SİVAS UNIVERSITY OF SCIENCE AND TECHNOLOGY
FACULTY OF ENGINEERING AND NATURAL SCIENCES

CIRCUIT THEORY - II LABORATORY

Experiments Manual Report

Supervisor: Asst. Prof. Sıtkı AKKAYA

Prepared by: Res. Asst. Arda Ahmet ATEŞ
Res. Asst. Berker ÇOLAK
Res. Asst. Berke Can TURAN
Res. Asst. İremnur DURU

SİVAS

Experiment 1: Impedance

1. Objective of the Experiment

The objective of this experiment is to familiarize students with the fundamental electrical equipment commonly used in the laboratory environment. Through hands-on experience, students will gain practical knowledge of operating frequently used instruments such as the digital multimeter and DC power supply. This experiment aims to ensure that students develop the necessary skills to handle these devices effectively and safely, laying the groundwork for future laboratory work and more advanced experiments in electrical and electronics engineering.

2. Theoretical Background

Impedance is a concept used to describe how an electrical circuit resists the flow of alternating current (AC). It extends the idea of resistance, which applies to direct current (DC) circuits, to situations involving AC, where the current and voltage can change with time. Impedance is a complex quantity because it includes both:

- **Resistance (R)**: Opposition to the flow of current that dissipates energy as heat, just like in DC circuits.

- **Reactance (X)**: Opposition to the flow of current that does not dissipate energy but instead stores energy temporarily in electric or magnetic fields, such as in capacitors and inductors.

Impedance mainly represented with ’Z’ and can be expressed as:

$$Z=R+jX$$

- Z is the total impedance (in ohms, Ω)
- R is the total resistance (in ohms, Ω)
- X is the total reactance (in ohms, Ω)
- j^2 is the imaginary unit

a) Type of Reactance

- **Capacitive Reactance (X_C):** Caused by capacitors in the circuit. Capacitors oppose changes in voltage, causing lagging in voltage. Its value is:

$$X_C = -\frac{1}{\omega C}$$

where ω is the angular frequency (in radians per second), and C is the capac-itance (in farads).

- **Inductive Reactance (X_L):** Caused by inductors in the circuit. Inductors oppose change in current, causing lagging in current. Its value is:

$$X_L = -\omega L$$

where L is the inductance (in henries).

b) Impedance in Polar Form

The impedance can also be expressed in polar form, representing the magnitude and phase angle:

$$Z = |Z|\angle\theta^{\circ}$$

where:

- $|Z|$ is the magnitude of the impedance given by $\sqrt{R^2 + X^2}$.
- θ is the phase angle, given by $tan^{-1}(\frac{X}{R})$

c) Kirchoff's Laws in AC

Ohm's Law

In AC circuits, current I is represented as a phasor of current, so it can have both magnitude and phase. Since current and voltage are related by impedance in AC circuits, Ohm's Law in AC is expressed as:

$$I = \frac{V}{Z}$$

where:

- I is the phasor current,
- V is the phasor voltage,
- Z is the phasor impedance of the circuit elements,

Kirchoff's Current Law (KCL)

If multiple currents with different magnitudes and phases are flowing through a node, their complex currents (phasor forms) are added algebraically as:

$$\sum I_{in} = \sum I_{out}$$

Example: If three current I1, I2, and I3 enter a node, and one current I4 leaves the node, KCL would state:

$$I_1 + I_2 + I_3 = I_4$$

All the currents must be added as phasors, meaning both their magnitudes and phases must be considered.

Kirchoff's Current Law (KCL)

The sum of the phasor voltages around any closed loop in an AC circuit is zero.

$$\sum V = 0$$

where V represents the phasor voltage across different elements (resistors, capacitors, inductors) in the loop.

Example: Consider an AC circuit loop containing a resistor (R), inductor (L), and capacitor (C). According to KVL, the sum of the voltage drops around the loop is zero:

$$V_R + V_L + V_C = 0$$

Expressing each voltage as the product of current and impedance:

$$IZ_R + IZ_L + IZ_C = 0$$

where $Z_R = R$, $Z_L = j\omega L$, and $Z_C = \frac{-1}{j\omega C}$. Substituting the given expressions and factor out the current would yield:

$$I(R + j\omega L + \frac{1}{j\omega C}) = 0$$

PRELAB WORK

- Solve the circuitry given in Figure 1 for Vo with given parameters in the Table 1, and fill the "Calculated Vo", and "Calculated Io" columns.
- Solve the circuitry given in Figure 2 for Vo with given parameters in the Table 2, and fill the "Calculated Vo", and "Calculated Io" columns.

3. Materials Used

- Function Generator
- Oscilloscope
- Resistors
- Capacitors
- Inductors

4. Procedure

- Turn on the electrical table.
- Turn on the devices that are going to be used.
- Built the circuitry given in Figure 1 with the R1 as 1kΩ, and L1 as given values in Table 1.
- Change the frequency according to the ω given in the Table 1 and take note of your measurements and fill the Table 1.
- Built the circuitry given in Figure 2 with the R2 as 1kΩ, and C1 as given values in Table 2.
- Change the frequency according to the ω given in the Table 2 and take note of you measurements and fill in the Table 2.

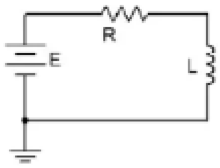


Figure 1: RL Circuit

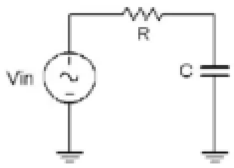


Figure 2: RC Circuit

5. Evaluation of Results

L_1	ω	Calculated V_o	Calculated I_o	Measured V_o	Measured I_o
10μ H	500 kHz				
10μ H	1 MHz				
10μ H	20 MHz				
100μ H	500 kHz				
100μ H	1 MHz				
100μ H	20 MHz				
220μ H	500 kHz				
220μ H	1 MHz				
220μ H	20 MHz				

Table 1: Results of Circuit given in Figure 1.

C_1	ω	Calculated V_o	Calculated I_o	Measured V_o	Measured I_o
10*n* F	500 kHz				
10*n* F	1 MHz				
10*n* F	20 MHz				
100*n* F	500 kHz				
100*n* F	1 MHz				
100*n* F	20 MHz				
1μ F	500 kHz				
1μ F	1 MHz				
1μ F	20 MHz				

Table 2: Results of Circuit given in Figure 2.

6. Safety Precautions

- Always ensure that all equipment is properly connected and powered off before making any changes to the circuit.

- Do not exceed the voltage or current limits of the instruments, especially when using the DC power supply.

- Handle all instruments with care, and avoid using damaged or frayed test leads.

- Be cautious when working with live circuits; avoid touching exposed wires or terminals.

- Keep the working area dry and organized to prevent accidental short circuits or equipment damage.

- Follow the instructor's guidelines and report any malfunctioning equipment immediately.

- Make sure to disconnect the power supply before assembling or disassembling any part of the circuit.

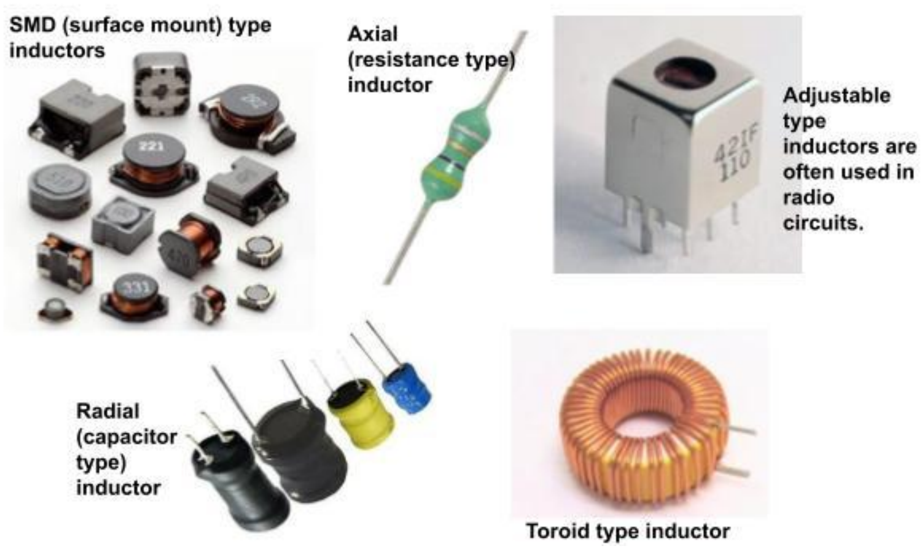
Experiment 2: Series R-L Circuit Analysis

1. Objective of the Experiment

The objective of this experiment is to analyze the electrical behavior of a series R-L circuit under alternating current (AC) conditions. Specifically, the experiment aims to examine how the inductive reactance $\mathbf{X_L}$ varies with frequency and how this affects the total impedance and phase relationship between voltage and current. Through both theoretical calculations and practical measurements using laboratory equipment such as oscilloscopes and function generators, students will gain an understanding of the voltage drops across resistive and inductive components and will learn to interpret phase shifts and impedance in R-L circuits.

2. Theoretical Background

(a) Coil Types



Consider a simple RL circuit in which resistor, R, and inductor, L are connected in series with a voltage supply of Vin (Fig.1). The current flowing in the circuit is I, and the current through resistor R and inductor L is IR and IL respectively. However, the resistor and inductor are connected in series, that is why the current passing through both elements is the same. i.e.,

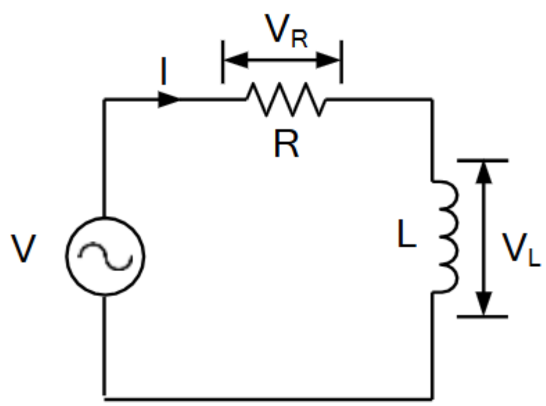


Figure 1: Basic circuit for series R-L

$$I_R = I_L = I \quad (1)$$

The voltages VR and VL are the voltage drops across the resistor and inductor. By applying the Kirchhoff voltage law (The summation of the drop voltages across R and L equal to the input voltage Vin) to this circuit, we get:

$$V_{in} = V_R + V_L \quad (2)$$

Before drawing the phasor diagram of a series RL circuit, one should know the relationship between voltage and current in the case of resistor and inductor.
In the case of the **resistor R,** the voltage and current are in the **same phase**, or we can say that the phase angle difference θ between voltage and current is zero.

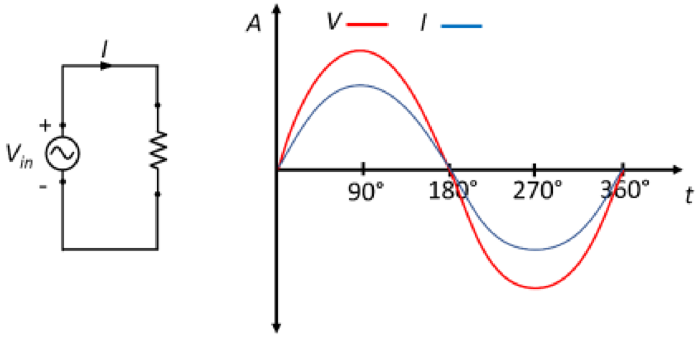


Figure 2: illustrates that the voltage and current wave are in phase in a purely resistive load

In the case of the **inductor L,** the voltage and current are **not in the same phase**. The voltage leads the current by 90°. This means the voltage reaches its maximum when the current attains the zero value.

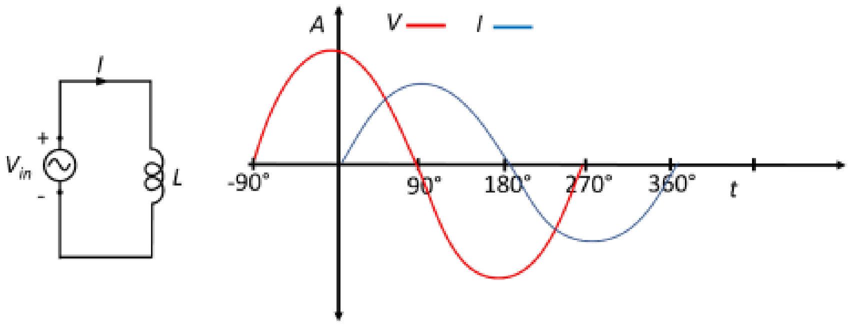


Figure 3: illustrate the voltage and current phase shift in a purely inductive load

The inductor is basically a coil or loops of wire that are either wound around a hollow tube former (air cored) or wound around some ferromagnetic material like iron core to increase their inductive value (inductance).

The inductor stores its energy in the form of a magnetic field that is created when a voltage is applied across an inductor. The growth of the current flowing through the inductor is not instant but is determined by the inductors own self-induced or back emf value. Then for an inductor coil, this back emf voltage VL is proportional to the rate of change of the current flowing through it. In an AC circuit, the opposition to the current flowing through the coils not only depends upon the inductance of the coil but also the frequency f of the applied voltage waveform as it varies from its positive to negative values. The actual opposition to the current flowing through a coil in an AC circuit is determined by the AC Resistance of the coil with this AC resistance being represented by a complex number. But to distinguish a DC resistance value from an AC resistance value, which is also known as Impedance, the term Reactance is used.

Like resistance, reactance is measured in Ohm's but is given the symbol X to distinguish it from a purely resistive "R" value, and as the component in question is an inductor, the reactance of an inductor is called **Inductive Reactance**, X_L and is measured in Ohms. Its value can be found from the formula.

$$X_L = 2\pi f L \tag{3}$$

Where X_L is inductive reactance in (Ω), π is the numeric constant of 3.142, ƒ is the frequency in Hz, and L = inductance in H.

Whenever a sinusoidal voltage is applied to an inductor, the back emf opposes the rise and fall of the current flowing through the coil and in a purely inductive coil which has zero resistance, this impedance (which can be a complex number) is equal to its inductive reactance. Also, reactance is represented by a vector as it has both a magnitude and a direction (angle). See Fig. 3.

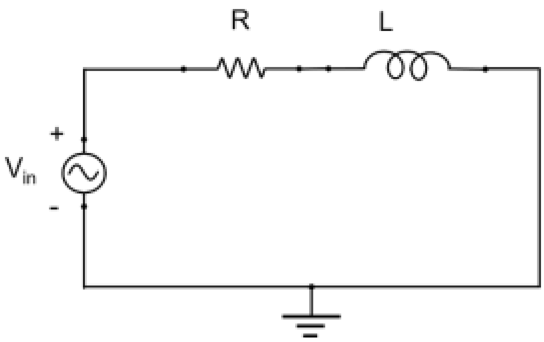


Figure 4: Schematic diagram illustrate an RC circuit connected in series

This simple circuit above consists of a pure inductance of L Henries (H), connected in series with a resistor R (Ohm) and a sinusoidal voltage given by the expression:

$$V_{in} = V_{max} sin(\omega t)\ ,\ \omega=2\pi f \tag{4}$$

This sinusoidal voltage will cause a current to flow and rise from zero to its maximum value. This rise or change in the current will induce a magnetic field within the coil which in turn will oppose or restrict this change in the current.

But before the current has had time to reach its maximum value as it would in a DC circuit, the voltage changes polarity causing the current to change direction. This change in the other direction once again is delayed by the self-induced back emf in the coil, and in a circuit containing a pure inductance only, the current is delayed by 90°.

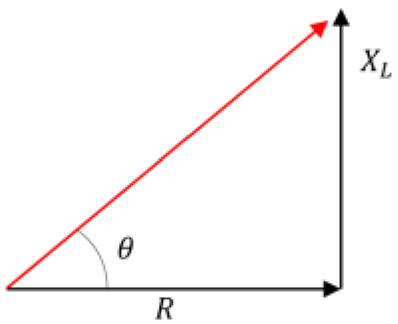


Figure 5: For an RL circuit, θ depends on the values of the R and X_L.

In an RL circuit, a phase shift occurs as well between the voltage across the inductor VL and the current I. As the circuit is a resistive-inductive load, the voltage V leads the current I, as shown in Fig. 4. The phase shift can also be calculated using equation 5.

$$\theta = tan^{-1}\frac{V_L}{V_R} \text{ or } tan^{-1}\frac{X_L}{R} \quad (5)$$

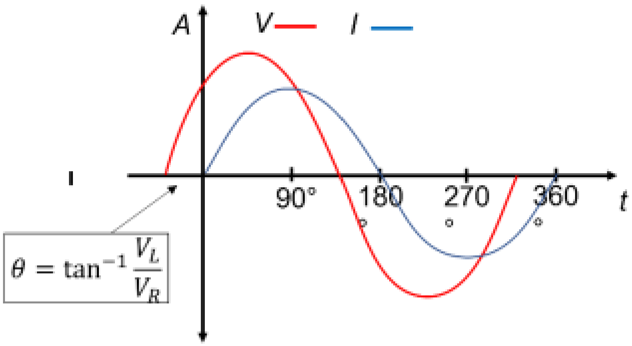


Figure 6: illustrates the voltage and current phase shift of a resistive-capacitive load.

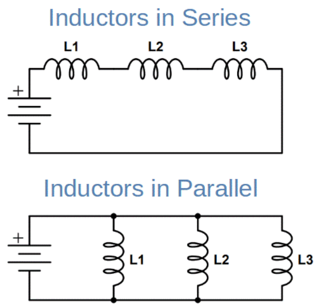
Table 1, shows important equations required to theoretically calculate V_R, V_L, Vs, R, XL, and Z.

For Voltage	For Impedance
$\lvert V_R\rvert = \lvert V_s\rvert \times cos(\theta)$	$\lvert R\rvert = \lvert Z\rvert \times cos(\theta)$
$\lvert V_L\rvert = \lvert V_s\rvert \times sin(\theta)$	$\lvert X_L\rvert = \lvert Z\rvert \times sin(\theta)$
$\lvert V_s\rvert = \sqrt{\lvert V_R\rvert^2 + \lvert V_L\rvert^2}$	$\lvert Z\rvert = \sqrt{\lvert R\rvert^2 + \lvert X_L\rvert^2}$

Key Eqs.	
The voltage of the inductor can be expressed by the following formula.	$v_L(t) = L\frac{di_L(t)}{dt}$
Inductor current can be expressed as follows.	$i_L(t) = 1/L\int_{-\infty}^{t} V(x)dx$
With rL being the internal resistance of the coil, the value of L can be calculated by the following formula.	$L = \frac{1}{2\pi f}\sqrt{\left(\frac{V_{RMS}}{I_{RMS}}\right)^2 - (r_L)^2}$
The power equation of the inductor is as follows.	$P_L(t) = I_L(t).V_L(t) = i_L(t)L\left(\frac{di_L(t)}{dt}\right)$
Since power is the variation of energy concerning time, the energy stored in the inductor is as follows.	$W_L(t) = \frac{1}{2}Li_L^2(t)$

The equivalent of n inductors connected in series is as follows.
Leq = L1 + L2 + L3 +…..+Ln

The equivalent of n inductors connected in parallel is as follows.
1/Leq = 1/L1 + 1/L2 + …..+1/Ln



PRELAB WORK

a. What are the applications of the RL circuit?

b. Simulate the circuit given in Figure 7 using the **Proteus** program and take its image with the oscilloscope image. (R=5 ohms, L=10 mH, 10VRMS, 50 Hz)

 i. V_R oscilloscope output?

 ii. V_L oscilloscope output?

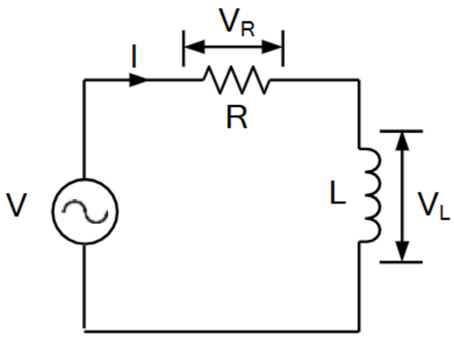


Figure 7

c. Calculate the X_L values according to the table below. Interpret the X_L value according to the frequency increase.

Freq (Hz)	L	X_L
50	2H	
500	2H	
1000	100mH	
2000	100mH	

3. Materials Used

- Function Generator
- Oscilloscope
- Resistors
- Inductors
- Multimeters
- Breadboard

4. Procedure

1- Build, and connect the circuit shown in Figure 8 using a 1kΩ resistor and a 100 uH inductor.

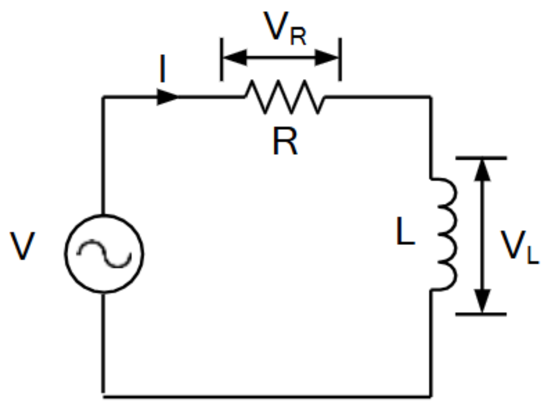


Figure 8

2- Set the input voltage at Vpp=4 V and frequency at 500 Hz.
3- Using the oscilloscope, read the voltage across the 1kΩ resistor and the 100 uH inductor.
4- Change the input frequency from 500 to 1000 Hz, 1500 Hz, 2000 Hz, 2500 Hz, and 3000 Hz.
5- Repeat step 3, measuring the voltage across the 1kΩ resistor and the 100 uH inductor.
6- Based on the experimental measurement, Calculate the phase shift (θ) between VR and VL theoretically using equation 5.
7- Write down all the measured and calculated values.

5. Evaluation of Results

Freq (Hz)	**500**	**1000**	**1500**	**2000**	**2500**	**3000**
I_R (mA)						
I_L (mA)						
V_R (V)						
V_L (V)						
Z_L (ohm)						
Calculated Phase shift (θ)						

6. Safety Precautions

- Ensure all power sources are turned off before assembling or modifying the circuit.
- Double-check all circuit connections to avoid short circuits or component damage.
- Use appropriate resistor and inductor values that can safely handle the applied voltage and current levels.
- Avoid touching live wires or terminals while the circuit is powered to prevent electric shock.
- Use the multimeter and oscilloscope probes correctly to prevent incorrect measurements or equipment damage.
- Keep the working area dry and free from clutter to minimize the risk of accidents.
- Follow the instructor’s guidelines and report any malfunctioning equipment immediately.
- Make sure the function generator output does not exceed the voltage ratings of the components used.

Experiment 3: OPAMP Characteristics

1. Objective of the Experiment

The objective of this experiment is to investigate the fundamental characteristics of an operational amplifier (Op-Amp) by examining parameters such as input offset voltage, input bias current, open-loop gain, slew rate, input and output impedance, and common-mode rejection ratio (CMRR). Through this experiment, students will gain a deeper understanding of the ideal and practical behavior of Op-Amps and observe how real-world deviations influence circuit performance.

2. Theoretical Background

An operational amplifier (Op-Amp) is a high-gain, direct-coupled amplifier with differential inputs and a single-ended output. Ideally, an Op-Amp possesses infinite input impedance, zero output impedance, infinite open-loop gain, and zero offset voltages. However, real Op-Amps deviate from these ideal conditions due to practical limitations of internal components.

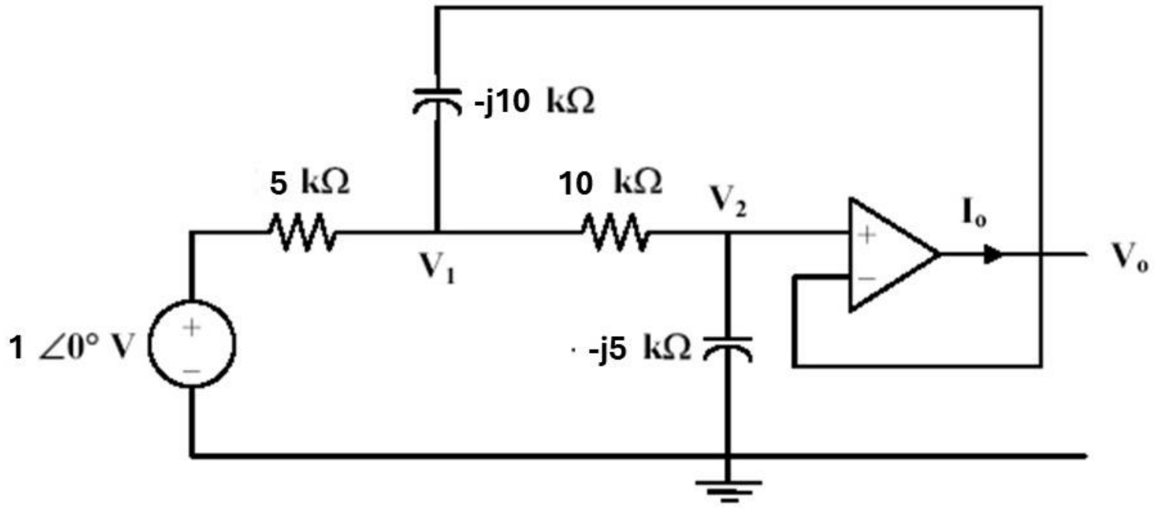
The key parameters that define the performance of an Op-Amp include:

- **Input Offset Voltage (Vos):** The differential DC voltage required between the inputs of the Op-Amp to make the output zero.
- **Input Bias Current (Ib):** The average of the DC currents entering the inverting and non-inverting terminals.
- **Open-Loop Gain (Aol):** The gain of the Op-Amp without any feedback, typically very high ($>10^5$).
- **Slew Rate (SR):** The maximum rate at which the output voltage can change, usually specified in V/µs.
- **Input and Output Impedance:** Practical Op-Amps have high but finite input impedance and low output impedance.
- **Common-Mode Rejection Ratio (CMRR):** The ability of the Op-Amp to reject common-mode signals, ideally infinite.

Understanding these characteristics is essential for accurate analog circuit design and ensures proper application of Op-Amps in amplifiers, filters, and signal processing systems.

PRELAB WORK

1- Solve the circuit below theoretically and calculate the values of V_0, I_0. f=800 Hz



3. Materials Used

Function Generator

Oscilloscope

Resistors

Capacitor

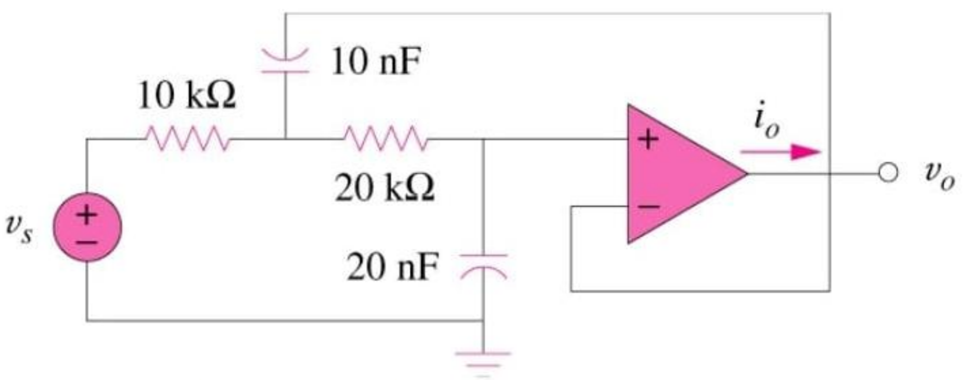
Multimeters

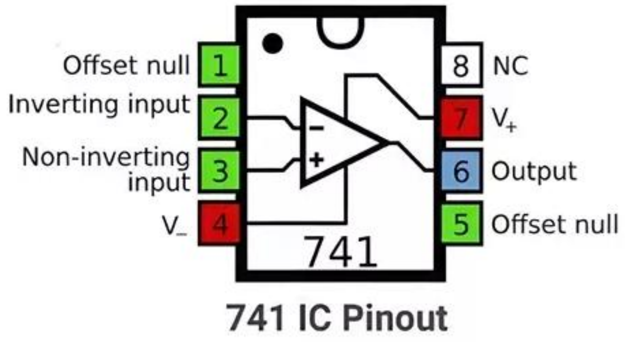
Breadboard

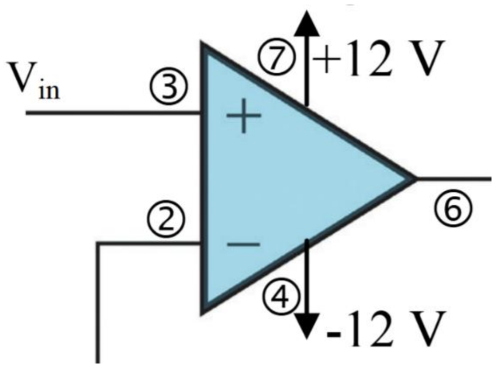
LM741-Opamp

4. Procedure

Build, and connect the circuit shown in Figure using 10kΩ, 20kΩ resistors, and a 20 nF capacitor.







Set the input voltage at Vpp=2 V and frequency at 800 Hz.

Using the oscilloscope, read the output voltage

Change the input frequency from 800 to 1000 Hz, 1500 Hz, 2000 Hz, 2500 Hz, and 3000 Hz.

Repeat step 3, measuring the output voltage.

Write down all the measured and calculated values.

5. Evaluation of Results

Freq (Hz)	**800**	**1000**	**1500**	**2000**	**2500**	**3000**
V_{max}(out)						
I_{0max}(out) ***Report***						

For Exp: I_{0max}(out) =50 sin (4000t) µA

6. Safety Precautions

- Write down all the calculated values.
- Interpret the relationship between the increase in frequency and the current values obtained.
- Always verify all connections before powering the circuit to prevent short circuits and component damage.
- Do not exceed the voltage or current ratings of any component used in the experiment.
- Use resistors with adequate power ratings to avoid overheating or burning.
- Ensure that capacitors are connected with correct polarity if they are electrolytic.
- Be cautious when handling live circuits; power off the supply before making changes.
- Avoid touching conductive parts while the circuit is energized.

Experiment 4: OPAMP Characteristics

1.Objective of the Experiment

The objective of this experiment is to investigate the fundamental characteristics of an operational amplifier (Op-Amp) by examining parameters such as input offset voltage, input bias current, open-loop gain, slew rate, input and output impedance, and common-mode rejection ratio (CMRR). Through this experiment, students will gain a deeper understanding of the ideal and practical behavior of Op-Amps and observe how real-world deviations influence circuit performance.

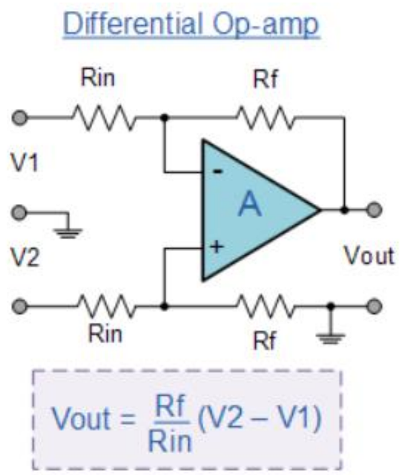
2. Theoretical Background

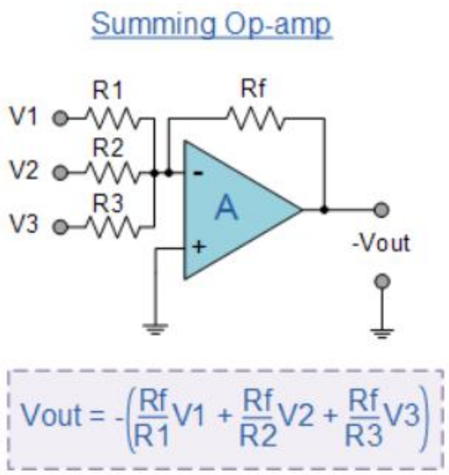
An operational amplifier (Op-Amp) is a high-gain, direct-coupled amplifier with differential inputs and a single-ended output. Ideally, an Op-Amp possesses infinite input impedance, zero output impedance, infinite open-loop gain, and zero offset voltages. However, real Op-Amps deviate from these ideal conditions due to practical limitations of internal components.

The key parameters that define the performance of an Op-Amp include:

- **Input Offset Voltage (Vos):** The differential DC voltage required between the inputs of the Op-Amp to make the output zero.
- **Input Bias Current (Ib):** The average of the DC currents entering the inverting and non-inverting terminals.
- **Open-Loop Gain (Aol):** The gain of the Op-Amp without any feedback, typically very high ($>10^5$).
- **Slew Rate (SR):** The maximum rate at which the output voltage can change, usually specified in V/µs.
- **Input and Output Impedance:** Practical Op-Amps have high but finite input impedance and low output impedance.
- **Common-Mode Rejection Ratio (CMRR):** The ability of the Op-Amp to reject common-mode signals, ideally infinite.

Understanding these characteristics is essential for accurate analog circuit design and ensures proper application of Op-Amps in amplifiers, filters, and signal processing systems.





3. PRELAB WORK

In below figure R1 = R2 = R3 = R4 = 1kΩ, V1 = 5∠0, V2 = 2∠0, Vout?

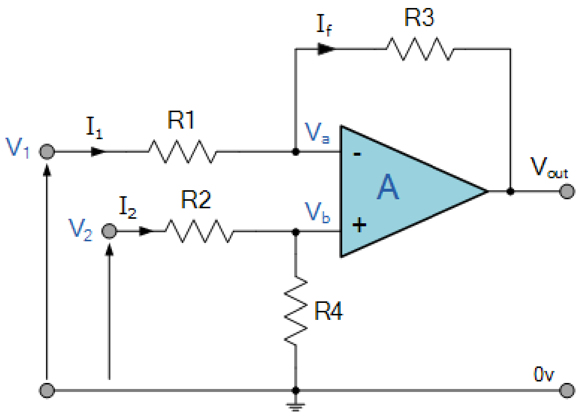


Figure 1

In below figure Rin = Rf = 1kΩ, V1 = 5∠0 (AC), V2 = 2∠0 (AC), V3 = 3V (DC), Vout?

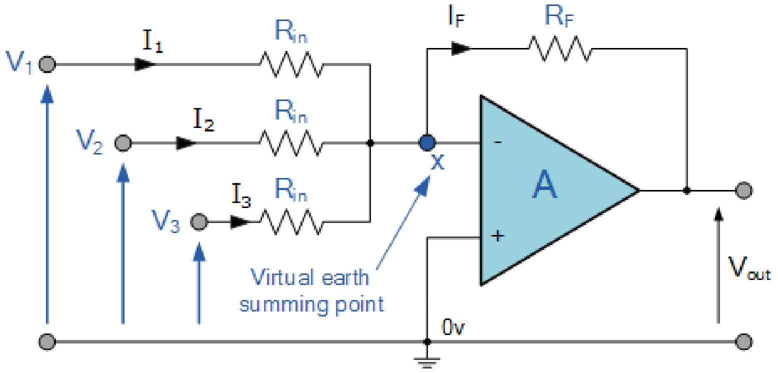


Figure 2

4. Materials Used

Function Generator

Oscilloscope

Resistors

Capacitor

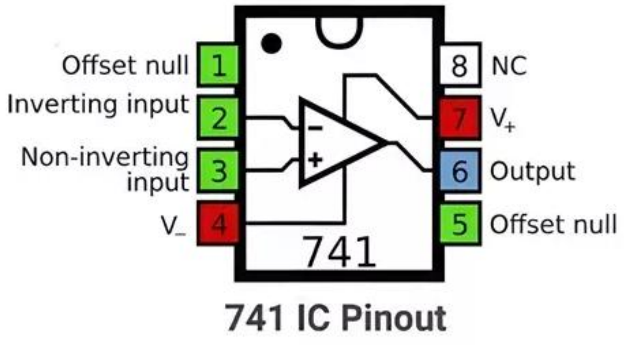
Multimeters

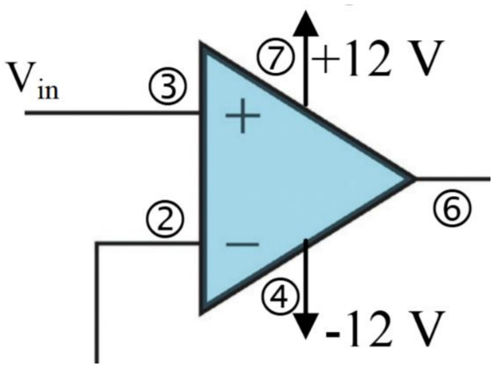
Breadboard

LM741-Opamp

5. Procedure

Build, and connect the circuit shown in Figure 1 with R1 = R2 = R3 = R4 = 1kΩ.
Set the input voltage values as in Table 1.
Using the oscilloscope, record the output voltage (screenshot, picture), and fill in Table 1.
Build, and connect the circuit shown in Figure 2 with Rin = Rf = 1kΩ.
Set the input voltage values as in Table 2.
Using the oscilloscope, record the output voltage (screenshot, picture), and fill in Table 2





6. Evaluation of Results

For Figure 1-Table 1

Values	Vo
V1=1∠0 V (Vmax) V2=2∠0 V (Vmax) Freq= 1kHz	
V1=3∠0 V (Vmax) V2=4∠0 V (Vmax) Freq= 1kHz	

For Figure 2 -Table 2

Values	Vo
V1=1∠0 V V2=1∠0 V V3=1∠0 V Freq= 1kHz	
V1=2∠0 V V2=2∠0 V V3=2∠0 V Freq= 1kHz	

7. Safety Precautions

- Write down all the calculated values.
- Interpret the relationship between the increase in frequency and the current values obtained.
- Always verify all connections before powering the circuit to prevent short circuits and component damage.
- Do not exceed the voltage or current ratings of any component used in the experiment.
- Use resistors with adequate power ratings to avoid overheating or burning.
- Ensure that capacitors are connected with correct polarity if they are electrolytic.
- Be cautious when handling live circuits; power off the supply before making changes.
- Avoid touching conductive parts while the circuit is energized.

Experiment 5: OPAMP-III

1.Objective of the Experiment

The objective of this experiment is to investigate the behavior of operational amplifier circuits configured as integrators and differentiators. By applying various waveform inputs such as square, sinusoidal, and triangular signals, students will observe how an op-amp processes these inputs based on its configuration and the values of surrounding passive components. The experiment aims to provide practical understanding of how op-amps can perform basic calculus operations, integration and differentiation, in analog signal processing. Students will gain hands-on experience in constructing and testing these configurations using the LM741 op-amp, visualizing their input-output relationships via oscilloscope, and interpreting the influence of signal frequency and waveform shape on circuit behavior.

2.Theoretical Background

Operational amplifiers (op-amps) can be configured to perform various mathematical operations on analog signals, including integration and differentiation. These functions are realized by strategically placing resistors and capacitors around the op-amp in specific configurations. In an integrator circuit, a resistor is placed at the input and a capacitor in the feedback path. When a time-varying input voltage is applied, the output of the op-amp is proportional to the negative integral of the input voltage over time. Mathematically, the output voltage V_{out} is given by:

$$V_{out} = -\frac{1}{RC}\frac{dV_{in}(t)}{d_t}$$

This configuration is useful in waveform generation, analog computation, and signal conditioning. In contrast, a differentiator circuit places a capacitor at the input and a resistor in the feedback path. In this case, the output voltage is proportional to the rate of change (derivative) of the input voltage. The mathematical expression is:

$$V_{out} = -RC\frac{dV_{in}(t)}{d_t}$$

This circuit responds strongly to rapid changes in input, making it sensitive to high-frequency components and ideal for edge detection and signal analysis applications. These behaviors assume ideal op-amp conditions, such as infinite gain, infinite input impedance, and zero output impedance, but in practice, real components (e.g., LM741) introduce limitations such as bandwidth constraints and noise sensitivity. As illustrated in Fig. 5.1, integrator and differentiator circuits can be constructed using the LM741 op-amp, where the choice of input waveform (e.g., square, triangular, sinusoidal) directly affects the resulting output shape. By observing these responses, students can develop intuition about the dynamic behavior of analog systems and the practical implementation of calculus-based operations in electronic circuits.

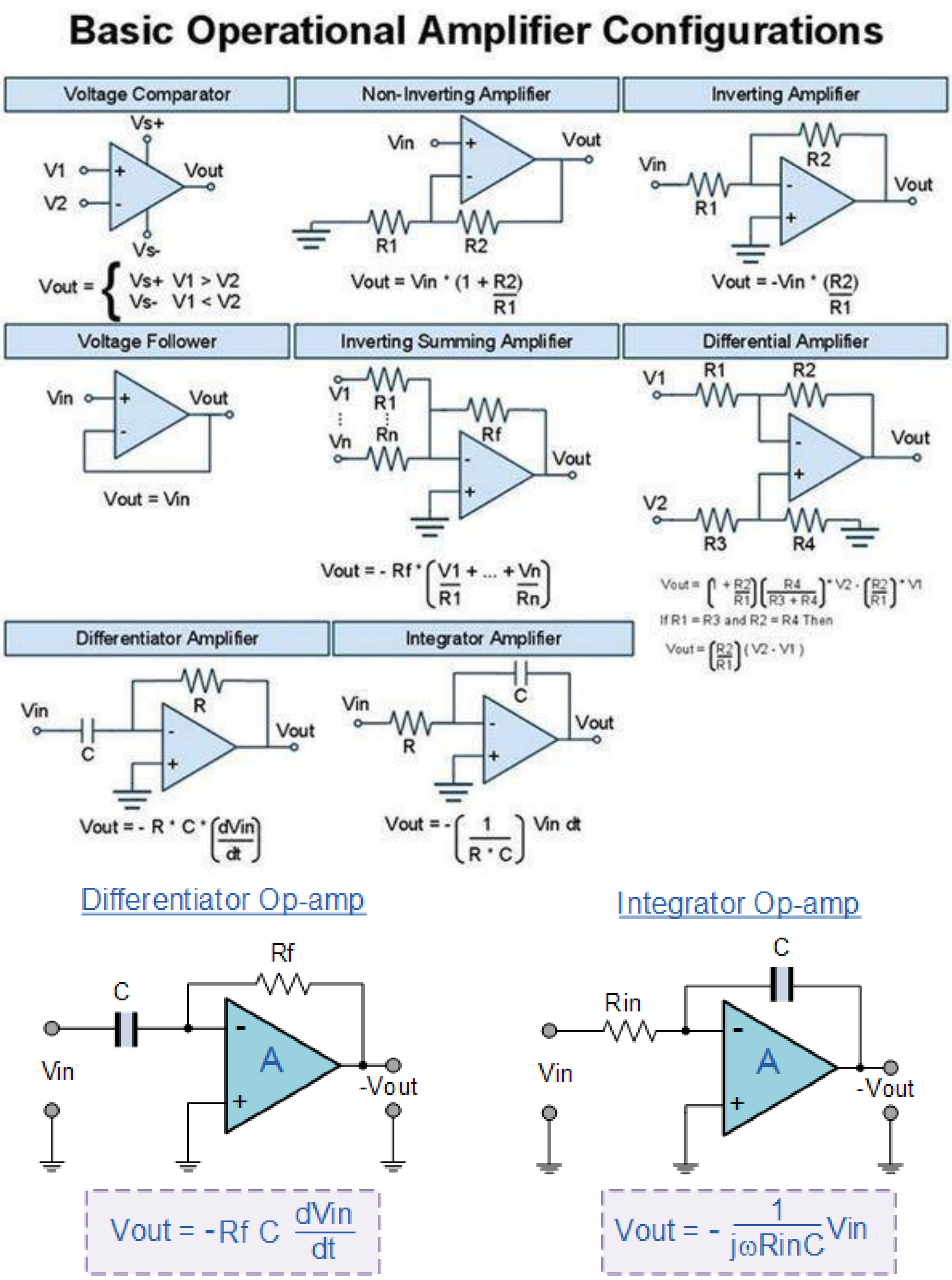


Fig. 5.1. Circuit diagrams showing the standard op-amp configurations for integration and differentiation using resistors and capacitors.

3. Materials Used

- Function Generator
- Oscilloscope
- One 10kΩ resistor, one 0.01µF capacitor and one LM741-Opamp
- Multimeters
- Breadboard

4.Preliminary (PreLab)

In the fig.5.2 , V_g=cos(100t) V, V_{out}?

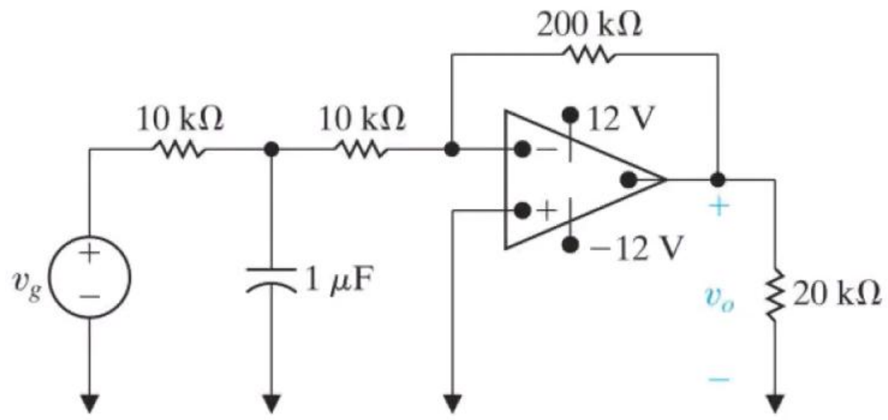


Fig. 5.2. Circuit for prelab section.

In the fig.5.3 ω=50 rad/s, V_{out}?

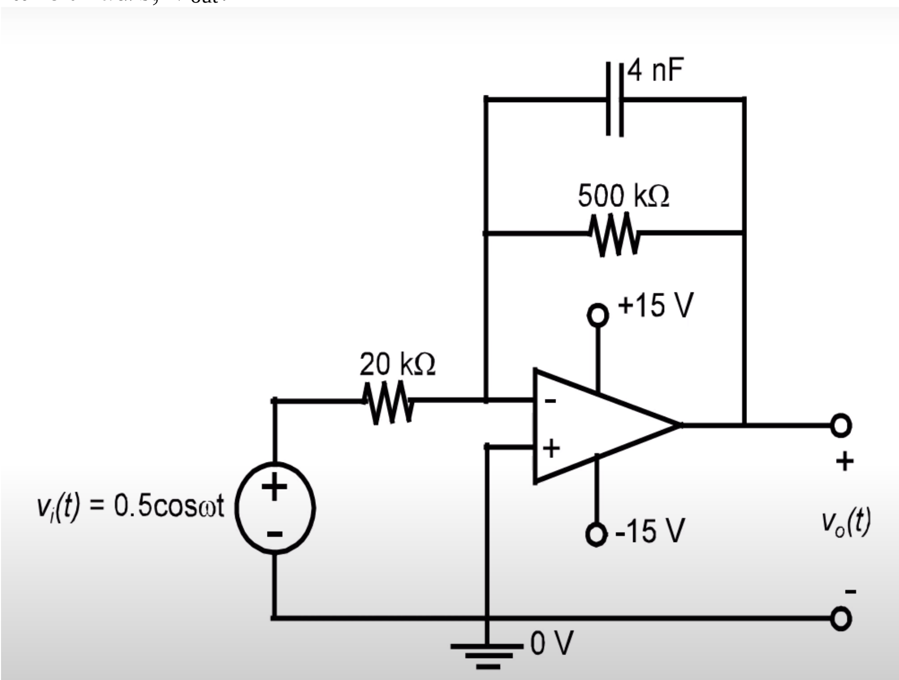


Fig. 5.3. Circuit for prelab section.

5.Procedure

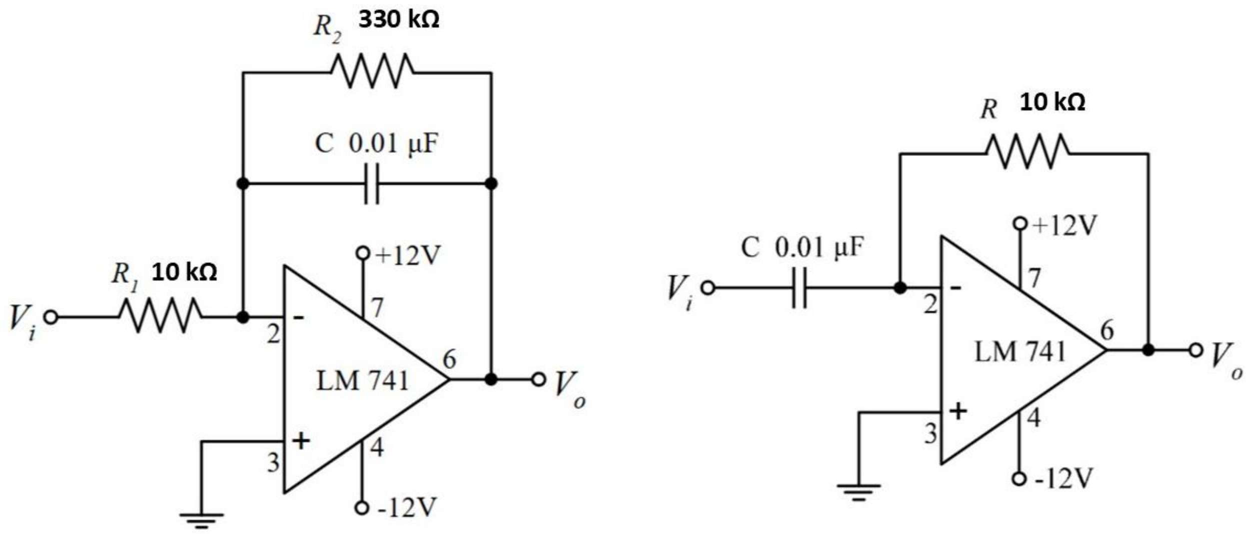
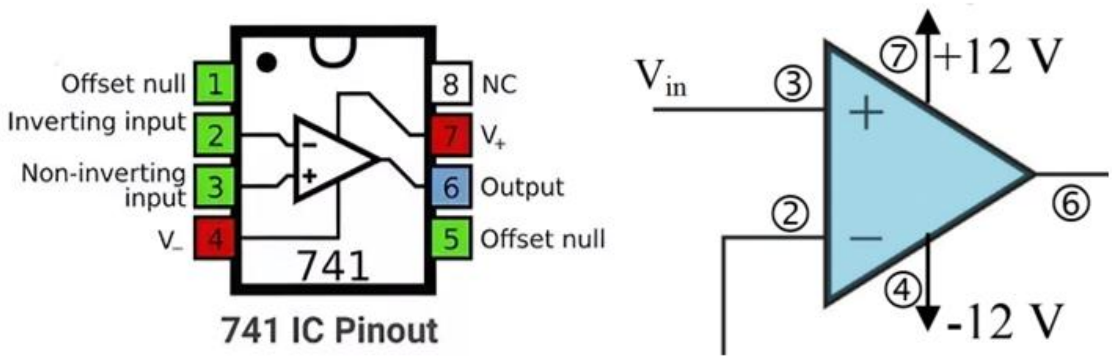


Fig. 5.4. Experimental circuits of integrator (left) and differentiator (right) configurations using the LM741 operational amplifier.

1. Build and connect the circuit shown in Fig 5.4. with R_1 = 10 kΩ and R_2 = 330 kΩ.
2. Apply the input voltage V_i as given in the table, observe the input and output using an oscilloscope, and record the output with a screenshot or picture.
3. Similarly, apply V_i and repeat step 2 for the differentiator circuit.



LM741 operational amplifier pin configuration and symbolic representation showing power supply, input, and output terminals.

6.Evaluation of Results

Put the visuals of the oscilloscope screen taken during the experiment.

Table 1. Oscilloscope waveforms obtained from the experiment

Values	**Integrator**	**Differentiator**
	V_0	**V_0**
Square Vin= 10 Vpp Freq = 500 Hz		
Square Vin= 10 Vpp Freq = 1 kHz		
Square Vin= 10 Vpp Freq = 1.5 kHz		
Sinusoidal Vin= 10 Vpp Freq= 500 Hz		
Sinusoidal Vin= 10 Vpp Freq = 1 kHz		
Sinusoidal Vin= 10 Vpp Freq = 1.5 kHz		
Triangular Vin= 10 Vpp Freq = 500 Hz		
Triangular Vin= 10 Vpp Freq = 1 kHz		
Triangular Vin= 10 Vpp Freq = 1.5 kHz		

7. Safety Precautions

- Ensure all circuit connections are correct before powering the circuit to avoid short circuits or damage to components.
- Always turn off the DC power supply before modifying the circuit or changing component values on the breadboard.
- Use appropriate resistor values as specified, and avoid exceeding their power ratings to prevent overheating.
- Do not touch conductive parts while the circuit is powered; even low voltages can cause burns or shocks in case of faulty connections.
- Place measurement probes securely when using a multimeter to avoid accidental slips that may cause short circuits.
- Keep the work area dry and organized, and handle all instruments with dry hands to minimize the risk of electrical accidents.
- Check power supply voltage settings before connecting to the circuit to ensure they match the experiment requirements.

Exp-6: PASSIVE FILTERS (HIGH, LOW, BAND PASS)

Objective:

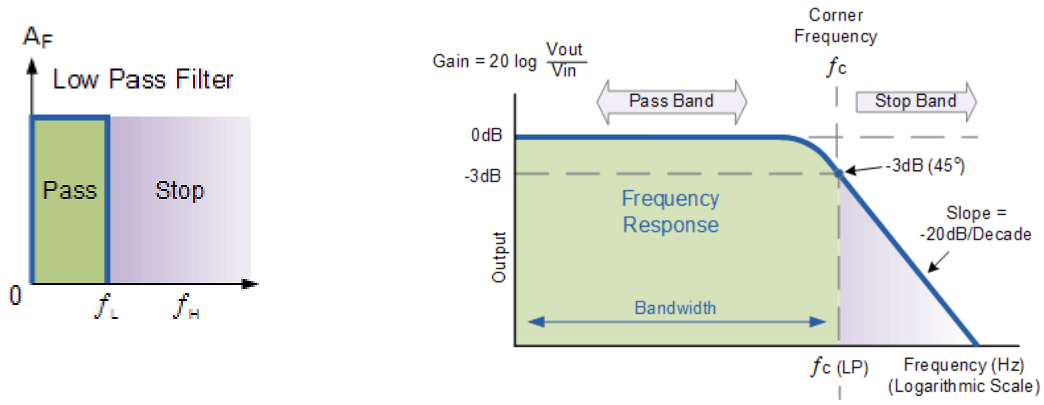
The objective of this experiment is to analyze the frequency response of a series-parallel RLC circuit under alternating current (AC) excitation. Specifically, the aim is to investigate the behavior of the circuit near its resonance condition by measuring the output voltage across a parallel LC branch as the input frequency varies. Through both theoretical calculation and experimental measurement, the resonance Frequency at which the circuit exhibits maximum impedance and voltage gain will be identified and compared. This experiment also reinforces the understanding of AC circuit analysis, resonance behavior, and the practical use of instruments such as signal generators and oscilloscopes in observing sinusoidal waveforms and voltage variations.

Required Equipment:

- Function Generator
- Oscilloscope
- 1 kΩ, 2 kΩ Resistors
- 1 uF, 2 uF Capacitor
- Breadboard

Theoretical Background:

a. Low Pass Filters



a **b**

Figure 1: a) Ideal Low Pass Filter frequency response, b) Realistic Low Pass Filter frequency response

A low pass filters output readings should be high in amplitude when applied input has a low frequency components, vice-versa output readings should be low when applied input has high frequency components shown in Figure 1. To obtain this characteristic a circuitry given in Figure 2 can be used.

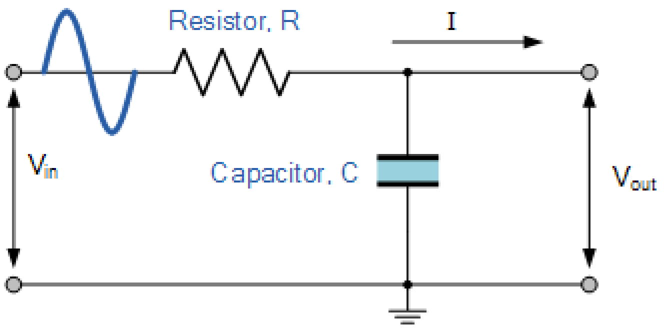
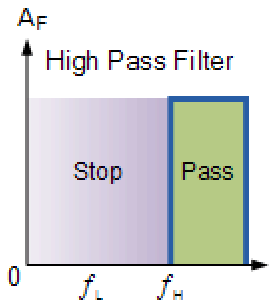


Figure 2: Low Pass Filter Circuitry Schematic

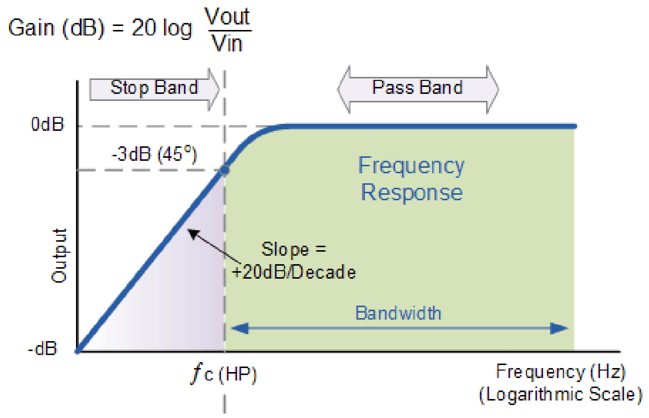
Cutoff frequency (f_c) can be found via Eq. 1.

$$f_c = \frac{1}{2\pi RC} \qquad \text{Eq. 1}$$

b. High Pass Filters



a



b

Figure 3: a) Ideal High Pass Filter frequency response, b) Realistic High Pass Filter frequency response

A high pass filters output readings should be low in amplitude when applied input has a low frequency components, vice-versa output readings should be high when applied input has high frequency components shown in Figure 3. To obtain this characteristic a circuitry given in Figure 4 can be used.

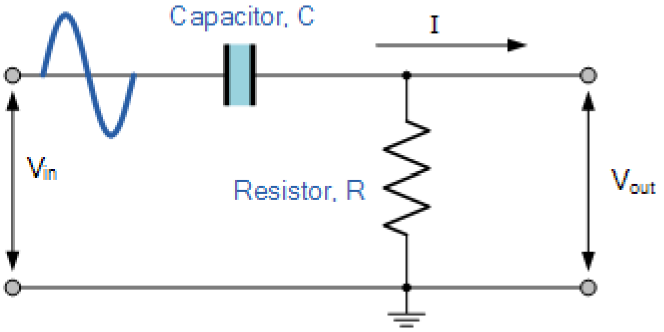
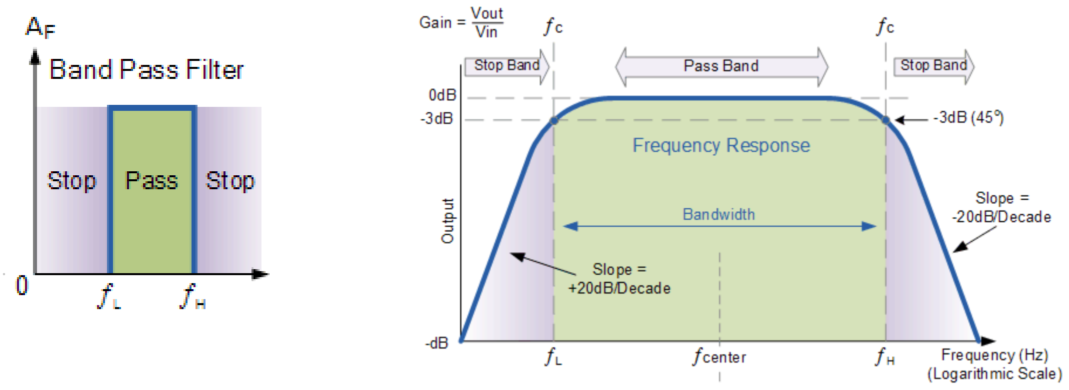


Figure 4: High Pass Filter Circuitry Schematic

Cutoff frequency (fc) can be found via Eq. 1, as well.

c. Band Pass Filters



a **b**

Figure 5: a) Ideal High Pass Filter frequency response, b) Realistic High Pass Filter frequency response

Band Pass Filters are combinations of a low pass filter and a high pass filter as cutoff frequency of low pass filter (f_L) is being smaller than cutoff frequency of high pass filter (f_H). The circuitry given in Figure 6 provides a band pass filter characteristic shown in Figure 5.

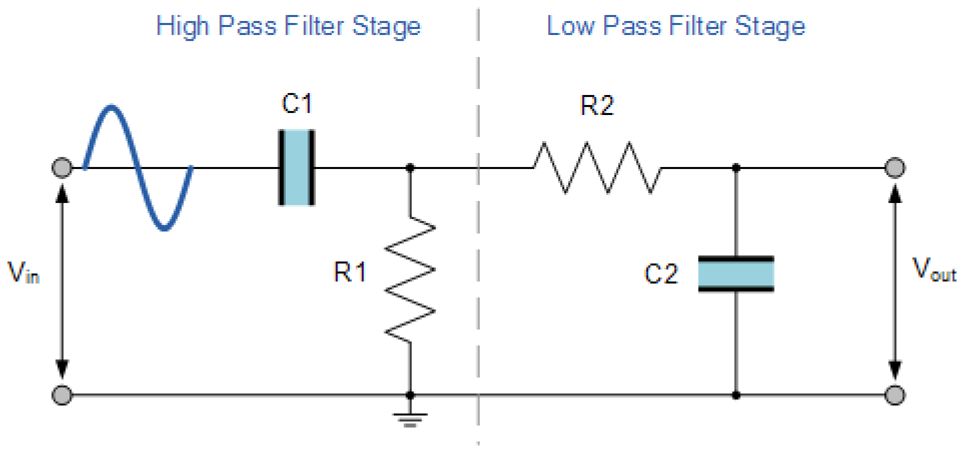


Figure 6: Band Pass Filter Circuitry Schematic

Pre-Lab Preperation

Before coming to the laboratory, complete the following tasks and bring your results with you:

1. Calculate the theoretical cutoff frequency (fc) using the formula for given R and C values below:
   a. R = 1 kΩ, C = 1 μF
   b. R = 1 kΩ, C = 2 μF
   c. R = 2 kΩ, C = 1 μF
   d. R = 2 kΩ, C = 2 μF
2. Calculate the resistance values (R1-R2) for a bandpass filter for circuitry given in Fig. 6 with the values given below:
   a. LF = 2 kHz, HF = 3kHz
   b. C1 = 1 μF, C2 = 2 μF

Experimental Procedure

1. Build and connect the circuit shown in Figure 2 with R = 1 kΩ and C = 1 uF.
2. Apply the input voltage Vin as given in the Table 1, observe the input and output using an oscilloscope, and record the output voltage peak to peak value to the Table 1.
3. Build and connect the circuit shown in Figure 4 with R = 1 kΩ and C = 1 uF and repeat step 2 for Table 2.
4. Build and connect the circuit shown in Figure 6 with R1 = 1 kΩ, R2 = 2 kΩ, C1 = 1 uF, and C2 = 2 uF and repeat step 2 for Table 3.

Results

Put the visuals of the oscilloscope screen taken during the experiment.

Table 1. Result of Low Pass Filter

Vin (4 Vpp sine)	Vout (Vpp)
10Hz	
20Hz	
50Hz	
100Hz	
200Hz	
500Hz	
1000Hz	

Table 2. Result of High Pass Filter

Vin (4 Vpp sine)	Vout (Vpp)
10Hz	
20Hz	
50Hz	
100Hz	
200Hz	
500Hz	
1000Hz	

Table 3. Result of Band Pass Filter

Vin (4 Vpp sine)	Vout (Vpp)
10Hz	
20Hz	
50Hz	
100Hz	
200Hz	
500Hz	
1000Hz	

Conclusion

In this experiment, the frequency response of passive RC-based low pass, high pass, and band pass filters was examined. It was observed that the output voltage amplitude varied significantly with input frequency, confirming the theoretical behavior of each filter type. The low pass filter allowed low-frequency signals while attenuating high frequencies; the high pass filter behaved in the opposite way. The band pass filter showed maximum output within a specific frequency band. The measured cutoff frequencies were consistent with the calculated theoretical values, verifying the expected performance of the filters. These results demonstrate the practical applications of basic RC filters in signal conditioning and frequency selection.

Discussion Questions

1. Why does the amplitude of the output voltage decrease significantly beyond the cutoff frequency in a low pass filter, and what would happen to the cutoff frequency if the capacitor value were increased?
2. In a high pass filter, what role does the resistor play in shaping the filter's frequency response, and how would doubling the resistance affect the cutoff frequency?
3. How does combining a high pass and a low pass filter result in a band pass filter, and what determines the bandwidth of such a filter?
4. What are the limitations of passive filters compared to active filters in real-world signal processing applications?
5. If noise appears at high frequencies in a circuit, which filter type would be most appropriate to suppress it, and why?

Exp-7: ANALYSIS OF RLC CIRCUIT IN ALTERNATING CURRENT (AC)

Objective:

The objective of this experiment is to analyze the frequency response of a series-parallel RLC circuit under alternating current (AC) excitation. Specifically, the aim is to investigate the behavior of the circuit near its resonance condition by measuring the output voltage across a parallel LC branch as the input frequency varies. Through both theoretical calculation and experimental measurement, the resonance Frequency at which the circuit exhibits maximum impedance and voltage gain will be identified and compared. This experiment also reinforces the understanding of AC circuit analysis, resonance behavior, and the practical use of instruments such as signal generators and oscilloscopes in observing sinusoidal waveforms and voltage variations.

Required Equipment:

- AC Power Supply
- Oscilloscope
- 1kΩ Resistor
- 220nF Capacitor
- 47mH Inductor

Theoretical Background:

Figure 1 shows a series-parallel RLC circuit powered by an AC power source. This circuit also functions as a **band-pass filter** with a **parallel resonance circuit.** The parallel LC circuit, also known as a **tank circuit**, behaves like a voltage divider when placed in series with the resistor. The output voltage is measured across the parallel LC branches.

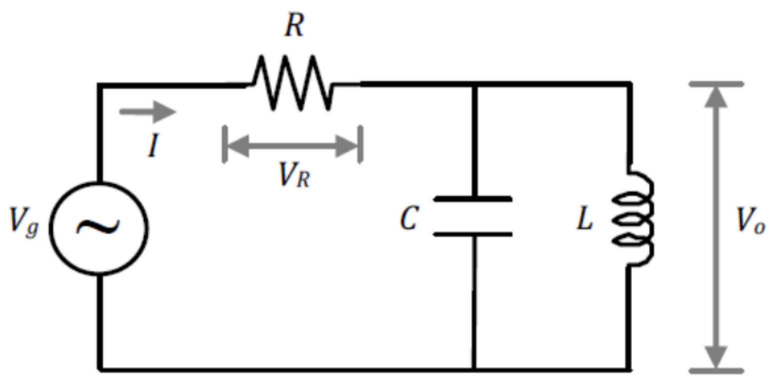


Figure 1: Series-Parallel RLC Circuit

The capacitive reactance X_C, inductive reactance X_L, and therefore the impedance of the parallel LC circuit vary with frequency. At a specific frequency called the resonance frequency (f_0), X_L equals X_C, and the parallel LC circuit operates in resonance. The resonance frequency is defined by the following equation:

$$f_0 = \frac{1}{2\pi\sqrt{LC}} \quad \text{Eq. 1}$$

At resonance, the parallel LC circuit exhibits maximum impedance. Under this condition, the circuit current reaches its minimum value (ideally zero). Therefore, when the frequency of the input voltage matches the resonance frequency, the impedance of the parallel LC circuit and the voltage across its terminals become maximum. At resonance, the amplitude of the output voltage taken from the terminals of the parallel LC circuit becomes approximately equal to the input voltage. The variations of the output voltage for different input frequencies are shown in Figure 2.

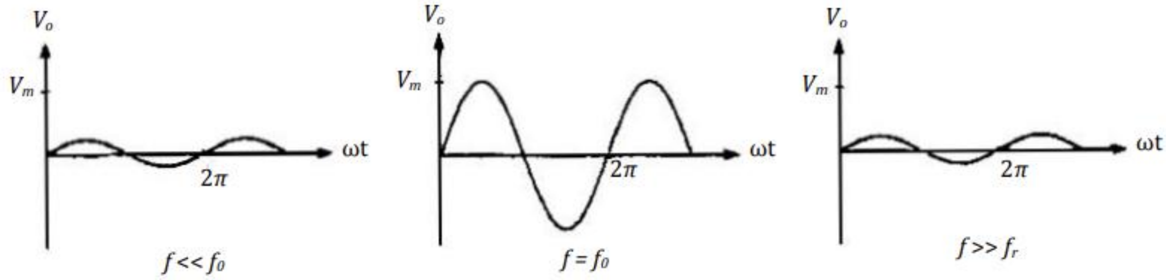


Figure 2: Output voltage variation of the parallel LC circuit with respect to different input frequencies

Phasor Expression of the Output Voltage:

$$\overline{V_0} = \frac{\overline{Z_p}}{R + \overline{Z_p}}\overline{Vg} \qquad \text{Eq. 2}$$

$$\bar{Z}_p = Z_L // Z_C = \frac{jX_L(-jX_C)}{jX_L - jX_C} \qquad \text{Eq. 3}$$

Pre-Lab Preperation

1. For the RLC circuit shown in Figure 3, consider an AC voltage source with a frequency of f = 1 kHz, amplitude of V = 5 V RMS, and component values R = 1 kΩ, L = 47 mH, and C = 220 nF:
   a) Calculate the RMS value of the output voltage (V_o) at the given frequency.
   b) Calculate the resonance frequency (f_0) of the circuit.
   c) Calculate the maximum RMS output voltage V_{omax} at the resonance frequency.

Experimental Procedure

5. Construct the series-parallel RLC circuit as shown in Figure 3 using the components:
   R = 1 kΩ, L = 47 mH, and C = 220 nF.
6. Set the function selector of the signal generator to the sine wave position.
7. Connect Channel 1 (CH1) of the oscilloscope to the output of the signal generator.
8. Using the amplitude and frequency controls on the signal generator, adjust the signal to produce a sinusoidal waveform with a frequency of 1 kHz and an RMS voltage of 5 V. Observe the waveform on the oscilloscope screen.
9. Connect the adjusted AC signal to the input of the RLC circuit.
10. To observe the input voltage V_g and output voltage V_O waveforms on the oscilloscope, connect the oscilloscope as shown in Figure 3:
    - Connect CH1 to node a,
    - Connect CH2 to node b,
    - Connect the GND terminals to node c of the circuit.

11. Measure the peak-to-peak value of the output voltage V_{oPP} from the oscilloscope screen and record it in Table 1.
12. Gradually increase the frequency of the signal generator from a low value until the output voltage reaches its maximum peak-to-peak amplitude $V_{oPP.}$ From the oscilloscope, determine the resonance frequency (f_0) and record both values in Table 1.
13. Calculate the resonance frequency (f_0) using Equation (1) and record the result in Table 1.
14. Compare the measured and calculated resonance frequency values.
15. Based on your pre-lab calculations, compare the results with the measured values in Table 1, and provide your interpretation and analysis.

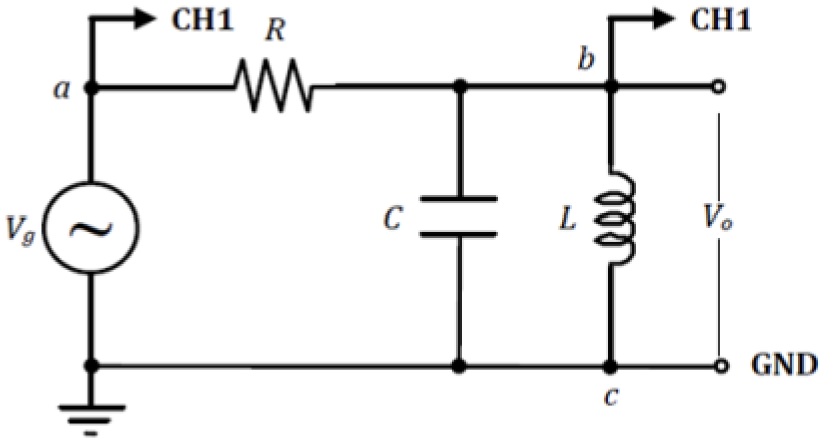


Figure 3: Circuit Diagram for Experimental Measurements

Results

Table 1:Theoretical Calculations and Measurement Data

Measured Output Voltage V_{oPP} **(V)**		**Resonance Frequency** f_0 **(Hz)**	
f=1kHz	$f = f_0$	**Calculated**	**Measured**

Conclusion

The experiment focused on analyzing a series-parallel RLC circuit in AC and identifying its resonance behavior. The resonance frequency was both theoretically calculated and experimentally measured. Results showed that at resonance, the impedance of the LC branch was maximized, and the current through the circuit minimized. This condition caused the output voltage across the LC branch to reach its peak value, confirming the theoretical expectation. The measured resonance frequency closely matched the calculated value, validating the theoretical model and demonstrating how RLC circuits can be used for selective frequency amplification in practical applications.

Discussion Questions

1. What would be the effect on the resonance frequency and the sharpness of the resonance peak (bandwidth) if the resistance in the circuit were increased?
2. Why is the output voltage at its maximum at resonance in a series-parallel RLC circuit, and how is this behavior linked to impedance?
3. What is the significance of using a parallel LC branch instead of a series LC in this circuit configuration?
4. In practical implementations, what sources of error could affect the accuracy of the measured resonance frequency?
5. How could this type of RLC circuit be used in real-world applications such as wireless communication or audio electronics?

Experiment 8: 3-Phase AC Δ-Δ Connected Circuits

1.Objective of the Experiment

The objective of this experiment is to analyze the behavior of a balanced three-phase AC circuit composed of inductive (L) and resistive (R) components. Using sinusoidal sources with a 120° phase shift and equal amplitude, the experiment aims to measure the line and phase currents, verify phase relationships, and observe the impact of inductance and resistance on the overall current and power distribution in the circuit. This setup helps students gain practical understanding of three-phase systems, phasor relationships, and the interaction between resistive and inductive loads under AC excitation.

2.Theoretical Background

A three-phase AC system is a widely used method for electric power generation and distribution, especially in industrial environments. In this system, three alternating voltages of equal magnitude and frequency are phase-shifted by 120° from one another. This arrangement ensures continuous power delivery and better efficiency compared to single-phase systems. In the experimental setup (see Fig.8.1), each phase includes a resistor and an inductor connected in series. When AC voltage is applied to such a circuit, the current does not align perfectly with the voltage. This is due to the inductor causing a delay in the current waveform, a phenomenon known as phase shift. As a result, the current lags the voltage in each phase. The current in each branch of the circuit is determined by applying Ohm's Law, considering the resistive and inductive components. Since the voltage sources are balanced and phase-shifted by 120°, the phase currents will also be balanced and equally phase-shifted. This experiment helps students understand how three-phase systems operate, how resistive and inductive loads behave under AC excitation, and how phase shifts influence the current and power in each branch of the system.

3. Materials Used

- Function Generator
- Oscilloscope
- Three 50 mH inductors and three 600 Ω resistors.
- Multimeters
- Breadboard

4. Procedure

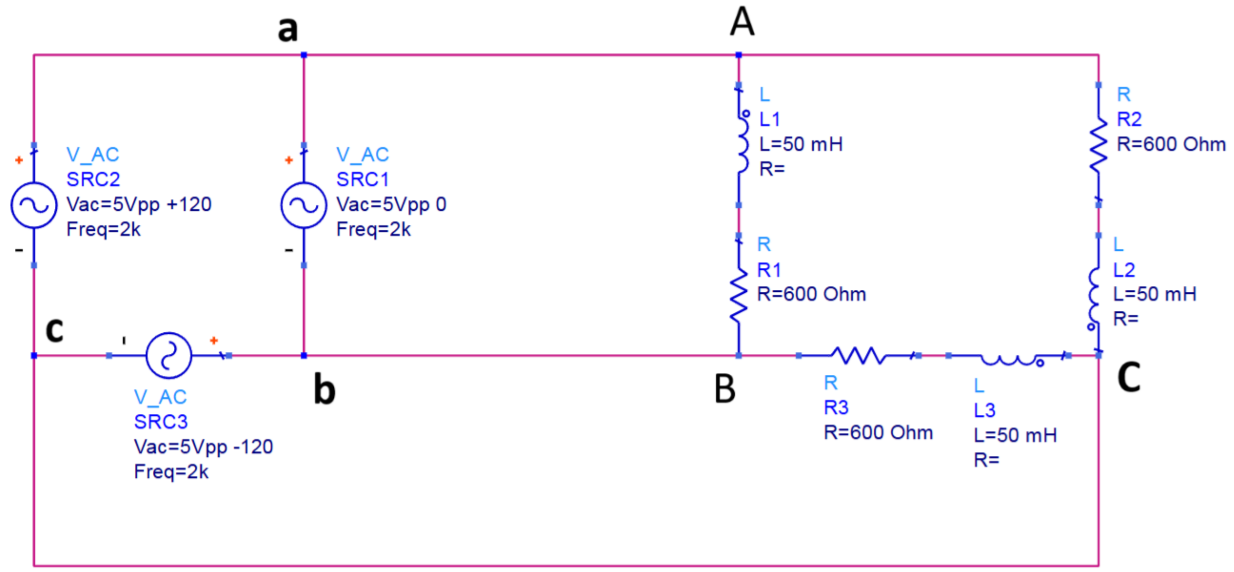


Fig. 5.4. Circuit for experiment.

1. Connect three function generators to produce 3-phase sinusoidal signals as shown in Fig. 8.1:

- v_a =5Vpp ∠0 ∘
- v_b=5V pp ∠−120 ∘
- v_c=5Vpp ∠+120 ∘
- All at a frequency of 2 kHz.

2. Construct the three-phase circuit using:
   - Three inductors, each with L=50mH
   - Three resistors, each with R=600Ω(or two 330 Ω)
3. Make the connections according to Fig. 8.1, ensuring the correct phase sequence and node labeling (a–b–c for sources, A–B–C for the load side).
4. Power on the function generators and verify the phase shift between the waveforms using an oscilloscope.
5. Measure the line currents I_{AA}, I_{BB}, and I_{CC}.
6. Measure the voltages V_{AB}, V_{BC}, V_{CA}.
7. Record all voltage and current values.

5. Evaluation of Results

Tabulate the data obtained during the tests.

.

Table 1. Oscilloscope waveforms obtained from the experiment

	V_{AB}	V_{BC}	V_{CA}
V(Volt)			

Table 2. Oscilloscope waveforms obtained from the experiment

	I_{AA}	I_{BB}	I_{CC}
I(mA)			

6. Safety Precautions

- Ensure all circuit connections are correct before powering the circuit to avoid short circuits or damage to components.
- Always turn off the DC power supply before modifying the circuit or changing component values on the breadboard.
- Use appropriate resistor values as specified, and avoid exceeding their power ratings to prevent overheating.
- Do not touch conductive parts while the circuit is powered; even low voltages can cause burns or shocks in case of faulty connections.
- Place measurement probes securely when using a multimeter to avoid accidental slips that may cause short circuits.
- Keep the work area dry and organized, and handle all instruments with dry hands to minimize the risk of electrical accidents.
- Check power supply voltage settings before connecting to the circuit to ensure they match the experiment requirements.

REFERENCES

- Alexander, C. K., Sadiku, M. N., & Sadiku, M. (2007). Fundamentals of electric circuits (pp. 34-39). Boston, MA, USA: McGraw-Hill Higher Education.

- Laboratory Manual – Circuit Analysis I, Department of Electrical and Electronics Engineering.