual

Name	
RegistrationNumber	
Class	
Instructor’sName	

Introduction

This is the Lab Manual for EEE – 121 Electric Circuit Analysis I. The labs constitute 25 % of the total marks for this course.

During the labs you will work in groups (no more than three students per group). You are required to complete the ‘Pre-Lab’ section of the lab before coming to the lab. You will be graded for this and the ‘In-Lab’ tasks during the in-lab viva. You will complete the ‘Post-Lab’ section of each lab before coming to the next week’s lab.

You are not allowed to wander in the lab or consult other groups when performing experiments. Similarly the lab reports must contain original efforts. CIIT has a zero tolerance anti-plagiarism policy. You are not allowed to copy anything from Lab manual or internet, you should draw your own circuit diagram using LTSPICE. Take snapshot of your physical circuit using camera and explain the theory in your own words.

Apart from these weekly labs you will complete two projects. One mini-project that will count towards your Lab Sessional II score and a Final Project which will be graded as Lab Final Exam. The grading policy is already discussed in the Course Description File.

Acknowledgement

The labs for EEE-121Electrical Circuit Analysis I were designed by the team of Dr. Bilal Ijaz and Mr. Adeel Iqbal. The manuals were prepared by Mr. Adeel Iqbal, Mr.Imtiaz Ur Rehman Mr. Mirza Osama and Ms. Nida Asim. The first version was completed in Session Spring 2016, The second version was completed during the summer break of 2016. Typesetting and formatting of this version was supervised by Dr. Omar Ahmad and was carried out by Mr. Abdul Rehman, Mr Suleman & Mr Baqir Hussain. Two open ended labs are introduced by Ms Sadaf Iqbal in Fall Session 2018

History of Revision

Version and Date of Issue	Team	Comments
Version 1. May 2016	Dr. Bilal Ijaz Mr. Adeel Iqbal Mr. Imtiaz Ur Rehman Mr. Mirza Osama Ms. Nida Asim	This is the first OBE based draft of EEE–121 lab manual. Major formatting changes were performed. Few labs are upgraded. All labs are transformed into OBE based Labs. For comments and suggestions please contact: adeeliqbal@comsats.edu.pk
Version 2. September 2018	Ms Sadaf Iqbal	Two open ended Labs are introduced

Safety Precautions

- Be calm and relaxed, while working in lab.
- First check your measuring equipment.
- When working with voltages over 40 V or current over 10 A, there must be at least two people in the lab at all time.
- Keep the work area neat and clean.
- Be sure about the locations of fire extinguishers and first aid kit.
- No loose wires or metals pieces should be lying on the table or neat the circuit.
- Avoid using long wires, that may get in your way while making adjustments or changing leads.
- Be aware of bracelets, rings, and metal watch bands (if you are wearing any of them). Do not wear them near an energized circuit.
- When working with energize circuit use only one hand while keeping rest of your body away from conducting surfaces.
- Always check your circuit connections before power it ON.
- Always connect connection from load to power supply.
- Never use any faulty or damage equipment and tools.
- If an individual comes in contact with a live electrical conductor.
  - Do not touch the equipment, the cord, the person.
  - Disconnect the power source from the circuit breaker and pull out the plug using insulated material.

Table of Contents

Lab 01: Part 01: Introduction to lab Instruments

Objective:

To get a comprehensive understanding of various laboratory instruments

Equipment:

DC Power Supply, Digital Multi-meter and Bread-board

General Laboratory Operating Procedures:

Listed below are the operating procedures that you are expected to follow in the laboratory.

- Please treat the instruments with care as they are very expensive.
- Read the laboratory documentation prior to each lab meeting.
- Follow the instructions carefully.
- Return the components to the correct bin when you are finished with them.
- Before leaving the lab place the stools under the lab bench.
- Before leaving the lab, turn off the power to all instruments.

Figure 1

Pre-Lab Tasks:

DC Power Supply:

DC Power Supply is used to generate either a constant voltage or a constant current. That is, it may be used as either a DC voltage source or a DC current source. You will be primarily as a voltage source. Recall that DC is an acronym for direct current. DC means constant with respect to time.

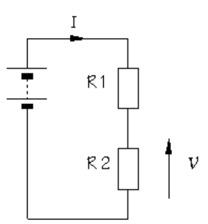
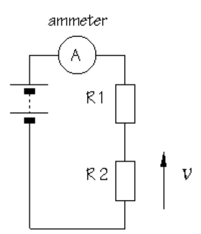
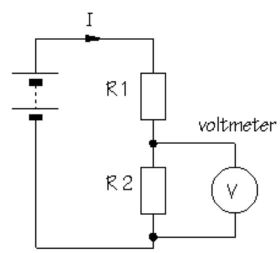
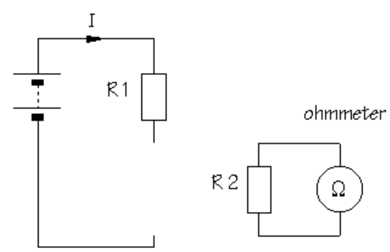
DC power supply has digital display including 0-30vD C and 0-3A. Adjustable voltage (coarse and fine) and current limiting (2 range meter). The voltage produced by the power supply is controlled by the knob labeled voltage. The current is limited by adjusting the knob labeled current. As long as the circuit does not attempt to draw more current than the value set by the current knob, the voltage will remain constant. Current limiting can prevent damage to equipment and parts which may be unable to handle excessive currents

Multimeter

What do meter measure?

A meter is a measuring instrument. An **ammeter** measures current, a **voltmeter** measures the potential difference (voltage) between two points, and an **ohmmeter** measures resistance. A **multi-meter** combines these functions and possibly some additional ones as well, into a single instrument.

Before going in to detail about multi-meters, it is important for you to have a clear idea of how meters are connected into circuits. Figures *A* and *B* below show a circuit before and after connecting an ammeter:

Figure A	Figure B	Figure C	Figure D
			

- to measure current, the circuit must be broken to allow the ammeter to be connected in series
- ammeters must have a LOW resistance

Think about the changes you would have to make to a practical circuit in order to include the ammeter. To start with, you need to break the circuit so that the ammeter can be connected in series. All the current flowing in the circuit must pass through the ammeter. Meters are not supposed to alter the behavior of the circuit, or at least not significantly, and it follows that an ammeter must have a very LOW resistance.

Figure C shows the same circuit after connecting a voltmeter:

- to measure potential difference (voltage), the circuit is not changed:the voltmeter is connected in parallel
- voltmeters must have a HIGH resistance

This time, you do not need to break the circuit. The voltmeter is connected in parallel between the two points where the measurement is to be made. Since the voltmeter provides a parallel pathway, it should take as little current as possible. In other words, a voltmeter should have a very HIGH resistance.

Figure 2

Which measurement technique do you think will be the more useful? In fact, voltage measurements are used much more often than current measurements.

The processing of electronic signals is usually thought of in voltage terms. It is an added advantage that a voltage measurement is easier to make. The original circuit does not need to be changed. Often, the meter probes are connected simply by touching them to the points of interest.

An ohmmeter does not function with a circuit connected to a power supply. If you want to measure the resistance of a particular component, you must take it out of the circuit altogether and test it separately, as shown in figure D

- To measure resistance, the component must be removed from the circuit altogether
- Ohmmeters work by passing a current through the component being tested

Ohmmeters work by passing a small current through the component and measuring the voltage produced. If you try this with the component connected into a circuit with a power supply, the most likely result is that the meter will be damaged. Most multi-meters have a fuse to help protect against misuse.

Digital multimeter:

Multimeters are designed and mass produced for electronics engineers. Even the simplest and cheapest types may include features which you are not likely to use. Digital meters give an output in numbers, usually on a liquid crystal display. The diagram below shows a digital multimeter:

Parts of a Digital-Multimeter(DMM):

A multimeter has three parts:

- Display
- Selection Knob
- Ports

The **display** usually has four digits and the ability to display a negative sign. A few multimeters have illuminated displays for better viewing in low light situations.

The **selection knob** allows the user to set the multimeter to read different things such as current in milliamperes (mA),microamperes (uA)& amperes(A) , voltages in millivolts (mv),microvolts(uv)& volts (V) and resistance in ohms(Ω), kiloohms (kΩ) & megaohms(MΩ).

Two probes are plugged into two of the **ports** on the front of the unit. **COM** stands for common and is almost always connected to Ground or '-' of a circuit. The **COM** probe is conventionally black but there is no difference between the red probe and black probe other than color. **10A** is the special port used when measuring large currents (greater than 200mA). **mAVΩ** is the port that the red probe is conventionally plugged in to. This port allows the measurement of current (up to 200mA), voltage (V), and resistance (Ω).

Measuring Voltage:

To start, let's measure voltage on a AA battery: Plug the black probe into **COM** and the red probe into **mAVΩ**. Set the multimeter to "2V" in the DC (direct current) range.Connect the black probe to the battery's ground or '-' and the red probe to power or '+'. Squeeze the probes with a little pressure against the positive and negative terminals of the AA battery. If you've got a fresh battery, you should see around 1.5V on the display (this battery is brand new, so its voltage is slightly higher than 1.5V).

If you're measuring DC voltage (such as a battery) you want to set the knob where the V has a straight line. AC voltage like what comes out of the wall can be dangerous, so we rarely need to use the AC voltage setting (the V with a wavy line next to it). If you're messing with AC,

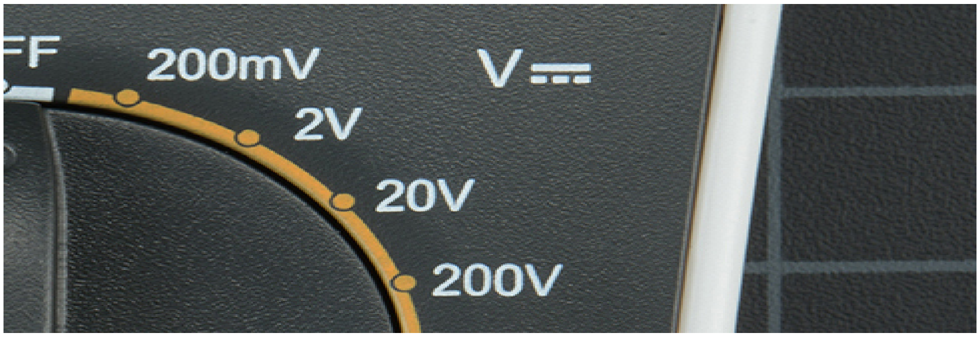


Figure 3:**Use the V with a straight line to measure DC Voltage**

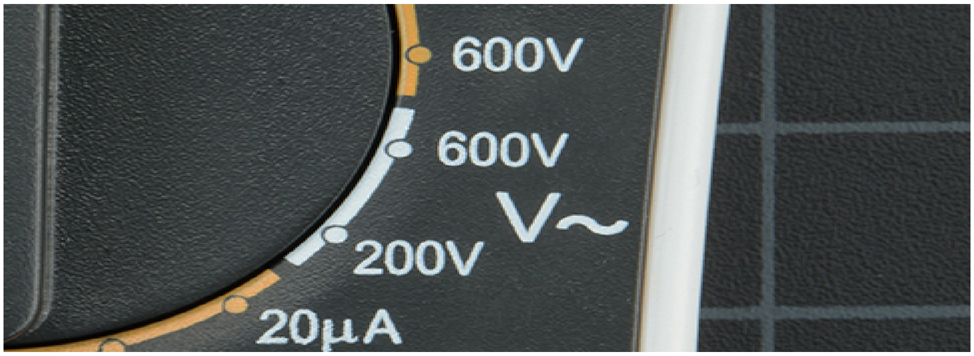


Figure 4: Use the V with a wavy line to measure AC Voltage

What happens if you switch the red and black probes? The reading on the multimeter is simply negative. Nothing bad happens!

Multimeters are generally not auto-ranging. You have to set the multimeter to a range that it can measure. For example, **2V** measures voltages **up to 2 volts**, and **20V** measures voltages **up to 20 volts**. So if you've measuring a 12V battery, use the 20V setting. If you set it incorrectly, you will probably see the meter screen change and then read '1'.

Now let's construct a simple circuit to demonstrate how to measure voltage in a real world scenario. The circuit is simply a 1kΩ and LED powered with a Power Supply .Let's measure the whole circuit first. Measuring from where the voltage is going in to the resistor and then where ground is on the LED, we should see the full voltage of the circuit, expected to be around 5V.

Figure 5

We can then see how much voltage the LED is using. This is what is referred to as the **voltage drop** across the LED

Figure 6: This LED is using 2.66V of the available 5V supply to illuminate

Overload:

What happens if you select a voltage setting that is too low for the voltage you're trying to measure? Nothing bad. The meter will simply display a 1. This is the meter trying to tell you that it is overloaded or out-of-range. Whatever you're trying to read is too much for that particular setting. Try changing the multimeter knob to the next highest setting.

Figure 7: Reading the 5V across this circuit is too much for the 2V setting on the multimeter

Measuring Resistance:

Normal resistors have color codes on them. If you don't know what they mean, that's ok! a multimeter is very handy at measuring resistance. Pick out a random resistor and set the multimeter to the 20kΩ setting. Then hold the probes against the resistor legs with the same amount of pressure you when pressing a key on a keyboard.

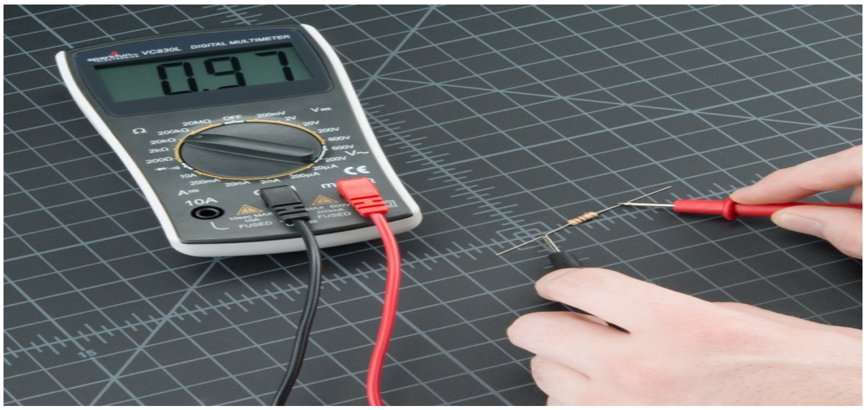


Figure 8

The meter will read one of three things, **0.00**, **1**, or the **actual resistor value**.

- In this case, the meter reads 0.97, meaning this resistor has a value of 0.97kΩ, or about 1kΩ or 1000 Ω (remember you are in the 20kΩ or 20,000 Ohm mode so you need to move the decimal three places to the right or 9,900 Ohms).
- If the multimeter **reads 1** or displays **OL**, it's overloaded. You will need to try a higher mode such as **200kΩ**mode or **2MΩ** (mega ohm) mode. There is no harm if this happen, it simply means the range knob needs to be adjusted.
- If the multimeter reads **0.00** or nearly zero, then you need to lower the mode to **2kΩ** or **200Ω**.

Measuring Current:

Reading current is one of the trickiest and most insightful readings in the world of embedded electronics. It's tricky because you have to measure current in series. Where voltage is measure by poking at +ive & -ive (in parallel), to measure current you have to physically interrupt the flow of current and put the meter in-line. To demonstrate this, we'll use the same circuit we used in the measuring voltage section.

The first thing we'll need is an extra piece of wire. As mentioned, we'll need to physically interrupt the circuit to measure the current. Said another way, pull out the +ive wire going to the resistor, add a wire where that wire was connected, and then probe from the power pin on the power supply to the resistor. This effectively “breaks” power to the circuit. We then insert the multimeter in-line so that it can measure the current as it “flows” through the multimeter into the bread board.

We used alligator clips. When measuring current, it’s often good to watch what your system does over time, for a few seconds or minutes. While you might want to stand there and hold the probes to the system, sometimes it’s easier to free up your hands. These alligator clip probes can come in handy.

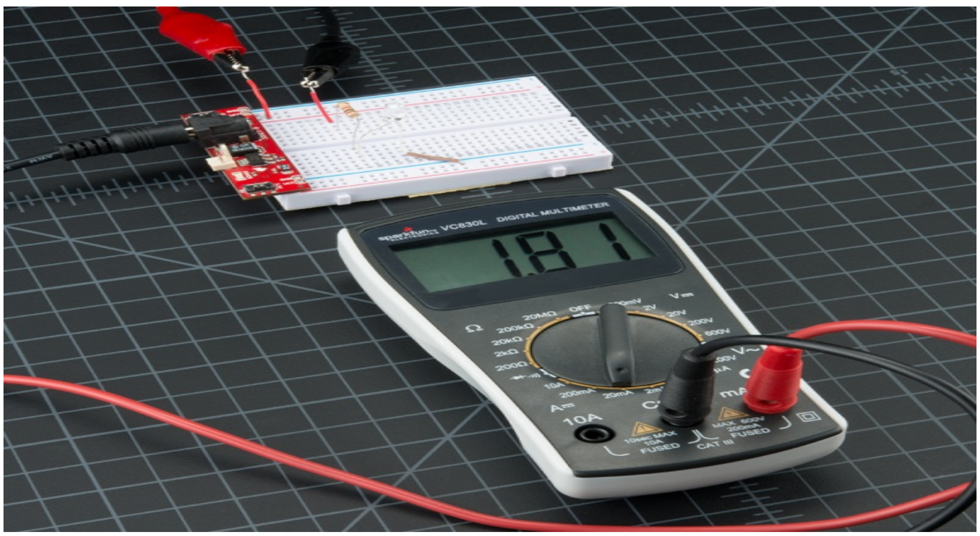


Figure 9

This circuit was only pulling 1.8mA at the time of measurement.Similar to the other measurements, when measuring current, the color of the probes does not matter. What happens if we switch probes? Nothing bad happens! It simply causes the current reading to become negative.Measuring current can be tricky the first couple of times.

Remember! When you're done using the meter, always return the meter to read voltage (return the probes to the voltage port, set the meter to read the DC voltage range if necessary). It's common to grab a meter and begin to quickly measure the voltage between two pins. If you have left your meter in 'current' mode, you won't see the voltage on the display. Instead you'll see '0.000' indicating that there is no current between +ive and -ive. Within that split second you will have connected +ive to -ive through your meter and the 200mA fuse will blow which is not good.

Continuity:

Continuity testing is the act of testing the resistance between two points. If there is very low resistance (less than a few Ωs), the two points are connected electrically, and a tone is emitted. If there is more than a few Ωs of resistance, than the circuit is open, and no tone is emitted. This test helps insure that connections are made correctly between two points. This test also helps us detect if two points are connected that should not be.

Continuity is quite possibly the single most important function for embedded hardware gurus. This feature allows us to test for conductivity of materials and to trace where electrical connections have been made or not made.

Set the multimeter to 'Continuity' mode. It may vary among DMMs, but look for a diode symbol with propagation waves around it (like sound coming from a speaker).

Figure 10: Multimeter is set to continuity mode.

Now touch the probes together. The multimeter should emit a tone (Note: Not all multimeter have a continuity setting, but most should). This shows that a very small amount of current is allowed to flow without resistance (or atleast a very small resistance) between probes.

Warning! In general, turn OFF the system before checking for continuity.

Changing the Fuse:

One of the most common mistakes with a new multimeter is to measure current on a bread board by probing from +ve to -ve (bad!). This will immediately short power to ground through the multimeter causing the bread board power supply to burn out. As the current rushes through the multimeter, the internal fuse will heat up and then burn out as 200mA flows through it. It will happen in a split second and without any real audible or physical indication that something is wrong.

Remember that measuring current is done in series. If you try to measure the current with a blown fuse, you'll probably notice that the meter reads '0.00' and that the system doesn't turn on like it should when you attach the multimeter. This is because the internal fuse is broken and acts as a broken wire or open.

Breadboard:

Breadboards are one of the most fundamental pieces when learning how to build circuits. you will learn how to use one. Once you are done you should have a basic understanding of how breadboards work and be able to build a basic circuit on a breadboard.

To understand this, first you have to understand how a breadboard's holes are electrically connected. You can see the breadboard in the figure 1 given below on which circuits are build. two rows of blocks A,D&G of breadboard are internally & separately horizontally connected from 0 to 30 after that there is no horizontal connection i.e. there is a gap after that, than again from 31 to 60 these 2 rows are internally & separately horizontally connected as show by arrows in blocks A,D&G of figure 1,2&3 below. There is no internal connection between these two rows These two rows are mostly used for power connections when making circuit on breadboard. Blocks B,C, E&F are internally vertical connected as shown in figure 1,2&3.These blocks are mostly used to place components in order to complete the circuit while making it on breadboard.Remember that there is no internal connection between Blocks A,B,C,D,E,F&G.

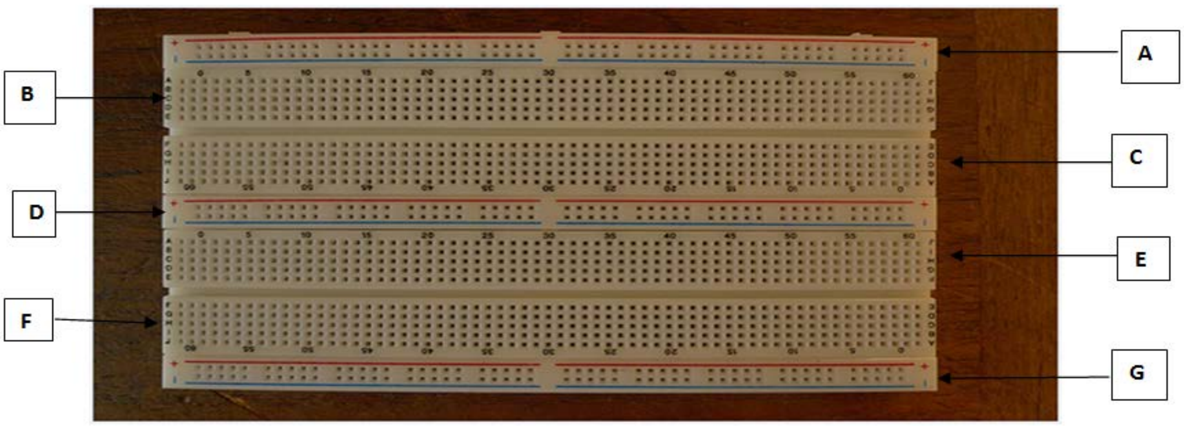


Figure 11

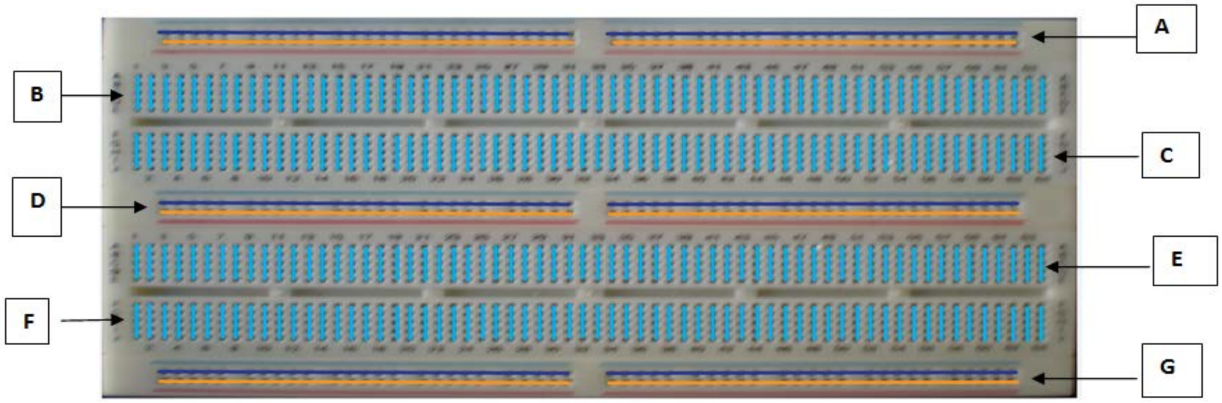


Figure 12

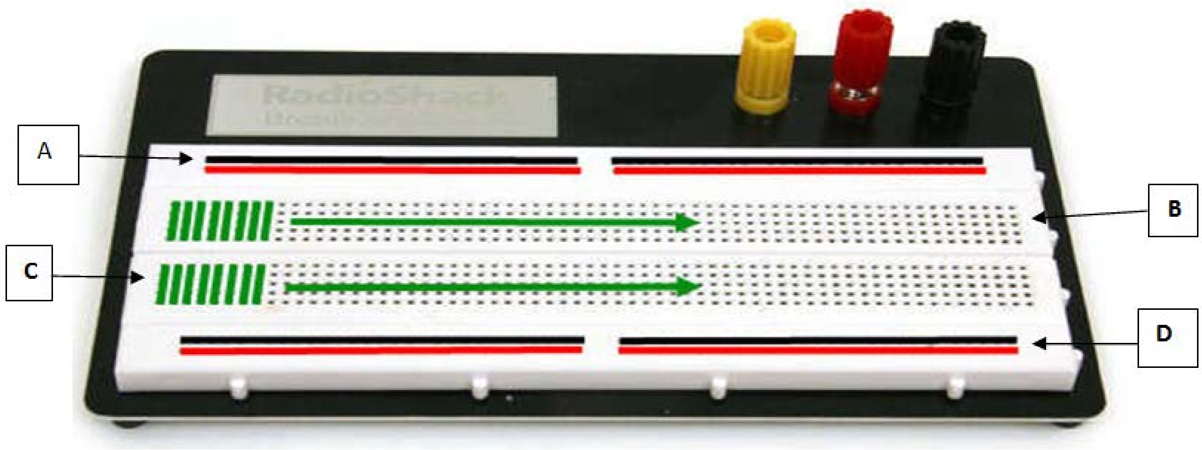


Figure 13

There are different ways to change the physical layout of a circuit on a breadboard without actually changing the electrical connections. For example, two circuits in figure 4&5 below are electrically identical; even though the leads of the LED have moved, there is still a complete path (called a **closed circuit**) for electricity to flow through the LED.

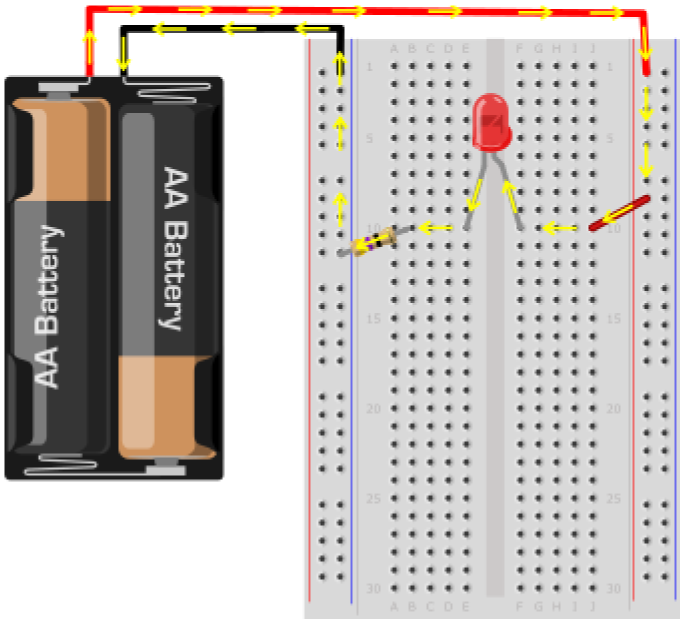


Figure 14

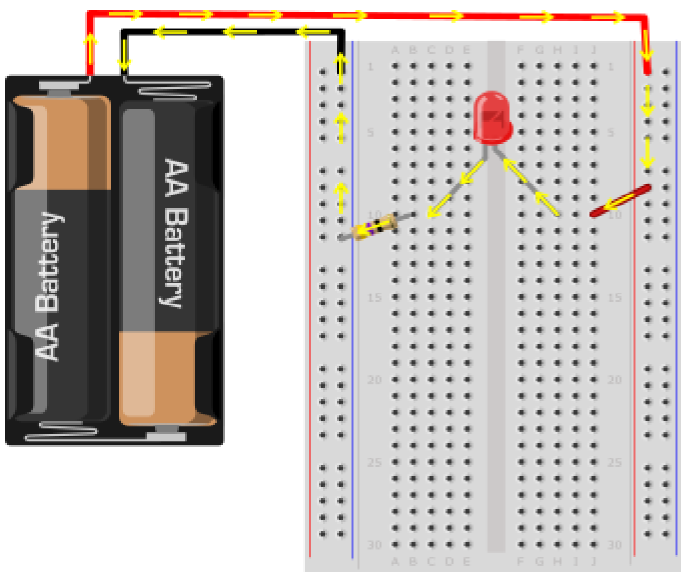


Figure 15

However, you can also completely rearrange the components on the breadboard. As long as the circuit is electrically equivalent, it will still work. Even though the circuit in figure 6 looks different than the previous two because the components have been rearranged, electricity still follows an equivalent path through the LED and the resistor.

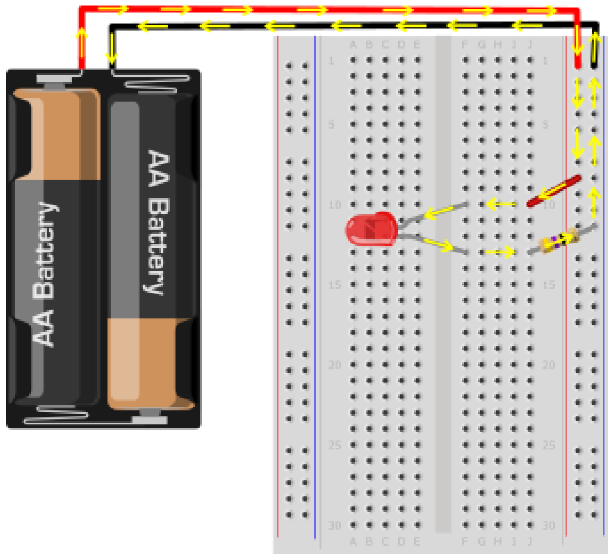


Figure 16

Lab 01: Part 02: Identification of resistor color codes:

In-Lab Tasks:

Objectives:

- To find the value of a resistor and its tolerance by color coding.
- To measure the value of the resistor by Digital Multi-meter (DMM).

Theory:

Resistor values are marked onto the body of the resistor using a series of colored bands. These

Figure 1: Four band resistor color

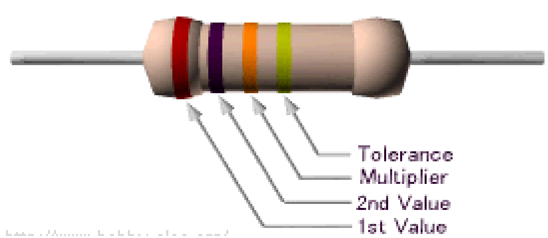
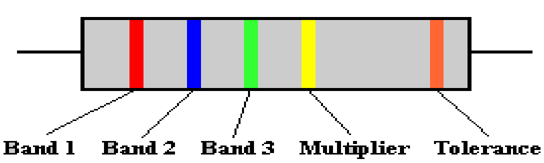


Figure 2: Five band resistor



4-band Resistor Color Codes:

	Band 1	Band 2	Multiplier
Black	0	0	1
Brown	1	1	10
Red	2	2	100
Orange	3	3	1000
Yellow	4	4	10000
Green	5	5	100000
Blue	6	6	1000000
Violet	7	7	
Grey	8	8	
White	9	9	

5-band Resistor Color Codes:

Color	Band 1	Band 2	Band 3	Multiplier
Black	0	0	0	1
Brown	1	1	1	10
Red	2	2	2	100
Orange	3	3	3	1000
Yellow	4	4	4	10000
Green	5	5	5	100000
Blue	6	6	6	1000000
Violet	7	7	7	10^{-1}
Grey	8	8	8	10^{-2}
White	9	9	9	

Color:	None	Silver	Gold	Red	Brown
Tolerance:	20%	10%	5%	2%	1%

give the value of the resistor as well as other information including the tolerance and sometimes the temperature coefficient. The band closest to the end of the resistor body is taken to be Band 1.In case of four color bands, the first two bands are the significant figures of the value, the third band is a multiplier (number of zeros) and fourth band is the tolerance band, i.e. red black brown gold would be 2 0 x 10 + 5% or 200+5% ohms. In case of five color bands first three bands are the significant figures of the value, the fourth band is a multiplier and fifth band is the tolerance band. In case of six color bands the sixth band is the temperature coefficient band. Following tables give the values allocated to different colors to find the value of a resistor through the color bands it has on it.

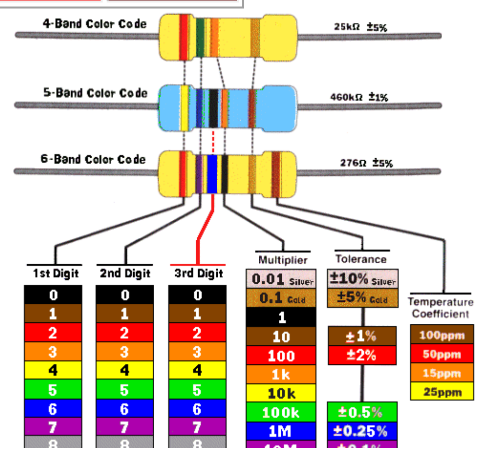


Figure 3: How to read the resistor values through color codes

The resistance value of the resistor is not the only thing to consider when selecting a resistor for use in a circuit. The "tolerance" and the electric power ratings of the resistor are also important. The tolerance of a resistor denotes how close it is to the actual rated resistance value. For example, a ±5% tolerance would indicate a resistor that is within ±5% of the specified resistance value. The power rating indicates how much power the resistor can safely tolerate. The maximum rated power of the resistor is specified in Watts. Power is calculated using the square of the current (I^2) x the resistance value (R) of the resistor. If the maximum rating of the resistor is exceeded, it will become extremely hot and even burn. Resistors in electronic circuits are typically rated 1/8W, 1/4W, and 1/2W. 1/8W is almost always used in signal circuit applications.

Task#1:

Select ten resistors (5% tolerance) between 1Ω and 1MΩ. Verify the value of resistance and tolerance of resistors using color codes, and complete the following table:

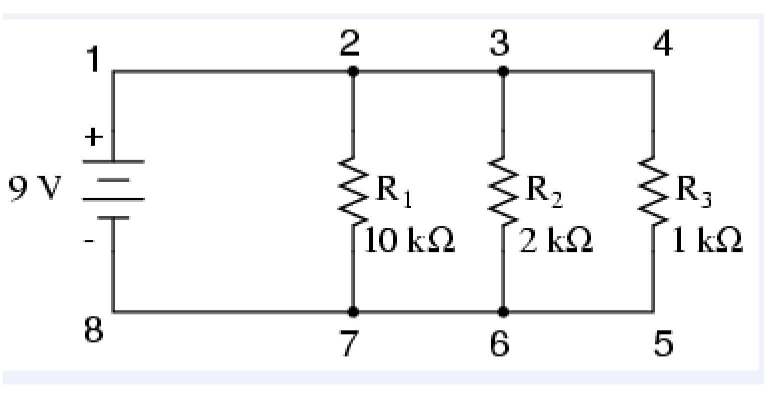
Task 02					
No.	Resistance value through color codes (Ω)	Tolerance (%)	Range (Ω)	Value of Resistance measured by the DMM (Ω)	Error (%)
1.					
2.					
3.					
4.					
5.					
6.					
7.					
8.					
9.					
10.					

Task #2:

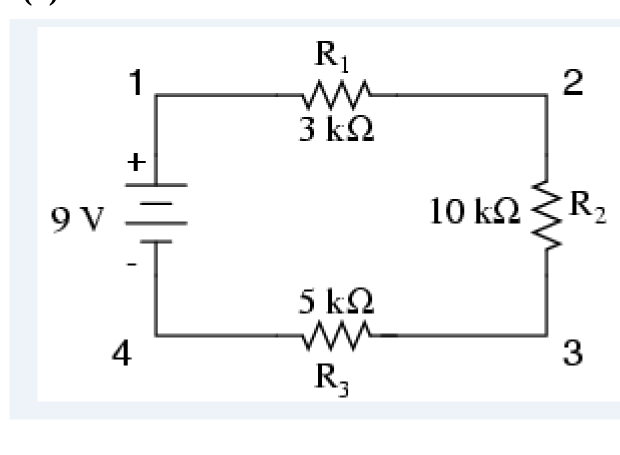
This is repetition of lab 4.

Build the circuits given below on the Breadboard and show it to the lab instructor?

(a):



(b):



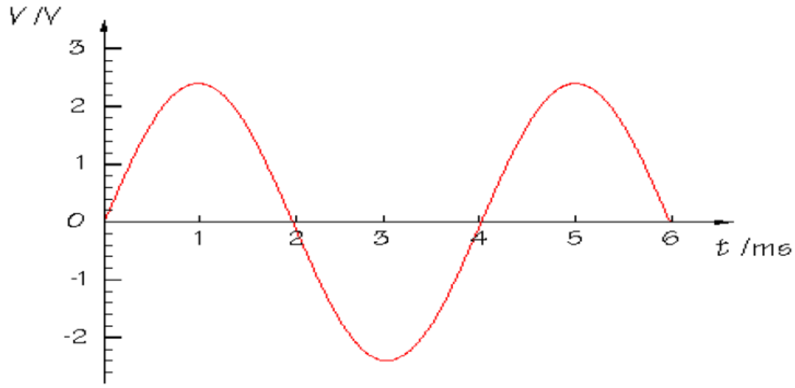
Post-Lab Task:

Answer the following Questions:functionality of oscilloscope is not mentioned in this lab

(1) Calculate the frequency of waveforms with periods of (a) 10 s, (b) 5 ms, (c) 200 μs.

(2) What is the period of waveforms with frequencies of (a) 20 Hz, (b) 150 kHz,(c) 0.5 Hz.

(3) Find values for the period, frequency, peak amplitude, peak-to-peak amplitude and rms amplitude for the sine wave shown below:



(4) What are the important features of ammeters, voltmeters and ohmmeters?

(5) What is the value of resistor having colors red, red, yellow and gold?

(6) What is the largest deviation in Table of task 2? Would it ever be possible to find a value that is outside the stated tolerance? Why or why not?

(7) Do the measured values of Table of task 2 represent the exact values of the resistors tested? Why or why not?

(8) What does "short circuit" mean?

(9) What does "open circuit" mean?

Critical Analysis / Conclusion

<table>
<tr><th colspan="3">Lab Assessment</th></tr>
<tr><td>Pre Lab</td><td>/5</td><td rowspan="5">/25</td></tr>
<tr><td>Performance</td><td>/5</td></tr>
<tr><td>Results</td><td>/5</td></tr>
<tr><td>Viva</td><td>/5</td></tr>
<tr><td>Critical Analysis</td><td>/5</td></tr>
<tr><td colspan="3">Instructor Signature and Comments</td></tr>
</table>

Lab 02: Experimental verification of ohm's law through simulation software (LTSPICE/circuit maker)

Objective

- To verify Ohm's law experimentally and to find the relationship between voltage, current, and resistance in a circuit.
- To learn how to create and simulate the circuits in LTSPICE

Equipment Required

Resistors, DMM, breadboard, DC power supply, and connecting wires.

Knowledge Level

- Before working on this lab, students should have good understanding of Ohm's Law.
- Students should be able to theoretically solve the circuit shown in circuit diagram.

Pre-Lab Task:

EXPERIMENTAL VERIFICATION OF OHM'S LAW

Introduction:

Ohm's law states that "the voltage ***v***(or potential difference) across a resistor is directly proportional to the current ***i*** flowing through it." As the current increases the voltage drop also increases provided that the resistance is kept constant; and the current decreases as the resistance increases provided that the voltage across the resistor terminals remains the same. Mathematically,

$$v \propto i$$

$$v = iR$$

Where***R*** is the constant of proportionality and is the resistance of the resistor element.

Material that obeys Ohm's Law is called **"ohmic"** or "**linear"** because the potential difference (or voltage) across its terminals varies linearly with the value of

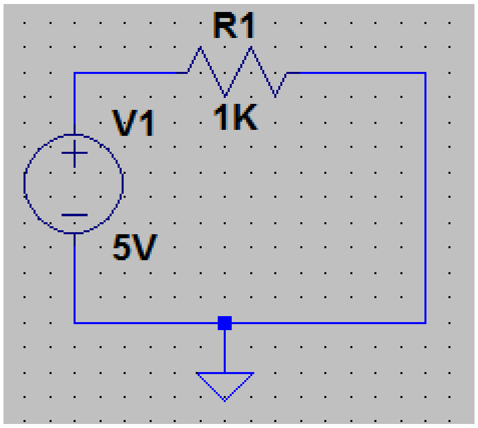


Figure 1

current flowing through it.

Task # 1:

Solve the circuit shown in figure below by hand before coming to the lab and find the Current across the Resistor by varying voltages (from 0.5V to 5V in stepwise manner) for three values of resistance R1 as show in table 1 below and record these values in table . The values of resistors should be taken as 1KΩ, 2.2KΩ and 5KΩ. (The values of resistor available in laboratory may slightly vary.

Task # 2:

INTRODUCTION TO LTSPICE:

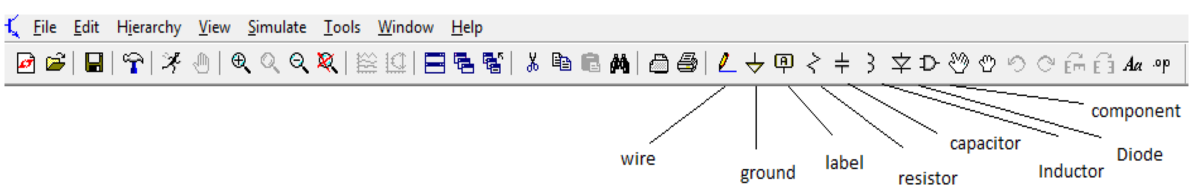
LTSPICE is an abbreviation for Linear Technology Simulation Program with Integrated Circuit Emphasis. It uses mathematical models to describe circuit elements and allows DC and time transient analysis of nonlinear circuits (transistors, diodes, capacitors, etc., also digital circuitry).

DOWNLOADINGANDINSTALLINGLTSPICE:-

LT spice can be downloaded fromhttp://www.linear.com/designtools/software/ltspice.jspThe downloaded file is a .exe file which directly installs LTspice.

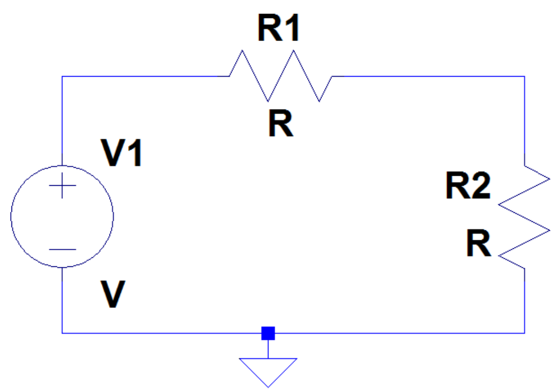
CREATINGASIMPLECIRCUIT:-

1. Open the LTspice software.
2. ChooseFile->NewSchematic.
3. FromTools menu the color preferences can be changed, thegrid can be turned on or off from the view menu.
4. The tool bar is explained below

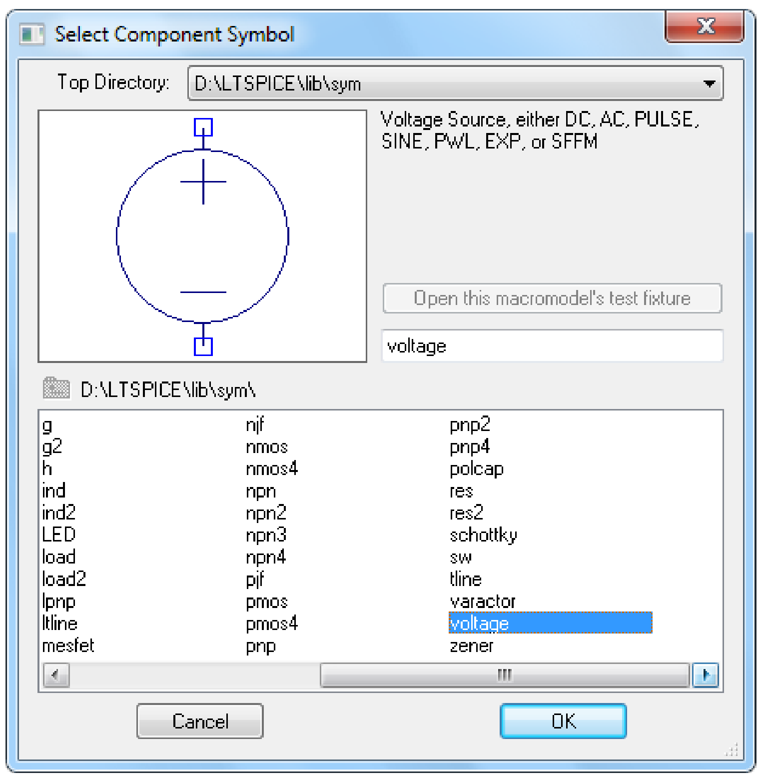


5. The component button can be used to put any circuit component on the schematic diagram. The wire button can be used to connect different components.
6. The label button can be used to give labels to different nodes. Otherwise a default name isgivento each node.
7. To delete a component from the diagram either use F5 or click the scissors button and click on the component to be deleted.

8. To make a simple circuit as shown below click on the component button.



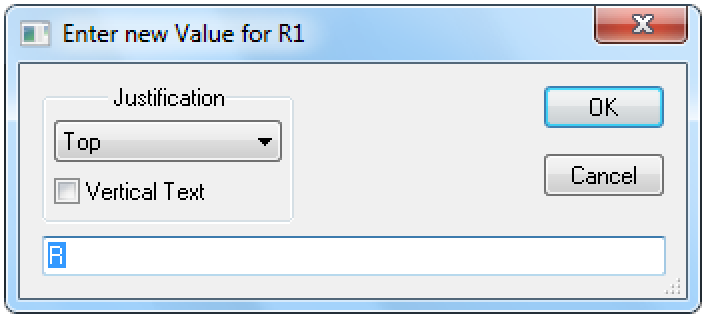
9. The following window appears.



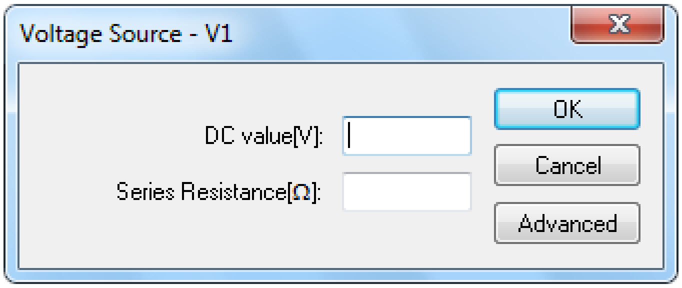
10. This window contains a collection of basic component; to make the circuit as shown above choose the voltage source and place it on the schematic diagram.
11. Place resistors on the schematic diagram and join those using wires to make the complete circuit. To rotate aresistor so that it can be placed as in the given circuit,

selectthe resistor and press ***ctrl+r***. Similarly ***ctrl+e*** is used to mirror a resistor. Place the ground at the lower node.

12. The circuit iscomplete. To set the values of observe that each component has two labels attached to it. One represents the name and other represents the value of the component .To change the name or the value of any component left click on the corresponding label e.g. each resistor comes with a label R1, R2 etc that represents its name. Each resistor is also accompanied by a label R that represents its value. To change the value of the resistor use left click on the label.Thefollowing window appears.



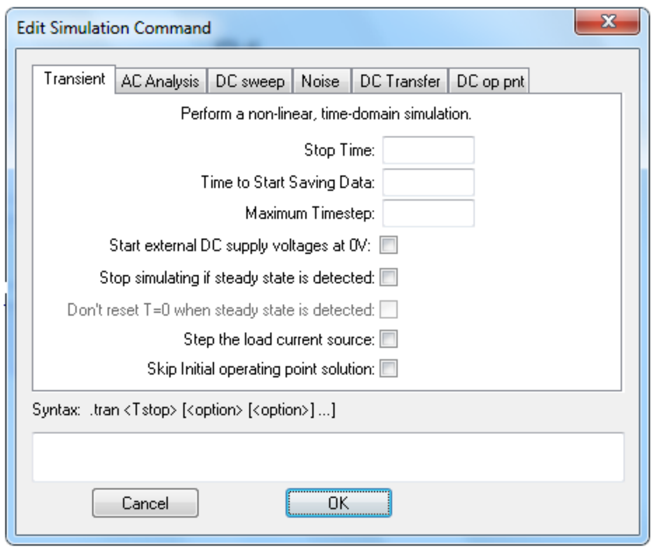
13. Enter the value in the text field and click ok
14. Another way setting different properties of a component is by using left click on the component itself e.g. if we use left click on the voltage source the following window appears.



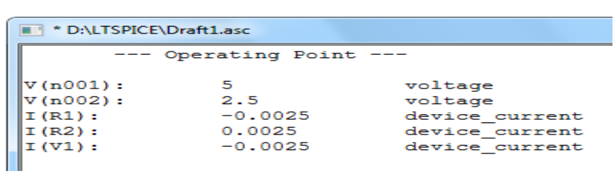
15. Now the DC value and the source internal resistance can be set from this window. The advanced button can be used to change the voltage source from DC to other types which shall be explored in other tasks.

TASK1: Simulating a Simple Circuit to Obtain DC Bias Point: -

1. Afterthecircuithasbeenmadeandvaluesaresetasexplainedabovewecansimulatethe circuit todeterminetheDCbias pointi.e. all node voltages and currents.
2. Supposewe usetheDCvoltagesourceequalto5Vandbothresistorsaresetequalto1K.(The symbolforprefixessuchaskiloandmilliandmegaarecaseinsensitivecanbeconfusinge.g.the symbolforkiloisKork, formilliitisMor mandfor megaitisMEGorMeg.
3. Now click Simulate-> run from the top menuor click the run button on the toolbar. The following window appears



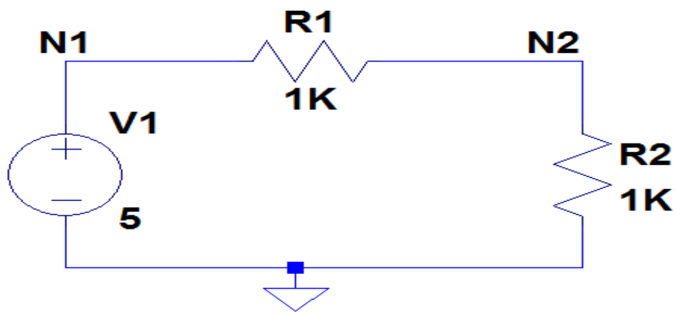
4. It shows thepossible type of analyses LTspice can perform. At the moment we are only interested in the DC biaspointso click the DC oppnt button on the top menu of this window and click ok.
5. The operating point iscalculated and thefollowing results appear.

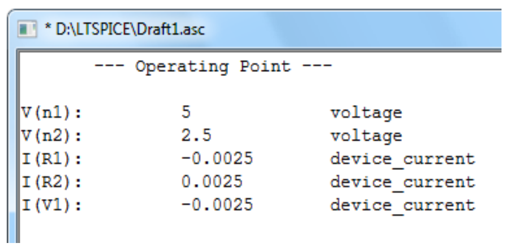


6. Since we placed no label on the nodes so they are given names n001 and n002.The node

with ground connected is named 0.

7. Now we place our own label son the nodes by using the label net button on the tool bar and run the simulation again.

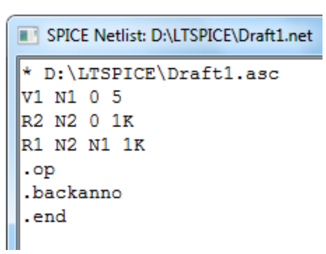




```
* D:\LTSPICE\Draft1.asc
       --- Operating Point ---

V(n1):         5              voltage
V(n2):         2.5            voltage
I(R1):         -0.0025        device_current
I(R2):         0.0025         device_current
I(V1):         -0.0025        device_current
```

8. So the node voltages and current through each component are listed.Note that the current through the resistor R1 is negative. The reason is that R1 was rotated before being placed in the circuit .LTspice defines a predetermined direction of current through each resistor. A negative value shows that the actual direction of current is opposite to the predetermined assumed direction.To check whatdirection LTspice has assumed click View->Spice Netlist from the top menu. A net-list is at extversion of the schematic diagram. The following window appears



```
SPICE Netlist: D:\LTSPICE\Draft1.net
* D:\LTSPICE\Draft1.asc
V1 N1 0 5
R2 N2 0 1K
R1 N2 N1 1K
.op
.backanno
.end
```

9. Shows that R1 is connected between nodes N2 and N1 and hence the assumed direction of current is from N2 to N1. Whereas the actual current flows from N1 to N2 and hence the output generated a negative sign.

10. To connect R1 i.e. the assumed direction is from N1 to N2 select the resistor by using the moveordrag button (the buttons with the symbol of open or closed hand) from the tool bar and press ctrl+e to mirror the resistor. Now run the simulation and view the Spice Netlist.
11. The current through Voltage source is negative as it should be by passive sign convention

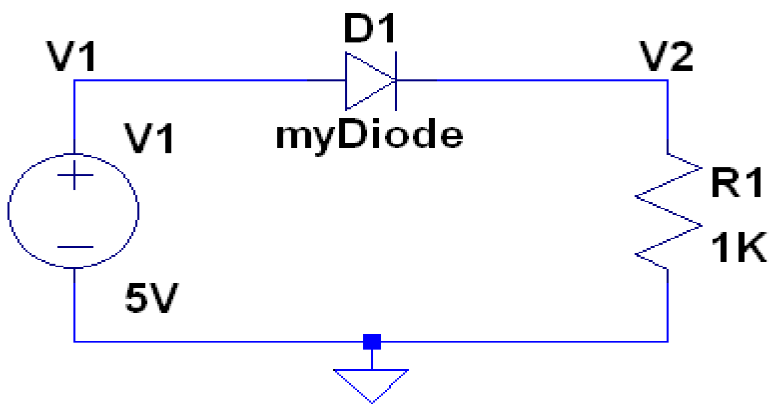
DC SWEEP ANALYSIS: -Students have no idea

What diodes are…they study them in electronics 1…

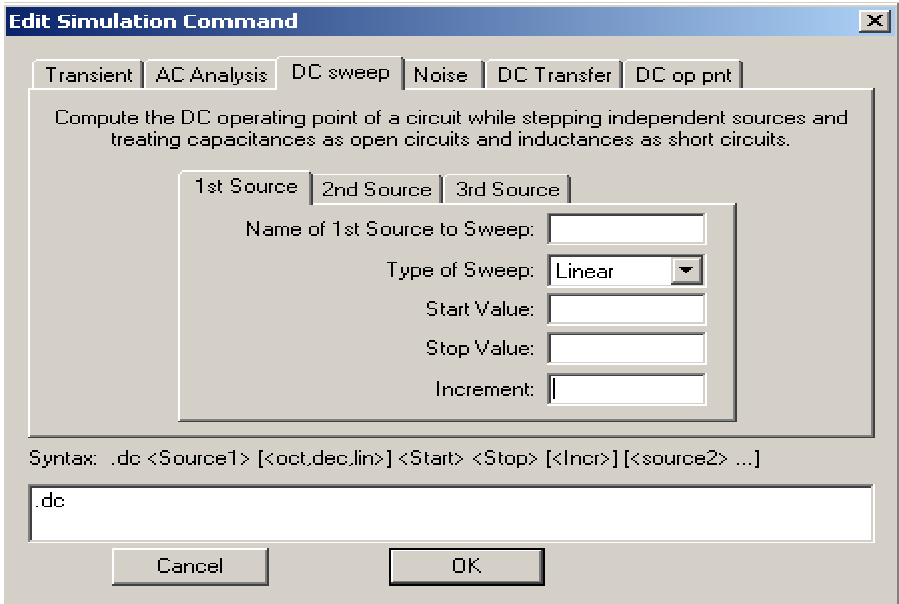
DC Sweep is a type of simulation in LTSpice where the DC voltage of one or more than one source(s) is varied in a step-wise manner. At each step the DC bias point is calculated, there ultsare usually represented in the graphs. This type of analysis is most suitable when plotting the V-I curves of different devices or when designing a specific DC bias point for a particular circuit.

TASK 2: Plotting the V-I curve of a real diode

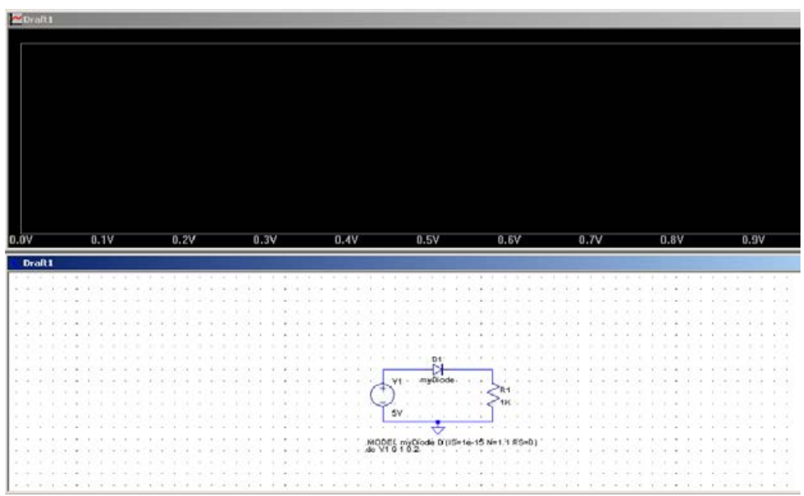
1. Create a new schematic and draw the following circuit. Remember to label them asV1andV2 as shown.



2. To perform the DC sweep analysis click the run command and choose the 'DC Sweep' button on the window that appears.



3. Provide the name of the DC source which will perform the sweep i.e. V1 in our case. Provide a start value lets say 0V and an end value lets say 2V and an increment (i.e. Step) value letssay 0.05V.
4. The simulation will be performed and agraphical window would appear.

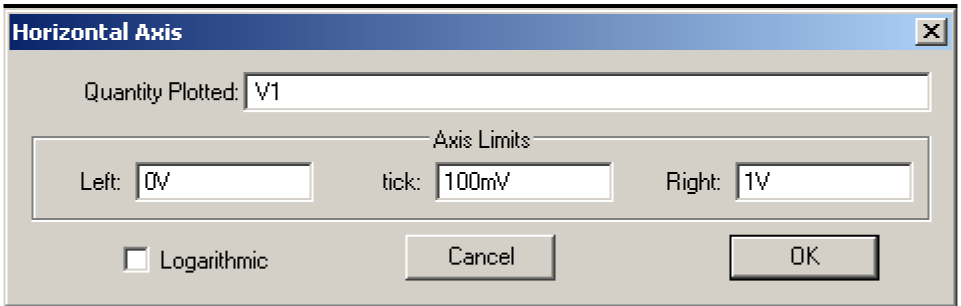


GRAPHICAL ANALYSIS (TRACE): -

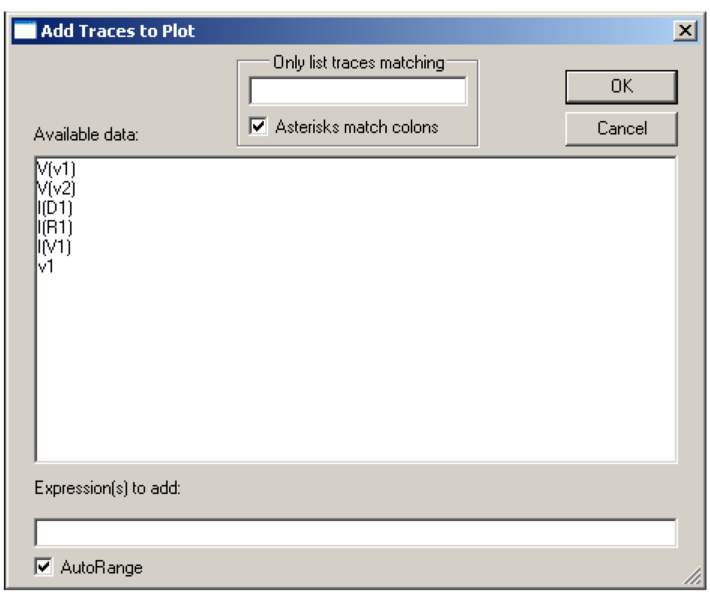
1. Since the results of the DC Sweep are best viewed using a graphical utility so shall use the graphical analysis of LTSPICE also called TRACE. When use a DC Sweep analysis

the value of the DC voltage source varies from the initial to the final value in the form of incremental step, at each step all voltages and currents are measured and stored.Here we want to plot the diode current as the value of diode voltage varies.

2. Maximize the graph window;take them cursor over the horizontal axis,Use the 'right-click' button and a window would appear.



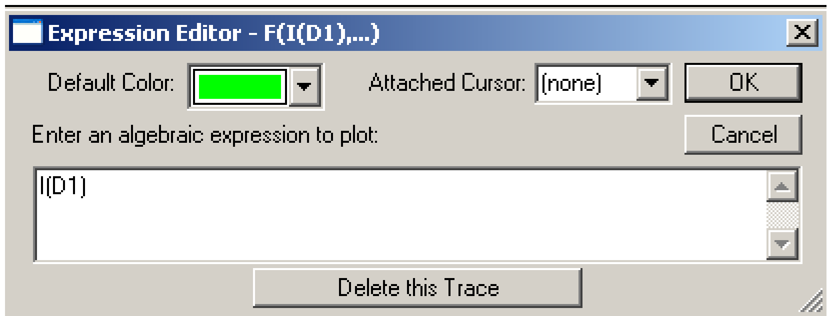
3. This window tells us that the quantity plotted on the horizontal-axis is the DC source voltage V1. It also tells what themaximum and minimum value on the axis is and where the ticks are placed. We can change all these values .Since we want to plot diode current vs diode voltages we should place diode voltage on the horizontal-axis. To do so change the value of 'Quantity plotted' from 'V1' to **V(V1)–V(V2).**
4. Now move the mouse cursor somewhere on the graphical screen and use 'left click', from the drop-down menu that appears click'AddTrace'.The following window appears



5. It lists all the voltages and current which have been calculated during the DC sweep

simulation. Choose I(D1) which is the diode current.

6. The V-I curve is plotted on the screen.
7. A number of mathematical operations can be performed on the graphs. A constant may be added, subtracted, multiplied or dived from the graph. Two or more graphs may be added, subtracted,multiplied or divided. Similarly the logarithm or some trigonometric function of the graph may be plotted as well.



8. In this window any algebraic expression may be written.
9. By using right click on the graph,thennumerical values at different points can be observed.

Note: Now you have learn how to simulate the circuits on LTspice and plot Graphs.

<u>Task # 3:</u>

Create the circuit shown in figure 1 below on LTSPICE and simulate using **DC Sweep Analysis**by varying voltages (from 0.5V to 5V in stepwise manner) for three values of resistance R1 and plots the V-I curves of a Resistor R1 before coming to the lab and bring the screen shorts with you in lab.The values of resistors should be taken as 1KΩ, 2.2KΩ and 5KΩrespectively. (The values of resistor available in laboratory may slightly vary. Also simulate the circuit using **DC Bias-Point** and observe the values of corresponding current **'Simulated Value'**. Record these values along with calculated in the table below:

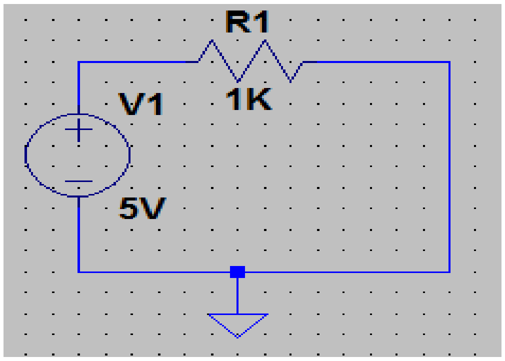


Figure 2

In-Lab Task:

Task# 1: Procedure

Task 1 Assemble on the bread-board the circuit shown in Figure 2.6 using the same voltage setting on the power supply and the same resistor as shown.Set the multi-meter to measure dc current, make sure that the leads are correctly set for current measurement.Measure the current flowing through the resistor. Does this value agree with the calculations using Ohm's Law $\left(i = \frac{v}{R} \right)$?

Task 2 Measure the current flowing through the resistor in the opposite direction. This is done by reversing the leads of the ammeter. Does this value agree with the calculations using Ohm's Law $\left(i = \frac{v}{R} \right)$?. Record these Values and Complete the table shown in Measurements Section

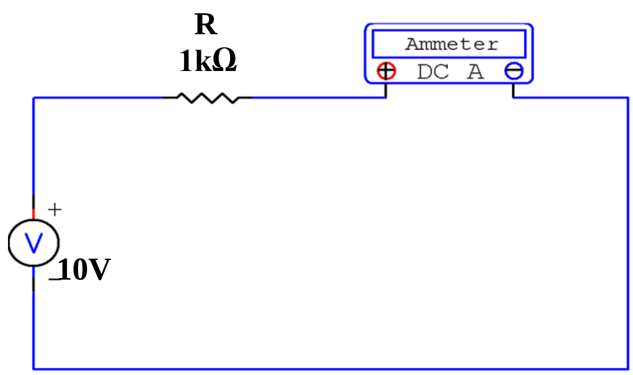


Figure 3: A simple resistive circuit to verify Ohm's law

Lab Exercise:

Safety Precautions

- ☑ Look at each exercise carefully before connecting the circuits.
- ☑ Make sure all power is off before connecting or disconnecting components.
- ☑ Ask your TA to check the circuit before turning on the power.
- ☑ When measuring voltage or current, make sure the DMM is correctly set for what you need to measure.

Measurement Section:Task-1

V (volts)	R = 1K Ω			R = 2.2K Ω			R = 5K Ω		
	I (mA) Measured	I (mA) Calculated	I (mA) Simulated	I (mA) Measured	I (mA) Calculated	I (mA) Simulated	I (mA) Measured	I (mA) Calculated	I(mA) Simulated
0.5									
1.0									
1.5									
2.0									
2.5									
3.0									
3.5									
4.0									
4.5									
5.0									

Table 1

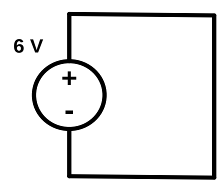
Post-Lab Tasks:

1. Plot the values Measured in the lab with voltage on x-axis and current on y-axis on graph paper.

2. Answer the following Questions?

a. What is the advantage of using LTSPICE in circuit analysis?

b. What would happen if a wire having no resistance at all (0 Ω) was connected directly across the terminals of a 6 volt battery? How much current would result, according to Ohm's Law?



c. How would you place a DC current source with downward direction on LTSPICE schematic?

d. When you simulate the circuit (Figure 2.6) in LTSPICE, the magnitude of current through all elements is same; however, negative sign appears with current through voltage source. What is the reason?

Critical Analysis / Conclusion

Lab Assessment		
Pre Lab	/5	/25
Performance	/5	
Results	/5	
Viva	/5	
Critical Analysis	/5	
Instructor Signature and Comments		

Lab 03: Resistor Combinations - Series and Parallel

Pre Lab

Theory: Resistance in series

The circuit in which the current remains the same and the voltage is different across each resistor is called a series circuit, as shown in Figure 3.1. In a series circuit the total resistance is the sum of individual resistance values. If k number of resistors is connected in series then the equivalent resistance Req is given by,

$$R_{eq} = R_1 + R_2 + \ldots + R_k \qquad (3.1)$$

Or

$$R_{eq} = \sum_{i=1}^{k} R_i \qquad (3.2)$$

In Lab

Objectives

- Learn how to connect the resistors in series and parallel on breadboard.
- To measure the equivalent resistance of series and parallel combination of resistors using digital multi-meter (DMM), and compare with its theoretical value.
- To verify that same amount of current flows through each series circuit element.
- To verify that equal voltage appears across each parallel circuit element.

Equipment Required

Resistors, DMM, Breadboard, DC Power Supply, Connecting wires

Knowledge Level

Before working on this lab you should have a basic understanding of Ohm’s law and use of DMM for resistance measurement.

Task(1)

Task 1. Take any three resistors of your choice. Resistance values are marked onto the body of the resistor using a series of colored bands. Find their individual resistance values through color code identification and record these in Table 3.2.

Task 2. Connect the three resistors in series as shown in Figure 3.2.
Calculate the value of equivalent series resistance R_{eq} and record the value in Table 3.2.

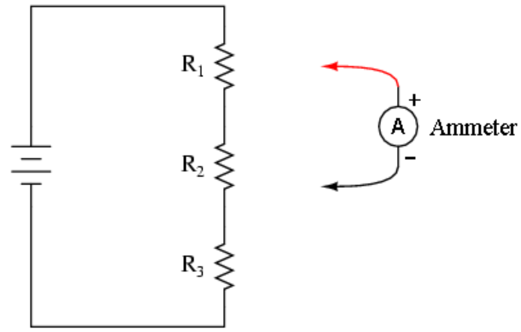


Figure 1: Resistors in Series

Task 3. Measure the equivalent resistance of the circuit on breadboard using the Digital Multi-meter (DMM) being set for resistance measurement; and record the value in Table 3.2. Do the measured and calculated equivalent resistance values agree?

Task 4. Apply 5V across the terminals of the series combination of three resistors on breadboard. The terminals are circled and shown in Figure 3.2.

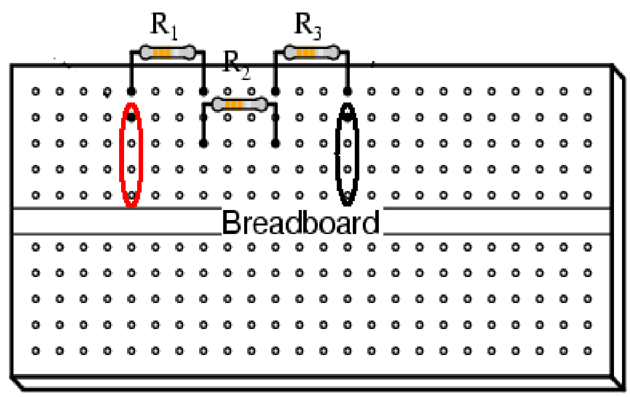


Figure 2: Resistors connected in series on the breadboard

Task 5. Use Ohm's law to calculate the value of current i_T flowing in the circuit. Record this value in Table 3.2.

Task 6. Use DMM being set as ammeter to measure the value of total current i_T flowing through the circuit as shown in Figure 3.3. Record the value in Table 3.2. Do the measured and calculated current values agree?

Task 7. Connect the ammeter in series with the first resistor R_1. The value would give the current I_1 flowing through R_1. Record the value of current in Table 3.2. Similarly, connect the ammeter in series with the remaining two resistors R_2 and R_3; and record the I_2 and I_3 current values in Table 3.2. All these values should be same, which shows that when connected in series, the resistors have same amount of current flowing through them.

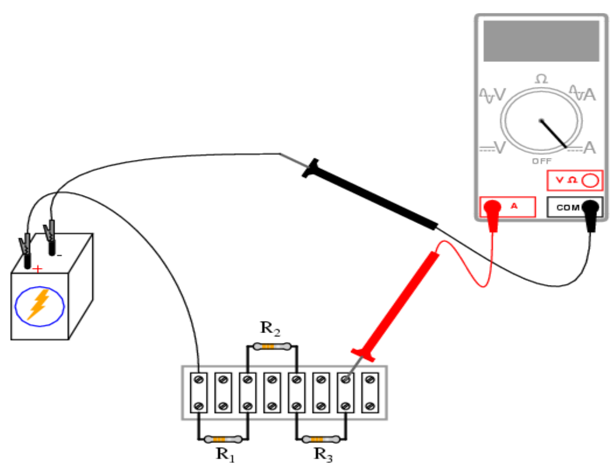


Figure 3

Task 8. Create the circuits shown in figure 3.1 in LTSPICE. Simulate the circuit and observe all the branch voltages and currents. Record the values mentioned in measurement tables.

Resistances in parallel

In a parallel circuit voltage across all the resistors remains the same and the supply current or total current is the sum of the individual currents in different parallel paths. The sum of the reciprocal of parallel resistances connected in the circuit is equal to the reciprocal of the equivalent resistance connected in the circuit. If *k* number of resistors is connected in parallel then the equivalent resistance R_{eq} is given by,

$$\frac{1}{R_{eq}} = \frac{1}{R_1} + \frac{1}{R_2} + \dots + \frac{1}{R_k} \tag{3.3}$$

Value of R_1 (Ω)	Value of R_2 (Ω)	Value of R_3 (Ω)	R_{eq} calculated(Ω)	R_{eq} measured(Ω)
i_T (mA) calculated	i_T (mA) measured	V_1 (V) measured	V_2 (V) measured	V_3 (V) measured
i_T (mA) Simulated		V_1 (V) Simulated	V_2 (V) Simulated	V_3 (V) Simulated

Table 3.1

Task(2)

Task 1. Take any three resistors of your choice. Find their individual resistance values through color code identification and record these in Table 3.3.

Task 2. Connect the three resistors in parallel as shown in Figure 3.5.

Task 3. Calculate the value of equivalent parallel resistance R_{eq} and record the value in Table 3.3.

Task 4. Measure the equivalent resistance of the circuit on breadboard using the Digital Multi-meter (DMM) being set for resistance measurement; and record the value in Table 3.3. Do the measured and calculated equivalent resistance values agree?

Task 5. Apply 5V across the terminals of the parallel combination of three resistors on breadboard. The terminals are circled and shown in Figure 3.5.

Task 6. Use DMM being set as voltmeter to measure the voltages across the three resistors. Record the values in Table 3.3. All these values should be equal to the supply voltage. Do the individual resistor voltages is equal to the supply voltage?

Task 7. Use Ohm's law to calculate the total amount of current i_T flowing in the circuit. Record this value in Table 3.3.

Task 8. Connect the ammeter in series with the parallel combination of resistors to measure the total current flowing through the circuit, as shown in Figure 3.6. The value would give the current i_T flowing through equivalent parallel resistance. Record the value of current in Table 3.3.

Task 9. Create the circuits shown in figure 3.4 in LTSPICE. Simulate the circuit and observe all the branch voltages and currents. Record the values mentioned in measurement tables.

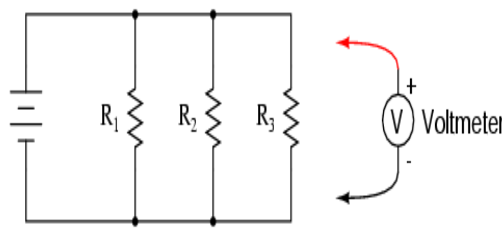


Figure 4: Resistors in Parallel

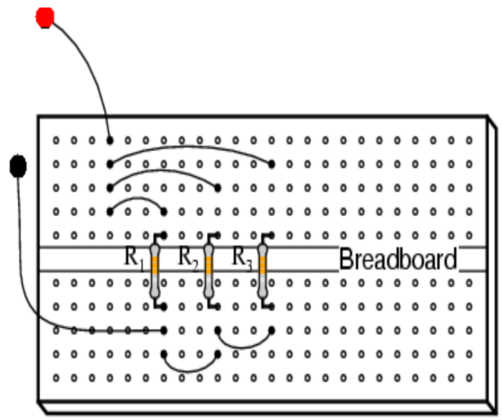


Figure 5: Resistors connected in parallel on breadboard

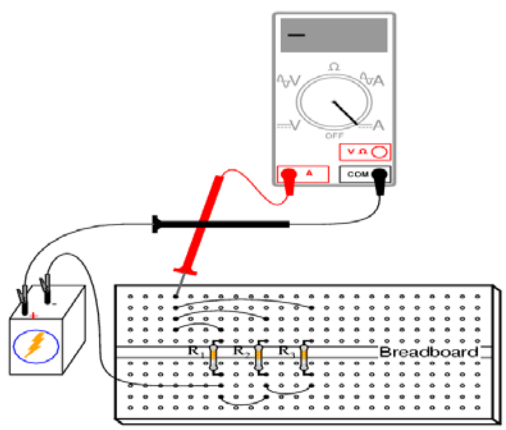


Figure 6: Ammeter connected to measure the total current flowing in the circuit

Measurement Tables:

Value of R_1 (Ω)	Value of R_2 (Ω)	Value of R_3 (Ω)	R_{eq} calculated(Ω)	R_{eq} (Ω) measured
i_T (mA) calculated	i_T (mA) measured	I_1 (mA) measured	I2 (mA) measured	I3 (mA) measured
i_T (mA) Simulated		I_1 (mA) Simulated	I_2 (mA) Simulated	I_3 (mA) Simulated

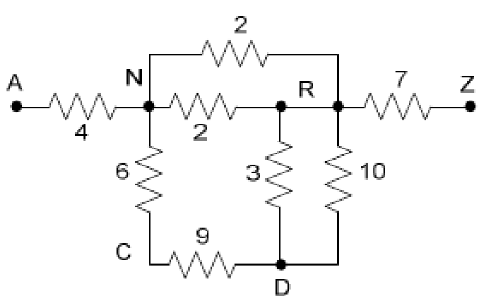
Table 3.2

Post lab

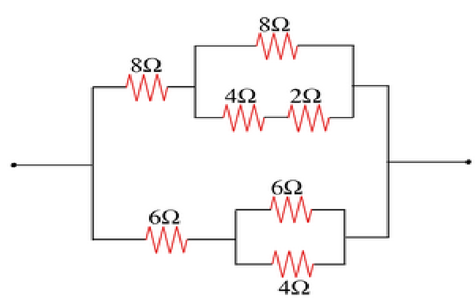
Questions

1. Assume that you have a 100 Ω resistor. You want to add a resistor in series with this 100 Ω resistor in order to limit the current to 0.5 amps when 110 volts is placed across the two resistors in series. How much resistance should you use?

2. Identify the resistances pairs that are in parallel in the following circuit:



3. What is the equivalent resistance of the following resistance combination?



Critical Analysis / Conclusion

Lab Assessment		
Pre Lab	/5	/25
Performance	/5	
Results	/5	
Viva	/5	
Critical Analysis	/5	
Instructor Signature and Comments		

Lab 04: Kirchhoff's Laws &Voltage-and-Current Division Principles

Pre Lab

Kirchhoff's Laws

Kirchhoff's Laws are based on energy and charge conservation. Kirchhoff's voltage law is based on energy conservation and states that the algebraic sum of the potential (voltage) drops around a complete path is equal to zero. For example, in the circuit shown in figure 4.1, the relations between the circuit elements and resistances for the path ABCDA is:

$$-V + I_1R_1 + I_2R_2 + I_5R_5 = 0 \quad (4.1)$$

Kirchhoff's current law is based on charge conservation, and states that the algebraic sum of the currents entering a node is zero.

A node is a point such as "B" in figure

4.1, where I_1 enters and I_2 and I_3 leave. The other nodes in figure 4.1 are "E", "D", and "C". Referring to figure 4.1, at node "B" we have:

$$I_1 - I_2 - I_3 = 0 \quad (4.2)$$

While at node "E" the relation is:

$$I_3 - I_4 - I_6 = 0 \quad (4.3)$$

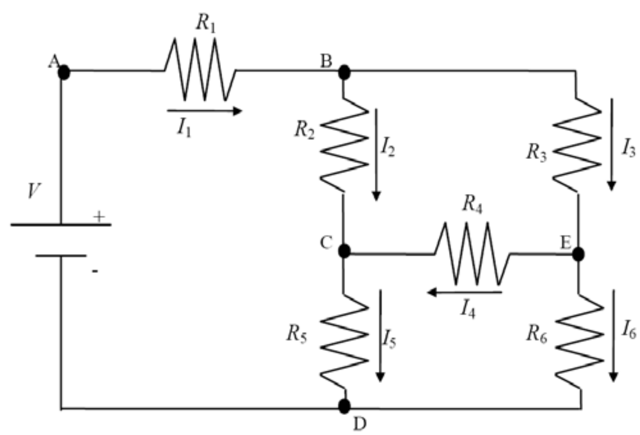


Figure 1: Example circuit to illustrate the experiment

Pre-Lab Tasks:

Solve the circuits shown in figure 4.2 and 4.3 before coming to the lab. You can chose any value for resistors…..and take v as 5 v Calculate the voltages V_1 through V_5 using Voltage Divider Rule for figure 4.2 and currents I_1 through I_3 using Current Divider Rule for figure 4.3. Bring the results with you.

(The values of resistors available in laboratory may slightly vary).

In Lab

Objective

- To study the validity of Kirchhoff's voltage and current laws.
- To study the validity of the voltage and current division principles.

Equipment Required

Resistors, DMM, breadboard, DC power supply, and connecting wires.

Knowledge Level

- Before working on this lab, students should have good understanding of Kirchhoff's voltage and current laws.
- Students should be able to theoretically solve the circuit shown in circuit diagram.
- Students should know how to simulate the electric circuit using LTSPICE.

Since we are not supposed to use the recipe based lab…hence I think the values of resistors should not be mentioned here….students should use the pre lab task values here

Task (1):Validation of Kirchhoff's Voltage Law and Voltage Divider Rule

Task 1. Using the provided equipment, construct the circuit as shown in figure 4.2. Make sure that every connection is clean and accurate.

Task 2. Turn the power supply 'on'. Adjust the supply voltage to 5V.

Task 3. Use the DMM to measure the power supply output voltage and the potential drops across each of the 5 resistors. Record these measurements in Table 4.1.

Task 4. Use the following relationship to verify Kirchhoff's voltage law:

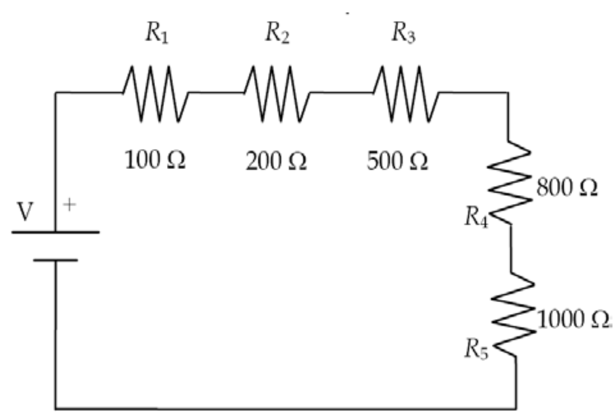


Figure 2: Circuit to validate Kirchhoff's voltage law

$$V = V_1 + V_2 + V_3 + V_4 + V_5 \tag{4.4}$$

Comment on your results.

Task 5. Use the voltage divider rule to determine the voltage drop across each of 5 resistors. Calculate the sum of the potential drops across all 5 resistors. Does this calculated sum agree with the measured sum found in the previous step? Comment on the validity of Kirchhoff's voltage law and the voltage divider rule based on your measurement results.

Table 1

Element		Voltage across element (V)	Calculated	Measured	Simulated
R1 (Ω)		**V1**			
R2(Ω)		**V2**			
R3(Ω)		**V3**			
R4(Ω)		**V4**			
R5(Ω)		**V5**			

Table 4.1

Verification of KVL:

The following relationship is verified: $V_1+V_2+V_{3+}V_{4+}V_5 = V$

Again the values of resistors should not be mentioned here…students should be provided with a range of values for resistors

Task (2): Validation of Kirchhoff's Current Law and Current Divider Rule

Task 1. Connect the circuit as shown in figure 4.3 and make sure all the connections are appropriate.

Task 2. Turn on the DC supply. Adjust the source voltage to 5V.

Task 3. Use the DMM to measure the total current flowing through the circuit. Similarly measure the current flowing through each of 3 resistors. Record these measurements in Table 4.2.

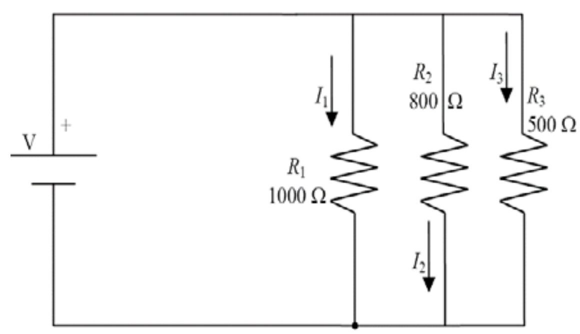


Figure 3: Circuit to validate Kirchhoff's current law

Task 4. Use the following relationship to verify Kirchhoff's current law:

$$I = I_1 + I_2 + I_3 \tag{4.5}$$

Comment on your results.

Task 5. Use the current divider rule to determine the current flowing through each of 3 resistors. Calculate the sum of the currents flowing through all 3 resistors. Does this calculated sum agree with the measured sum found in the previous step? Comment on the validity of Kirchhoff's current law and the current divider rule based on your measurement results.

Task 6. Calculate the equivalent resistance R_{eq} of the three resistors used in this circuit. With supply voltage V and R_{eq}, determine the total current I flowing in the equivalent circuit. Compare the calculated and measured current values as tabulated in Table 4.2. Do they agree? What is the reason for the slight variation

Table 2

Element		**Current through element (mA)**	**Calculated**	**Measured**	**Simulated**
		I (through Voltage Source)			
R1 (Ω)		**I1**			
R2(Ω)		**I2**			
R3(Ω)		**I3**			

Table 4.2

Verification of KCL:

The following relationship is verified: $I_1+I_2+I_3 = I$

Post Lab

Questions

1. What is path/loop? How many paths/loop are there in Fig 4.1?

2. Two resistors R_1 and R_2 are connected in series. The voltage drop across R_1 is larger than R_2. What can we infer about comparative values of the resistances? Is $R_1>R_2$ or $R_1<R_2$.

Critical Analysis / Conclusion

Lab Assessment		
Pre Lab	**/5**	**/25**
Performance	**/5**	
Results	**/5**	
Viva	**/5**	
Critical Analysis	**/5**	
Instructor Signature and Comments		

Lab 05: Voltmeter and Ammeter Design Using Galvanometer

A) Voltmeter Design Using Galvanometer
B) Ammeter Design Using Galvanometer
C) Determine The Internal Resistance of a Voltage Source

Pre Lab

Theory

A galvanometer is a sensitive device which can measure very small currents accurately. A galvanometer itself may not be very useful for measuring currents in most of the circuits where current is usually in milli amperes. However by slight alterations a galvanometer can be converted into a voltmeter or an ammeter with a reasonably larger range. It is basically a current measuring device but by knowing its internal resistance and using ohm's law we can use it to measure voltage across a circuit element. However such a usage of galvanometer has two serious limitations. First, since the internal resistance of a galvanometer is usually small it would seriously affect the voltage reading across the element for which it is used. Second, as galvanometers can measure only small amounts of current (300 micro amperes) so the range of voltage which they can measure is very small as well. We can overcome both these limitations very easily. By connecting a very large resistance in series with the galvanometer we can make its total resistance significantly large. This would increase the range of measurable voltage and would decrease the loading effect of the galvanometer as well.

PART A: VOLTMETER DESIGN

The selected galvanometer can measure currents from 0-300 micro amperes. The internal resistance of different galvanometers is different but it ranges from 130-150 ohms. We can measure it by using DMM. We will convert our galvanometer into a voltmeter with a range of -5 to 5 volts. It should give maximum deflection when a voltage of 5V is applied across its terminals. We know that it would give maximum deflection only if the current through

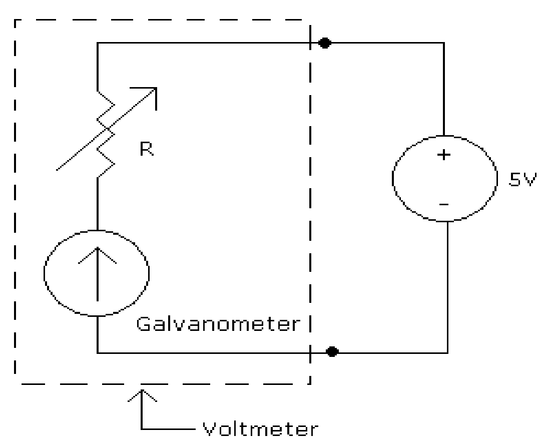


Figure 1: Voltmeter design using galvanometer

it is maximum, i.e. 300 microamperes.

If v = 5V then i = 300µA. Suppose the internal resistance R_m of the galvanometer is 140Ω then KVL around the closed loop would give the following equation:

$$v = iR + iR_m \tag{5.1}$$

$$R = \left(v - iR_m\right)/i \tag{5.2}$$

$$R = \left(5 - (300\mu)(140)\right)/(300\mu)$$

$$R = 16.5k\Omega$$

The design of the voltmeter as explained in the theory section has two constraints:

1) The actual value of the internal resistance (R_m) of the galvanometer is unknown.
2) If found the actual value of internal resistance (R_m) still, it would be really fortuitous if the calculated value of the resistance *R* is actually present in the laboratory.

So to overcome these two constraints a circuit as shown in figure 5.1 is build. A variable resistance (1 MΩ) has to be attached in series with the galvanometer. The value of the variable resistance is slowly varied until maximum deflection of the galvanometer is achieved, thus our voltmeter design is complete and calibrated for -5 to +5 volt measurement.

Measurement of the internal resistance of Galvanometer:

To calculate the actual value of the internal resistance of the galvanometer:

1) Measure the voltage across the sensitive galvanometer (v_g) in the circuit shown in figure 5.1 using DMM, and record the value in Table 5.1.
2) Measure the current flowing through the galvanometer (i_g) in the circuit shown in figure 5.1, and record the value in Table 5.1.
3) Write down in the calculated and measured value of the internal resistance R_m.
4) Determine the calculated value of *R* using eq. (5.2). Use DMM to measure the value of series resistance *R* and make a note in Table 5.1.

Vg(V)	**Ig(A)**	**Rm meas.(Ω)**	**Rm calc.(Ω)**	**R meas.(Ω)**	**R calc.(Ω)**

Table 5.1Data collection to measure the internal resistance of galvanometer

PART B: AMMETER DESIGN USING GALVANOMETER

The selected galvanometer can measure currents from 0-300 micro amperes. The internal resistance of different galvanometers is different but it ranges from 130-150 ohms. Suppose we wish to convert the galvanometer into an ammeter with a range of 0 to 10 milli amperes. Galvanometer should give maximum deflection when a current of 10mA flow through it. We know that the galvanometer would give maximum deflection only if the current through it is maximum, i.e. 300 micro amperes.

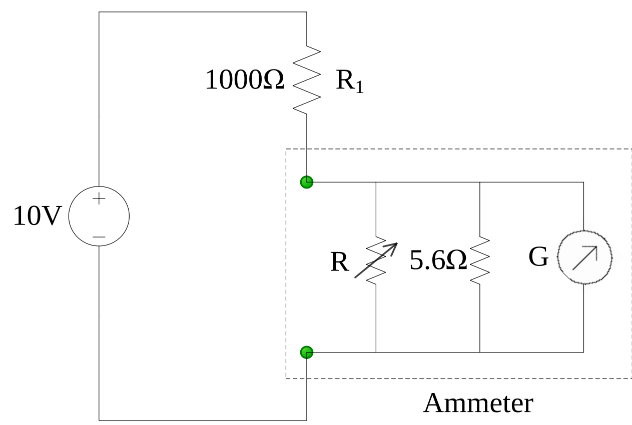


Figure 2: Ammeter design using galvanometer

If i = 10mA then i_g = 300µA and i_s = 9.7mA (i_s is the current flowing through the shunt resistance R_s). Suppose the internal resistance R_g of the galvanometer is 140Ω then according to Current Divider Rule, current through shunt resistance is:

$$i_s = \left(R_g / (R_g + R_s)\right) \cdot i \qquad (5.3)$$

$$R_s = R_g \left(i / i_s\right) - R_g \qquad (5.4)$$

$$R_s = (140)(10/9.7) - 140$$

$$R = 4.33\Omega$$

The design of the ammeter as explained in the theory section has two constraints: First, actual value of the internal resistance (Rg) of the galvanometer is unknown. Second, if found the actual value of internal resistance (Rg) still it would be really fortuitous if the calculated value of the resistance Rs is actually present in the laboratory. So we would make a circuit as shown in figure 5.4. A variable resistance R of 1kΩ has to be attachedin parallel with the galvanometer. The value of the variable resistance is slowly varied until maximum deflection of the galvanometer is achieved, thus our ammeter design is complete and calibrated for 0 to 10 mA measurement. You can verify that the designed ammeter can measure current up to 10 mA by finding the total current flowing through 10V source using the DMM.

Part C: Determine the Internal Resistance of a Voltage Source

Theory:

Any linear electric or electronic circuit or device which generates a voltage may be represented as an ideal voltage source in series with some impedance. This impedance is termed the **internal resistance** of the source. The internal resistance of a source cannot be measured using a conventional ohmmeter, since it requires a current to be observed. However, it can be calculated from current and voltage data measured from a test circuit containing the source and a load resistance. Since both the internal and load resistance (R_v and R_l) are in series with the ideal voltage source, Kirchhoff's laws and Ohm's law give

$$v_l = \left(R_v + R_l\right) i_l \qquad (5.5)$$

This equation can be solved for internal resistance:

$$R_v = \frac{v_l}{i_l} - R_l \qquad (5.6)$$

Where v_l is the voltage and i_l is the current associated with the load resistance R_l.

In Lab

Objective

- To convert a sensitive galvanometer into a voltmeter (measurement range: -5 volts to +5 volts)
- To convert a sensitive galvanometer into a ammeter (measurement range: 0 to 10 milli-amperes)
- To verify that voltages and currents measured by designed voltmeter and ammeter are comparable to that measured by Digital Multimeter
- To determine the internal resistance of voltage source

Equipment Required

Galvanometer, Variable resistor / potentiometer, Resistors, DMM, breadboard, DC power supply, and connecting wires.

Knowledge Level

- Before working on this lab, students should have good understanding of how the devices ammeter, voltmeter and galvanometer work.

- Students should be able to theoretically solve the circuit shown in circuit diagrams.

Task (1): Testing the designed voltmeter

Task 1. Remove the 5V supply from the designed voltmeter unit.

Task 2. Make a circuit of series resistors, shown in Figure 5.2, on the breadboard.

Task 3. Using the designed voltmeter, measure the voltage across each resistance and record the value in Table 5.2. Make sure that you correctly measure the voltage across each resistance. To measure the voltage across R_1 (1kΩ), attach the designed voltmeter as shown in Figure 5.3.

Task 4. The designed voltmeter would provide voltage in terms of divisions shown by the deflection of galvanometer needle. Map these divisions into voltage values. Voltage across other resistances can be measured in a similar manner.

Task 5. Use DMM to verify that the voltage values measured by designed voltmeter and DMM are comparable.

Task 6. Record the difference in both the voltage values in table 5.2.

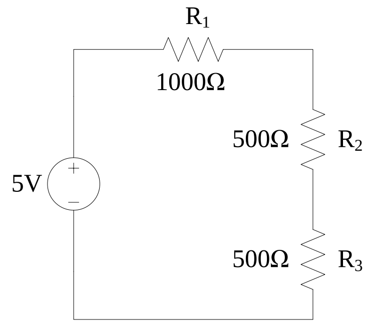


Figure 3: Test circuit to validate the voltmeter design

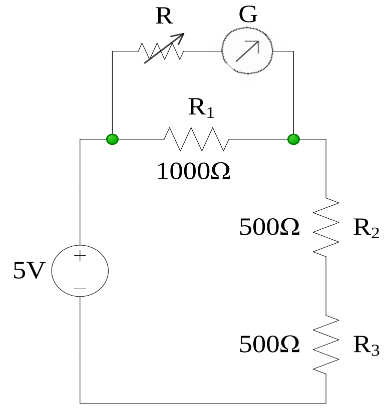


Figure 4: Measuring voltage across the resistor R1

Task 7. Comment on the results.

Measurement Table 1:

Value of resistance (Ω)	V measured by the designed voltmeter (V)	V measured by the DMM (V)	% difference
R1 =			
R2 =			
R3 =			

Table 5.2

Task (2): Testing the designed ammeter:

Task 1. Remove the 10V supply and the R_1 (1kΩ) resistance from the designed ammeter unit.

Task 2. Make a circuit of parallel resistors, shown in Figure 5.5, on the breadboard.

Task 3. Using the designed ammeter, measure the current through each resistance. Remember that designed ammeter is to be connected in series with the resistance as DMM is connected when used as ammeter. To measure the current through R_1 (1kΩ), attach the designed ammeter as shown in Figure 5.6. Current through other resistances can be measured in a similar manner. Record the values in Table 5.3.

Task 4. Use DMM to verify the current values.

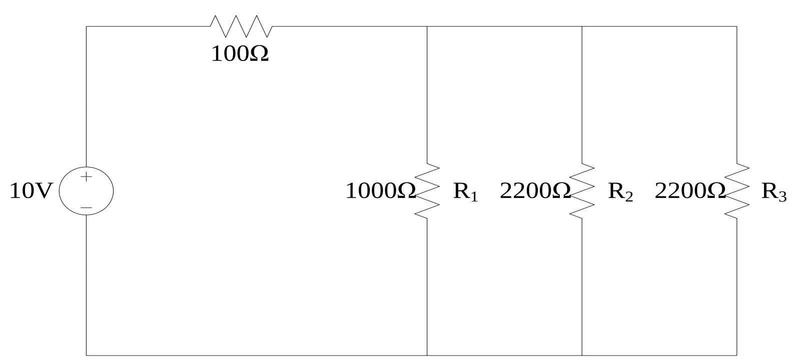


Figure 5: Test circuit to validate the ammeter design

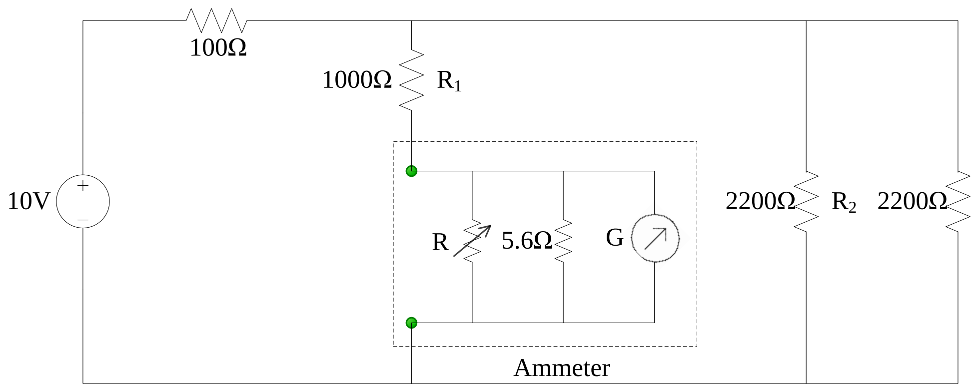


Figure 6: Measuring current through the resistor R1

Task 5. Comment on the results.

Measurement Table 2:

Value of resistance (Ω)	**Current measured by the designed ammeter (A)**	**Current measured by the DMM (A)**	**% difference**
R_1 =			
R_2 =			
R_3 =			

Table 5.3

Task (3): Measuring internal resistance of voltage source

Task 1. Construct on breadboard the circuit shown in Figure 6.1. Leave nodes ‘a’ and ‘b’ open to connect different values of resistance R_x between them.

Task 2. Use DMM to measure the voltage and current for each value of the R_x resistance and record the data in Table 5.4.

Task 3. Plot the values of current against voltage. Make sure that the values of current are measured in Amperes, not milli amperes. The graph should be approximately a straight line (linear). Find out the slope of the graph. Take absolute value of the slope, if negative, and subtract 100 from it (this is the value of resistance attached in series with the voltage source). Take again the absolute value of the answer, if negative. This is the value of the internal resistance R_v of voltage source in ohms.

Task 4. Support your answer by showing the calculations.

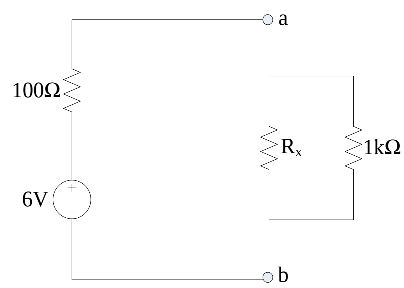


Figure 7: Test circuit to find the internal resistance of a voltage source

Measurement Table 3:

Value of the test resistance R_x (Ω)	**Measured value of the current through R_x, i_x (A)**	**Measured value of the voltage across R_x, v_x(V)**
0 (short circuit)		
100		
220		
470		
1k		
3.3k		
4.7k		
10k		
33k		
100M		
∞ (open circuit)		

Table 5.3

Post Lab

Questions:

1. What do you mean by short and open circuit? What are the values of voltages and currents in open and short circuits?

2. Why high resistance is a desirable attribute of voltmeter?

3. What is the basic motivation behind converting galvanometer into ammeter?

Critical Analysis / Conclusion

Lab Assessment		
Pre Lab	**/5**	**/25**
Performance	**/5**	
Results	**/5**	
Viva	**/5**	
Critical Analysis	**/5**	
Instructor Signature and Comments		

Lab 06: Wheatstone bridge, Delta to Wye & Wye to Delta transformation

Pre Lab

Wheatstone Bridge

The Wheatstone bridge gives a precise method to measure resistance against a known standard. Within a Wheatstone bridge, a comparative device measures two additional relative resistances from two separate resistors. The relative resistance equals the lengths of a divided wire wound in a coil of a potentiometer, a device allowing the manipulation of this resistance ration. Thus, the Wheatstone bridge utilizes repetitive comparisons of potentials to find the potential settings. In this experiment, a voltmeter is used as the null detector and is placed as shown in Fig. 1. The Wheatstone bridge achieves balance when the following condition is satisfied and no current flows through the voltmeter.

$$\frac{R_1}{R_2} = \frac{R_x}{R_s}$$

$$R_x = \frac{R_1}{R_2} R_s$$

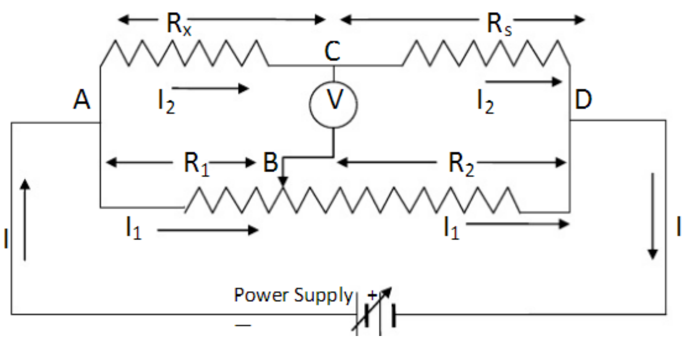


Figure 1

In Lab

Objectives

- To find the value of unknown resistor using Wheatstone bridge.
- To verify experimentally the principle of delta-wye and wye-deltatransformation.

EquipmentRequired

Fixed Resistors, Variable Resistor, DMM, Breadboard, DC Power Supply, Connecting wires

Knowledge Level

Before working on this lab you should have a basic understanding of the use of DMM for resistance meter, ammeter and voltmeter. Student should be able to use voltage divider rule.

Task (1): Measure Unknown Resistance using Wheatstone Bridge

Task 1. Connect the circuit as shown in Fig. 6.1, using a potentiometer for a known resistance R_s.

Task 2. Balance the bridge by moving the slider of the potentiometer. The current flow (or the voltage drop) through the voltmeter must be zero.

Task 3. When the bridge is in the balanced state, measure R_1, R_2, R_s and R_x using a DMM.

Task 4. Use Wheatstone bridge relationship and determine R_x.

Task 5. Find the error in the resistance value and note these readings in Table 6.2.

Task 6. Repeat this experiment twice for two different values of R_s by rebalancing the bridge in accordance with the new resistance values.

Theory: Verification of Delta-Wye and Wye-Delta Conversion

Situations open arise in circuit analysis when the resistors are neither in parallel nor in series. Many circuits of the type can be simplified by using three terminal equivalent networks. These networks are the Wye(Y) or Tee (T) and the Delta (Δ) or Pi (π) (as shown in Fig. 6.3). These networks occur by themselves or as part of larger network. They are used in three-phase networks, electrical filters and matching networks.

Sometimes it is more convenient to work with a Delta Network in place where the circuit contains a Wye configuration and vice-versa. In such situations it is convenient to transform the given circuit to its equivalent circuit. The following relations hold for transforming resistive networks:

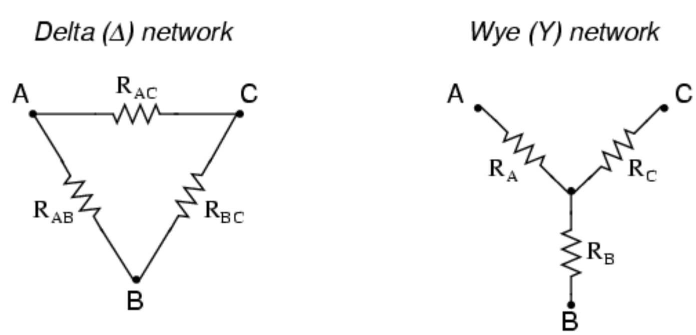


Figure 2: Delta to Wye conversion

To convert a Delta (Δ) to a Wye (Y)

$$R_A = \frac{R_{AB}\,R_{AC}}{R_{AB}+R_{AC}+R_{BC}}$$

$$R_B = \frac{R_{AB}\,R_{BC}}{R_{AB}+R_{AC}+R_{BC}}$$

$$R_C = \frac{R_{AC}\,R_{BC}}{R_{AB}+R_{AC}+R_{BC}}$$

To convert a Wye (Y) to a Delta (Δ)

$$R_{AB} = \frac{R_AR_B+R_AR_C+R_BR_C}{R_C}$$

$$R_{BC} = \frac{R_AR_B+R_AR_C+R_BR_C}{R_A}$$

$$R_{AC} = \frac{R_AR_B+R_AR_C+R_BR_C}{R_B}$$

Measurement Table 1:

R1 (Ω)	R2 (Ω)	R3 (Ω)	Rx(Ω) using Wheatstone Bridge	Rx(Ω) Using DMM	Error (%)

Table 6.1

Task (2): Verification of Equivalence of Delta-Wye and Wye-Delta Conversion

Task 1. Connect the circuit as shown in Fig. 6.4 (a) using 3kΩ resistors and identify the nodes **A, B**and **C** containing the Delta configuration of resistors.

Task 2. Prior to converting this Delta configuration, first determine voltage across the resistors $R_{AB}, R_{AC}, R_{BC}, R_4$ and R_5. Also determine the currents flowing through these resistors and note these reading in Table 6.3. In Table 6.3 E is the voltage between the nodes (e.g. nodes **A** and **B** for R_{AB}) while I is the current through resistor (e.g.R_{AB}) and so on.

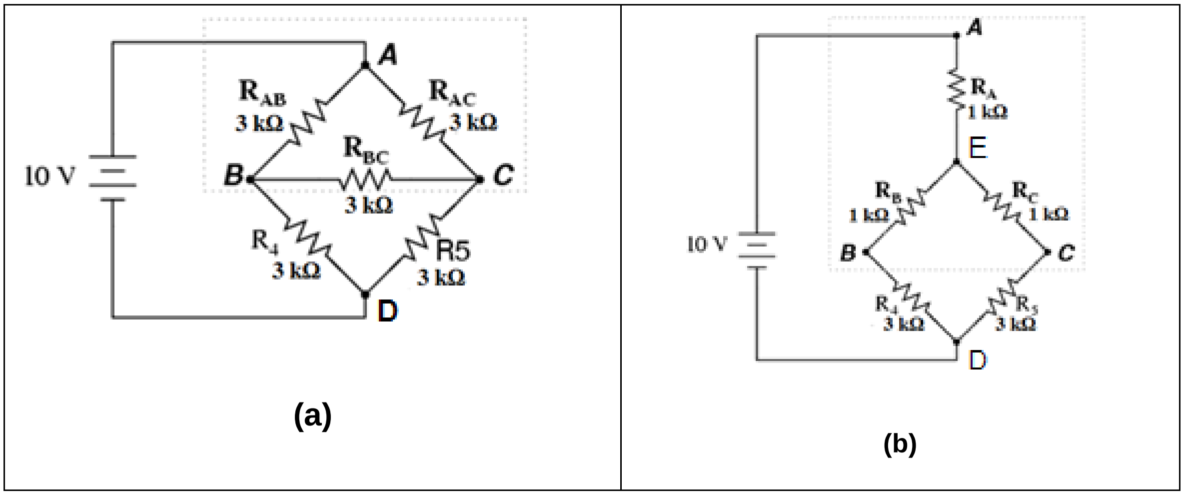


Figure 3

Task 3. Next convert the Delta to its equivalent Wye circuit. For conversion to hold true the voltage difference between the nodes in the converted circuit must be same as that of the difference between the nodes of the original delta circuit.

Task 4. Converted circuit as shown in Fig. 6.4 (b) contains a Wye instead of a Delta. Values of the resistances determined using the relationships mentioned above should be equal to 1kΩ.

Task 5. Determine the voltage across all the resistors and current through them and note the values in Table 6.4.

Task 6. The voltage difference between nodes in the Wye circuit and corresponding nodes in delta circuit should be same. Do keep in mind the positive/negative signs while taking the voltage difference.

Measurement Table 2:

Voltage/ Current	R_{AB} Ω	R_{BC} Ω	R_{AC} Ω	R_{BD} (R_4)Ω	$R_{CD}/ R5$ (Ω)
E (V)					
I (mA)					

Table 6.2 Delta Circuit

Voltage/ Current	RAB(Ω)	RBC(Ω)	RAC(Ω)	RBD/ R4(Ω)	RCD/ R5 (Ω)
E (V)					
I (mA)					

Table 6.3 Wye Circuit

Post Lab

Questions?

1. How is the operation of Wheatstone bridge affected by changes in the input power supply voltage? Would there be an advantage in using a higher voltage?

2. Wheatstone bridge can be used to determine resistance of resistors made of a variety of materials. Is it possible to adapt Wheatstone bridge for determining hot resistance of an electric lamp? Explain your response.

3. How much voltage needs to be dropped across resistor R_1 in order to make voltage V_{AB} equal to zero (as shown in Fig. 6.2)?How much resistance must R_1 possess in order to drop that amount of voltage?

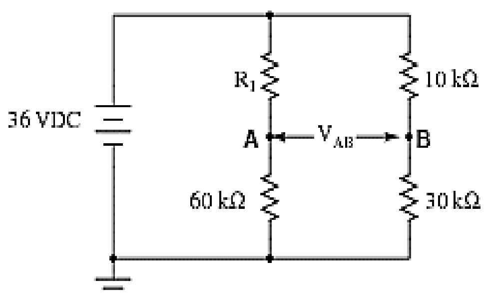


Figure 4

4. Is it possible to find the current through a branch or to find a voltage across the branch using Wye-Delta/ Delta-Wye conversions only? If so, justify your answer.

5. Find the value of R_{eq} for the circuit given below when the switch is open and when the switch is closed?

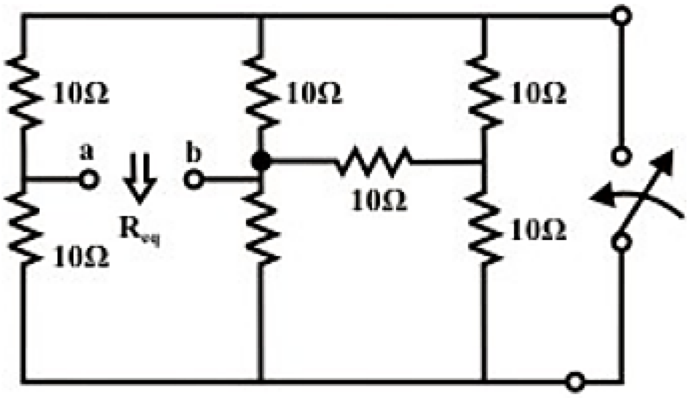


Figure 5

Critical Analysis / Conclusion

<table>
<tr><th colspan="3">Lab Assessment</th></tr>
<tr><td>Pre Lab</td><td>/5</td><td rowspan="5">/25</td></tr>
<tr><td>Performance</td><td>/5</td></tr>
<tr><td>Results</td><td>/5</td></tr>
<tr><td>Viva</td><td>/5</td></tr>
<tr><td>Critical Analysis</td><td>/5</td></tr>
<tr><td colspan="3">Instructor Signature and Comments</td></tr>
</table>

Lab 07: Validation of node voltage method

Pre Lab

Theory

The node-voltage method uses KCL equations that are written at all non-reference nodes. Recall that KCL states that the sum of all the currents at a node is zero. The node-voltage method is comprised of the following steps:

1. Identify all the nodes in a circuit.
2. Choose one of the nodes as the reference node (usually a low potential node).
3. Assign variable names to each of the essential (non-reference) nodes. Mark voltages as per the variable names assigned to each node e.g. v_a, v_b, v_c, etc.
4. Determine the direction of currents i.e. mark voltage polarities across each circuit element.
5. Write a KCL equation in terms of node-voltages at each of the non-reference nodes. The result will be *(n-1)* equations where *n* is the number of essential nodes plus a reference node.
6. Solve the equations and calculate values of node voltages a, b and c. Also calculate voltages and currents through resistances R_1 through R_4. Record the values in Table 7.1.

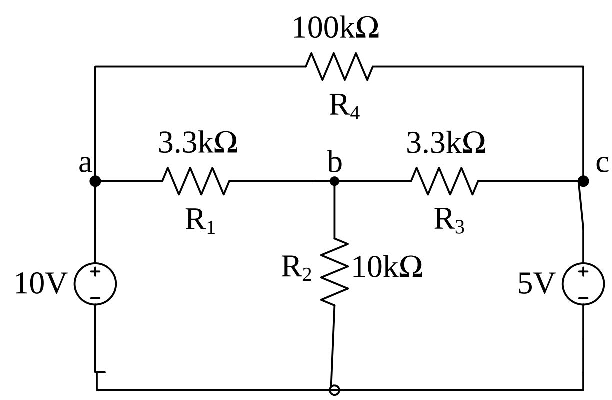


Figure 1: Example circuit to test the validity of the node-voltage method

Pre-Lab Task

Students should solve the circuit shown in figure 7.1 using nodal analysis and calculate all the node voltages. The values of currents through and voltages across all the resistances should also be calculated.

In Lab

Objective

- ➢ To implement the Node Voltage method in lab
- ➢ To verify that measured values of branch voltages and currents are comparable to the values obtained by solving the circuit using Node Voltage method

Equipment Required

Resistors, DMM, breadboard, DC power supply, and connecting wires.

Knowledge Level

- Before working on this lab, students should have good understanding of Node-Voltage method.
- Students should be able to theoretically solve the circuit shown in circuit diagram.
- Students should know how to simulate the electric circuit using LTSPICE.

Task (1)

Task 1. Connect the components in figure 7.1 on breadboard. Measure node voltages and branch voltages and currents. Record all these values in Table 7.1.

Measurement Table 1

Nodes	**Calculated Voltage (V)**	**Measured Voltage (V)**	**Simulated Voltage (V)**
A			
B			
C			

Table 7.1 (a)

Resistance values	**Calculation results**		**Measurement results**		**Simulation result**	
	Voltage (V)	**Current (A)**	**Voltage (V)**	**Current (A)**	**Voltage (V)**	**Current (A)**
R1 =						
R2 =						
R3 =						
R4 =						

Table 7.1 (b)

Post Lab

Questions?

1) What is the difference between Nodal Analysis and Mesh Analysis methods?
2) What is the difference between node and essential node?
3) What is the criterion for selecting reference node out of various essential nodes?

Critical Analysis / Conclusion

<table>
<tr><th colspan="3">Lab Assessment</th></tr>
<tr><td>Pre Lab</td><td>/5</td><td rowspan="5">/25</td></tr>
<tr><td>Performance</td><td>/5</td></tr>
<tr><td>Results</td><td>/5</td></tr>
<tr><td>Viva</td><td>/5</td></tr>
<tr><td>Critical Analysis</td><td>/5</td></tr>
<tr><td colspan="3">Instructor Signature and Comments</td></tr>
</table>

Lab 8: Validation of Norton's & Thévenin's Theorem

Pre Lab

Part a) Norton's theorem

Theory:

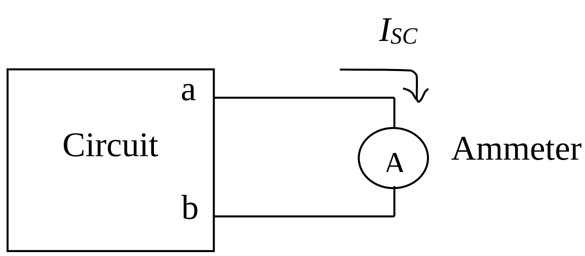
Norton's theorem states that any linear two-terminal circuit can be replaced by an equivalent circuit consisting of a current source I_N in parallel with a resistance R_N where

– I_N is the short-circuit current flowing through the terminals, and
– R_N is the equivalent resistance seen thru the terminals provided that all the independent sources are turned off. R_N is exactly the same as R_{TH}.

Please note that the Thévenin's and Norton's equivalent circuits are related by a source transformation.

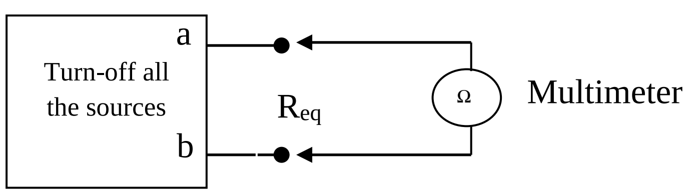
Finding IN: Determine the short circuit current ISC flowing through the load terminals. This is known as the Norton's current IN.

$$I_N = I_{SC} \text{ (short-circuit current)} \quad (10.1)$$



Finding RN: Remove all sources from the circuit, i.e. replace all voltage sources with a short-circuit and current sources with an open-circuit. Then with the help of a multimeter find the resistance between the points 'a' and 'b', denoted by Req.

$$R_N = R_{eq} \quad (10.2)$$



Please note that R_N is exactly the same as R_{TH} and can be determined through many ways as discussed in the last experiment [Thevenin's Theorem].

Part b) Thevenin's Theorem

Thevenin's theorem states that any linear two-terminal circuit (Fig. 10.5a) can be replaced by an equivalent circuit (Fig. 10.5b) consisting of a voltage source VTH in series with a resistance RTH where

– V_{TH} is the open-circuit voltage at the terminals, and
– R_{TH} is the equivalent resistance seen thru the terminals provided that all the independent sources are turned off.

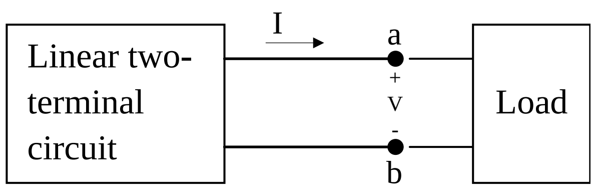


Figure 10.5.a: A linear two-terminal circuit

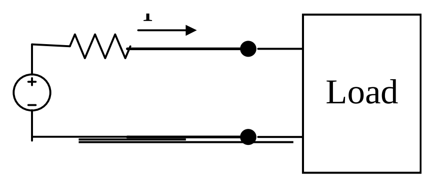


Figure 10.5b: Thevenin equivalent circuit

Pre-Lab Task

Students should solve the circuits shown in figure 10.1 and figure 10.7 for the values of I_N, R_N using Norton's theorem and values of V_{Th}, R_{Th} using Thevenin's theorem.

In Lab

Objective

- Determine the value of Norton's current I_N and the Norton's resistance R_N in a DC circuit theoretically and experimentally
- Determine the Thevenin Voltage V_{TH} theoretically and experimentally
- Find the Thevenin's resistance R_{TH} by various methods and compare values.
- Verify that the values of current through and voltage across the load resistance are comparable in original and Thevenin and Norton Equivalent circuits.
- Demonstrate that maximum power is delivered to load when its value is equal to R_{TH}

Equipment Required

Resistors, DMM, breadboard, DC power supply, and connecting wires.

Knowledge Level

- Before working on this lab, students should have good understanding of Norton's and thevenin's method.
- Students should be able to theoretically solve the circuit shown in circuit diagram.
- Students should know how to simulate the electric circuit using LTSPICE.

Task (1): (Calculating and Measuring RN and IN)

Task 1. Find the Norton's equivalent circuit of the circuit shown in Fig. 10.1. Draw the Norton's equivalent circuit in the space provided in Fig. 10.2. Record the calculated values in Table 10.1.

Task 2. Connect on breadboard the circuit (Fig. 10.1). Remove the load resistance. Turn off all the independent sources. Replace the voltage sources with short-circuits and current sources with open-circuits. (Fig 10.4). Then using an ohmmeter find the equivalent resistance between load points 'a' and 'b'. This is R_N.

Task 3. Short the points 'a' and 'b' (Fig 10.3). Measure the current flowing through 470Ω resistance using ammeter. This is Norton current **'I_N'**. Record the value in Table 10.1

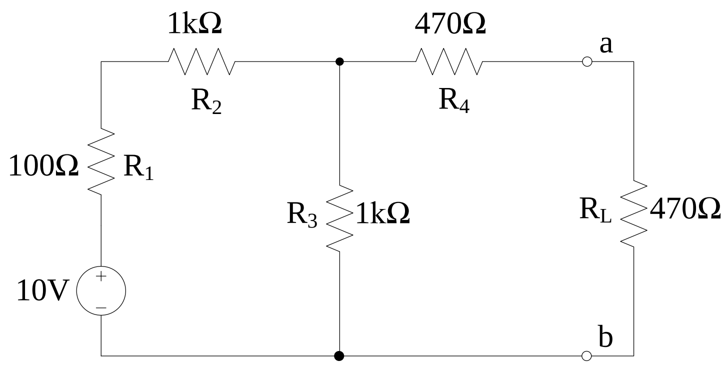


Figure 1: Example circuit to test the validity of the Norton's theorem

(a). R_L = 470Ω	**(b). R_L = 1000Ω**

Figure 2: Norton's equivalent circuit of the circuit shown in Fig 1

Task (2): (Finding IL using the Norton's equivalent circuit)

Task 1. Calculate the current flowing through the load resistance R_L shown in Fig. 10.1. Solve the circuit for R_L=470Ω and R_L=1000Ω and record the values in Table 10.2. The values must coincide with the current values determined in task 1.

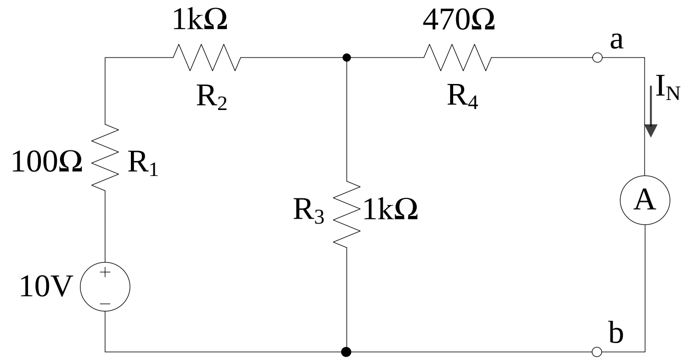


Figure 3: Finding I_N

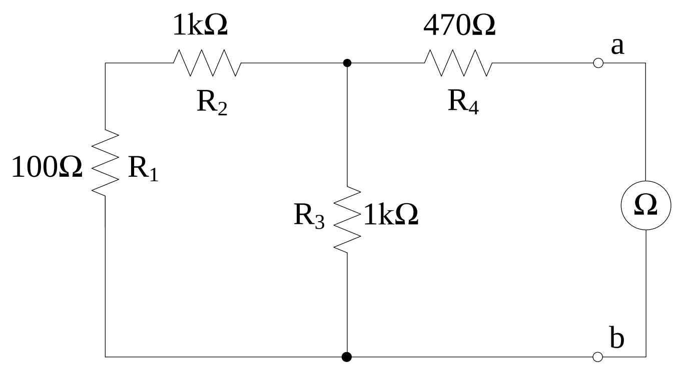


Figure 4: Finding R_N

Measurement Tables: (Part a)

Table 1

Norton Resistance RN (Ω)		**Norton Current IN (A)**	
Calculated	**Measured**	**Calculated**	**Measured**

Table 10.1

Table 2

RL(Ω)	**Current through load resistance IL(mA)**	
	Original circuit	**Norton Equivalent circuit**
470		
1000		

Table 10.2

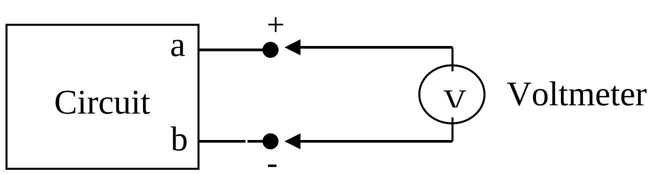
Task (1) (Measuring Thevenin Voltage VTH)

Task 1. Connect on breadboard the circuit shown in Fig. 10.6. The aim is to determine the current through and the voltage across the 1kΩ resistance R_1 connected between the two nodes A and B.

Task 2. Remove the load resistor R_1 from the circuit on breadboard.

Task 3. Calculate the voltage at nodes A and B (preferably using nodal analysis). Determine the open-circuit voltage (voltage between points A and B i.e. V_{AB}) using a voltmeter. The voltage V_{AB} is the Thevenin voltage calculated as V_{OC} (open-circuit voltage) = V_{TH} = V_{AB} = V_A-V_B. Record the value in Table 10.3.

Task 4. Repeat the experiment for terminals C-D and record the values.



Measurement Table 1

Terminals	Node Voltages VA/ VC(volts)			Node Voltages VB/ VD(volts)			VTH = VA-VB/ VC-VD (volts)
	Calculated Values	Measured Values	Simulated Values	Calculated Values	Measured Values	Simulated Values	
A – B							
C – D							

Table 10.3

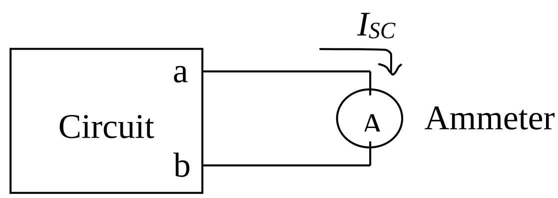
Task (2) (Calculating and Measuring RTH)

There are various methods to calculate the value of R_{TH}as described below.

Task 1. Method A: (By measuring ISC – short-circuit current)

Short the points 'a' and 'b', then measure the short-circuit current I_{SC} using an ammeter. Calculate R_{TH} using equation 10.3

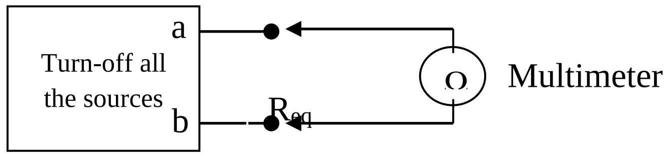
$$R_{TH} = V_{TH} / I_{SC} \tag{10.3}$$



Task 2. Method B: (By measuring the equivalent resistance)

Remove all sources from the circuit, i.e. replace all voltage sources with a short-circuit and current sources with an open-circuit. Then with the help of a multimeter find the resistance between the points 'a' and 'b', denoted by R_{eq}.

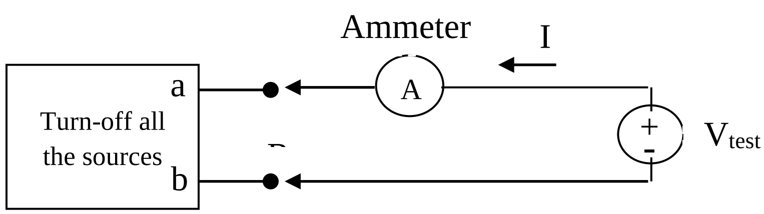
$$R_{TH} = R_{eq} \tag{10.4}$$



Task 3. Method C: (By applying known voltage/ test source and measuring source current)

Apply a known DC voltage (1V) between points A and B. Then using an ammeter find the current '*I*' through the test voltage source. Suppose 1 Vis applied as the test voltage then, use equation 10.5 to find R_{TH}.

$$R_{TH} = V_{test} / I \tag{10.5}$$

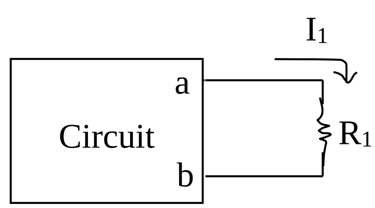


Task 4. Method D: (By inserting two different resistors and measuring current)

Insert resistors R_1 and R_2 (between points A and B) one by one, write expression for the current (measured) through the resistors as given below, and substitute values in equations (10.6)-(10.7).Simultaneously solving these two resultant equations will give the value of R_{TH}.

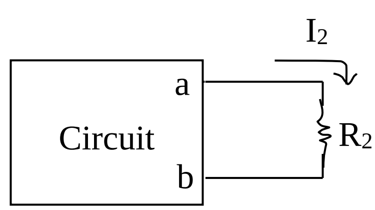
Current through the resistor R_1 :-

$$I_1 = V_{TH} / (R_{TH} + R_1) \tag{10.6}$$



Current through the resistor R_2 :-

$$I_2 = V_{TH} / (R_{TH} + R_2) \tag{10.7}$$



Simultaneously solving equations (10.6)-(10.7) will give the value of R_{TH}.

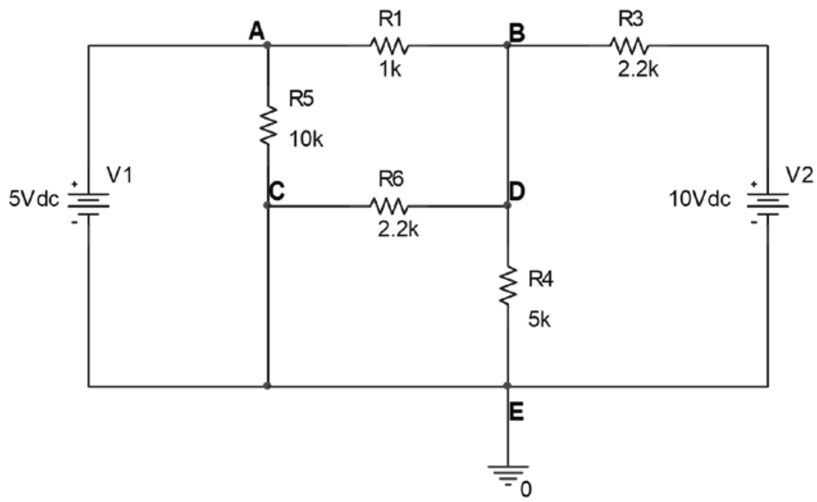


Figure 5: Example circuit to test the validity of the Thevenin's theorem

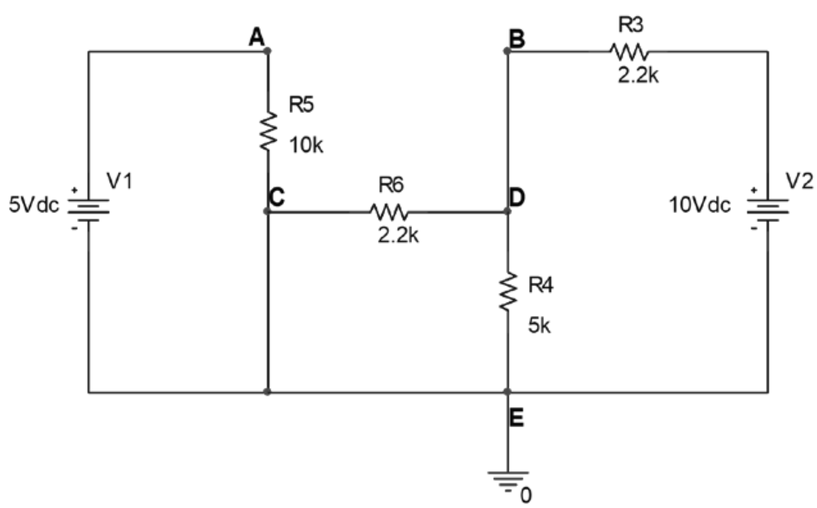


Figure 6: Finding V_{TH}

1. Determine the Thevenin resistance at terminals A-B and C-D using the four methods mentioned above. Fill the table 10.4 with calculated and measured values.
2. Compare the values of R_{TH} being found using different methods. Do they agree?

Measurement Table 2

Terminals	Thevenin Resistance RTH(Ω)				
	Calculated Values	Measured Values			
		Method A	Method B	Method C	Method D
A - B					
C - D					

Table 10.4

Task (3) (Simulation Task)

Task 1. Simulate the original circuit shown in figure 10.6 and note the value of current through and voltage across the load resistance w.r.t. terminals A-B and C-D (denote them by $\mathbf{I_{L'}}$ and $\mathbf{V_{L'}}$).

Task 2. Simulate the Thevenin equivalent circuit consisting of voltage source '$\mathbf{V_{TH}}$', resistance '$\mathbf{R_{TH}}$' and load resistance. Note the value of current through and voltage across the load resistance w.r.t. terminals A-B and C-D. (denote them by $\mathbf{I_L}$ and $\mathbf{V_L}$). Record the values in table 10.5.

Task 3. Compare the corresponding values.

Measurement Table 3

Terminals	Current through load resistance (mA)		Voltage across load resistance (volts)	
	IL	IL'	VL	VL'
A-B				
C-D				

Table 10.5

Post Lab

Questions:

1) What is the theoretical and practical advantage of using Thevenin and Norton theorem in circuit analysis?

2) Which theorem is typically used to determine the range of output voltages for a series-parallel circuit with a variable load?

3) A circuit has a voltage source of 15 volts and three 15 Ω resistors connected in parallel across the source. What Thevenin resistance (RTH) would a load "see" when connected to this circuit?

4) Which theorem could be used (along with Ohm's law) to calculate the bridge current in a Wheatstone bridge?

5) Briefly describe a practical application of Norton theorem.

6) Which of the following statements is true?
    a. RN acts as a voltage divider with the load resistance
    b. RN acts like a current divider with the load resistance.
    c. RTH acts like a current divider with the load resistance

Critical Analysis / Conclusion

Lab Assessment		
Pre Lab	/5	/25
Performance	/5	
Results	/5	
Viva	/5	
Critical Analysis	/5	
Instructor Signature and Comments		

Lab9 : Natural Response of an RC Circuit

Pre Lab

Oscilloscope

We should be familiar to the following four things about oscilloscope.

- What does an oscilloscope do?
- How does it work?
- Setting Up
- Other Controls

An oscilloscope is easily the most useful instrument available for testing circuits because it allows you to see the signals at different points in the circuit. The best way of investigating an electronic system is to monitor signals at the input and output of each system block, checking that each block is operating as expected and is correctly linked to the next. With a little practice, you will be able to find and correct faults quickly and accurately. An oscilloscope is an impressive piece of kit.

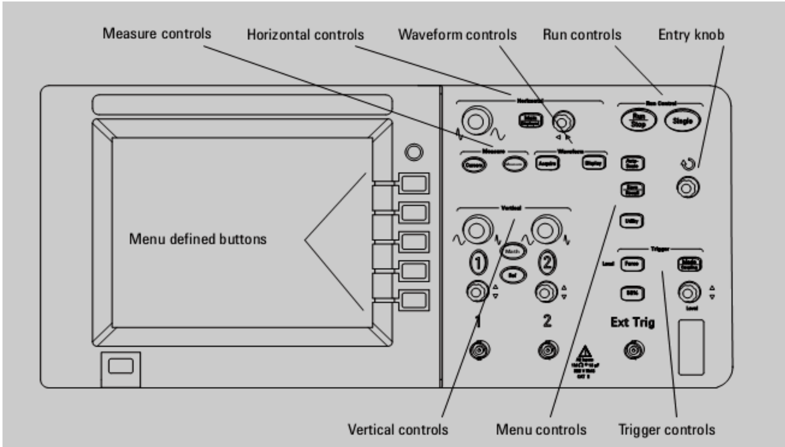
Digital Oscilloscope:

The digital oscilloscope is an indispensable tool for anyone designing, manufacturing or repairing electronic equipment. In today's fast-paced world, engineers need the best tools available to solve their measurement challenges quickly and accurately. As the eyes of the engineer, digital oscilloscopes are the key to meeting today's demanding measurement challenges.

Digital oscilloscopes are used by everyone from physicists to repair technicians. An automotive engineer uses a digital oscilloscope to correlate analog data from sensors with serial data from the engine control unit. A medical researcher uses a digital oscilloscope to measure brain waves.

Front Panel Controls:

The front panel has knobs and buttons. Knobs are used most often to make adjustments. Buttons are used for run controls and to change other oscilloscope settings via menus.



The definitions of the buttons and the knobs are as follows:

Measure controls: Cursors and Measure menu buttons.

Waveform controls: Acquire and Display menu buttons.

Menu controls: Save/Recall and Utility menu buttons.

Vertical controls: Vertical position knobs, vertical scale knobs, channel (1,2) Math, and Ref menu buttons.

Horizontal controls: Position knob, Main/Delayed menu button, and scale knob.

Trigger controls : Trigger Level knob, 50% , Mode/Coupling, and Force buttons.

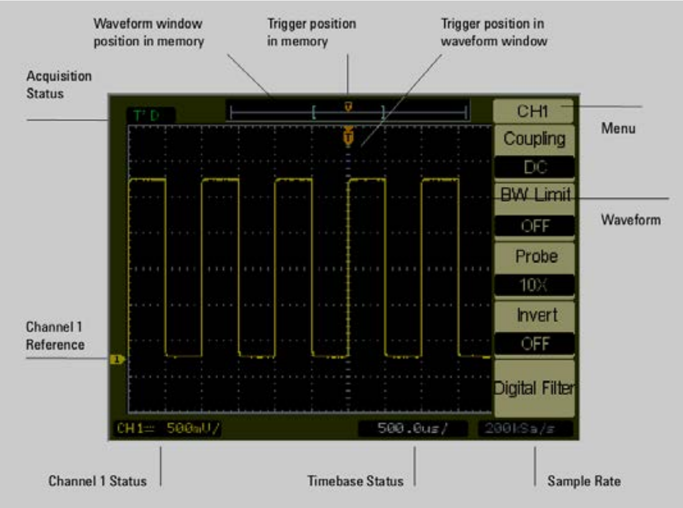
Run controls: Run/Stop, Single, and Auto-Scale buttons.

Menu defined buttons: Five gray buttons from top to bottom on the right-hand side of the screen, which select the adjacent menu items in the currently displayed menu.

Entry knob : For the adjustment defined contr oscilloscope display

Oscilloscope display:

The small description of Oscilloscope display as below



Many of the controls of the oscilloscope allow you to change the vertical or horizontal scales of the *V*/*t* graph, so that you can display a clear picture of the signal you want to investigate. 'Dual channel ' oscilloscopes display two *V*/*t* graphs at the same time, so that simultaneous signals from different parts of an electronic system can be compared. The auto scale button is use for calibration of Oscilloscope.

Auto- scale Button :

The oscilloscope has an auto- scale feature that automatically sets the oscilloscope controls for the input waveforms present.

Auto- scale requires waveforms with a frequency greater than or equal to 50 Hz and a duty cycle greater than 1%.

1. Press Auto-Scale.
2. The oscilloscope turns on all channels that have waveforms applied and sets the vertical and horizontal scales appropriately. It also selects a time base range based on the trigger source.

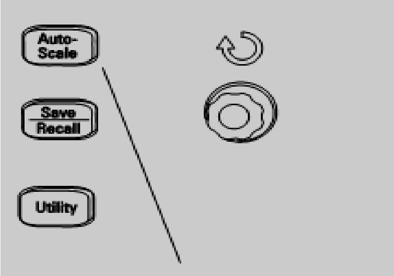


Figure 6 Auto-Scale Button

The trigger source selected is the lowest numbered channel that has a waveform applied

Low Frequency Compensation and Calibration of Oscilloscope:

For the compensation and calibration of Digital Scope we have to

1. Set the Probe hooktip to the 10x using menu , ensure a proper connection by firmly inserting the tip onto the probe.
2. Attach the probe tip to the probe compensation connector and the ground lead to the probe compensator ground connector.
3. Press the Auto-Scale front panel button.
4. compensation adjustment on the probe for the flattest square wave possible

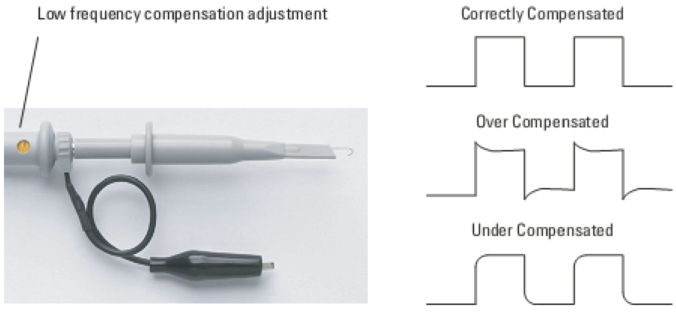


Figure 7 Low Frequency Probe Compensation

Function Generator:

What is a function generator?

A function generator is a device that can produce various patterns of voltage at a variety of frequencies and amplitudes. It is used to test the response of circuits to common input signals. The electrical leads from the device are attached to the ground and signal input terminals of the device under test.

Features and controls

Most function generators allow the user to choose the shape of the output from a small number of options.

- Square wave - The signal goes directly from high to low voltage.
- Sine wave - The signal curves like a sinusoid from high to low voltage.
- Triangle wave - The signal goes from high to low voltage at a fixed rate.

The amplitude control on a function generator varies the voltage difference between the high and low voltage of the output signal.

The direct current (DC) offset control on a function generator varies the average voltage of a signal relative to the ground.

The frequency control of a function generator controls the rate at which output signal oscillates. On some function generators, the frequency control is a combination of different controls. One set of controls chooses the broad frequency range (order of magnitude) and the other selects the precise frequency. This allows the function generator to handle the enormous variation in frequency scale needed for signals.

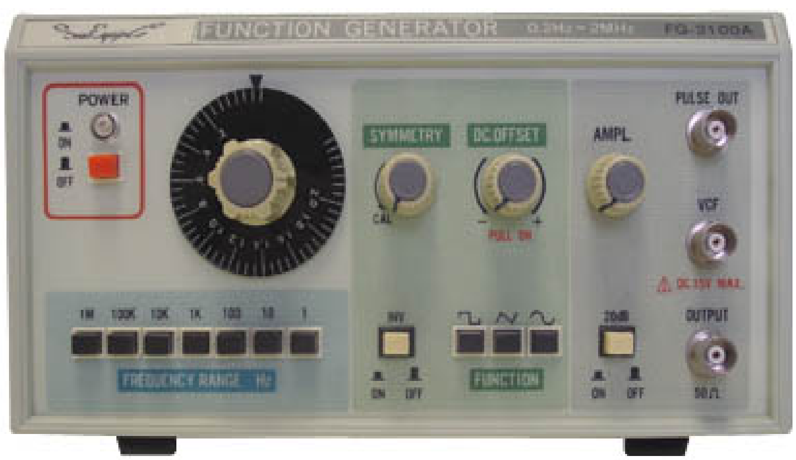


Figure 1: Function Generators FG-2100A

The duty cycle of a signal refers to the ratio of high voltage to low voltage time in a square wave signal.

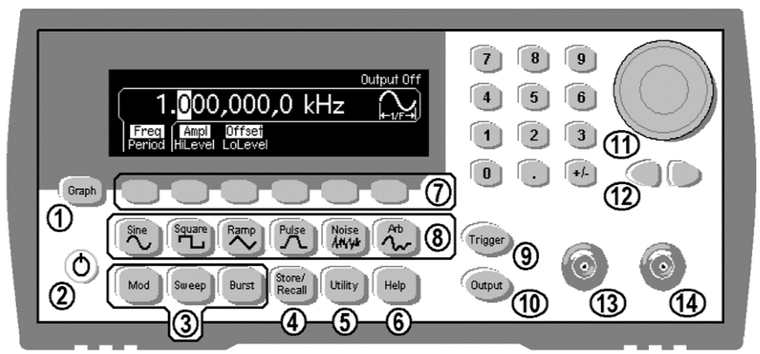
How to use a function generator

After powering on the function generator, the output signal needs to be configured to the desired shape. Typically, this means connecting the signal and ground leads to an oscilloscope to check the controls. Adjust the function generator until the output signal is correct, then attach the signal and ground leads from the function generator to the input and ground of the device under test. For some applications, the negative lead of the function generator should attach to a negative input of the device, but usually attaching to ground is sufficient.

Following is the diagram and specifications of the function generator, which will be used by the students in the laboratory.

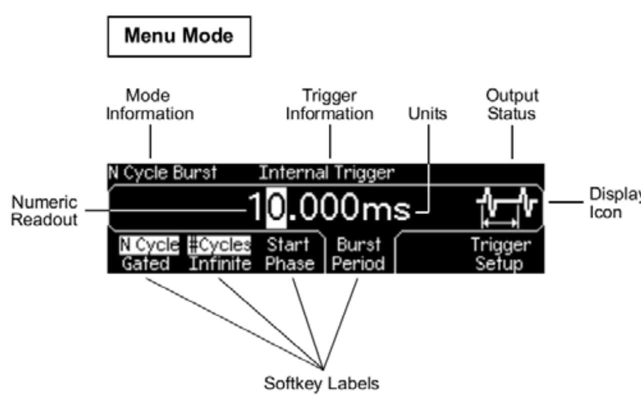
Description:

The FG-2100A Function Generator produces a variety of waveforms, including sine, square, triangle, ramp and pulse signal over a frequency range of 0.2Hz to 2MHz. The unit can be used for diverse applications for frequency characteristic measurement of audio/video equipment and the testing of automatic control devices.

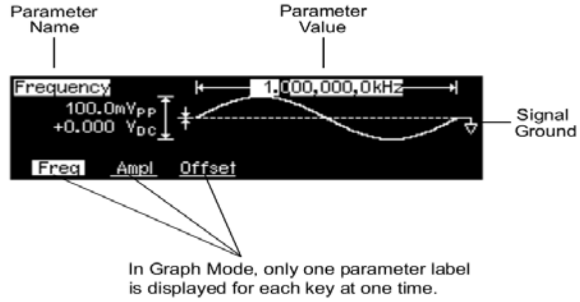


Agilent Function Generator:

1. Graph Mode/Local Key
2. On/Off Switch
3. Modulation/Sweep/Burst Keys
4. State Storage Menu Key
5. Utility Menu Key
6. Help Menu Key
7. Menu Operation Softkeys
8. Waveform Selection Keys
9. Manual Trigger Key (used forSweep and Burst only)
10. Output Enable/Disable Key
11. Knob
12. Cursor Keys
13. Sync Connector
14. Output Connect

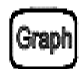


The Front-Panel Display at a Glance:



Agilent function generator has two modes for display

1. Menu Mode
2. Graphic Mode

To enter or exit the Graph Mode, press 

the key

Front-Panel Number Entry:

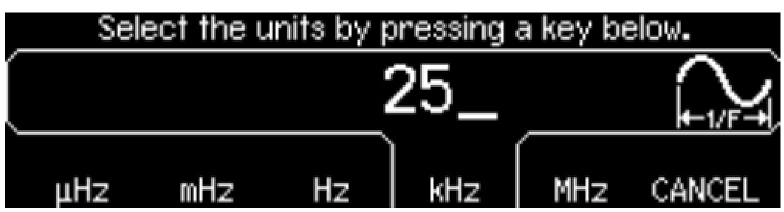
You can enter numbers from the front-panel using one of two methods.

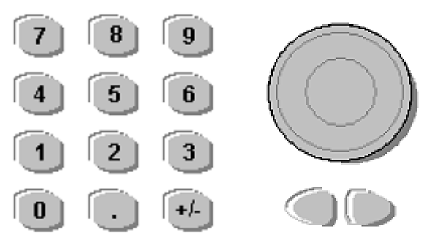
Use the knob and cursor keys to modify the displayed number.

1) Use the keys below the knob to move the cursor left or right.
2) Rotate the knob to change a digit (clockwise to increase).

Use the keypad to enter numbers and the soft keys to select units.

1) Key in a value as you would on a typical calculator.
2) Select a unit to enter the value.





Theory of Natural Response of RC Circuits:

Any circuit configuration that consists only of resistors and capacitors is known as an RC circuit. An example RC circuit is shown below. 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.

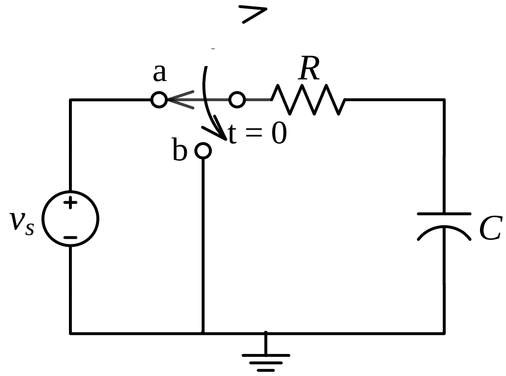


Figure 2: An example RC circuit (t < 0)

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 buildup 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, current, and charge dissipate exponentially in time. For t ≥ 0, the above circuit is reduced to:

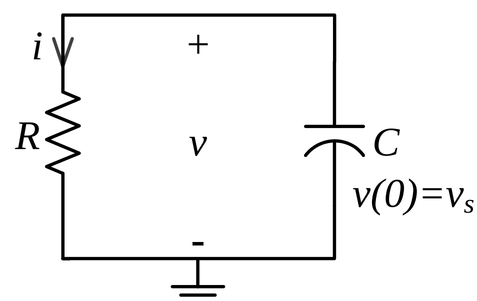


Figure 3: Reduced RC circuit (t ≥ 0)

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

$$C\frac{dv}{dt}+\frac{v}{R}=0 \qquad (11.1)$$

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

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

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**.

Time Constant

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 ⅜.

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

$$\tau = RC \qquad (11.4)$$

In other words, when a time interval equaling the time constant has passed, the voltage across the capacitor is ⅜ 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.

In Lab

Objective

- To experimentally measure time constant of an RC circuit and compare with the theoretical expression of it.
- To analyze the behavior of a first-order RC circuit.
- To familiarize students with the usage of oscilloscope to make voltage measurements.

Equipment Required

Resistor (1kΩ), capacitor (1µF), oscilloscope, function generator, breadboard, and connecting wires.

Knowledge Level

- Before working on this lab, students should have good understanding of the concept of behavior of capacitors and time constant of RC circuit.
- Students should be able to theoretically solve the source free RC circuits.

Task (1) (Analyzing natural response of RC circuit)

Task 1. Consider the circuit diagram shown in figure 11.3. In this task we will use AC voltage source instead of DC source. During positive value of voltage, capacitor will be charged, while capacitor will be discharged during the negative (0V) cycle. Switch is eliminated in this way. Function generator is used to provide AC voltage, while charging and discharging behavior of capacitor (in terms of exponential rise or decay in voltage) can be observed simultaneously on the oscilloscope.

Task 2. Switch ON the function generator and the oscilloscope.

Task 3. The vertical axis on oscilloscope represents voltage and the horizontal axis represents time. Make sure that the trace of the oscilloscope (yellow line) is correctly set at 0.00 *divs*. The vertical position of the trace is established by adjusting the associated knob.

Task 4. Attach the BNC adapter cable to the function generator output and CH1 of the oscilloscope.

Task 5. Adjust the function generator to generate a square wave that is used at the input to analyze the natural response of an RC circuit. The parameters of this square wave are adjusted using function generator as:

- **Amplitude: 2V peak to peak**
- **Frequency: 100 Hz**
- **Duty cycle: 50%**
- **DC Offset: 1 volt**

Task 6. On the oscilloscope, press Auto Scale.

Task 7. Press channel (1) button, make sure from the menu that coupling is DC, bandwidth limit is off and probe is set at (1X).

Task 8. Play with the horizontal and vertical position and scale knobs and try to understand their effect. Finally, auto-scale again.

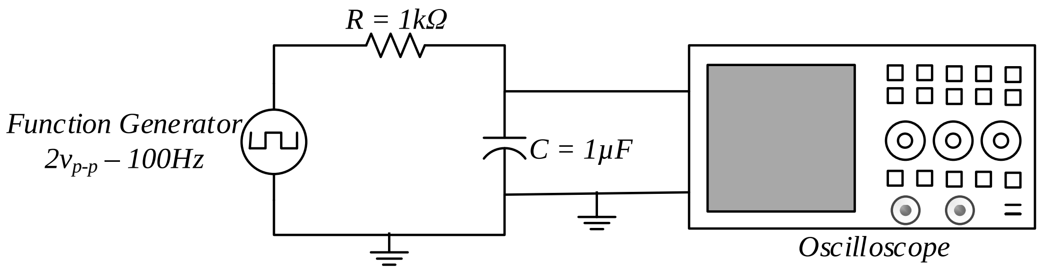


Figure 4: Circuit diagram for RC circuit with Oscilloscope

Task 9. Note down the values of Channel 1 Status and Time base Status. Interpret the graph displayed using these values.

Task 10. Connect on breadboard the circuit arrangement as shown in figure 11.3.

Task 11. Feed the function generator output to CH1 of the oscilloscope whereas the voltage drop across the capacitor is fed to CH2 of the oscilloscope.

Task 12. Record the values of channel 1 and channel 2 status and time base status in table.

Measurement Table 1:

CH1 Status	Time Base Status (CH1)	CH2 Status	Time Base Status (CH2)

Task (2) (Calculating and Measuring Time Constant of circuit)

Task 1. Adjust the HORIZONTAL POSITION control on the oscilloscope so that the cycle begins at an initial time of zero. An example trace is shown in figure 11.4.

Task 2. Notice that the voltage across the capacitor, in the above figure, decays through four units along the vertical; each of the four units can be divided into five divisions for a total of twenty divisions. The value at 7.5 divisions (along the vertical) marks the point where the voltage across the capacitor is ⅜ of the initial voltage.

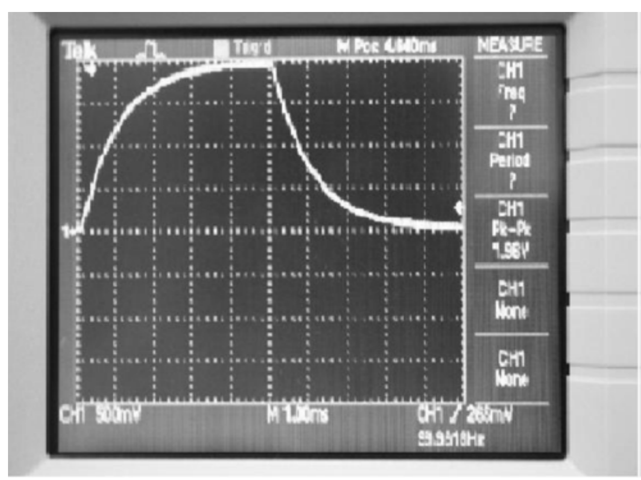


Figure 5: An example trace of the voltage across a capacitor in an RC circuit

$$\text{⅜ (total no. of divisions on vertical axis) = x divisions} \qquad \textbf{(11.5)}$$

Task 3. Measure and record the time constant τ corresponding to the initial voltage at *x* divisions. The experimental time constant is determined from the number of divisions along the horizontal axis where the capacitor voltage drops to 36.8% of its initial value i.e. at *x* divisions on the vertical axis.

$$\tau_{exp} = \frac{no\ of\ divisions\ along\ horizontal\ exis}{5}\ (time\ scale\ value) \qquad \textbf{(11.6)}$$

Task 4. Calculate the theoretical value of the time constant using values of resistance and capacitance.

$$\tau_{theo} = RC \qquad \textbf{(11.7)}$$

Task 5. Calculate the percent difference between the experimental and theoretical values for the time constant.

Task 6. **Repeat the task 2 for R = 2.2kΩ.**

Measurement Table 2:

R (Ω)	C (µF)	τ (theoretical)	τ (experimental)	% difference
1KΩ				
2.2KΩ				

Task (3) (Calculating and measuring natural response of circuit)

Task 1. Write equation of the measured natural response of the experiment RC circuit:

$$v(t) = v(0)e^{-t/\tau} \quad , \quad t \geq 0 \qquad (11.8)$$

Task 2. Calculate the natural response of the experiment RC circuit. Compare it with the measured response. Do they agree?

Repeat the task 3 for R = 2.2kΩ.

Measurement Table 3:

R=______ **C=______**	**Calculated: __________________________________** **Measured: __________________________________**
R=______ **C=______**	**Calculated: __________________________________** **Measured: __________________________________**

Post Lab

Questions?

1. What is duty cycle?

2. How a square wave having duty cycle of 25% would look like?

3. What do you mean by calibrating the oscilloscope?

4. Which parameters of square wave can be adjusted using function generator?

Critical Analysis / Conclusion

Lab Assessment		
Pre Lab	/5	/25
Performance	/5	
Results	/5	
Viva	/5	
Critical Analysis	/5	
Instructor Signature and Comments		

Lab 10: To analyze the sinusoidal response of RLC Circuit out of course

Pre Lab

Calculation of Square Wave response of RLC Circuit

I. Assemble the circuit shown in Figure 2 on the breadboard.With values R= 220Ω, C= 3.3uH, L= 5mH and f = 20 kHz, observe the waveform when the output is taken across the resistor. *You will use the function generator to provide* ***a square wave*** *a.c voltage of* ***6V peak to peak****, and the oscilloscope to measure the output.*

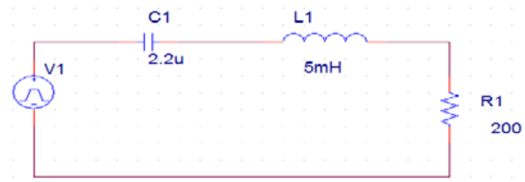


Figure 1

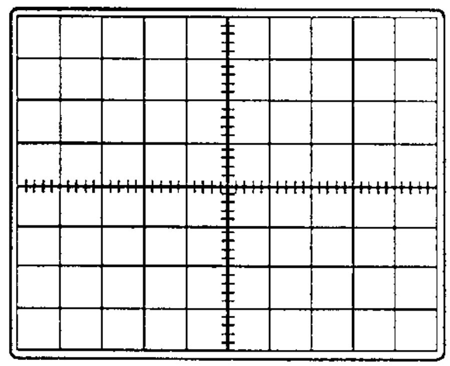


Figure 2

II. Sketch the waveform you observed on the oscilloscope on the graph given.

III. Simulate your circuit in Ltpice, select **New Simulation** and select **Bias Point (Transient) Analysis** in the simulation menu. Run to simulation to a 100ms. *To provide a square wave at the input use the VPULSEpart* from the part menu. Set the values as shown in figure 12.5. Save your simulation on the desktop.

τ and T1/2

The product of the resistance and capacitance, *RC*, governs the time scale with which the changes take place. For this reason it is called the time constant, which we call **τ (tau)**. It can be found indirectly by measuring the time required for the voltage to fall to *V*o/2 (see Figure below). This time interval is called the half-life, $T_{1/2}$, and is given by the equation $T_{1/2}$ = **(ln2)τ**, so**τ** = $T_{1/2}$ **/ln2 =** $T_{1/2}$ **/(0.693).**

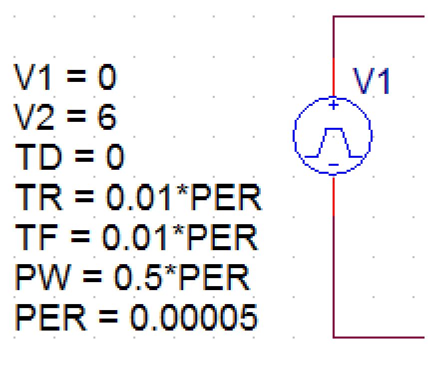


Figure 3

TECHNIQUE FOR FINDING HALF-LIFE

Here is a method for finding *T*1/2.

- Change oscilloscope gain (volts/cm) and sweep rate (ms/cm) until you have a large pattern on the screen. Make sure the sweep speed is in the "calibrated" position so the time can be read off the *x*-axis.
- Center the pattern on the screen so that the horizontal axis is in the center of the pattern. That is, so that the waveform extends equal distances above and below the axis.
- Move the waveform to the right until the start of the discharge of the capacitor is on the vertical axis (Figure 6b). You may find it helpful to expand, or magnify, the trace. The sweep time is now a factor of five or ten faster than indicated on the dial. Ask your instructor for details.

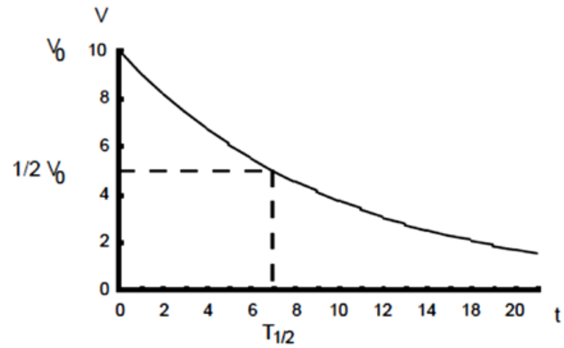


Figure 4

- The half-life, *T*1/2 is just the distance shown on Figure below.

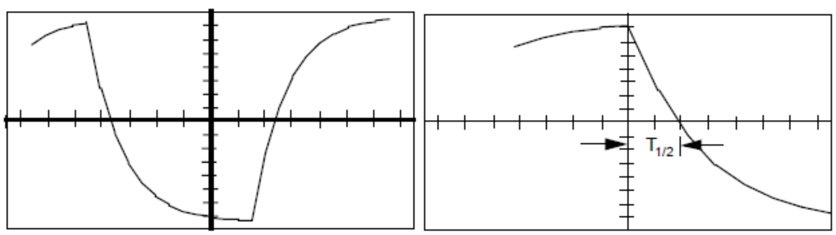


Figure 5

In Lab

Objective

- To investigate the behavior of circuits containing combinations of resistors, capacitors and inductors
- To observe the resultant waveforms using an oscilloscope
- Measure $T_{1/2}$ and τ of the RLC circuits

Equipment

The equipment to be used is listed below; however, the students are advised to also consult the user manuals for each piece of equipment.

Digital multi-meter (DMM), Dual channel oscilloscope, Function generator, LCR Meter, Breadboard, Resistors: 200Ω, Capacitor:2.2µF, Inductor: 5mH

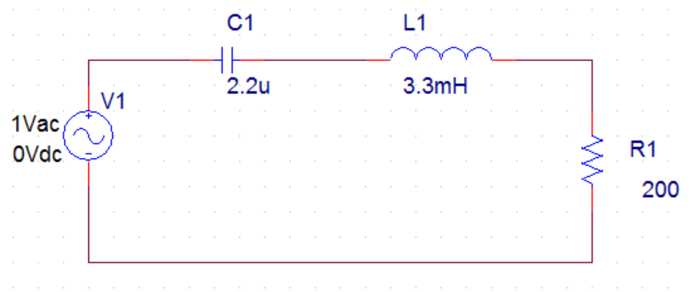


Figure 6

Calculation of Sinusoidal Response of RLC Circuit

Part - 1 Theoretical Analysis

Task 1. Using the LCR, measure the actual values of the circuit elements:

R = ________________, L = _______________, C = __________________

Task 2. For f = 30kHz, calculate circuit current I_s, and element voltages V_R, V_L and V_C.

2.1. $\omega = 2\pi f$ = ________________________

2.2. $R = Z_R$= ________________________

2.3. $X_L = Z_L = j\omega L$ ________________________ = _______________

2.4. $X_C = Z_C = 1/j\omega C$ ________________________ = _______________

2.5. $R+J(X_L - X_C) = Z_{in} = Z_R + Z_L + Z_C$ = ________________________ = ______

Task 3. For $V = 1/0^o$ volts

3.1. $I_S = V/Z_{in}$ = _____________/________________ = ______________________

3.2. $V_R = I_S \times Z_R$ = ____________________________= ______________________

3.3. $V_L = I_S \times Z_L$ = ___________________________ = ______________________

3.4. $V_C = I_S \times Z_C$ = ___________________________ = ______________________

Part -2 Practical Measurement

Task 4. Using the bread-board, connect the circuit elements ensuring that no connection is loose. Connect Ch 1 of the oscilloscope across the circuit input (i.e. function generator)

4.1. Using the oscilloscope, ensure that the source output is 1V amplitude (2VP-P) at 30 Khz.

4.2. Using multimeter, measure the source voltage. Note that the multimeter reads rms voltage = 1/1.414 = 0.707 volts.

4.3. Connect Ch 2 of the oscilloscope across the resistor and select “alt” on the display selection. Make the appropriate selections for voltage and time scales.
4.4. Measure the amplitude of VR = ________. Measure VR,rms = ____________ using multimeter.
4.5. Now measure the horizontal distance between the two waveforms. It is best to observe the difference at the zero crossing of both signals. You need to ensure that the x-axis of both signals accurately overlaps. This can be done by selecting GND on both channels and adjusting the up-down knobs.
4.6. Calculate the phase difference between the two signals.
4.7. Phase angle = (measured distance / Time period) x 360o = ________________ = ___________________
4.8. Calculate circuit current I = VR/ZR = ___________/__________=______________
4.9. Measure VL = _____________ Phase angle = ______________ = ____________
4.10. Measure VC = _____________ Phase Angle = ________________ = __________

Post Lab

Questions:

1) What is the difference between underdamped and overdamped response?

2) What is the behavior of RLC circuit at low frequencies and at high frequencies?

3) What is the significance of phase in sinusoidal circuit analysis?

Critical Analysis / Conclusion

<table>
<tr><th colspan="3">Lab Assessment</th></tr>
<tr><td>Pre Lab</td><td>/5</td><td rowspan="5">/25</td></tr>
<tr><td>Performance</td><td>/5</td></tr>
<tr><td>Results</td><td>/5</td></tr>
<tr><td>Viva</td><td>/5</td></tr>
<tr><td>Critical Analysis</td><td>/5</td></tr>
<tr><td colspan="3">Instructor Signature and Comments</td></tr>
</table>

Open Ended Lab 1

Lab 11: Validation of Mesh Analysis Method

Pre-Lab Task

Students should solve the circuit shown in figure 8.1 using mesh analysis and calculate all the mesh currents. The values of currents through and voltages across all the resistances should also be calculated.

In Lab

Objective

- To implement the Mesh Analysis method in lab
- To verify that measured values of branch voltages and currents are comparable to the values obtained by solving the circuit using mesh analysis method

Task 1:

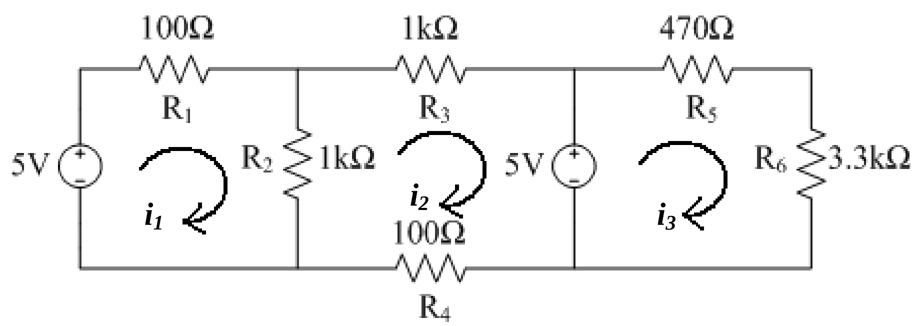


Table:

Resistance values(Ω)	Calculated results		Measurement results		Simulation Results		%age Difference between calculated and measured values
	Voltage (V)	Current (A)	Voltage (V)	Current (A)	Voltage (V)	Current (A)	
R_1 =							
R_2 =							

R_3 =							
R_4 =							
R_5 =							
R_6 =							

Post lab

Questions:

1) What is the difference between mesh and loop?
2) What is super mesh? Does the circuit in figure 8.1 contain any super mesh?

Opend Ended Lab2

Lab 12: Superposition Theorem

Pre-Lab Task

Students should solve the circuit shown in figure 9.1 using Superposition theorem. Students should calculate currents and voltages for resistances R_2 and $R_4$and also power dissipated by them.

In Lab

Objectives:

- To verify the superposition theorem experimentally for resistances
- To verify that superposition theorem does not apply to power (nonlinear quantity

Task 1:

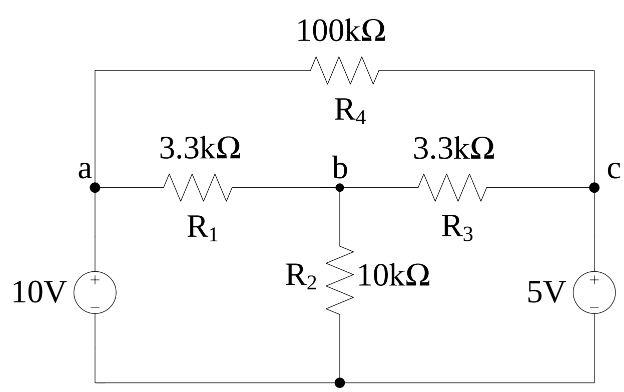


Figure 9.1: Example circuit to test the validity of the superposition theorem

Measurement Tables 1

Voltage / Current	R2=10KΩ			R4=100KΩ		
	Calculated results	Measured results	Simulation results	Calculated results	Measured results	Simulation results
v1 =						
v2 =						

v =						
i1 =						
i2 =						
i =						

Post Lab

Questions:

1. The Superposition Theorem works nicely to calculate voltages and currents in resistor circuits. But can it be used to calculate power dissipations as well? Why or why not?

2. What advantages and disadvantages are associated with Superposition Theorem?

3. How are dependent voltage and current sources dealt with when using Superposition Theorem?

4. What is power rating of resistor? A resistor sometimes becomes hot while connected in the circuit. What can be the reason for this?

Critical Analysis / Conclusion

Lab Assessment		
Pre Lab	/5	/25
Performance	/5	
Results	/5	
Viva	/5	
Critical Analysis	/5	
Instructor Signature and Comments		