

EECS LAB REPORT (ECS357)

SEM – 5

Author: Shlok Mehndiratta

Roll no. 23309



**Department of Electrical Engineering and Computer Science
Indian Institute of Science Education and Research (IISER)
Bhopal**

INDIA

Lab Experiment List

Experiment 1: Familiarization with lab Instruments

- **Objective 1.1:** Introduction to Lab Instruments
- **Objective 1.2:** Running a LT Spice Simulation

Experiment 2: Familiarization with DC and AC sources

- **Objective 2.1:** IV characteristics of ohm's law (DC input).
- **Objective 2.2:** IV characteristics of ohm's law (AC input).

Experiment 3: RC and RL circuit

- **Objective 3.1:** Charging and discharging characteristics of capacitor using DC source.
- **Objective 3.2:** Effect on the output of RC circuit for a square wave input signal.
- **Objective 3.3:** Experiment with RL circuit with DC source. Record the voltage and current across the L with time.

Experiment 4: Low and High Pass Filters

- **Objective 4.1:** Design a passive low-pass filter in LTspice and verify with experiment.
- **Objective 4.2:** Design a passive high-pass filter in LTspice and verify with experiment.

Experiment 5: RLC circuit

- **Objective 5.1:** For the given RLC circuit, apply input signals of different frequencies and vary the resistance values for a fixed value of L and C and measure output voltage, $V_{out}(t)$.
- **Objective 5.2:** Measuring V_{out} for these combinations for observing underdamp, overdamp and critically damped behavior after adjusting the appropriate values of RLC.

Experiment 6: Diode behavior

- **Objective 6.1:** I-V characteristics of a diode.
- **Objective 6.2:** Study of Full-wave and Half-wave rectifier circuit
- **Objective 6.3:** Study of Clipper and clamper circuit.

Experiment 7: Bipolar Junction Transistor (BJT) analysis

- **Objective 7.1:** Study of common emitter configuration (IC vs V_{CE} for different V_{BE}).

Experiment 8: Familiarization with Op-Amp circuits

- **Objective 8.1:** Design and realize Inverting amplifier using IC741Op-amp.
- **Objective 8.2:** To study the working of op-amp as differentiator.
- **Objective 8.3:** To study the working of op-amp as integrator.

Experiment 9: To design a Schmitt trigger using Op-amp IC-741.

- **Objective 9.1:** To design a Schmitt trigger using Op-amp IC-741.

Experiment 10: Digital Logic

- **Objective 10.1:** To design Logic gates using transistor
- **Objective 10.2:** To study about Demultiplexer

1. Familiarize with lab instruments

Objectives

1. Introduction to lab instruments
2. Running a LT Spice simulation

Introduction to lab instruments

Instruments studied

1. Multi-meter
2. Oscilloscope
3. Function generator
4. Small resistors
5. DC power supply

Determining the resistance of given resistor

We determined the resistance of the given resistor using a multimeter. Based on the resistor's colour code, the expected resistance was approximately $10\text{ k}\Omega$.



To measure the resistance, we set the multimeter dial to the resistance measurement mode and connected the test leads to the COM and Ω terminals. The multimeter's functionality was confirmed, as the measured resistance closely matched the calculated value, falling within the acceptable error range. This verified the accuracy of the multimeter in measuring resistance.

DC Power Supply and Voltage measurement

The DC power supply was set to 5V, and the output was then measured using a multimeter. To measure the voltage, the multimeter dial was set to voltage measurement mode, and the test leads were connected to the COM and V terminals.



The measured voltage closely matched the voltage set on the power supply, verifying the accuracy of both the power supply and the measurement procedure. This confirms that the setup and the method used to measure the voltage were correct.

Using the Oscilloscope and Function generator

We experimented with Oscilloscope and function generator. Making a function wave and trying to find its plot without the auto function, using knob. Adjusting scale and Location we finally get a good representation of the original function wave.



Left is function generator and right is oscilloscope. We generated a square wave function and observed the output on the oscilloscope.

1. First connect the wires with any one of the channels of oscilloscope and function generator.
 2. Design a square wave in function generator by going to menu and selecting square wave.
 3. Adjust the parameters on the oscilloscope to nicely visualize the wave output.
- As it's shown, oscilloscope plots the square wave voltage input.

Precautions

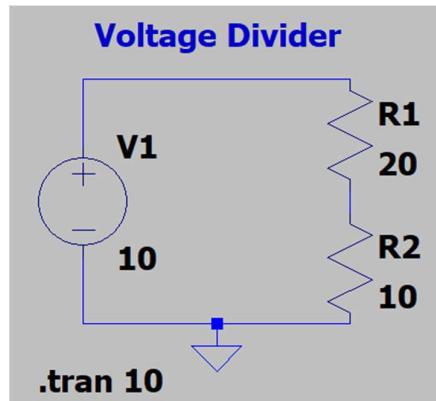
- One shouldn't operate at high voltages, as it can be dangerous. For our experiments, we should not go above 12 volts.
- Always turn off the power supply, multimeter, oscilloscope, and other equipment when not in use to conserve energy and prevent accidents.
- Before powering on any device, double-check all connections to ensure they are secure and correctly configured to prevent short circuits or equipment damage.
- Keep all instruments dry and away from liquids to prevent electrical hazards and equipment malfunction.
- Always ensure that all equipment is properly grounded to avoid electrical shocks and interference in measurements.

LT Spice Simulation

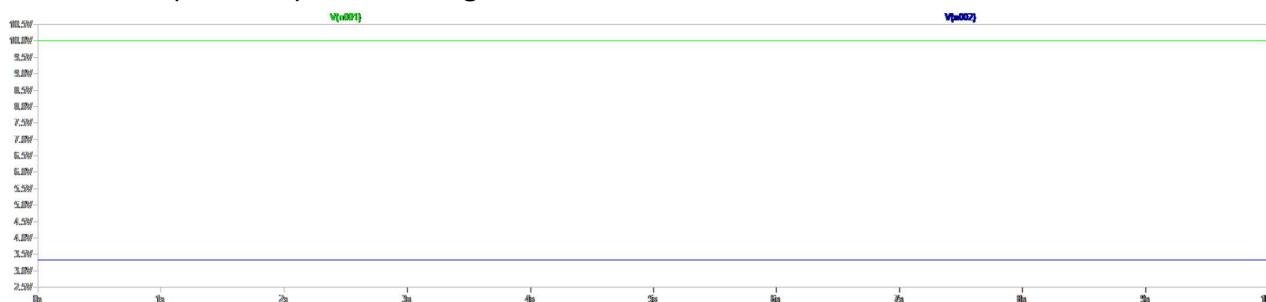
Prerequisites

Install LTSpice on your device. One can install it from [LTspice - EngineerZone \(analog.com\)](http://LTspice.EngineerZone.analog.com)

Schematic



We implemented a basic voltage divider on LTSpice with above schematic. We simulated it to compute and plot the voltages across both resistors.



Green is about both the resistors, while blue is about the 10Ω resistor $\approx 3.4V$

We thus learnt very basic functionality of LTSpice software. It can be later used in our course to simulate advanced electronic circuits.

LAB Report 2 ECS 327

Date: August 14, 2025

Name: Shlok Mehndiratta

Roll No: 23309

Title of Experiments:

Experiment 2: Familiarization with DC and AC sources

- **Objective 2.1:** Verification of ohm's law using IV characteristics (DC input).
- **Objective 2.2:** Verification of ohm's law using IV characteristics (AC input).

Objective 2.1:

IV characteristics of ohm's law (DC input)

Brief Description:

Current through an ohmic resistor increases proportionally with the increase in applied voltage i.e. $V = R \times I$, where V is applied voltage, and I is the current through the ohmic resistor and R is the proportionality constant which is known as resistance.

Equipment/Components Required:

Sl. No.	Component	Specifications/Value	Quantity
1.	Resistor	100 kΩ, 457 kΩ	01 each
2.	Variable Voltage Source	0–12 V with resolution 0.5 V or higher	01

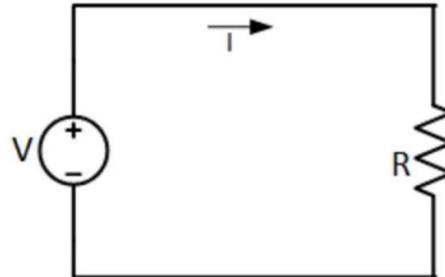
Schematic diagram:

Figure 1.1: Circuit diagram of the experiment

Results:

For 100 kΩ Resistance:

V	I	R
0 v	0 μA	100 kΩ
0.5V	4.8 μA	100 kΩ
1 V	9.7 μA	100 kΩ
1.5V	14.8 μA	100 kΩ
2 V	19.7 μA	100 kΩ

Slope : $\frac{\Delta V}{\Delta I} = \frac{(19.7 - 0) V}{(19.7 - 0) \mu A} \approx 101.5 \text{ k}\Omega$

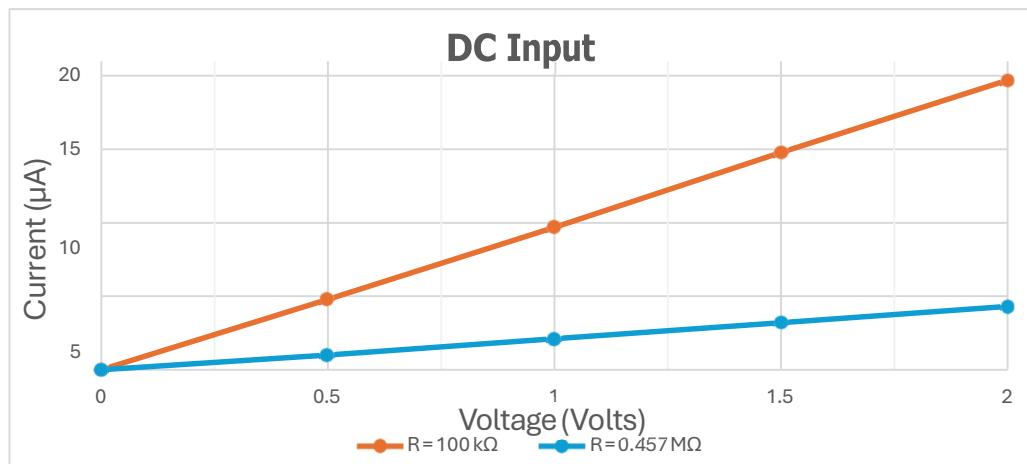
For 0.457 MΩ Resistance:

V	I	R
0 V	0 mA	0.457 MΩ
0.5 V	1 μA	0.457 MΩ
1 V	2.1 μA	0.457 MΩ
1.5 V	3.2 μA	0.457 MΩ
2 V	4.3 μA	0.457 MΩ

↙
Diagonal ↗

Slope : $\frac{\Delta V}{\Delta I} = \frac{(0.2 - 0)V}{(4.3 - 0) \mu A} = 0.465 \text{ M}\Omega$

IV characteristic plot:



Discussion:

In this experiment, we prepared a simple circuit with resistance in series. We used function generator to feed a DC input for various values of voltages and measured the current across the circuit using a digital multimeter for different input voltages. After this we plotted the IV graph for both the resistances and calculated the slope of both the curves. The slope of IV curve gives us the reciprocal of resistance.

Conclusion:

From the above results, we can conclude that the resistance values we obtained from the slopes are approximately equal to the applied resistances and the IV characteristic curve is almost linear.

Thus, **IV characteristics of a circuit at DC input obeys the ohm's law.**

Objective 2.2:

IV characteristics of ohm's law (AC input)

Brief Description:

Resistors are bidirectional electronic passive component and hence the behavior of the resistor does not change with the direction of current flowing through it. In AC circuits as shown in Fig. 1.2, the direction of the current flowing through a component changes with time and the change in the direction of has no effect on the behavior of the resistor, thus the current will rise and fall as the applied voltage rises and falls. In this case, the current through and voltage across the resistor reach to a maximum value, then fall through zero and reach to a minimum value at exactly the same time. i.e, they rise and fall simultaneously and are said to be “in-phase”.

Equipment/Components Required:

Sl. No.	Component	Specifications/Value	Quantity
1.	Resistor	100 kΩ, 457 kΩ	01 each
2.	Variable Voltage Source	0–12 V with resolution 0.5 V or higher	01

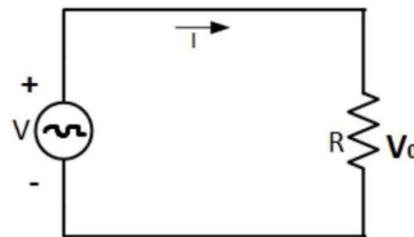
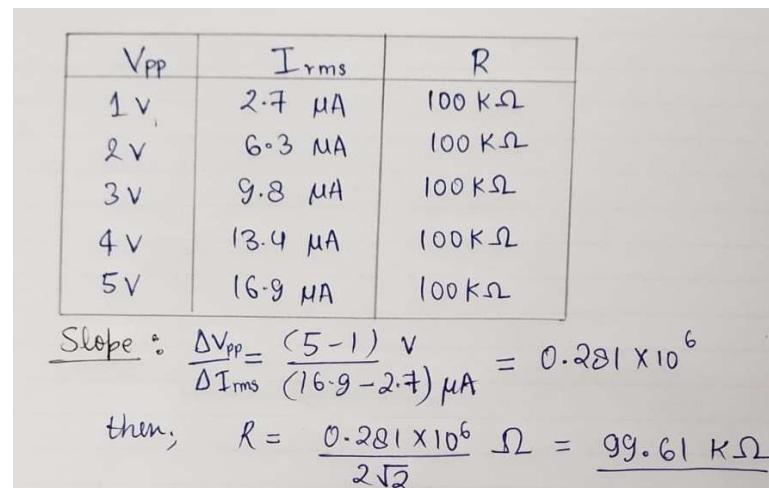
Schematic diagram:

Fig. 1.2: Circuit diagram of the experiment

Results:

For 100 kΩ Resistance:



For $0.457 \text{ M}\Omega$ Resistance:

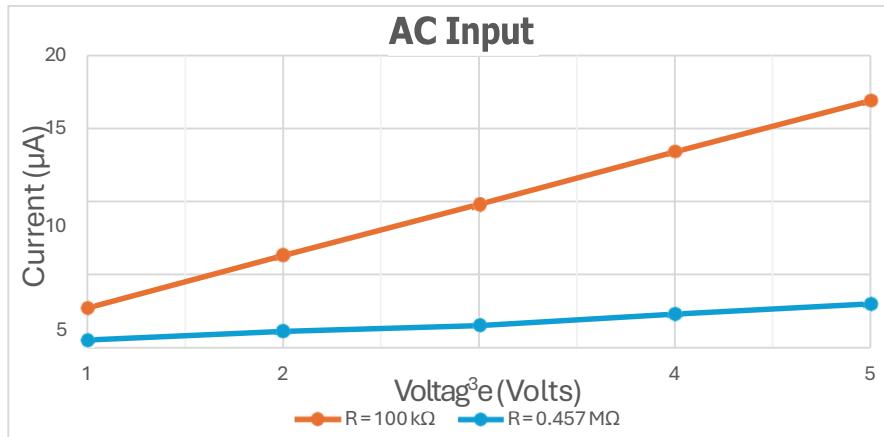
V_{pp}	I_{rms}	R
1 V	0.5 μA	$0.457 \text{ M}\Omega$
2 V	1.1 μA	$0.457 \text{ M}\Omega$
3 V	1.5 μA	$0.457 \text{ M}\Omega$
4 V	2.3 μA	$0.457 \text{ M}\Omega$
5 V	3.0 μA	$0.457 \text{ M}\Omega$

(Handwritten)

Slope $\frac{\Delta V_{pp}}{\Delta I_{rms}} = \frac{(5-1) \text{ V}}{(3 - 0.5) \mu\text{A}} = 1.6 \times 10^6$

then, $R = \frac{\Delta V_{pp}}{\Delta I_{rms}} \times \frac{1}{2\sqrt{2}} = \underline{565.7 \text{ k}\Omega}$

IV characteristic plot:



Discussion:

In this experiment, we prepared a simple circuit with resistance in series. We used function generator to feed a AC input for varying peak to peak values of voltages (V_{pp}) and then measured the current across the circuit using a digital multimeter for different values of V_{pp} . After this we plotted the IV graph for both the resistances and calculated the slope of both the curves. The slope of IV curve multiplied by $2\sqrt{2}$ (as $V_{rms} = V_{pp} / 2\sqrt{2}$) gives us the reciprocal of resistance.

Conclusion:

From the above results, we can conclude that the IV characteristic curve obtained is almost linear and resistance values we obtained from the slopes are approximately equal to the applied resistances.

Thus, **IV characteristics of a circuit at AC input obeys the ohm's law.**

LAB Report 3 ECS 327

Date: August 22, 2025

Name: Shlok Mehndiratta

Roll No: 23309

Title of Experiment:

Experiment 3: RC and RL circuit

- **Objective 3.1:** Charging and discharging characteristics of capacitor using DC source.
- **Objective 3.2:** Effect on the output of RC circuit for a square wave input signal.
- **Objective 3.3:** Experiment with RL circuit with DC source. Record the voltage and current across the L with time.

Objective 3.1:

Charging and discharging characteristics of capacitor using DC source.

Brief Description:

A discharged capacitor connected to a DC voltage source charges up with increase in the voltage of the source. Similarly, a charged capacitor discharges in opposite direction when the applied voltage of the DC source is reduced. Capacitors in this line can be compared with small batteries. The charge stored in a capacitor plates is proportional to the applied voltage and can be defined as, $Q = C \times V$, where Q is the stored charge, V is the applied voltage, and C is the proportionality constant which is known as capacitance.

The charging and discharging of capacitor generally takes some time and depends on the time constant (τ) i.e. the time taken for a certain percentage of charging and discharging of capacitor.

As per the RC circuit given in Fig. 1.3, the capacitor will charge up through the resistor connected in series with time until the voltage across the capacitor (V_C) becomes equal to supply voltage (V_S). The time constant specifies the rate of charge or discharge and can be defined as, $\tau = R \times C$, where R is in ohm and C is in Farad where the τ is in seconds. In the given circuit, at any point of time (t) the voltage across the capacitor can be given by, $V_C = V_S(1 - e^{-\frac{t}{\tau}})$.

From the above equation, it is clear that at $t = 5\tau$, the capacitor becomes $\sim 100\%$ charged, i.e. $V_C \approx V_S$; at $t = 4\tau$, the capacitor becomes $\sim 98\%$ charged, i.e. $V_C \approx 0.98V_S$; and at $t = \tau$, the capacitor becomes $\sim 63\%$ charged, i.e. $V_C \approx 0.63V_S$. In this situation, $t \leq 4\tau$ is known as transient period and $t > 4\tau$ is known as steady state where $t = 5\tau$, the capacitor can be treated as fully charged. Similar is the case for discharging condition.

Equipment/Components Required:

Sl. No.	Component	Specification Values	Quantity
1.	Resistor	100 kΩ	01
2.	Capacitors	100 μF	01
3.	DC Voltage source	0 – 12 V	01
4.	Multimeter/ Oscilloscope	Digital	01

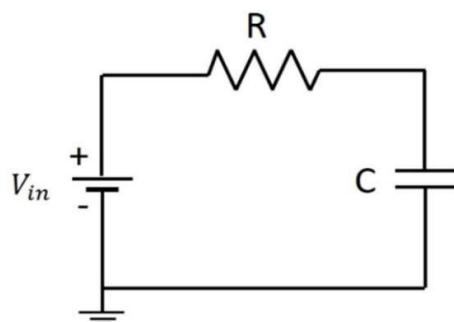
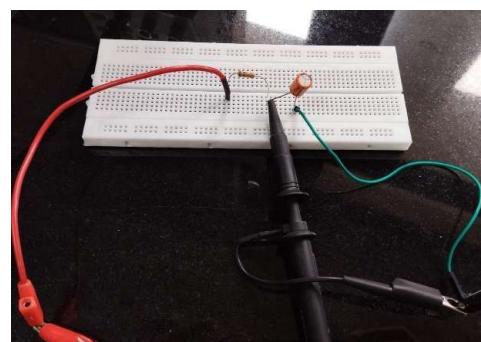
Schematic diagram:

Figure 1.3: Circuit diagram of the experiment



Results:

Charging of capacitor using DC source

$R = 100 \text{ k}\Omega$
 $C = 100 \mu\text{F}$

Time constant (T) = $RC = 10 \text{ sec}$

<u>Time</u>	<u>Output voltage across C</u>
4 sec	2.3 V
5 sec	2.8 V
10 sec	5.1 V
20 sec	7.6 V
30 sec	8.6 V
40 sec	8.8 V
50 sec	8.84 V

[Signature]

Discussion:

In this experiment, we prepared a simple circuit with resistor and capacitor in series. We used function generator to feed a DC input of 10 volts and then measured the rise in voltage across the capacitor using a digital multimeter during this charging cycle.

We observed that during the charging cycle as the charge gets accumulated on the capacitor, the voltage across it keeps increases until it is equal to the supply voltage.

Conclusion:

From the charging and discharging characteristics of capacitor using DC source, we can conclude that capacitors can act as small batteries that can store energy.

Objective 3.2:

Effect on the output of RC circuit for a square wave input signal.

Brief Description:

The basic charging discharging operation has been discussed in the previous section of this experiment. In this case, the difference is that the input is no longer a constant DC voltage instead, it is a square wave where the voltage will be high for a certain period of time and zero for a certain period consecutively. For the sake of understanding, one can argue that a square wave is a constant DC current source for a specific period time. From the theory discussed in previous section, it is now clear that in a RC circuit, a capacitor charges fully i.e. $\sim 100\%$ of the applied voltage in $t = 5\tau$. Similarly, discharges down to zero in $t = 5\tau$ when it is removed. In both RC charging and discharging circuits, the time $t = 5\tau$ always remains true as it is determined by the resistor-capacitor (RC) combination. In this case of square wave, if the voltage remains high for $t \geq 5\tau$, then the capacitor will charge completely. Similarly, if the voltage remains zero for $t \geq 5\tau$, the battery will discharge completely. Any time period $t < 5\tau$ for high and zero voltage will lead to a distorted or nearly equivalent triangular wave at the output across the capacitor.

Equipment/Components Required:

Sl. No.	Component	Specification Values	Quantity
1.	Resistor	470 k Ω	01 each
2.	Capacitors	10 nF	01 each
3.	AC voltage source/ Function Generator	0–3 MHz or Higher	01
4.	Digital Oscilloscope i.e. DSO	0–300 MHz or Higher	01

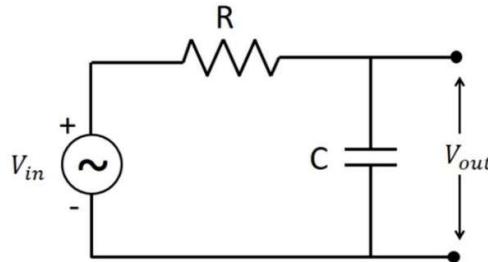
Schematic diagram:

Figure 2.2: RC circuit diagram of the experiment



Results:

Max	Min	Vpp	Top	Base	Amp	Avg	Vrms	Area	AreaP
CH1 3.20 V	-3.12 V	6.32 V	2.56 V	-2.48 V	5.04 V	40.3mV	2.52 V	4.82mVs	1.99mVs
CH2 1.72 V	-1.72 V	3.44 V	1.59 V	-1.59 V	3.18 V	-89.0mV	1.37 V	-10.6mVs	296uVs
Freq	Peri	+Duty	-Duty	+Wid	-Wid	+Rate	-Rate	Over	Pre
CH1 20.0 Hz	50.00ms	50.00 %	50.00 %	25.00ms	25.00ms	20.2kVs	-20.2kVs	12.69 %	12.69 %
CH2 20.0 Hz	50.00ms	50.00 %	50.00 %	25.00ms	25.00ms	322 Vs	-322 Vs	3.968 %	4.207 %
Rise	Fall	tVmax	tVmin	Vari	Vupper	Vmid	Vlower	VrmsP	
CH1 200.0us	200.0us	50.00ms	-25.00ms	6.37 V ²	2.06 V	40.0mV	-1.98 V	2.52 V	
CH2 7.900ms	7.900ms	-29.80ms	-2.600ms	1.86 V ²	1.27 V	3.79mV	-1.27 V	1.37 V	

Discussion:

In this RC Circuit, we gave an AC input of square wave and observed the output voltage in the oscilloscope. For the value of RC time constant (τ) = $R \times C = 4.7$ ms, we gave time period of input wave as 5 ms. We got the above voltage curve where current rises for every positive voltage input and decays for the negative voltage input.

For this output waveform, we have got the average **rise time** as **7.90 ms** and average **fall time** as **7.90 ms**.

Conclusion:

From the above experiment, we have studied the effect of square wave input signal on the output of RC circuit. We can conclude that for square wave, there is exponential rise and fall in the voltage and current values for alternating positive and negative input voltages in RC circuit.

Objective 3.3:

Experiment with RL circuit with square wave input signal. Record the voltage and current across the L with time.

Brief Description:

The series LR circuit is composed of an inductor (L) and resistor (R) connected in series along with a constant DC voltage source V. This is similar to that of RC circuit discussed in the previous experiment where capacitor (C) was used instead of inductor (L). Once the circuit is connected to a constant DC voltage source, the current starts flowing through the circuit but takes some time to reach to the maximum value as per ohms law i.e. V/R .

The reason behind the delay of reaching maximum current is the self-induced emf within the coil i.e. inductor due to the magnetic flux as per Lenz's law. Thus, it takes some time for the applied voltage source to neutralize the self-induced emf and thereafter the current becomes constant.

The rate of change of current and voltage depends on the time constant (τ), like that of a RC circuit. In the case of RL circuit, the time constant is determined and defined as $\tau = L/R$, where τ is in seconds when inductance L is in Henry and resistance R is in ohms.

The voltage across the L (V_L) at any time t can be given by $V_L = V e^{-t/\tau}$, where V is the applied voltage. Similarly, the current through the L (I_L) at any time t can be given by $I_L = \frac{V}{R}(1 - e^{-t/\tau})$, where V is the applied voltage and R is the resistance.

Equipment/Components Required:

Sl. No.	Component	Specification Values	Quantity
1.	AC Source/ Function Generator	0–3 MHz or Higher	01
2.	Resistors	1 KΩ	01
3.	Inductors	4.7 mH	01
4.	Digital Oscilloscope i.e. DSO	0–300 MHz	01
5.	Multimeter	Digital	01

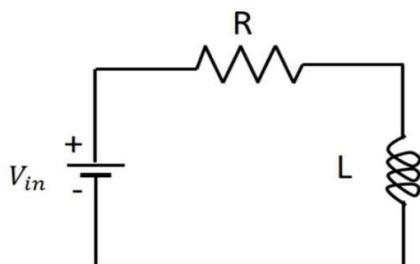
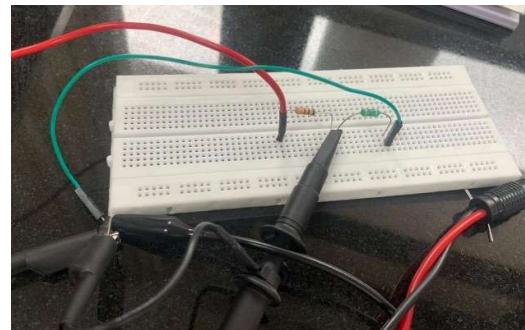
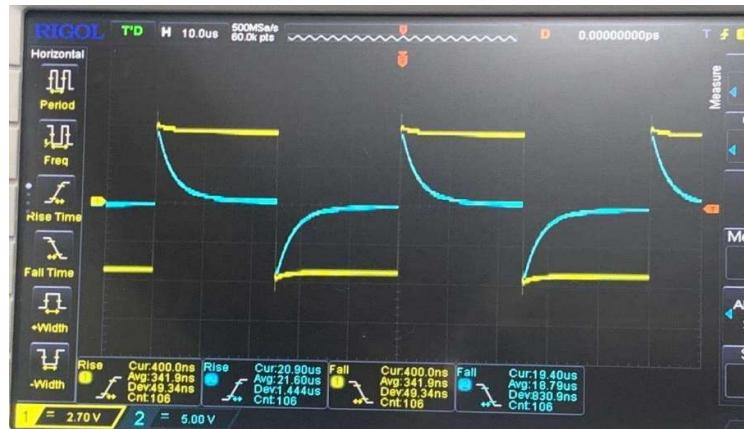
Schematic diagram:

Fig. 2.3: Circuit diagram of the LC experiment



Results:**Discussion:**

In this RL Circuit, we gave an AC input of a square wave and observed the output voltage in the oscilloscope. For the value of RL time constant (τ) = $L / C = 4.7 \mu s$, we gave the 5 ms input square wave and got the above voltage curve, where the current decays to zero for the input waveform.

For this output waveform, we have got the average **rise time** as **21.60 μs** and average **fall time** as **18.79 μs** .

Conclusion:

From the above experiment, we have studied the effect of a square wave input signal on the output of the RL circuit. We can conclude that for a square wave input to the RL series circuit, the output voltage decays and rises exponentially to zero for every positive and negative cycle respectively of the wave.

LAB Report 4 ECS 327

Date: August 29, 2025

Name: Shlok Mehndiratta

Roll No: 23309

Title of Experiment:

Experiment 4: Low and High Pass Filters

- **Objective 4.1:** Design a passive low-pass filter in LTspice and verify with experiment.
- **Objective 4.2:** Design a passive high-pass filter in LTspice and verify with experiment.

Objective 4.1:

Design a passive low-pass filter

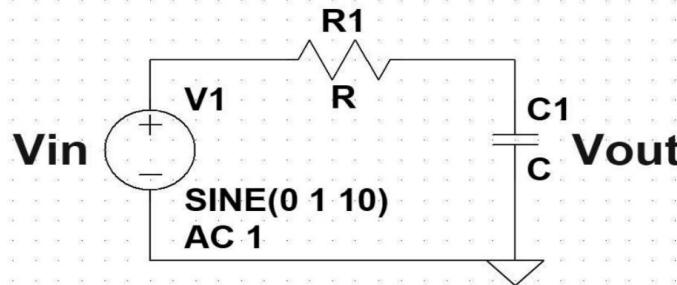
Schematic diagram:

Figure 1: Passive low-pass filter

Brief Description:

The RC low pass filter allows frequency signals below the cut-off frequency f_c .

$$f_C = \frac{1}{2\pi RC}$$

The output voltage signal attenuates when the supply frequency is greater than the cut-off frequency as shown in Figure 2. At the cut-off frequency, capacitor C's capacitive reactance is equal to resistor R's resistance, causing the output voltage to be 0.707 times

$$\text{Gain (dB)} = 20 \log \frac{V_{out}}{V_{in}}$$

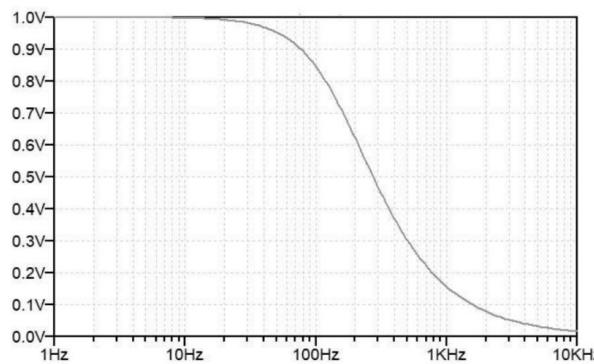
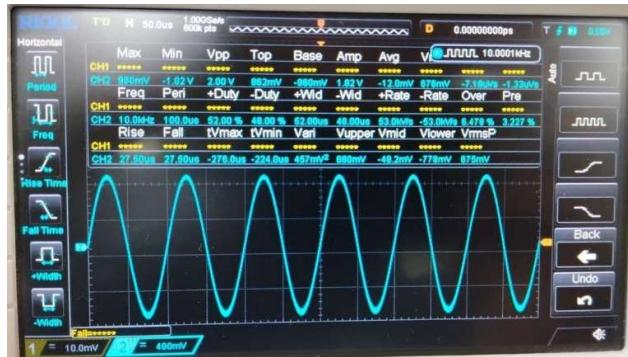
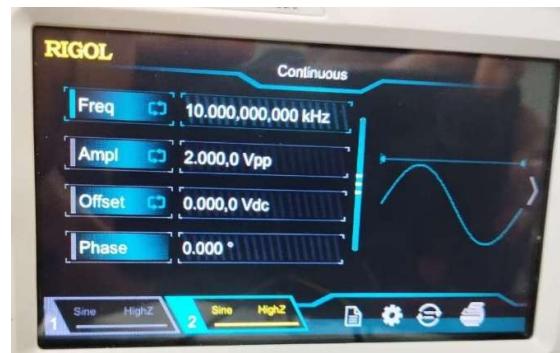
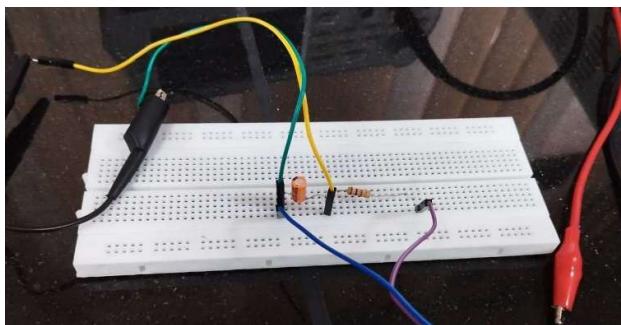


Figure 2: Frequency response of the low pass filter

the input voltage.

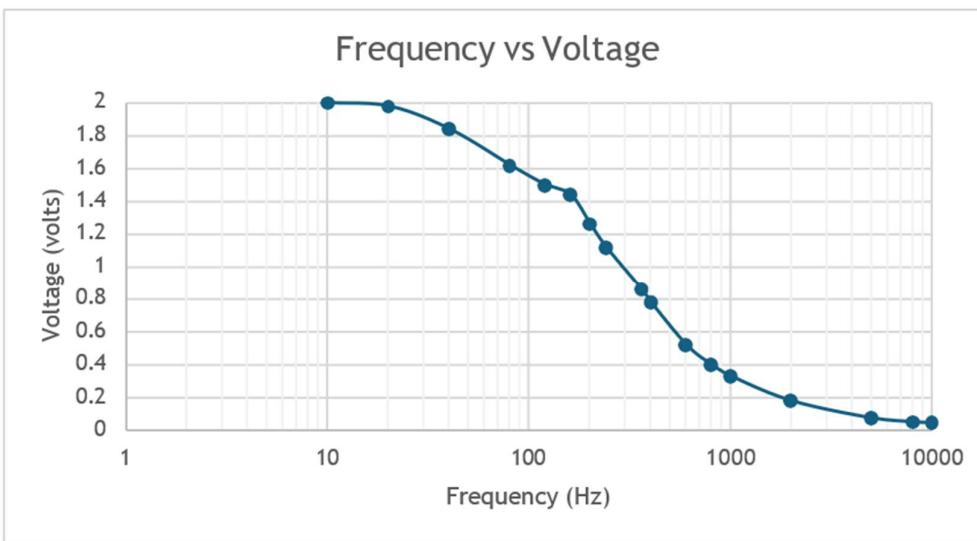
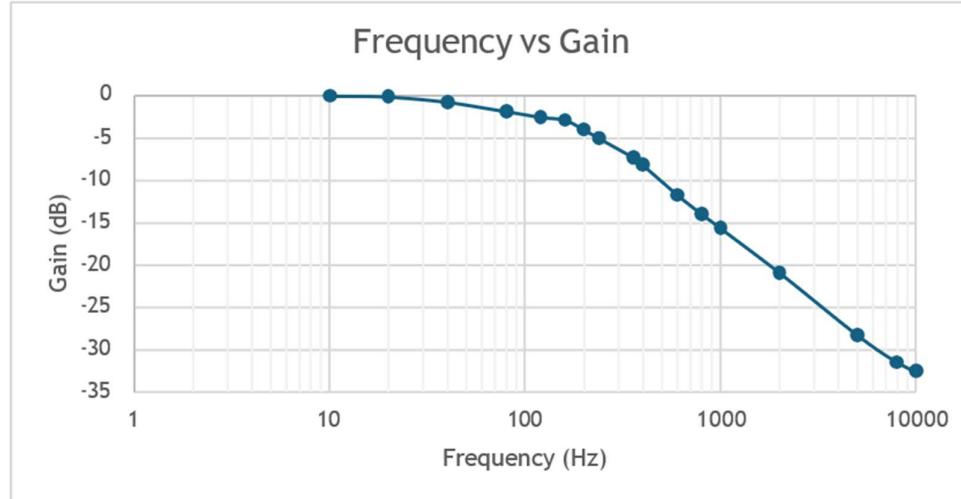
Equipment/Components Required:

Sl. No.	Component	Specification Values	Quantity
1.	Breadboard		1
2.	Signal generator		1
3.	Oscilloscope or multimeter		1
4.	Resistance	1 kΩ	1
5.	Capacitor	1 μF	1
6.	Connecting wires		As required

Results:

Across capacitor (low pass filter)

freq	Vout,pp	Gain (G)-dB
10 Hz	2.00 V	0
20 Hz	1.98 V	-0.087
40 Hz	1.84 V	-0.724
80 Hz	1.62 V	-1.830
120 Hz	1.50 V	-2.498
160 Hz	1.44 V	-2.853
200 Hz	1.26 V	-4.013
240 Hz	1.12 V	-5.036
360 Hz	0.86 V	-7.330
400 Hz	0.78 V	-8.178
600 Hz	0.52 V	-11.700
800 Hz	0.40 V	-13.979
1000 Hz	0.33 V	-15.650
2000 Hz	0.18 V	-20.915
5000 Hz	0.077 V	-28.29
8 KHz	0.053 V	-31.535
10 KHz	0.047 V	-32.578



Discussion:

For this RC circuit, we will first calculate the cut-off voltage, i.e. $f_c = 1/2\pi RC$. And for this RC combination, our cut-off frequency is 159.23 Hz. This means that for frequencies more than f_c , the output voltage across the Capacitor will attenuate. This is the basic functionality of low-pass filter, as it allows signals with frequencies lower than the cut-off frequency to pass through, while higher frequency signals face a barrier and are attenuated.

This can be clearly seen from the plot of Frequency vs Gain, for higher frequencies the gain is very less, while for frequencies less than f_c the gain is almost 0 dB, i.e. the output frequency almost resembles the input frequency.

Conclusion:

From the graphs plotted above, it can be seen clearly that the capacitor acts as a low-pass filter in an RC Circuit.

Objective 4.2:

Design a passive high-pass filter

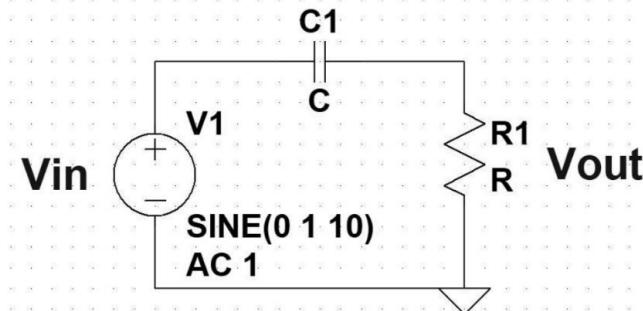
Schematic diagram:

Figure 3: Passive high-pass filter

Brief Description:

The RC high-pass filter allows frequency signals above the cut-off frequency f_c .

$$f_C = \frac{1}{2\pi RC}$$

The output voltage signal attenuates when the supply frequency is less than the cut-off frequency, as shown in Figure 4. At the cut-off frequency, capacitor C's capacitive reactance is equal to resistor R's resistance, causing the output voltage to be 0.707 times the input voltage.

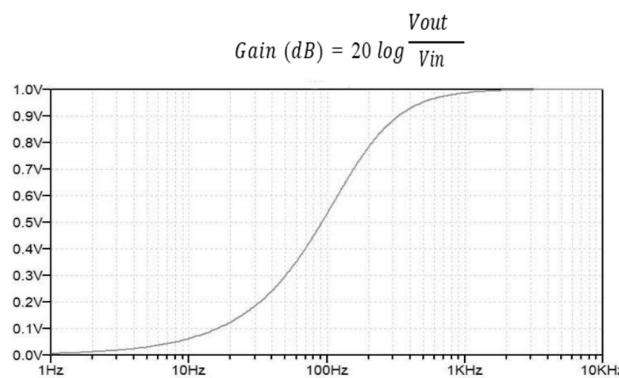


Figure 4: Frequency response of the high pass filter

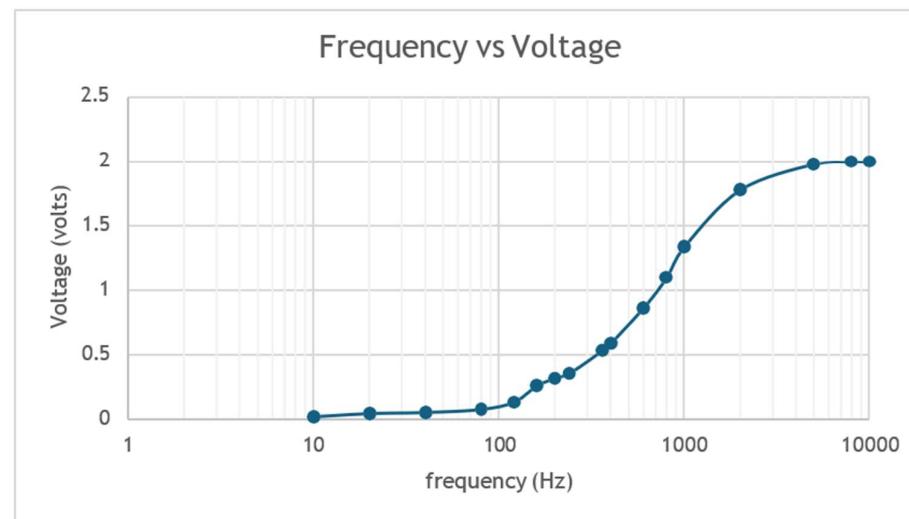
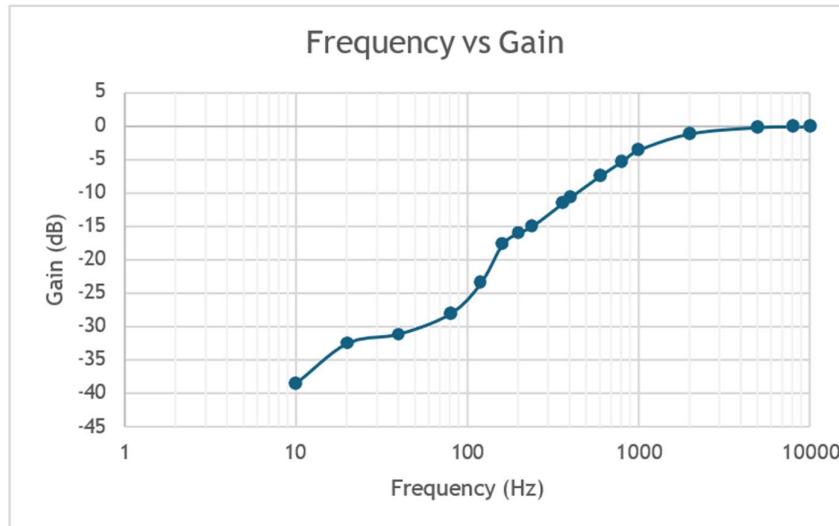
Equipment/Components Required:

Sl. No.	Component	Specification Values	Quantity
1.	Breadboard	-	1
2.	Signal generator	-	1
3.	Oscilloscope or multimeter	-	1
4.	Resistance	1 kΩ	1
5.	Capacitor	1 μF	1
6.	Connecting wires	-	As required

Results:

Across Resistor - (High pass filter)

<u>freq</u>	<u>Vout,pp</u>	<u>Gain (dB)</u>
10 Hz	0.024 V	-38.42
20 Hz	0.048 V	-32.39
40 Hz	0.056 V	-31.05
80 Hz	0.080 V	-27.95
120 Hz	0.136 V	-23.34
160 Hz	0.264 V	-14.59
200 Hz	0.320 V	-15.92
240 Hz	0.360 V	-14.89
360 Hz	0.536 V	-11.43
400 Hz	0.592 V	-10.54
600 Hz	0.864 V	-7.29
800 Hz	1.1 V	-5.19
1K Hz	1.34 V	-3.48
2K Hz	1.78 V	-1.01
5K Hz	1.98 V	-0.084
8K Hz	2.00 V	0
10K Hz	2.00 V	0



Discussion:

For this RC circuit, we will first calculate the cut-off voltage, i.e. $f_c = 1/2\pi RC$. And for this RC combination, our cut-off frequency is 159.23 Hz. This means that for frequencies less than f_c , the output voltage across the Resistor will attenuate. This is the basic functionality of a high-pass filter, as it allows signals with frequencies more than the cut-off frequency to pass through, while lower frequency signals face a barrier and are attenuated.

This can be clearly seen from the plot of Frequency vs Gain, for lower frequencies the gain is very less, while for frequencies more than f_c , the gain is almost 0 dB, i.e. the output frequency almost resembles the input frequency.

Conclusion:

From the graphs plotted above, it can be seen clearly that the Resistor acts as a high-pass filter in an RC Circuit.

LAB Report 5 ECS 327

Date: September 4, 2025

Name: Shlok Mehndiratta

Roll No: 23309

Title of Experiment:

Experiment 5: RLC Circuit

- **Objective 5.1:** For the given RLC circuit, apply input signals of different frequencies and vary the resistance values for a fixed value of L and C and measure output voltage $V_{out}(t)$.
- **Objective 5.2:** Measuring V_{out} for these combinations for observing underdamp overdamp and critically damped behavior after adjusting the appropriate values of RLC.

Objective 5.1:

For the given RLC circuit, apply input signals of different frequencies and vary the resistance values for a fixed value of L and C and measure output Current, $I(t)$.

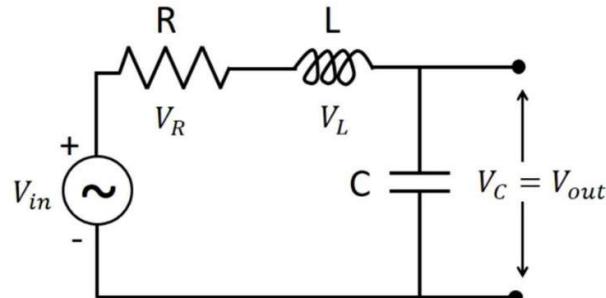
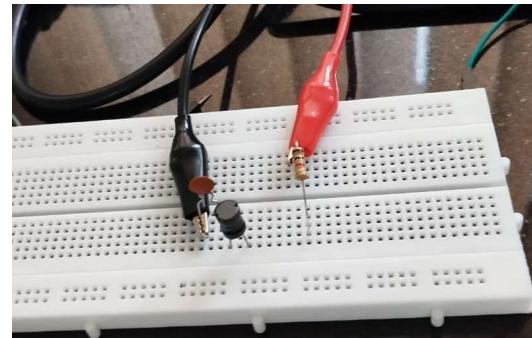
Schematic diagram:

Fig 3.1: Circuit diagram of the RLC experiment

**Brief Description:**

The series RLC circuit is composed of a resistor (R), a capacitor (C), and an inductor (L) connected in series as illustrated in Figure 3.1. In case of a pure ohmic resistor the voltage and current waveforms are in-phase with each other. In case of pure inductance surface the voltage waveform leads the current by 90° and for pure capacitance the voltage lags the current by 90° .

The mentioned phase difference between the voltage and current depends on the reactance (X) of circuit. For a purely resistive circuit element, $X = 0$; for purely inductive circuit element, $X > 0$; and for purely capacitive circuit element, $X < 0$.

The reactance and impedance of the circuit elements are,

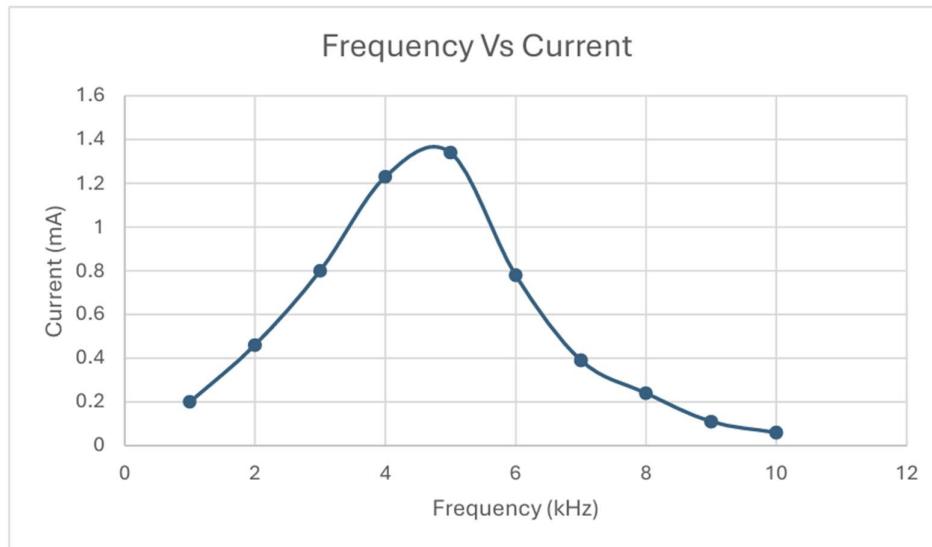
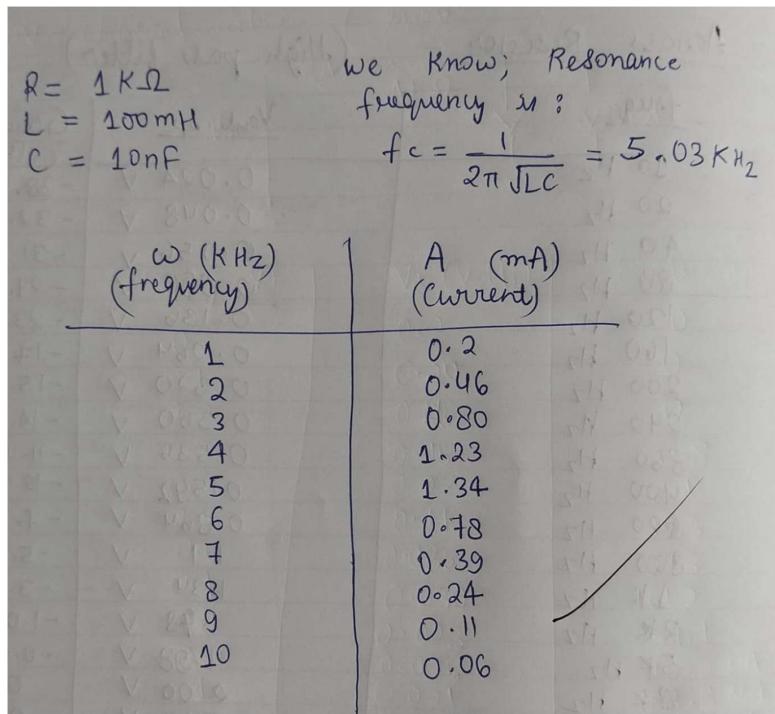
Element	Resistance	Reactance (X)	Impedance (Z)
Resistor	R	0	$Z = R$
Inductor	0	ωL	$z = j\omega L$
Capacitor	0	$\frac{1}{\omega C}$	$Z = \frac{1}{j\omega C}$

where ω is angular frequency

The RLC circuit in this case will help us to analyze each circuit element simultaneously. This circuit is similar to the series RL and RC circuits discussed previously, the difference in this case is that the reactance of both L and C will be counted together in overall reactance of the circuit. In the circuit under consideration consists a single loop with the same current flowing through all three circuit element. Since, the reactances of L and C are a function of frequency ($\omega = 2\pi f$, f is frequency), hence the sinusoidal response of the circuit will vary with input frequency, f. In this case, the individual voltage drop of each circuit element, R, L, and C will be out of phase with each other.

Equipment/Components Required:

S. No.	Component	Specification Values	Quantity
1.	AC power source/ Function Generator	0 – 10 kHz	1
2.	Resistance	1 kΩ	1
3.	Inductors	100 mH	1
4.	Capacitors	10 nF	1
5.	Digital Oscilloscope i.e. DSO	0 – 300 MHz	1
6.	Multimeter	Digital	1

Results:

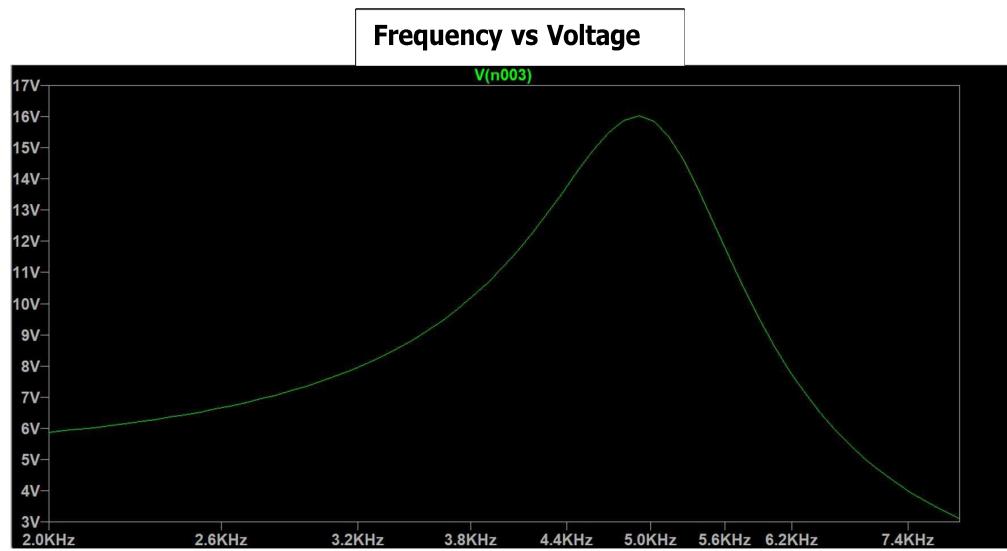
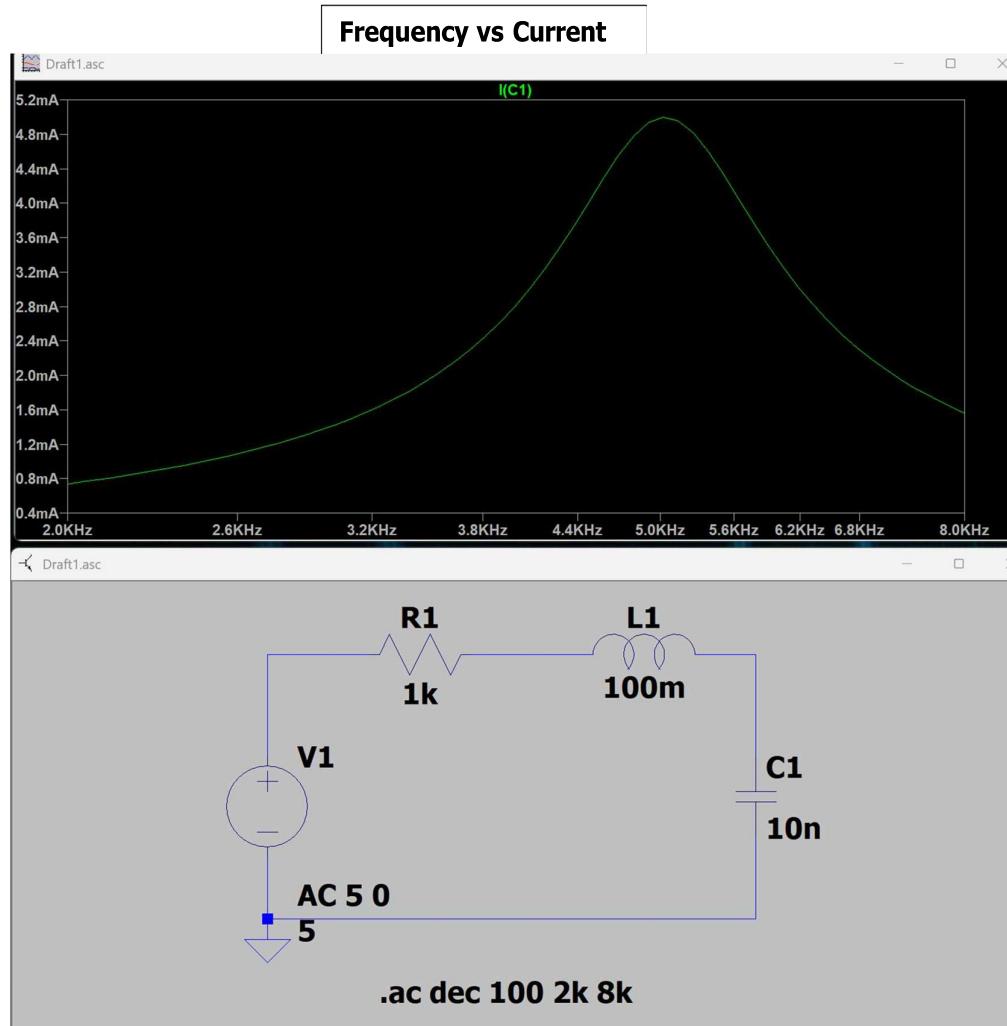
Discussion:

For this RLC circuit, we theoretically calculated the resonance frequency $f_0 = \frac{1}{2\pi\sqrt{LC}}$ (=5.03kHz).

We can verify that from the above frequency vs current graph too that resonance frequency ~ 5kHz. This means that for frequency ~5kHz, the current is maximum, and impedance is minimum in the circuit. The inductor and capacitor cancel each other's opposition, letting the circuit behave as if it were purely resistive, leading to maximum (series) current.

Conclusion:

Thus, we can conclude that- at resonance, energy sloshes back and forth between L and C with **no net reactive opposition**. And this RLC series circuit acts like a **band-pass filter** (passes resonance frequency strongly).

LT Spice:

Objective 5.2:

Measuring V_{out} for these combinations for observing underdamp, overdamp and critically damped behavior after adjusting the appropriate values of RLC.

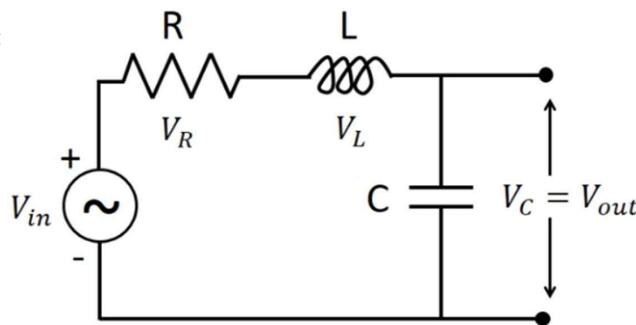
Schematic diagram:

Fig: 3.2: RLC circuit for the experiment.

Brief Description:

The basic discussion on the RLC circuit has been discussed in the previous experiment. If we consider KVL the, $VR + VL + VC = V(t)$ where VR , VL , and VC are the voltages across, R , L , and C , respectively and $V(t)$ is the input voltage signal. Substituting the values of the voltages across each electronic components we get,

$RI(t) + L \frac{dI(t)}{dt} + V_0 + \frac{1}{C} \int_0^t I(\tau) d\tau = V(t)$; for an unchanging voltage signal the same will reduce to,

$$\frac{d^2}{dt^2} I(t) + \frac{R}{L} \frac{d}{dt} I(t) + \frac{1}{LC} I(t) = 0$$

This can be written as the following form, $\frac{d^2}{dt^2} I(t) + 2\alpha \frac{d}{dt} I(t) + \omega_0^2 I(t) = 0$ Where $\alpha = \frac{R}{2L}$, is the attenuation or Neper frequency and $\omega_0 = \frac{1}{\sqrt{LC}}$, is the angular frequency.

The term α decides how fast the transient response will settle down. Depending on the electrical component values the circuit can be categorized as Overdamped, critically damped, and under damped. The following are the conditions for the same.

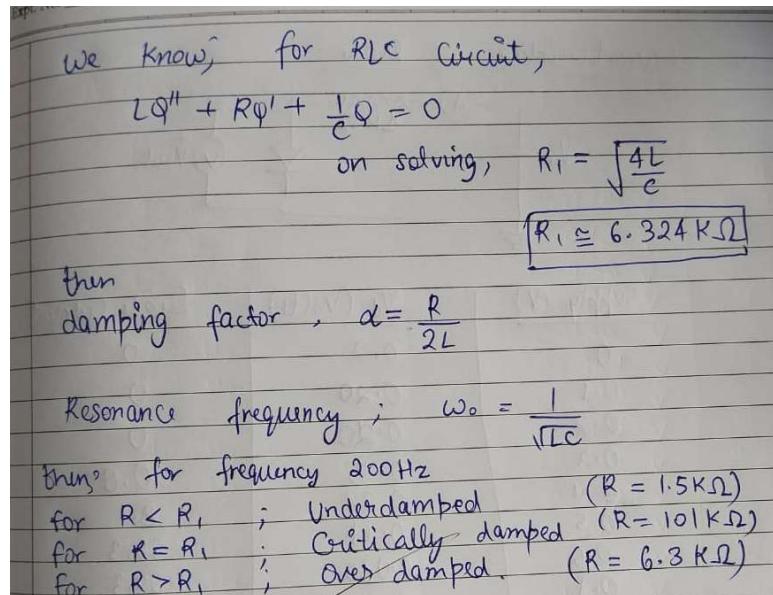
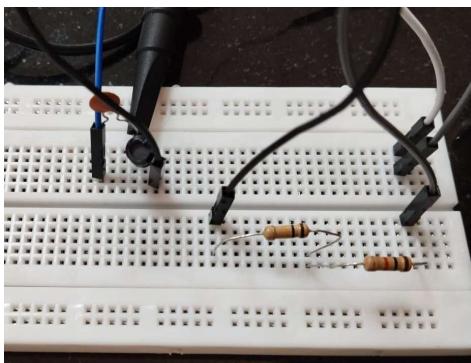
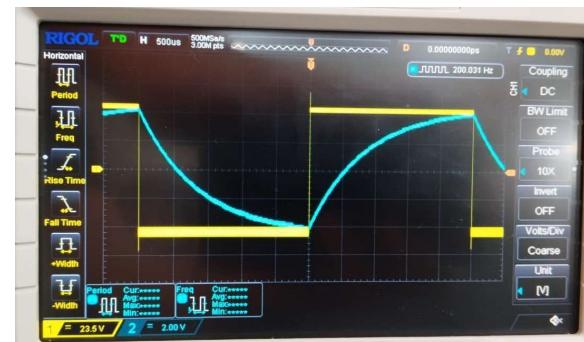
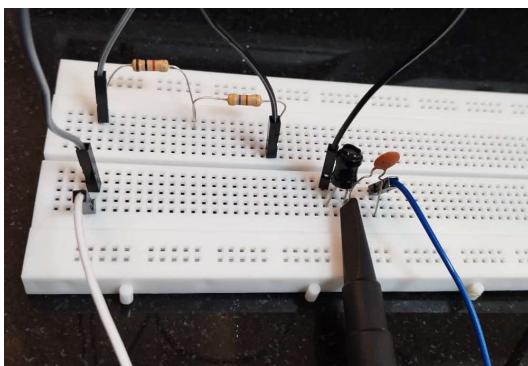
If $\alpha > \omega_0$ then it is overdamped

If $\alpha = \omega_0$ then it is critically damped

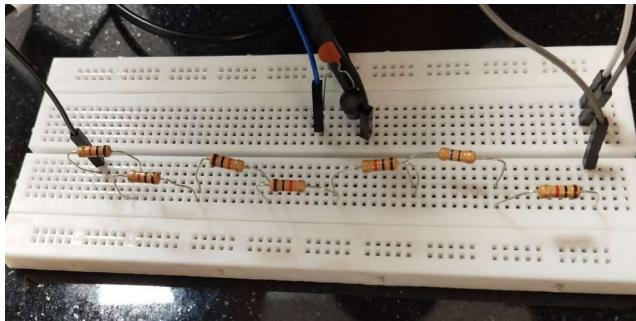
If $\alpha < \omega_0$ then it is underdamped

Equipment/Components Required:

Sl. no.	Component	Specification Values	Quantity
1.	AC power Source	-	1
2.	Resistance	1 k Ω , 100k Ω , 6.3 k Ω , 0.5 k Ω	1 each
3.	Inductors	100 mH	1
4.	Capacitors	10 nF	1
5.	DSO	-	1

Results:**1. Under-Damped****2. Critically damped**

3. Over-Damped



Discussion:

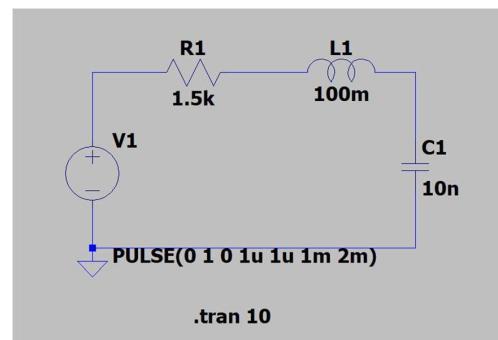
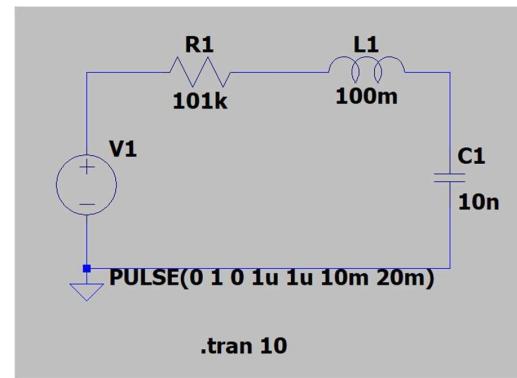
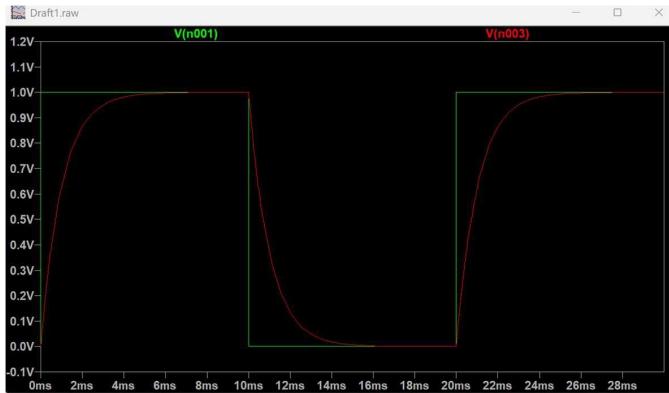
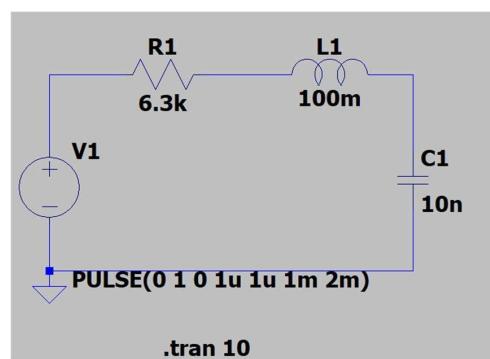
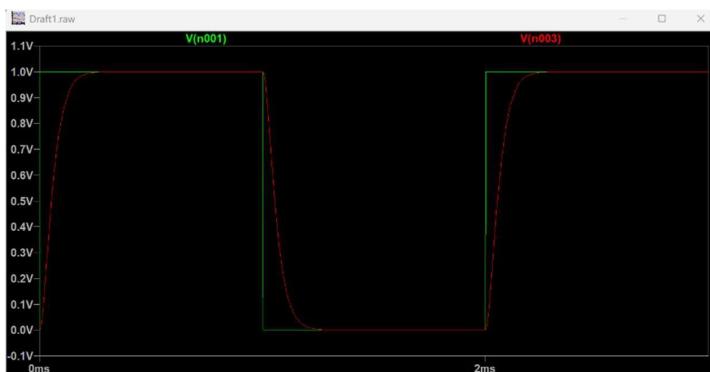
In this experiment, we studied the transient response of a series RLC circuit by observing the output voltage V_{out} across different combinations of R , L , and C .

By varying the resistor values while keeping L and C fixed ($L=100\text{ mH}$, $C=10\text{ nF}$), the transition between these damping conditions was observed on the DSO. The waveform confirmed the theoretical predictions:

- With **low resistance**, oscillations persisted before decay (underdamped).
- With **critical resistance**, the response returned to steady state quickly without oscillation.
- With **high resistance**, the response decayed slowly and non-oscillatory (overdamped).

Conclusion:

The experiment successfully demonstrated the effect of damping in a series RLC circuit. The transient response strongly depends on the relative magnitudes of resistance R , inductance L , and capacitance C .

LT-Spice Simulation -**1. Under-Damped****2. Critically Damped****3. Over Damped**

LAB Report 6 ECS 327

Date: September 12, 2025

Name: Shlok Mehndiratta

Roll No: 23309

Title of Experiment:

Experiment 6: Diode Behaviour

- **Objective 6.1:** I-V characteristics of a diode.
- **Objective 6.2:** Study of Full-wave and Half-wave rectifier circuit.
- **Objective 6.3:** Study of Clipper and clamper circuit.

Objective 6.1: I-V characteristics of a diode.

Schematic diagram:

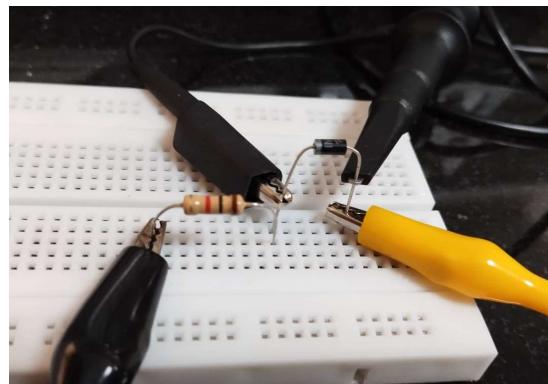
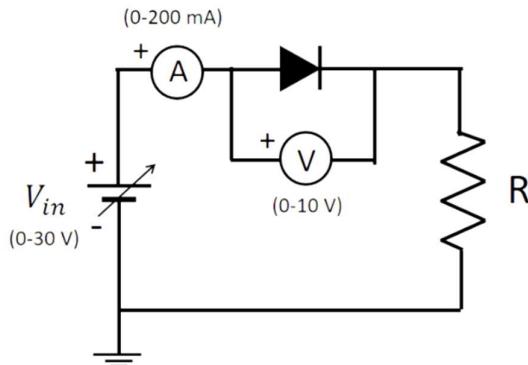


Fig. 4.1: circuit diagram of PN diode

Brief Description:

Diode is a unidirectional passive device and thus polarity of the power supply connected to a diode matters. An ideal diode is a perfect conductor for one direction of current whereas a perfect insulator for the opposite direction. However, in practice the diode does not behave as ideal conductor or insulator in any case. A junction diode is basically a semiconductor PN junction made of p-type and n-type semiconducting regions.

If the positive (negative) terminal of a voltage source is connected to P-side (N-side) of the diode then the diode is said to be in forward bias or ON, otherwise it is reverse biased or OFF. A real semiconductor diode made of Si needs approximately 0.7 V forward bias before it starts conducting current. In reverse bias, it conducts a negligible current which ideally should be zero. The forward current flows from p to n region.

The typical voltage – current (V-I) relationship in a real PN junction diode can be given by the following equation,

$$I = I_S \left(e^{\frac{V_D}{nV_T}} - 1 \right)$$

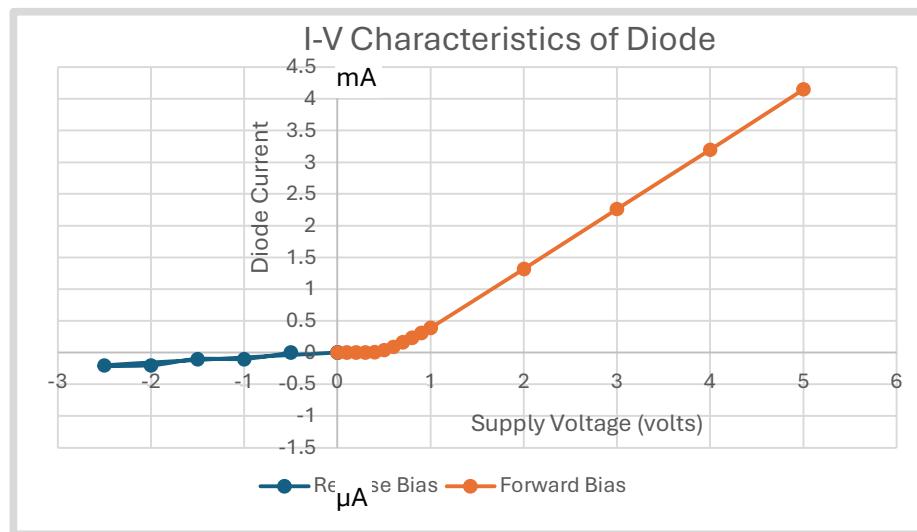
Where, I is diode current, I_S is the reverse bias saturation current or scale current, V_D is the applied voltage across the diode, V_T is the thermal voltage ($(kT)/q$, k – Boltzmann constant, T – Temperature in K, q – charge of electron), and n is the ideality factor.

Equipment/Components Required:

S. No.	Component	Specification Values	Quantity
1.	PN Diode	1N4007	1
2.	Multimeters	Digital Type	2
3.	Voltage Supply Source	DC regulating type	1
4.	Load Resistance	1 kΩ	1

Results:

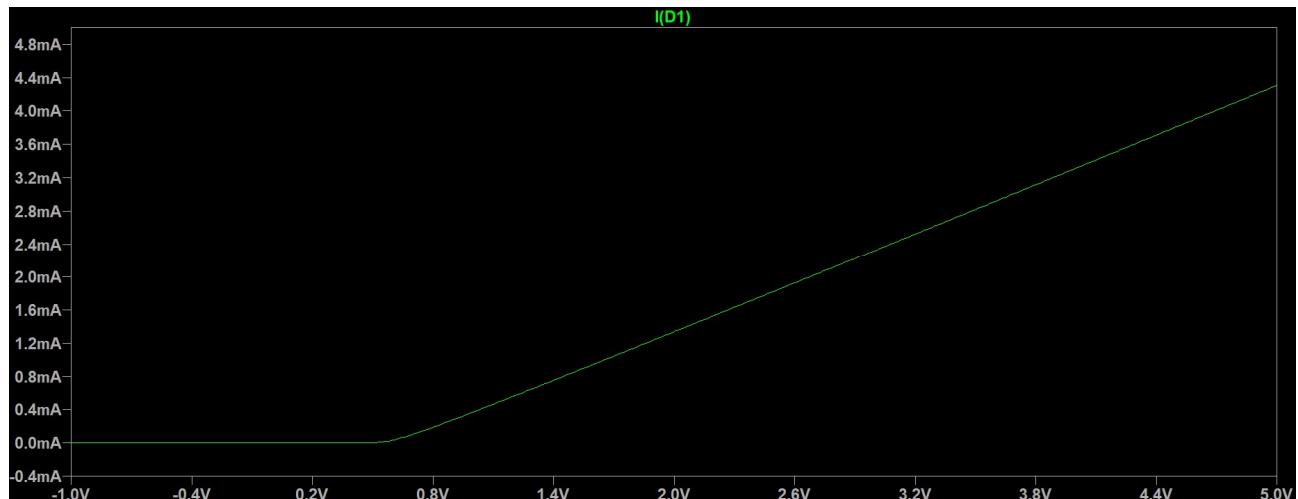
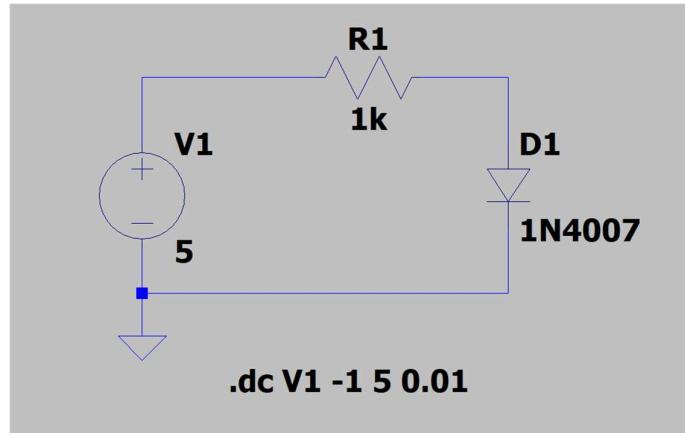
Supply Voltage (Volts)	Diode Voltage (Volts)	Diode Current
0	0.2	0
0.1	0.2	0
0.2	0.2	0
0.3	0.15	0.8 μ A
0.4	0.22	8.1 μ A
0.5	0.17	39.3 μ A
0.6	0.035	94.3 μ A
0.7	0.03	161.2 μ A
0.8	0.023	234.1 μ A
0.9	0.026	310.2 μ A
1	0.021	388.8 μ A
2	0.027	1.32 mA
3	0.026	2.26 mA
4	0.025	3.2 mA
5	0.026	4.15 mA
-0.5	0.024	0
-1	0.023	-0.1 μ A
-1.5	0.027	-0.1 μ A
-2	0.026	-0.2 μ A
-2.5	0.022	-0.2 μ A

**Discussion:**

The V-I characteristics of the PN junction diode clearly show that in forward bias the current remains very small until the cut-in voltage (~ 0.7 V for Si) is reached, after which it increases rapidly with voltage. In reverse bias, the current stays nearly constant and very small (in μ A range), representing the reverse saturation current. This verifies the rectifying behavior of the diode. Minor deviations from ideal behavior are due to leakage currents and practical limitations.

Conclusion:

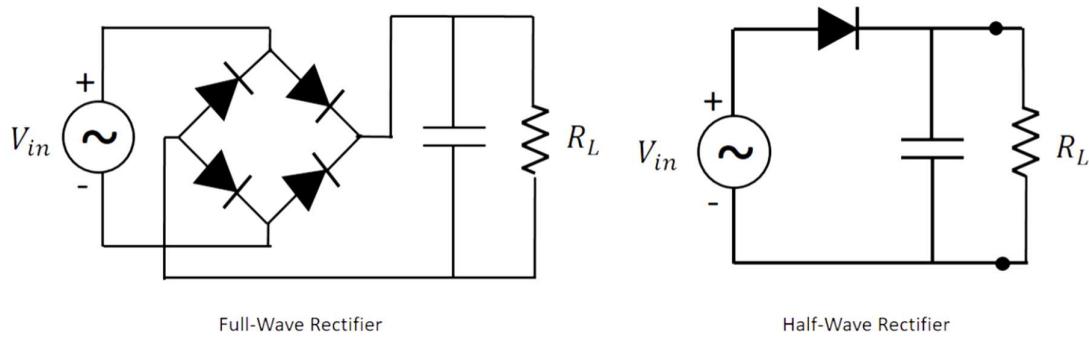
The experiment confirms that a diode conducts significantly only after its threshold voltage in forward bias and allows negligible current in reverse bias, thereby acting as a rectifier.

LT Spice Simulations :

I-V Characteristic graph for a diode

Objective 6.2: Study of Full-wave and Half-wave rectifier circuit

Schematic diagram:



Full-Wave Rectifier

Half-Wave Rectifier

Fig 4.2: Circuit Diagram of Full-Wave and Half Wave Rectifier

Brief Description:

Rectifiers are the electrical circuits that converts the alternating currents (i.e. the currents that changes its direction with time) to a direct current (i.e. the current that has only one direction) by either removing a section of the signal with one polarity or converting the polarity of the signal to one. Most used two types of rectifiers are the half-wave and full wave rectifiers. The associated circuits are given in Figure 4.2.

In case of half-wave rectifier, the diode conducts current only during the positive cycle of the sinusoidal input and blocks the negative cycle of the input. Hence, the output of the rectifier is only the positive cycle of the input signal. Thus, it is called half-wave rectifier as it only rectifies the half part of the full signal.

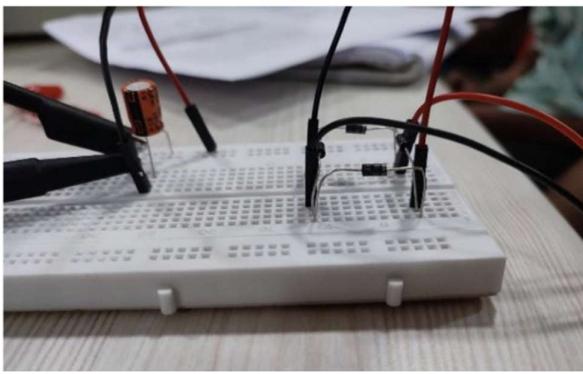
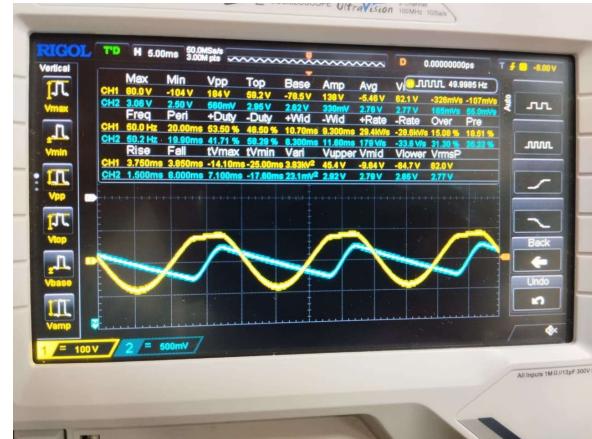
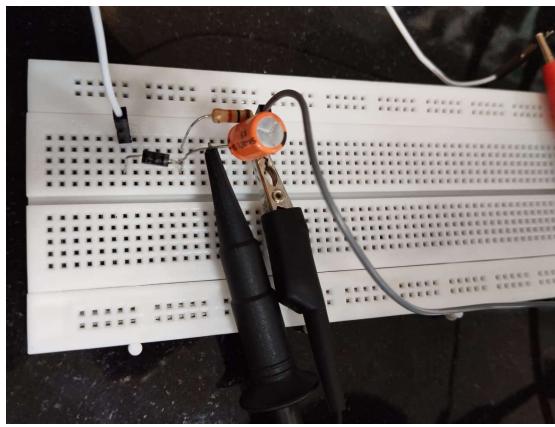
In case of full-wave rectifier, the 4 diodes are arranged in such a way that during positive cycle two of the 4 diodes conduct and during negative cycle the other two of the 4 diodes will conduct. In both the cases, the current follows different paths and reflect as a positive signal at the output.

Equipment/Components Required:

Sl. No.	Component	Specification Values	Quantity
1.	AC source	Function Generator	1
2.	PN Diodes	1N4007 or Similar	4
3.	Capacitors	100 μ F	1
4.	Resistors	1 k Ω	1
5.	Digital Oscilloscope (DSO)	0-300 MHz or Higher	1

Precautions:

1. Connections should be verified before clicking run button.
2. The resistance to be chosen should be in k Ω range.
3. Best performance is being obtained within 50 Hz to 1 MHz

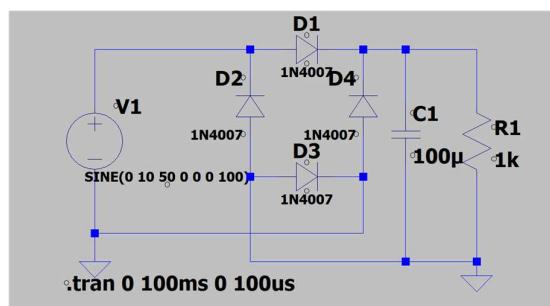
Results:**1. Full-wave Rectifier****2. Half-Wave Rectifier****Discussion:**

The experimental study of half-wave and full-wave rectifiers demonstrates the role of diodes in converting AC to DC. In the half-wave rectifier, the diode allows conduction only during the positive half-cycle of the input sinusoidal signal, blocking the negative half-cycle. As a result, the output waveform consists of only positive cycles, but with significant ripples and lower efficiency.

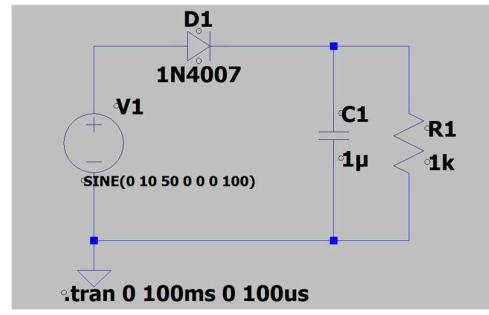
In the full-wave rectifier, four diodes are connected in a bridge configuration so that during both positive and negative half-cycles of the input, a unidirectional current flows through the load. This doubles the output frequency compared to the input and produces a smoother DC output than the half-wave rectifier. The use of a filter capacitor further reduces the ripple and brings the output closer to pure DC.

Conclusion:

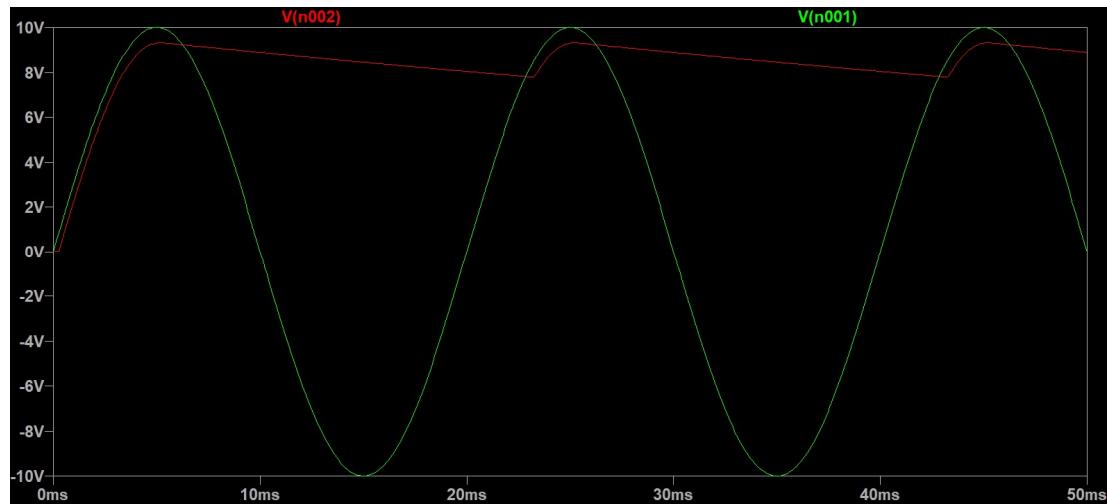
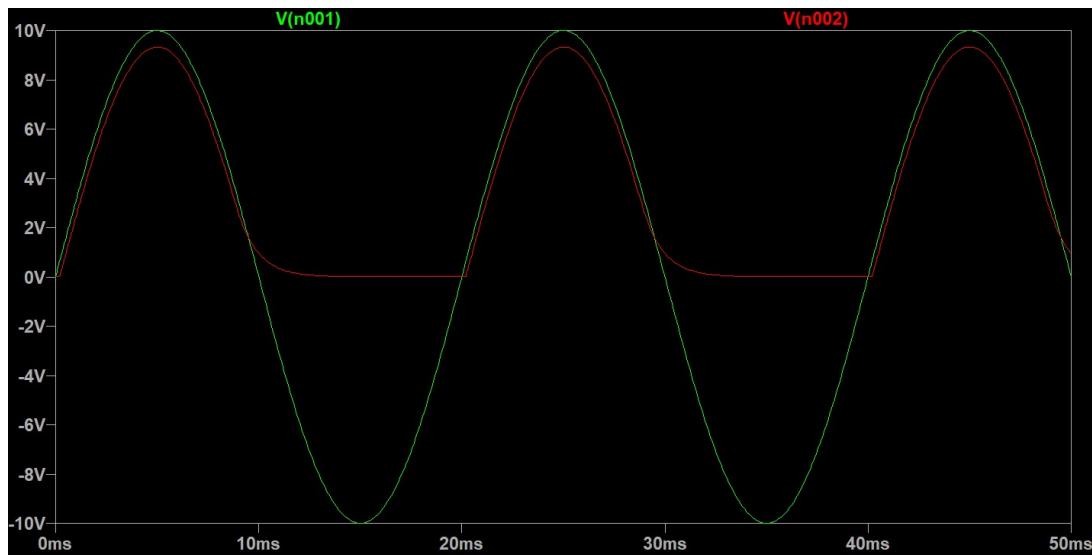
The experiment shows that a half-wave rectifier gives low efficiency with high ripple, while a full-wave rectifier provides higher efficiency and less ripple. Adding a capacitor filter further smoothes the DC output, making full-wave rectifiers more suitable for practical power supplies.

LT-Spice Simulations -

Full Wave Rectifier



Half Wave Rectifier

1. Full-wave Rectifier**2. Half-Wave Rectifier**

Objective 6.3: Study of Clipper and clamper circuit.

Schematic diagram:

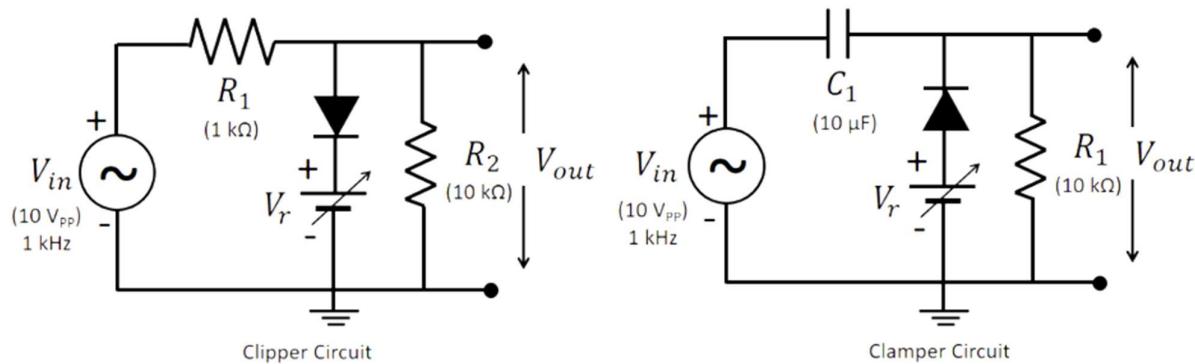


Fig 4.3: Clipper and Clamper circuit

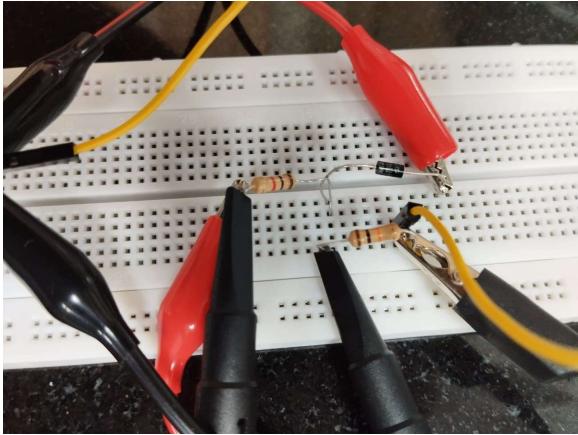
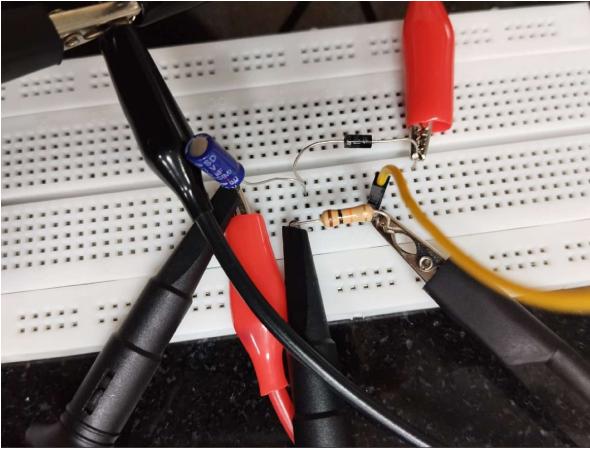
Brief Description:

Clipper Circuit: Clipping circuits are used to remove a part of a signal which is above or below a reference level. Clipping circuits are also known as limiters, amplitude selectors, or slicers. Half-wave rectifier is also a good basic example of clipper circuit where the reference level is zero and the signal below zero voltage (i.e. negative) are not allowed to pass through. To alter the reference level to a desired value, a DC voltage source is put in series with the diode. Depending on the polarity of DC source and direction of diode the circuit will clip the input signal above or below the reference level set by the user. The circuit is given in Figure 4.3.

Clamping Circuits: A clamper circuit, on the other hand, shifts a signal to a defined value. Basically, this circuit adds a DC component to the input signal. The circuit can work with a bias or no-bias condition. If the signal shifts above the central line of a input wave, then it calls a positive clamper circuit and if it shifts downwards, then it is called a negative clamper circuit. The circuit arrangement of a diode clamper circuit is given in Figure 4.3.

Equipment/Components Required:

Sl. No.	Component	Specification Values	Quantity
1.	Function Generator	0-3 MHz or Higher	1
2.	Oscilloscope	0-300 MHz or Higher	1
3.	DC power supply	0-12 V DC	2
4.	Resistor	1, 10, 100, 330 kΩ	4
5.	Diode	1N4007 or Similar	4

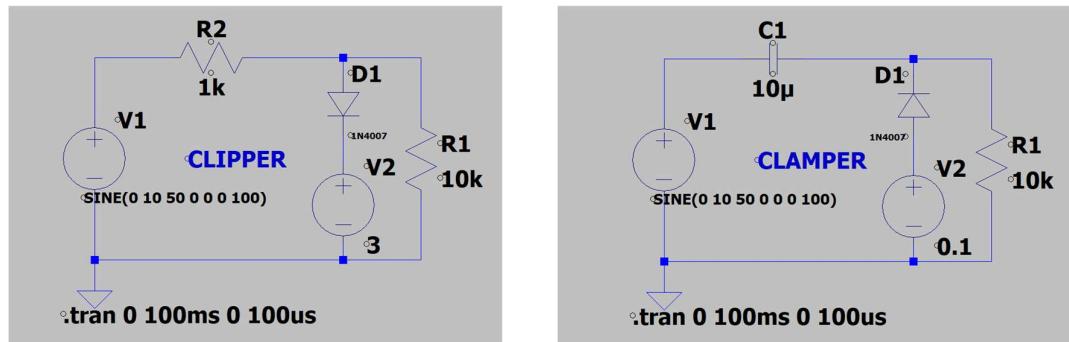
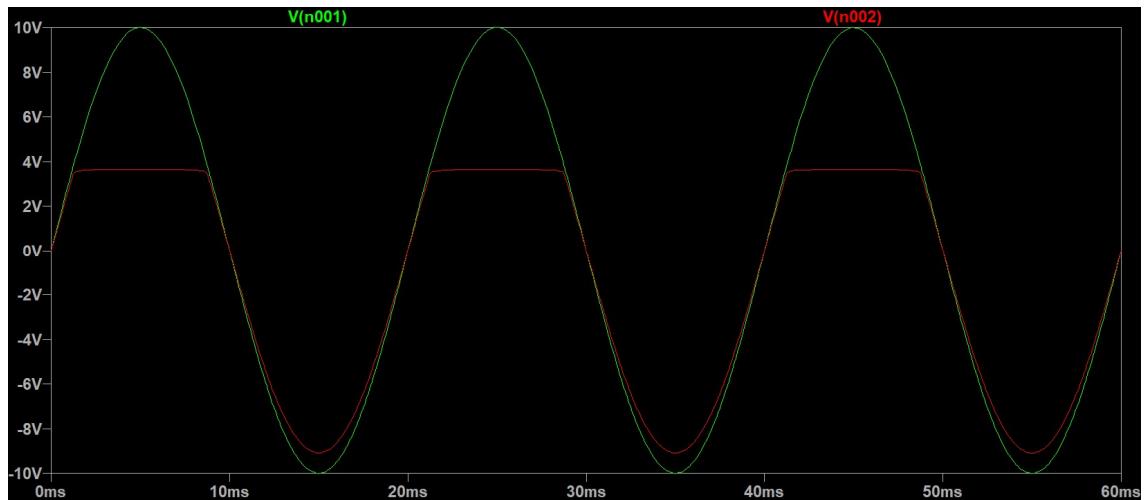
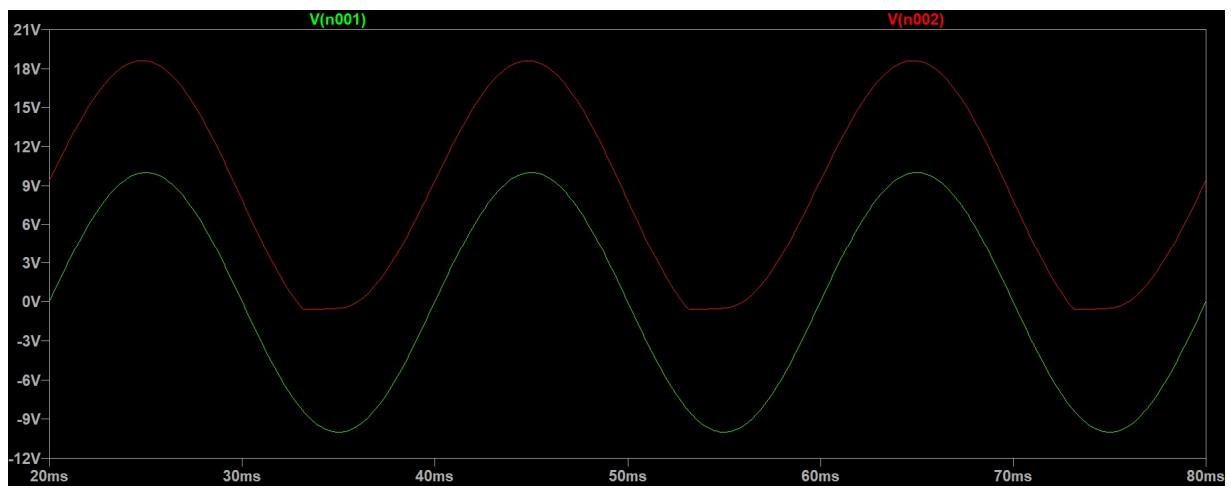
Results:**1. Clipper Circuit****2. Clamper Circuit****Discussion:**

The clipper circuit restricts the output voltage by removing parts of the input signal that exceed a certain reference level. By adjusting the DC bias and diode orientation, the circuit can clip either the positive or negative peaks. This demonstrates how clippers are useful in shaping signals and protecting circuits from over-voltage.

The clamper circuit, instead of cutting off peaks, shifts the entire signal either upward or downward by adding a DC level. Depending on diode polarity and bias, it can be configured as a positive or negative clamper. This shows how clampers can shift signal baselines while preserving shape and amplitude.

Conclusion:

Thus, clippers are mainly used for signal limiting and protection, while clampers are applied for level shifting in communication and electronic systems.

LT-Spice Simulation –**1. Clipper****2. Clamper**

Experiment – 6

Assignment Answers

1. Explain any of the breakdown mechanisms of diodes.

When a very high voltage is applied to a diode in the reverse-biased direction, it can fail and conduct a large current. This is called diode breakdown. One of the primary mechanisms for this is **Avalanche Breakdown**.

Avalanche Breakdown: This type of breakdown occurs in diodes that have a relatively wide depletion region. The process happens in a chain reaction:

1. Acceleration: The high reverse-bias voltage creates a strong electric field across the depletion region. This field accelerates the few minority charge carriers (electrons and holes) to extremely high speeds, giving them significant kinetic energy.
2. Collision: These high-speed carriers collide with stationary atoms within the semiconductor crystal lattice.
3. Carrier Generation: The collision is so forceful that it knocks valence electrons free from the atoms, creating new electron-hole pairs.
4. Chain Reaction: These newly created carriers are also accelerated by the electric field, leading to more collisions and creating even more electron-hole pairs.

This rapid multiplication of charge carriers is like a snowball causing an avalanche, leading to a sudden and massive increase in reverse current that effectively "breaks down" the diode's resistance.

A second mechanism, primarily seen in heavily doped diodes at lower voltages (<5V), is Zener Breakdown, where the electric field becomes so strong it can directly pull electrons across the narrow depletion region in a process called quantum tunneling.

2. Calculate V_{rms} of the half-wave and full-wave rectifier, and compare your results.

The RMS voltage is the "effective" value of a waveform. It's calculated based on the peak voltage (V_p) of the AC input.

Half-Wave Rectifier (HWR)

For a half-wave rectified signal, where one half of the AC sine wave is blocked, the RMS voltage is half of the peak voltage.

$$V_{rms}(FWR) = \frac{V_p}{2} \approx 0.5 V_p$$

Full-Wave Rectifier (FWR)

For a full-wave rectified signal, where both halves of the AC sine wave are used, the RMS voltage is the same as that of a standard AC sine wave

$$V_{rms}(FWR) = \frac{V_p}{\sqrt{2}} \approx 0.707 V_p$$

The RMS voltage of a full-wave rectifier is higher than that of a half-wave rectifier by The factor of 2 (approximately 1.414).

This is because the full-wave rectifier utilizes both halves of the AC cycle, while the half-wave rectifier discards one half. Since power delivered to a resistor is proportional to the square of the RMS voltage ($P=V_{rms}^2/R$), a full-wave rectifier delivers exactly twice the average power to a load compared to a half-wave rectifier for the same input signal. This makes it far more efficient for power supply applications

3. How do the capacitor and diode work together to shift the DC level of an AC input signal?

This combination of a capacitor, diode, and resistor forms a Clamper circuit, which is designed to shift the entire AC signal up or down so that its positive or negative peak is "clamped" to a specific DC level (usually 0V).

Here's how it works, using a positive clamper as an example (clamps the negative peak to 0V):

1. Diode as a Switch: The diode acts as a one-way switch. In a positive clamper, it is oriented to conduct only when the input signal's voltage goes negative.
2. Capacitor Charging: During the very first negative half-cycle of the AC input, the diode turns ON. This provides a path for the capacitor to quickly charge up to the peak voltage of the input (V_p). The capacitor holds this charge, storing a DC voltage equal to V_p .
3. Diode Turns Off: After the first negative peak, the diode turns OFF because the voltage stored in the capacitor prevents it from becoming forward-biased again. It now acts as an open switch.
4. DC Level Shift: The charged capacitor now acts like a small DC battery with voltage V_p placed in series with the AC input signal. The output voltage (V_{out}) is the sum of the input voltage and the capacitor's voltage ($V_{out}=V_{in}+V_C$).

Because the capacitor adds a constant DC voltage ($V_C=V_p$) to the time-varying input (V_{in}), the entire signal is shifted vertically. An AC signal that originally swung from $+V_p$ to $-V_p$ is now shifted up to swing from $+2V_p$ to $0V$.

LAB Report 7 ECS 327

Date: October 09, 2025

Name: Shlok Mehndiratta

Roll No: 23309

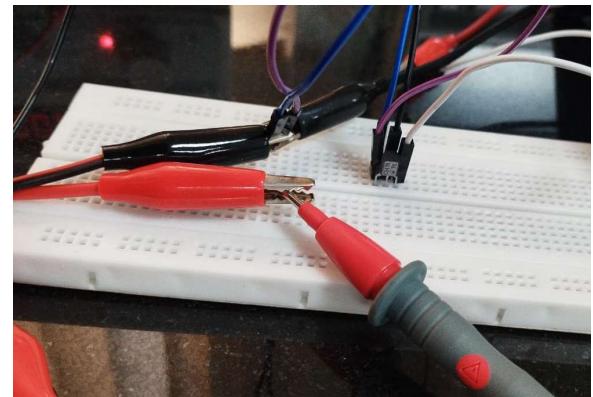
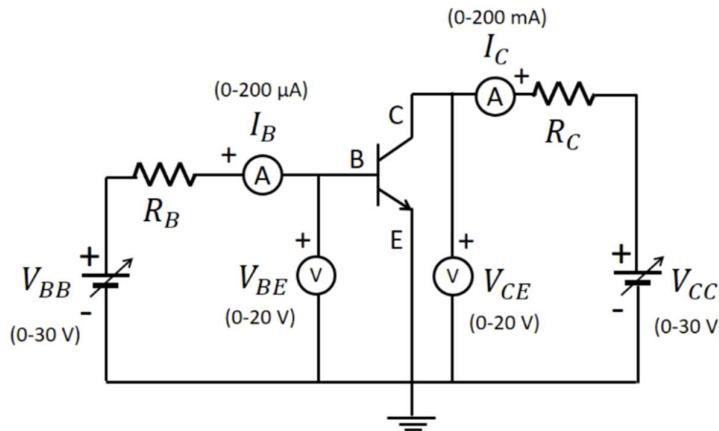
Title of Experiment:

Experiment 7: Bipolar Junction Transistor (BJT) analysis

- **Objective 7.1:** Study of common emitter configuration (I_C vs V_{CE} for different V_{BE}).

Objective 7.1: Study of Common emitter configuration (I_C vs V_{CE} for different V_{BE})

Schematic diagram:



Brief Description:

Bipolar Junction Transistor (BJT) has three terminals namely, emitter (E), base (B), and collector (C). A BJT is composed of two PN junctions and the operation of BJT is mainly based on the PN junction characteristics. In case of a npn transistor in active region, under forward biased emitter-base (EB) junction, the majority carrier electrons in n-type emitter region are injected to thin p-type base region where the electrons as minority carrier diffuse towards the collector through the reverse biased collector-base (CB) junction. Some of the electrons recombine with holes in the thin p-type base region to produce a small base current (I_B) and the remaining reach collector as a collector current (I_C). Hence, if there is no current from emitter (I_E), then there will be almost no I_C . Combining all the currents, the total emitter current, $I_E = I_B + I_C$.

In case of pnp transistor, the polarity of voltage sources must be reversed. Depending on the biasing of two junctions (i.e. EB and CB) transistor, the transistor can be said to be in different modes of operation.

Operating Region	EB Junction	CB Junction	Remark
Cut-off	Reverse	Reverse	$I_E \approx I_B \approx I_C \approx 0$, Off-state, $V_{BE} < 0.7 V$
Active	Forward	Reverse	Amplifier gain (100-1000)
Saturation	Forward	Forward	Conducting
Reverse Saturation	Reverse	Forward	Reverse gain

Output characteristics are obtained between the output voltage and output current at constant input current. It is plotted between V_{CE} and I_C at constant I_B in CE configuration.

Equipment/Components Required:

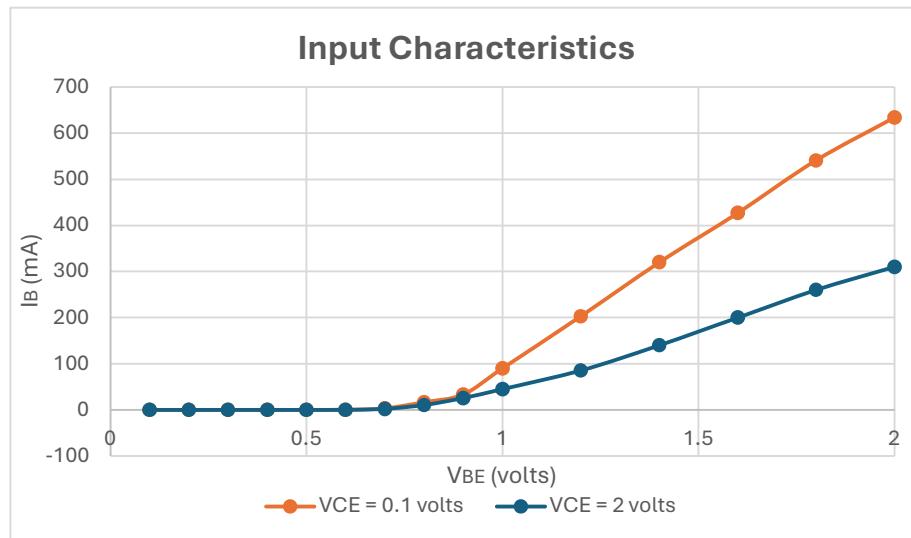
S. No.	Component	Specification Values	Quantity
1.	DC power Supply	0-12 V	2
2.	NPN transistor	BC 547	2
3.	Resistors	1 kΩ, 100 kΩ	4
4.	Multimeter	Digital	3
5.	Connecting Wires	Male - Male	10

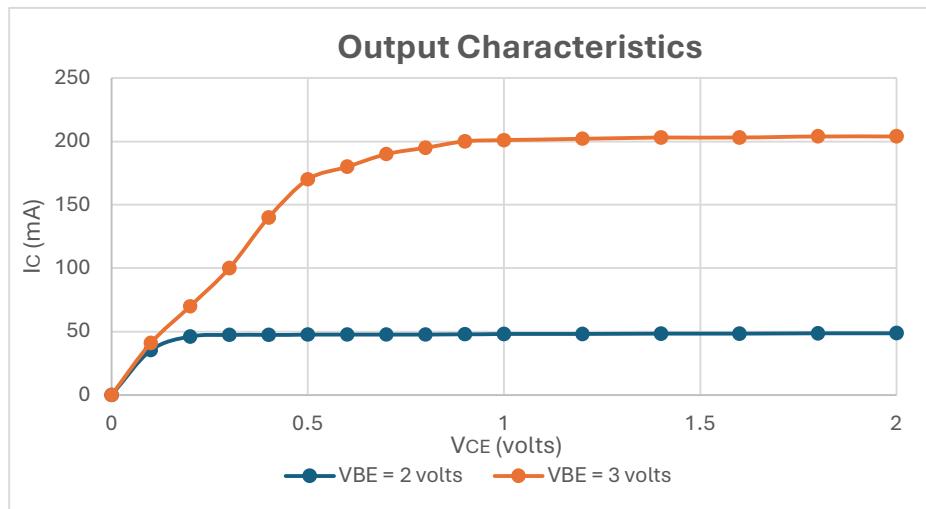
Results:**Input Characteristics:**

V _{BE} (volts)	V _{CE} = 0.1 volts	V _{CE} = 2 volts
	I _B (mA)	I _B (mA)
0.1	0.0104	0
0.2	0.024	0
0.3	0.0504	0
0.4	0.0693	0.002
0.5	0.13	0.01
0.6	0.29	0.12
0.7	3.28	2.2
0.8	16.54	10.5
0.9	33.35	25.5
1	90	45
1.2	203	85.5
1.4	320	140
1.6	427	200
1.8	541	260
2	634	310

Output Characteristics:

V _{CE} (volts)	V _{BE} = 2 volts	V _{BE} = 3 volts
	I _C (mA)	I _C (mA)
0	0	0
0.1	35.25	41
0.2	46	70
0.3	47.3	100
0.4	47.4	140
0.5	47.5	170
0.6	47.6	180
0.7	47.7	190
0.8	47.7	195
0.9	47.9	200
1	48	201
1.2	48.1	202
1.4	48.3	203
1.6	48.4	203
1.8	48.6	204
2	48.7	204



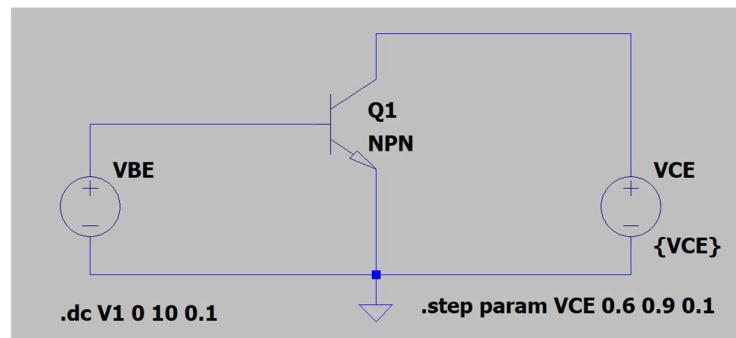
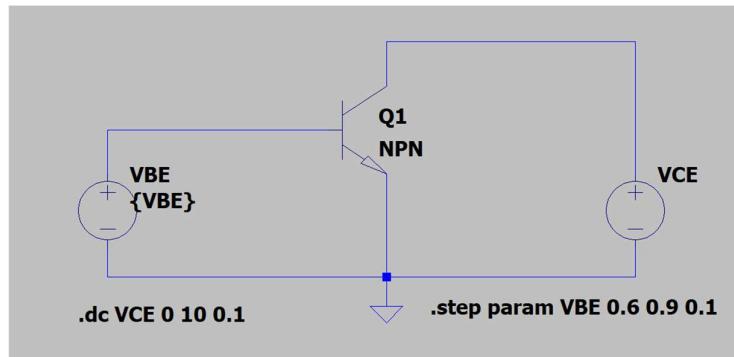


Discussion:

The input and output characteristics of an NPN transistor in common emitter configuration were studied. The input curve (I_B vs V_{BE}) shows diode-like behaviour — base current remains negligible until $V_{BE} \approx 0.7$ V, after which it increases rapidly, confirming forward biasing of the base-emitter junction. The output characteristic (I_C vs V_{CE}) shows that at low V_{CE} the transistor is in saturation, while at higher V_{CE} the collector current becomes nearly constant for each fixed V_{BE} , indicating active region operation. These results match the expected theoretical behavior of a BJT.

Conclusion:

The experiment confirms that in a common emitter configuration, the transistor exhibits three regions of operation — cut-off, active, and saturation. The collector current mainly depends on the base current in the active region, demonstrating the transistor's current amplification property. The observed characteristics validate the typical behavior of a silicon NPN transistor.

LT-Spice Simulation –**1. Input Characteristics:****2. Output Characteristics:**

LAB Report 8 ECS 327

Date: October 16, 2025

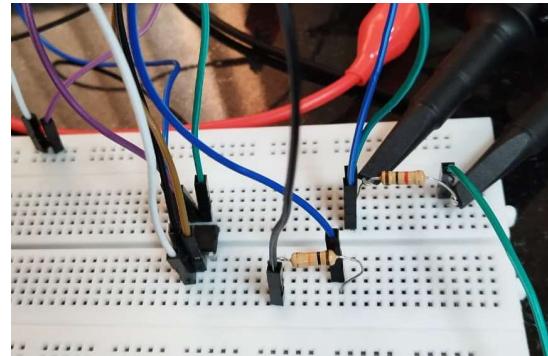
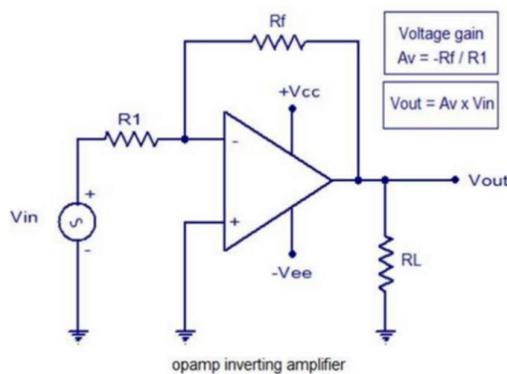
Name: Shlok Mehndiratta

Roll No: 23309

Title of Experiment:

Experiment 8: Familiarization with Op-Amp circuits

- **Objective 8.1:** Design and realize Inverting amplifier using IC741Op-amp.
- **Objective 8.2:** To study the working of op-amp as differentiator.
- **Objective 8.3:** To study the working of op-amp as integrator.

Objective 8.1: Design and realize Inverting amplifier using IC741Op-amp.**Schematic diagram:****Brief Description:**

An inverting amplifier utilizing an operational amplifier is a configuration in which the output waveform is phase-inverted relative to the input waveform. The input waveform will be amplified by the factor A_v (voltage gain of the amplifier) in magnitude, and its phase will be inverted. The signal intended for amplification is introduced to the inverting input of the operational amplifier via the input resistor R_1 . R_f denotes the feedback resistor. R_f and R_1 collectively dictate the amplifier's gain. The gain of an inverting operational amplifier can be articulated using the equation

$$A_v = -\frac{R_f}{R_1}$$

Negative sign implies that the output signal is negated.

where

A_v is voltage gain

R_f is the feedback resistor value

R_1 is the input resistor value

There are two very important rules to remember about Inverting Amplifiers or any operational amplifier for that matter and these are.

- No Current Flows into the Input Terminals
- The Differential Input Voltage is Zero as $V_1 - V_2 = 0$ (Virtual Earth)

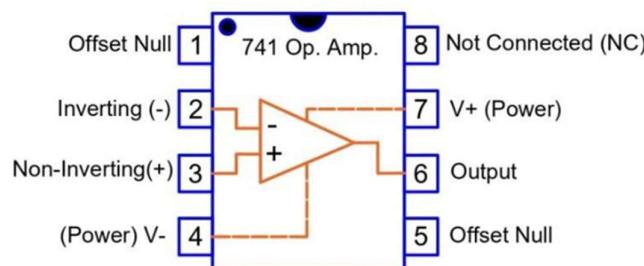


Fig.1.2: schematic of 741 OP-AMP

$$i = \frac{V_{in} - V_{out}}{R_{in} + R_f}$$

therefore, $i = \frac{V_{in} - V_2}{R_{in}} = \frac{V_2 - V_{out}}{R_f}$

$$i = \frac{V_{in}}{R_{in}} - \frac{V_2}{R_{in}} = \frac{V_2}{R_f} - \frac{V_{out}}{R_f}$$

so, $\frac{V_{in}}{R_{in}} = V_2 \left[\frac{1}{R_{in}} + \frac{1}{R_f} \right] - \frac{V_{out}}{R_f}$

and as, $i = \frac{V_{in} - 0}{R_{in}} = \frac{0 - V_{out}}{R_f} \quad \frac{R_f}{R_{in}} = \frac{0 - V_{out}}{V_{in} - 0}$

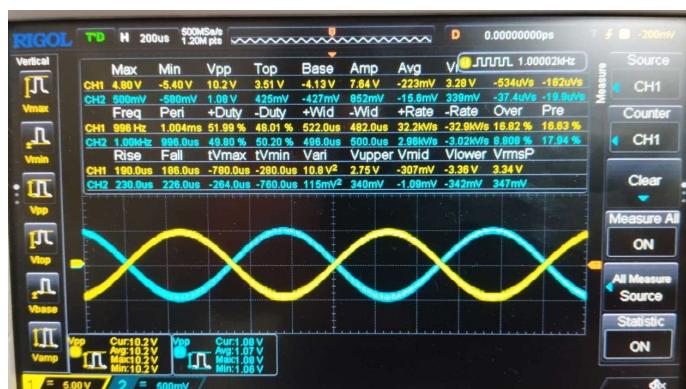
the Closed Loop Gain (A_v) is given as, $\frac{V_{out}}{V_{in}} = -\frac{R_f}{R_{in}}$

Equipment/Components Required:

S. No.	Component	Specification Values	Quantity
1.	Dual power supply for supply of 741 IC	0-12 V	1
2.	AFM (arbitrary function generator)	Digital	1
3.	Resistors	1 kΩ, 10 kΩ	2
4.	Op Amp	IC-741	1
5.	DSO	-	1

Results:

S. No.	V _{in}	V _{out}	R _{in} (kΩ)	R _f (kΩ)	A _v
1.	Sine wave	Inverted Sine wave	1	10	9.44
2.	Triangular wave	Inverted Triangular	1	10	9.44
3.	Square Wave	Inverted Square	1	10	7.28
4.	DC	DC	1	10	-



Input: Sine Wave



Input: Triangular Wave



Input: Square Wave



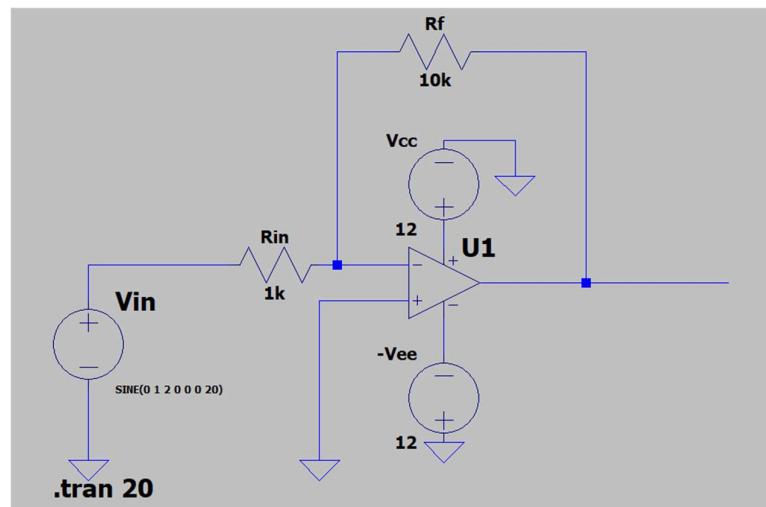
Input: DC Voltage

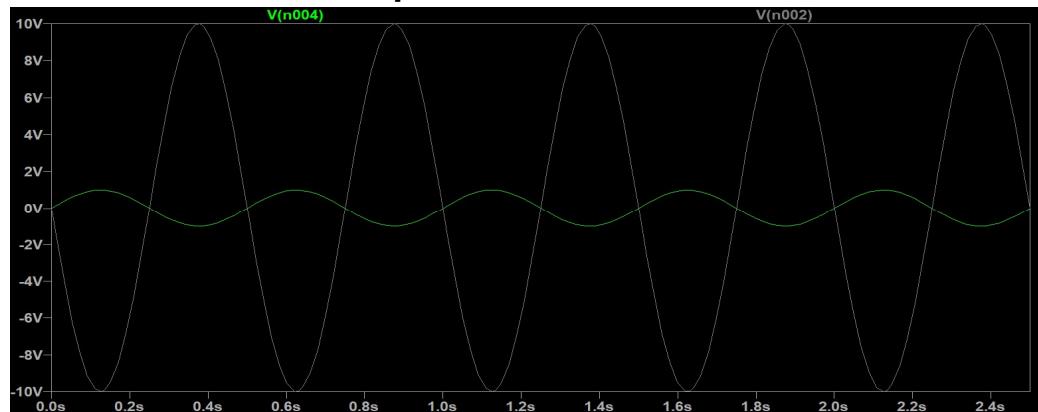
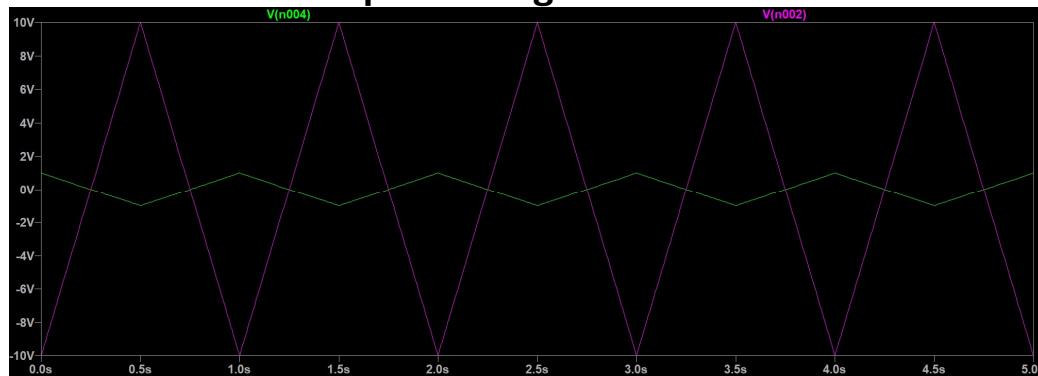
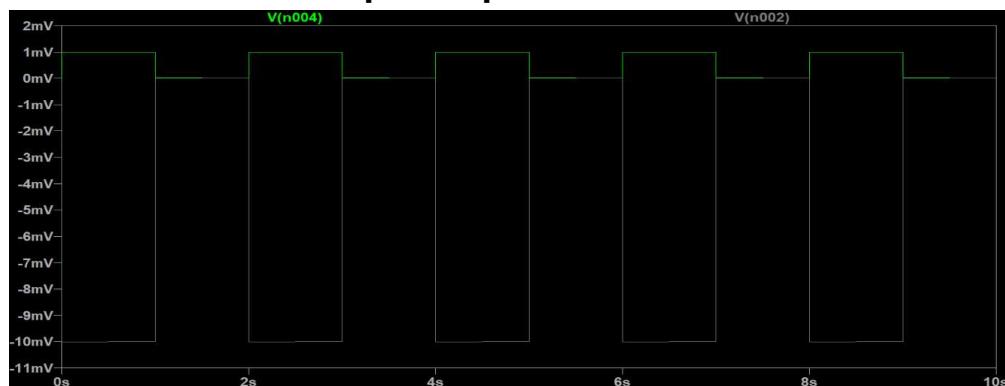
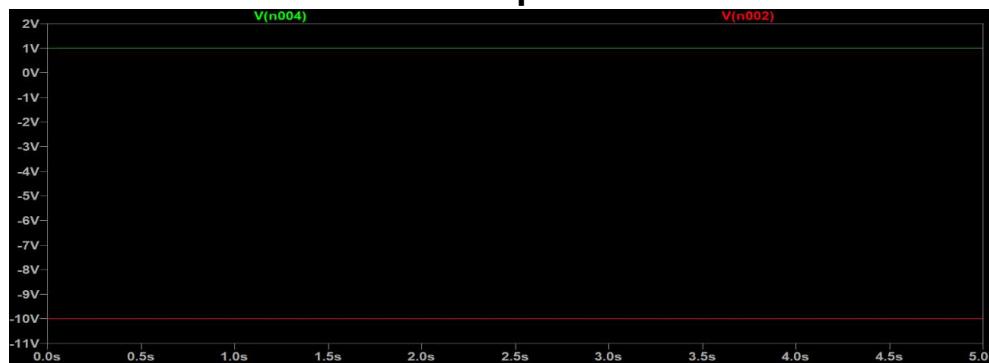
Discussion:

The inverting amplifier using the 741 op-amp showed a clear 180° phase inversion for all input types—sine, triangular, square, and DC. The observed gain values closely followed the theoretical relation $A_v = -\frac{R_f}{R_1}$. Sine and triangular waveforms were linearly amplified, while square waves exhibited finite rise and fall times due to slew-rate limits. Slight deviations in amplitude and symmetry were caused by input bias currents, offset voltage, and the limited output swing of the 741.

Conclusion:

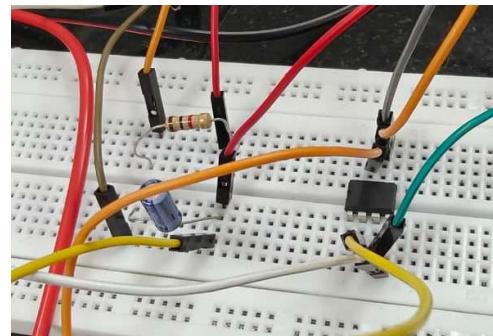
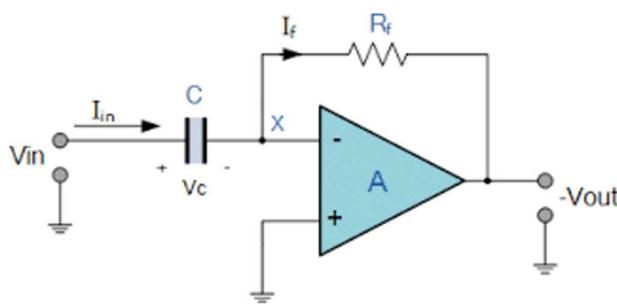
- The circuit successfully demonstrated the behaviour of an inverting amplifier.
- Output phase inversion and gain relation $A_v = -\frac{R_f}{R_1}$ were verified.
- Amplification remained linear within supply and slew-rate limits.
- Practical non-idealities of the 741 accounted for observed minor discrepancies.

LT Spice Simulations:

Input: Sine Wave**Input: Triangular Wave****Input: Square Wave****DC Input**

Objective 8.2: To study the working of op-amp as differentiator.

Schematic diagram:



Brief Description:

An operational amplifier differentiator is a circuit arrangement that generates an output voltage amplitude proportionate to the rate of change of the input voltage. It signifies that a variation in the input voltage signal will promptly result in a corresponding change in the output voltage.

$$V_{OUT} = -R_f C \frac{dV_{IN}}{dt}$$

The output voltage V_{out} is a constant equal to $-R_f * C$ multiplied by the temporal derivative of the input voltage V_{in} . The minus sign (-) signifies a 180° phase shift due to the connection of the input signal to the inverting input terminal of the operational amplifier.

Equipment/Components Required:

S. No.	Component	Specification Values	Quantity
1.	Dual power supply for supply of 741 IC	0-12 V	1
2.	AFM (arbitrary function generator)	Digital	1
3.	Resistors	10 kΩ	1
4.	Op Amp	IC-741	1
5.	Capacitor	10 μF	1

Results:

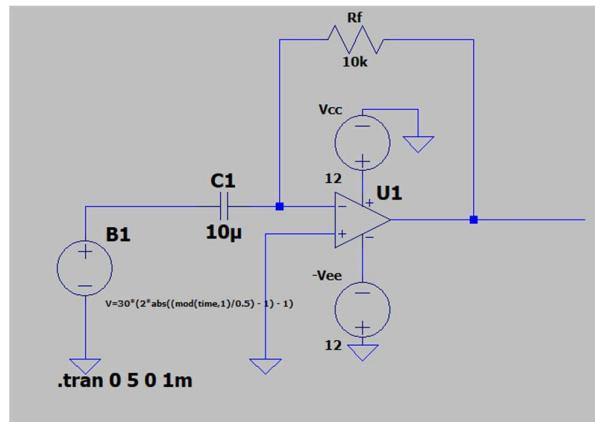
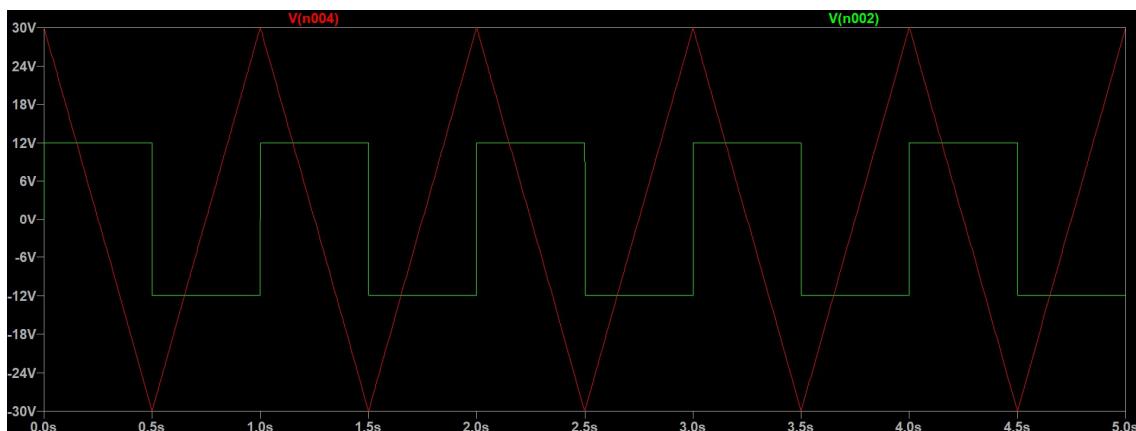


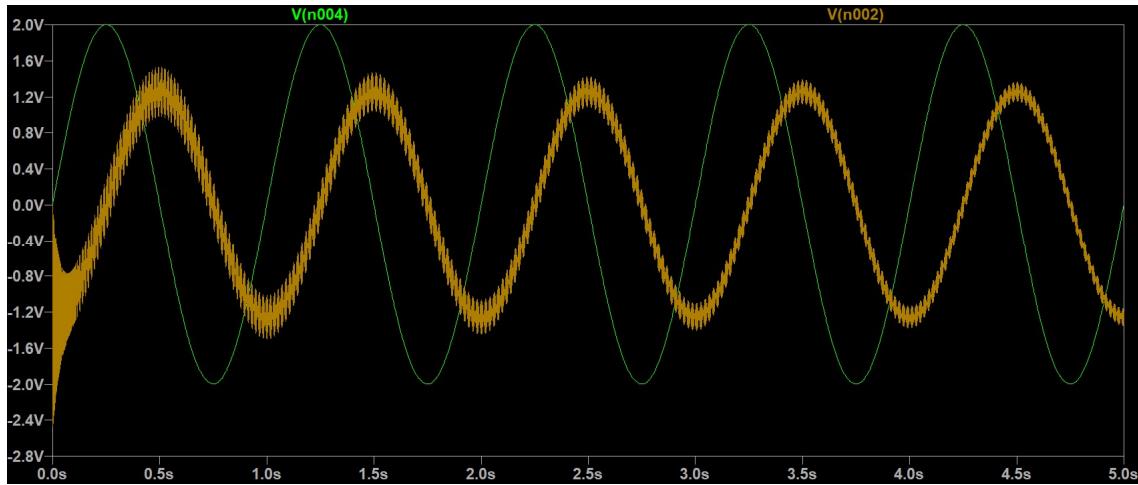
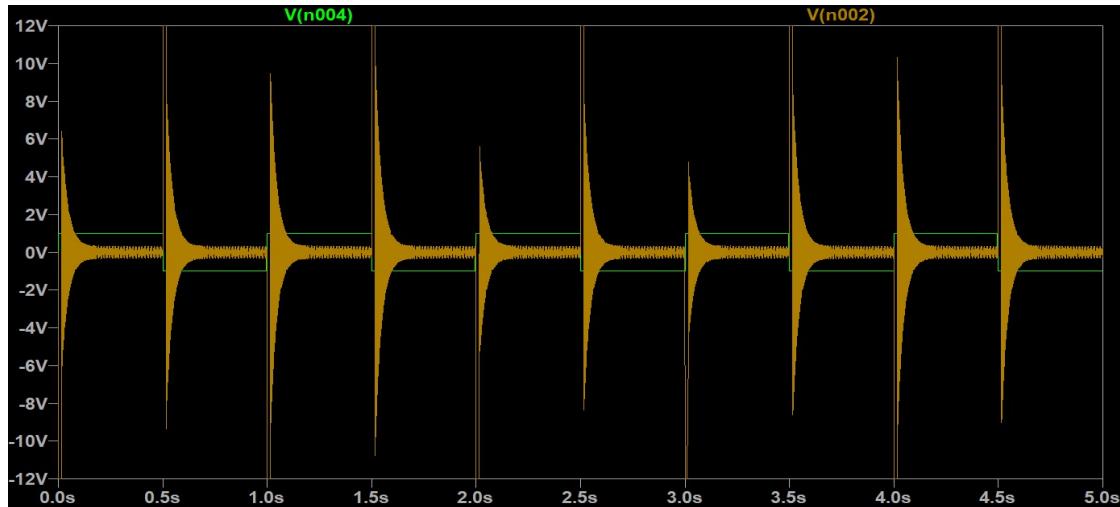
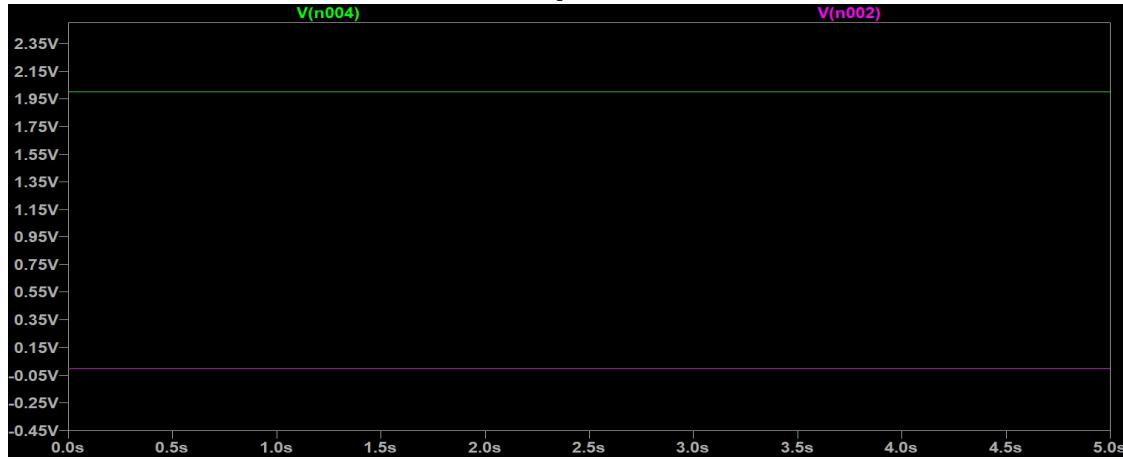
Discussion:

The differentiator circuit using the 741 op-amp produced outputs proportional to the rate of change of the input signal, verifying the relation $V_{out} = -R_f C \frac{dV_{in}}{dt}$. For a **square-wave input**, sharp positive and negative spikes appeared at transitions, indicating high $\frac{dV}{dt}$. A **sine-wave input** yielded a cosine-shaped output, confirming phase shift and differentiation behaviour. The **triangular input** produced square pulses, consistent with the derivative of a linearly varying waveform. Minor distortions and noise were observed due to high-frequency sensitivity and limitations of the 741 op-amp's slew rate and bandwidth.

Conclusion:

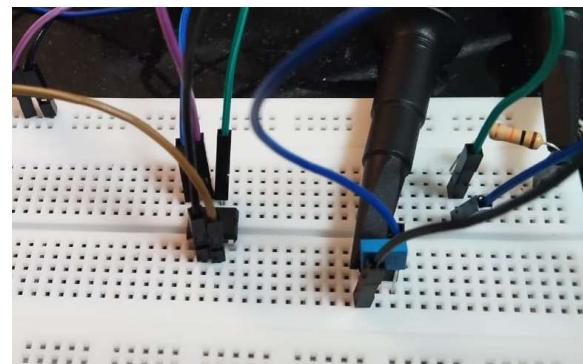
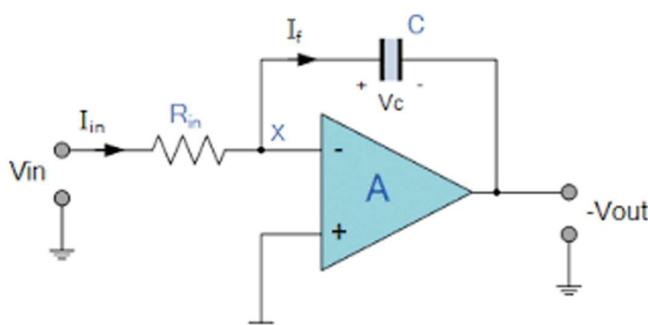
The experiment successfully demonstrated the differentiating action of an op-amp. The output waveforms for different inputs matched the theoretical derivatives—spikes for square waves, cosine for sine waves, and square pulses for triangular waves. The results confirm that the op-amp differentiator produces an output proportional to $\frac{dV_{in}}{dt}$, with small deviations arising from practical non-idealities and noise amplification at high frequencies.

LT Spice Simulations**Input: Triangular Wave**

Input: Sine Wave**Input: Square Wave****DC Input**

Objective 8.3: To study the working of op-amp as integrator.

Schematic diagram:



Brief Description:

An op-amp integrator is an inverting amplifier whose output voltage is proportional to the negative integral of the input voltage, thereby replicating mathematical integration. The output voltage, V_{out} , is equivalent to the constant $-1/RC$ multiplied by the integral of the input voltage, V_{in} . Integrates (and inverts) the input signal $V_{in}(t)$ across the time period t , where $t_0 < t < t_1$, resulting in an output voltage at time $t = t_1$ of

$$V_{OUT} = -\frac{1}{R_{IN}C} \int_0^t V_{IN} dt = -\int_0^t V_{IN} \frac{dt}{R_{IN}C}$$

Equipment/Components Required:

S. No.	Component	Specification Values	Quantity
1.	Dual power supply for supply of 741 IC	0-12 V	1
2.	AFM (arbitrary function generator)	Digital	1
3.	Resistors	10 kΩ	1
4.	Op Amp	IC-741	1
5.	Capacitor	10 μF	1

Results:

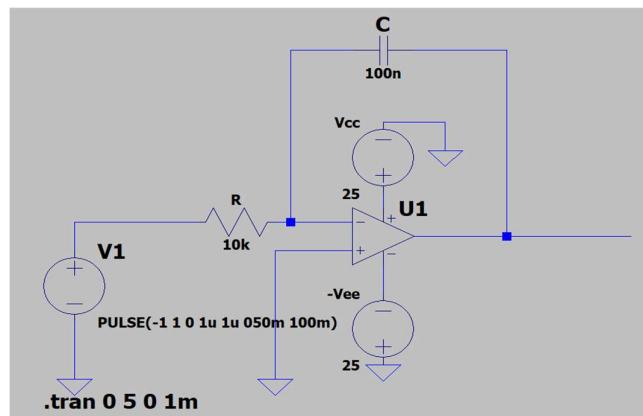
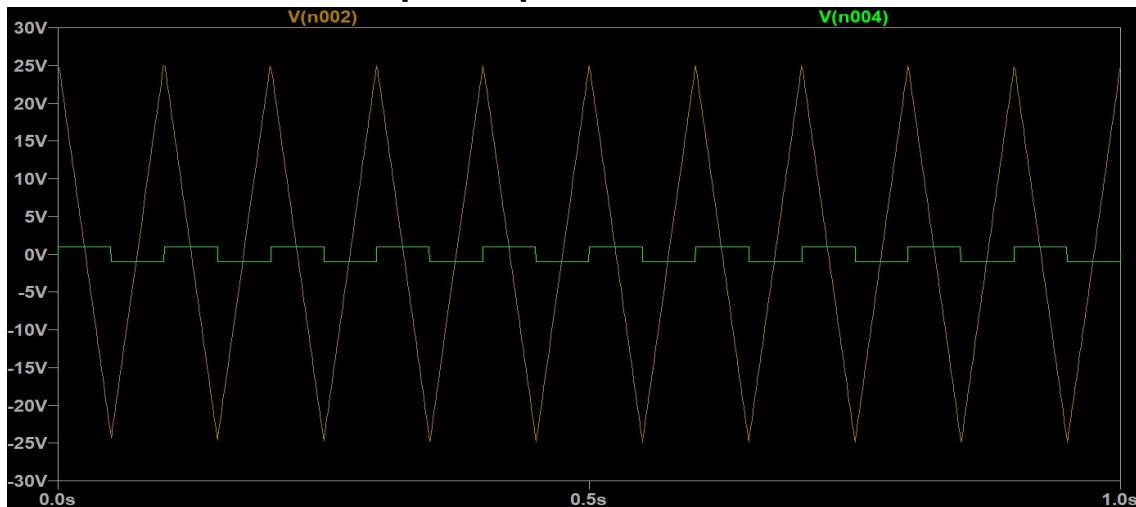


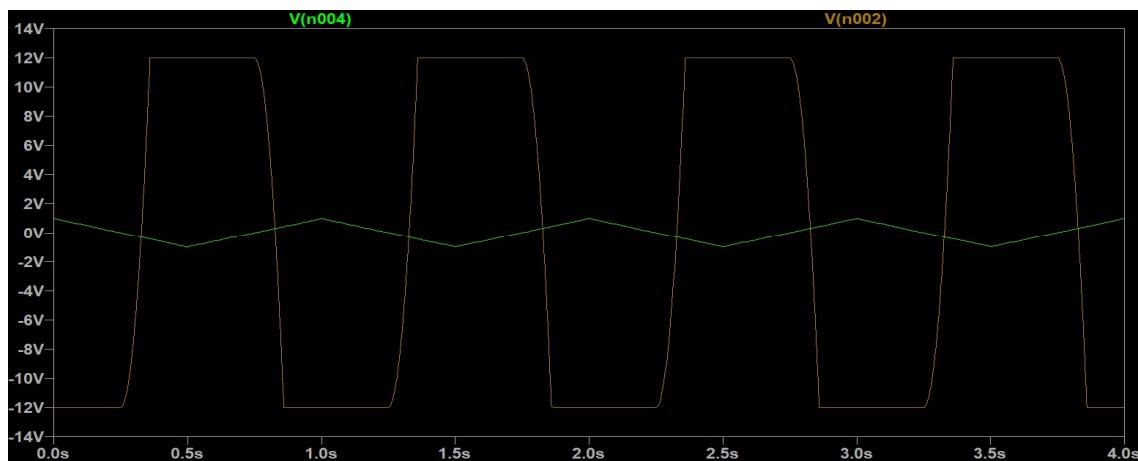
Discussion:

The op-amp integrator circuit using the 741 demonstrated an output proportional to the time integral of the input voltage, following the relation $V_{out} = -\frac{1}{RC} \int V_{in} dt$. For a **square-wave input**, the output was a triangular waveform, confirming integration of a constant-level signal. A **sine-wave input** produced a cosine-shaped output with a 90° phase shift, while the **triangular input** resulted in a parabolic waveform, validating the expected integral behaviour. Minor deviations from the ideal waveforms were due to capacitor leakage, offset voltage, and the finite bandwidth and slew rate of the 741 op-amp.

Conclusion:

The experiment successfully verified the working of the op-amp as an integrator. The observed output waveforms corresponded to the mathematical integrals of the applied inputs, confirming theoretical predictions. The circuit effectively performed signal integration within the limits of the op-amp's practical characteristics, demonstrating the fundamental principle $V_{out} = -\frac{1}{RC} \int V_{in} dt$.

LT Spice Simulations**Input: Square Wave**

Input: Sine Wave**Input: Triangular Wave**

LAB Report 9 ECS 327

Date: October 30, 2025

Name: Shlok Mehndiratta

Roll No: 23309

Title of Experiment:

Experiment 9: To design a Schmitt trigger using Op-amp IC-741.

- **Objective 9.1:** To design a Schmitt trigger using Op-amp IC-741.

Objective 9.1: To design a Schmitt trigger using Op-amp IC-741.

Schematic diagram:

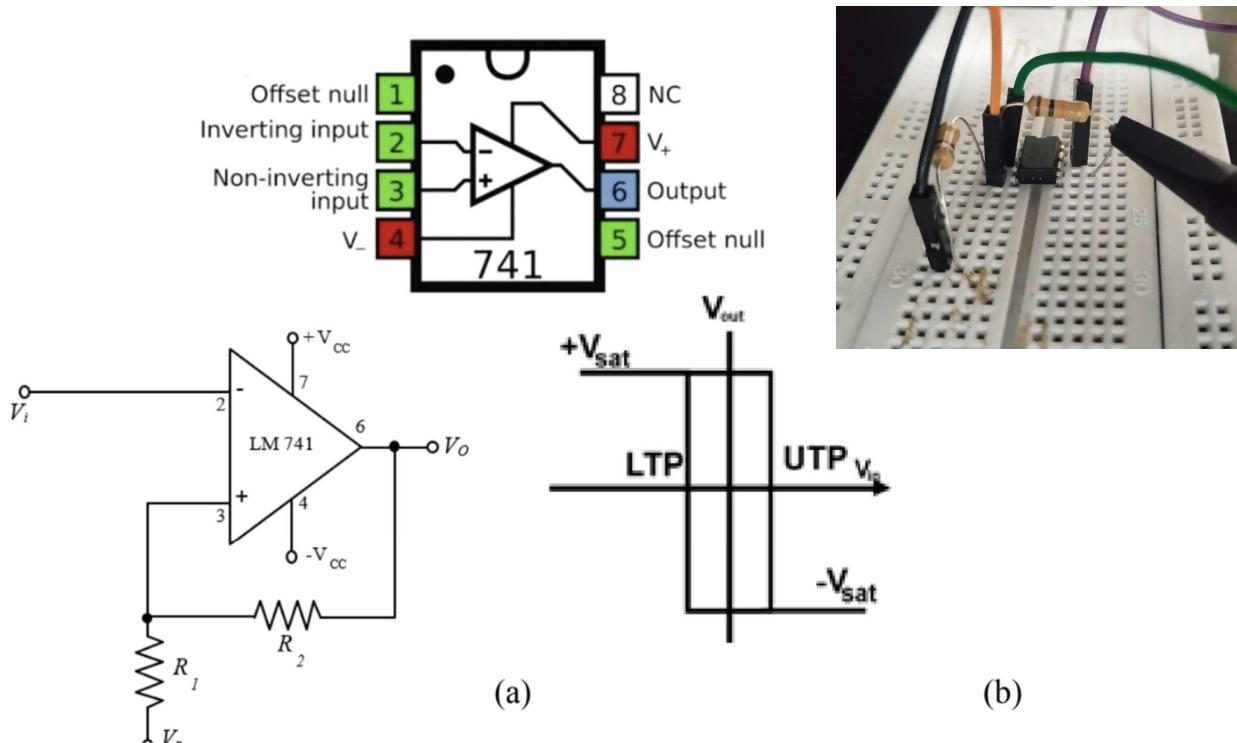


Fig.1 (a) Schmitt trigger circuit and (b) Transfer characteristic.

Brief Description:

Schmitt Trigger circuits are designed with feedback that provides hysteresis in the transfer characteristics. It is basically a comparator with positive feedback. Fig.1 (a) and (b) shows a typical Schmitt trigger circuit along with its transfer characteristic respectively. Schmitt trigger converts an irregular-shaped waveform to a square wave or pulse. This circuit is also known as a squaring circuit. As the input voltage increases it reaches a threshold voltage (the upper threshold point UTP) at which the output voltage goes to negative saturation. As the input voltage decreases it reaches another threshold voltage (the lower threshold point LTP) at which the output voltage goes to positive saturation. The output remains stable with the voltage difference between UTP and LTP larger than the noise.

The input voltage V_{in} triggers (changes the state of) the output V_o every time it exceeds certain voltage levels called Upper threshold voltage, V_{UTP} , and Lower threshold voltage, V_{LTP} . These threshold voltages are calculated as follows:

$$V_{UTP} = \left(\frac{R_2}{R_1+R_2} \right) V_{ref} + \left(\frac{R_1}{R_1+R_2} \right) (V_{sat}) \quad \text{when } V_o = V_{sat}$$

$$V_{LTP} = \left(\frac{R_2}{R_1+R_2} \right) V_{ref} + \left(\frac{R_1}{R_1+R_2} \right) (-V_{sat}) \quad \text{when } V_o = -V_{sat}$$

The hysteresis width is the difference between these two threshold voltages i.e.

$$H = V_{UTP} - V_{LTP}$$

The output of the Schmitt trigger is a square wave when the input is a sine wave or triangular wave, whereas if the input is a saw tooth wave, the output is a pulse wave.

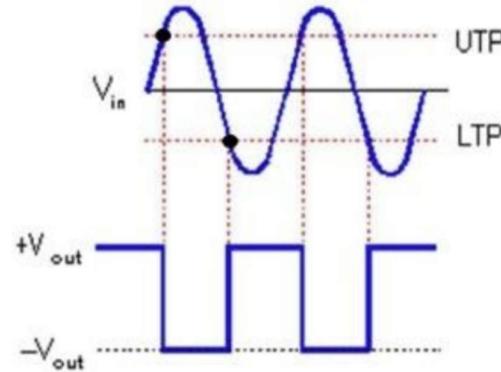


Fig.2 Output response of the Schmitt trigger circuit for sine wave input.

Equipment/Components Required:

S. No.	Component	Specification Values	Quantity
1.	Dual power supply for supply of 741 IC	0-12 V	1
2.	AFM (arbitrary function generator)	Digital	1
3.	Resistors	10 kΩ	2
4.	Op Amp	IC-741	1
5.	Oscilloscope	-	1
6.	Multimeter	Digital	1

Results:

$V_{in} = 6$ volts peak to peak, 10 kHz

$V_{cc} = 5$ volts, $-V_{cc} = -5$ volts

1. $V_{ref} = 0$ volts, $R_1 = 10$ kΩ, $R_2 = 10$ kΩ



Here,

$V_{sat} = 3.96$ volts,

$-V_{sat} = -3.21$ volts

$$V_{UTP} = \left(\frac{R2}{R1+R2} \right) Vref + \left(\frac{R1}{R1+R2} \right) (Vsat) = 1.98 \text{ volts}$$

$$V_{LTP} = \left(\frac{R2}{R1+R2} \right) Vref + \left(\frac{R1}{R1+R2} \right) (-Vsat) = -1.60 \text{ volts}$$

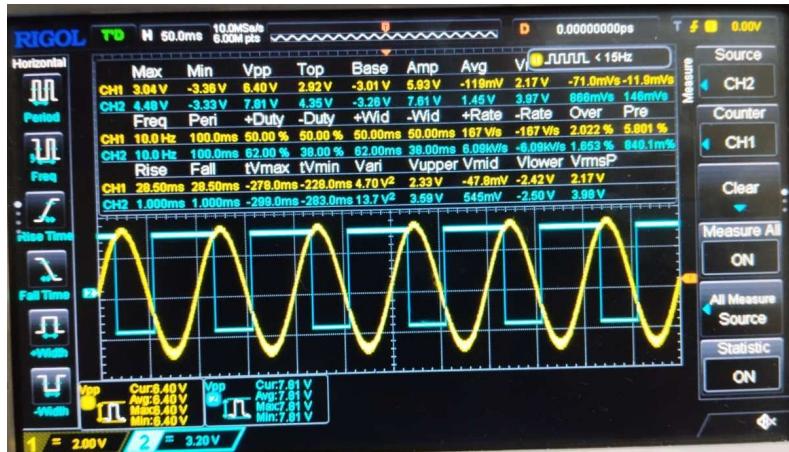
\therefore Hysteresis = $V_{UTP} - V_{LTP} = 3.58 \text{ volts}$

Experimental Results:

$$V_{UTP} = 1.86 \text{ volts}$$

$$V_{LTP} = -1.79 \text{ volts}$$

2. $V_{ref} = 1 \text{ volt}, R1 = 10 \text{ k}\Omega, R2 = 10 \text{ k}\Omega$



Here,

$$Vsat = 4.35 \text{ volts}, \quad -Vsat = -3.26 \text{ volts}$$

$$V_{UTP} = \left(\frac{R2}{R1+R2} \right) Vref + \left(\frac{R1}{R1+R2} \right) (Vsat) = 2.68 \text{ volts}$$

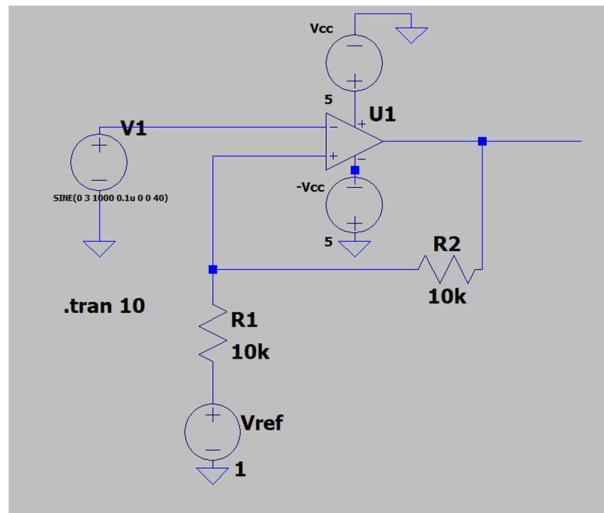
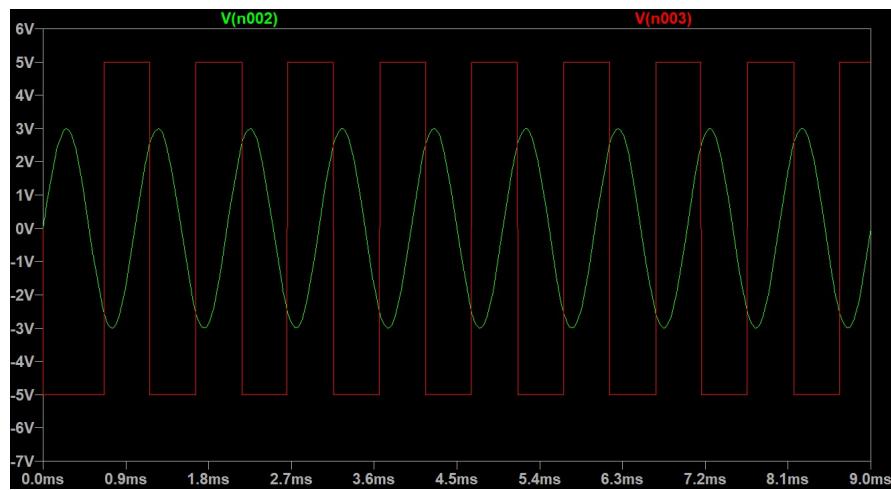
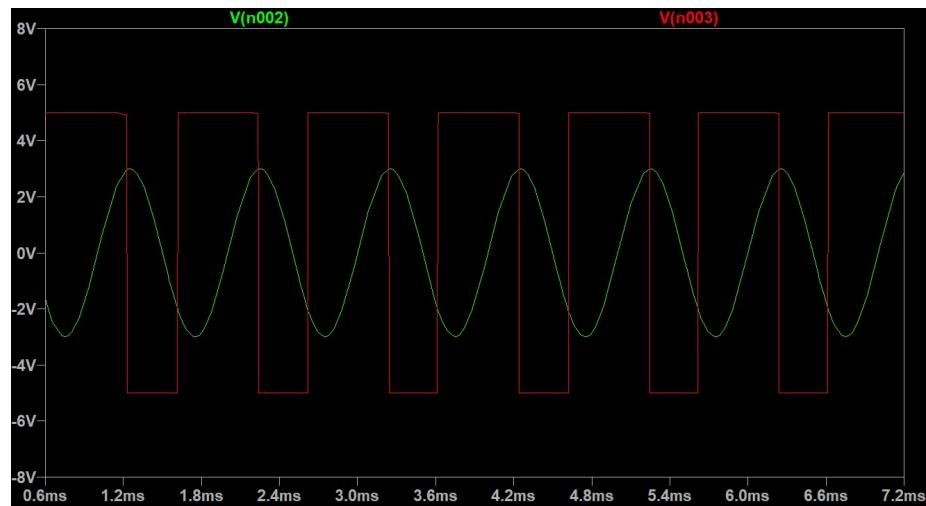
$$V_{LTP} = \left(\frac{R2}{R1+R2} \right) Vref + \left(\frac{R1}{R1+R2} \right) (-Vsat) = -1.13 \text{ volts}$$

\therefore Hysteresis = $V_{UTP} - V_{LTP} = 3.81 \text{ volts}$

Experimental Results:

$$V_{UTP} = 2.52 \text{ volts}$$

$$V_{LTP} = -1.02 \text{ volts}$$

LT Spice Simulations:1. $V_{ref} = 0$ volts2. $V_{ref} = 1$ volts

LAB Report 10 ECS 327

Date: October 31, 2025

Name: Shlok Mehndiratta

Roll No: 23309

Title of Experiment:

Experiment 10: Digital Logic

- **Objective 10.1:** To design Logic gates using transistors

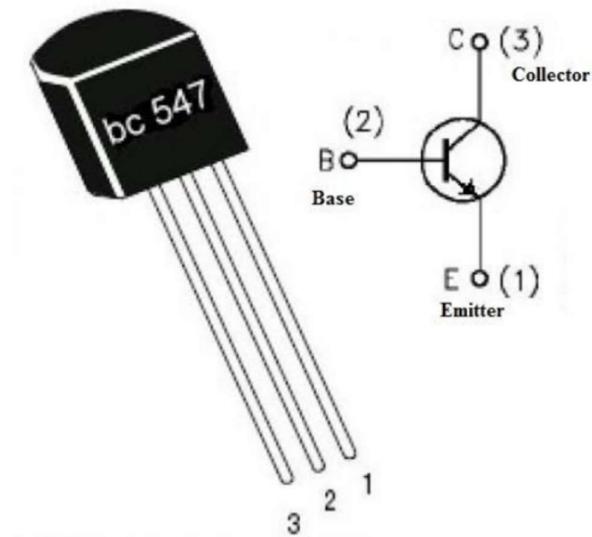


Fig. 1 Diagram of BC 547

Objective 10.1: To design Logic gates using transistors.

Brief Description:

Transistors

Transistors are made of materials like silicon or germanium that allow electrical current to flow through them in a controlled manner. The materials of transistors are doped, or “treated,” with impurities to create a structure called a p-n junction. The notations p and n refer to the type of dopant atoms (impurities) added to the semiconductor material. A transistor consists of three main parts: the emitter, the base, and the collector. The emitter serves as the source of electrons, the collector as the drain, and the base as the control terminal. The BC547 transistor is an NPN transistor. The working states of BC547 transistor include the following:

- Forward bias
- Reverse bias

In a forward bias mode, the two terminals like emitter and collector are connected to allow the flow of current through it. Whereas in a reverse bias mode, it doesn't allow the flow of current through it because it works as an open switch. Transistors are used in a wide variety of electronic devices and equipment like computer, cell phones, space and military applications, etc.

Logic gates

A gate is an electronic device which is used to compute a function on a two valued signal. Generally, all logic gates have one output and two inputs. Some logic gates like NOT or Inverter has only one input and one output. The inputs of the logic gates are designed to receive only binary data (only low 0 or high 1) by receiving the voltage input. Logic gates can be used in manufacturing binary counters, calculators, in decision making regarding automatic control, etc.

The main advantages of logic gates built using transistors can be summarized as given below:

- Transistors are cheaper than ICs.
- Transistorized logic gates can be operated with voltages as low as 1.5 V, while the IC counterparts need a minimum of 3 V.

A transistorized logic gate can be customized to control heavier loads, which an IC based logic gate cannot do.

Advantages of Transistor based Logic gates

The main advantages of logic gates built using transistors can be summarized as given below:

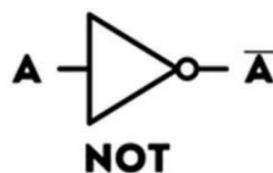
- Transistors are cheaper than ICs.
- Transistorized logic gates can be operated with voltages as low as 1.5V, while the IC counterparts need a minimum of 3 V.

- A transistorized logic gate can be customized to control heavier loads, which an IC based logic gate cannot do.

Equipment/Components Required:

S. No.	Component	Specification Values	Quantity
1.	Dual power supply for supply of 741 IC	0-12 V	1
2.	NPN Transistors	BC547	3
3.	Resistors	1 kΩ, 100 kΩ	2
4.	LED	-	1
5.	Breadboard	-	1
6.	Jumper Wires	-	10

NOT GATE



NOT Gate	
A	\bar{A}
0	1
1	0

Fig. 2 Symbol and truth table for NOT Gate

Circuit Diagram:

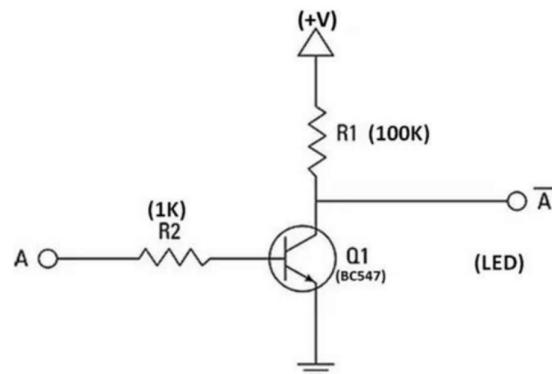
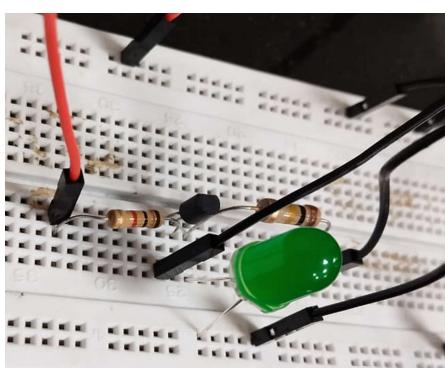
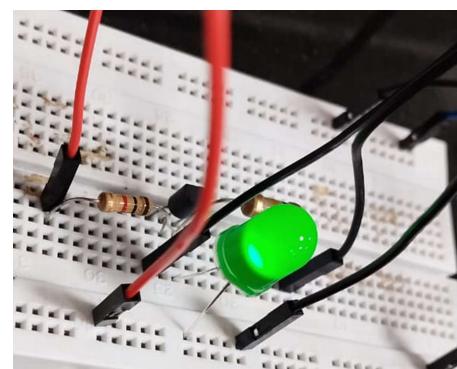


Fig. 2 Transistor based NOT Gate



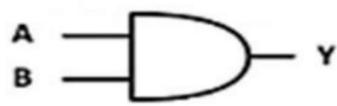
A: 1



A: 0

AND GATE

Inputs		Output
A	B	Y
0	0	0
0	1	0
1	0	0
1	1	1



$$Y = A \cdot B$$

Fig. 3 Symbol and truth table for AND Gate

- Circuit Diagram:**

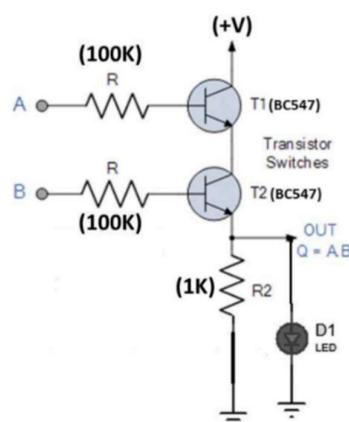
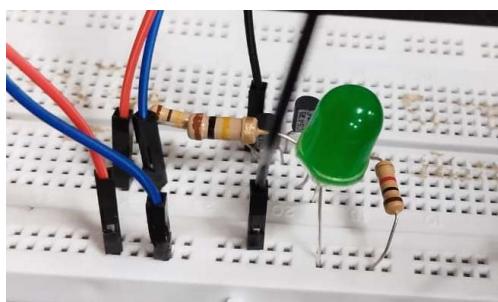
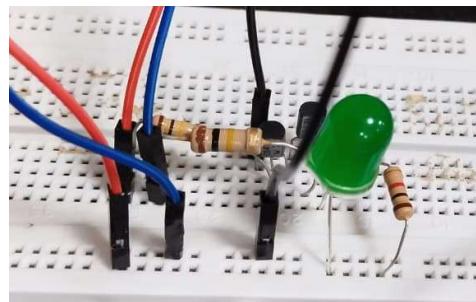


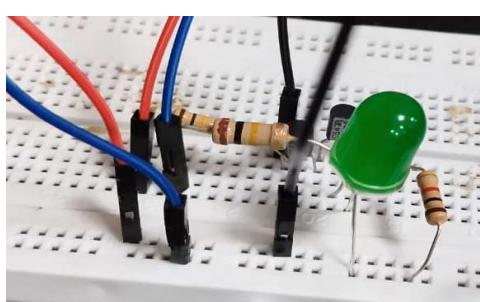
Fig. 4 Transistor based AND Gate



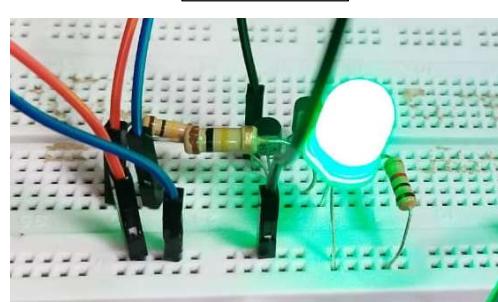
A: 0, B:0



A: 0, B:1



A: 1, B:0



A: 1, B:1

NAND GATE

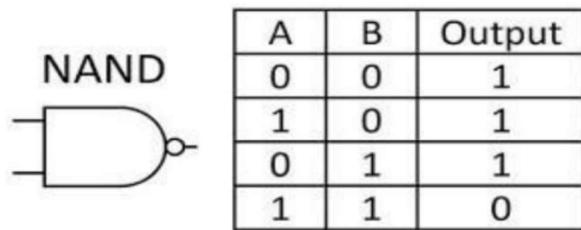


Fig. 5 Symbol and truth table for NAND Gate

- Circuit Diagram:**

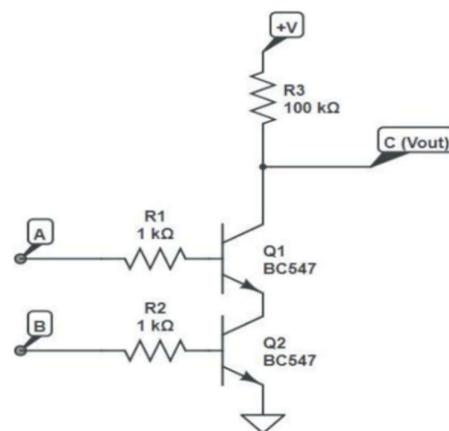
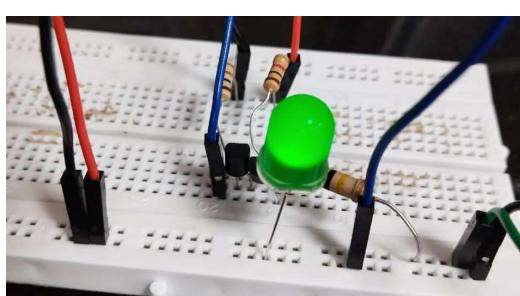
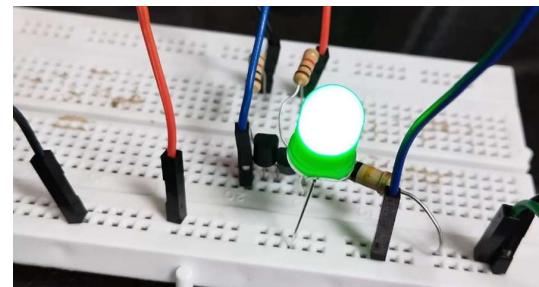


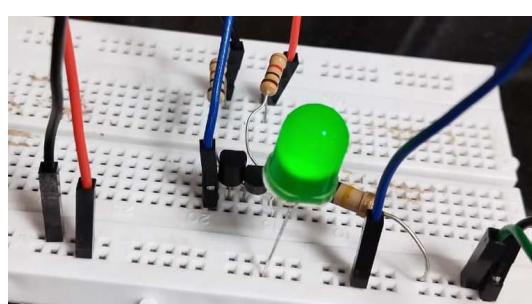
Fig. 6 Transistor based NAND Gate



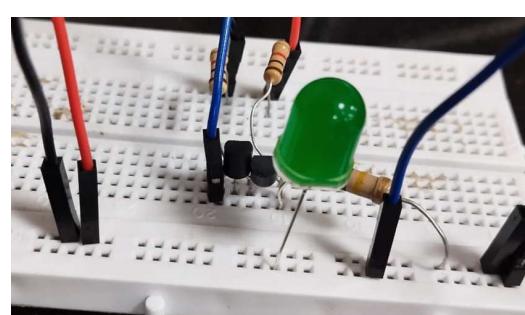
A: 0, B:0



A: 0, B:1



A: 1, B:0



A: 1, B:1

OR GATE

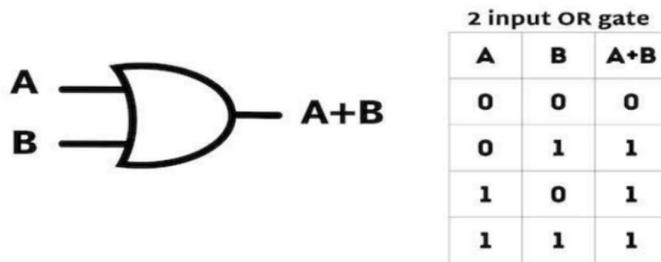


Fig. 7 Symbol and truth table for OR Gate

- **Circuit Diagram:**

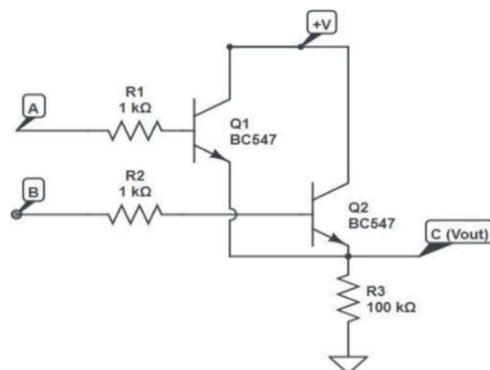
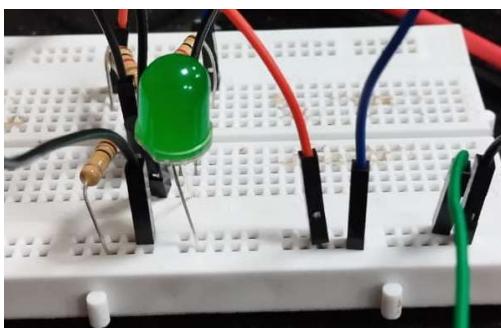
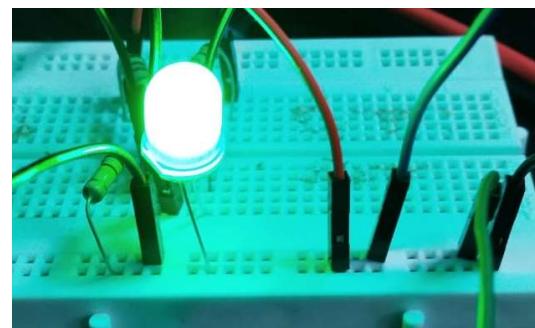


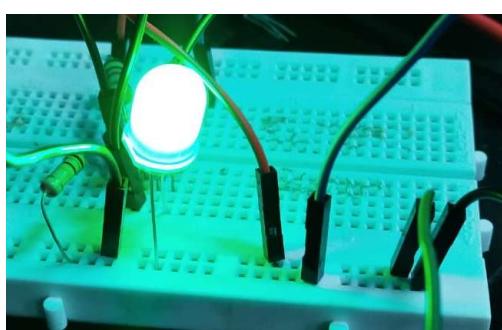
Fig. 8 Transistor based OR Gate



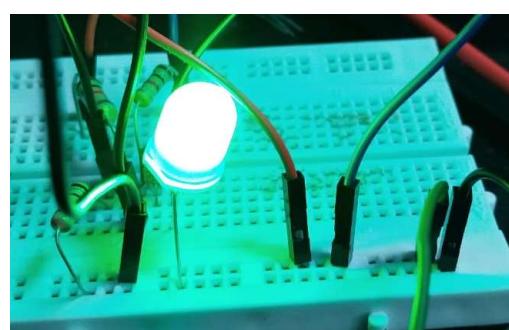
A: 0, B:0



A: 0, B:1

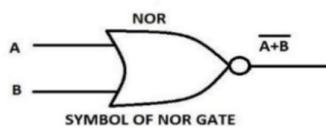
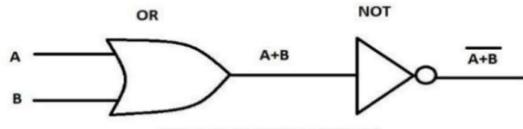


A: 1, B:0

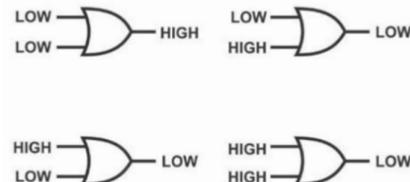


A: 1, B:1

NOR GATE



Input		Output
A	B	Y
0	0	1
0	1	0
1	0	0
1	1	0



- Circuit Diagram:

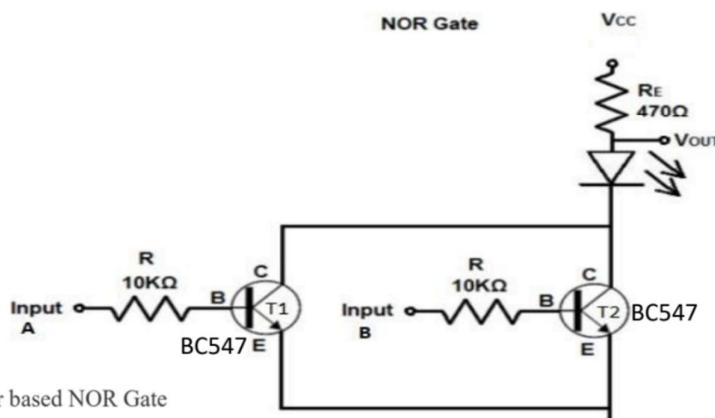
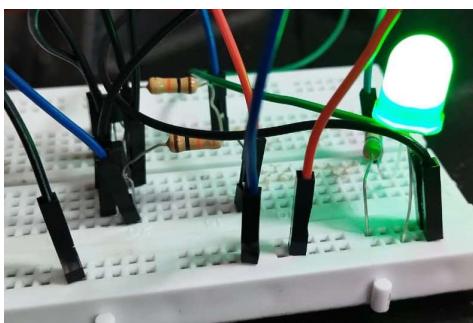
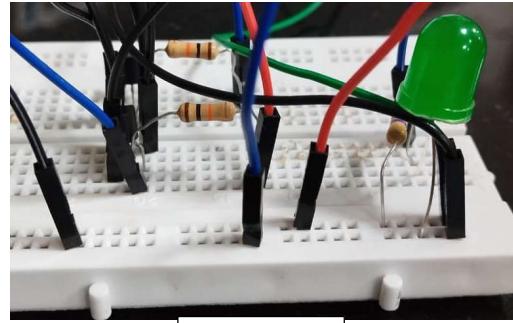


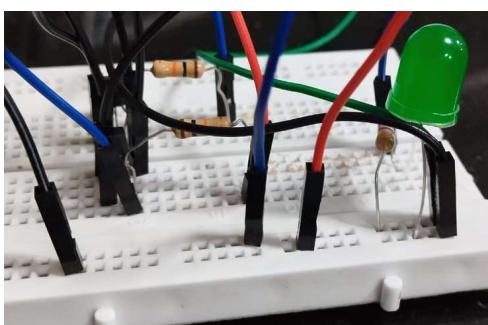
Fig. 10 Transistor based NOR Gate



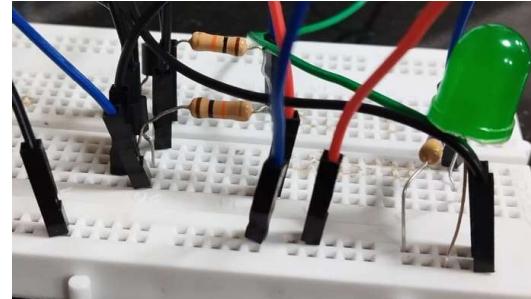
A: 0, B:0



A: 0, B:1

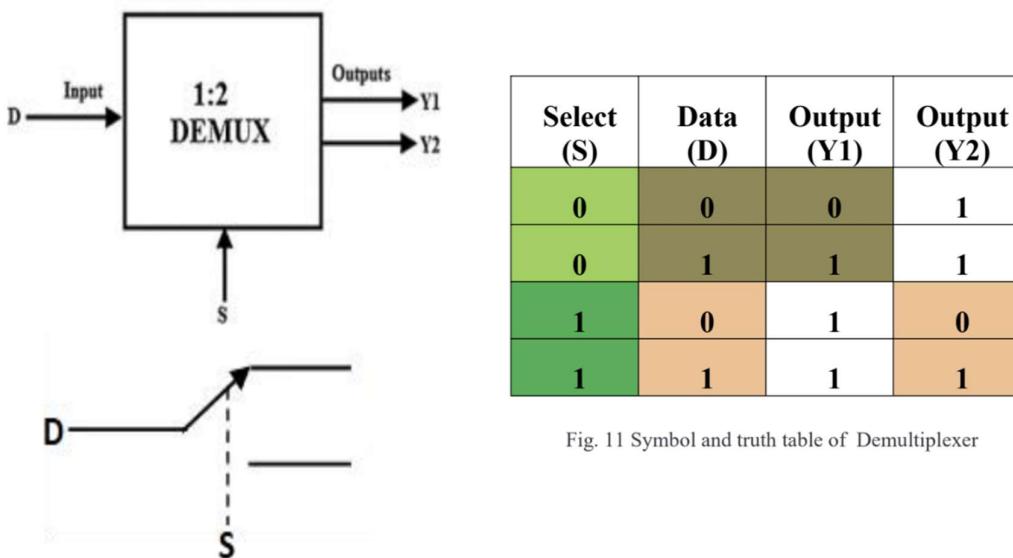


A: 1, B:0



A: 1, B:1

Demultiplexer



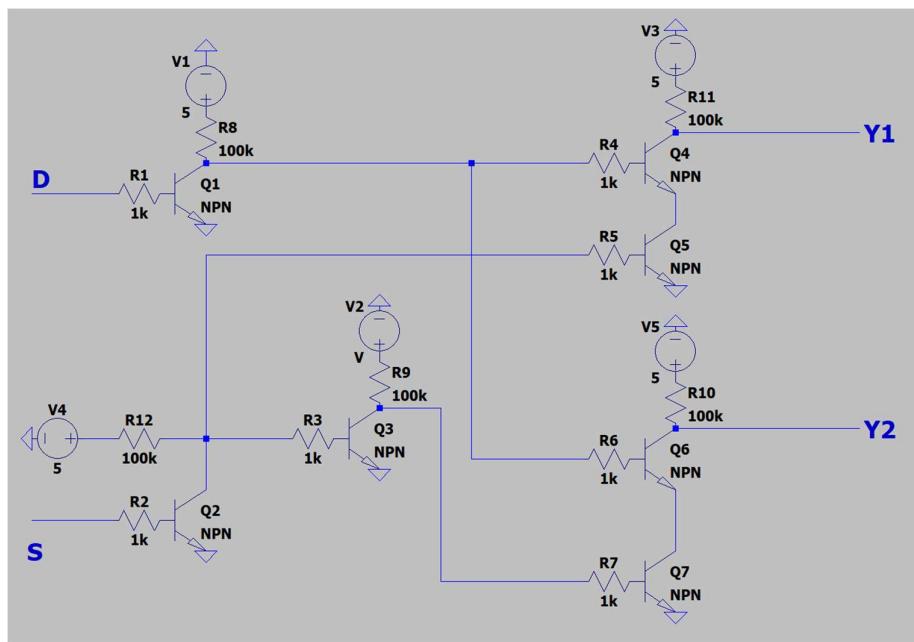
A 1-of-2 demultiplexer with three NOT gates and two NAND circuits is shown in upper Figure. The appropriate output is chosen using the one-bit “Select Line S” (decide the selection of outputs), which may be either Y1 or Y2, while the driving information is applied to the circuit using the “Data input D”. The circuit operates most effectively when the data rate is maintained under 10 kHz.

The D input is supplied with the required signal, which inverts the incoming data at Q3's collector. The output of Q1 is inverted the Select line input S. At the Q1's collector, output is divided into two paths. In the first path, Q1's output is supplied to Q5's base (one of a two-input NAND gate's legs). In the second path, Q1's high output is simultaneously supplied into the input of another NOT gate (Q2). After undergoing a double inversion, Q2's output is same as Select line input S. This Select line input S is supplied Q7's base (one terminal of a second NAND gate, made up of Q6 and Q7).

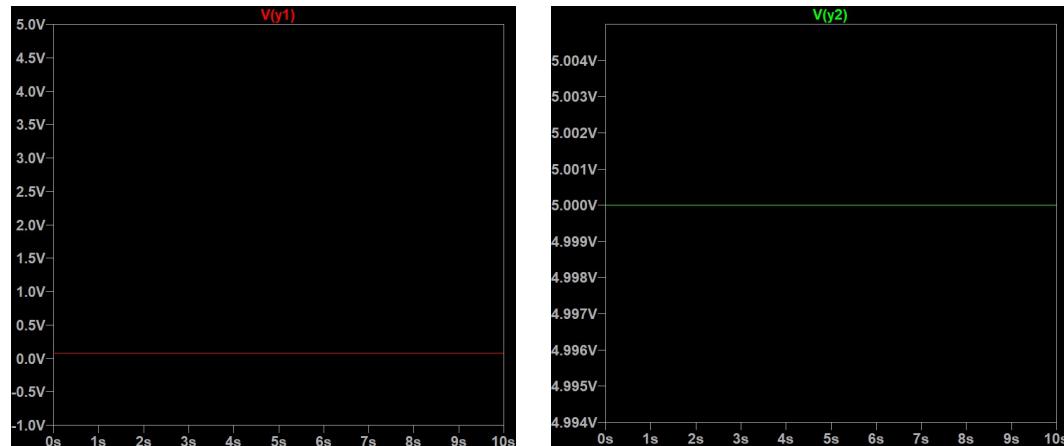
- When S=0. Output Y1 becomes active, which is equal to the data input D.
- When S=1. Output Y2 becomes active which is equal to the data input D.

Equipment/Components Required:

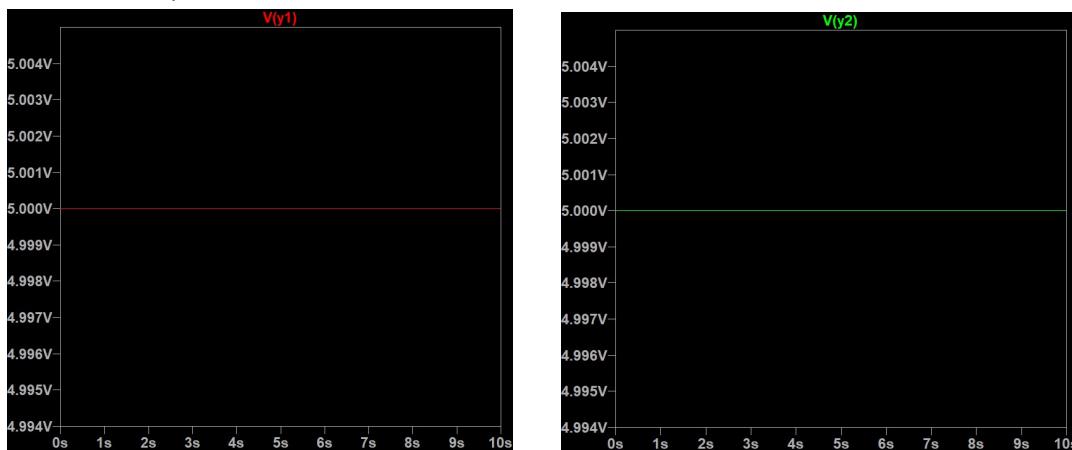
S. No.	Component	Specification Values	Quantity
1.	Dual power supply for supply of 741 IC	0-12 V	1
2.	NPN Transistors	BC547	7
3.	Resistors	1 kΩ	7
4.	Resistors	100 kΩ	5
5.	Breadboard	-	1

LT Spice:

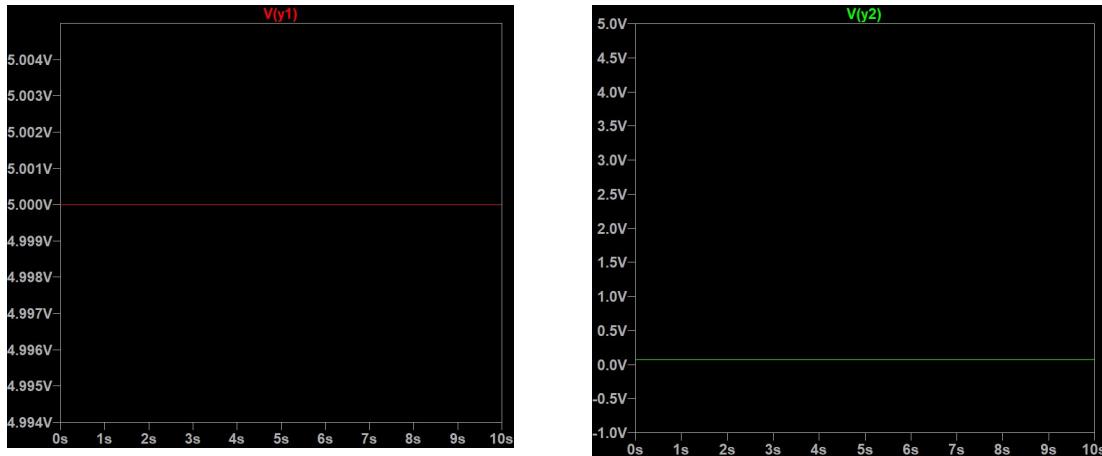
Case 1: S:0, D:0



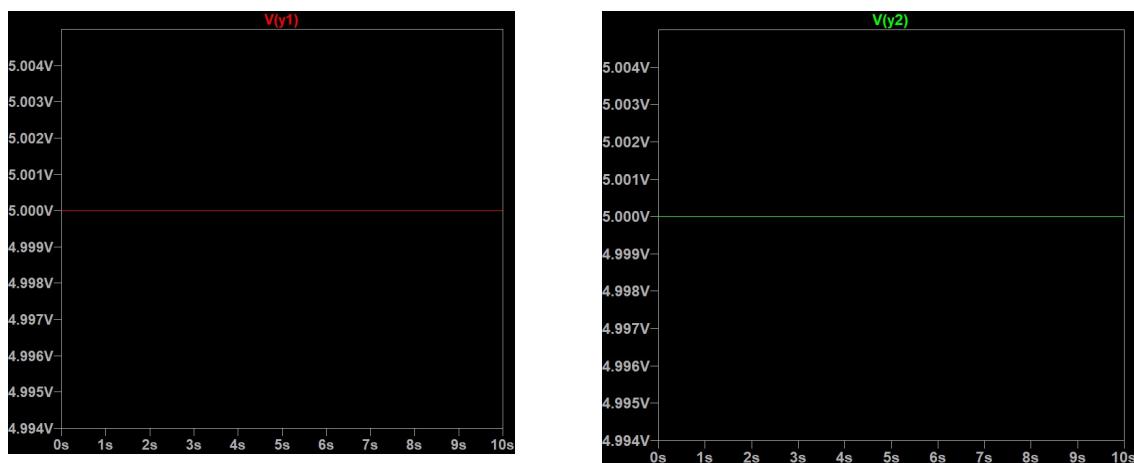
Case 2. S:0, D:1



Case 3. S:1, D:0



Case 4: S:1, D:1



-----End-----