# GOVT CO-ED POLYTECHNIC

BYRON BAZAR RAIPUR (C.G.)

**LAB MANUAL**

Branch : Electrical Engineering

Year & Semester : 3rd Year / 5th Semester

*2024561(025) – Electrical & Electronics Simulation (Lab)*

# CONTENTS

# Experiment No: 1

**AIM:** Connect one or more source and discrete components and complete the circuit in simulation window.

**System Requirements:** PC with Windows 10 (4GB RAM), MultiSim software.

## Theory

MultiSim software is very flexible for designing and analysing circuits. The simulation tools available are easy to modify. Selecting and placing components from the drop-down menu, across the simulation platform can be easily edited.

In this experiment, a simple lamp is connected to a DC voltage source and its current is controlled through a resistor. A switch is placed in the circuit to start the simulation. The value of current and voltage is displayed using voltage and current probe.

## Procedure:

- Open the MultiSim software working space.
- From the drop-down menu of components list select a DC voltage source, a switch, a lamp, a resistor, a ground terminal.
- Make the circuit connection as per the Fig. 1.1.
- Start the simulation by pressing play button  in task bar menu.
- Note the value of current and voltage in circuit using suitable probe.

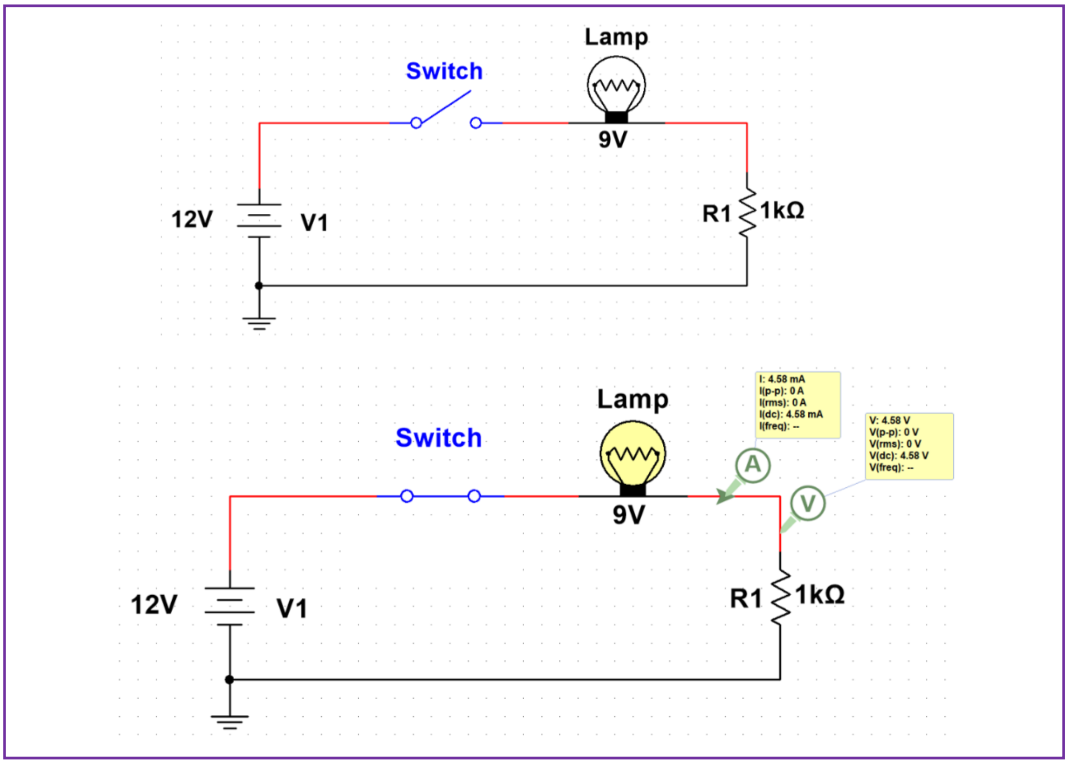


Fig 1.1: Circuit Diagram Showing Components Connected in MultiSim Software

## Result:

Thus, the circuit has been successfully simulated in MultiSim software. We noticed, how the lamp glows after running the simulation. No lag is observed while simulation. Current and voltage probe is easy to use.

## Viva-Voce:

Q 1. How the value of component is varied?
Q 2. What is the power rating of lamp?
Q 3. What is the value of current flowing in the circuit?
Q 4. Can you simulate the circuit without ground terminal?
Q 5. How the switch is operated in simulation?
Q 6. Can you delete any component, while the simulation is running?

# Experiment No: 2

**AIM:** Connect the resistors in series and parallel combination and measure the current and voltage in the circuit using simulation tool.

**System Requirements:** PC with Windows 10 (4GB RAM), MultiSim software.

## Theory

MultiSim software is very flexible for designing and analysing circuits. The simulation tools available are easy to modify. Selecting and placing components from the drop-down menu, across the simulation platform can be easily edited.

In this experiment, three loop based resistive network is designed. First, the KVL is applied theoretically and then simulated in the MultiSim software.

## Procedure:

- Open the MultiSim software working space.
- From the drop down menu of components list select a DC voltage source, a few resistor, a ground terminal.
- Make the circuit connection as per the Fig. 2.1.
- Start the simulation by pressing play button  in task bar menu.
- Note the value of current and voltage in circuit using suitable probe.

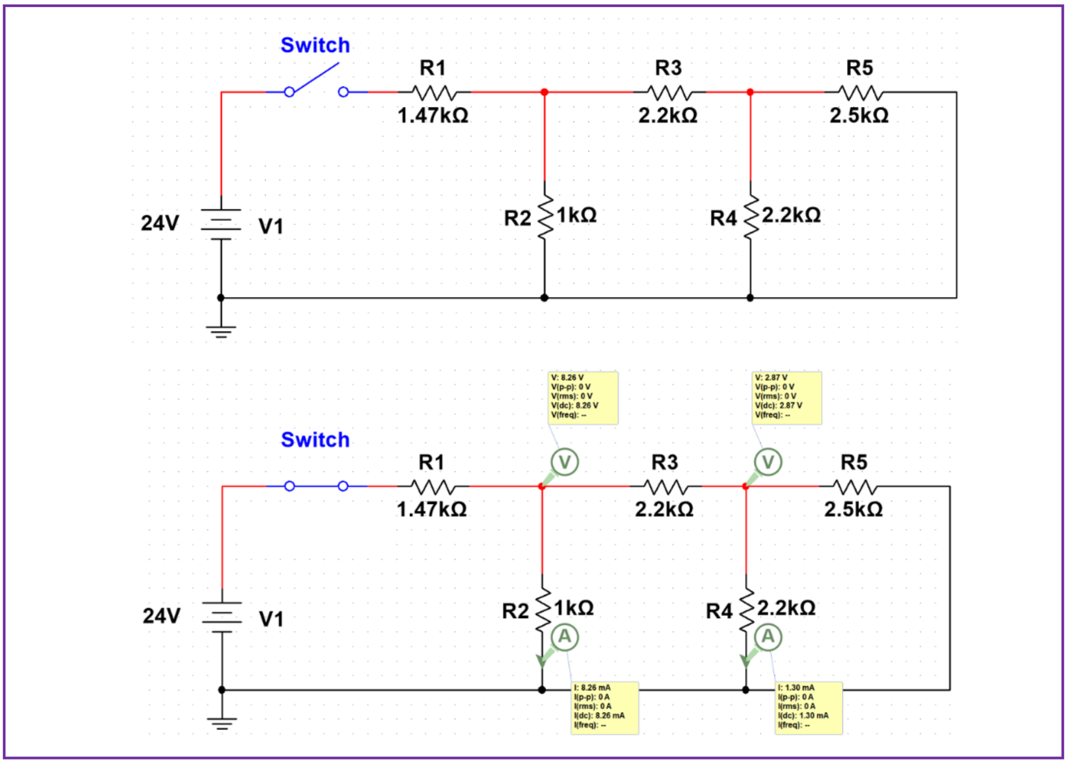


Fig 2.1: Circuit Diagram Showing Resistive Network Connected in MultiSim Software

## Observation Table:

| S. No. | Current through/ Voltage across | Ampere/Volts |
|---|---|---|
| 1 | R1 | |
| 2 | R2 | |
| 3 | R3 | |
| 4 | R4 | |
| 5 | R5 | |

## Result:

Thus, the circuit has been successfully simulated in MultiSim software. No lag is observed while simulation. Current and voltage probe is easy to use.

## Viva-Voce:

Q 1. How the value of component is varied?
Q 2. What is the power rating of each resistor?
Q 3. What is the value of current flowing in loop one of the circuit?
Q 4. Can you simulate the circuit without ground terminal?
Q 5. How the switch is operated in simulation?
Q 6. Can you delete any component, while the simulation is running?

# Experiment No: 3

**AIM:** Design and test a Zener voltage regulator & plot the Zener current vs voltage.

**System Requirements:** PC with Windows 10 (4GB RAM), MultiSim software.

## Theory

MultiSim software is very flexible for designing and analysing circuits. The simulation tools available are easy to modify. Selecting and placing components from the drop-down menu, across the simulation platform can be easily edited.

In this experiment, three loop based resistive network is designed. First, the KVL is applied theoretically and then simulated in the MultiSim software.

## Procedure:

- Open the MultiSim software working space.
- From the drop down menu of components list select a DC voltage source, a few resistor, a ground terminal, a Zener diode, a potentiometer, an SPST switch.
- Make the circuit connection as per the Fig. 3.1.
- Start the simulation by pressing play button  in task bar menu.
- Press button A to change the value of potentiometer resistance, which in turn vary the input voltage across the Zener diode.
- R_Zener is selected to limit the value of excess current through Zener diode under reverse biased operation.
- Note the value of current through Zener diode and voltage across load resistor, $R_L$ in circuit using suitable probe.

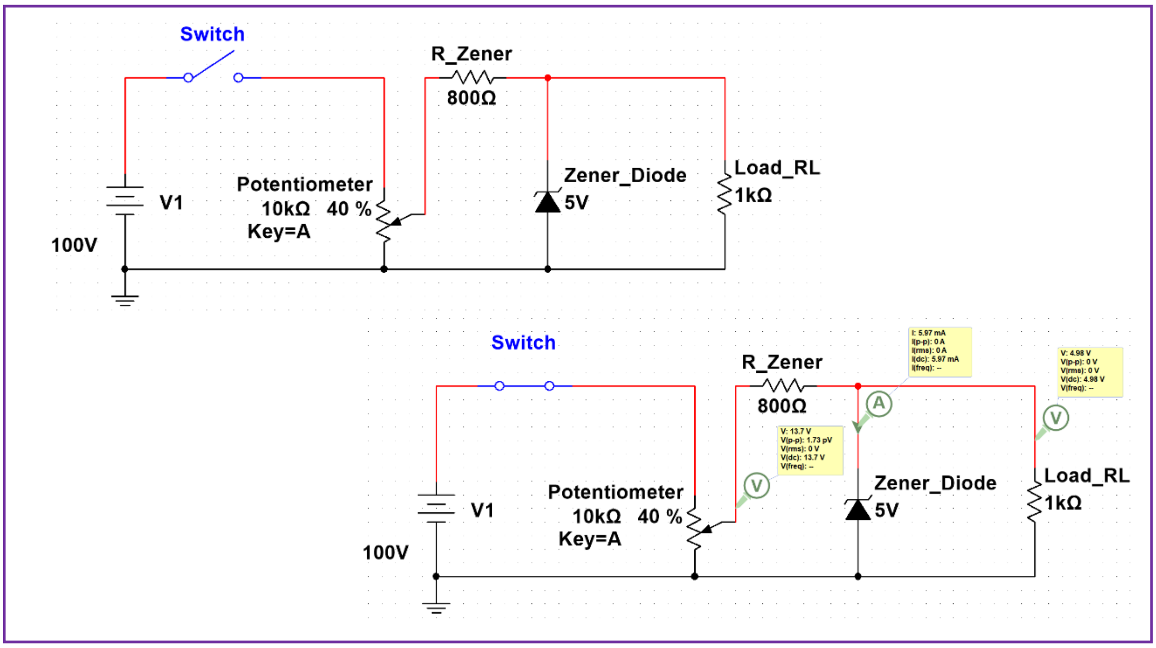


Fig 3.1: Circuit Diagram Showing Zener Diode Based Voltage Regulator Circuit Connected in MultiSim Software

## Observation Table:

| S.No. | Input Voltage | Zener Current | Load Voltage |
|---|---|---|---|
| 1 | | | |
| 2 | | | |
| 3 | | | |
| 4 | | | |
| 5 | | | |
| 6 | | | |
| 7 | | | |
| 8 | | | |
| 9 | | | |
| 10 | | | |

## Result:

Thus, the circuit has been successfully simulated in MultiSim software. No lag is observed while simulation. The voltage across load resistor remains almost constant as per the theoretical limits. Current and voltage probe is easy to use.

## Viva-Voce:

Q 1. How the value of component is varied?
Q 2. What is the power rating of Zener diode?
Q 3. What is the value of current flowing in the Zener diode?
Q 4. How the Zener current is regulated in the circuit?
Q 5. Can you simulate the circuit without ground terminal?
Q 6. How the potentiometer is operated in simulation?
Q 7. Can you delete any component, while the simulation is running?

# Experiment No: 4

**AIM:** Simulate and test summing amplifier circuit.

**System Requirements:** PC with Windows 10 (4GB RAM), MultiSim software.

## Theory

MultiSim software is very flexible for designing and analysing circuits. The simulation tools available are easy to modify. Selecting and placing components from the drop-down menu, across the simulation platform can be easily edited.

In this experiment, an ideal op-amp is used to simulate summing operation of two variable DC sources. The output is first theoretically calculated and then simulated in the MultiSim software.

## Procedure:

- Open the MultiSim software working space.
- From the drop-down menu of components list select two DC voltage source, a few resistors, a ground terminal, an ideal Op-Amp, an SPST switch.
- Make the circuit connection as per the Fig. 4.1.
- Start the simulation by pressing play button  in task bar menu.
- Press space button to change the state of switch, which in turn vary the input voltage to summing amplifier.

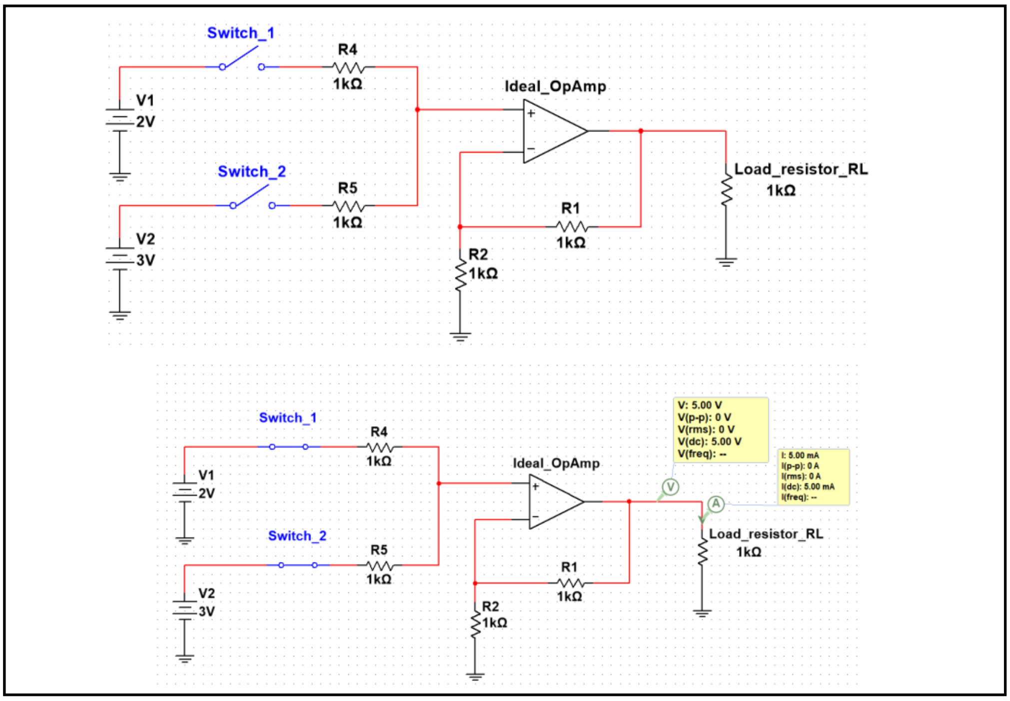


Fig 4.1: Circuit Diagram Showing Summing Amplifier Circuit Connected in MultiSim Software

## Observation:

| | |
|---|---|
| Output Voltage, $V_o$ = | When V1 is only active. |
| Output Voltage, $V_o$ = | When V2 is only active. |
| Output Voltage, $V_o$ = | When V1 and V2 is active. |

## Result:

Thus, the circuit has been successfully simulated in MultiSim software. No lag is observed while simulation. The voltage across load resistor comes out to be sum of individual input voltages. Current and voltage probe is easy to use.

## Viva-Voce:

Q 1. How the value of component is varied?
Q 2. What is the power rating of Op-Amp?
Q 3. What is the value of current flowing in the load resistor?
Q 4. How the feedback resistor works in the circuit?
Q 5. Can you simulate the circuit without ground terminal?
Q 6. Can you delete any component, while the simulation is running?

# Experiment No: 5

**AIM:** Simulate and test integrator circuit.

**System Requirements:** PC with Windows 10 (4GB RAM), MultiSim software.

## Theory

MultiSim software is very flexible for designing and analysing circuits. The simulation tools available are easy to modify. Selecting and placing components from the drop-down menu, across the simulation platform can be easily edited.

In this experiment, an ideal op-amp is used to simulate integrator amplifier circuit. The circuit utilises two variable AC sources. One source generates square pulses with time period of 1 mili-sec & other generates sinusoidal wave with operating frequency being 1 kHz. The circuit simply integrates the input signal and the output is visible on the CRO module of MultiSim software. The output is first theoretically calculated and then simulated in the MultiSim software.

## Procedure:

- Open the MultiSim software working space.
- From the drop-down menu of components list select two voltage source, a few resistors, a ground terminal, an ideal Op-Amp, an SPDT switch.
- Make the circuit connection as per the Fig. 5.1.
- Start the simulation by pressing play button  in task bar menu.
- Press space button to change the state of switch, which in turn vary the input voltage to integrator amplifier.

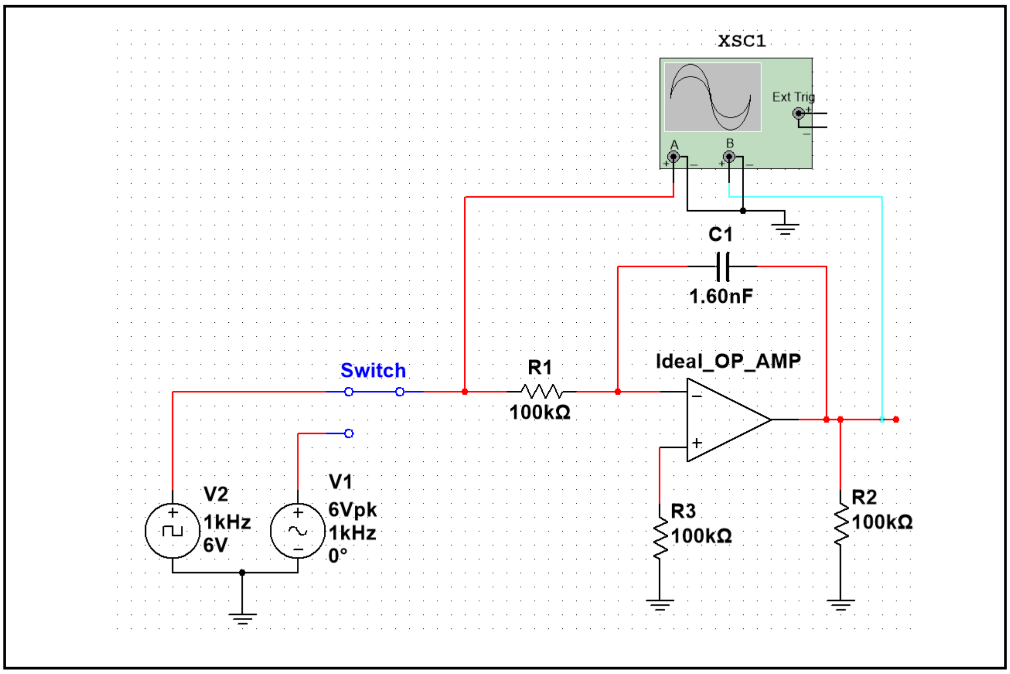


Fig 5.1: Circuit Diagram Showing Integrator Amplifier Circuit Connected in MultiSim Software

## Result:

Thus, the circuit has been successfully simulated in MultiSim software. No lag is observed while simulation. The voltage waveform across load resistor changes as per the law of integration.

## Viva-Voce:

Q 1. How the value of component is varied?
Q 2. What is the power rating of Op-Amp?
Q 3. What is the value of current flowing in the load resistor?
Q 4. How the feedback resistor works in the circuit?
Q 5. Can you simulate the circuit without ground terminal?
Q 6. Can you delete any component, while the simulation is running?

# Experiment No: 6

**AIM:** Develop a series R-L-C circuit and analyse the relationship of V and I waveform in under damped, critically damped & over damped condition.

## Theory

MultiSim software is very flexible for designing and analysing circuits. The simulation tools available are easy to modify. Selecting and placing components from the drop-down menu, across the simulation platform can be easily edited.

In this experiment, the transient response of an R-L-C series circuit is simulated to study its behaviour to step input signal. The relationship of voltage and current for underdamped, critically damped and overdamped situation is studied. The output is first theoretically calculated and then simulated in the MultiSim software.

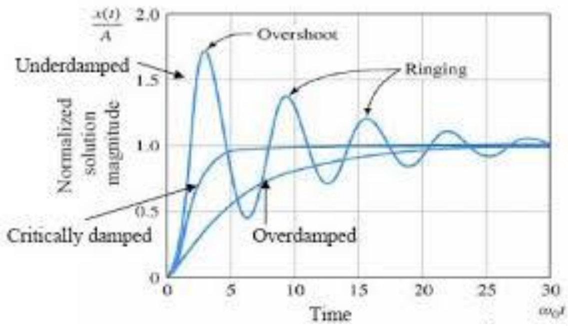


Fig 6.1: Step Response of RLC Circuit

## Procedure:

- Open the MultiSim software working space.
- From the drop-down menu of components list select step voltage source, a resistor, an inductor, a capacitor, a ground terminal.
- Make the circuit connection as per the Fig. 6.2.
- Place a voltage probe across the capacitor and change the mode of simulation to transient analysis. Set initial condition to zero and end time to 0.005 sec.
- Set maximum step time to 1e-005 and press save icon on the menu.
- *FOR UNDERDAMPED CONDITION*
  - Set R = 40 Ω, L = 10 mH and C = 1 uF
- Start the simulation by pressing play button  in task bar menu & plot.
- *FOR CRITICALLY DAMPED CONDITION*
  - Set R = 200 Ω, L = 10 mH and C = 1 uF
- Start the simulation by pressing play button  in task bar menu & plot.
- *FOR OVER DAMPED CONDITION*
  - Set R = 1 kΩ, L = 10 mH and C = 1 uF
- Start the simulation by pressing play button  in task bar menu & plot.

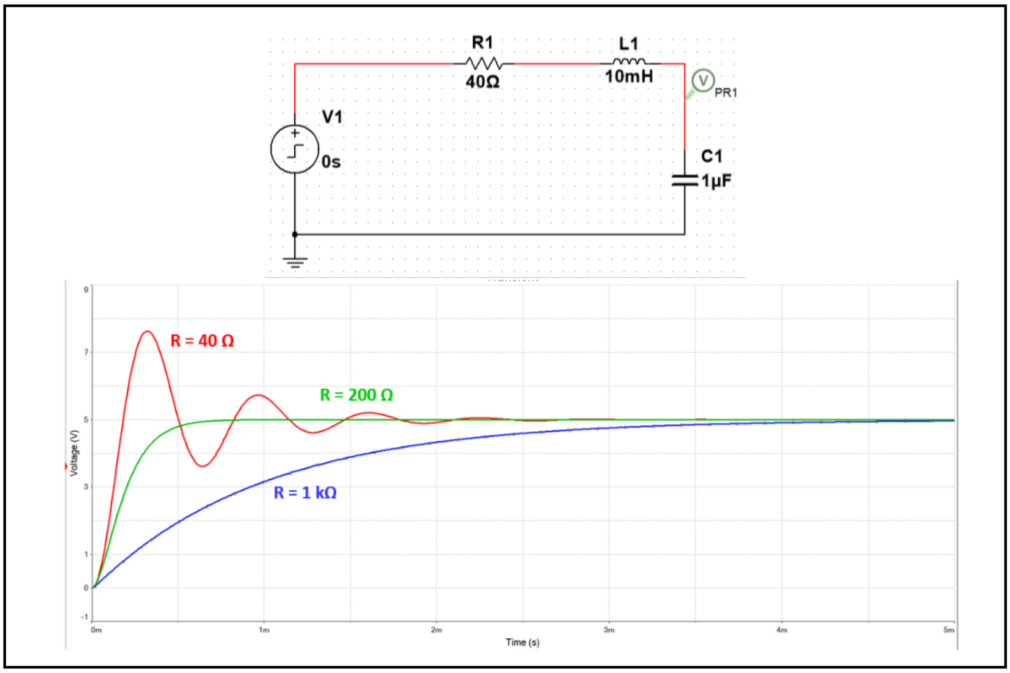


Fig 6.2: Series RLC Circuit Along with its Output Waveform
For Different Damping Conditions

## Result:

Thus, the circuit has been successfully simulated in MultiSim software. No lag is observed while simulation. The voltage waveform for step response, for different damping conditions, has been successfully plotted.

## Viva-Voce:

Q 1. How the value of component is varied?
Q 2. What is the value of current flowing in the resistor?
Q 3. What is the resonant frequency of the circuit?
Q 4. Can you simulate the circuit without ground terminal?
Q 5. Can you delete any component, while the simulation is running?
Q 6. Define the term damping?
Q 7. State the condition of critical damping.

# Experiment No: 7

**AIM:** Use program file to plot the rotor speed of a three-phase slip ring induction motor with varying rotor resistance and constant load torque.

## Theory

SciLab software is very flexible for designing and analysing circuits. The simulation tools available are easy to modify. Selecting and placing components from the drop-down menu, across the simulation platform can be easily edited.

In this experiment, an ideal op-amp is used to simulate integrator amplifier circuit. The circuit utilises two variable AC sources. One source generates square pulses with time period of 1 mili-sec & other generates sinusoidal wave with operating frequency being 1 kHz. The circuit simply integrates the input signal and the output is visible on the CRO module of MultiSim software. The output is first theoretically calculated and then simulated in the MultiSim software.

## Procedure:

- Open the MultiSim software working space.
- From the drop-down menu of components list select two voltage source, a few resistors, a ground terminal, an ideal Op-Amp, an SPDT switch.
- Make the circuit connection as per the Fig. 5.1.
- Start the simulation by pressing play button  in task bar menu.
- Press space button to change the state of switch, which in turn vary the input voltage to integrator amplifier.

## Result:

Thus, a program to calculate sum of first ‘N’ natural numbers using while loop has been compiled successfully.

# Experiment No: 8

**AIM:** Use program file to plot the efficiency of a given transformer as a function of the load currents.

## Algorithm:

## Result:

Thus, a program to check a given number is prime or not using loop with break statement has been compiled successfully.

# Experiment No: 9

**AIM:** Develop a half wave-controlled rectifier circuit with R-L load and analyse the voltage and current waveform across the load.

## Theory

MultiSim software is very flexible for designing and analysing circuits. The simulation tools available are easy to modify. Selecting and placing components from the drop-down menu, across the simulation platform can be easily edited.

In this experiment, a half wave-controlled rectifier is designed using SCR and by controlling its firing angle, a suitable amount of power can be delivered to it R-L load. The circuit utilises an AC source and a gate triggering pulse voltage source. One source generates sinusoidal waves to be delivered to R-L load & pulses are required to trigger SCR. The output is first theoretically calculated and then simulated in the MultiSim software.

## Procedure:

- Open the MultiSim software working space.
- From the drop-down menu of components list select an AC voltage source and a pulse voltage source, a resistor, a ground terminal, SCR, an SPST switch, an inductor.
- Make the circuit connection as per the Fig. 9.1.
- Start the simulation by pressing play button  in task bar menu.
- Press space button to change the state of switch.

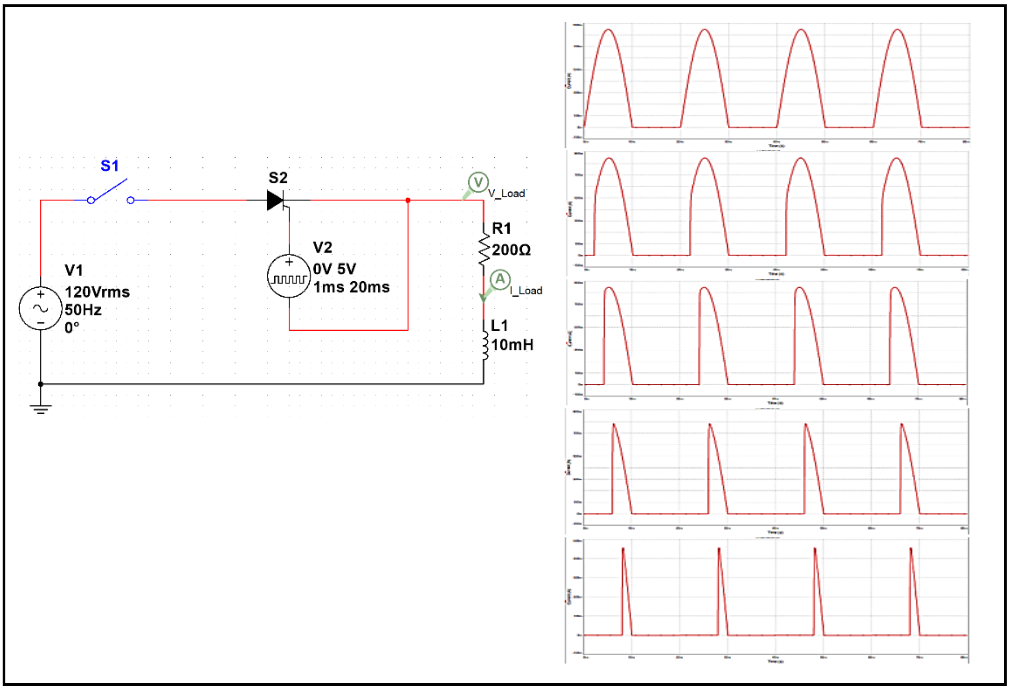


Fig 9.1: Circuit Diagram Showing SCR based Half Wave-Controlled Rectifier circuit with R-L load. Waveform Showing Output for Different Firing Angle.

## Observation Table:

| S.No. | Firing Angle | Load Current | Load Voltage |
|---|---|---|---|
| 1 | | | |
| 2 | | | |
| 3 | | | |
| 4 | | | |
| 5 | | | |

## Result:

Thus, the circuit has been successfully simulated in MultiSim software. No lag is observed while simulation. The voltage/current waveform across R-L load for different values of firing angle, has been successfully plotted.

## Viva-Voce:

Q 1. How the value of component is varied?
Q 2. What is the value of current flowing in the resistor?
Q 3. What is the operating frequency of SCR in the circuit?
Q 4. Can you simulate the circuit without ground terminal?
Q 5. Can you delete any component, while the simulation is running?
Q 6. Define the term commutation?
Q 7. State the reverse recovery time.

# Experiment No: 10

**AIM:** Design and test buck converter circuit using SCR.

## Theory

MultiSim software is very flexible for designing and analysing circuits. The simulation tools available are easy to modify. Selecting and placing components from the drop-down menu, across the simulation platform can be easily edited.

In this experiment, a buck converter is designed using SCR and by controlling its firing angle, and forced commutation (class-B) via L1 and C1, suitable amount of load voltage can be delivered to load resistance. The circuit utilises a DC source and a pulse voltage source required to trigger SCR. The output is first theoretically calculated and then simulated in the MultiSim software.

## Procedure:

- Open the MultiSim software working space.
- From the drop-down menu of components list select a DC voltage source and a pulse voltage source, a resistor, a ground terminal, SCR, an SPST switch, an inductor, a capacitor, a CRO module, voltage probe.
- Make the circuit connection as per the Fig. 10.1.
- Start the simulation by pressing play button  in task bar menu.
- Press space button to change the state of switch.

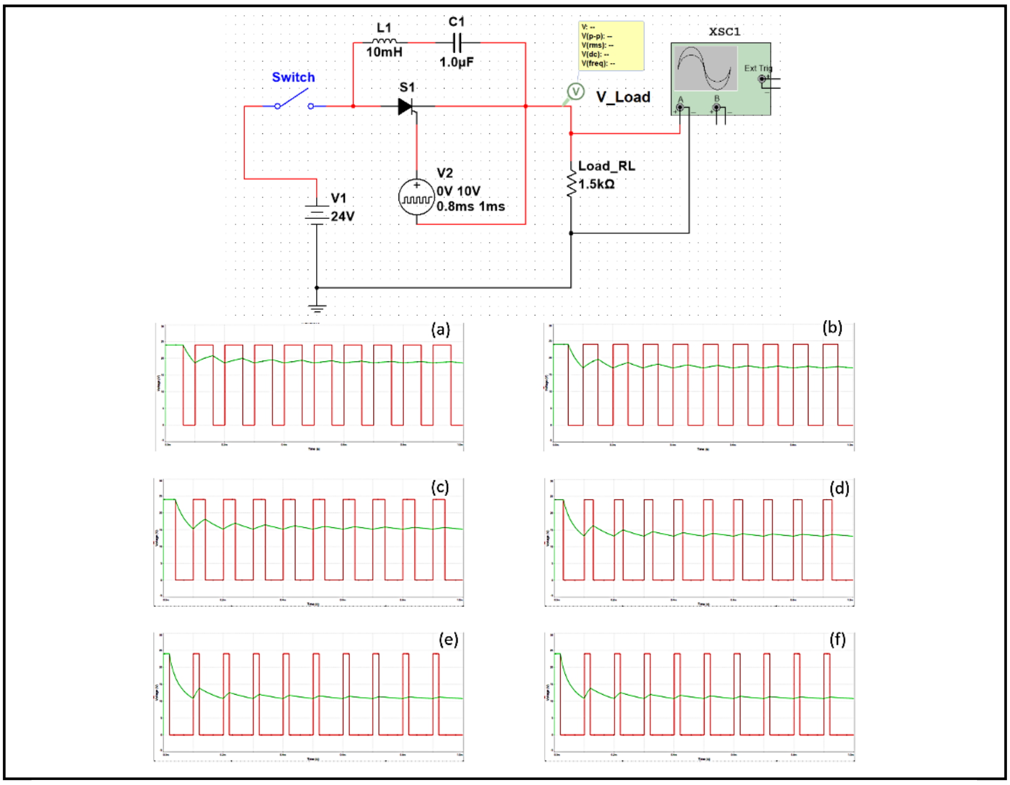


Fig 10.1: Circuit Diagram Showing SCR based Buck Converter circuit with R load and Class-B commutation.
Different duty cycle, **a. 60% b. 50% c. 40% d. 30% e. 20% f. 10%.**

## Result:

Thus, the circuit has been successfully simulated in MultiSim software. No lag is observed while simulation. The voltage/current waveform across R load for different values of duty cycle, has been successfully plotted.

## Viva-Voce:

Q 1. How the value of component is varied?
Q 2. What is the value of current flowing in the resistor?
Q 3. What is the operating frequency of SCR in the circuit?
Q 4. Can you simulate the circuit without ground terminal?
Q 5. Can you delete any component, while the simulation is running?
Q 6. Define the term forced commutation?
Q 7. State the reverse recovery time.

# Dos & Don’ts in the Computer Lab

| Dos | Don’ts |
|---|---|
| • Enter/exit lab quietly.<br>• Raise your hand before asking any doubt.<br>• Always have a clean & dry hand.<br>• Touch keyboard & mouse gently.<br>• Keep your work space clean.<br>• Search only approved websites | • No food or drinks in the lab.<br>• Do not mark on any part of computer.<br>• Do not change any key settings of computer.<br>• No magnets allowed in computer lab<br>• Do not pull any cable/cord of any system.<br>• Ask teacher before taking any printout. |