VIT-AP UNIVERSITY

Fundamentals of Electrical and Electronics Engineering Lab

Name : Arka Lodh

Regn No. : 20BCE7349

INDEX

[1] **Date : 16/01/2021**

Name : Arka Lodh

ID no. : 20BCE7349

<u>Experiment 1</u>

<u>ANALYSIS OF RESISTANCE USING COLOUR CODING</u>

Objectives :

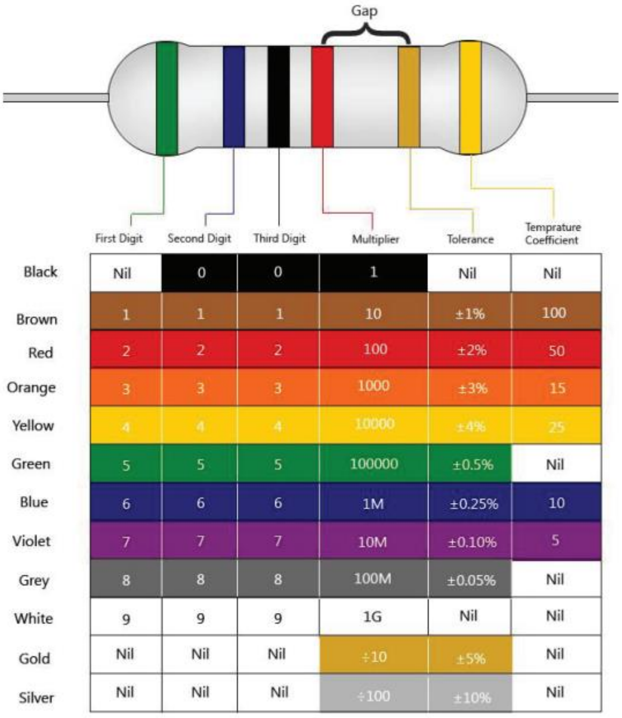
1. To learn Resistor colour code
2. To determine the stated value of a resistor by interpreting the colour code indicated on the resistor.

Software Used : Ni Multisim 14.0

Theory : The resistors used in laboratory are carbon composition resistors, consisting of graphite or some other type of carbon embedded in a filler material. Graphite is a moderately good conductor, so by varying the graphite-filler mix, a large range of resistance values can be obtained (the less graphite, the higher the resistance) [1]. Carbon resistors are cheap and reliable, however, their tolerances (5 to 20% deviation from nominal values) indicate that larger errors can be expected. Other types of resistors include wire wound, metal film, and carbon film [1]. The nominal value and tolerance of a carbon resistor can be determined from the color-coded stripes that appear on the resistor. The first three bands represent the two significant figures of the resistance, while the fourth band indicated the number of zeros that follow. If there are only three bands, the resistor has a 20% tolerance. If the three colour bands are followed by a silver band, the resistor has a 10% tolerance. A gold band following the three colour bands indicates a 5% tolerance, a red band indicates a 2% tolerance, and a brown band indicates a 1% tolerance.

[2]

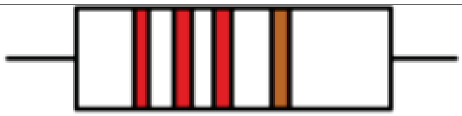
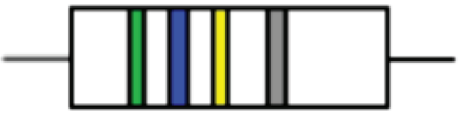
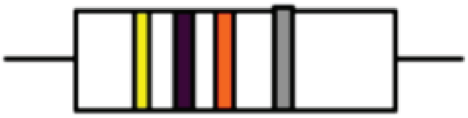
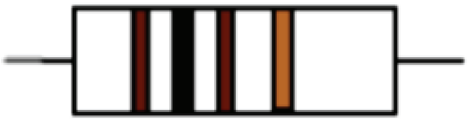
Circuit Diagram :



	First Digit	Second Digit	Third Digit	Multiplier	Tolerance	Temprature Coefficient
Black	Nil	0	0	1	Nil	Nil
Brown	1	1	1	10	±1%	100
Red	2	2	2	100	±2%	50
Orange	3	3	3	1000	±3%	15
Yellow	4	4	4	10000	±4%	25
Green	5	5	5	100000	±0.5%	Nil
Blue	6	6	6	1M	±0.25%	10
Violet	7	7	7	10M	±0.10%	5
Grey	8	8	8	100M	±0.05%	Nil
White	9	9	9	1G	Nil	Nil
Gold	Nil	Nil	Nil	÷10	±5%	Nil
Silver	Nil	Nil	Nil	÷100	±10%	Nil

Figure 1: Colour code for a resistor.

Result & Observation :

	Resistance Value	Tolerance
	$22 \times 10^{2} \Omega$	±5%
	$56 \times 10^{4}\ \Omega$	±10%
	$47 \times 10^{3}\ \Omega$	±10%
	$10 \times 10^{1}\ \Omega$	±5%

Therefore we are able to deduce the resistance of a resistor from the colour code.

 Date : 19/01/2021

Experiment – 2

VERIFICATION OF OHM'S LAW

Objective : To verify Ohm's Law

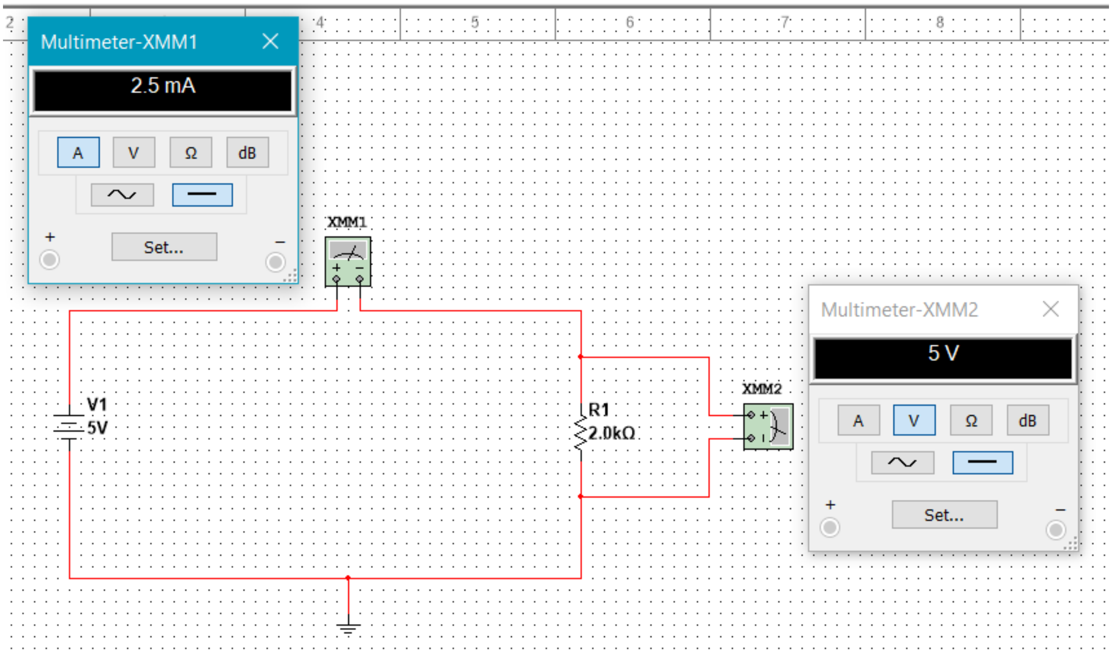
Software Used : NI Multisim 14.0

Theory : Georg Ohm found that, at a constant temperature, the electrical current flowing through a fixed linear resistance is directly proportional to the voltage applied across it, and also inversely proportional to the resistance.

Ohm's Law expressed of a simple equation, describing how voltage current and resistance interrelate.

V=IR is the equation for Ohm's Law

Circuit Diagram :



Result & Observations :

			Theoretical		Experimental	
S. No.	Source Voltage (V)	Resistance (kΩ)	Current (A)	Voltage Out(V)	Current	Voltage Out(V)
1	5	1	5×10^{-3}	5	5mA	5
2	5	2	2.5×10^{-3}	5	2.5mA	5
3	5	3	0.1×10^{-3}	3	1mA	3
4	5	4	0.5×10^{-3}	2	500μA	2
5	5	5	0.4×10^{-3}	2	400μA	2

Hence we can see that the theoretical and experimental values match. Therefore Ohm's Law has been verified.

[4] Date : 19/01/2021

Experiment – 3

VERIFICATION OF KCL AND KVL

Objective : To verify Kirchhoff's Current Law and Kirchhoff's Voltage Law

Software Used : NI Multisim 14.0

Theory : Gustav Robert Kirchhoff stated 2 basic laws concerning the relationship between currents and voltages in an electrical network. These laws are called Kirchhoff's Laws.

Kirchhoff's Voltage Law : This is based on the principle of conservation of energy. Mathematically KVL states that the algebraic sum of all voltages around a closed path is zero.

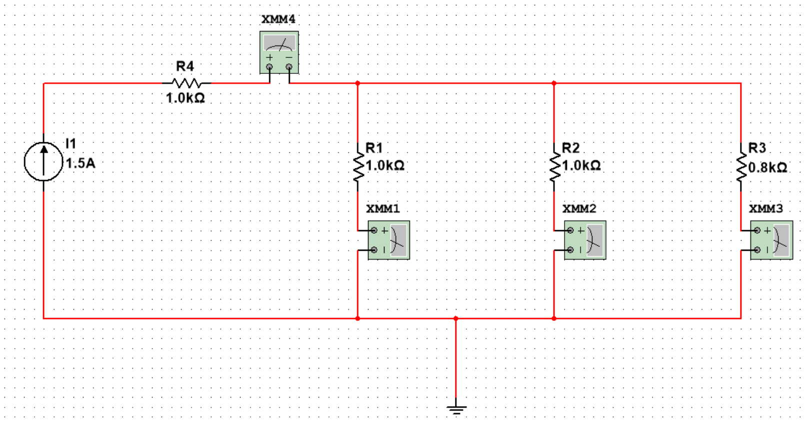
Sum of voltage drops = Sum of voltage gains

Kirchhoff's Current Law : This is based on the principal of conservation of charge. KCL states that the algebraic sum of currents entering a node is zero.

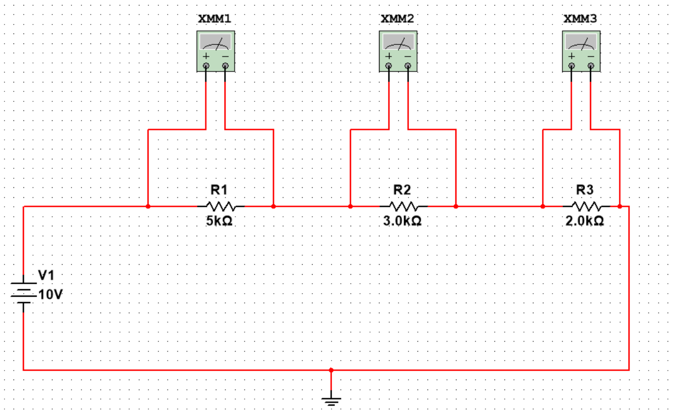
The sum of currents entering the node = The sum of currents leaving the node.

Circuit Diagram :

1) **KCL :**



2) **KVL :**



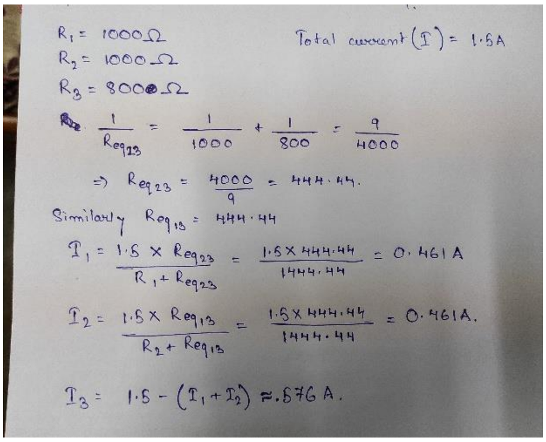
[5]

Result & Observations :

1) Experimental Result :

The current through R1 = 1.0k ohm is 461.538mA
The current through R2 = 1.0k ohm is 461.538mA
The current through R3 = 0.8k ohm is 576.923mA

Theoretical Result :



$R_1 = 1000\Omega$

$R_2 = 1000\Omega$

$R_3 = 800\Omega$

Total current $(I) = 1.5A$

$$\frac{1}{R_{eq23}} = \frac{1}{1000} + \frac{1}{800} = \frac{9}{4000}$$

$$\Rightarrow R_{eq23} = \frac{4000}{9} = 444.44.$$

Similarly $R_{eq13} = 444.44$

$$I_1 = \frac{1.5 \times R_{eq23}}{R_1 + R_{eq23}} = \frac{1.5 \times 444.44}{1444.44} = 0.461\,A$$

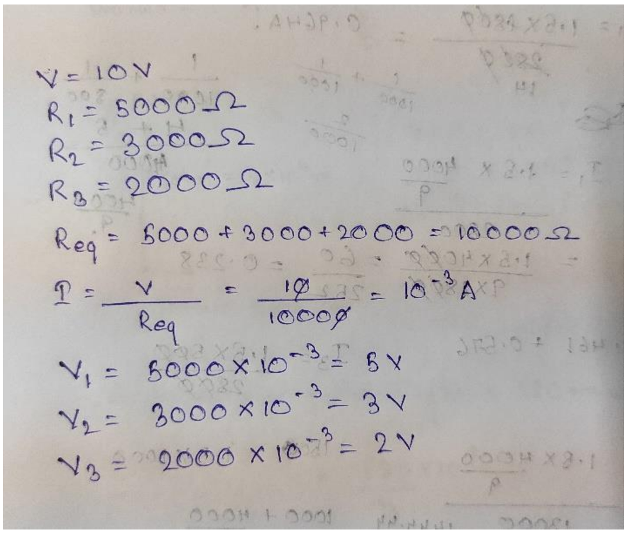
$$I_2 = \frac{1.5 \times R_{eq13}}{R_2 + R_{eq13}} = \frac{1.5 \times 444.44}{1444.44} = 0.461A.$$

$$I_3 = 1.5 - (I_1 + I_2) \approx .576\,A.$$

2) Experimental Result :

The voltage across R1 = 5k ohm is 5V
The voltage across R2 = 3k ohm is 3V
The voltage across R3 = 2k ohm is 2V

Theoretical Result :



$V = 10V$

$R_1 = 5000\Omega$

$R_2 = 3000\Omega$

$R_3 = 2000\Omega$

$R_{eq} = 5000 + 3000 + 2000 = 10000\Omega$

$$I = \frac{V}{R_{eq}} = \frac{10}{10000} = 10^{-3}A$$

$V_1 = 5000 \times 10^{-3} = 5V$

$V_2 = 3000 \times 10^{-3} = 3V$

$V_3 = 2000 \times 10^{-3} = 2V$

Therefore we can see that theoretical and experimental results match. Hence KCL and KVL are verified.

 Date : 20/01/2021

Experiment – 4

VERIFICATION OF CURRENT AND VOLTAGE DIVISION RULE

Objective : To verify current and voltage division rule

Software Used : NI Multisim 14.0

Theory :

Voltage Division Rule : The voltage is divided between two series resistors in direct proportion to their resistance. In a series circuit, voltage is divided, whereas the current remains the same.

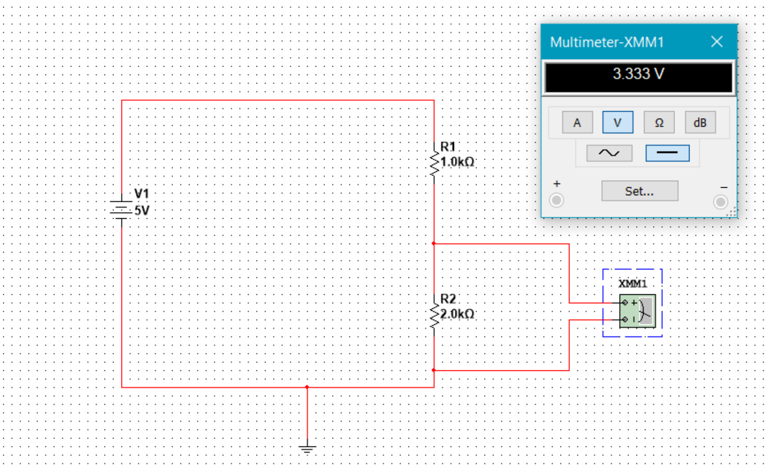
$V_{out} = (V_s * R_2)/(R_1 + R_2)$

Current Division Rule : A parallel circuit acts as a current divider as the current divides in all the branches in a parallel circuit, and the voltage remains the same across them. The current division rule determines the current across the circuit impedance.

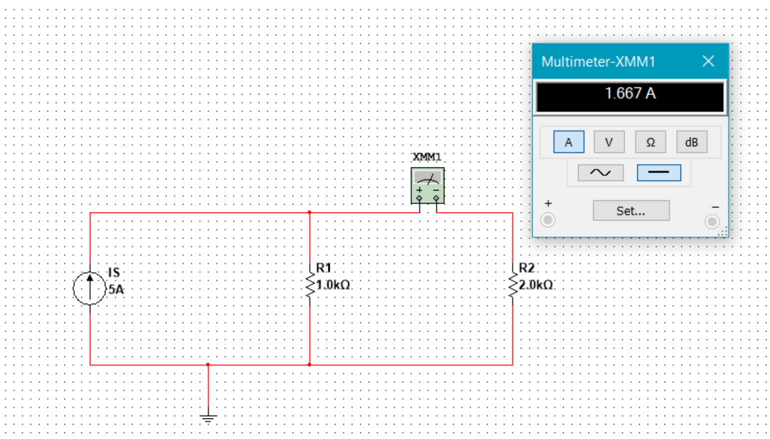
$I_2 = (I_S * R_2)/(R_1 + R_2)$

Circuit Diagram :

- **Circuit for Voltage Division Rule :**

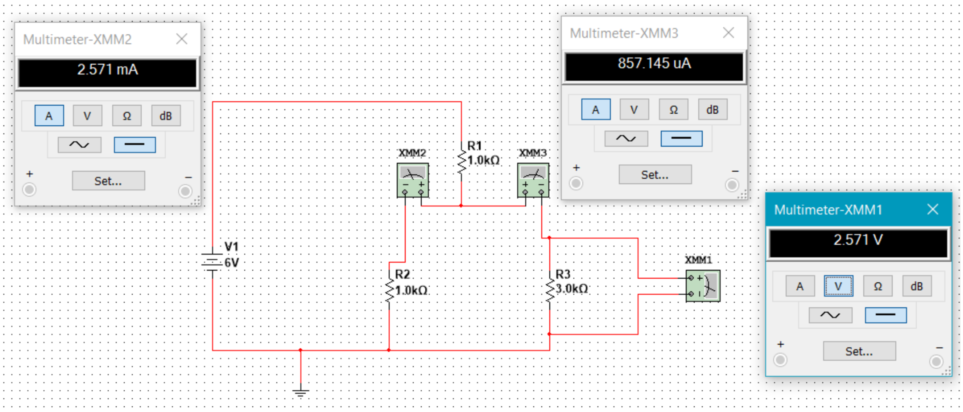


- **Circuit for Current Division Rule :**



[7]

- **Circuit 3 :**



Result & Observations :

- **Voltage Division Rule :**

				Theoretical	Experimental
S No.	**Source Voltage (V)**	**Resistance 1 (kΩ)**	**Resistance 2 (kΩ)**	**Voltage Out (V)**	**Voltage Out (V)**
1	5	1	2	3.33	3.33
2	5	2	3	3	3
3	5	2.5	3.1	2.76	2.76
4	5	2.6	3.2	2.75	2.75
5	5	3	5.25	3.18	3.18

- **Current Division Rule :**

				Theoretical	Experimental
S No.	**Source Voltage (V)**	**Resistance 1 (kΩ)**	**Resistance 2 (kΩ)**	**Voltage Out (V)**	**Voltage Out (V)**
1	5	1	2	1.66	1.66
2	5	2	3	2	2
3	5	2.5	3.1	2.23	2.23
4	5	2.6	3.2	2.24	2.24
5	5	3	5.25	1.81	1.81

- **Circuit 3 :**

				Theoretical	Experimental
S No.	**Source Voltage (V)**	**Current I_1 (A)**	**Current I_2 (A)**	**Voltage Out (V)**	**Voltage Out (V)**
1	5	2.1×10^{-3}	0.714×10^{-3}	2.142	2.143
2	6	2.55×10^{-3}	0.857×10^{-3}	2.571	2.571
3	7	3×10^{-3}	1×10^{-3}	3	3

Hence we see that theoretical and experimental values match. Therefore Voltage and Current division rules are verified.

 Date : 21/01/2021

Experiment – 5

VERIFICATION OF VOLTAGE CURRENT POWER RELATIONS FOR STAR DELTA CONNECTED LOADS

Objective : To verify the voltage, current, power relations for star and delta connected loads

Software Used : NI multisim 14.0

Theory :

Voltages and Currents in Y-Connection:

- The voltage induced in each winding is called the phase voltage and current in each winding is likewise known as phase current.
- However, the voltage available between any pair of terminals is called line voltage (V_L) and the current flowing in each line is called line current (I_L).

Line Voltages and Phase Voltages :

The p.d. between line 1 and 2 is $V_{RY} = E_R – E_Y$. Hence, V_{RY} is found by compounding E_R and E_Y reversed.

Obviously, the angle between E_R and E_Y reversed is 60°. Hence if $E_R = E_Y = E_B$ = say, E_{ph} – the phase e.m.f., then

$V_{RY} = 2 \times E_{ph} \times \cos(60°/2) = \sqrt{3}E_{ph}$

Similarly, $V_{YB} = E_Y - E_B = \sqrt{3}E_{ph}$ and $V_{BR} = E_B - E_R = \sqrt{3}E_{ph}$

Now $V_{RY} = V_{YB} = V_{BR}$ = line voltage, say V_L.

Hence, in star connection $V_L = \sqrt{3}E_{ph}$.

Line Currents and Phase Currents :

Line current in each line is the same as the current in the phase winding to which the line is connected.

Current in line 1 = I_R; Current in line 2 = I_Y; Current in line 3 = I_B

Since $I_R = I_Y = I_B =$ say, I_{ph} – the phase current

$\therefore$ line current $I_L = I_{ph}$.

Power :

The total active or true power in the circuit is the sum of the three phase powers.

Total active power = 3 × phase power or $P = 3 \times V_{ph} I_{ph} \cos\varphi$

$P = \sqrt{3} V_L I_L \cos\varphi$

[9]

Delta (Δ) or Mesh Connection :

- If the system is balanced then sum of the three voltages round the closed mesh is zero.
- This type of connection is also referred to as 3-phase, 3-wire system.

Line Voltages and Phase Voltages :

The voltage between lines 1 and 2 as V_{RY} and that between lines 2 and 3 as V_{YB}, we find that V_{RY} lead V_{YB} by 120°. Similarly, V_{YB} leads V_{BR} by 120º.

Let $V_{RY}= V_{YB}= V_{BR}$ = line voltage V_L. Then, it is seen that $V_L= V_{ph}$.

Line Currents and Phase Currents :

Current in each line is the vector difference of the two phase currents flowing through that line.

Current in line No. 1 is found by compounding I_R and I_B reversed

$I_1= 2 \times I_{ph}\times \cos(60º/2) = \sqrt{3}I_{ph}$

$I_2 = I_B – I_Y = \sqrt{3}I_{ph}$

$I_3 = I_B – I_Y = \sqrt{3}I_{ph}$

Since all the line currents are equal in magnitude i.e.

$I_1 = I_2 = I_3 = I_L$

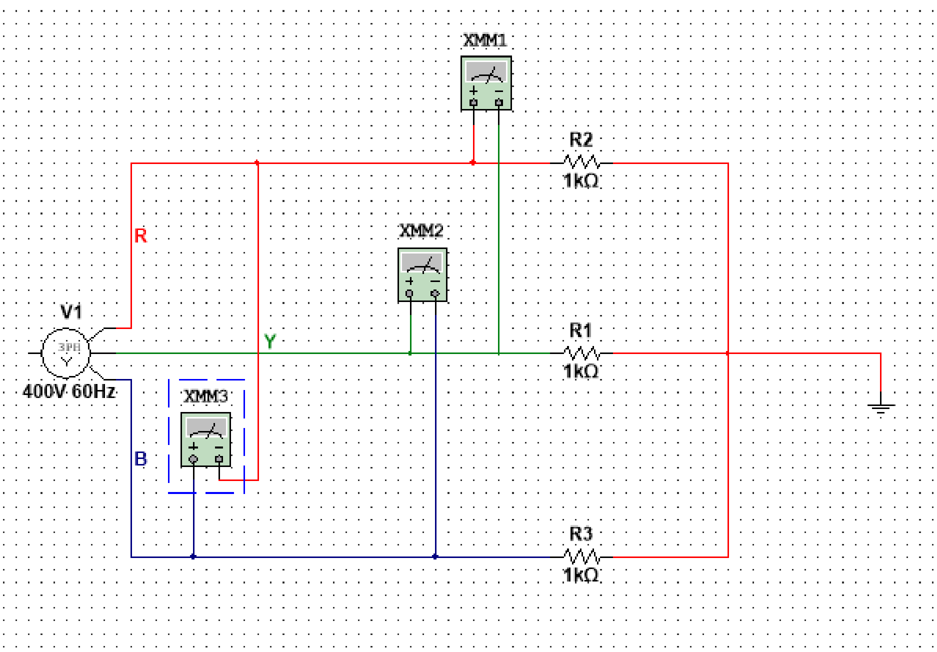
$\therefore I_L= \sqrt{3}I_{ph}$

Power :

Total power = $3 \times V_{ph} I_{ph} \cos\varphi$

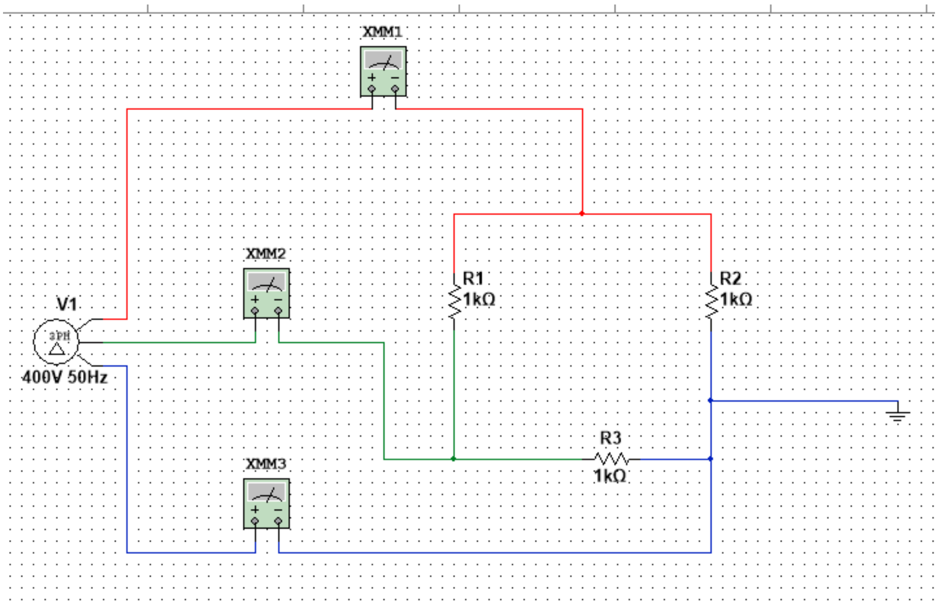
Circuit Diagram :

- **Star Connected Load :**



[10]

- **Delta Connected Load :**



Result & Observations :

- **Star connected Load :**

	Multisim	Theoretical
V_{RY}	692.872V	692.8V
V_{YB}	692.872V	692.8V
V_{BR}	692.872V	692.8V
I_R	400.03mA	400 mA
I_Y	400.016 mA	400 mA
I_B	400.023 mA	400 mA
P_1	240.037W	
P_2	240.037W	
$P = P_1 + P_2$	480.074W	480.057W

- **Delta connected Load :**

	Multisim	Theoretical
V_{RY}	400V	400V
V_{YB}	400V	400V
V_{BR}	400V	400V
I_R	692.861mA	692.820mA
I_Y	692.872mA	692.820mA
I_B	692.848mA	692.820mA
P_1	240.037W	
P_2	240.037W	
$P = P_1 + P_2$	480.074W	479.999W

Hence we are able to see that the theoretical and multisim values are same. Therefore the voltage, current and power relations of star and delta connected loads are verified.

 Date : 23/01/2021

Experiment- 6

CALCULATION OF TWO PORT NETWORK PARAMETERS

Objective :

- To understand the analysis of a two-port network.
- To understand the behavior of a two-port network using parametric analysis.
- To learn the measurement conditions and procedure for two-port analysis.

Software Used : NI Multisim 14.0

Theory : The electrical network with two pairs of terminals is a two-port network. The network can contain resistors, inductors, capacitors, transformers, transistors and in general any linear circuit device, including depending devices but no independent sources are allowed. The behavior of a linear two-port network is described by impedance (Z), admittance (Y), transmission (ABCD), or hybrid (h) parameter.

Z Parameter :

$$V_1 = Z_{11} I_1 + Z_{12} I_2$$

$$V_2 = Z_{21} I_1 + Z_{22} I_2$$

Parameter	Condition	Name
Z11=V1/I1	I2=0	Input impedance
Z21=V2/I1	I2=0	Transfer impedence
Z12=V1/I2	I1=0	Transfer impedence
Z22=V2/I2	I1=0	Output impedence

Y Parameter :

$$I_1 = Y_{11} V_1 + Y_{12} V_2$$

$$I_2 = Y_{21} V_1 + Y_{22} V_2$$

Parameter	Condition	Name
Y11=I1/V1	V2=0	Input admittance
Y21=I2/V1	V2=0	Transfer admittance
Y12=I1/V2	V1=0	Transfer admittance
Y22=I2/V2	V1=0	Output admittance

[12]

H Parameter :

$$V_1 = h_{11}\ I_1 + h_{12}\ V_2$$

$$I_2 = h_{21}\ I_1 + h_{22}\ V_2$$

Parameter	**Condition**	**Name**
h11=V1/I1	V2=0	SS input admittance
h21=I2/I1	V2=0	SS forward current gain
h12=V1/V2	I1=0	OS reverse voltage gain
h22=I2/V2	I1=0	OS output admittance

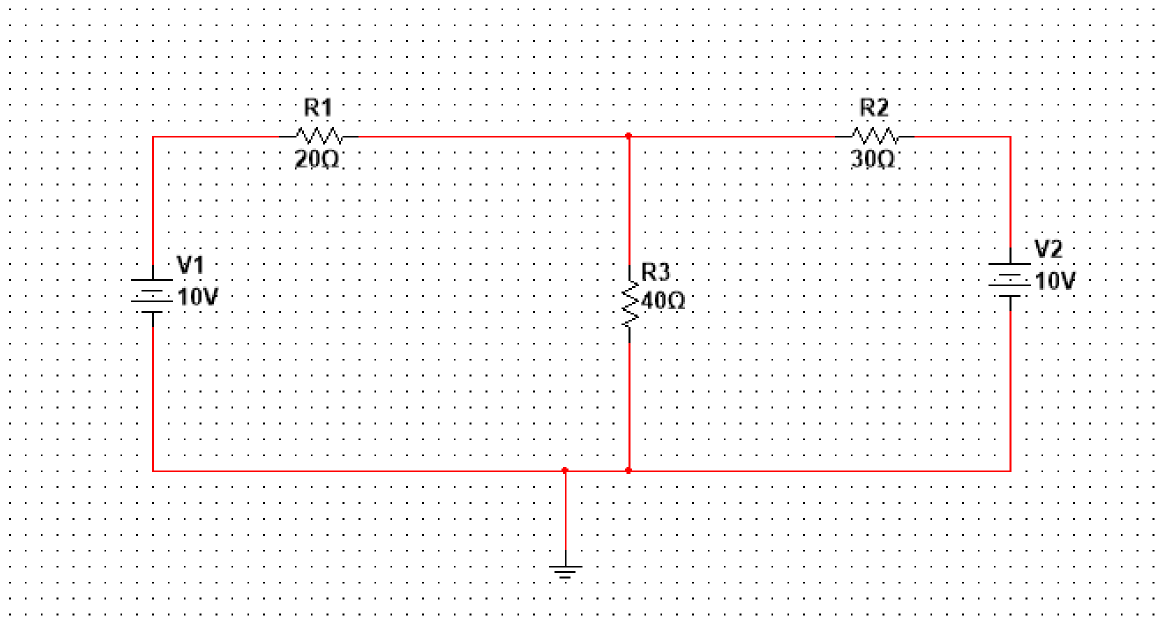
ABCD Parameter :

$$V_1 = A\ V_2 – B\ I_2$$

$$I_1 = C\ V_2\ \ - D\ I_2$$

Parameter	**Condition**	**Name**
A=V1/V2	I2=0	OC voltage ratio
B=V1/-I2	V2=0	OC transfer admittance
C=I1/V2	I2=0	-SC transfer impedance
D=I1/-I2	V2=0	-SC current ratio

Circuit Diagram :



[13]

Result & Observations :

Parameter	Calculated Value	Average value
Z_{11}	60Ω	59.998 Ω
Z_{21}	40 Ω	40.001 Ω
Z_{12}	40 Ω	39.998 Ω
Z_{22}	70 Ω	70 Ω
Y_{11}	0.0269 mho	0.0269 mho
Y_{21}	0.0153 mho	0.01538 mho
Y_{12}	0.0153 mho	0.01538 mho
Y_{22}	0.023 mho	0.023 mho
h_{11}	37.142 Ω	37.14 Ω
h_{21}	0.571	0.57
h_{12}	0.571	0.571
h_{22}	0.0142 mho	0.0142 mho
A	1.5	1.499
B	-65 Ω	-65 Ω
C	0.025 mho	0.0249 mho
D	-1.75	-1.75

Hence we can see that the calculated and experimental values match. Therefore all parameters of two port networks have been verified.

 Date : 23/01/2021

Experiment – 7

STUDY OF CHARACTERISTICS OF SEMICONDUCTOR DIODE TRANSISTORS

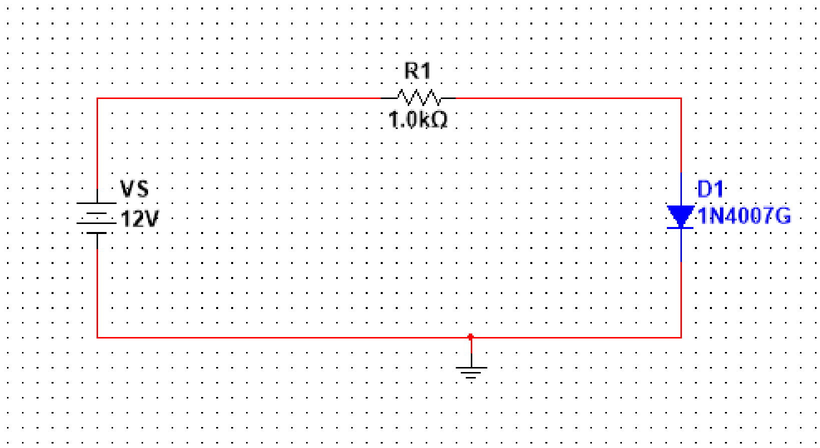
Objective : To study the input and output characteristics of semiconductor diodes

Software Used : NI Multisim 14.0

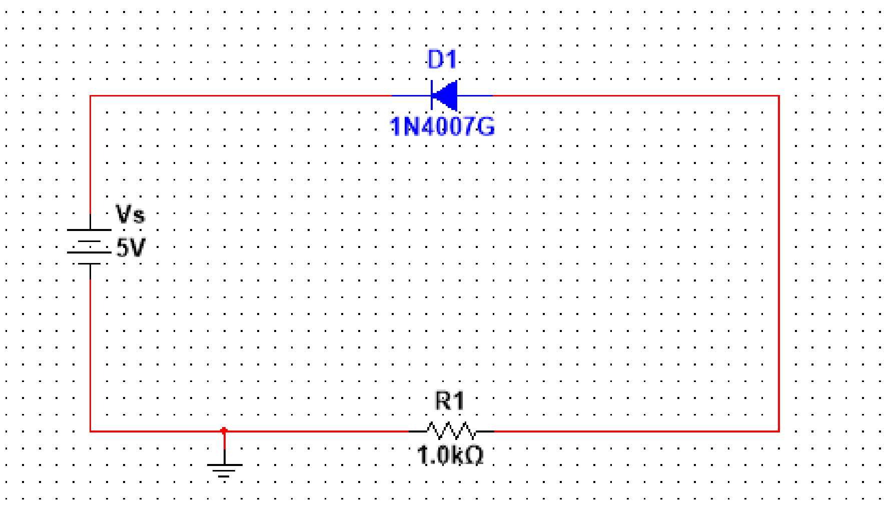
Theory : A semiconductor diode is a combination of p and n type semiconductors which in forward bias conducts current only above a certain voltage known as cutting voltage. After achieving cutting voltage the increase in current is almost exponential. A diode in reverse bias conducts negligible amount of current in the order of micro amperes.

Circuit Diagram :

- **Diode in Forward Bias :**

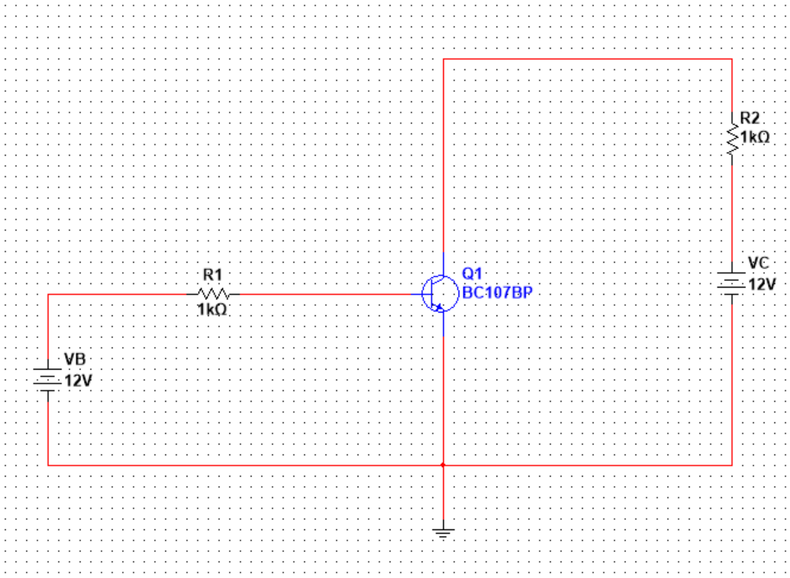


- **Diode in Reverse Bias :**



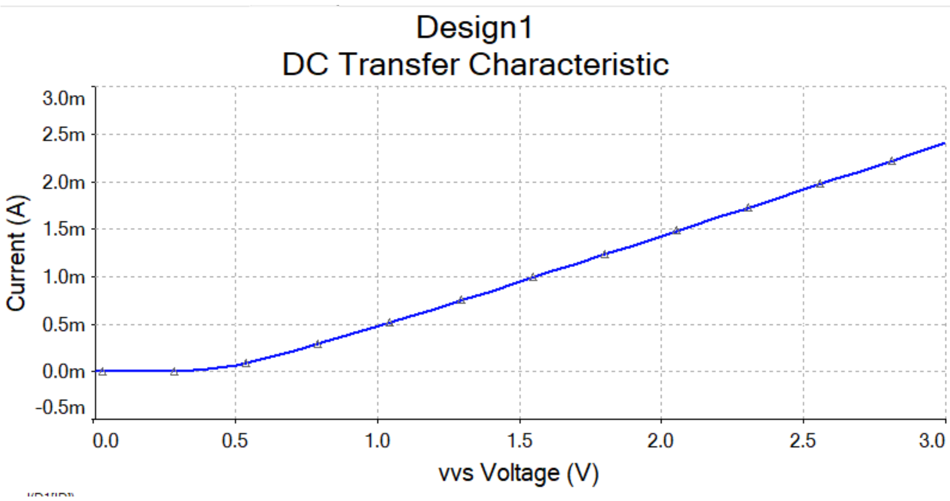
[15]

- **Common Emitter Configuration :**



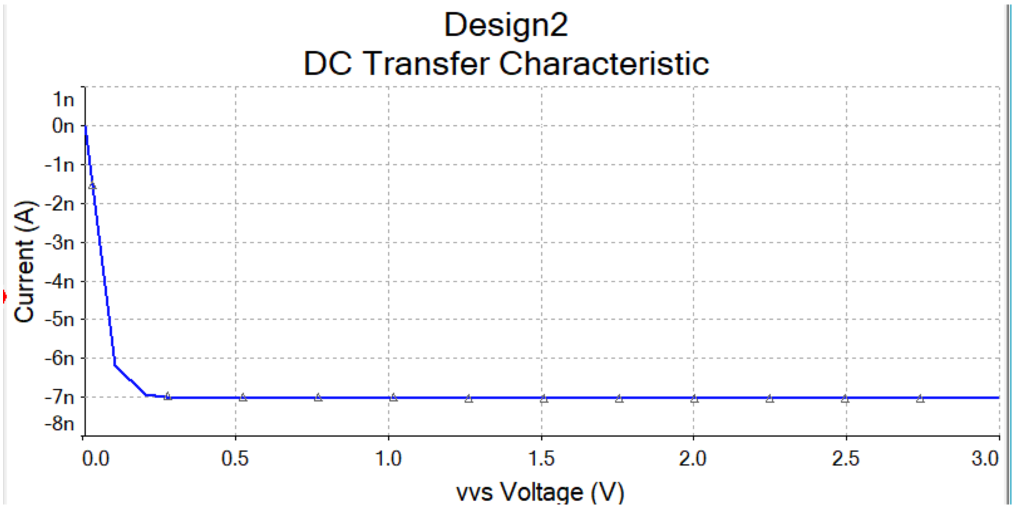
Result & Observations :

- **Diode is in Forward Bias :**
  The cutting voltage for the diode is 0.3V.
  Therefore the diode is a germanium diode.

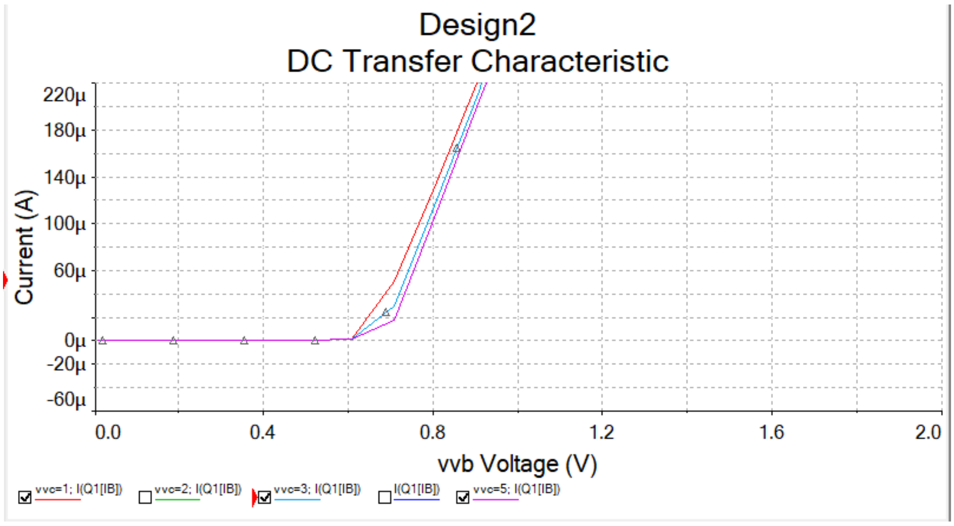


[16]

- **Diode is in Reverse Bias :**
  The leakage current is negligible or zero.



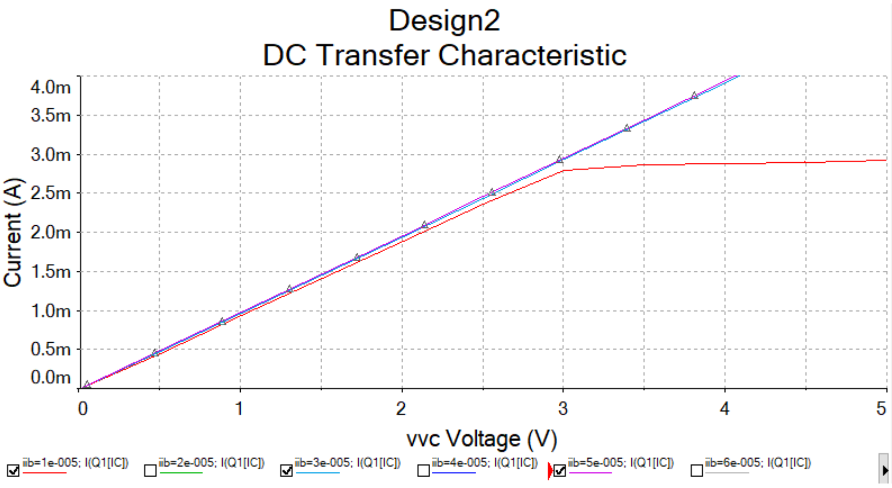
- **Common Emitter configuration Input Characteristics :**



	Voltage (V_C)		
	1V	**3V**	**5V**
Cutting Voltage (V_γ)	**0.500V**	**0.506V**	**0.510V**

[17]

- **Common Emitter Configuration Output Characteristics :**



	Base Current (I_B)		
	10µA	**30µA**	**50µA**
Collector Current (I_C)	**2.77mA**	**8.74mA**	**14.46mA**

 Date : 27/01/2021

Experiment – 8

DESIGN AND SIMULATION OF VARIOUS RECTIFIER CIRCUITS

Objective : To design and simulate various rectifier circuits.

Software Used : Ni Multisim 14.0

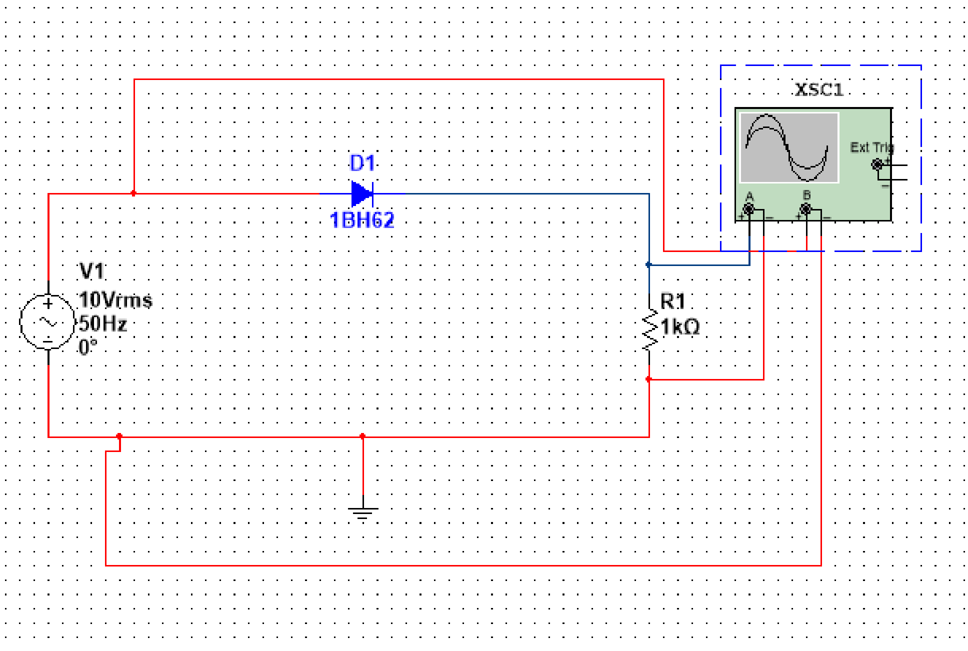
Theory : A rectifier is a circuit that converts the Alternating Current (AC) input power into a Direct Current (DC) output power. The input power supply may be either a single-phase or a multi-phase supply.

There are 2 types of rectifiers :

- **Half wave rectifier :** The power diode in a half-wave rectifier circuit passes just one half of each complete sine wave of the AC supply to convert it into a DC supply. Then this type of circuit is called a "half-wave" rectifier because it passes only half of the incoming AC power supply.
- **Full wave or Bridge rectifier :** We can define bridge rectifiers as a type of full-wave rectifier that uses four or more diodes in a bridge circuit configuration to efficiently convert alternating (AC) current to a direct (DC) current.

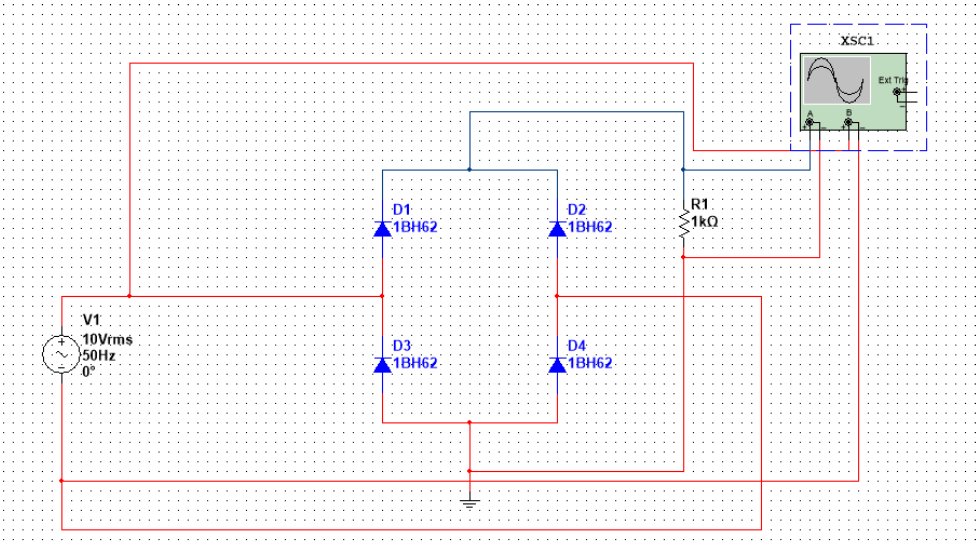
Circuit Diagram :

- **Half Wave Rectifier :**



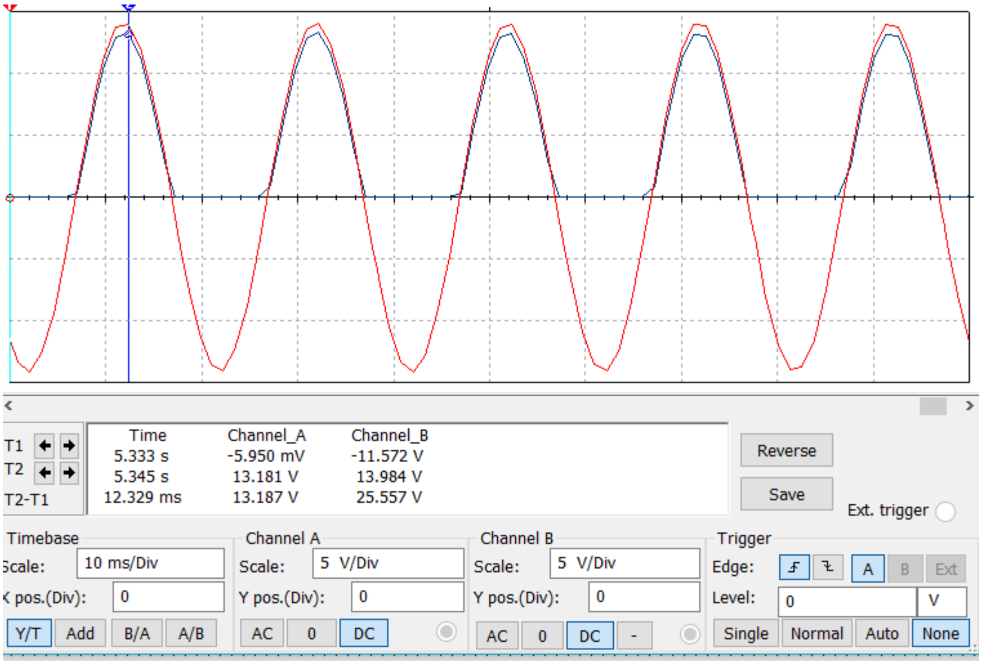
[19]

- **Full Wave Rectifier :**



Result & Observations :

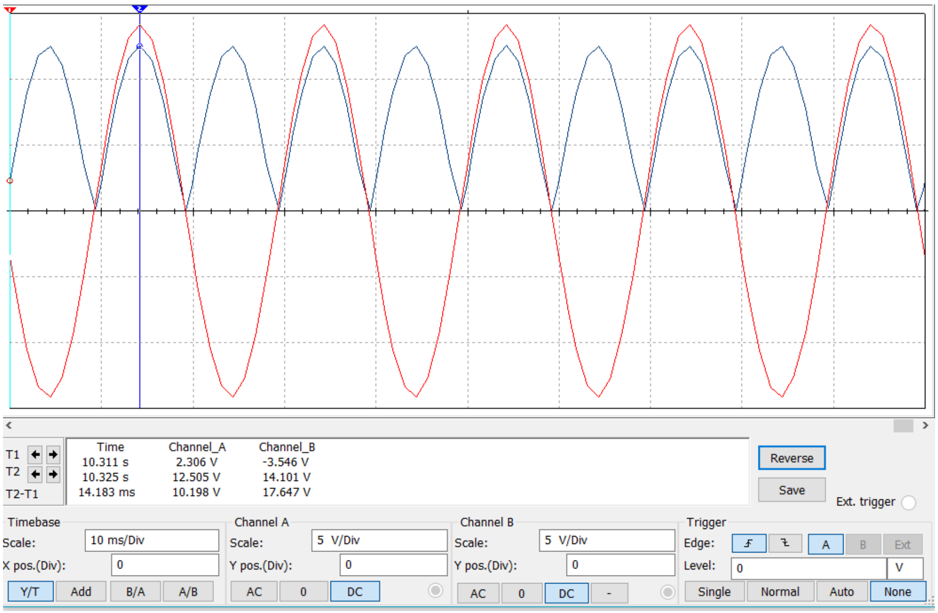
- **Half wave Rectifier :**



V_M		V_{RMS}		V_{DC}		Ripple factor (γ)	
Theory	**Multisim**	**Theory**	**Multisim**	**Theory**	**Multisim**	**Theory**	**Multisim**
20V	**13.181V**	**10V**	**6.590V**	**6.366V**	**4.195V**	**1.2114**	**1.2115**

[20]

- **Full wave Rectifier :**



V_M		V_{RMS}		V_{DC}		Ripple Factor (γ)	
Theory	**Multisim**	**Theory**	**Multisim**	**Theory**	**Multisim**	**Theory**	**Multisim**
14.14V	**12.505V**	**10V**	**8.843V**	**9.001V**	**7.960V**	**0.484**	**0.483**

Result : Hence we can see that the theoretical and experimental values match. Therefore we have a successful experiment.

Experiment-9

DESIGN AND IMPLEMENTATION OF MOSFET VOLTAGE AMPLIFIER

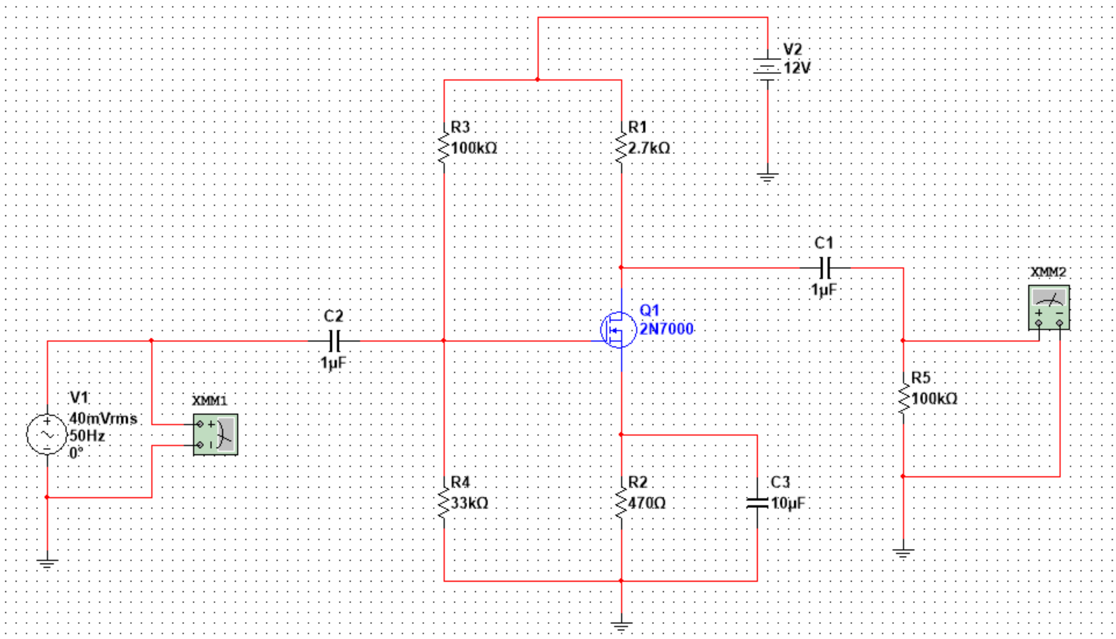
Objective : To study the MOSFET amplifier working in common source configuration with given specifications.

Software Used : NI Multisim 14.0

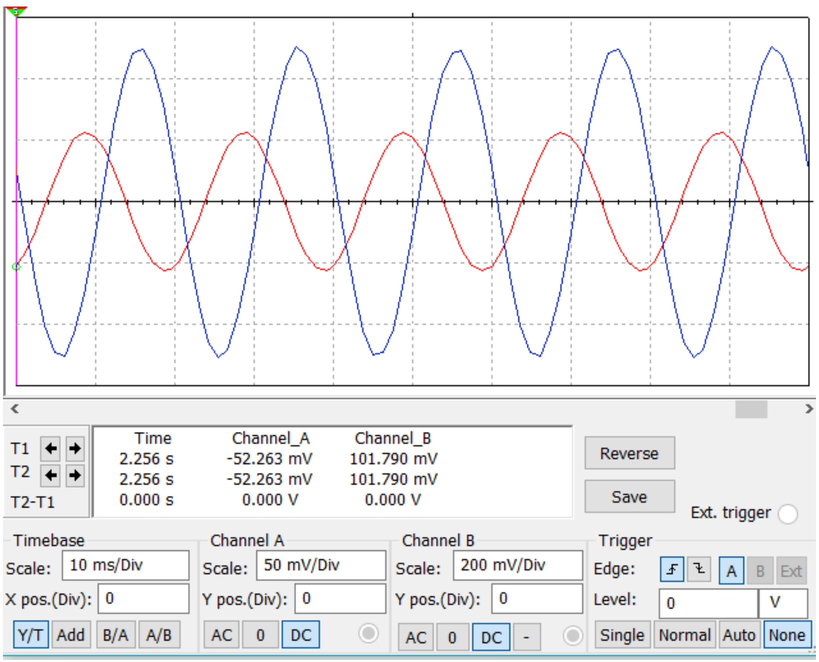
Theory : The MOSFET structure has become the most important device structure in the electronics industry. It dominates the integrated circuit technology in Very Large Scale Integrated (VLSI) digital circuits based on n-channel MOSFETs and Complementary n-channel and p-channel MOSFETs (CMOS). The technical importance of the MOSFET results from its low power consumption, simple geometry, and small size, resulting in very high packing densities and compatibility with VLSI manufacturing technology. Two of the most popular configurations of small-signal MOSFET amplifiers are the common source and common drain configurations. The common source circuit is shown below. The common sources, like all MOSFET amplifiers, have the characteristic of high input impedance. High input impedance is desirable to keep the amplifier from loading the signal source. This high input impedance is controlled by the bias resistors R1 and R2). Normally the value of the bias resistors is chosen as high as possible. However, too big a value can cause a significant voltage drop due to the gate leakage current. A large voltage drop is undesirable because it can disturb the bias point. For amplifier operation the MOSFET should be biased in the active region of the characteristics.

[22]

Circuit Diagram :



Result & Observations :



Input Voltage V_i (mV)	Output Voltage V_o (mV)	Voltage Gain $\frac{V_o}{V_i}$
40.021	355.734	8.888
50.016	442.937	8.856

Therefore we can see that the MOSFET gives a voltage gain of 8.8. Hence the MOSFET successfully works as a voltage amplifier.

 Date : 28/01/2021

Experiment – 10

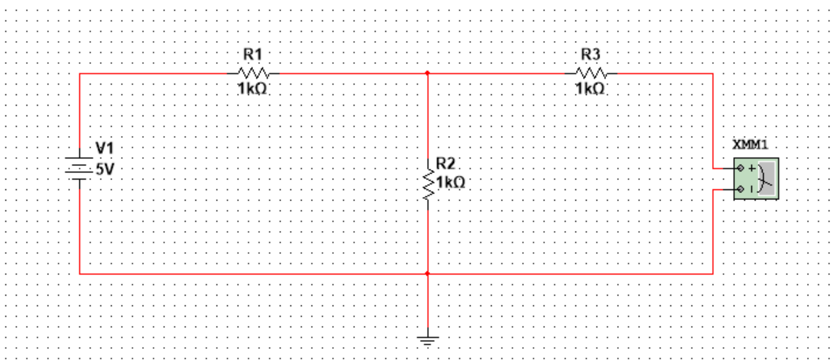
APPLICATION OF THEVENIN'S THEOREM

Objective : To Verify Thevenin's theorem.

Software Used : NI Multisim 14.0

Theory : Any linear, bilateral network having a number of voltage, current sources and resistances can be replaced by a simple equivalent circuit consisting of a single voltage source in series with a resistance, where the value of the voltage source is equal to the open circuit voltage and the resistance is the equivalent resistance measured between the open circuit terminals with all energy sources replaced by their ideal internal resistances.

Circuit Diagram :



Result & Observations :

				V_{Th}		R_{Th}	
V_S	R_1	R_2	R_3	Theoretical	Multisim	Theoretical	Multisim
5V	1kΩ	1 kΩ	1 kΩ	2.5V	2.5V	1.5k Ω	1.5k Ω
5.2V	1.1 kΩ	2.5 kΩ	3 kΩ	3.611V	3.611V	3.7638k Ω	3.764k Ω
10V	2.5 kΩ	3 kΩ	5.2 kΩ	5.454V	5.455V	6.5636k Ω	6.564k Ω
11V	3 kΩ	2.5 kΩ	6.1 kΩ	5V	5V	7.4636k Ω	7.464k Ω

Hence we can see that the theoretical values and experimental values match. Therefore Thevenin's Theorem has been verified.