



Electrical and Electronic Simulation Project

Hridoy Banik

Serial	Name of the Experiments
01	To measure voltage and current of a circuit using NI multisim .
02	Plot VI characteristics of a diode using NI multisim.
03	Half wave rectifier circuit simulation on NI Multisim.
04	Full wave bridge rectifier circuit simulation on NI Multisim.
05	Simulation of full wave center tape rectifier in NI Multisim
06	Simulation of Series Clipper Unbiased Circuit.
07	Simulation of Series Clipper biased Circuit.
08	Simulation of parallel Clipper biased Circuit.
09	Simulation of combination Clipper Circuit in NI Multisim
10	Simulation of unbiased Clamper Circuit.
11	Simulation of biased parallel clamper circuit at NI Multisim.
12	simulation of Zener diode and characteristics at NI Multisim of VI curve of Zener diode.
13	Simulation of Zener diode voltage regulation and graph analysis.
14	Simulation of input and output characteristics of Common Emitter BJT in NI Multisim.
15	Simulation of amplification of BJT
16	Comparison of Resistive, Inductive and Capacitive Circuit in NI Multisim.
17	Simulation of RLC circuit in NI Multisim.
18	Simulation of Voltage Doubler in NI Multisim.
19	Verification of Superposition theorem in NI Multisim.
20	Verification of Superposition theorem in NI Multisim.
21	Verification of Thevenin theorem in NI Multisim.
22	Verification of Norton theorem by using simulation in NI Multisim.
23	Simulation of Operational Amplifier as comparator.
24	Simulation of Converting 230V RMS voltage into regulated 5 V DC voltage.
25	Simulation of three phase power supply NI Multisim.
26	Simulation of summing amplifier using operational amplifier.
27	Simulation of Differential amplifier in NI Multisim.
28	Simulation of Operational amplifier as an integrator in NI Multisim
29	Simulation of Measurement of power using two wattmeter in a three phase balance wye connection.
30	Simulation of transistor as a switch and as an inverter in NI Multisim.

Experiment No: 01

Name of the experiment: To measure voltage and current of a circuit using NI multisim .

Objective:

1. To get familiar with the interface of the NI multisim software.
2. To setup a circuit properly
3. To connect the ammeter(series) and volt meter(parallel) in appropriate manner.
4. To test the basic law of electrical circuit such Ohm's law, Kirchhoff's law , voltage divider rule , current divider rule etc.

Component requires for simulation:

1. DC voltage source.
2. Resistor.
3. Multimeter.
4. Connecting wire.

Circuit Diagram:

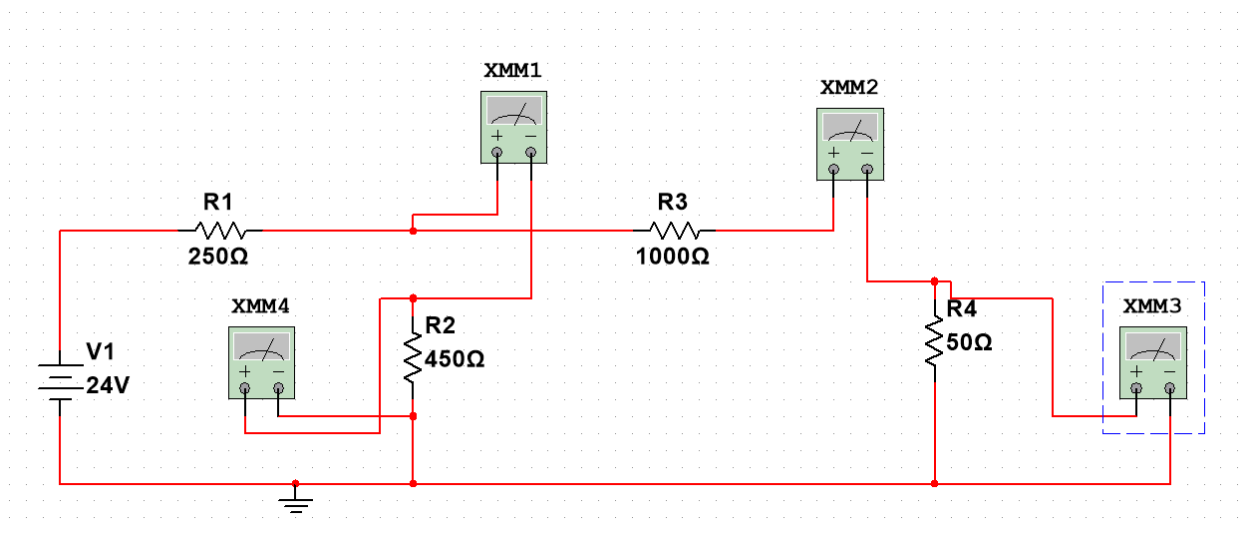


Fig 1.1 : Circuit Diagram of measurement of voltage and current in NI multisim.

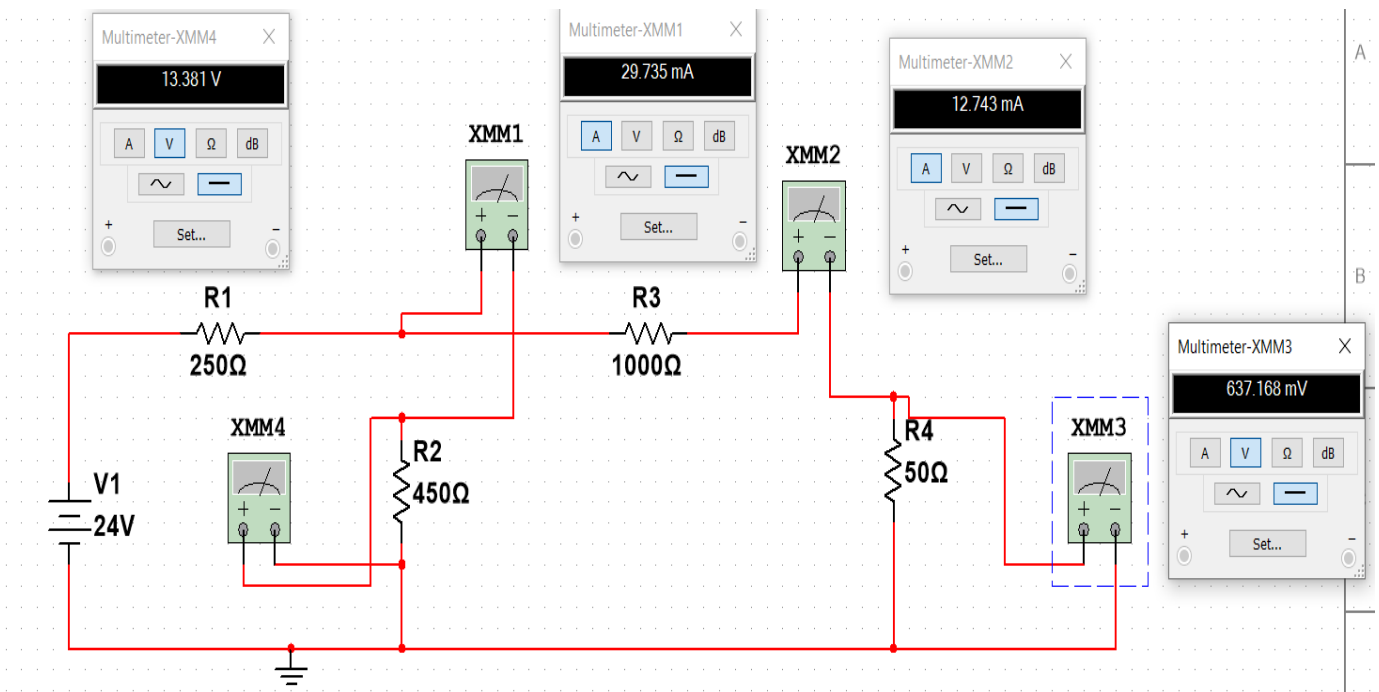


Fig 1.2: Output of the circuit.

Conclusion: Measuring current and voltage of a circuit is one of the fundamental experiments. We have taken a 24v dc power supply and 250, 450 , 1000, 50 ohm resistant respectively. We've connected ammeter in series and voltmeter in parallel. Although we have used multimeter in both the cases. Now we can see the current across the 450 ohm resistance is 29.735 mA, and the voltage is 13.381v again current through the 50 ohm resistance is 12.743mA and the voltage is 637.168 mV . Through this lab we have get familiar with the interface of NI multisim and measured the fundamental things of electrical engineering.

Experiment No : 02

Name of the experiment: Plot VI characteristics of a diode using NI multisim.

Objective:

1. To study the relation between voltage and current of a diode by applying various voltage.
2. To plot the curve for VI characteristics of a diode.
3. Connect the diode in forward and reverse bias and to observe how the diode changes its parameter according to corresponding value.

Component required at simulation:

1. IN4007 diode
2. Oscilloscope
3. Connecting wire
4. Resistance.

Circuit Diagram:

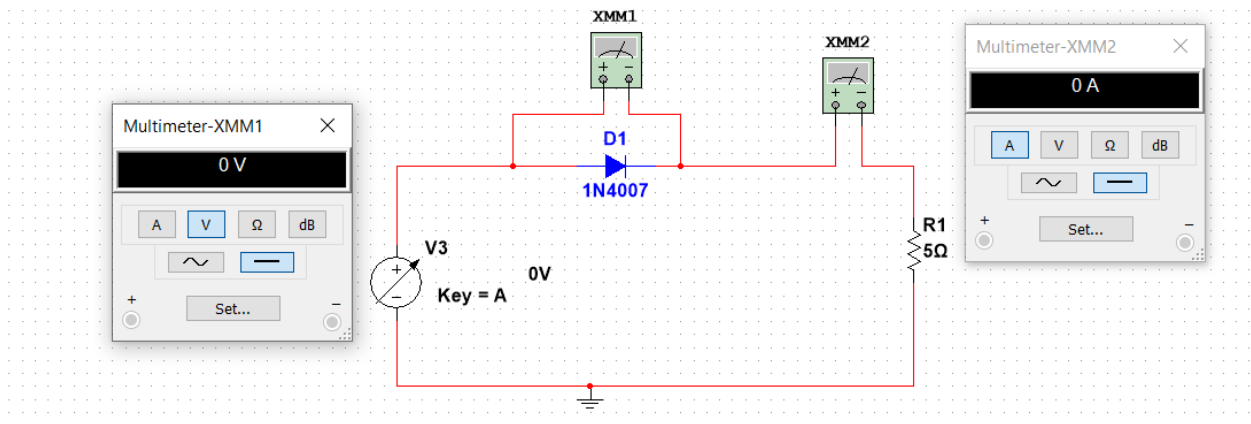


Figure 2.1 : Forward bias diode connection .

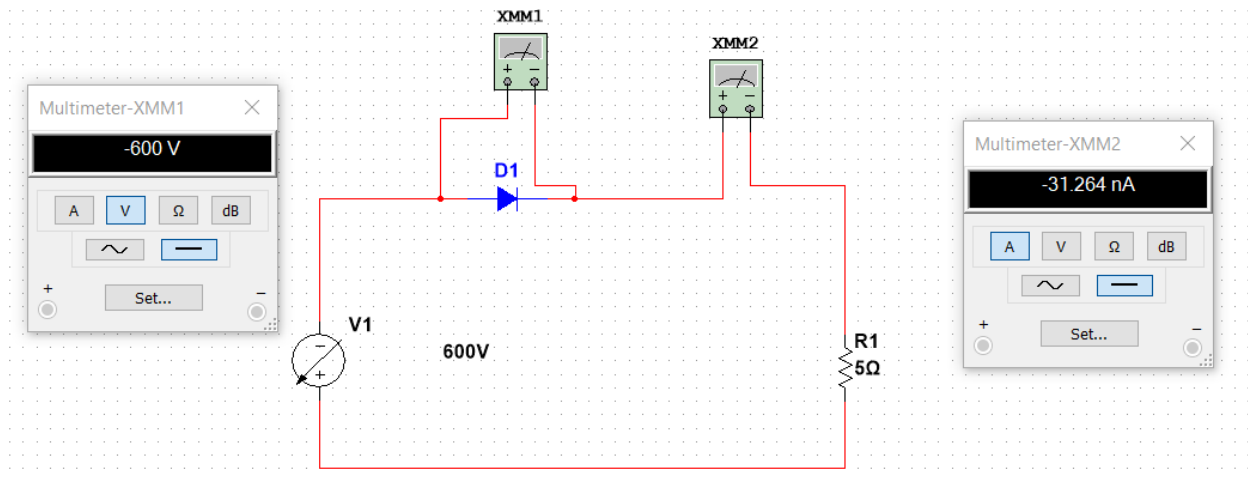


Figure 2.2 : Forward bias diode connection .

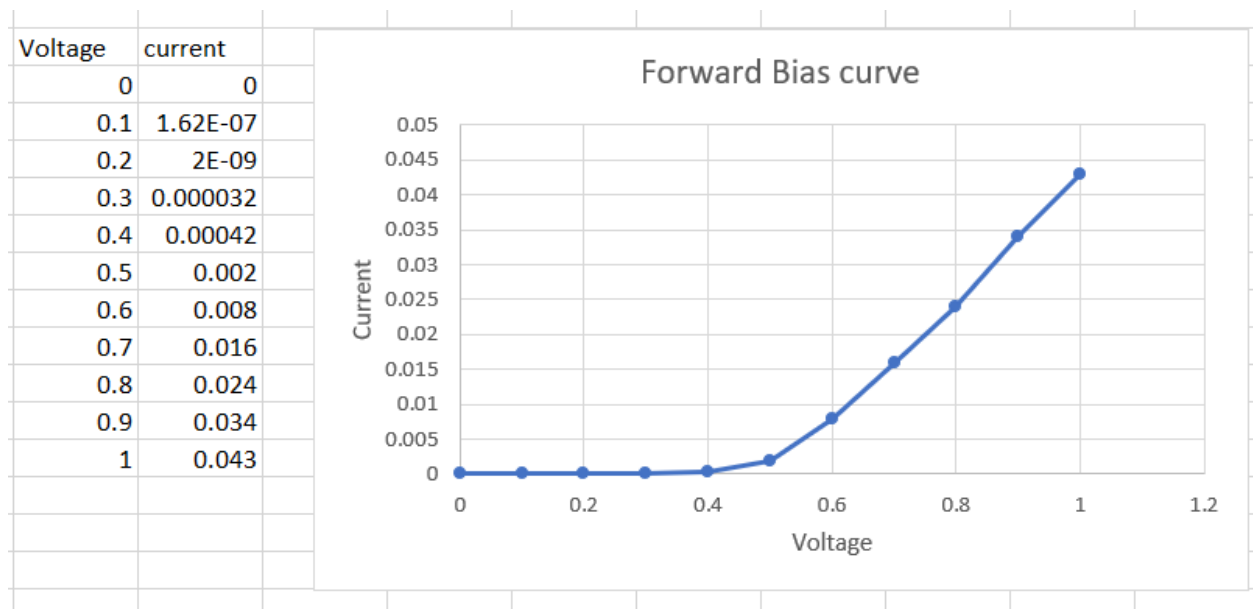


Figure 2.3: Forward bias PN junction curve

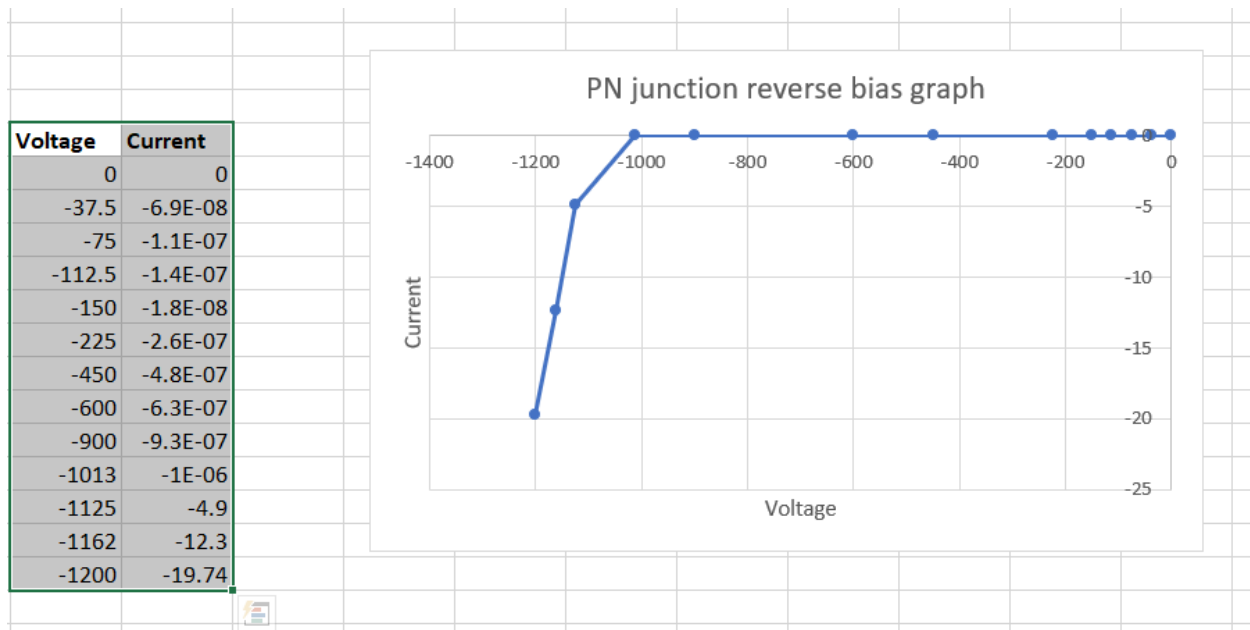


Figure 2.4: Reverse bias PN junction graph.

Conclusion: The application of a forward biasing voltage on the junction diode results in the depletion layer becoming very thin and narrow which represents a low impedance path through the junction thereby allowing high currents to flow. The point at which this sudden increase in current takes place is represented on the static I-V characteristics curve above as the “knee” point. On the other hand, reverse bias voltage on the junction results depletion layer getting wider hence ideally no current should flow but some of the current flows due to minority charge carrier. If we increasing negative voltage after sudden point we will have a huge negative current flowing this is known as breakdown region.

Experiment No : 03

Experiment Name: Half wave rectifier circuit simulation on NI Multisim.

Objective:

1. To design and simulate half wave rectifier on NI Multisim
2. To understand how half wave rectifier acts.
3. To know the characteristics of half wave rectifier.
4. Explain rectification for positive half cycle and negative half cycle.

Component requires at simulation:

- 1.Oscilloscope
- 2.IN4007 diode
- 3.Ac power source
- 4.Resistor
- 5.Connecting wire

Experimental Setup:

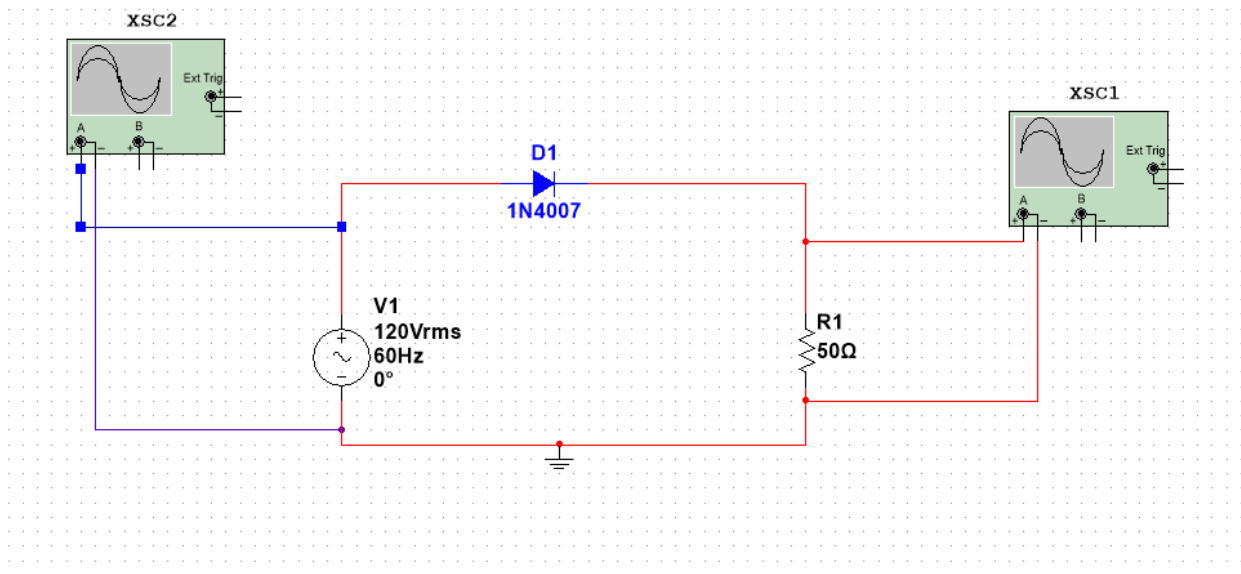


Figure 3.1: Half wave rectifier Circuit

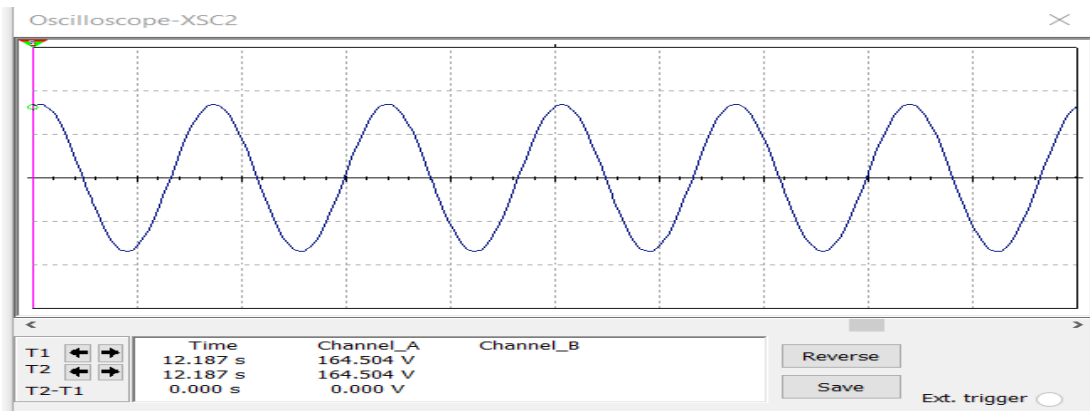


Figure 3.2 : Half wave rectifier Input graph

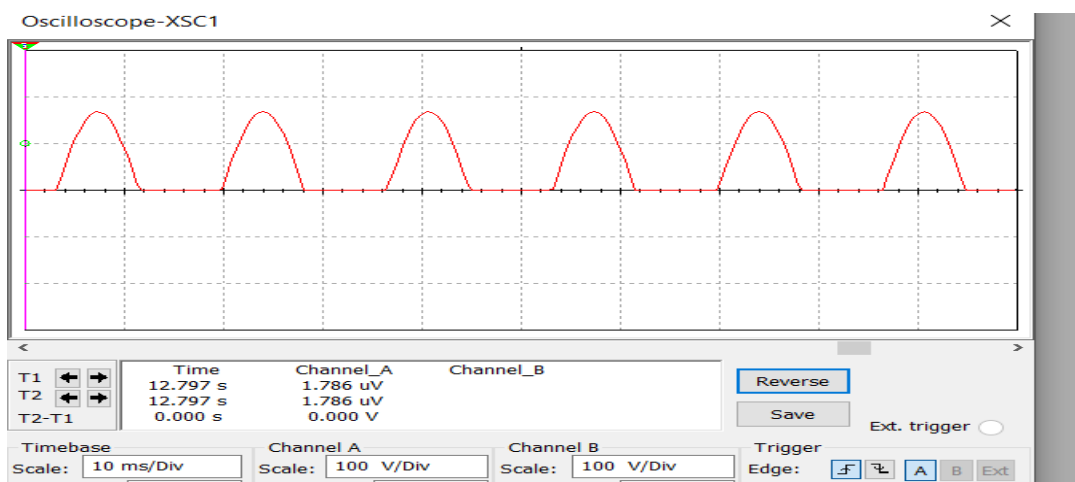


Figure 3.3 : Half wave rectifier output graph

Conclusion: In half wave rectifier we get the output when the diode is in forward bias but when the diode is in reverse bias we get 0v as output. That's why we get the wave shape at positive half cycle and 0v at negative half cycle.

Experiment No: 04

Experiment Name: Full wave bridge rectifier circuit simulation on NI Multisim.

Objective:

1. To design a full wave bridge rectifier.
2. To show that alternating current is rectified into direct current.
3. To design and simulate half wave rectifier on NI Multisim
4. To show that full wave rectifier has greater efficiency than half wave rectifier.

Component requires at simulation:

- 1.Oscilloscope
- 2.IN4007 diode
- 3.Ac power source
- 4.Resistor
- 5.Connecting wire

Experimental Setup:

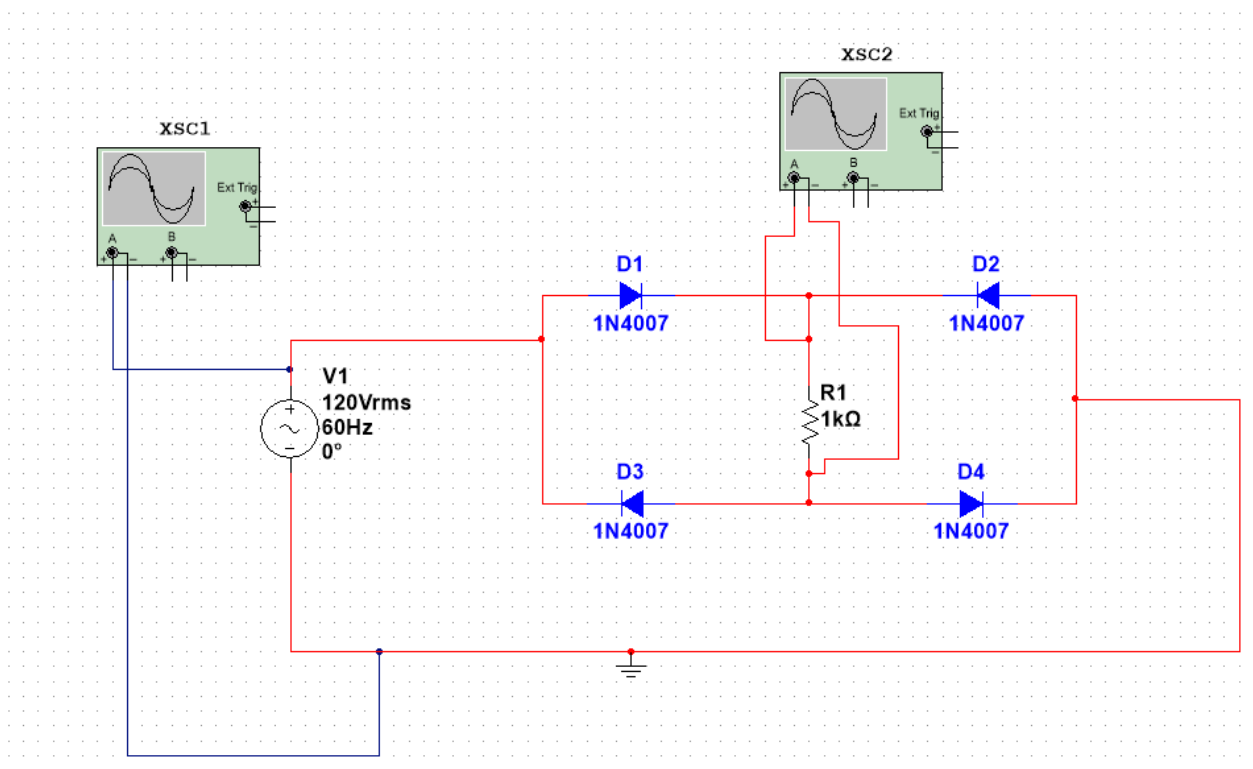


Figure4.1 : Full bridge rectifier circuit diagram

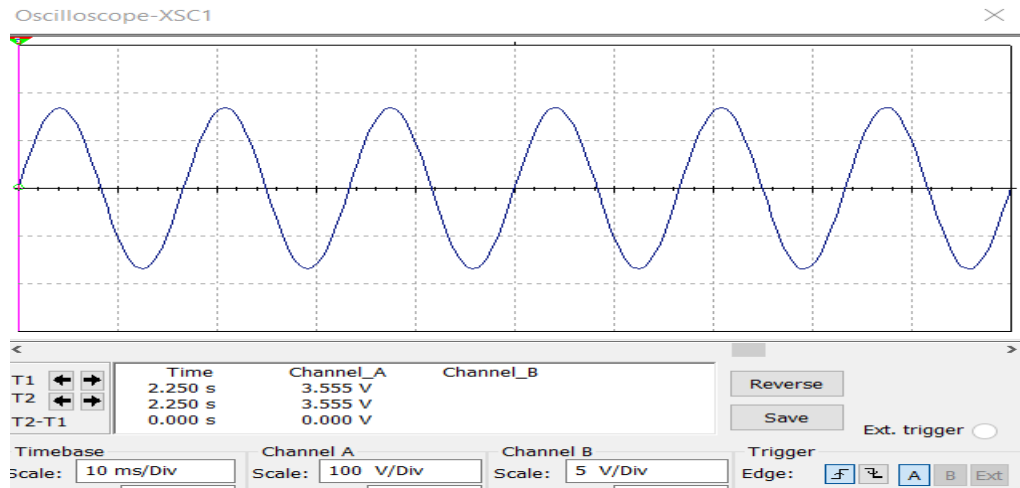


Figure 4.2 : Full wave rectifier voltage input waveshape

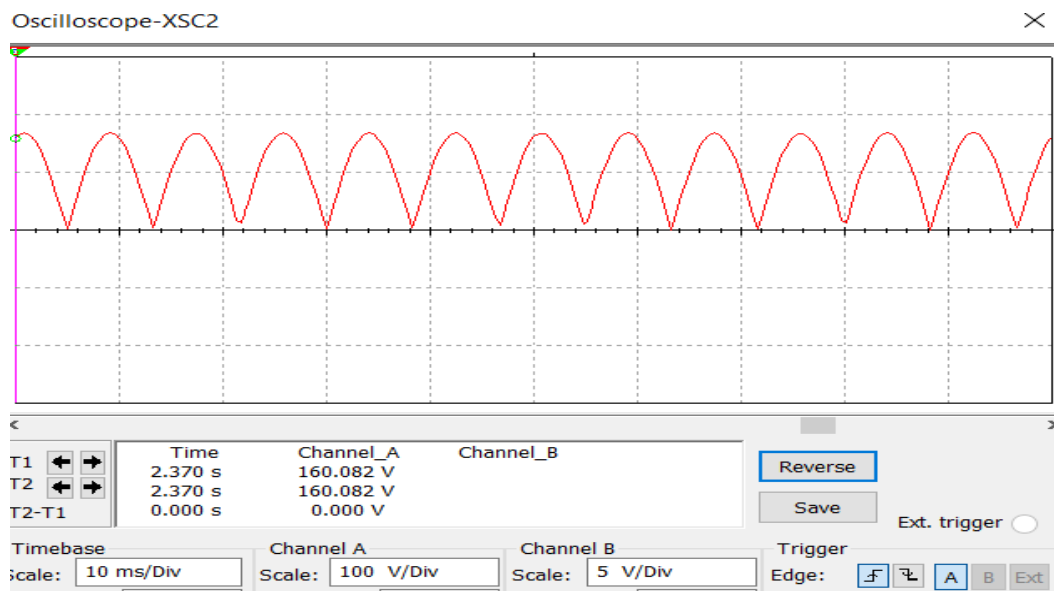


Figure 4.3: Full wave rectifier output waveshape

Conclusion: Full wave rectifier rectifies the whole input waveshape. That's why we get the full waveshape. In a full bridge rectifier the efficiency is almost double than the half wave rectifier. To obtain the full dc output from an ac source we use full wave rectifier.

Experiment No: 05

Experiment Name: Simulation of full wave center tape rectifier in NI Multisim

Objective:

1. Use of transformer as a converter.
2. To get direct current from any alternating current source.
3. To design an circuit with transformer in NI Multisim.

Component requires at simulation:

- 1.Oscilloscope
- 2.IN4007 diode
- 3.Ac power source
- 4.Resistor
- 5.Connecting wire
6. A center tape transformer

Experimental Setup:

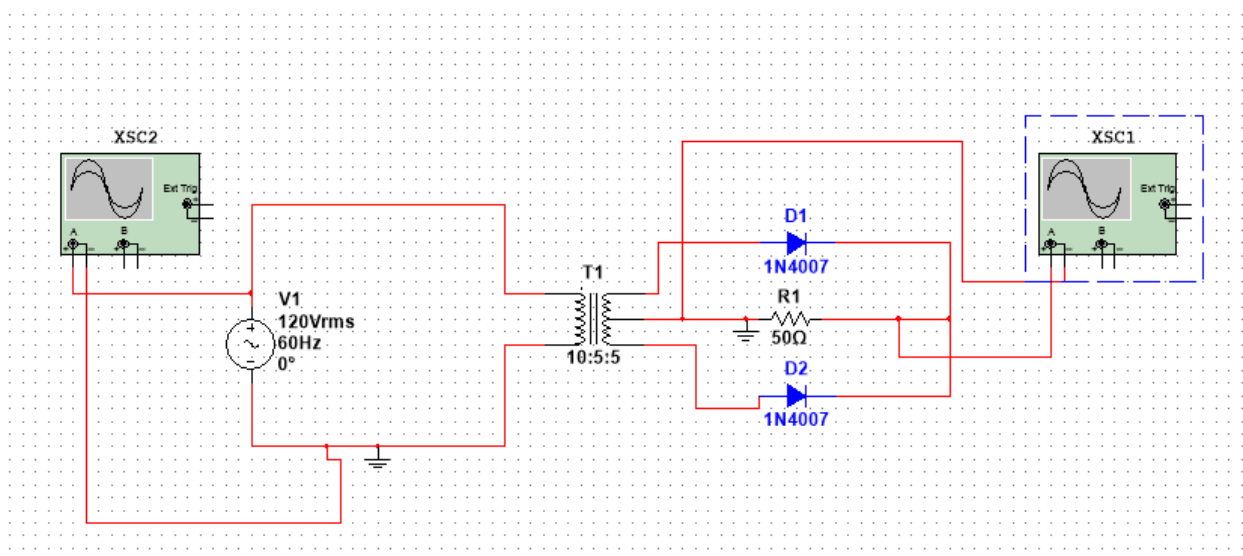


Figure 5.1 : Circuit diagram of Center tape rectifier.

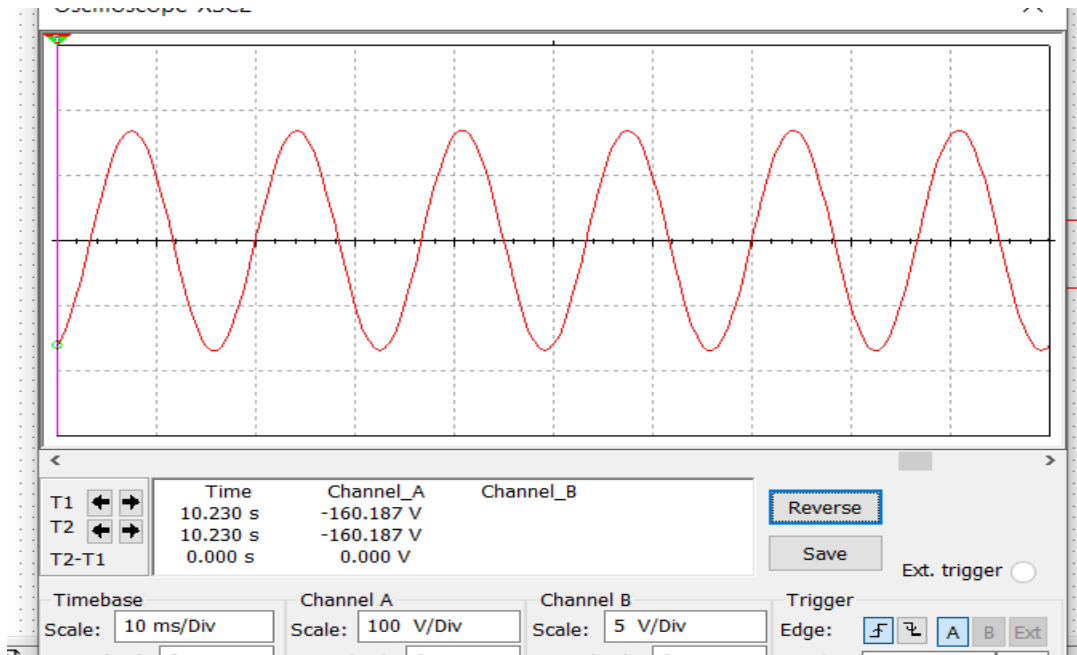


Figure 5.2: Center tape full wave rectifier input waveshape

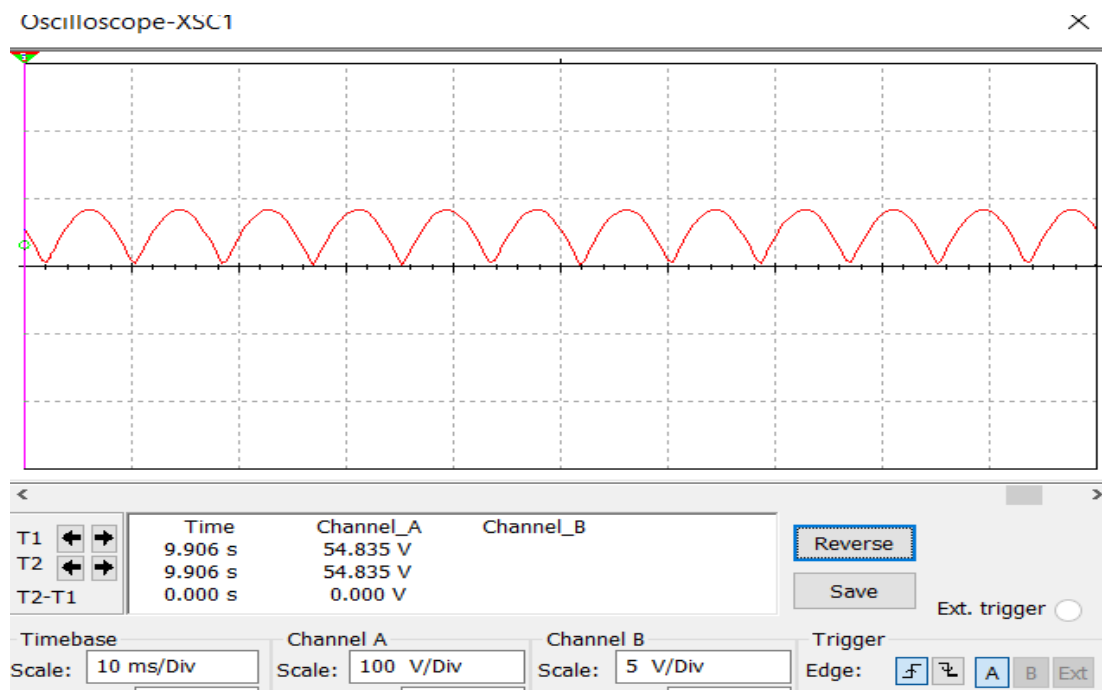


Figure 5.3: Center tape full wave rectifier output waveshape

Conclusion: Center tap rectifier has a similar working principle like full bridge rectifier. It also rectifies the full wave shape of the input signal. The basic difference we use a center tap transformer which divides the input voltage amplitude. Although the basic function is same .

Experiment No: 06

Name of the experiment: Simulation of Series Clipper Unbiased Circuit.

Objective:

1. To study how to form a clipper circuit with diode.
2. To understand the working principle of clipper circuit.
3. Design a circuit such a way that it can clip positive waveshape and negative waveshape.

Component requires at simulation:

1. Oscilloscope
2. IN4007 diode
3. Connecting wire
4. Resistor

Experimental Setup:

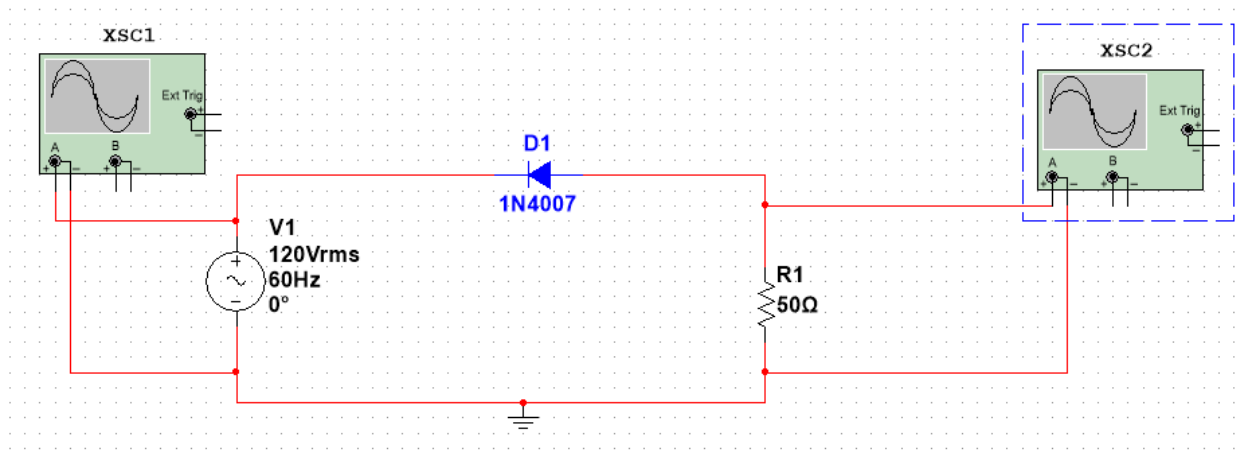


Figure 6.1 : Positive series clipper circuit

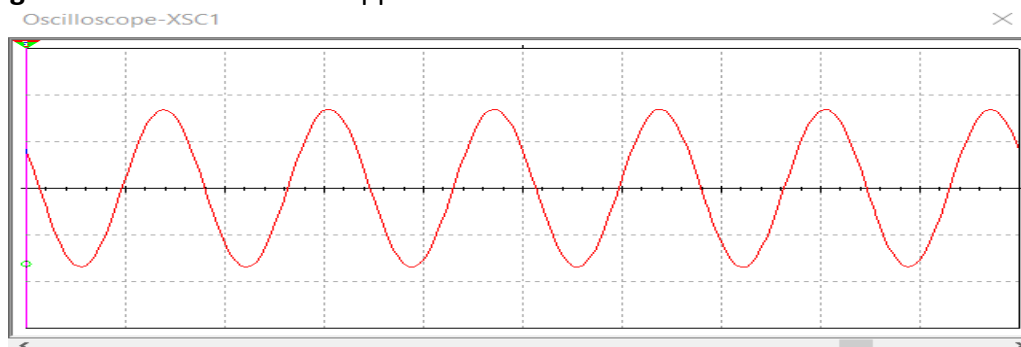


Figure 6.2: Input wave shape

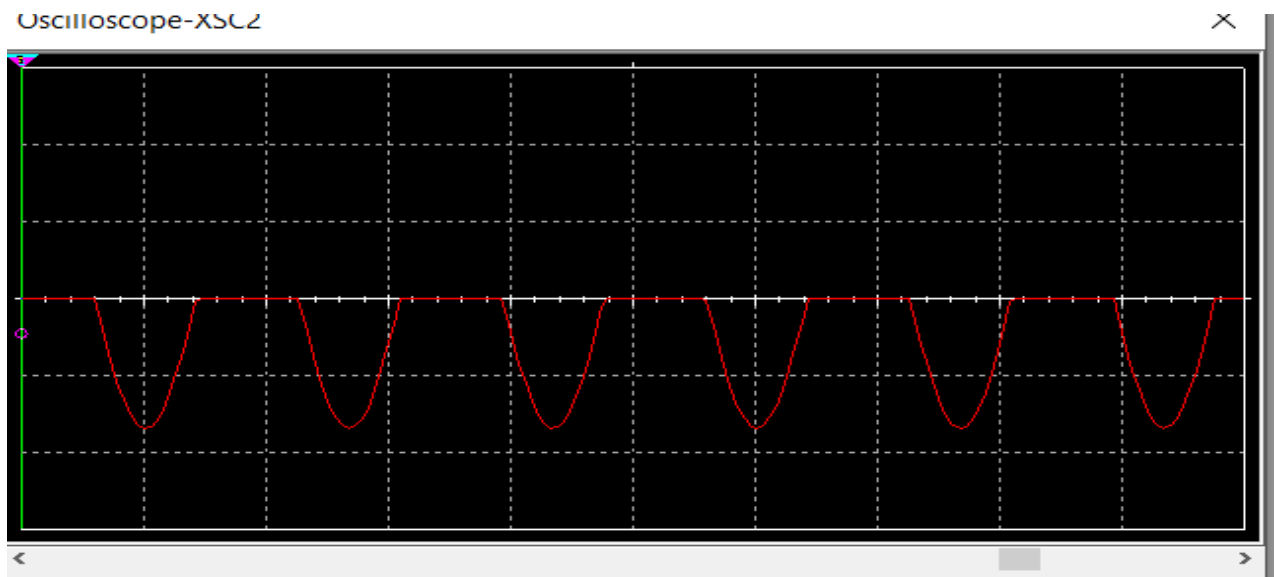


Figure 6.3 : Postive series clipper circuit Output

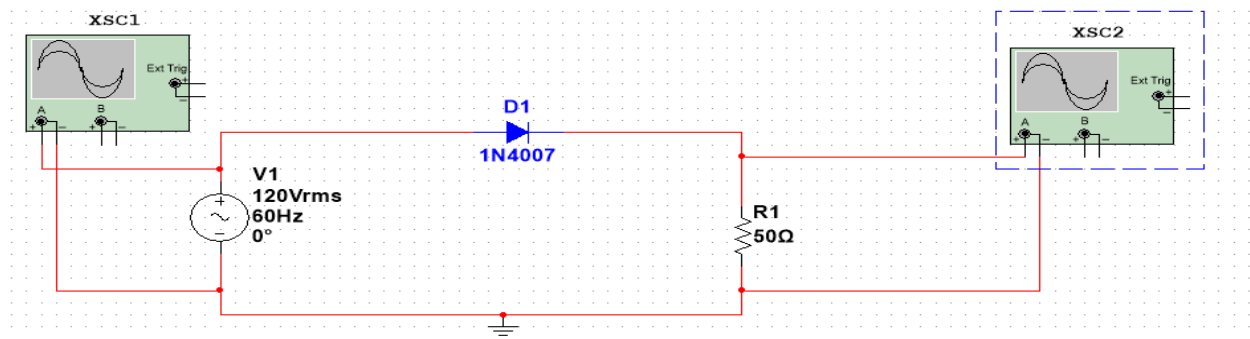


Figure 6.4 : Negative series Clipper Circuit

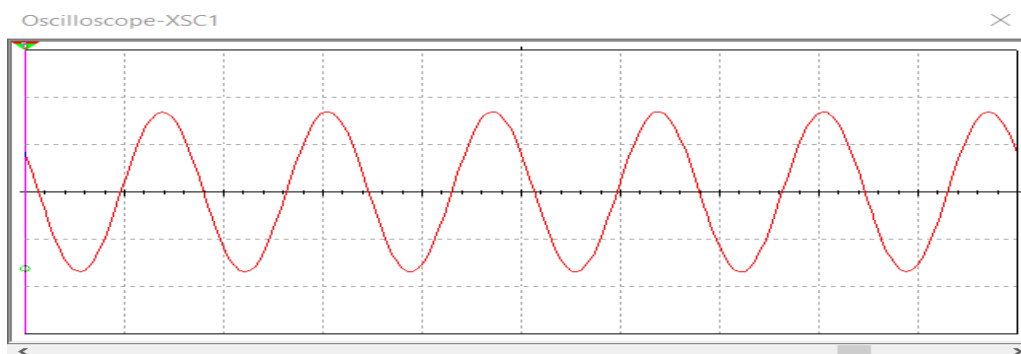


Figure 6.5 : Input waveshape

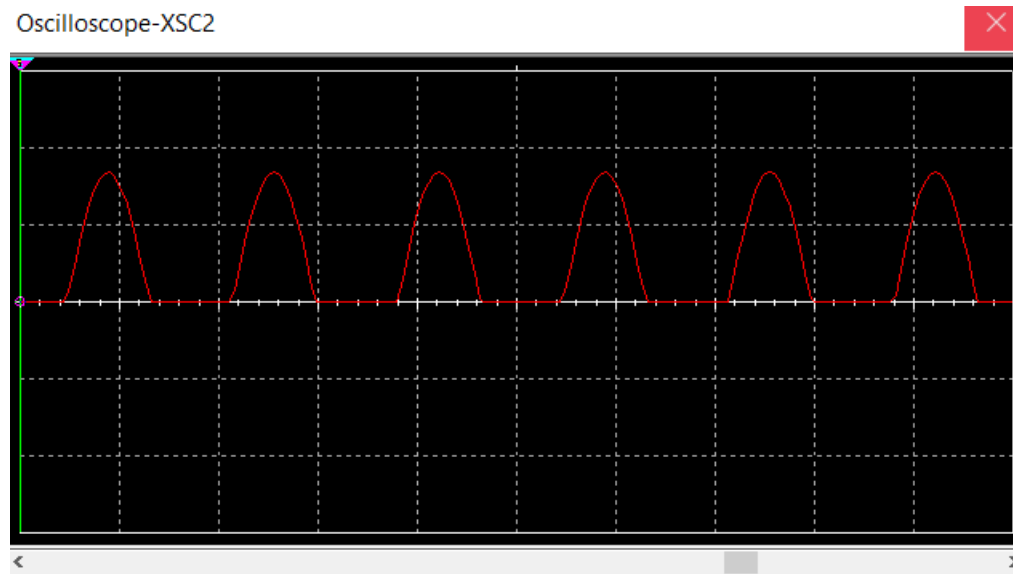


Fig 6.6 : Negative series clipper output

Conclusion:

Clipper circuit means to clip off any portion of any desired waveshape without distorting . We can use a clipper circuit in various purpose. Here unbiased means without using external dc source.

Experiment No: 07

Name of the experiment: Simulation of Series Clipper biased Circuit.

Objective:

- 1.To study how to form a biased clipper circuit with diode and a external dc source.
- 2.To understand the working principle of biased clipper circuit.
3. Design a circuit such a way that it can clip positive waveshape and negative waveshape with adding or subtracting required voltage waveshape.
4. To understand the impact of DC voltage source in the circuit.

Component requires at simulation:

- 1.Oscilloscope
- 2.IN4007 diode
- 3.Connecting wire
- 4.Resistor

Experimental Setup:

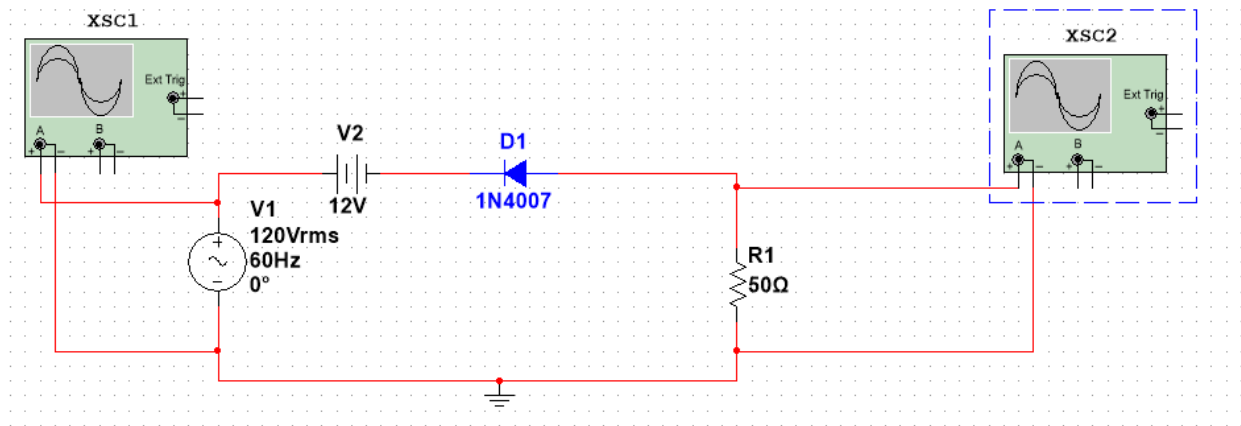


Figure 7.1 : Biased positive series Clipper

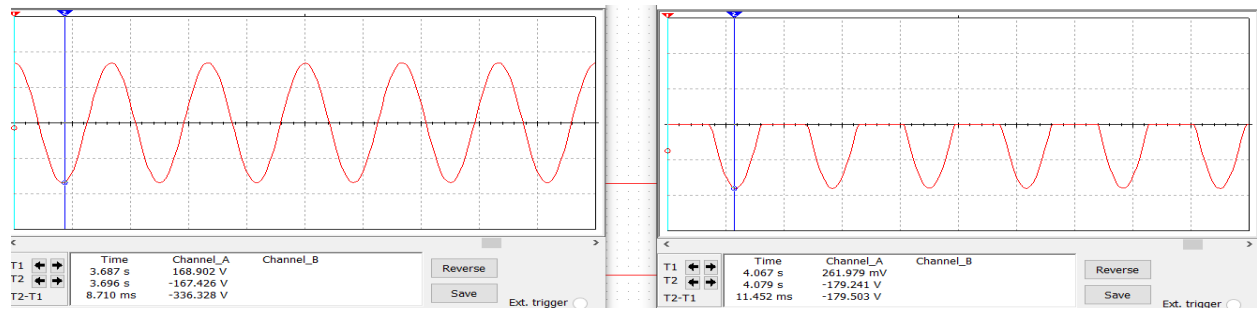


Figure 7.2 : Biased positive series clipper input and output graph.

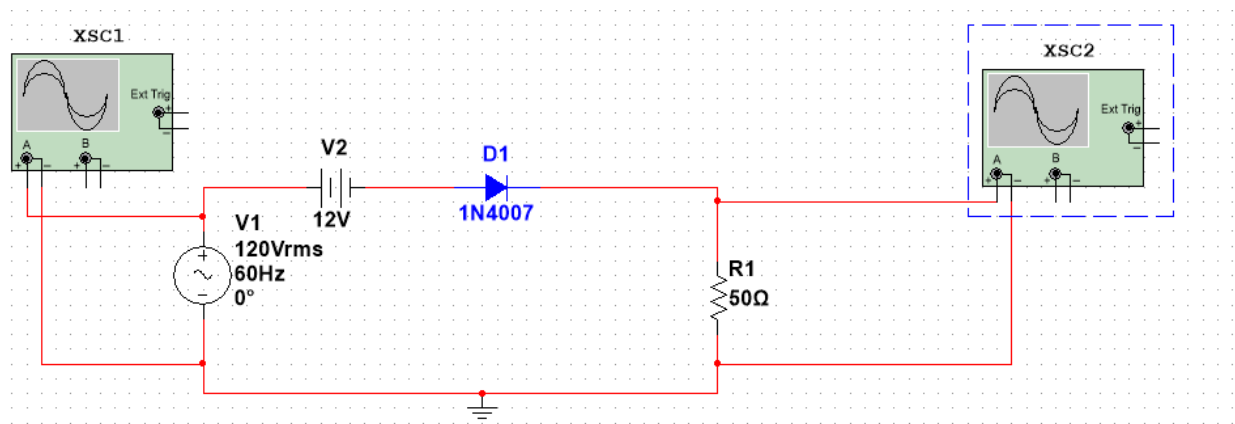


Figure 7.3 : Biased negative series clipper circuit.

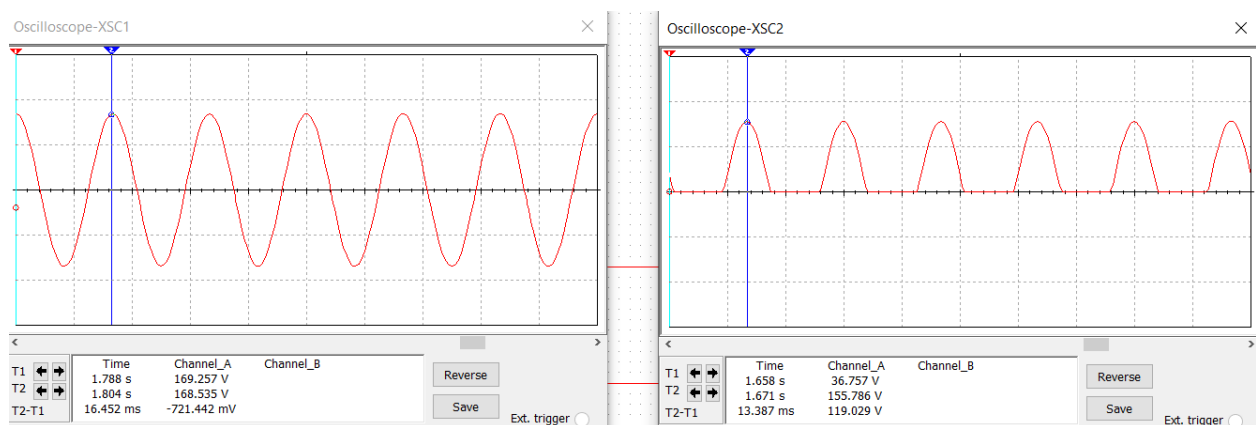


Figure 7.4 : Biased negative series clipper circuit input and output graph

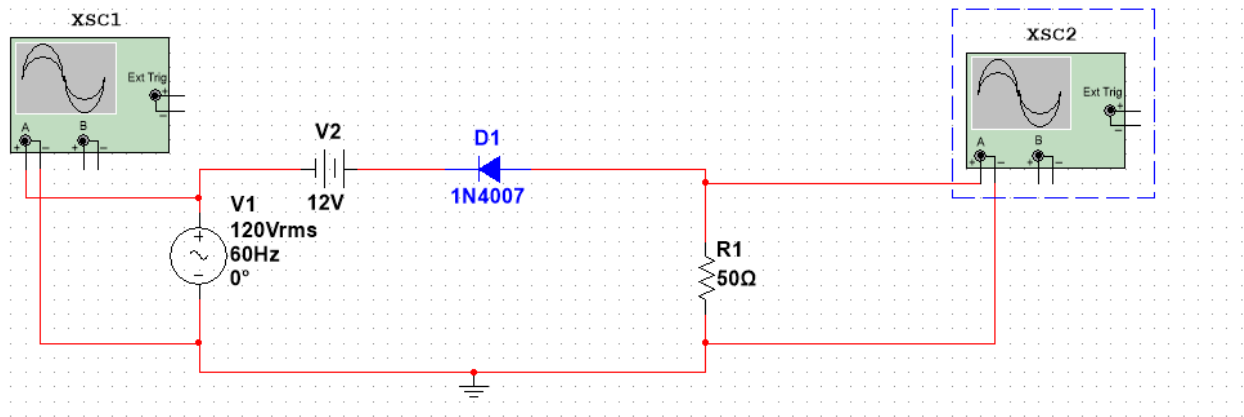


Fig 7.5: Biased positive series clipper.

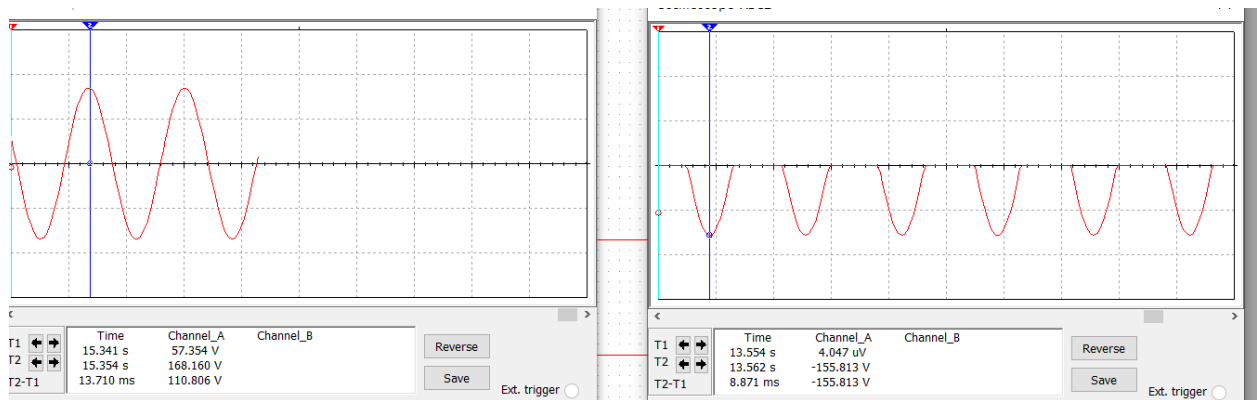


Fig 7.6: Biased positive series clipper output.

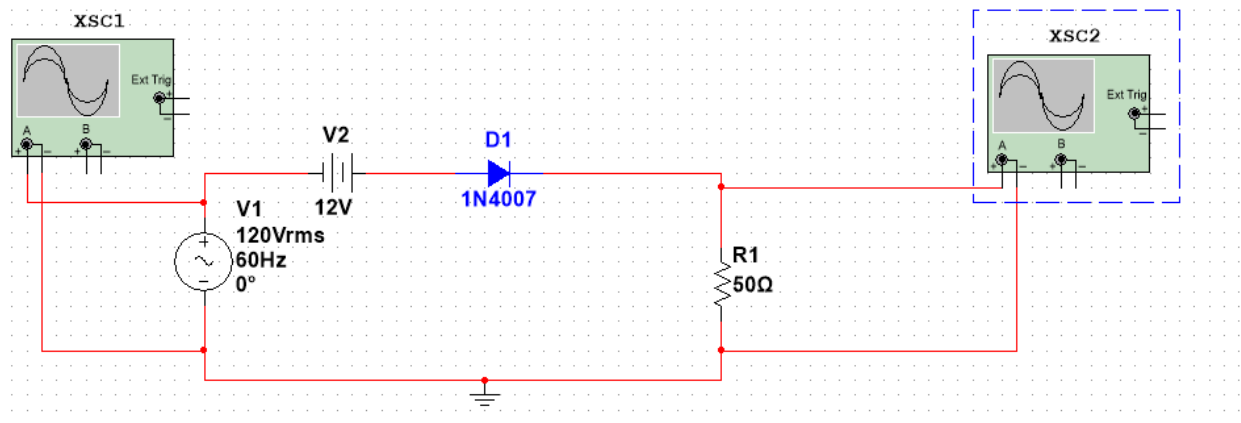


Fig 7.7: Biased negative series clipper.

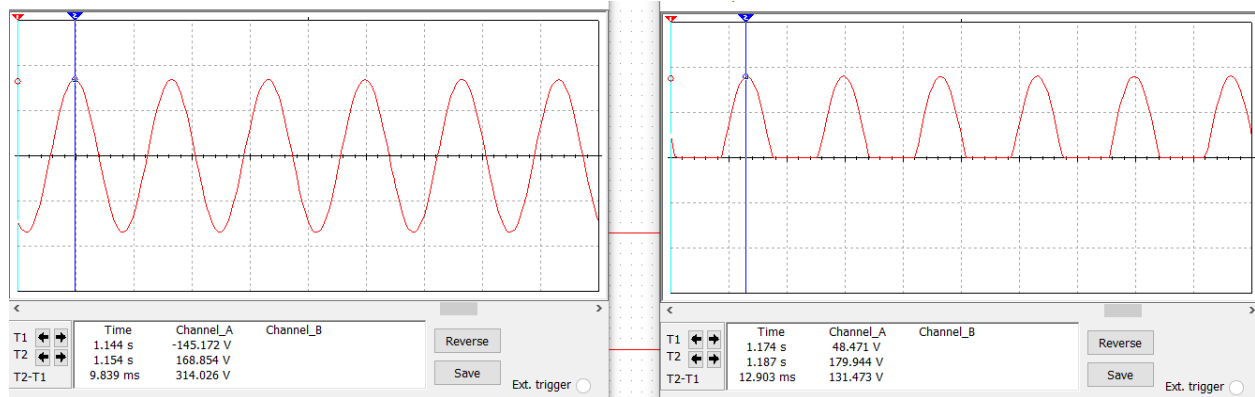


Fig 7.8: Biased negative series clipper output

Conclusion: We have simulated different combination of series positive and negative clipper circuit. And It has clearly shown that it has impact on circuit depends upon the connection.

Experiment No: 08

Name of the experiment: Simulation of parallel Clipper biased Circuit.

Objective:

- 1.To study how to form a parallel biased clipper circuit with diode and a external dc source.
- 2.To understand the working principle of parallel biased clipper circuit.
3. To understand the impact of DC voltage source in the parallel clipper circuit.

Component requires at simulation:

- 1.Oscilloscope
- 2.IN4007 diode
- 3.Connecting wire
- 4.Resistor

Experimental Setup:

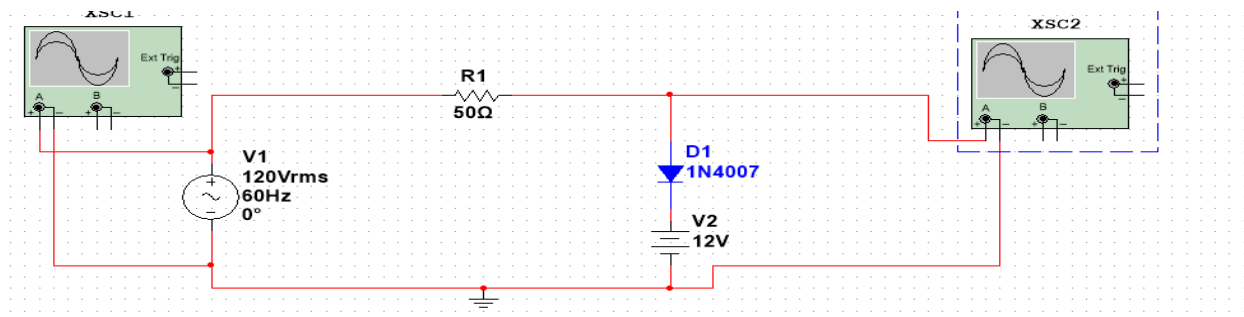


Figure 8.1: Biased positive parallel clipper.

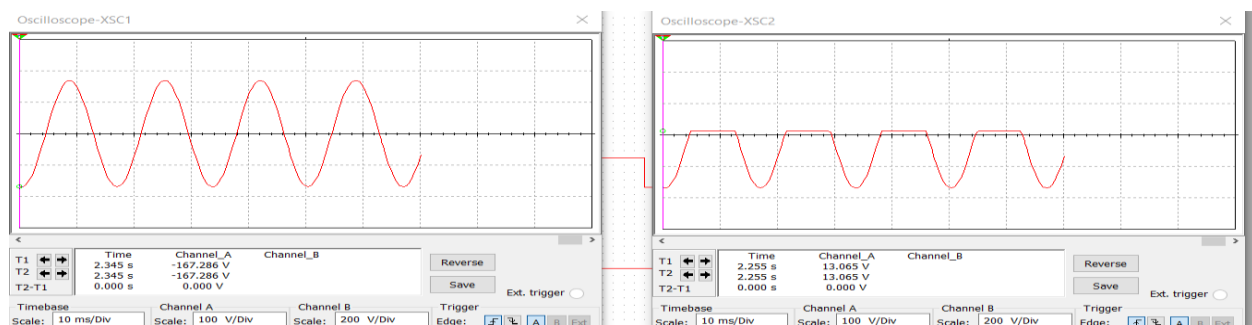


Figure 8.2: Biased positive parallel clipper output.

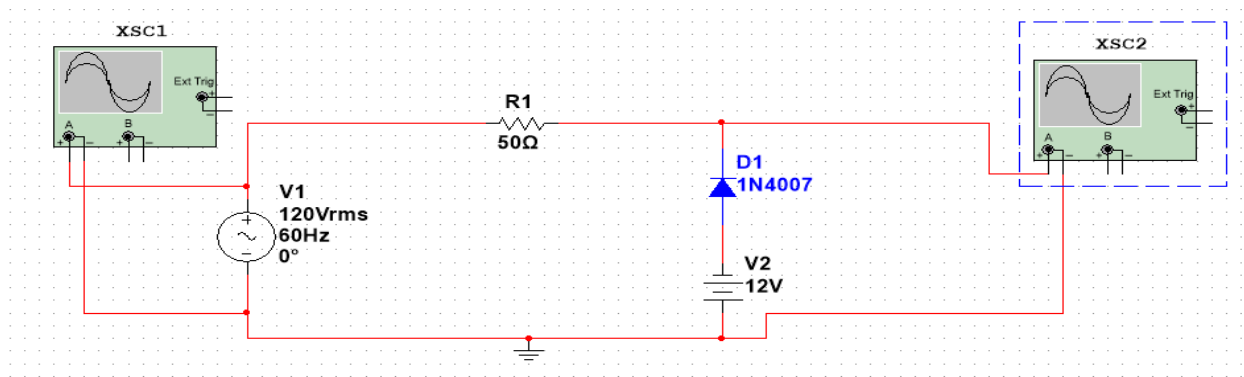


Figure 8.3: Biased negative parallel clipper.

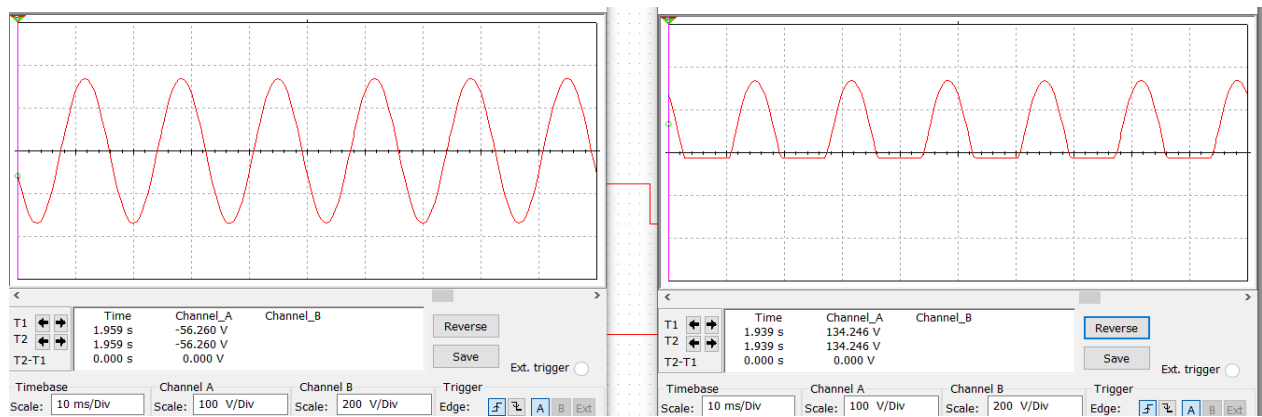


Figure 8.4: Biased negative parallel clipper output

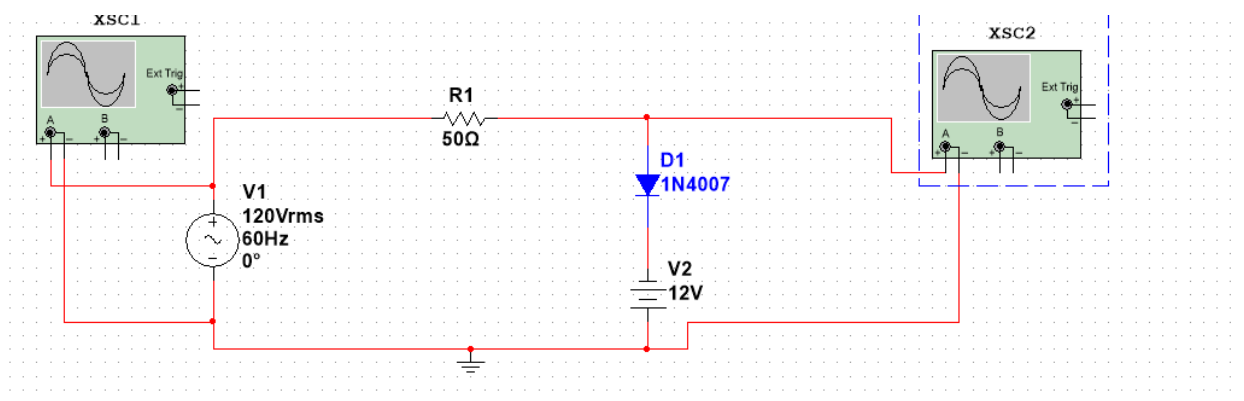


Figure 8.5: Biased positive parallel clipper.

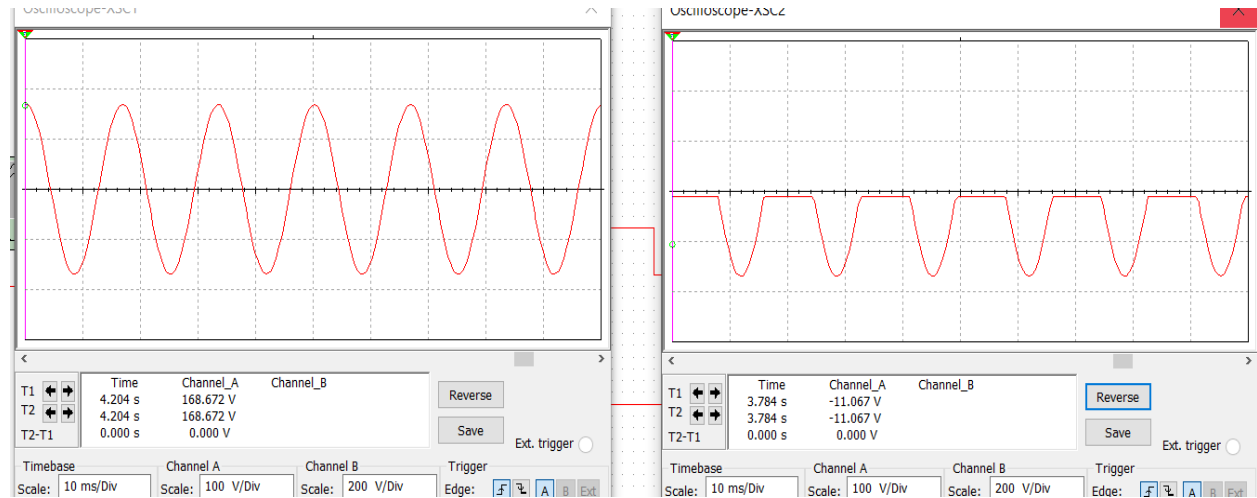


Figure 8.6: Biased positive parallel clipper output.

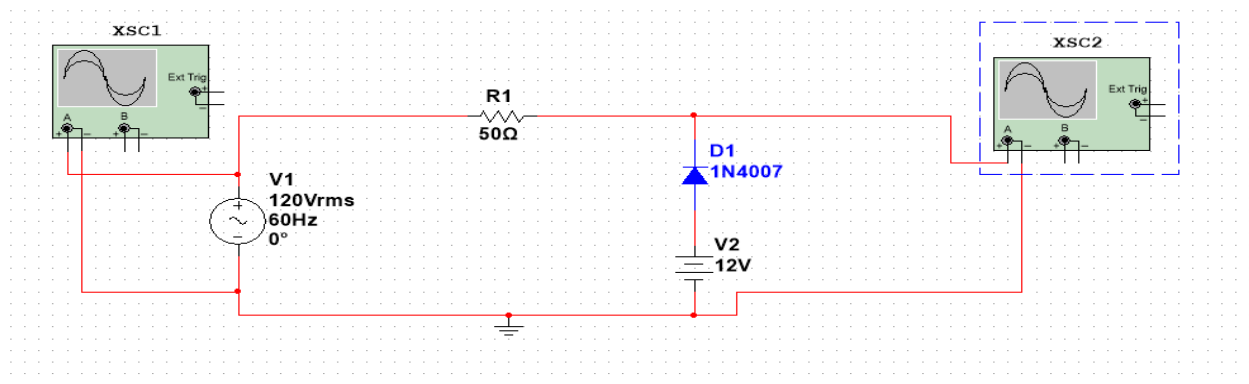


Figure 8.7: Biased negative parallel clipper.

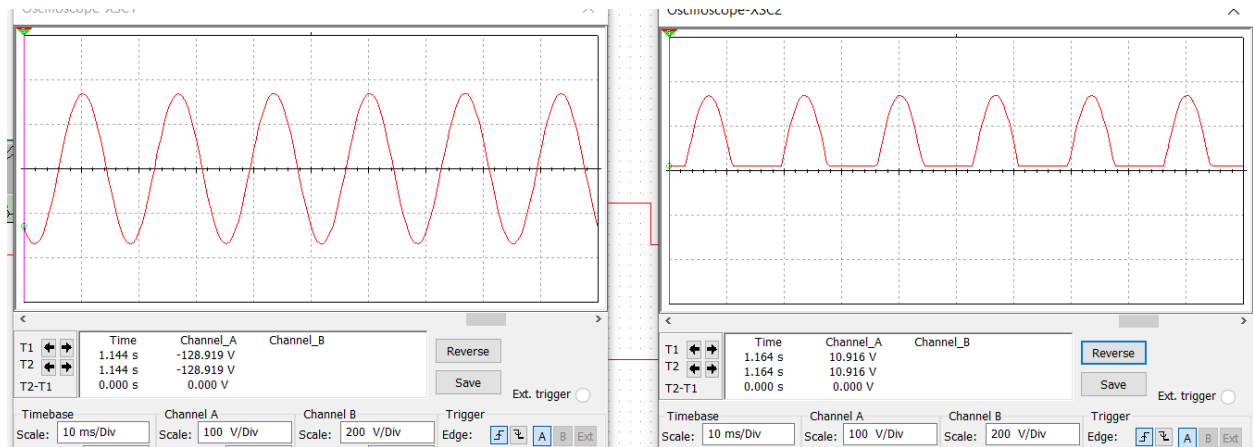


Figure 8.8: Biased negative parallel clipper output.

Conclusion : The clipper circuit design output waveform have been studied and the required parameter have been compared. The simulation is about the understanding of clipper circuits. And why they are formed and how to use diodes to make clipper circuits. Moreover, to analyze the working of diode in limiters or clippers and to find the waveforms.

Experiment No: 09

Experiment Name: Simulation of combination Clipper Circuit in NI Multisim

Objective:

1. To construct a Combination clipper circuit in NI Multisim.
2. To understand the working principle of combination clipper circuit.
3. To compare the output with simple clipper circuit.

Component requires at simulation:

- 1.Oscilloscope
- 2.IN4007 diode
- 3.Connecting wire
- 4.Resistor

Experimental Setup:

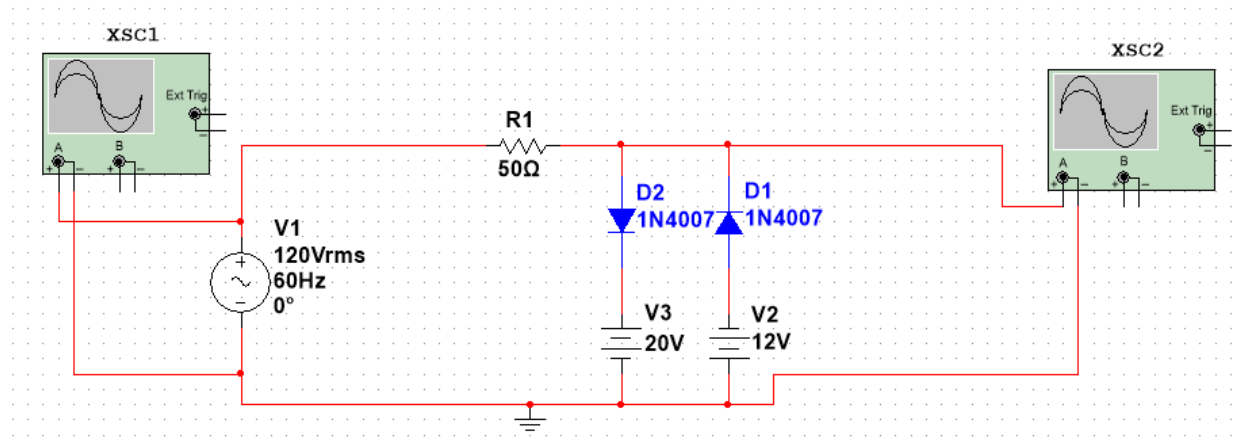


Figure 9.1: Circuit diagram of combination clipper circuit.

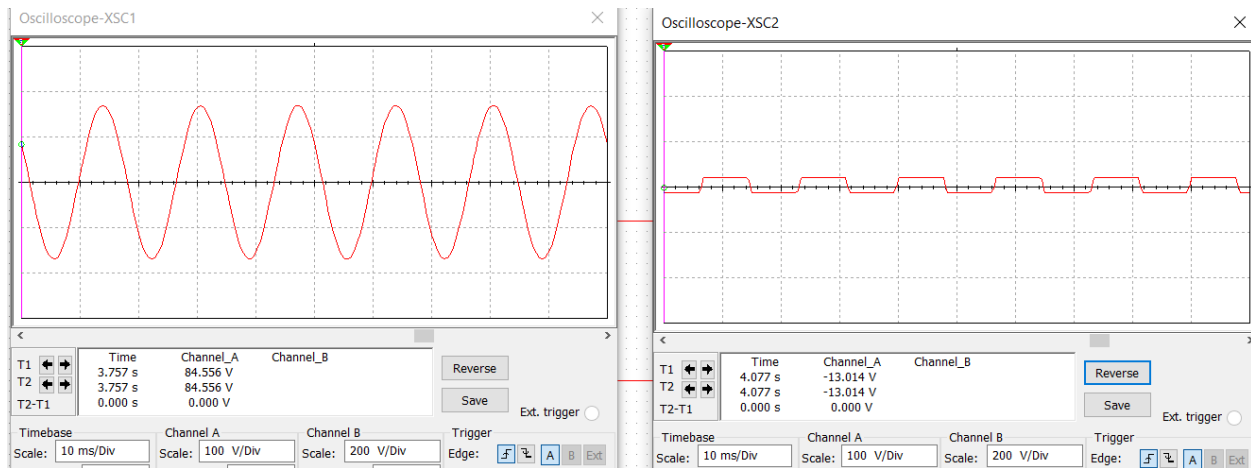


Figure 9.2: Wave shape of combination clipper circuit.

Conclusion: The reason combination clipper circuit is used because it can clip both from positive and negative input voltage. We can see from the output that it has clipped both from positive and negative side.

Experiment No: 10

Experiment Name: Simulation of unbiased Clamper Circuit.

Objective:

1. To construct a clamper circuit in NI Multisim.
2. To verify the waveshape of of the circuit.

Component requires at simulation:

- 1.Oscilloscope
- 2.IN4007 diode
- 3.Connecting wire
- 4.Resistor
5. capacitor

Experimental Setup:

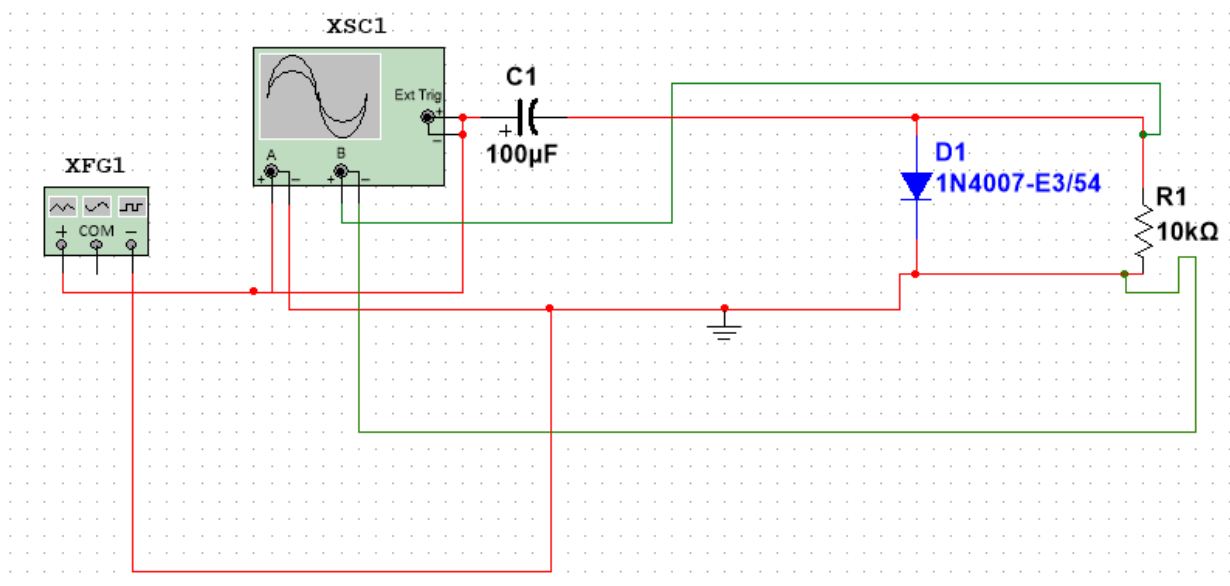


Figure 10.1: Negative Clamper circuit design .

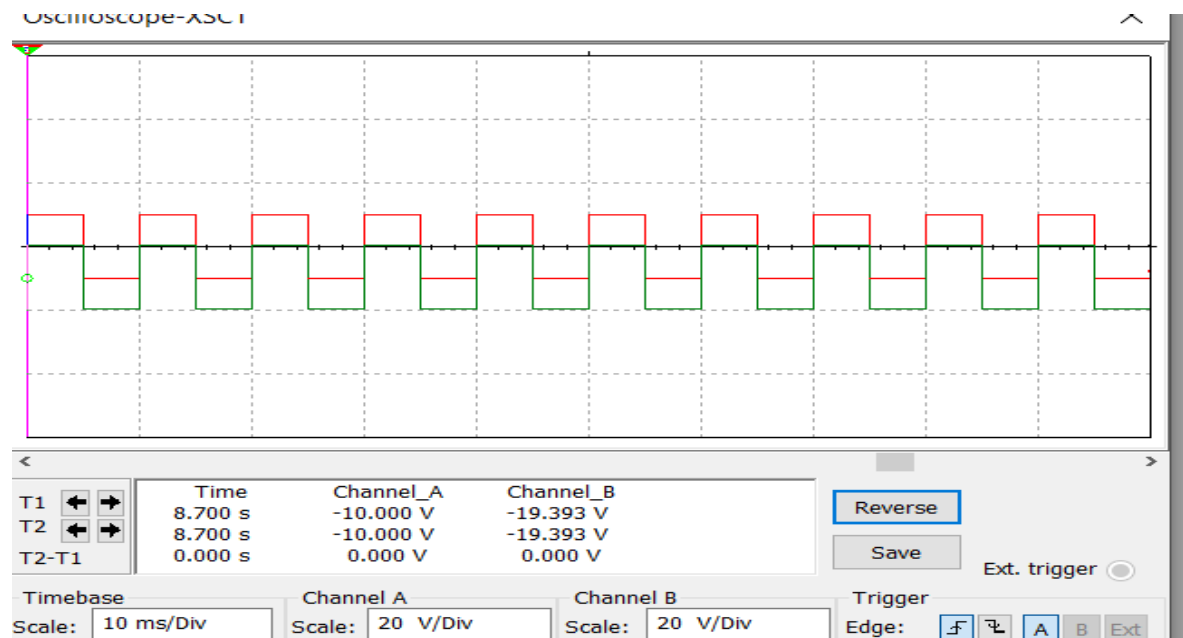


Figure 10.2: Negative Clamper circuit design input and output waveshape.

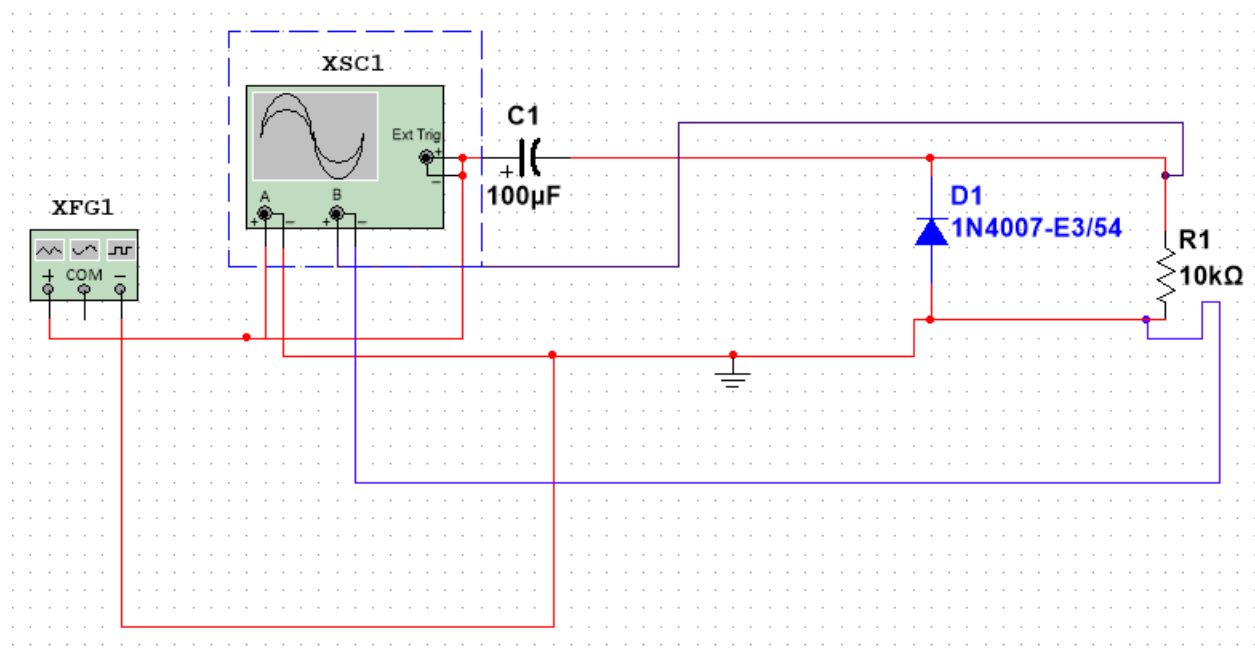


Figure 10.3: Positive Clamper circuit design .

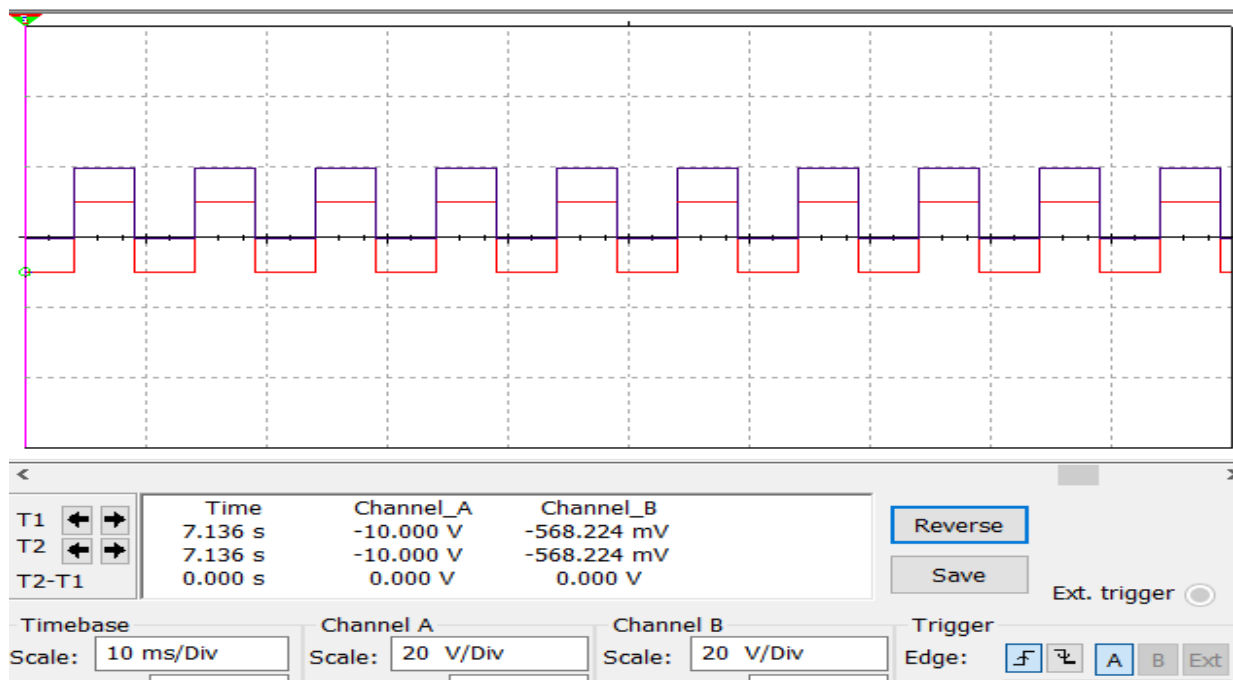


Figure 10.4: Positive Clamper circuit design input and output waveshape.

Conclusion: A clamper circuit is used to bind the upper or lower extreme of a waveform. Its used in removing the distortions and identifications of polarity of the circuits.

Experiment No: 11

Experiment Name: Simulation of biased parallel clamper circuit at NI Multisim.

Objective:

1. To design a biased clamper circuit at NI Multisim.
2. To analyze the effect of DC voltage source at the circuit.
3. To differentiate the resultant waveform of biased clamper circuit and unbiased clamper circuit.

Components requires at Simulation:

1. Function Generator
2. Capacitor
3. IN4007 diode
4. Resistor
5. Oscilloscope
6. Connecting Wire.

Experimental Setup:

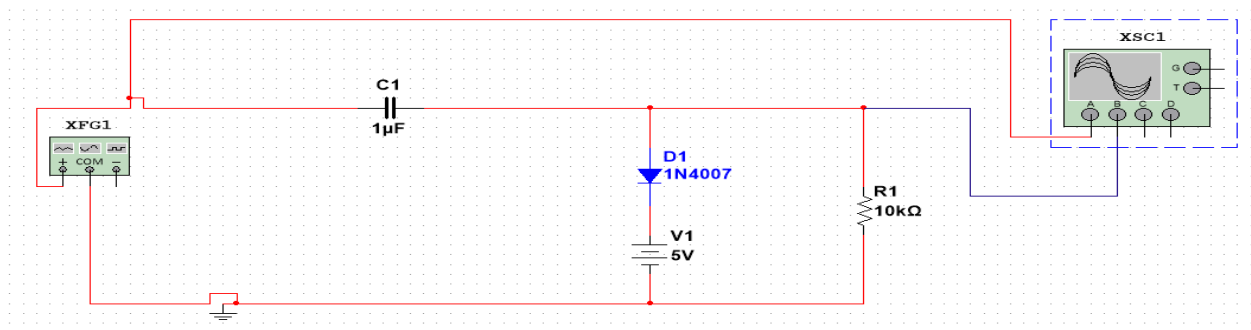


Figure 11.1: Biased negative parallel clamper circuit .

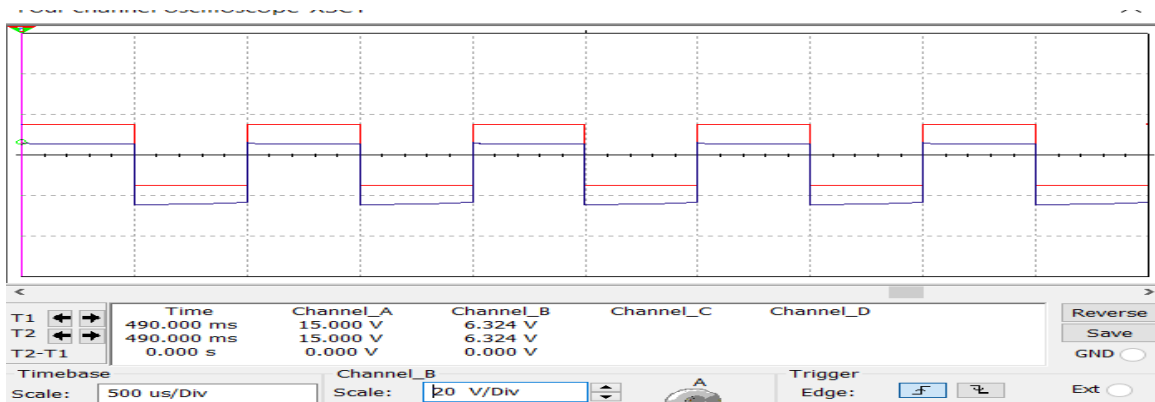


Figure 11.2: Biased negative parallel clamper circuit input and output waveform .

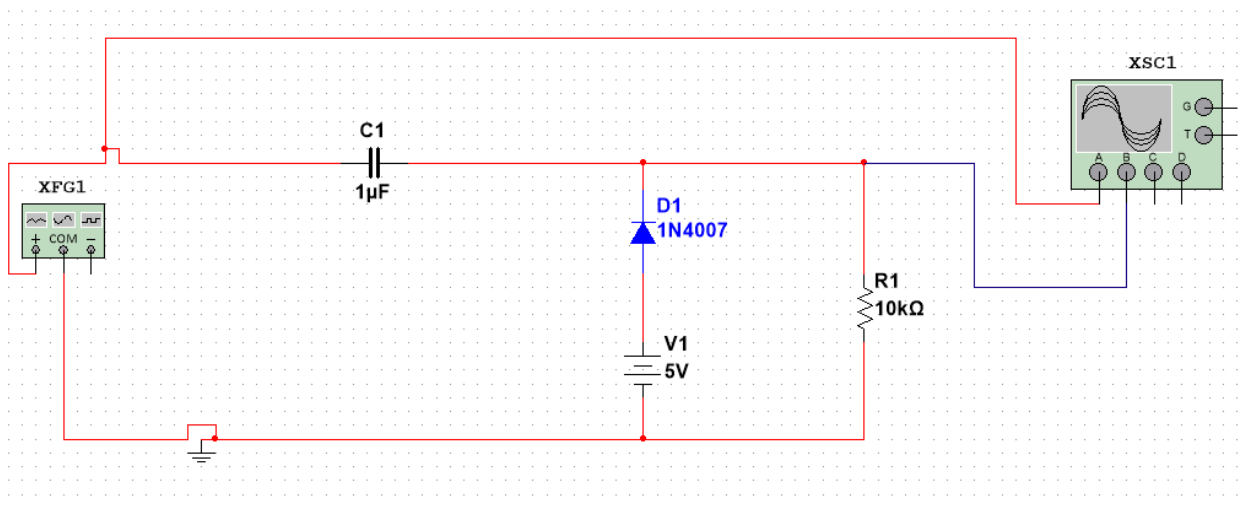


Figure 11.3: Biased positive parallel clamper circuit

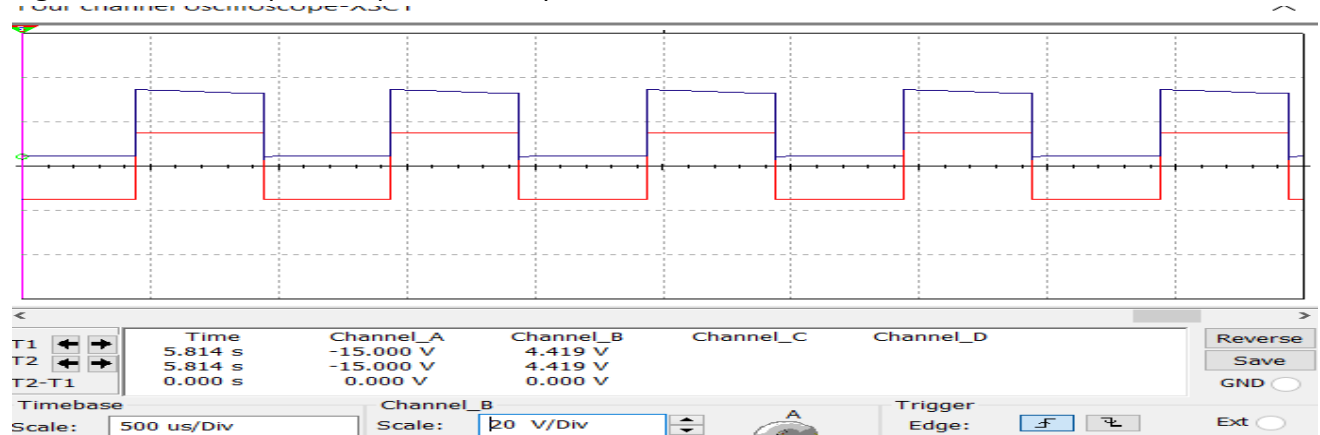


Figure 11.4: Biased positive parallel clamper circuit output.

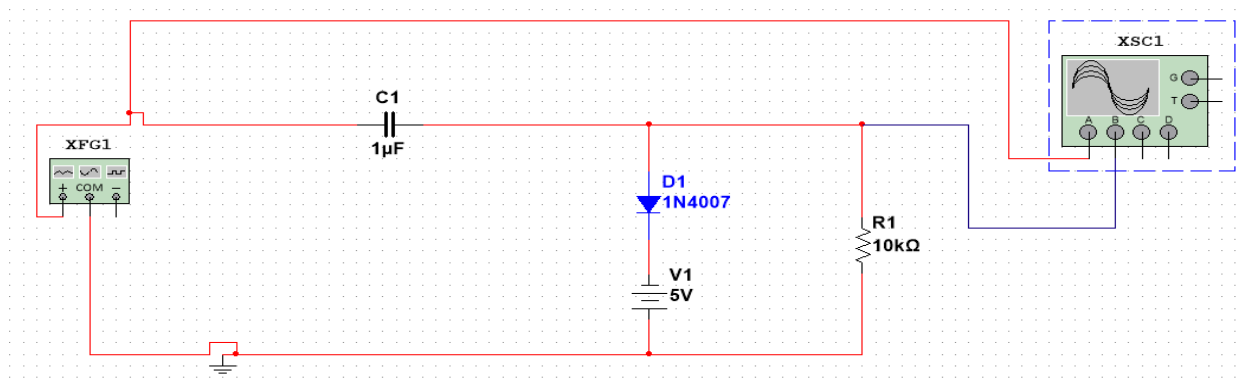


Figure 11.5: Biased negative parallel clamper circuit .

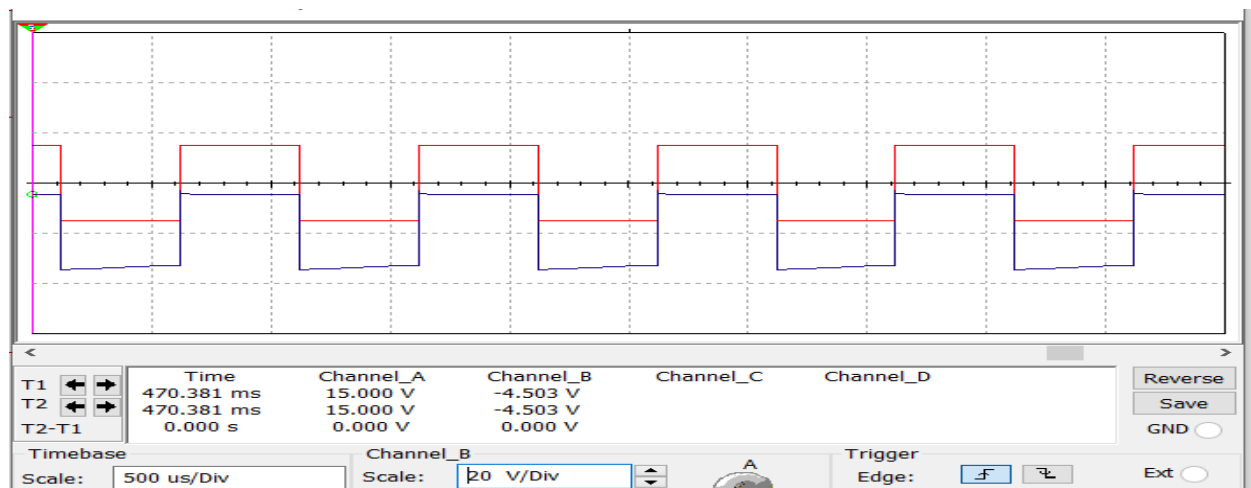


Figure 11.6: Biased negative parallel clamper circuit output.

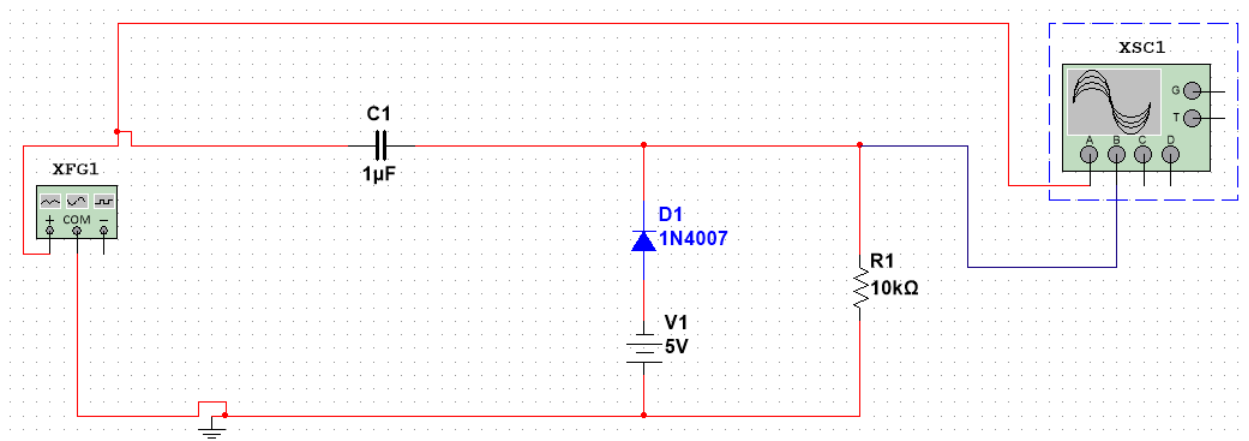


Figure 11.7 : Biased positive parallel clamper circuit .

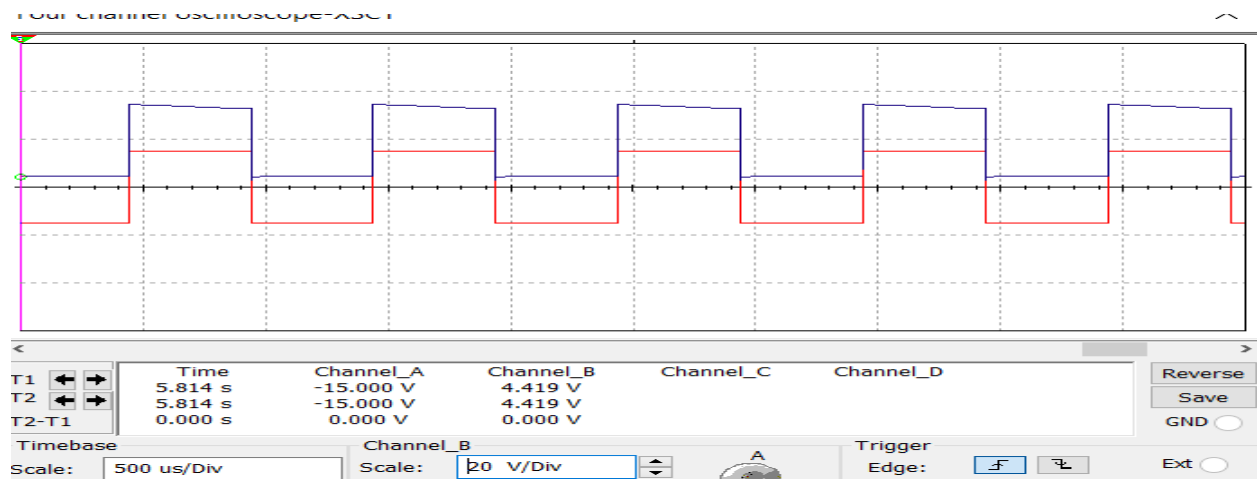


Figure 11.8 : Biased positive parallel clamper circuit output.

Conclusion: A clamper biased circuit is used to bind the upper or lower extreme of a waveform with the biasing DC voltage. Its used in removing the distortions and identifications of polarity of the circuits.

Experiment No: 12

Experiment Name: simulation of Zener diode and characteristics at NI Multisim of VI curve of Zener diode.

Objective:

1. To study and verify the voltage-current relation in Zener diodes by applying a voltage across it.
2. To construct a circuit with Zener diode.
3. To understand Zener diode working principle properly.

Component require at simulation:

1. Zener diode (1N5221B) .
2. Resistor
3. Variable voltage source.
4. Connecting Wire.

Experimental Setup:

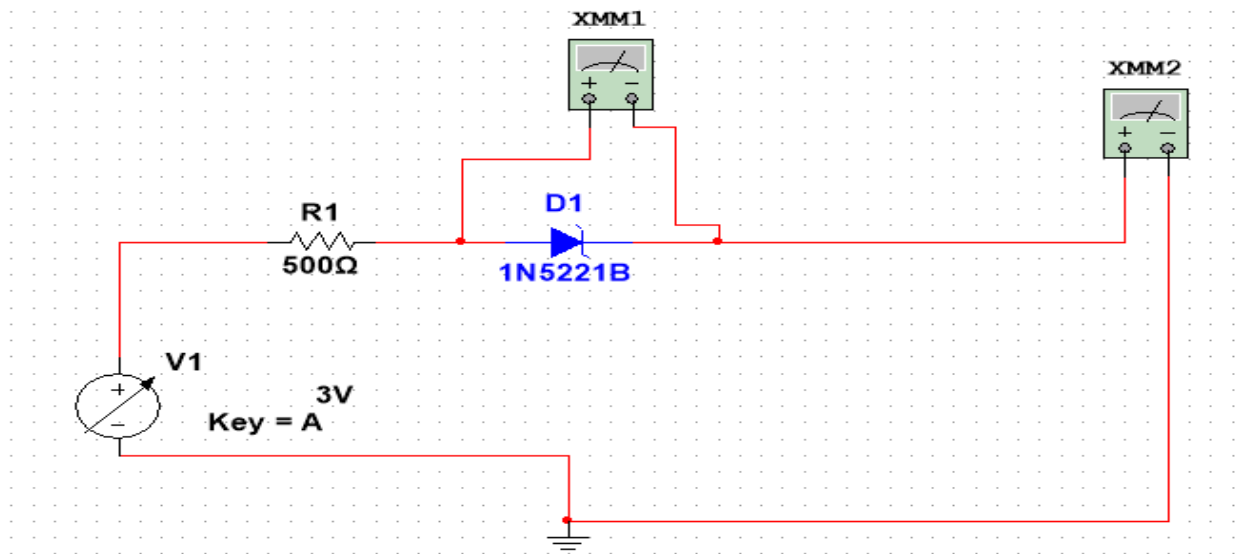


Figure 12.1 : Zener diode characteristics circuit for forward bias.

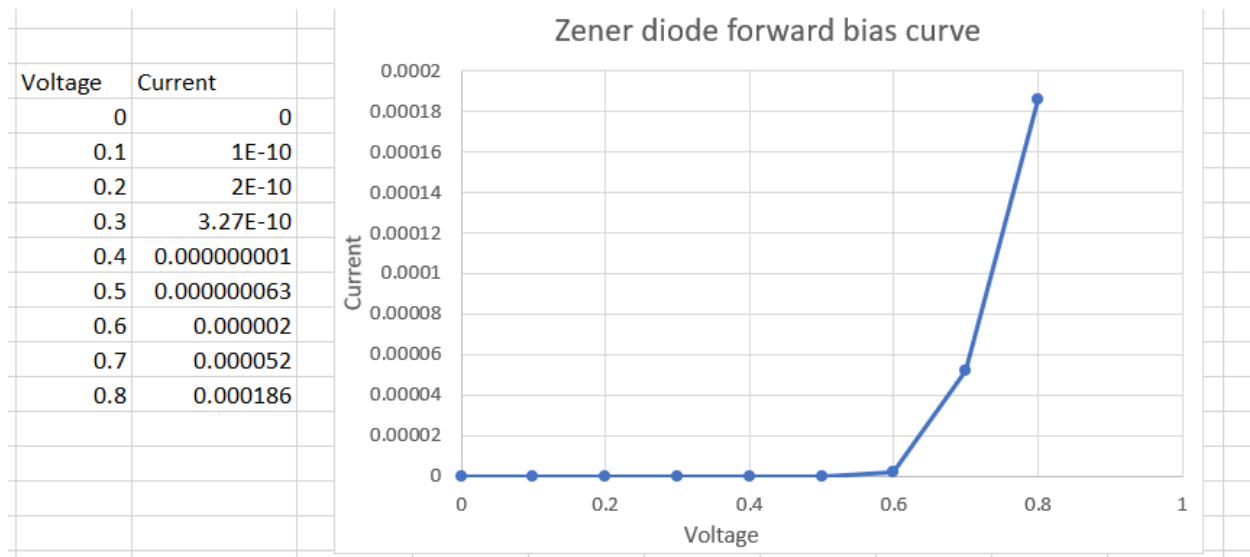


Figure 12.2 : Zener diode characteristics graph for forward bias.

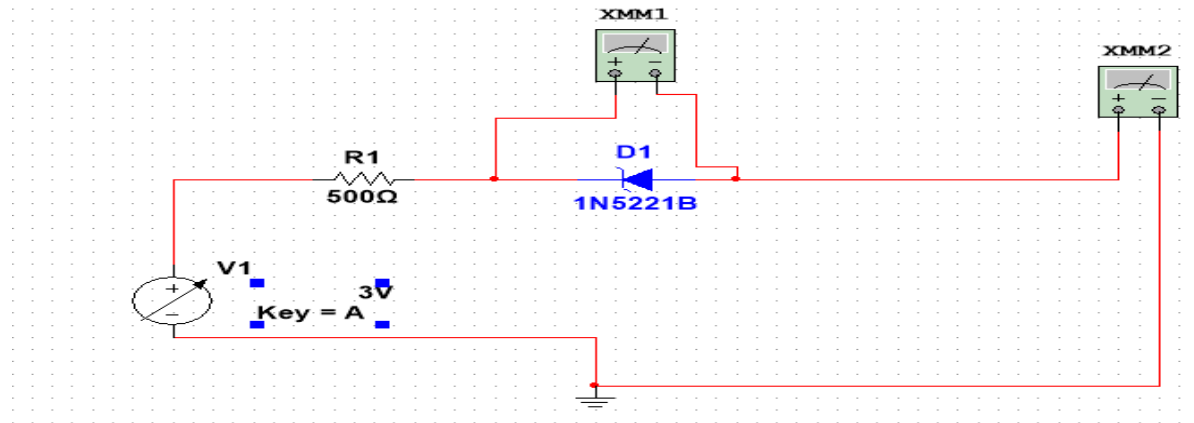


Figure 12.3 : Zener diode characteristics circuit for reverse bias.

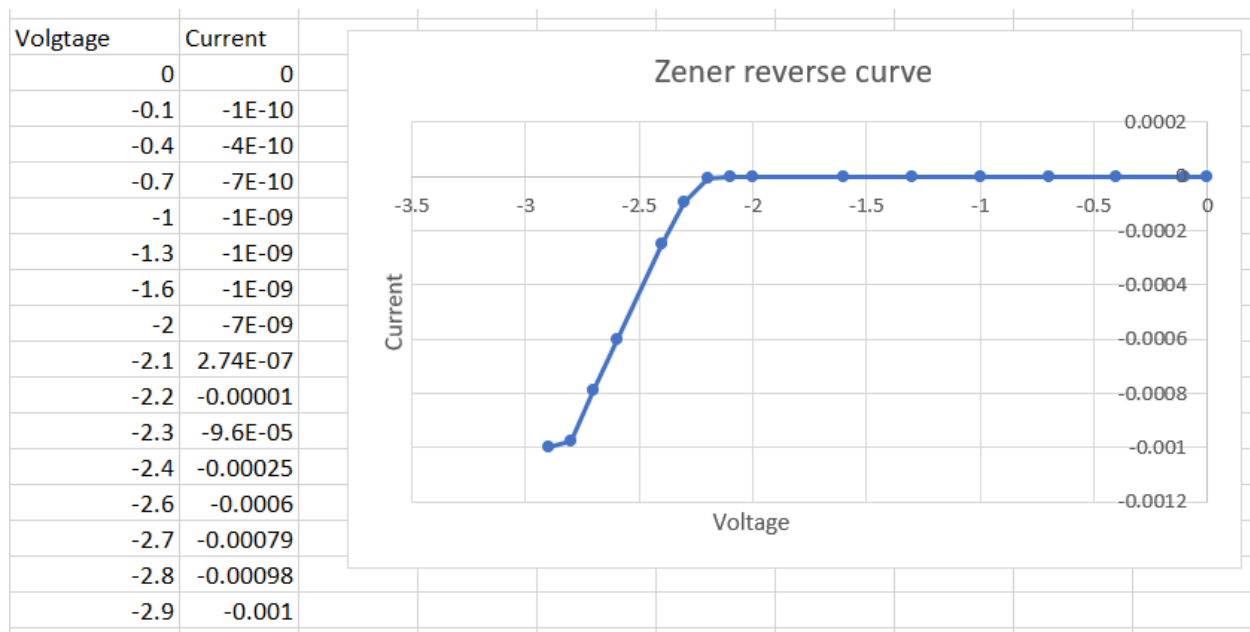


Figure 12.2 : Zener diode characteristics graph for reverse bias.

Conclusion: Zener diode are heavily doped than ordinary diodes. They have extra thin depletion region. When the diode is in forward bias then it acts like an forward bias of an ordinary diode. But the depletion region regains its original position after removal of the reverse voltage in Zener diode whereas in regular diode they don't and hence they get destroyed.

Experiment No: 13

Experiment Name: Simulation of Zener diode voltage regulation and graph analysis.

Objective:

1. To analyze how Zener diode works as a voltage regulator .
2. To develop a Zener diode voltage regulator circuit.

Component require for Simulation:

1. Resistor
2. Zener diode (1N5221B) .
3. Variable power source
4. Connecting Wire.

Experiment setup:

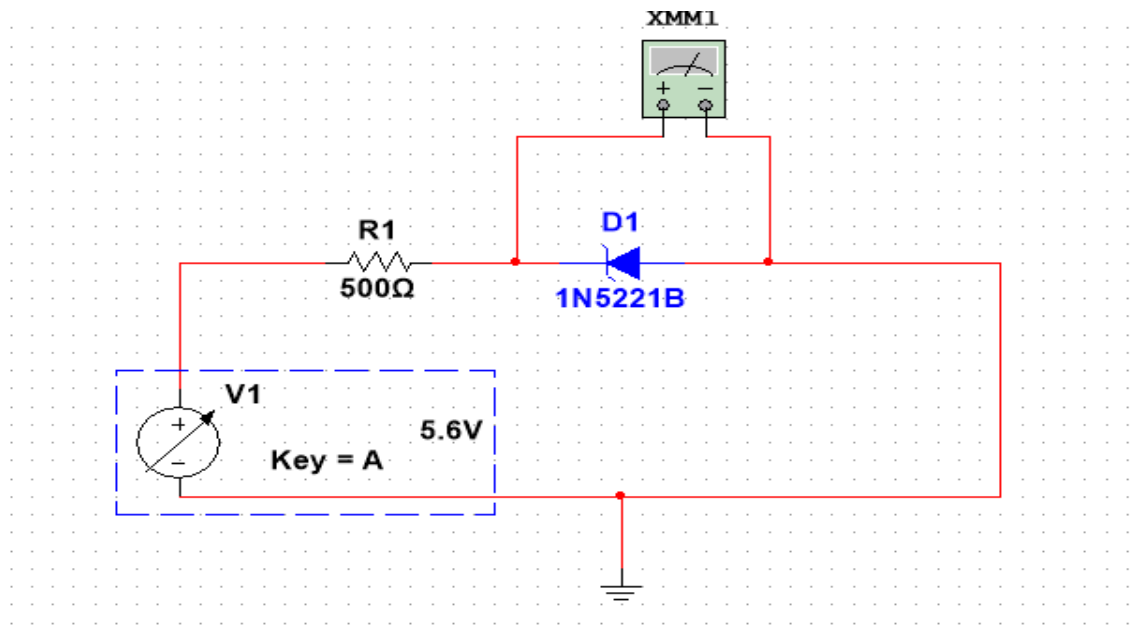


Figure 13.1 : Zener diode regulation circuit.

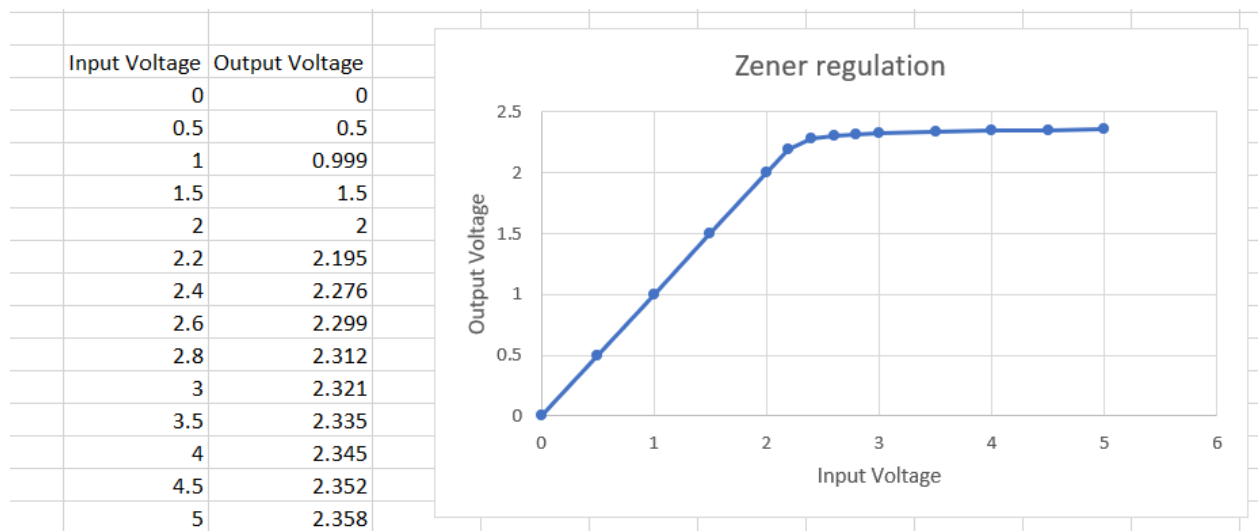


Figure 13.2: Zener diode regulation circuit output.

Conclusion: The function of a regulator is to provide a constant output voltage to a load connected in parallel with it in spite of the ripples in the supply voltage or the variation in the load current. Here the output voltage is almost 2.4 volt. That's why after increasing so much input voltage the output voltage is fixed at 2.3 to 2.4 v.

Experiment No: 14

Experiment Name: Simulation of input and output characteristics of Common Emitter BJT in NI Multisim.

Objective:

1. To find out the characteristics graph of common emitter graph of BJT in simulation.
2. To construct a circuit with transistor.

Component requires at simulation:

1. Transistor (BC548A)
2. Resistor
3. Connecting Wire
4. DC voltage source.

Experimental Setup:

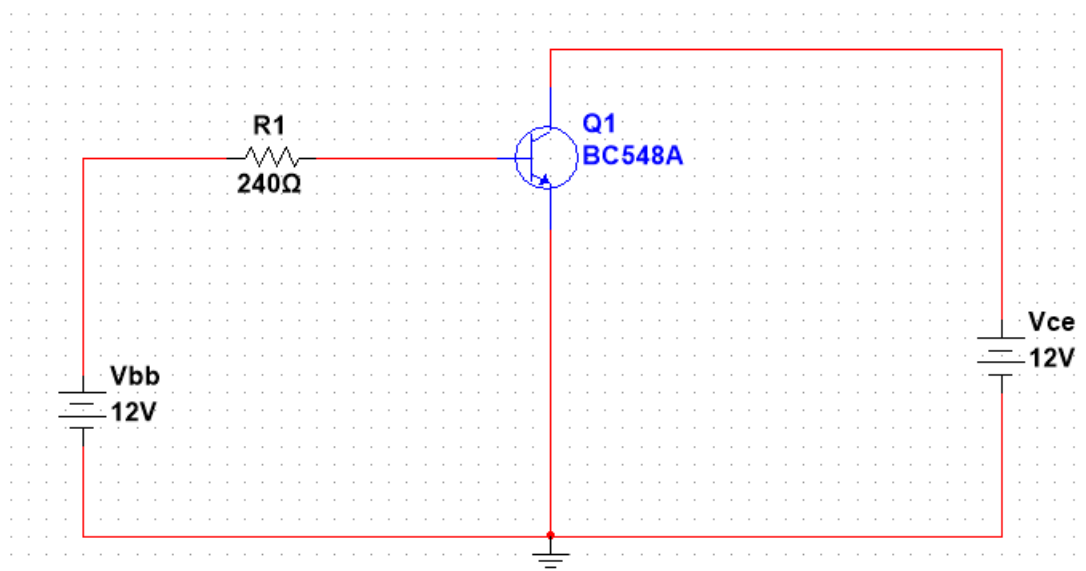


Figure 14.1: Experimental setup of input and output characteristics of Common Emitter

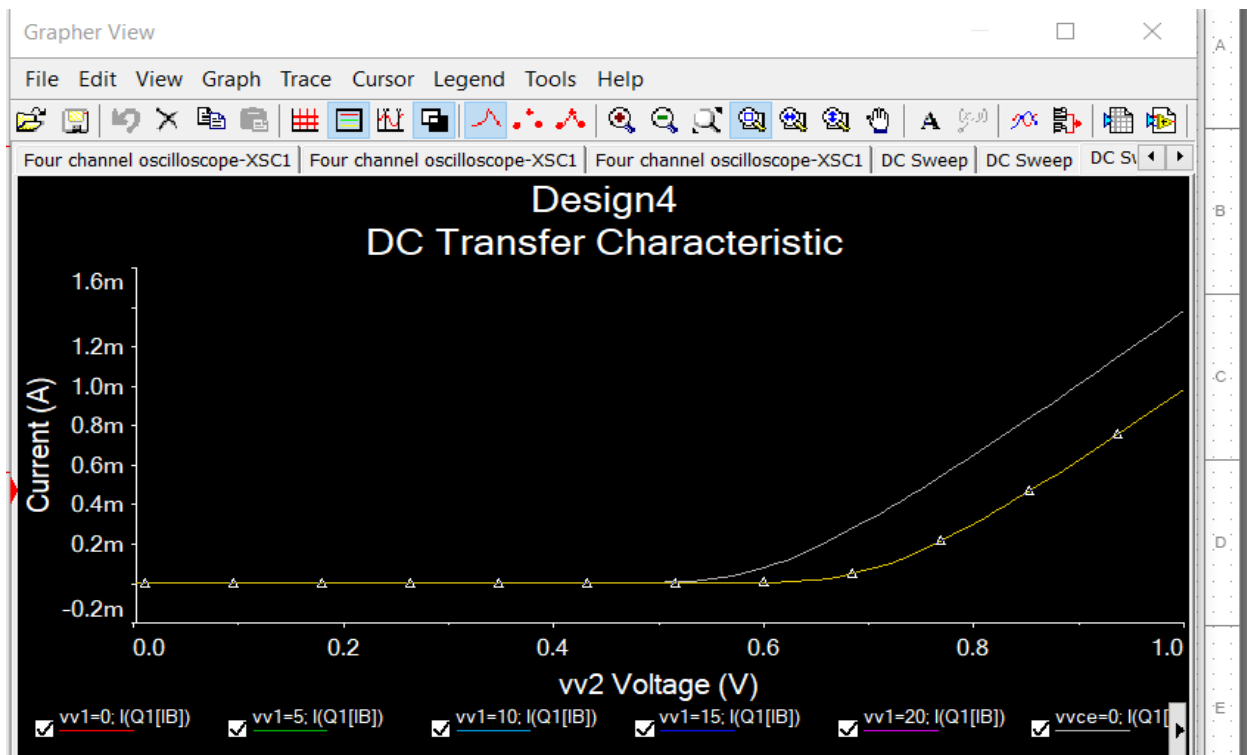


Figure 14.2: Graph of input characteristics of Common Emitter BJT.

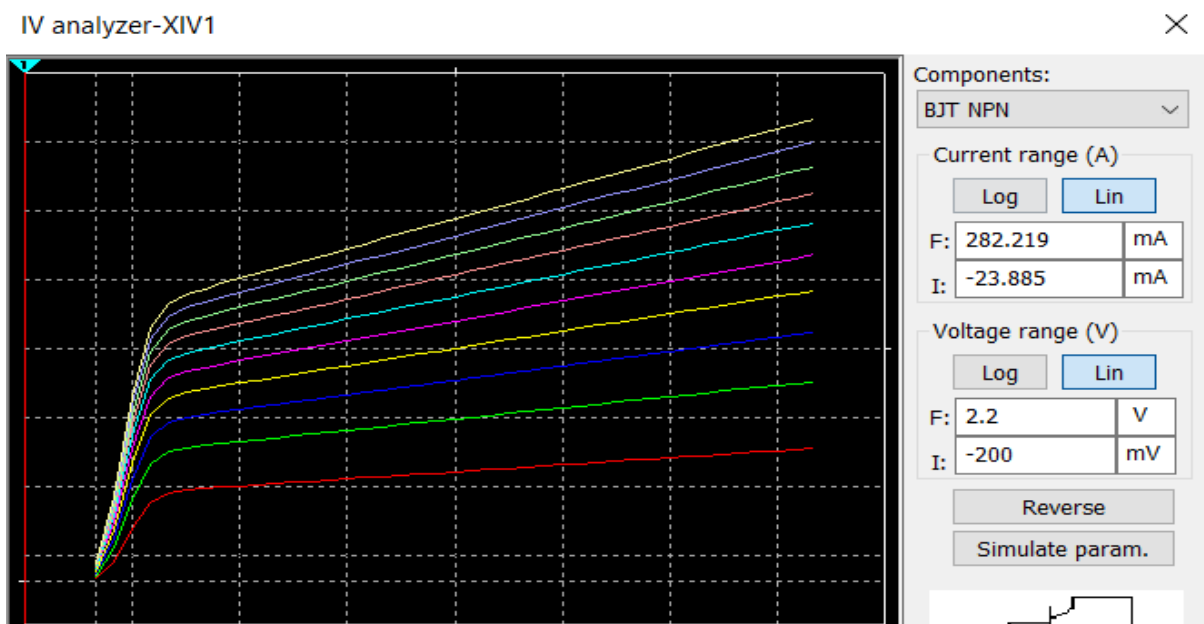


Figure 14.3: output characteristics of Common Emitter BJT.

Conclusion: From the input characteristics graph we have seen that the characteristics resembles that of a forward biased diode curve. This is expected since the base emitter section of transistor is a diode and it is forward biased.

The output characteristics graph shows that above knee voltage I_c is almost constant. However a small increase in I_c with increase V_{CE} is caused by the collector depletion layer getting wider and capturing a few more majority carriers before electro hole combination occur in the base area.

Experimental No: 15

Experimental Name: Simulation of amplification of BJT

Objective:

1. To design a common emitter BJT NPN transistor which will be used for amplification.
2. To show the amplification in oscilloscope.
3. To show the phase shift of a transistor.

Component Requires for simulation:

1. Function Generator
2. Capacitor.
3. DC voltage source.
4. Resistor
5. Transistor (BC107BP)
6. Oscilloscope.

Experimental Setup:

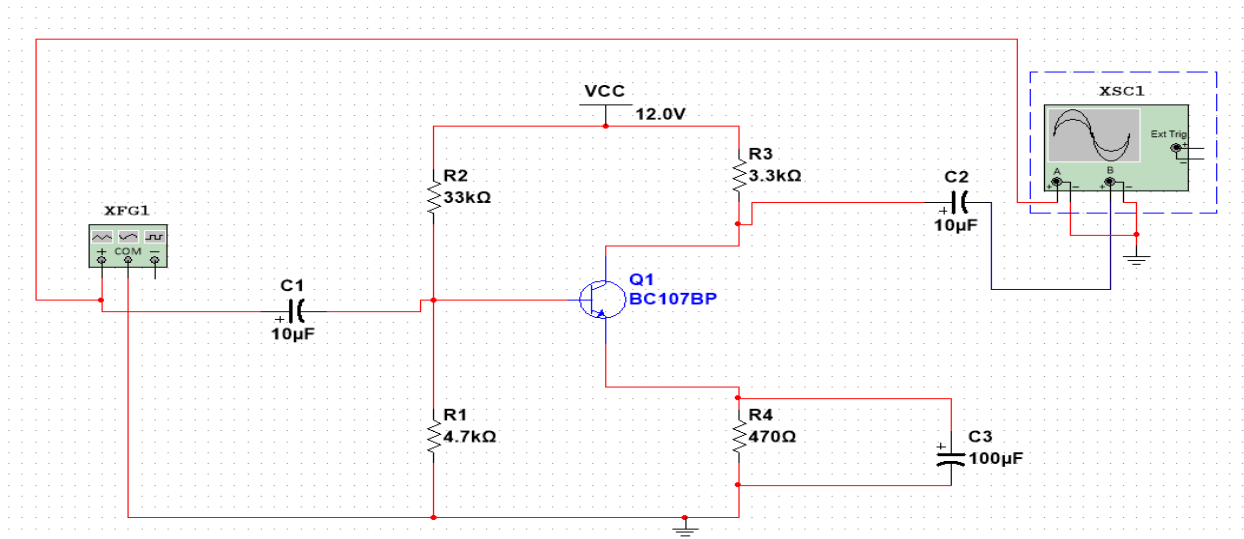


Figure 15.1: Circuit of BJT amplifier.

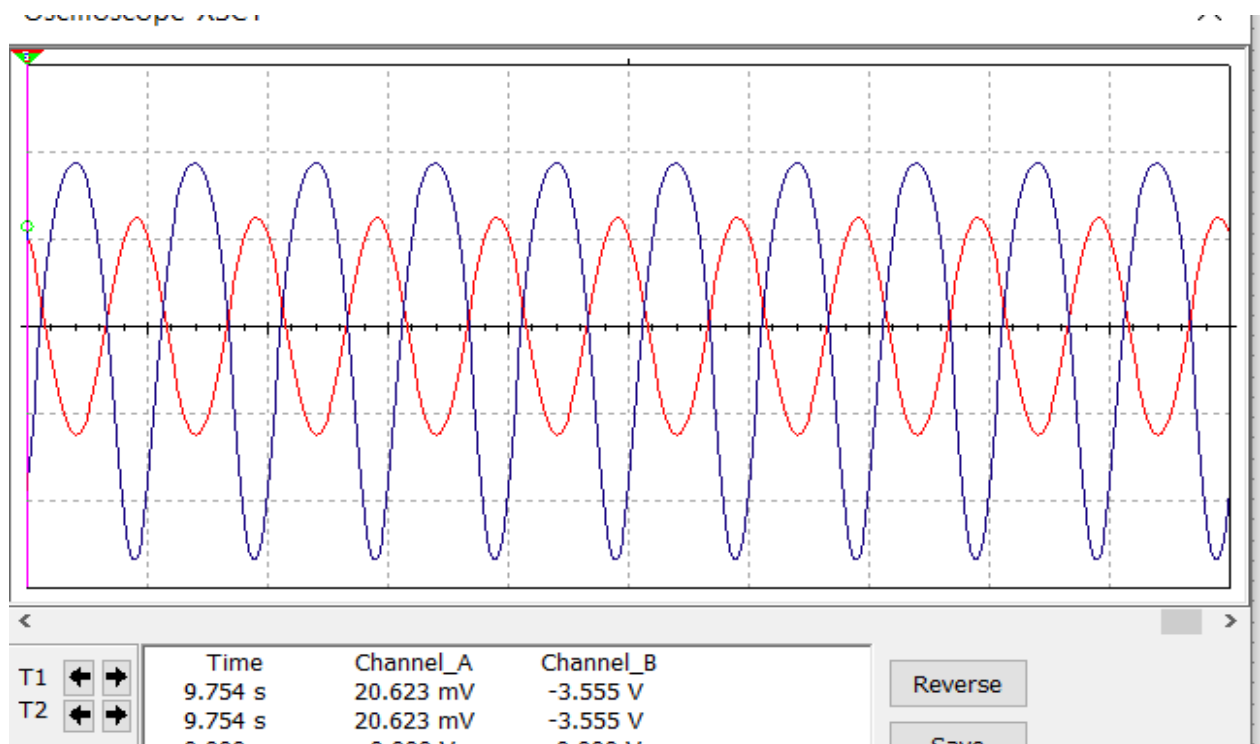


Figure 15.2: Output of amplification of BJT amplifier.

Conclusion: In this experiment we have learnt that what a common emitter amplifier is and how it works. We have seen that it not only amplifies this signal but also change the phase shift of the signal. Thus how a common emitter amplifier works.

Experiment No: 16

Experiment Name: Comparison of Resistive, Inductive and Capacitive Circuit in NI Multisim.

Objective:

1. To construct different combination of circuit using resistor, Inductor and capacitor.
2. To analyze the change of waveform for each circuit.

Component requires for simulation:

1. AC source.
2. Resistor
3. Capacitor
4. Inductor.
5. Oscilloscope
6. Connecting wire.

Experimental Setup:

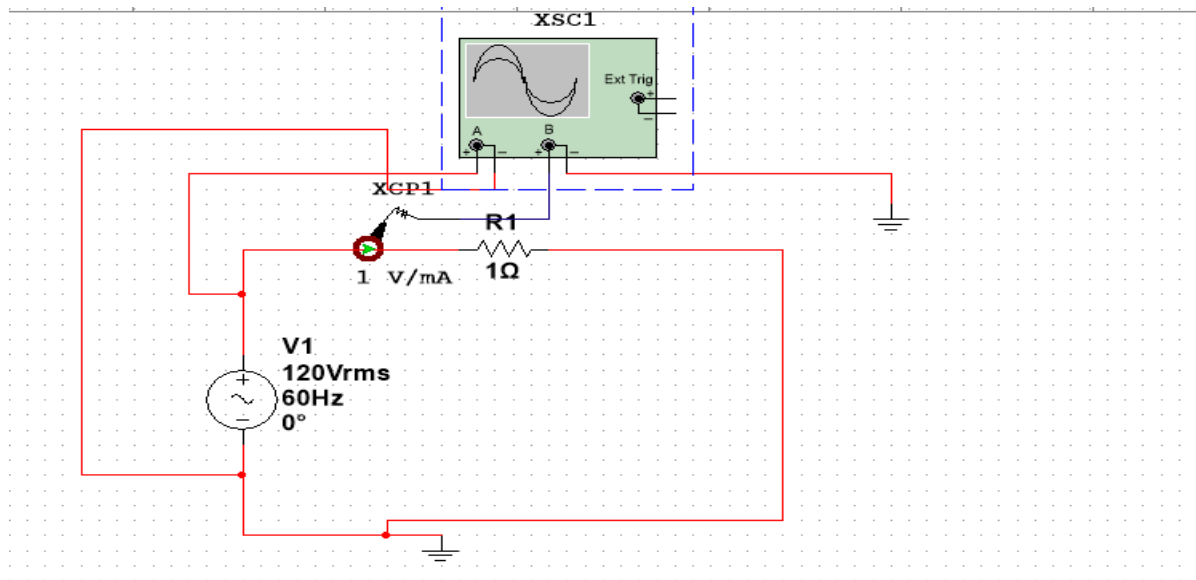


Figure 16.1: Resistive circuit diagram.

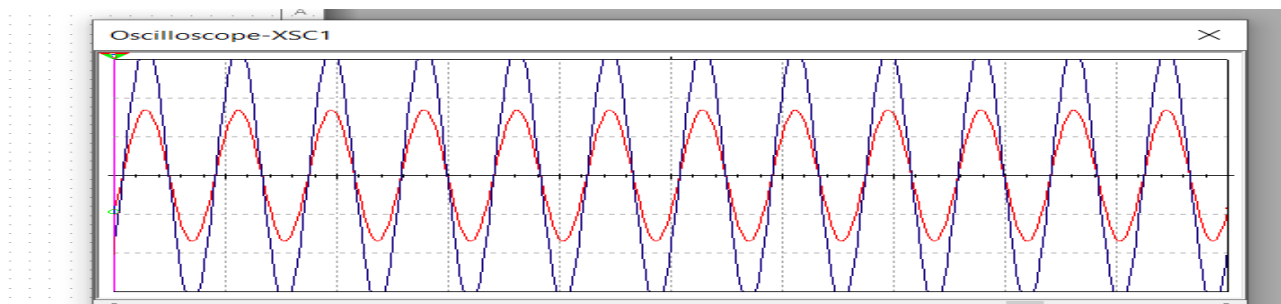


Figure 16.2: Resistive circuit output.

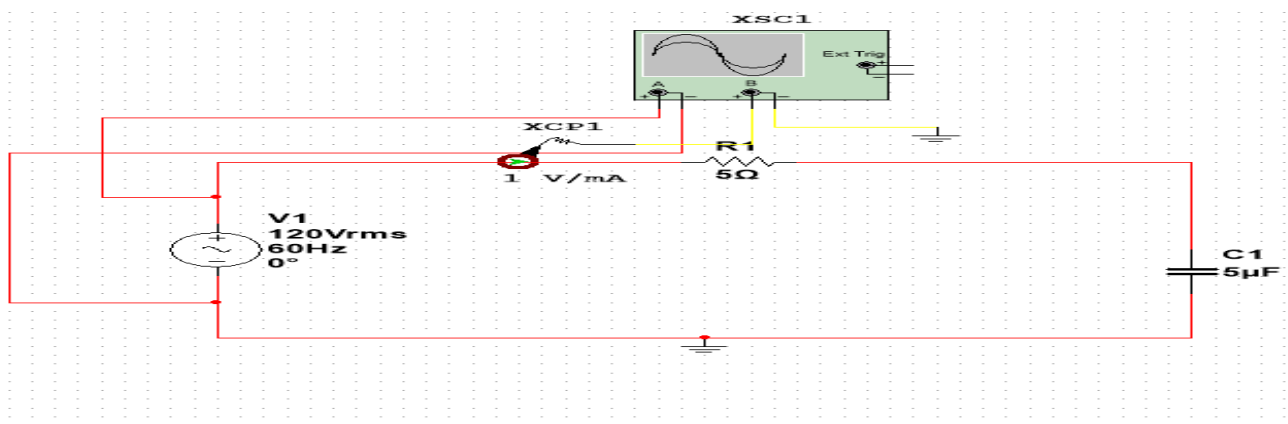


Figure 16. 3: RC circuit diagram.

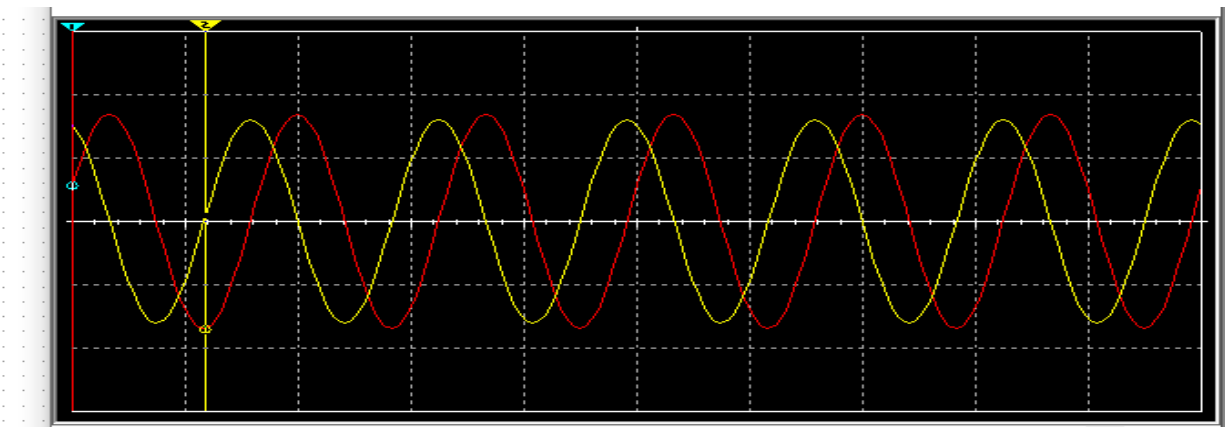


Figure 16. 4: RC circuit output

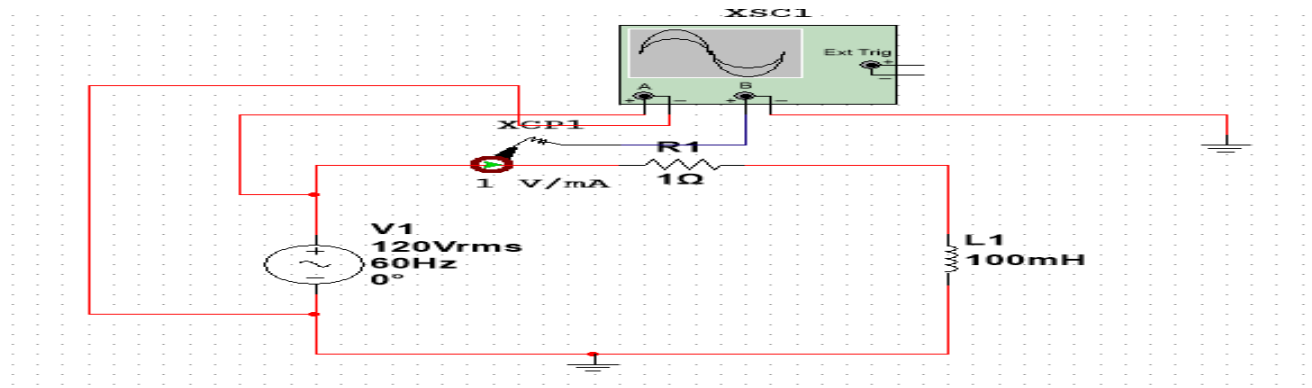


Figure 16.5: RL circuit diagram.

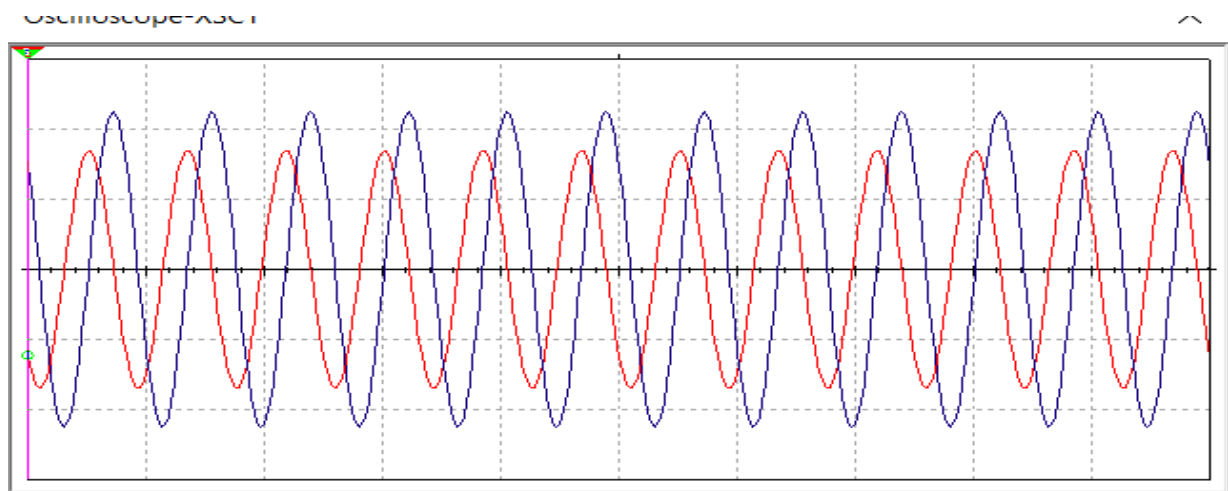


Figure 16.6: RL circuit output.

Conclusion: If we analyze the output of among the three circuits then we can see that the circuit having resistive element is having the same phase shift with the input voltage. Then again in the RC circuit current is leading voltage is lagging on the other hand in the RL circuit voltage is leading and current is lagging.

Experiment No: 17

Experiment Name: Simulation of RLC circuit in NI Multisim.

Objective:

1. The main purpose of this simulation is to understand the electrical resonance of electrical circuit.
2. To understand the significance of resonance frequency.

Components requires for simulation:

1. AC power Supply
2. Resistor.
3. Inductor.
4. Capacitor.
5. Multimeter.

Experimental Setup:

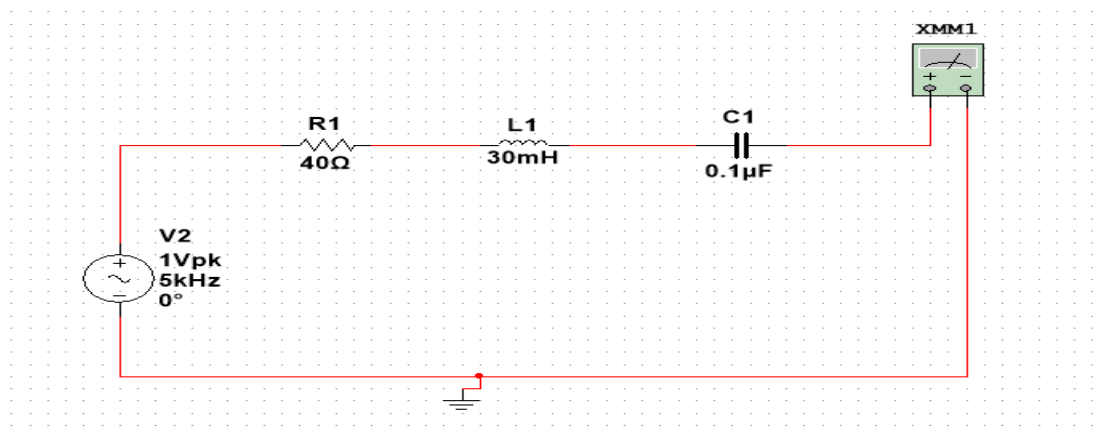


Figure 17.1: Series RLC circuit diagram.

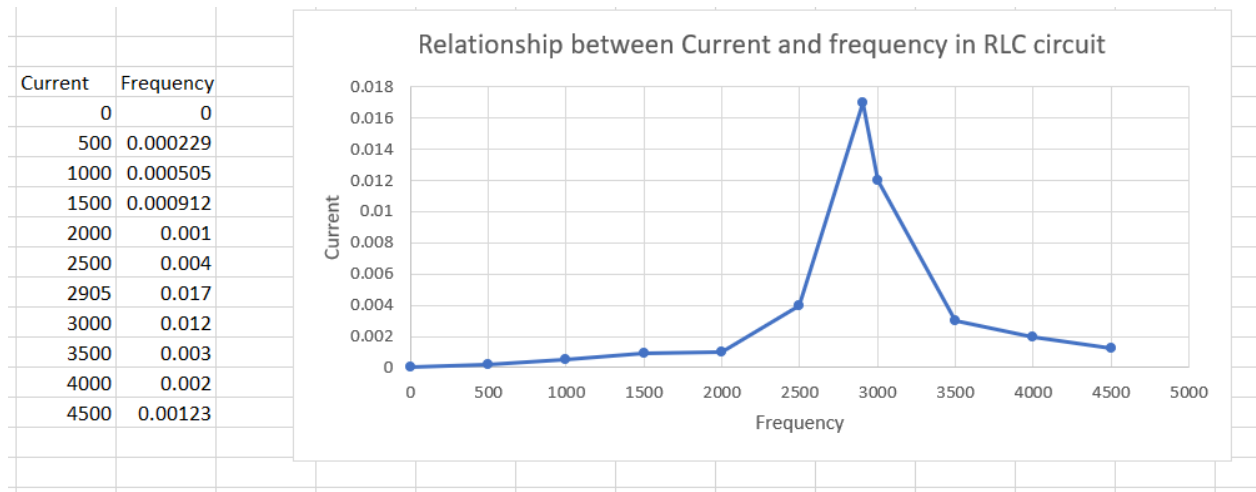


Figure 17.2: series RLC circuit resonance.

Conclusion: We can see the relation between current and frequency. The current increases up to resonant frequency which is here 2905 Hz. Then again fall down. This phenomenon is known as resonance.

Experiment No: 18

Experiment Name: Simulation of Voltage Doubler in NI Multisim.

Objective:

1. How to create an Voltage doubler circuit.
2. To verification of doubler circuit.

Component requires for simulation:

1. AC power source.
2. Transformer.
3. Diode (1N4007)
4. Capacitor.
5. Oscilloscope.

Experiment Setup:

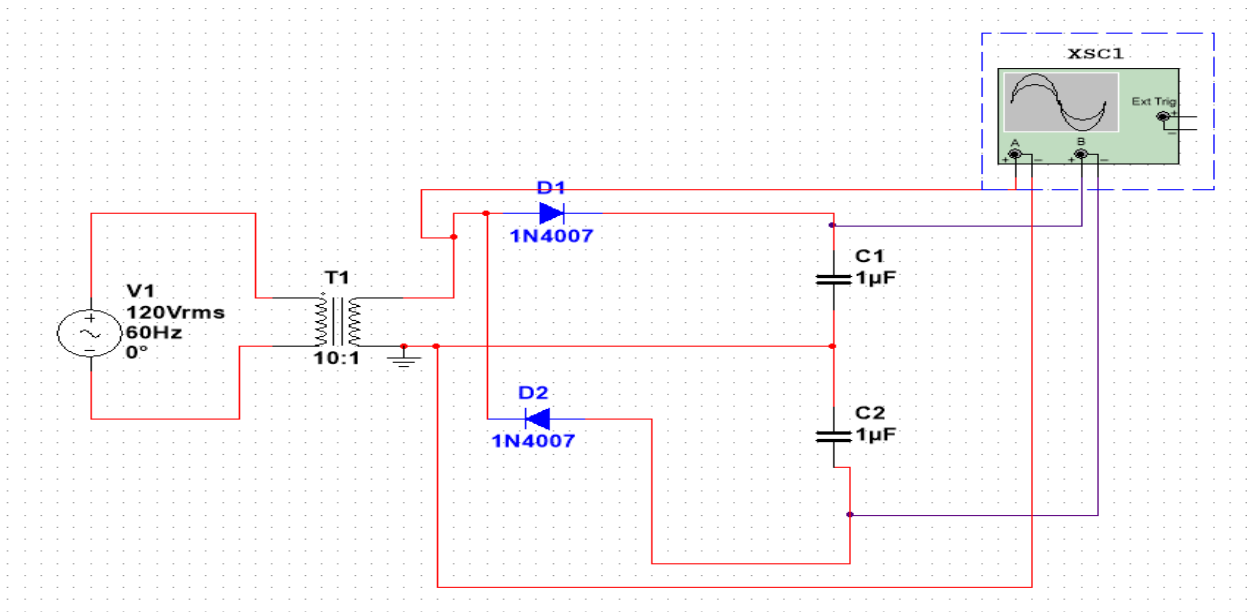


Figure 18.1: Circuit of Voltage doubler.

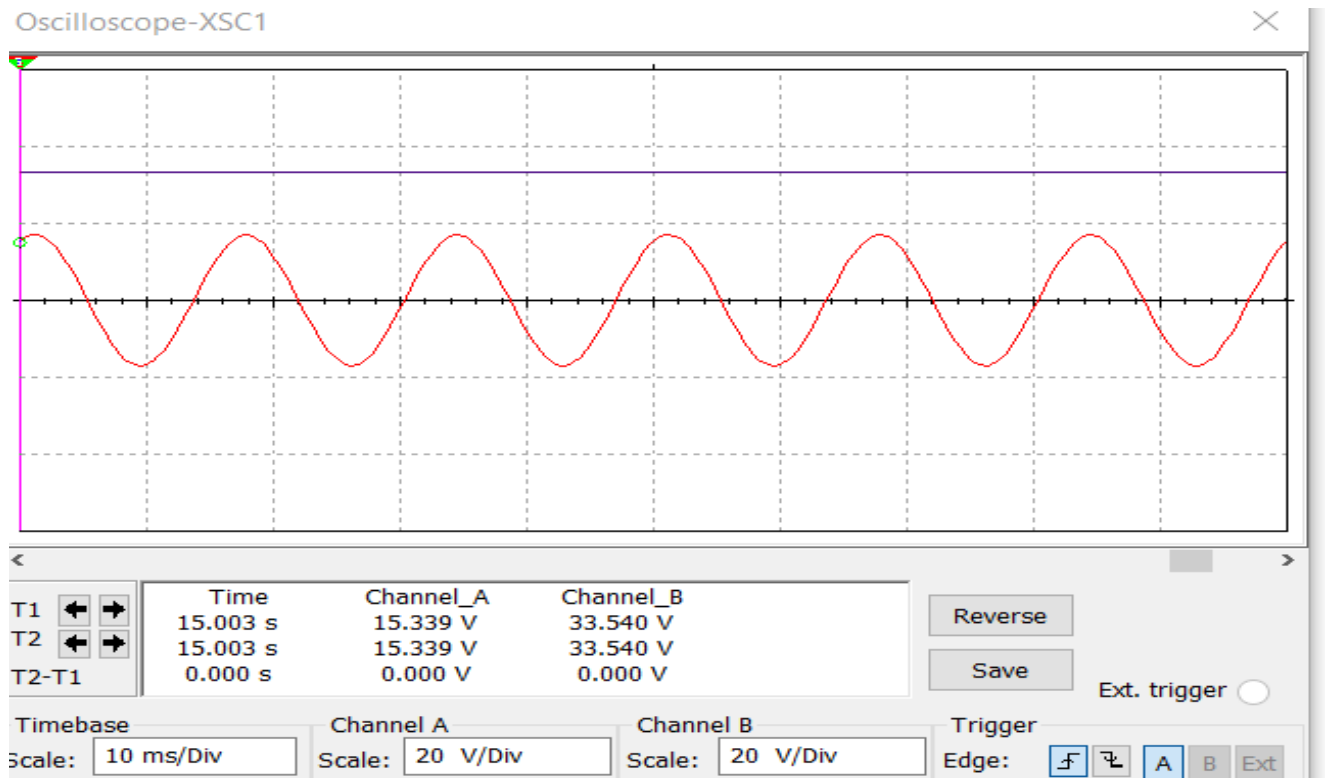


Figure 18.2: Output of Voltage Doubler.

Conclusion: A voltage doubler circuit is an electronic circuit which charges capacitors from the input voltage and switches these charges in such a way that in the ideal case exactly twice the voltage is produced at the output as its input.

Experiment No: 19

Experiment Name: Simulation of Voltage multiplier circuit NI Multisim.

Objective:

1. To create a higher voltage than a lower power source.
2. To convert an AC electrical power from lower voltage to a higher DC voltage.

Component requires for simulation:

1. AC power source.
2. Transformer.
3. Diode (1N4007)
4. Capacitor.
5. Oscilloscope

Experimental Setup:

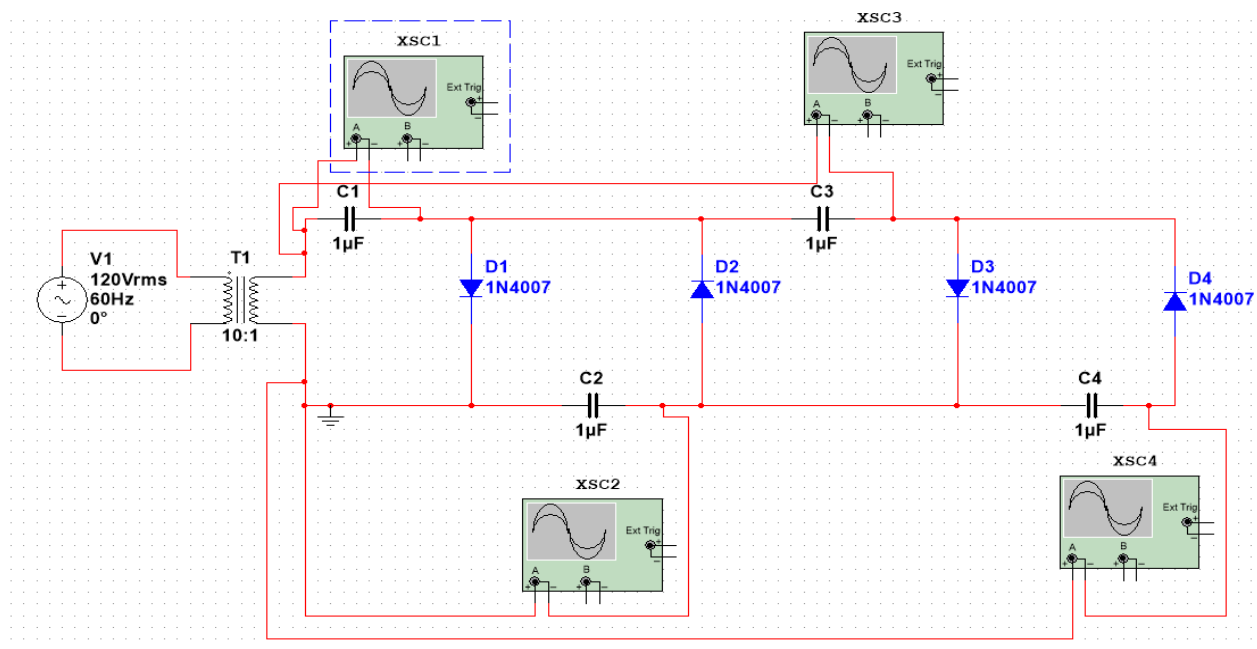


Figure 19.1: Voltage Multiplier Circuit.

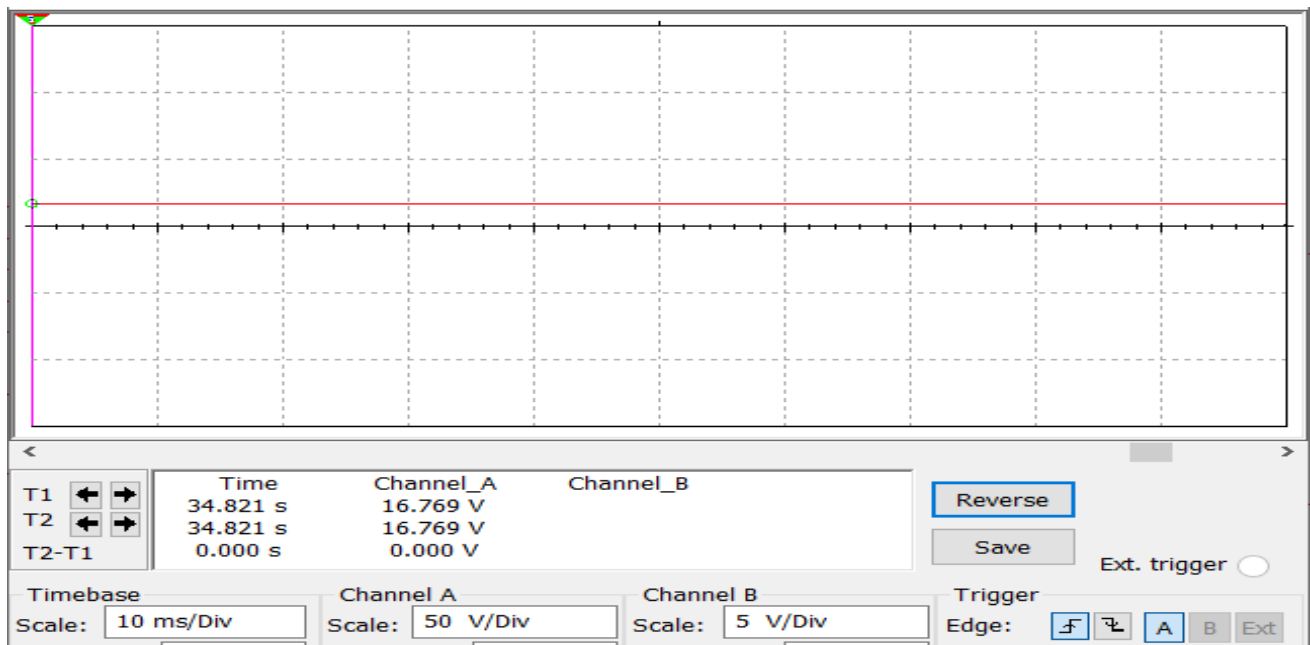


Figure 19.2: Output of First Capacitor 1(C1)

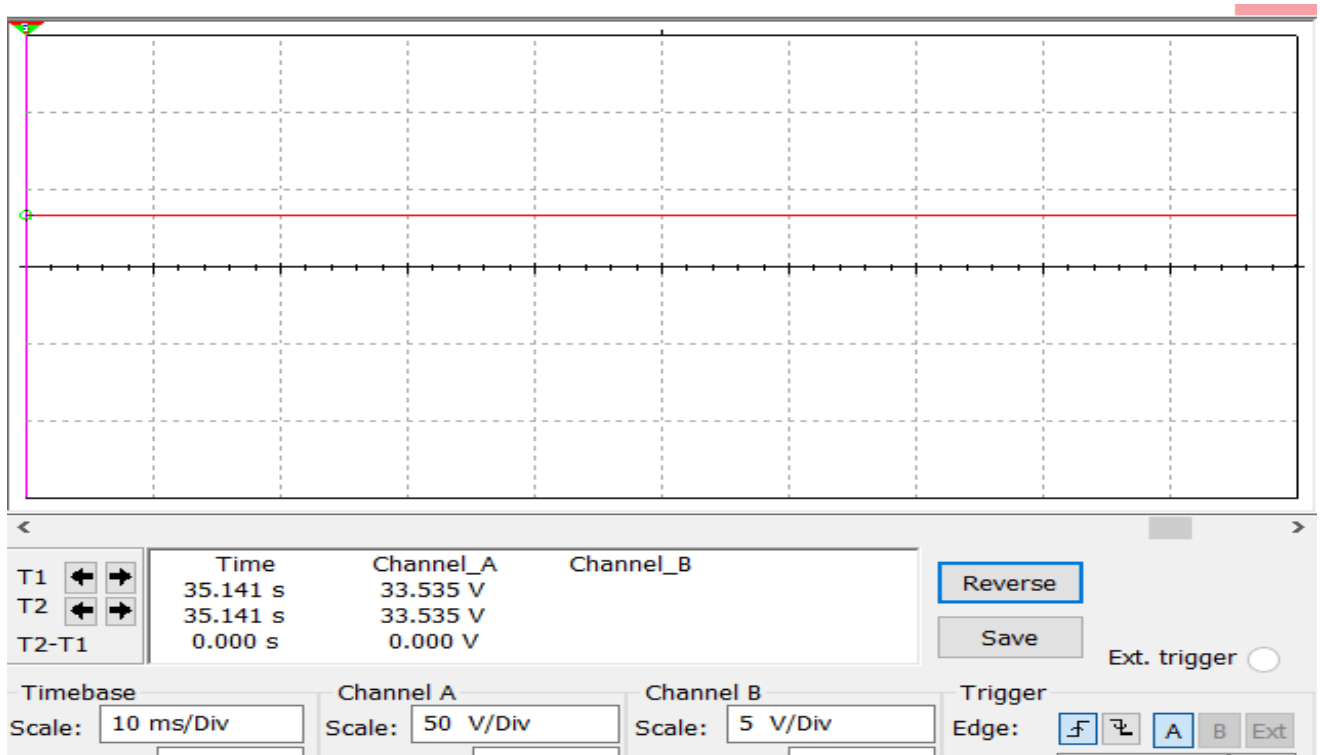


Figure 19.3: Output of Second capacitor (C2) which acts as a voltage doubler.

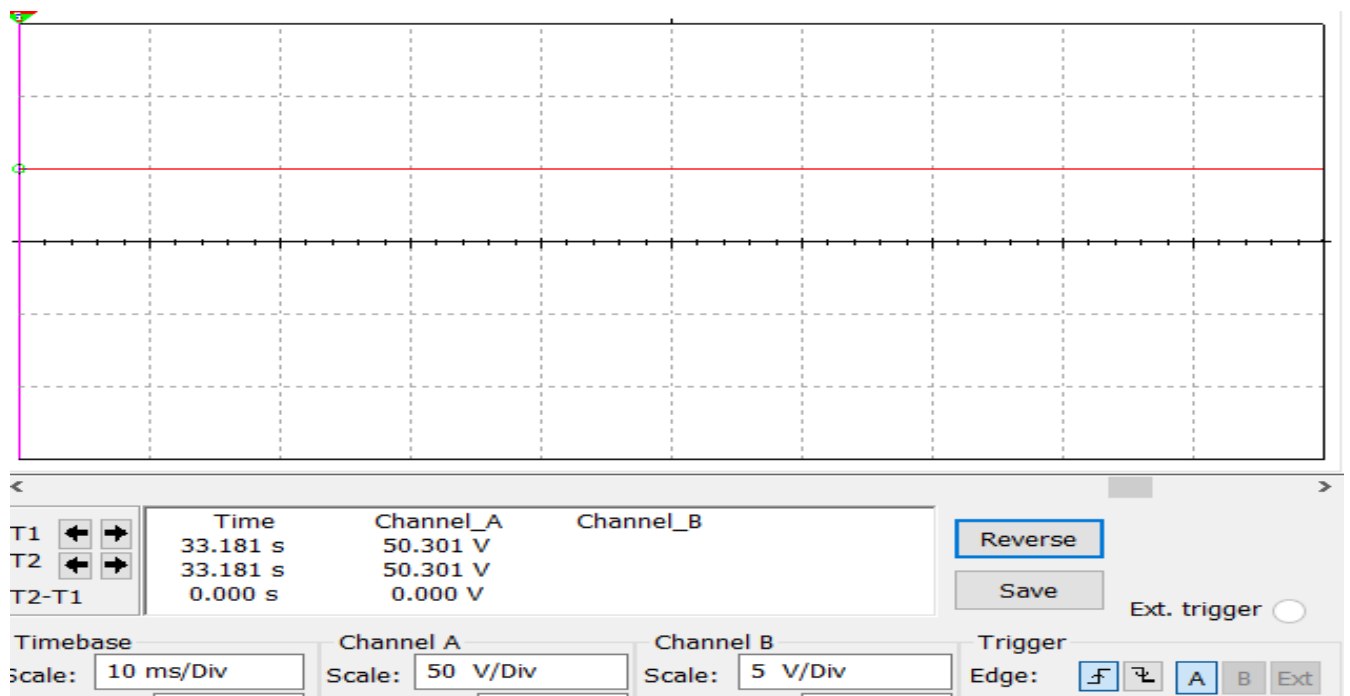


Figure 19.4: Output of third capacitor (C3) which acts as a voltage tripler.

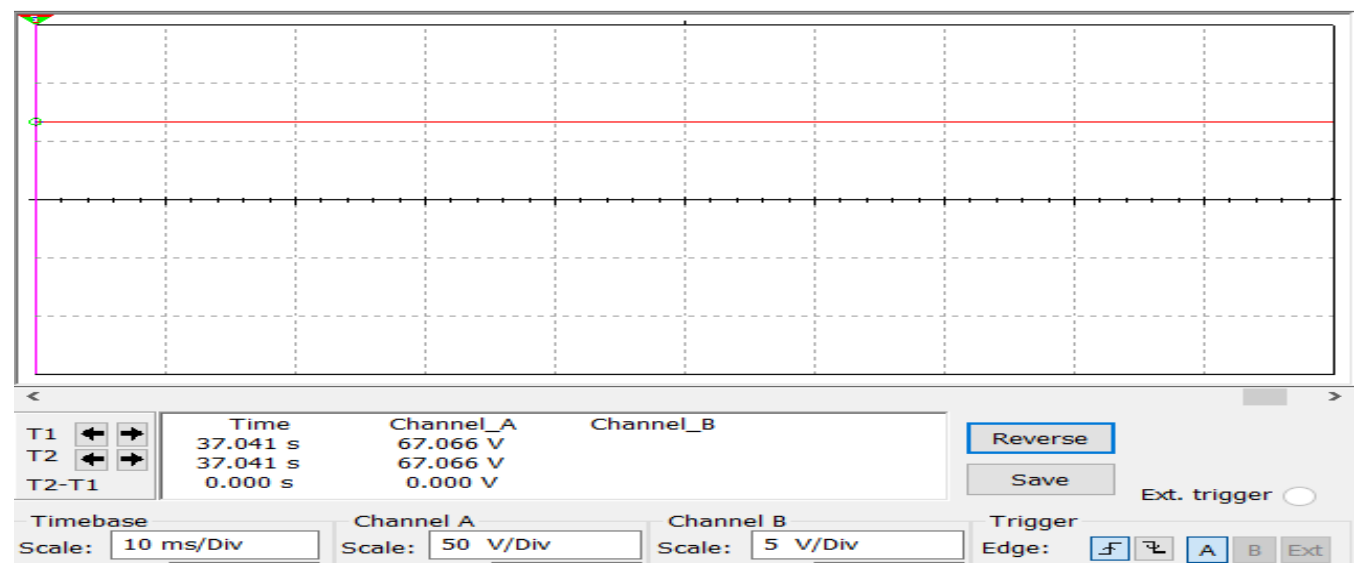


Figure 19.5: Output of third capacitor (C4) which acts as a voltage quadripolar.

Conclusion: A voltage multiplier circuit is an electronic circuit which charges capacitors from the input voltage and switches these charges in order of the capacitor to multiply the input voltage into twice, thrice and fourth time than the input voltage.

Experiment No: 20

Name of the experiment: Verification of Superposition theorem in NI Multisim.

Objective:

1. To apply the superposition theorem to linear circuit with two or more than one voltage source.
2. To construct a circuit with two voltage sources, solve for the currents and voltage throughout the circuit, and verify our computation by simulation.

Components requires for the simulation:

1. AC voltage source.
2. Resistor.
3. Connecting wire.
4. Multimeter.

Experimental Setup:

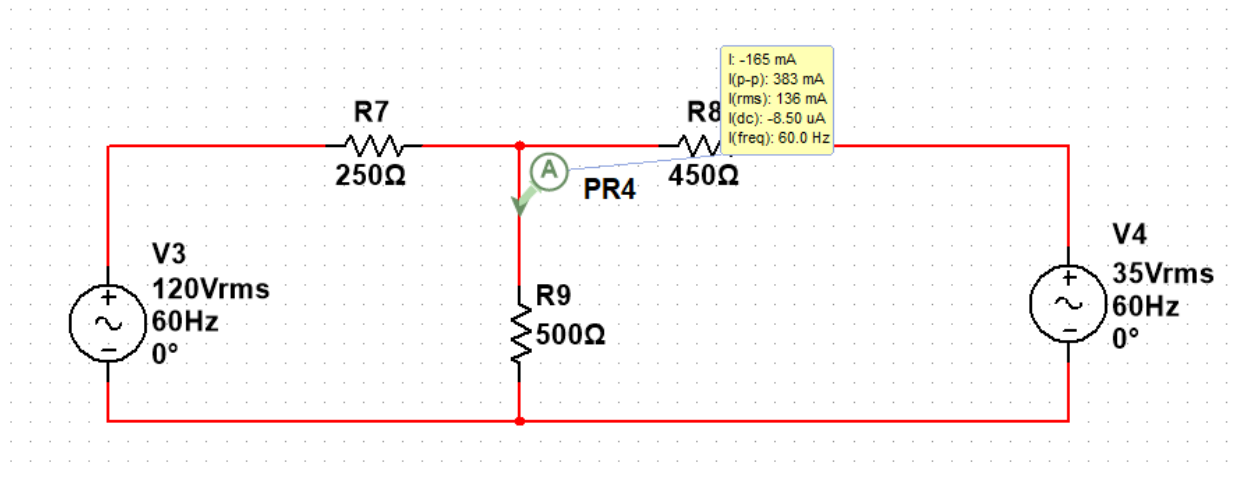


Figure 20.1: Superposition circuit with two source.

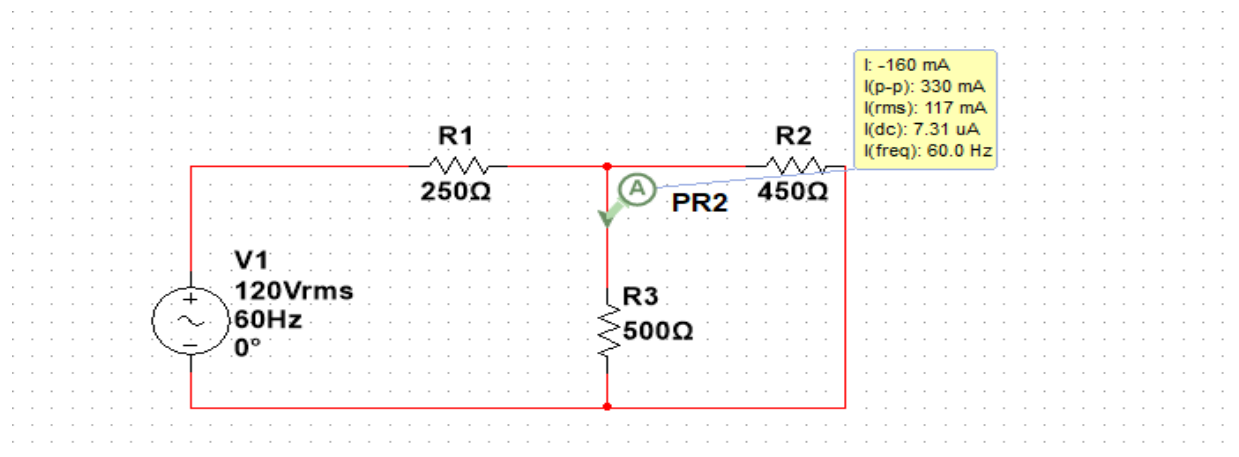


Figure 20.2: Superposition theorem with only one source.

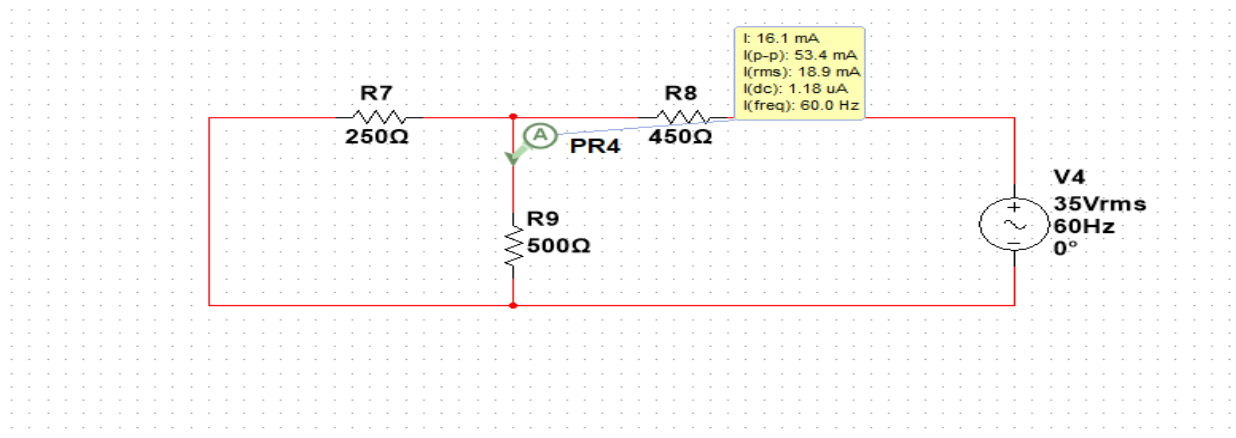


Figure 20.3: Superposition theorem with another source.

Conclusion: Superposition theorem uses to find the currents through keep of a single source in the circuit and then find a total current. Then remove source and put next source and so on.

Experiment No: 21

Experiment Name: Verification of Thevenin theorem in NI Multisim.

Objectives:

1. To determine the Thevenin equivalent voltage (V_{th}) and resistance (R_{th}) of a circuit.
2. To verify the Thevenin theorem.

Components requires for the simulation:

1. AC voltage source.
2. Resistor.
3. Connecting wire.
4. Multimeter.

Experimental Setup:

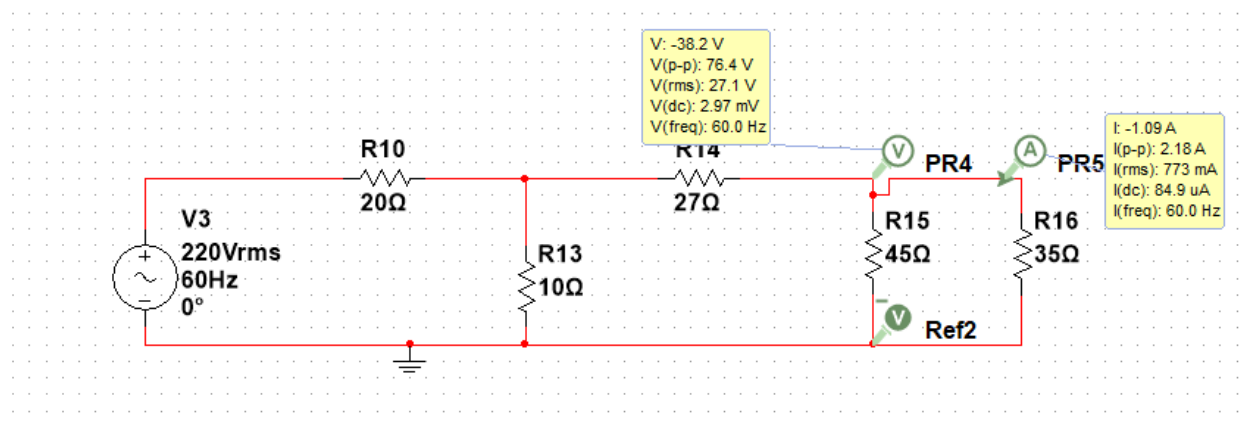


Figure 21.1: Thevenin theorem circuit diagram.

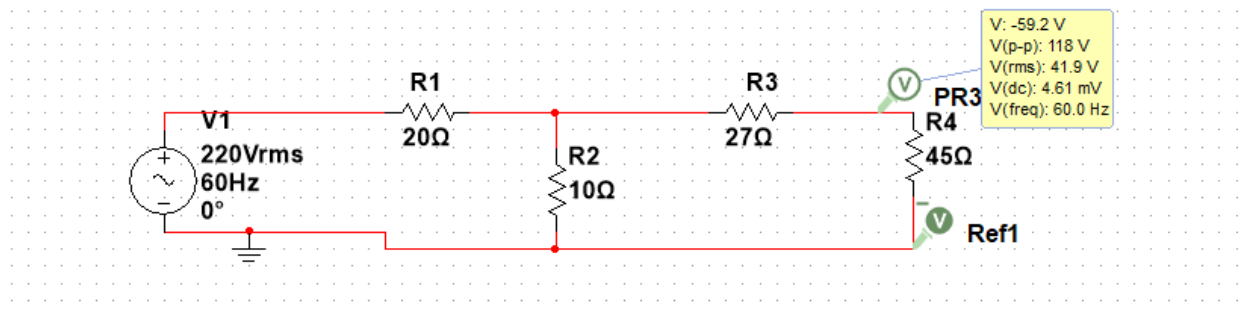


Figure 21.2: Measuring Thevenin Voltage (V_{th})

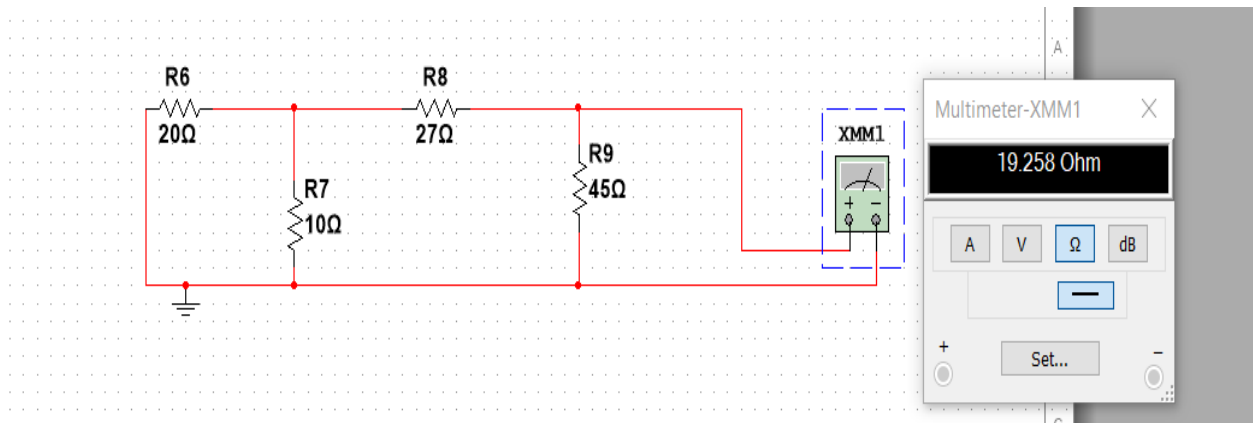


Figure 21.3: Measuring Thevenin resistance.

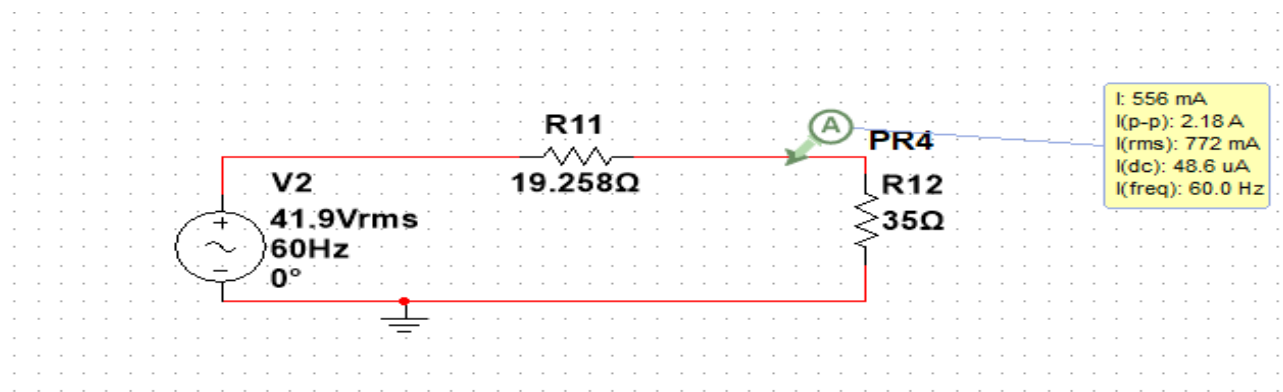


Figure 21.4: Thevenin converted circuit.

Conclusion: Thevenin theorem states that it is possible to simplify any linear circuit, no matter how complex, to an equivalent circuit with just a single voltage source and series resistor connected to a load. Throughout this experiment, the resistance are simplified into only one that is R_{th} .

Experiment No: 22

Experiment Name: Verification of Norton theorem by using simulation in NI Multisim.

Objective:

1. To verify Norton theorem.
2. To solve a complex circuit through Norton theorem.
3. To measure different parameter of any complicated circuit through Norton theorem.

Components requires for the simulation:

1. AC voltage source.
2. Resistor.
3. Connecting wire.
4. Multimeter.

Experimental Setup:

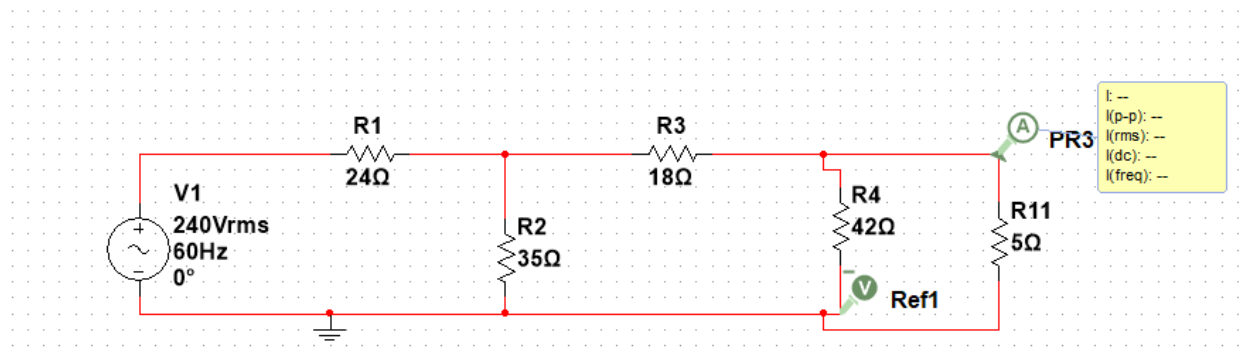


Figure 22.1: Norton theorem experiment circuit.

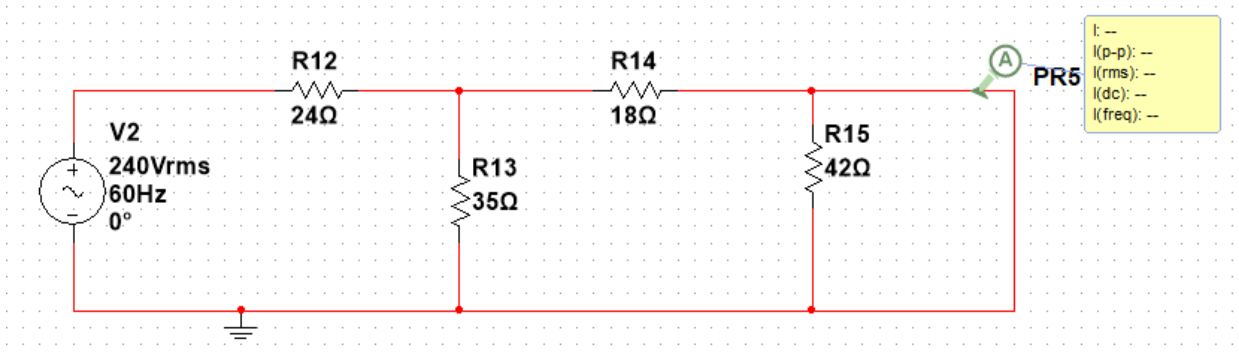


Figure 22.2: Norton current calculation.

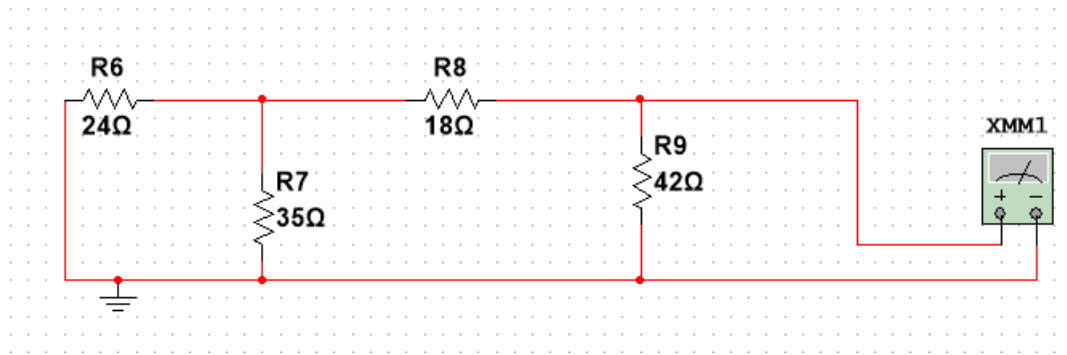


Figure 22.3: Norton equivalent resistant.

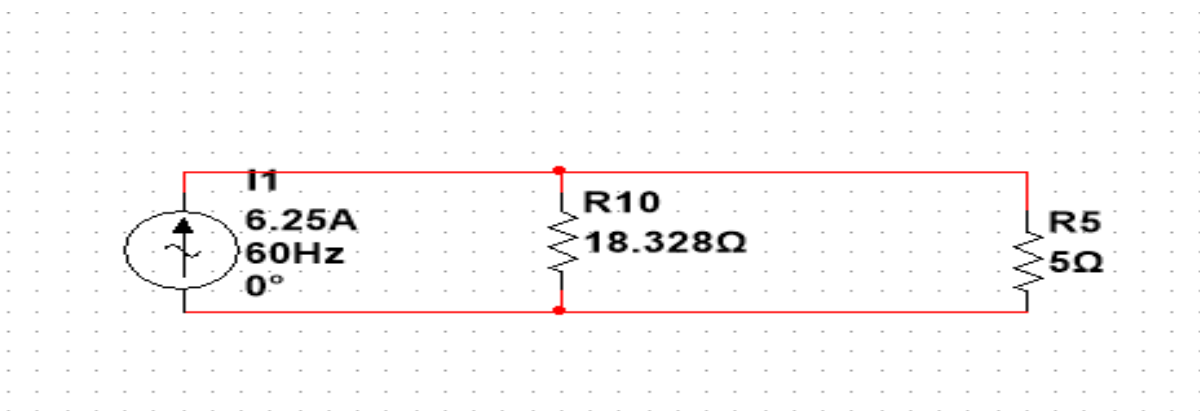


Figure 22.4: Norton equivalent circuit.

Conclusion: The objective of this experiment is to study Norton theorem and the application of circuit analysis. Norton theorem is identical to Thevenin theorem except the equivalent circuit is a source transformation of Thevenin equivalent circuit.

Experiment No: 23

Name of the experiment: Simulation of Operational Amplifier as comparator.

Objective:

1. To construct a circuit with operational amplifier.
2. To see the variation of output with changing the input values in inverting and non-inverting terminal.

Component requires for simulation:

1. DC voltage.
2. Operational Amplifier. (741).
3. Red LED.
4. Four channel oscilloscopes.
5. VCC
6. VEE
7. Resistor
8. Connecting wire.

Experimental Setup:

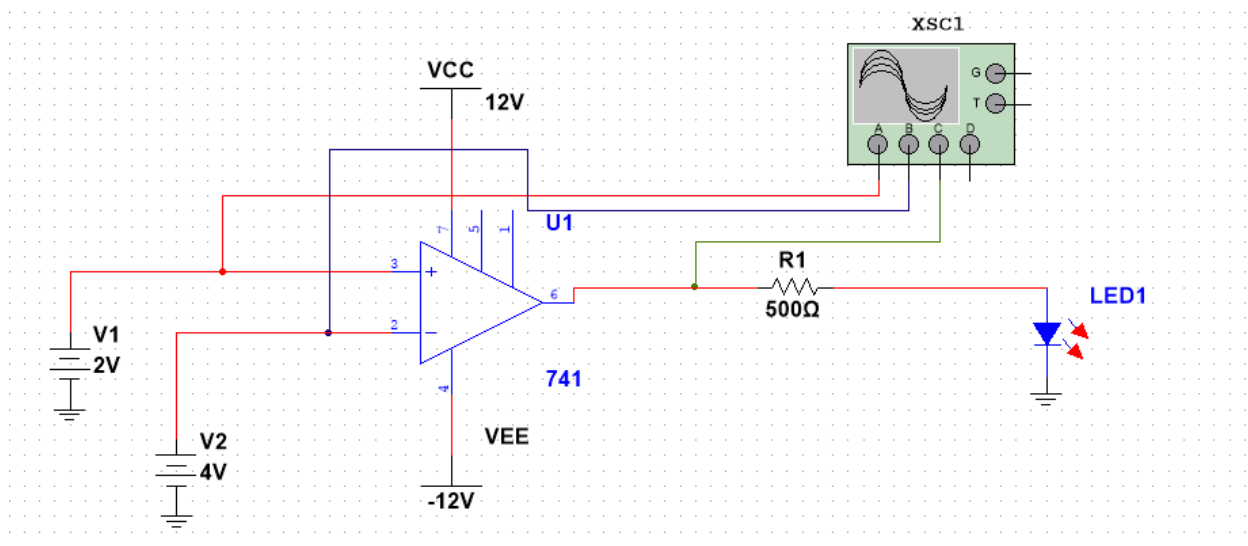


Figure 23.1: Circuit diagram when negative terminal has greater potential.

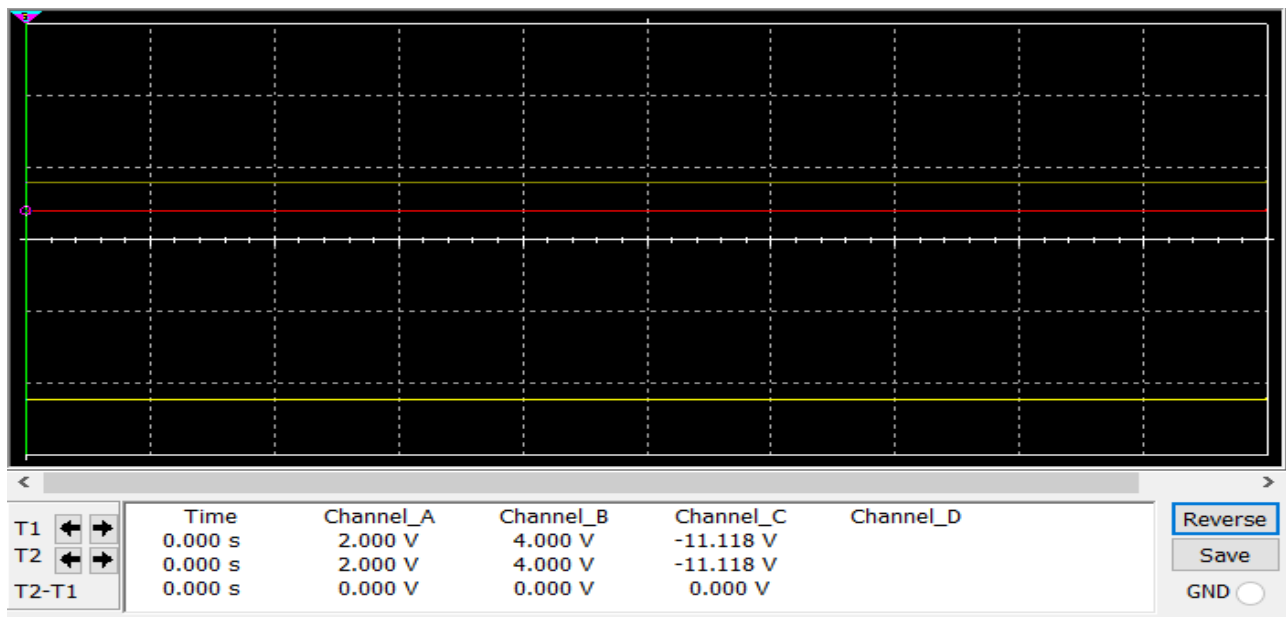


Figure 23.2: Output when negative terminal has greater potential.

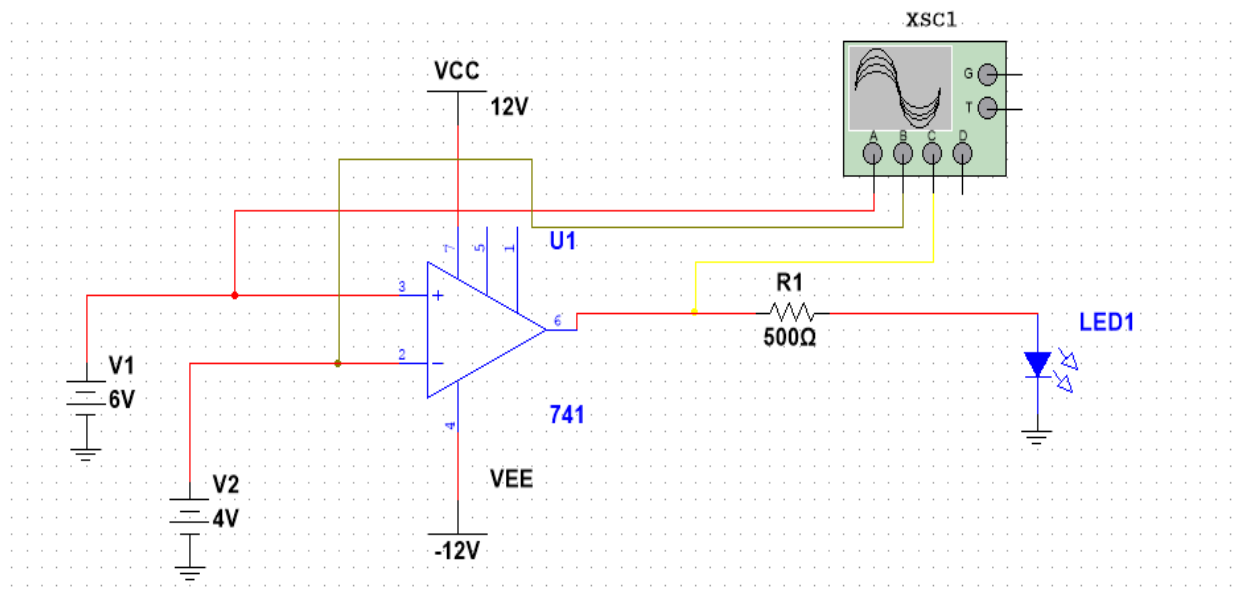


Figure 23.3: Circuit diagram when positive terminal has greater potential.

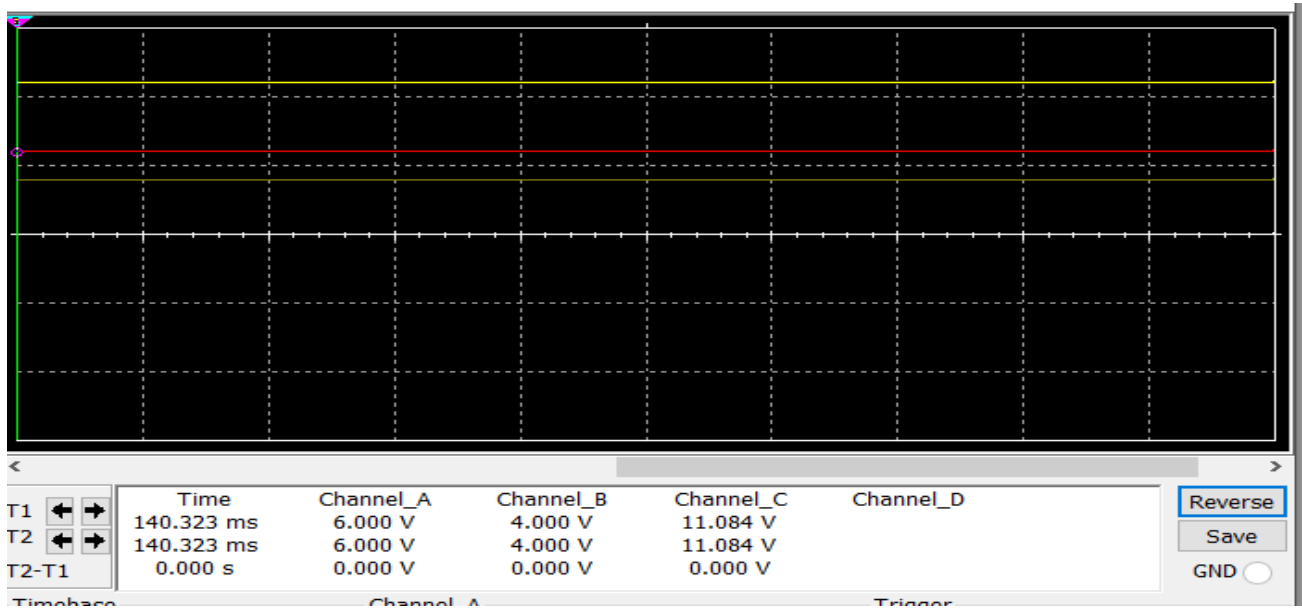


Figure 23.3: Output when positive terminal has greater potential.

Conclusion: When input is given in inverting and non-inverting terminal both , then the output depends on the difference between both terminals. In first case when negative terminal input was greater than positive terminal then the output was negatively amplified. But it was increased in such a way that it does not cross the limit of maximum biasing voltage which is V_{EE} . Same thing goes for the maximum positive terminal voltage.

Experiment No: 24

Experiment Name: Simulation of Converting 230V RMS voltage into regulated 5 V DC voltage.

Objective:

1. To create a AC to DC converting circuit.
2. To use an IC in Multisim.
3. To get to know how an IC works in an circuit.

Component requires for Simulation:

1. AC source.
2. Transformer
3. Diode
4. LM7805CT IC
5. Capacitor
6. Oscilloscope
7. Multimeter
8. Connecting wire.

Experimental Setup:

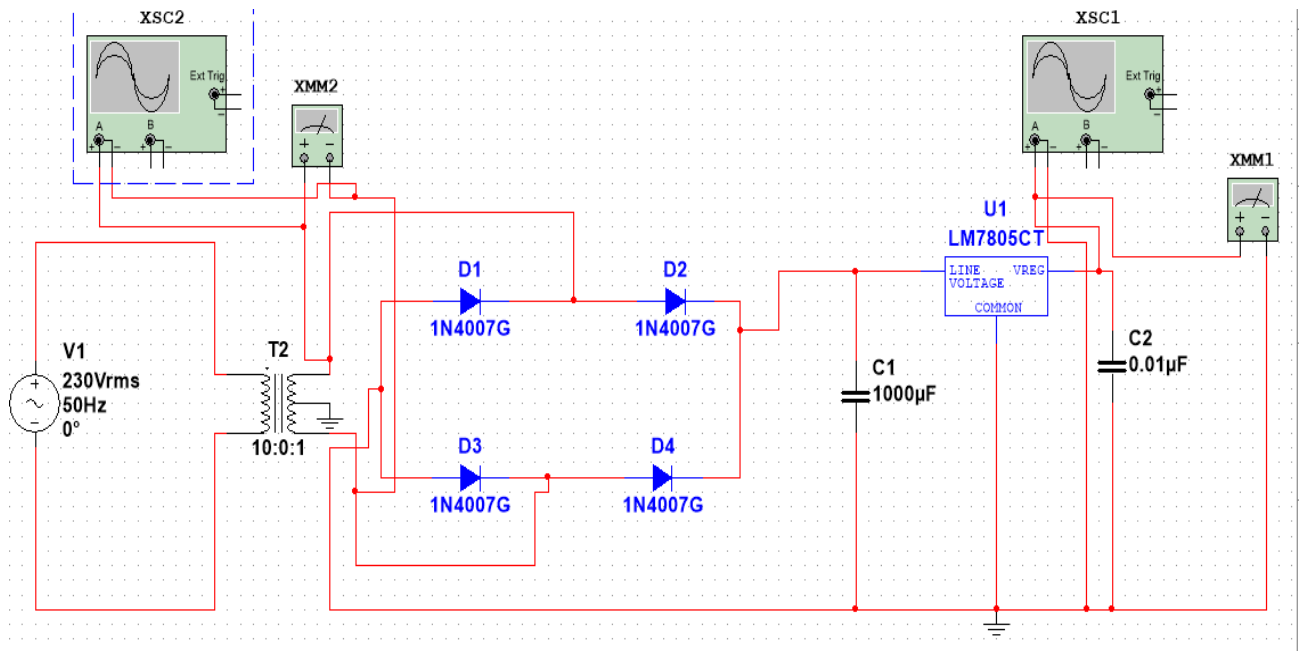


Figure 24.1: 230V RMS AC to DC 5V regulated circuit

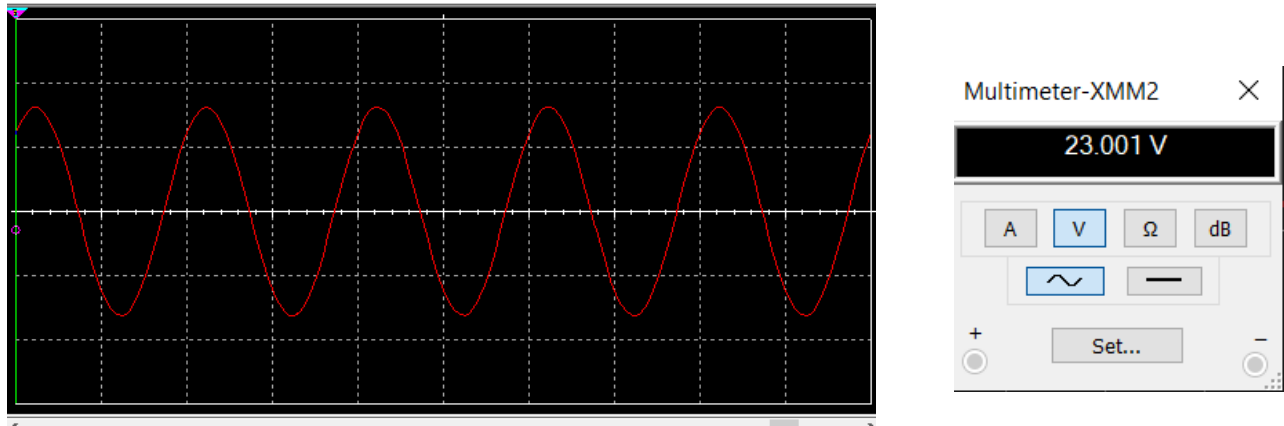


Figure 24.2: Input circuit voltage.

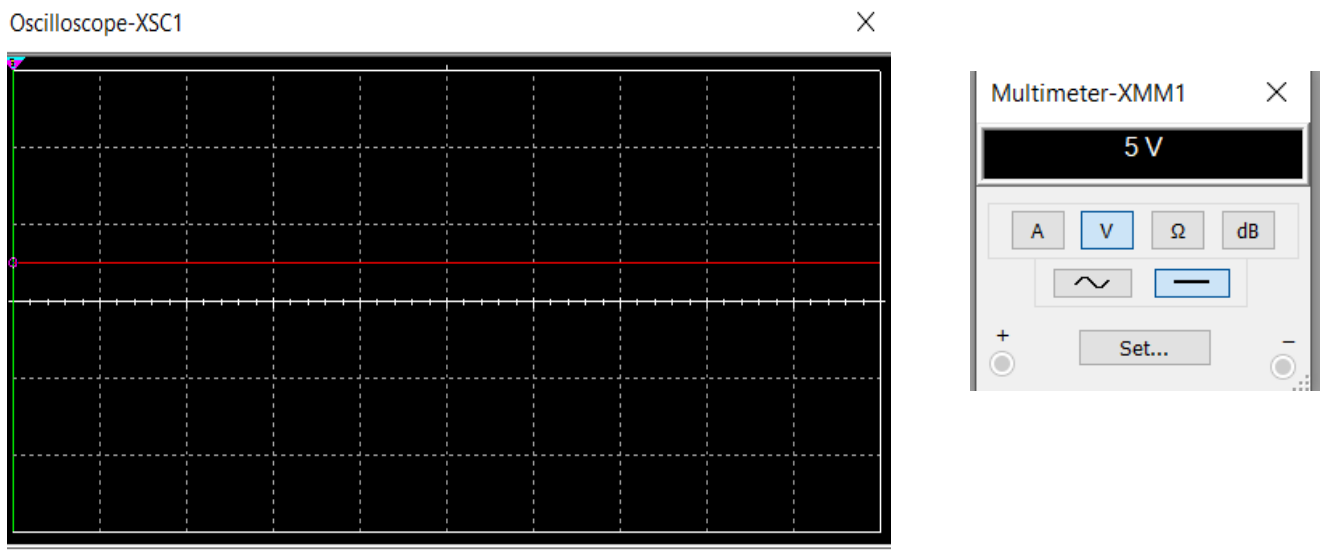


Figure 24.3: Output of regulated circuit.

Conclusion: To use electronic component we need to convert our input AC voltage into DC voltage. Mobile charger is one of the most common example which uses regulated DC circuit. Here IC 7805CT keeps the voltage inside 5.2 volt.

Experiment No: 25

Experiment Name: Simulation of three phase power supply NI Multisim.

Objective:

1. To prepare a three-phase power supply.
2. To verify the phase shift of the output.
3. To make a star connection of load.

Component require for simulation:

1. AC power supply.
2. Resistor.
3. Connecting wire.
4. Oscilloscope.

Experiment setup:

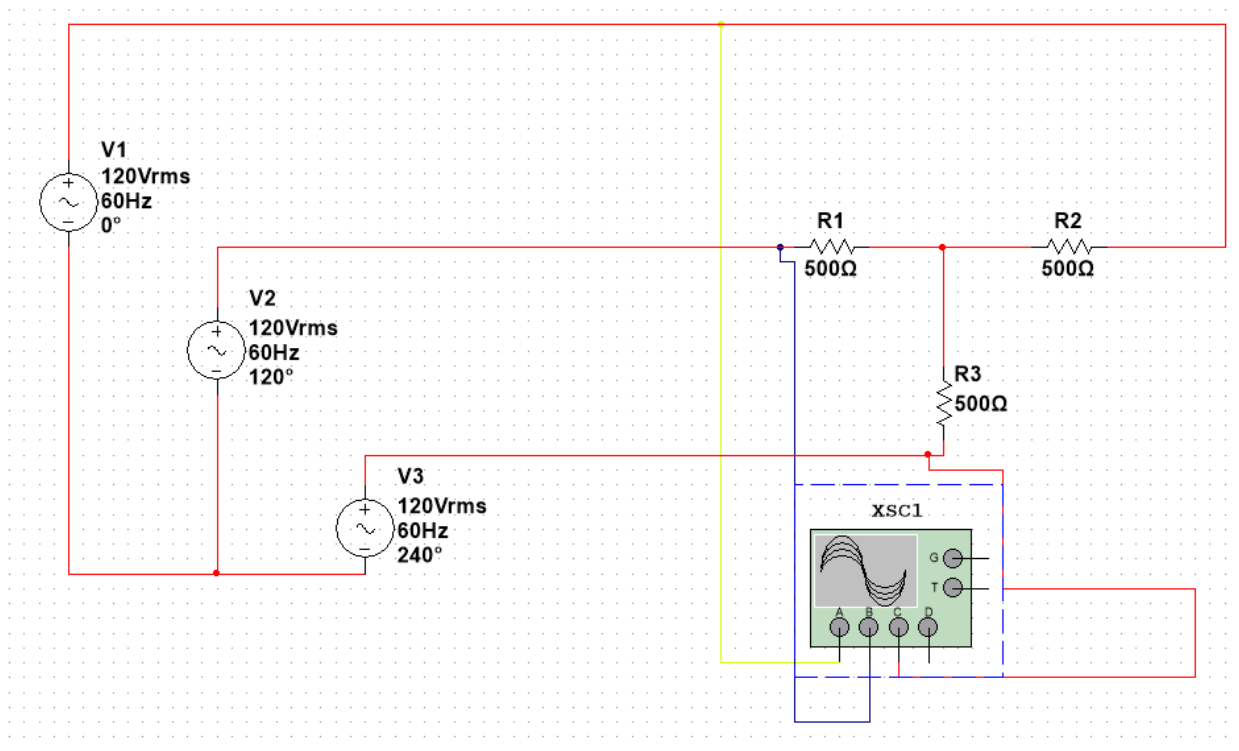


Figure 25.1: Circuit diagram of three phase power supply.

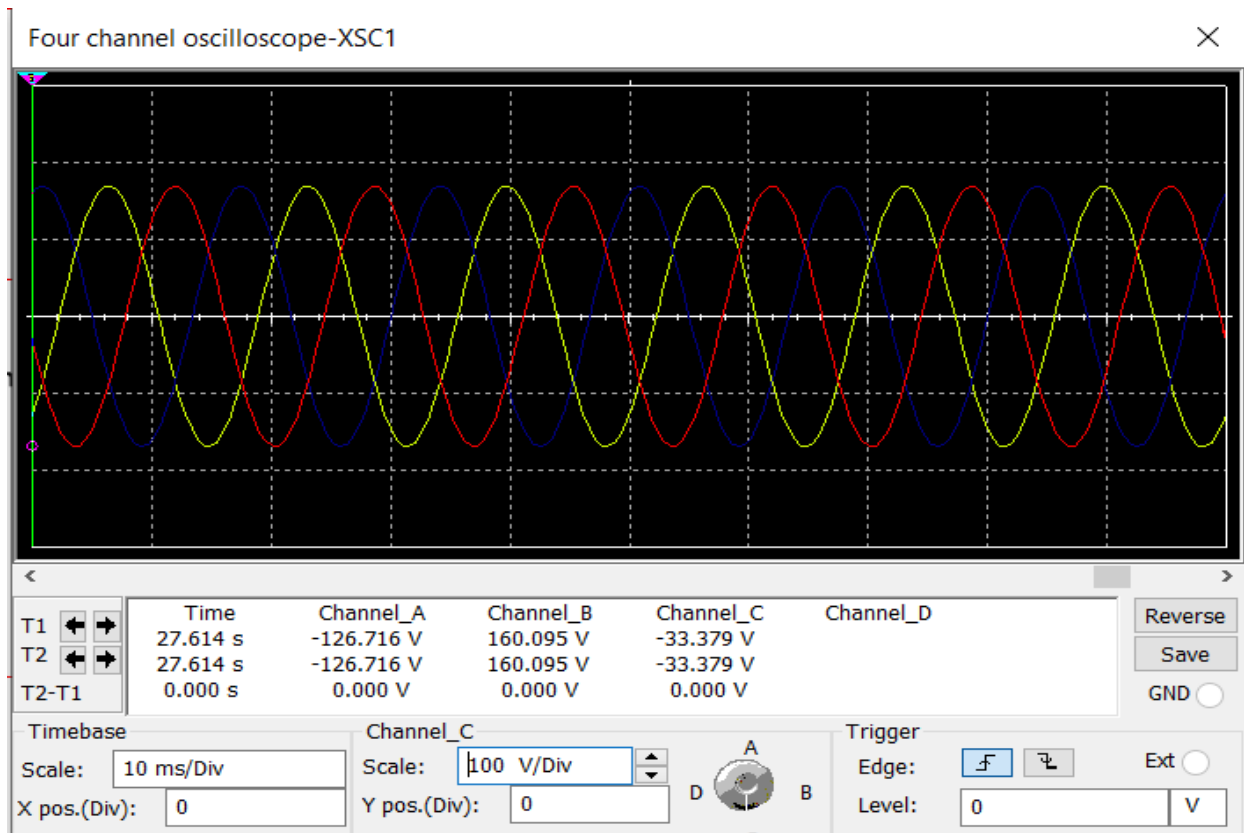


Figure 25.2: Output phase shift of three phase power supply.

Conclusion: We need three phase power supply in daily basis of our requirement. Three phase power supply is made such a way that is phase difference is 120 degree.

Experiment No: 26

Experiment Name: Simulation of summing amplifier using operational amplifier.

Objective:

1. Construct a summing amplifier circuit.
2. To understand the working principle of summing amplifier.
3. To verify the output of summing amplifier.

Component require for simulation:

1. Resistor.
2. AC power source.
3. Operational Amplifier.
4. Connecting wire.
5. Four Channel oscilloscope.
6. Connecting wire.

Experimental Setup:

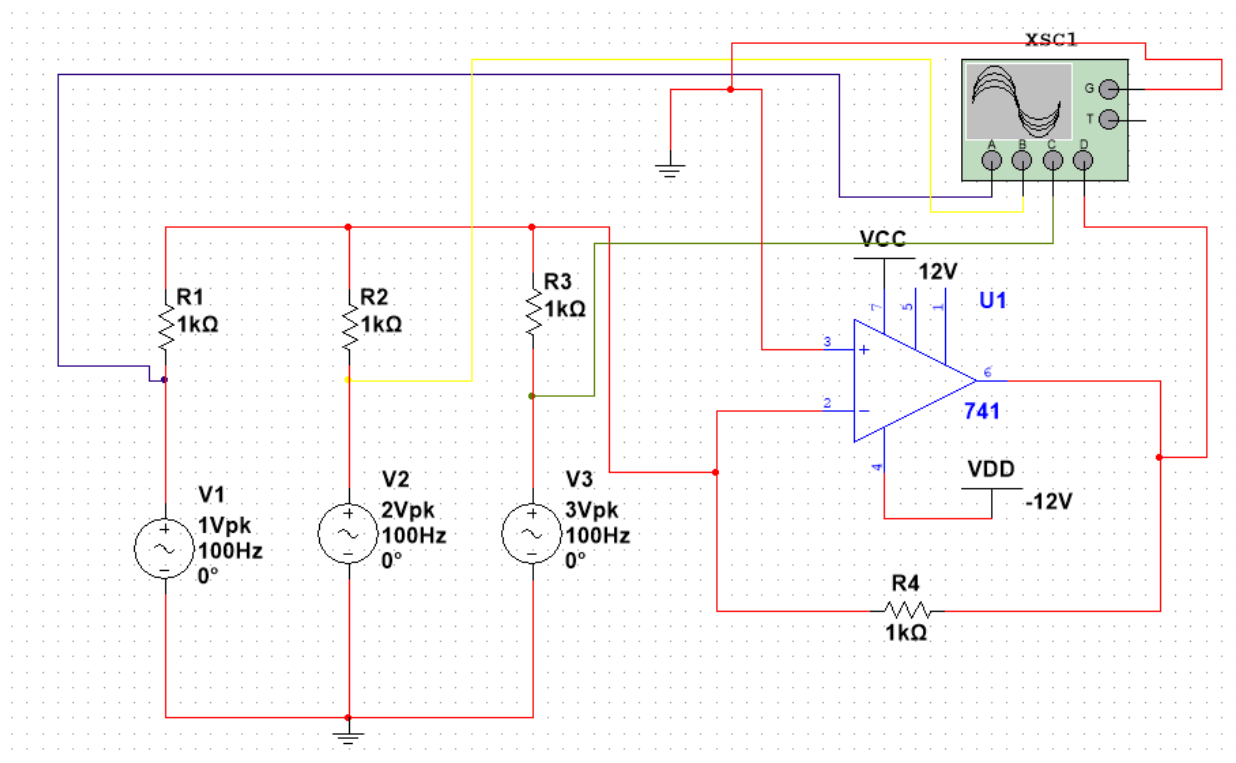


Figure 26.1: Summing operational Amplifier circuit.

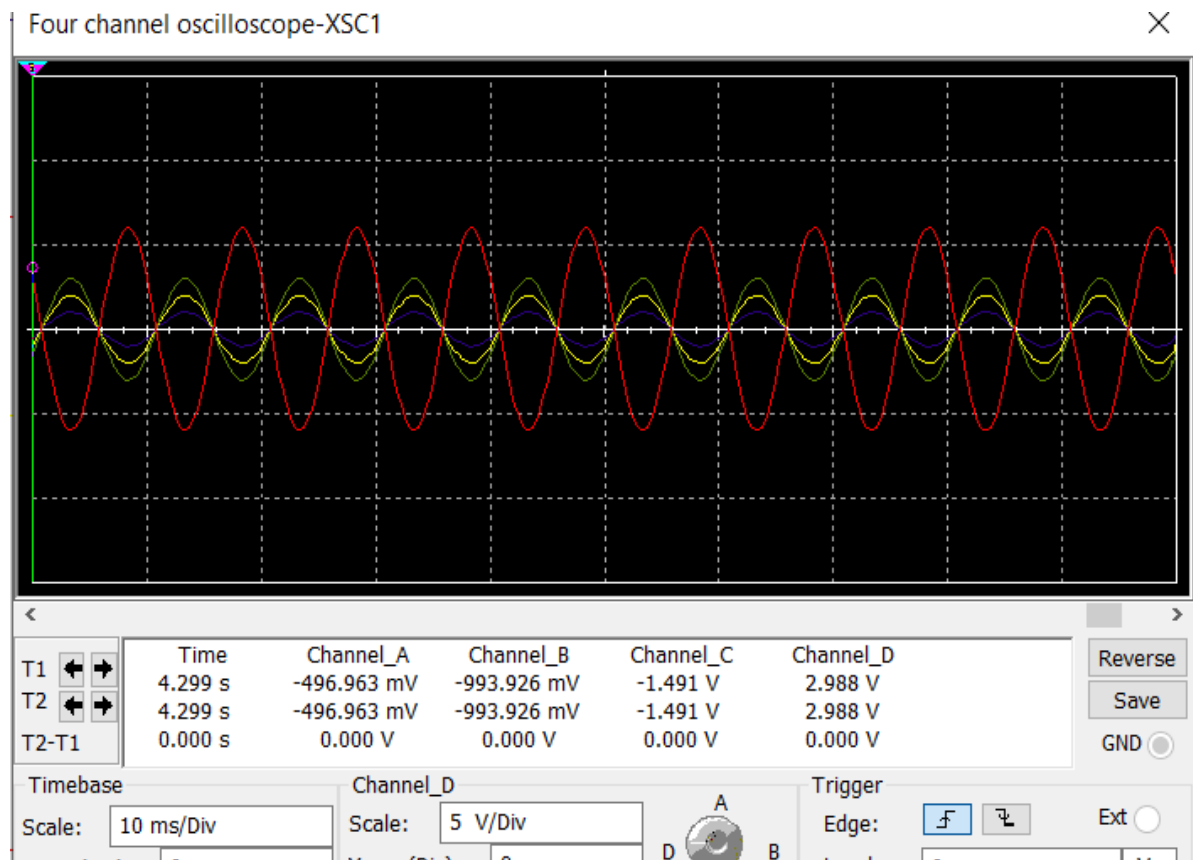


Figure 26.2: Summing amplifier output.

Conclusion: Here, the red wave shape is the output of the amplifier. It's the summation of the input signal. Summing amplifier is a circuit which adds all the signals in the input and gives the resultant output as a summation of the input signal. It's basically used to process analog signals.

Experiment No: 27

Experiment Name: Simulation of Differential amplifier in NI Multisim.

Objective:

1. To construct a differential amplifier circuit in NI Multisim.
2. To understand the working principle of differential amplifier.
3. To verify the waveshape of the differential amplifier.

Component require for simulation:

1. Resistor.
2. AC power source.
3. Operational Amplifier.
4. Connecting wire.
5. Four Channel oscilloscope.
6. Connecting wire.

Experimental Setup:

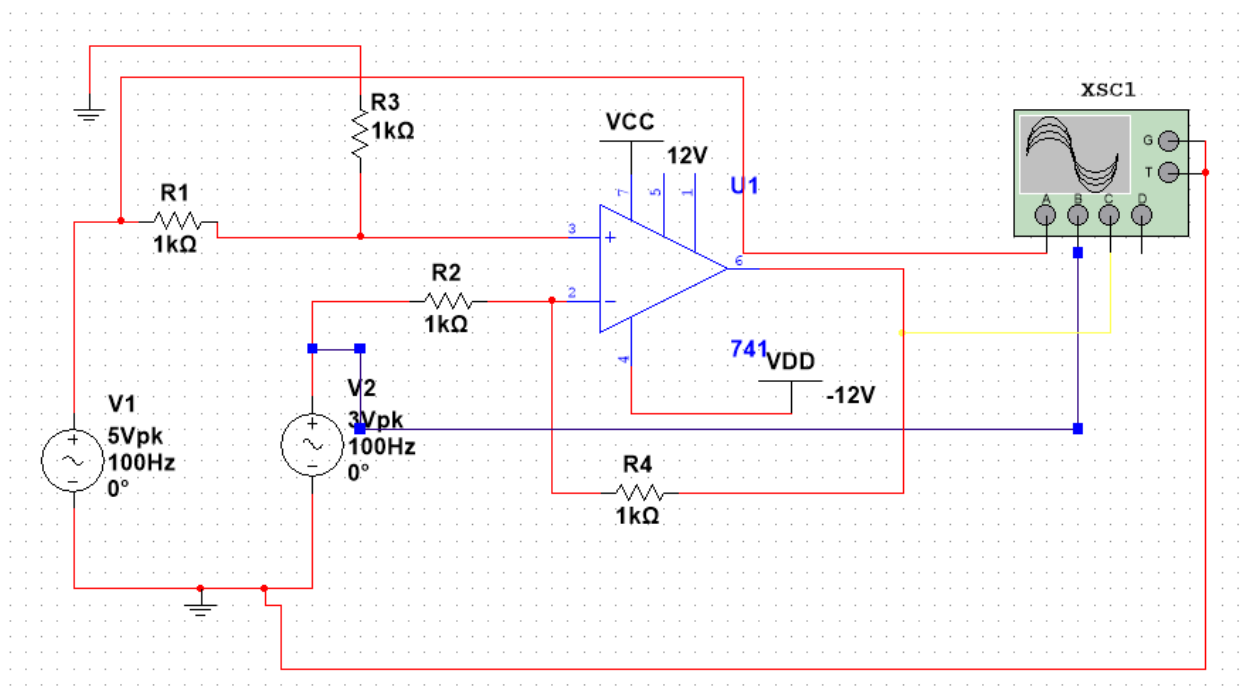


Figure 27.1: Circuit diagram of Differential amplifier.

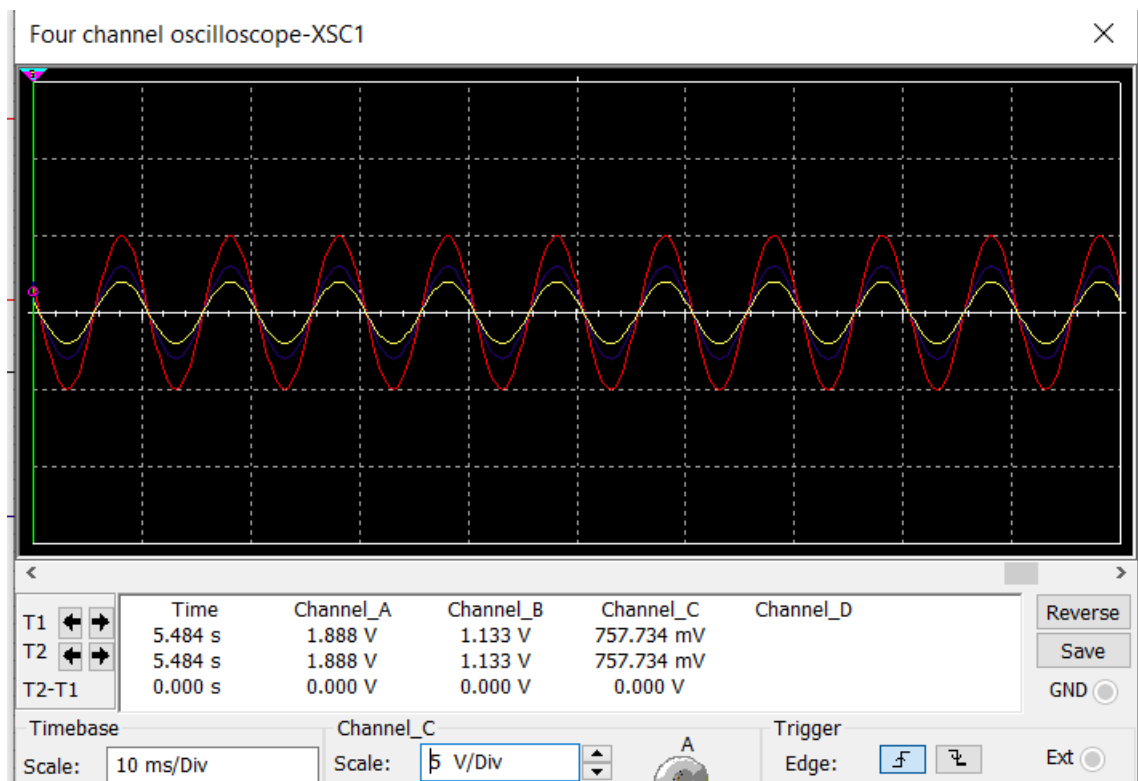


Figure 27.2: Output of differential amplifier.

Conclusion: Here, the yellow line in the oscilloscope is the required output. We can see that is the difference of the two-input signal. It is used to suppress noise. We can remove the noise from the input to get required output.

Experiment No: 28

Experiment Name: Simulation of Operational amplifier as an integrator in NI Multisim.

Objective:

1. To perform the mathematical operation of integration with respect to time.
2. To construct a integrator circuit of operational amplifier.
3. To generate ramp function.
4. To verify its working principle.

Component require for simulation:

1. Resistor.
2. AC power source.
3. Operational Amplifier.
4. Connecting wire.
5. Four Channel oscilloscope.
6. Connecting wire.
7. Capacitor.

Experimental Setup:

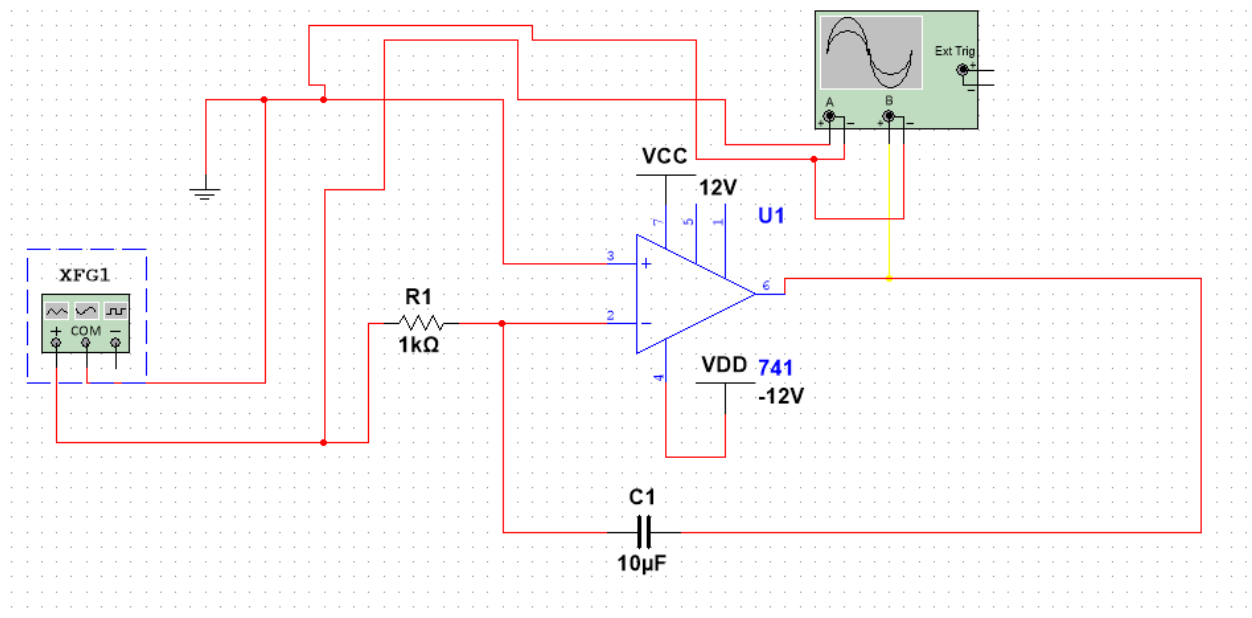


Figure 28.1: Circuit diagram of integrator of operational amplifier.

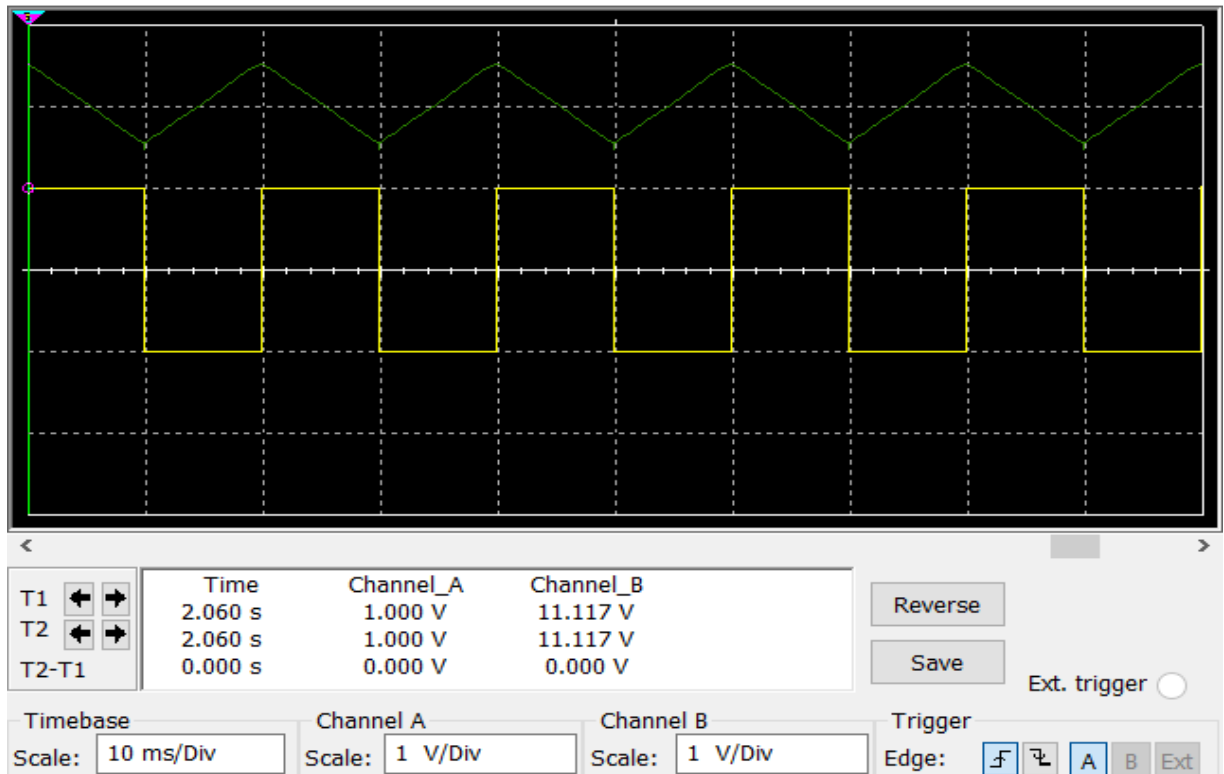


Figure 28.2: Output of integrator circuit of operational amplifier.

Conclusion: Here, the green signal showing in the oscilloscope is the output of integrator circuit. We can see that it generates ramp function in the output. We use this circuit to cause the output to respond to changes in the input voltage. We use it analog computers, analog to digital converters etc.

Experiment No: 29

Experiment Name: Simulation of Measurement of power using two wattmeter in a three phase balance wye connection.

Objective:

1. To get familiar with the connection of wattmeter in the circuit.
2. To measure the power of three phase system.
3. To measure the power factor of the three-phase system.

Component requires for simulation:

1. Wattmeter
2. Resistor
3. Three phase power supply

Experimental Setup:

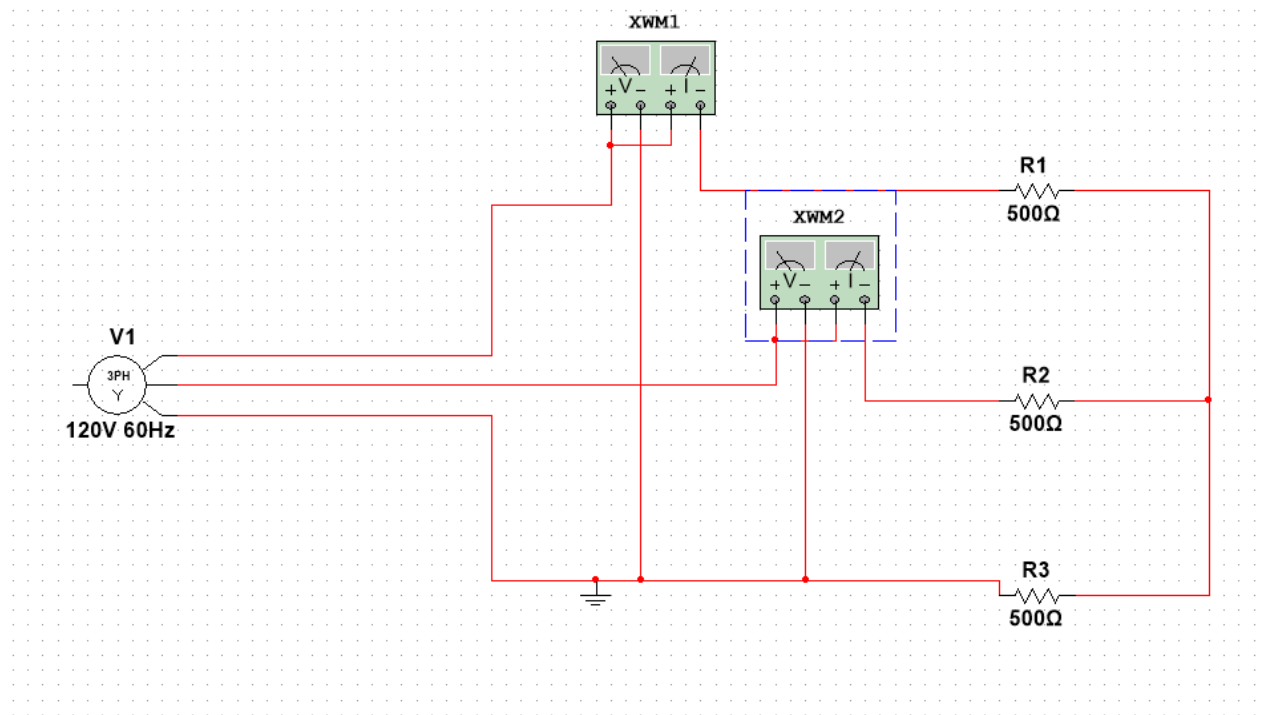


Figure 29.1 : Circuit diagram of three phase power supply measuring with two wattmeter.

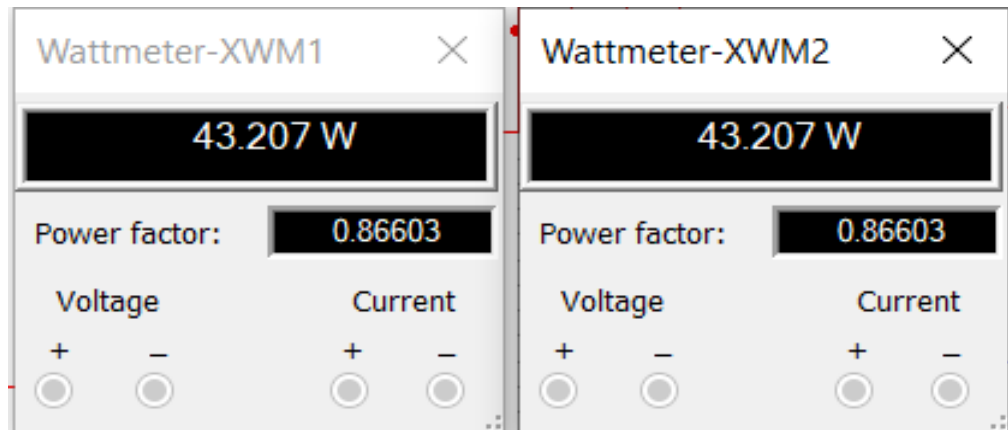


Figure 29.2: Reading of the wattmeter.

Conclusion: The two-wattmeter method reduces from six to four number of input channels required to measure power on a three-phase system. We can measure both balance system and unbalance system through this method.

Experiment No: 30

Experimental Name: Simulation of transistor as a switch and as an inverter in NI Multisim.

Objective:

1. To construct a circuit which can acts as a switch and as an inverter.
2. To verify both the operation in simulation.
3. To understand the working principle of these operation.

Component requires for simulation:

1. Transistor.
2. Resistor.
3. DC source.
4. Switch.
5. Connecting wire.
6. LED
7. Oscilloscope.

Experimental setup:

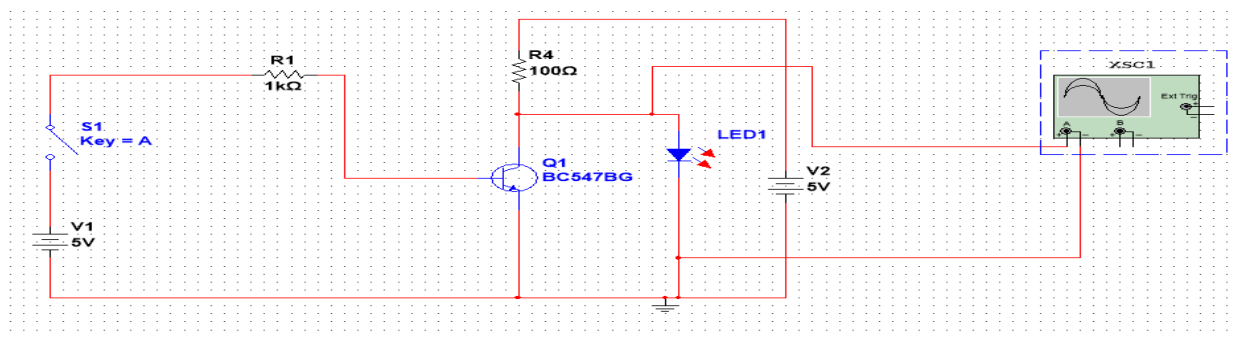


Figure 30.1: Circuit diagram when switch is open.

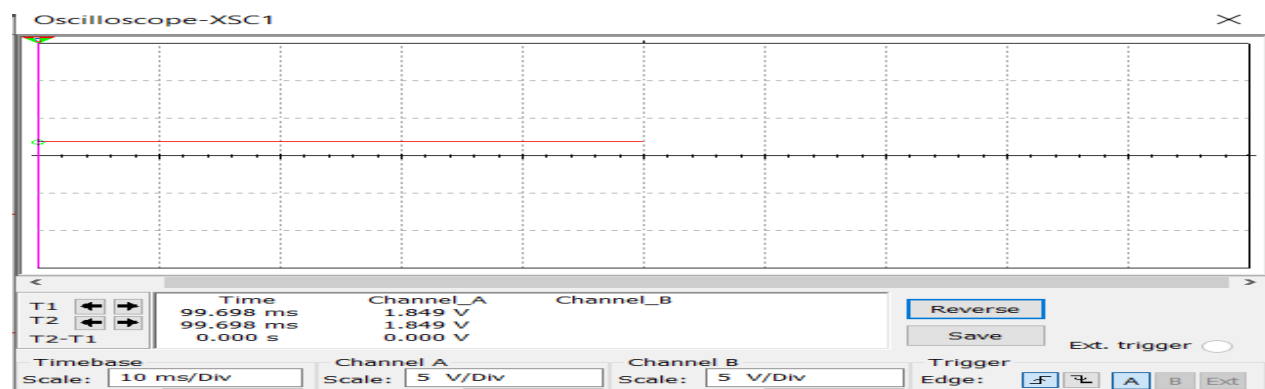


Figure 30.2: Oscilloscope output when switch is open.

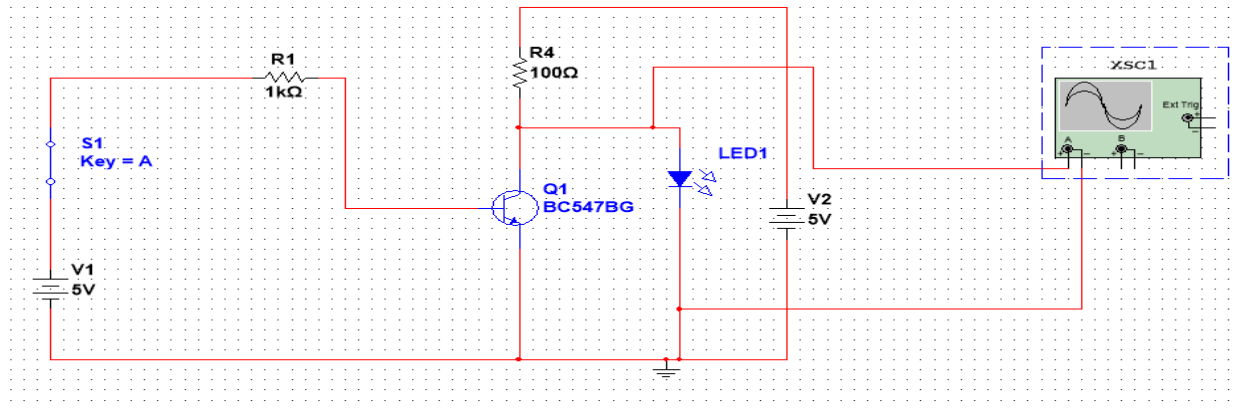


Figure 30.3: Circuit diagram when switch is closed

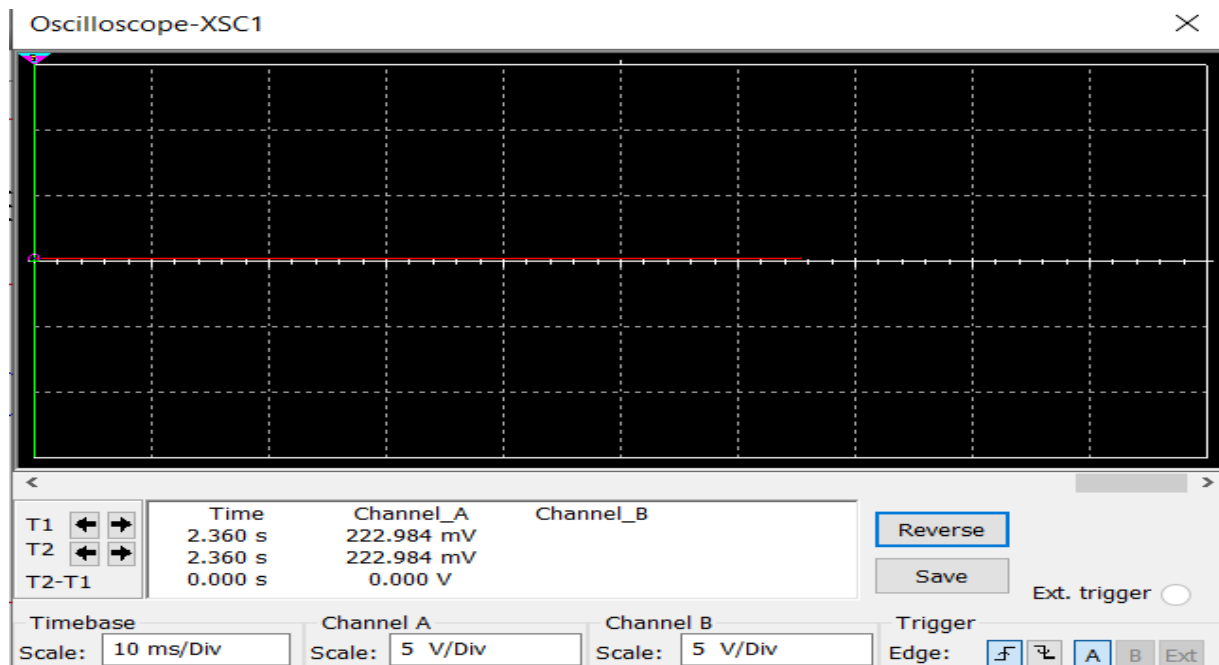


Figure 30.4: Oscilloscope output when switch is closed

Conclusion: If we observe the circuit we can see that when the switch was open the LED was blinking and we have founded a waveshape in the oscilloscope again when the switch was closed the LED was off and we have no signal in the output. So, giving positive signal we got no output and without giving signal we have a signal that means it is inverting the input and turning on and off the LED was done by switching operation.