EE-110: Circuit Analysis & Design

(Lab Manual)

Electrical Engineering Department, University of Engineering & Technology Lahore, Pakistan.

LABORATORY SAFETY RULES

(Safety Committee, Electrical Department UET Lahore)

The following general rules and safeguards should be followed always in the EED laboratories.

Administrative

1. Do not block access to these electrical panels/boards and shut-off switches.
2. Smoking is prohibited.
3. No food or drinks allowed.
4. Do not run or engage in reckless behavior.
5. Wear Personal Protective Equipment (PPE) if required.
6. Work space should be clear of unnecessary material such as extra books, papers, purses, and clothes.
7. Never wear rings, watches, bracelets, necklaces, or other electrically conductive jewelry.
8. No chatting or gaming is allowed
9. Inappropriate use of Internet (chatting, gaming and pornography etc.) is strictly prohibited.

Electrical safety

1. Working alone and unsupervised is forbidden unless getting permission from HSE incharge, security incharge, and safety officer.
2. When checking an operating circuit, keep one hand either in a pocket or behind your back to prevent current from passing through your chest cavity and injuring your heart.
3. Be familiar with the electrical hazards associated with your workplace.
4. Avoid contacting circuits with wet hands or wet materials.
5. Use electrical cords only if they are in good condition.
6. Avoid being grounded. Stay at least 6 inches away from all metal materials, walls, and water sources.
7. When unplugging a power cord, pull on the plug, not on the cable.
8. Follow lockout-tag out safety procedure.
9. Equipment found to be faulty in any way should be reported to the lab incharge immediately.
10. When attaching a high voltage power supply always switch off the supply.
11. When disassembling a circuit, first remove the source of power.

Emergency Response

1. Be familiar with emergency phones, fire alarm posters, emergency exit routes, first aid boxes, evacuation plans, eye wash, assembly points, and location of fire extinguishers.
2. Inform your lab instructor immediately after any incident, injury, fire or explosion.

Common Sense

1. It is better not to touch anything with which you are not familiar.
2. Your personal laboratory safety depends mostly on you.

Issued By: HSE Incharge Approved By: Chairman

ELECTRICAL SAFETY RULES FOR ELECTRICAL ENGINEERING LABORATORIES

(Safety Committee, Electrical Department UET Lahore)

1. For experiments where the voltages are 50 V AC (RMS)/DC or higher there should be at least two students in the laboratory and they must be supervised by either an instructor or a laboratory staff.
2. Switch off power by operating an associated switch or a circuit breaker before working on a circuit. It is a good practice to unplug the power cord from the source.
3. Complete your wiring, think about it, discuss it with your partners and re-check before switching power on. If unsure, ask an instructor or a laboratory staff.
4. Any alteration/modification in the circuit must only be done after the power to the circuit has been switched off (point 2).
5. When working with energised circuits, always use one hand: the other hand must be at your back. Think about this: it is absolutely important for your safety. It will save from possible electrocution.
6. After completing an experiment, switch off the power before dismantling the circuit.
7. Switch off power before checking or replacing a fuse. Identify and correct the cause of a blown-up fuse or a tripped circuit breaker before replacing the fuse or re-setting the circuit breaker.
8. Do not use damaged cords/leads, cords/leads which become hot or cords/leads with exposed wiring. Report to the laboratory staff if this happens.
9. If measurements must be made on live or energised circuits, use well-insulated meter probes. Remember: work with only one hand!
10. Use extension cords when necessary, and only on a temporary basis. Do not join leads together to make a longer lead/connection in a circuit.
11. Do not come to a laboratory wearing a chappal or slippers. The students working with electrical machines or with other rotating parts should not wear loose clothes.
12. Do not bring and consume edible items and drinks/beverages in a laboratory.
13. Avoid working with wet hands and clothing.
14. You should remove loose metallic bangles, bracelets, necklaces, ear rings and watchstraps before working on an electrical circuit. You should not have long loose hair as well!
15. Always check the electrical ratings of the equipment you work on and make sure you operate it within its ratings.
16. Never over-load an electrical circuit.
17. The fuses and circuit breakers must never be by-passed. Never replace a low-current fuse with a higher current rating fuse.
18. Make sure chassis or cabinets are grounded.
19. Safely discharge capacitors in equipment before working on a circuit.

Electrical Emergency Response

- It is the duty of a laboratory director, instructor and laboratory staff to make the students aware of the available Emergency Power-Off arrangement in their laboratory, and when and how they should operate it.

Dr. Muhammad Asghar Saqib
Convener, Department's Safety Committee

University of Engineering and Technology Lahore

Section Course Outline Report

Department: Electrical Engineering **Printed Date:** August 17, 2024

Section Course Detail	
Semester	FALL 2024
Department	Electrical Engineering
Section	A
Subject Title	EE-110L Circuit Analysis & Design
Subject Domain	Engineering
Subject Knowledge	Engineering Foundation
Contact	rabia.nazir@uet.edu.pk

Measureable Student Learning Outcomes

CLOs	Description	PLOs	Domain	Domain Level	Assessments
CLO1	Demonstrate the basic concepts related to RC, RL and RLC circuits.	PLO01	Psychomotor	3. Precision	null
CLO2	Demonstrate the basic operation of passive and active filters.	PLO02	Psychomotor	3. Precision	null
CLO3	Simulate and develop a filter with given specifications.	PLO09	Psychomotor	4. Articulation	null

Class Timings

Section Content

Week (Lec)	Topics	CLO's
week1	Introduction to Laboratory Equipment	CLO1
week2	Experiment 1: (a) To examine the pulse response of a series RL network	CLO1

University of Engineering and Technology Lahore

Section Course Outline Report

Department: Electrical Engineering **Printed Date:** August 17, 2024

Section Content		
Week (Lec)	**Topics**	**CLO's**
week3	Experiment 1: (b) To examine the pulse response of a series RL network	CLO1
week4	Experiment 2: (a) To examine the steady state sinusoidal response of RL and phasors	CLO1
week5	Experiment 2: (b) To examine the steady state sinusoidal response of RC and phasors	CLO1
week6	Experiment 3: To determine the resonant frequency of a series & parallel RLC circuit.	CLO1
week7	Experiment 4: Analysis of circuits using MATLAB	CLO3
week8	Experiment 5: S-domain analysis of circuits using MATLAB	CLO3
week9	Experiment 6: Circuit analysis with Laplace transform	CLO3
week10	Experiment 7: Analysis of first and second order circuits (Frequency Response) using MATLAB	CLO3
week11	Experiment 8: (a) To plot the frequency response (magnitude and phase) of passive filters (RC low-pass and high pass)	CLO2
week12	Experiment 8: (b) To plot the frequency response (magnitude and phase) of passive filters (RL low-pass and high pass)	CLO2

University of Engineering and Technology Lahore

Section Course Outline Report

Department: Electrical Engineering **Printed Date:** August 17, 2024

Section Content		
Week (Lec)	**Topics**	**CLO's**
week13	Experiment 9: To plot the magnitude and phase response of a series resonant band-pass filter	CLO2
week14	Experiment 10: To plot the magnitude and phase response of a series resonant band-stop filter	CLO2
week15	Experiment 11: Implementation of first order Active low pass and high pass filter	CLO2
week16	Open Ended Lab	CLO3, CLO2, CLO1

EE—-: Circuit Analysis & Design (Lab Manual)

November 18, 2018

[1]

[1]Electrical Engineering Department, University of Engineering & Technology Lahore, Pakistan.

Name: ______________________ Registration #: ______________________

EXPERIMENT NO 1

To examine the pulse and steady state response of a series RL & RC network

Objectives:

In this experiment, a pulse waveform is applied to series RL & RC circuits, to analyze the transient response of the circuits. The pulse-width relative to the circuits time constant determines how it is affected by the RL circuit.

Apparatus:

- Oscilloscope
- Signal generator
- Inductors
- Capacitor
- Resistors

Theoretical Background (RL Series Circuit)

At the instant when step voltage is applied to an RL network, the current increases gradually and takes some time to reach the final value. The reason current does not builds up instantly to its final value is that as the current increases, the self-induced *e.m.f.* in inductor opposes the change in current (Lenzs Law). Mathematically,

$$i(t) = \frac{V}{R}(1 - e^{-t/\tau}) \tag{1}$$

Where t = time elapsed since pulse is applied and $\tau = L/R$ = Time constant of the circuit.
During the next half cycle of pulse, when the pulse amplitude is zero, the current decreases to zero exponentially. Mathematically,

$$i(t) = \frac{V}{R}e^{-t/\tau} \tag{2}$$

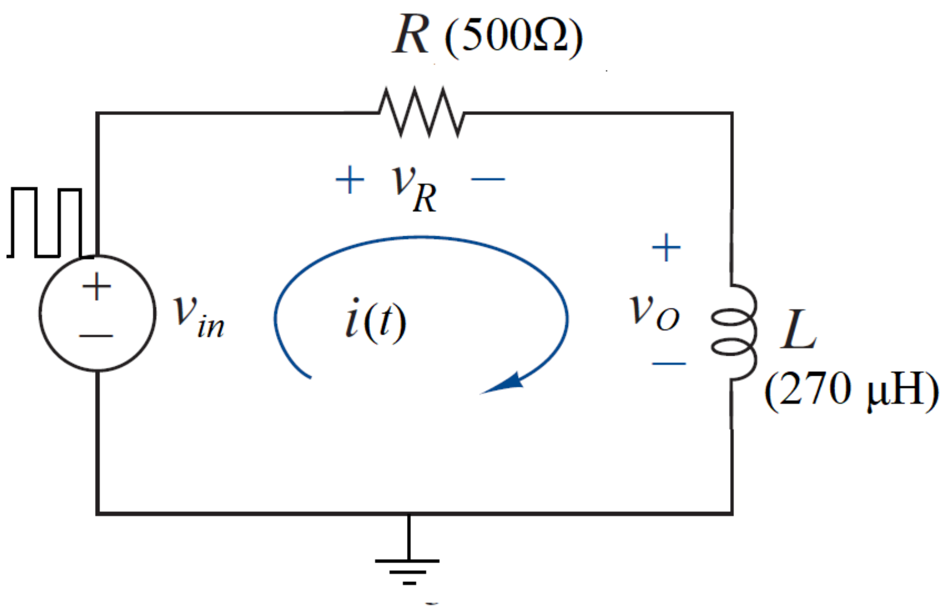


Figure 1: A series RL circuit

Task1:

1. Set the output of the function generator to a square-wave with peak-to-peak amplitude of 5V and adjust the frequency that should allow the inductor to charge and discharge fully *i.e.* f should be less than or equal to $f_{max} = \frac{1}{10\tau}$. Use dc offset to obtain pulse train ranging from 0 to 5 volts from square wave.

2. Patch the circuit given in Fig. 1 on the breadboard and apply pulse train to it.

3. Display simultaneously voltage $v_{in}(t)$ across the function generator (on CH 1) and $v_o(t)$ across the inductor L (on CH 2). Ensure common ground for both channels and the signal generator.

4. Sketch the two measured waveforms $v_{in}(t)$ and $v_o(t)$, calculate and sketch the waveforms, $v_R(t)$ and $i(t)$. Label the time, voltage and current scales. Note that the voltage waveform $v_R(t)$ across resistor R, also represents the current $i(t)$ waveform.

5. Measure the time constant, τ, using the waveform $v_o(t)$. Expand the time scale and measure the time it takes for the waveform to complete 63% of its total change, *i.e.* 5V. Note the measured value of τ.

6. Compare values of theoretically expected and experimentally obtained time constants τ.
7. Now calculate and measure $i(t)$ for five different values of $\tau$for rise of current as well as fall of current.

Observations & Calculations

Write down the values of following parameters:
$R =$ ____________, $L =$ ______________________-, $T =$ _______________________- , $10\tau =$ ____________________, $f_{max} =$ ___________________

Rise of Current

Calculate current using $i(t) = \frac{V}{R}(1 - e^{-t/\tau})$ and measure current (from oscilloscope)

No. of Time Constants	Calculated Current (amp)	Measured Current (amp)
τ		
2τ		
3τ		
4τ		
5τ		

Fall/Decay of Current

Calculate current using $i(t) = \frac{V}{R}(e^{-t/\tau})$ and measure current (from oscilloscope)

No. of Time Constants	Calculated Current (amp)	Measured Current (amp)
τ		
2τ		
3τ		
4τ		
5τ		

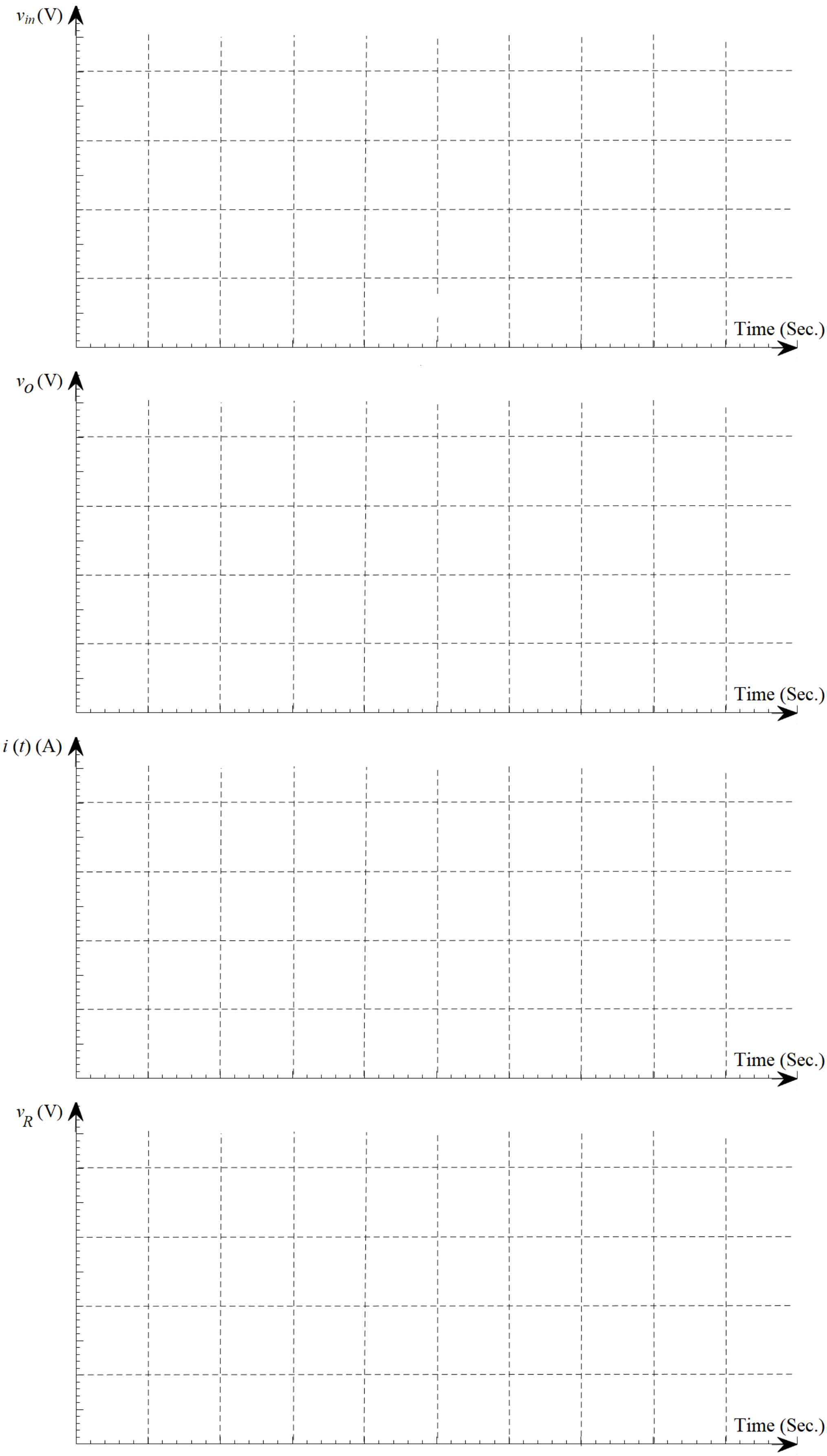


Figure 2: Input voltage, output voltage, loop current and resistor voltage of a series RL circuit.

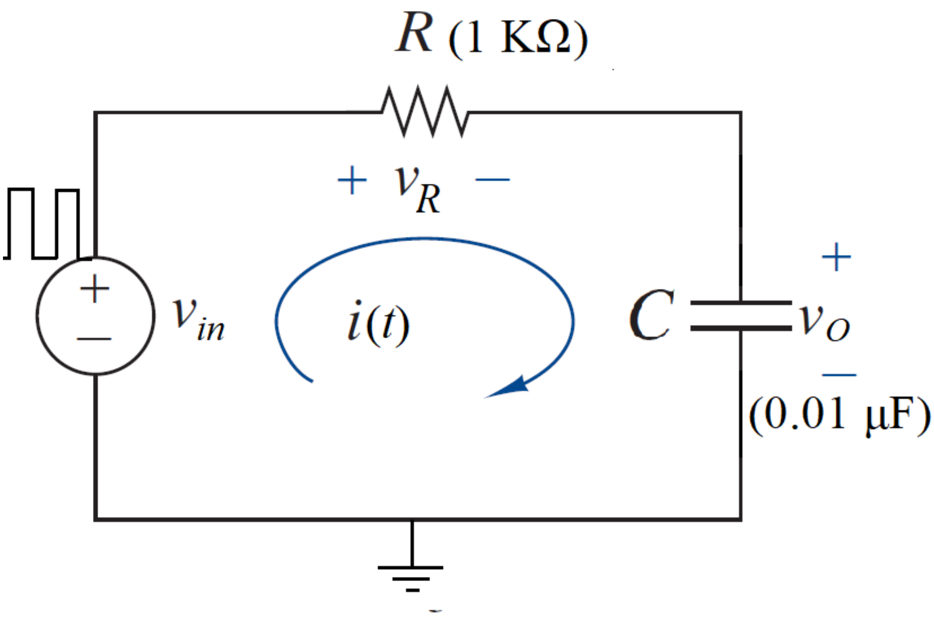


Figure 3: A series RC circuit

Theoretical Background (RC Series Circuit)

A capacitor can be charged by connecting its two terminals to the two terminals of a battery. If the capacitor is then disconnected from the battery it will retain this charge and the potential across the capacitor will remain that of the battery. If a resistor is then connected across the capacitor, charge will flow through the resistor until the potential difference between the two terminals goes to zero. The potential will decrease with time according to the relation:

$$v(t) = v_0 e^{\frac{-t}{\tau}} \tag{3}$$

where v_0 represents the voltage at time $t = 0$, and τ represents the "time constant" or time that it takes for the voltage to decrease by a factor of 1/e.

If the frequency of the square wave v_{in} is too high (*i.e.* if $f \gg 1/RC$), then v_o and v_R will not have enough time to reach their asymptotic values. If the frequency is too low (*i.e.* if $f \ll 1/RC$), the decay time will be very short relative to the period of the waveform and thus the exponential decay will be difficult to observe. As a rough guideline, the period of the square wave should be chosen such that it is approximately equal to $10RC$, in order for the responses to be readily observed on an oscilloscope.

Task2:

1. Set the output of the function generator to a square-wave with peak-to-peak amplitude of

5V and adjust the frequency that should allow the capacitor to charge and discharge fully *i.e.* f should be less than or equal to $f_{max} = \frac{1}{10\tau}$. Use dc offset to obtain pulse train ranging from 0 to 5 volts from square wave.

2. Patch the circuit given in Fig. 3 on the breadboard and apply pulse train to it.

3. Display simultaneously voltage $v_{in}(t)$ across the function generator (on CH 1) and $v_o(t)$ across the capacitor C (on CH 2). Sketch the two measured waveforms $v_{in}(t)$ and $v_o(t)$,also calculate, sketch and label the waveforms, $v_R(t)$ and $i(t)$ on the graphs given in Fig. 4.

4. Measure the time constant, τ, using the waveform $v_o(t)$. Expand the time scale and measure the time it takes for the waveform to complete 63% of its total change, *i.e.* 5V. Compare values of theoretically expected and experimentally obtained time constants τ. Calculated τ = ————————————-, Measured τ = ————————————-

Discussion:

Q1: What would be effect on v_o and v_R, if the resistance value is halved in Fig. 1?

____

____

____

____

____

____

Q2: What would be effect on v_o and v_R, if the inductance value is halved in Fig. 1?

____

____

____

____

____

____

Q3: What would be effect on v_o and v_R, if the capacitance value is doubled in Fig. 3?

____

____

____

____

____

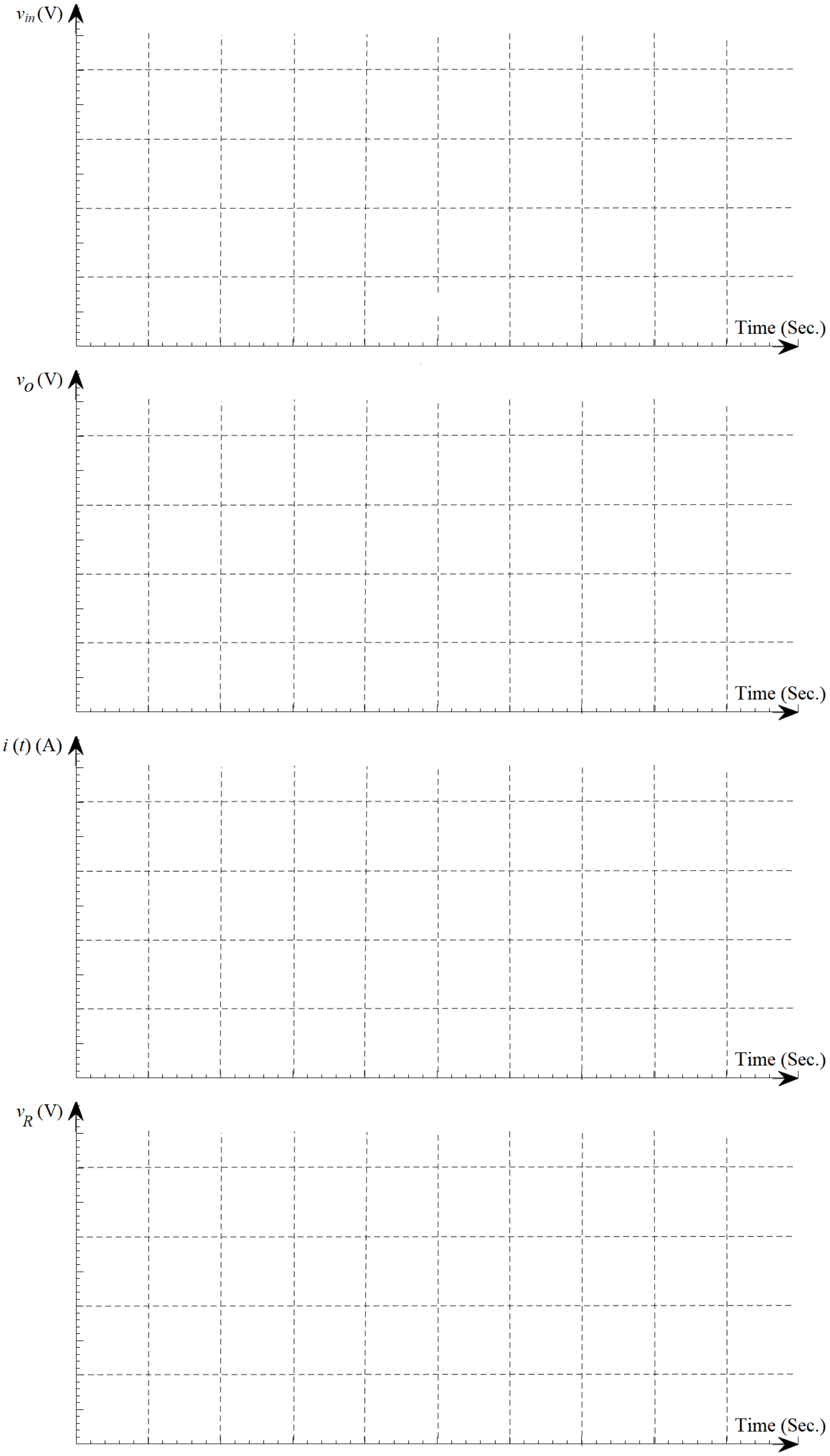


Figure 4: Input voltage, output voltage, loop current and resistor voltage of a series RC circuit.

Name: ______________________ Registration #: ______________________

EXPERIMENT NO 2

Steady state sinusoidal response (RC & RL circuits) and Phasors.

Objectives:

The response of an electrical network to a sinusoidal inputs is an extremely important characteristic. This lab investigates the amplitude and phase relationships between voltages and currents in electrical networks driven by sinusoidal sources. In particular, the concepts of phasors and impedance are examined.

Apparatus:

- Oscilloscope
- Signal generator
- Inductors
- Capacitor
- Resistors

Theoretical Background

Fig. 1 (a) shows a block-diagram representation of the system. The input is a cosine function with amplitude A and phase angle θ. The output is also a cosine function with amplitude B and phase angle φ. Both the input and output waveforms have radian frequency ω (recall that an important property of linear systems is that the steady state response of a linear system to a sinusoidal input is a sinusoid with the same frequency as the input sinusoid). The analysis of the circuit of Fig. 1 (a) can be simplified by representing the sinusoidal signals as phasors. The phasors provide the amplitude and phase information of the sinusoidal input and output signals. The input-output relationship governing the circuit then reduces to a relationship between the output and input signal amplitudes and the output and input signal phases/angles. The circuit can thus be

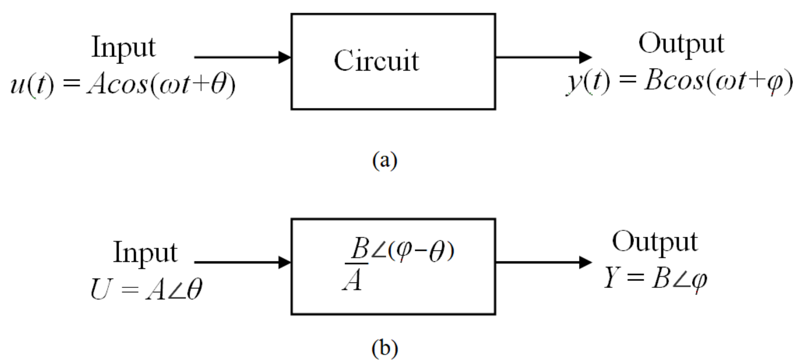


Figure 1: Steady state sinusoidal circuit analysis (a) physical circuit (b) phasor representation of circuit input output relationship.

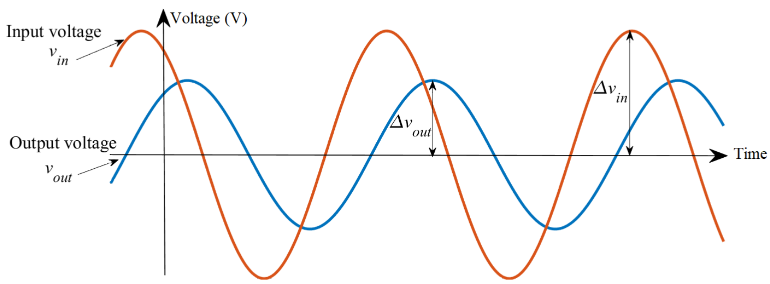


Figure 2: Input output voltage waveforms

represented in phasor form as an amplitude gain between the output and input signals and a phase difference between the output and input signals, as shown in Fig. 1 (b).

Gain of a system at a particular frequency is the ratio of the magnitude of the output voltage to the magnitude of the input voltage at that frequency *i.e.*:

$$Gain = \frac{\Delta v_{out}}{\Delta v_{in}} \tag{1}$$

Where Δv_{out} and Δv_{in} can be measured from the sinusoidal output and input voltages respectively, as shown in Fig. 2. The phase of a system at a particular frequency is a measure of the time shift between the output and input voltage at that frequency.

$$Phase = \frac{\Delta T}{T} \times 360^o \tag{2}$$

Where ΔT and T can be measured from the sinusoidal input and output voltages as shown in the Fig. 3

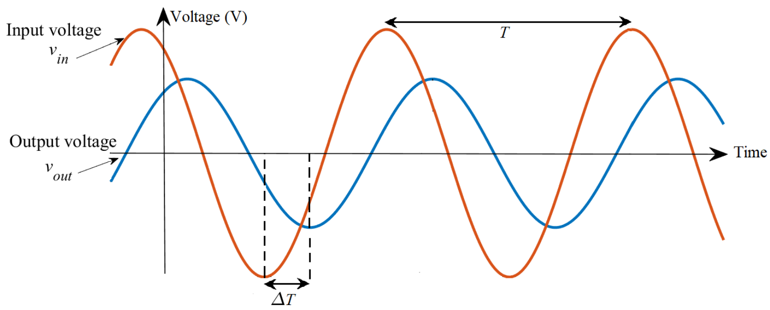


Figure 3: Input output voltage waveforms' phase relationship.

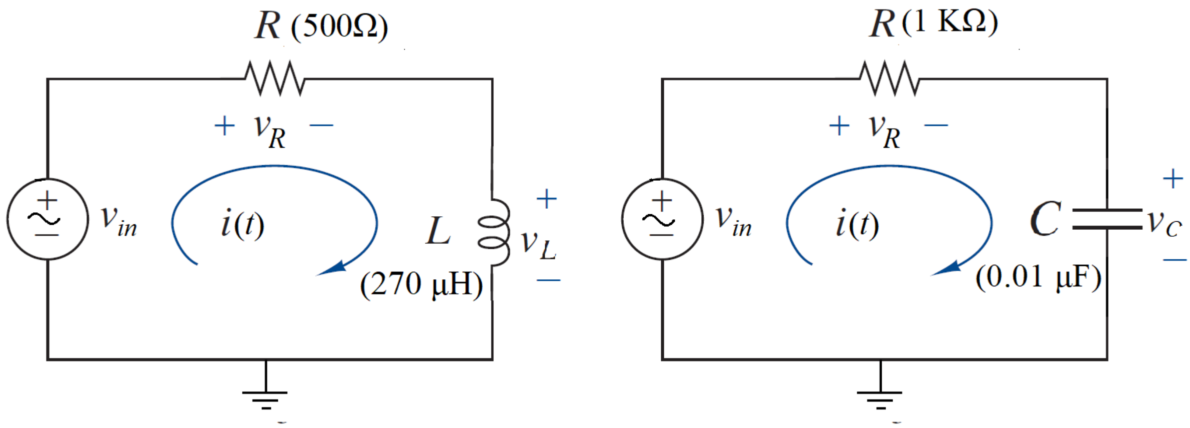


Figure 4: Response of series RL & RC circuits to sinusoidal inputs.

Sinusoidal response of RC & RL circuits:

Circuits containing inductance/capacitance and resistance appear in many electronic circuits ranging from power supplies to filters. In this experiment the response of series RL & RC circuit to sinusoidal inputs is investigated. A difficulty that arises in such circuits is that real inductors/capacitors are not like the ideal inductors/capacitors. A real inductor is formed of coiled wire so it possesses resistance as well as inductance. Similarly capacitors have internal shunt resistance as well. Furthermore for some inductors the resistance is dependent on frequency as well. As a consequence, the resistance R inserted in series with the Inductor does not represent the total resistance of the circuit. In this lab task R will be set to a fairly large value so that internal resistance of these components becomes negligibly small as compared to external R *i.e.* L & C can be assumed ideal.

For RL circuit given in Fig. 4,

$$Z_L = X_L = j\omega L \qquad \text{(Assuming resistance of coil to be zero)} \tag{3}$$

$$Z_{Total} = R + j\omega L \tag{4}$$

where $\omega = 2\pi f$ = angular frequency and Z_{Total} can be expressed in rectangular as well as polar co-ordinates.

$$I = \frac{V_{in}}{Z_{Total}} \tag{5}$$

where V_{in} in phasor notation is $5\angle 0^o$.

$$V_R = IR \qquad \text{and} \qquad V_L = I \times Z_L \tag{6}$$

For RC circuit given in Fig. 4,

$$Z_C = X_C = \frac{1}{j\omega C} \qquad \text{(Assuming internal resistance of capacitor to be zero)} \tag{7}$$

$$Z_{Total} = R + X_C = R - \frac{j}{\omega C} \tag{8}$$

where $\omega = 2\pi f$ = angular frequency and Z_{Total} can be expressed in rectangular as well as polar co-ordinates.

$$I = \frac{V_{in}}{Z_{Total}} \tag{9}$$

where V_{in} in phasor notation is $5\angle 0^o$.

$$V_R = IR \qquad \text{and} \qquad V_C = I \times Z_C \tag{10}$$

Task1

1. Patch the RL circuit given in Fig. 4 on the breadboard and apply sinusoidal signal from the signal generator. Adjust the magnitude and frequency of the sinusoidal signal to 5V peak and 1 kHz.
2. Display simultaneously voltage V_{in} across the function generator (on CH 1) and V_L across the inductor L (on CH 2). Ensure common ground for both channels and the signal generator. Measure magnitude of V_L and its phase with respect to V_{in}, and note in Table 1.
3. Interchange the resistor and inductor positions in the circuit. Display simultaneously voltage V_{in} across the function generator (on CH 1) and V_R across the resistor R (on CH 2). Ensure common ground for both channels and the signal generator. Measure magnitude of V_R and its phase with respect to V_{in}, and note in Table 1.
4. Calculate Z_L, Z_{Total}, using the measured values and note in the Table.
5. Calculate Z_L, Z_{Total}, I, V_R and V_L for a source voltage V_{in} of 5V peak and a frequency of 1 kHz and then note down these quantities in Table 1.
6. Draw the Phasor diagram of the RL circuit in Fig. 5.

Table 1: Parameters of an RL circuits

Parameters	$\boldsymbol{Z}_L(\Omega)$	$\boldsymbol{Z}_{Total}(\Omega)$	$\boldsymbol{I}(amp)$	$\boldsymbol{V}_R(V)$	$\boldsymbol{V}_L(V)$
Measured values					
Calculated values					

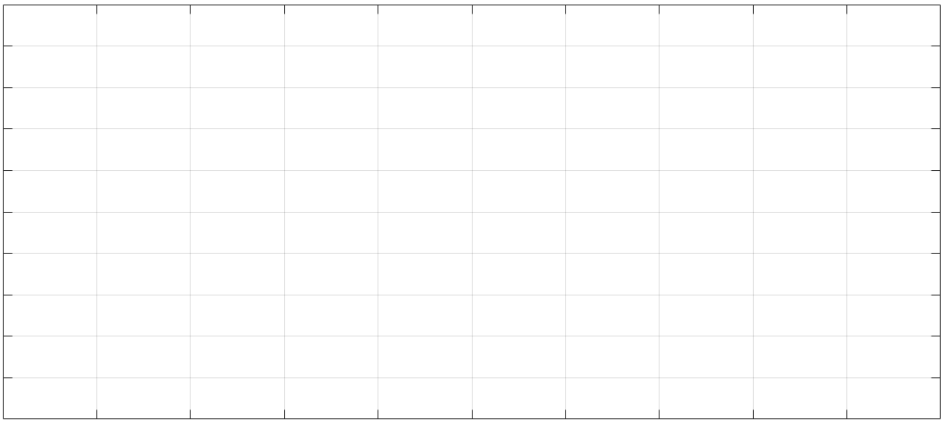


Figure 5: Phasor diagram of an RL Circuit

Task2

1. Set the sinusoidal input voltage from function generator, V_{in}, of 5V peak and a frequency of 1 kHz.

2. Patch the RC circuit given in Fig. 4 on the breadboard and apply sinusoidal signal from the signal generator.

3. Display simultaneously voltage V_{in} across the function generator (on CH 1) and V_C across the capacitor C (on CH 2). Ensure common ground for both channels and the signal generator. Measure magnitude of V_C and its phase with respect to V_{in}, and note in Table 2.

4. Interchange the resistor and inductor positions in the circuit. Display simultaneously voltage V_{in} across the function generator (on CH 1) and V_R across the resistor R (on CH 2). Ensure common ground for both channels and the signal generator. Measure magnitude of V_R and its phase with respect to V_{in}, and note in Table 2.

5. Calculate Z_C, Z_{Total}, using measured values and note in the Table.
6. Calculate Z_C, Z_{Total}, I, V_R and V_C for a source voltage V_{in} of 5V peak and a frequency of 1 kHz and then note down these quantities in Table 2.
7. Draw the Phasor diagram of the RC circuit in Fig. 6.

Table 2: Parameters of an RC circuits.

Parameters	$\boldsymbol{Z}_C(\Omega)$	$\boldsymbol{Z}_{Total}(\Omega)$	$\boldsymbol{I}$ (amp)	$\boldsymbol{V}_R(V)$	$\boldsymbol{V}_C(V)$
Measured values					
Calculated values					

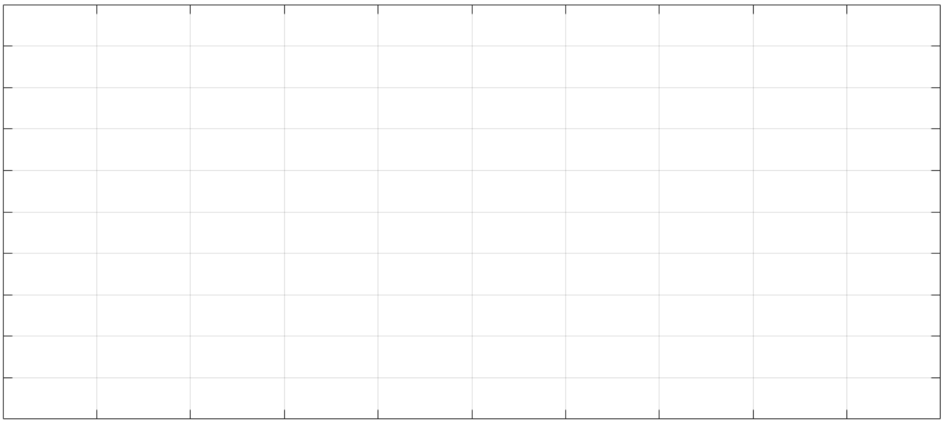


Figure 6: Phasor diagram of an RC Circuit.

Conclusion/Summary

Name: ______________________________ Registration #: ______________________________

EXPERIMENT NO 3

Resonant frequency calculation of series and parallel RLC circuits.

Objectives:

The response of a circuit containing both inductors and capacitors in series or in parallel depends on the frequency of the driving voltage or current. This lab will explore one of the more dramatic effects of the interplay of capacitance and inductance, namely, resonance, when the inductive and capacitive reactances cancel each other. The objective of this lab is to:

- Analyze behavior of series and parallel LC circuits at resonance.
- Understand the resonance frequency, cut-off frequency, bandwidth and quality factor of a resonance circuit.
- Determine if a circuit is inductive or capacitive and to understand the circuit behavior at resonance

Apparatus:

- Oscilloscope
- Signal generator
- Inductors
- Capacitor
- Resistors (10kΩ / 20kΩ Use resistor of standard value available in the Lab)

Theoretical Background

Resonant circuits form the basis for filters that have better performance than first order (RL, RC) filters in passing desired signal or rejecting undesired signals that are relatively close in frequency.

The *Resonance Frequency* is defined as the frequency at which the impedance of the circuit is purely real, that is, with zero reactance. For the reactance to be zero, impedance of the inductor must equal that of the capacitor. At resonance, the impedance of a branch with LC in series is equal to zero, which is equivalent to a short, and the admittance of a branch with LC in parallel is equal to zero, which is equivalent to an open. As the frequency increases, the magnitude of an inductive reactance increases, while the magnitude of a capacitive reactance decreases. A circuit is said to be inductive if the total reactance is positive, and a circuit is said to be capacitive if the total reactance is negative. Fig. 5 shows a series RLC circuit. Using KVL:

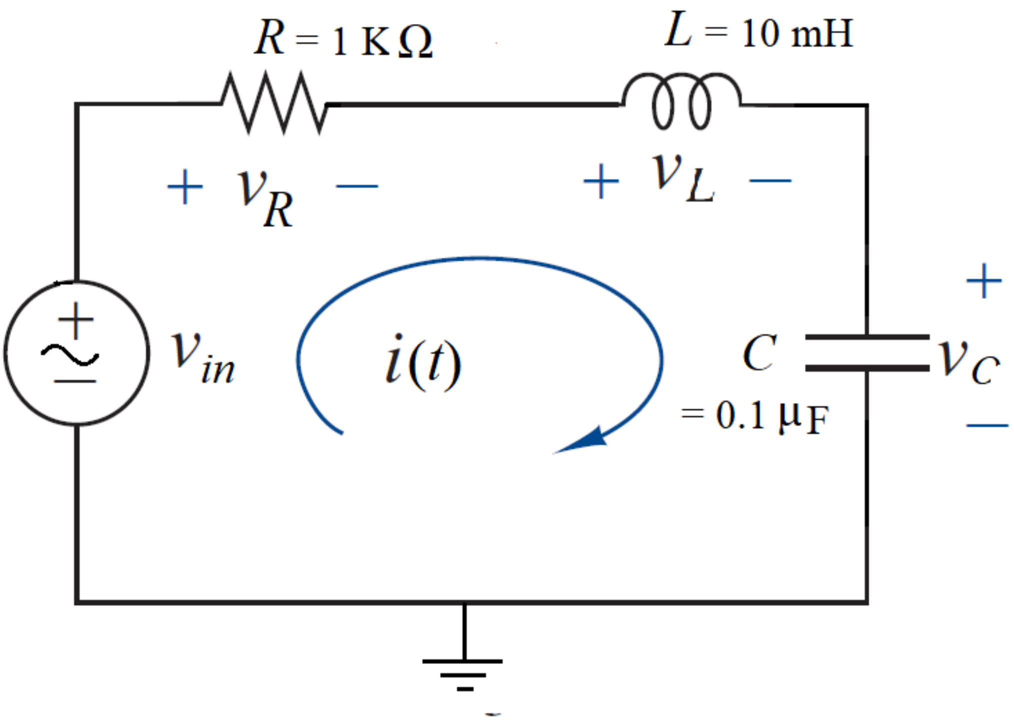


Figure 1: A series RLC Circuit.

$$v_{in} = v_R + v_L + v_C \tag{1}$$

In experiment 2, we checked that $v_L = IX_L = Ij\omega L$ and $v_C = IX_C = \frac{I}{j\omega C}$

$$\begin{aligned} v_{in} &= IR + Ij\omega L + \frac{I}{j\omega C} \\ &= IR + Ij\omega L - \frac{Ij}{\omega C} \\ &= I\big(R + j\omega L - \frac{j}{\omega C}\big) \end{aligned} \tag{2}$$

Thus

$$Z = R + j\omega L - \frac{j}{\omega C} = R + j\big(\omega L - \frac{1}{\omega C}\big) \tag{3}$$

When a series RLC circuit is in resonance, it possesses minimum impedance $Z = R$. Thus maximum circuit current I flows, as it is being limited by the value of R alone. This I is also in phase with V_{in}. Similarly, this current produces large voltage drops across L and C. These two drops being equal and opposite, cancel out each other. Taken together, L and C from part of a circuit across which no voltage develops however larger current is flowing. The frequency (ω or f) at which the net reactance of the series circuit is zero is called the resonant frequency (ω_R or f_R). Its value can be calculated as under:

$$Z_L + Z_C = 0 \tag{4}$$

$$j\big(\omega_R L - \frac{1}{\omega_R C}\big) = 0 \tag{5}$$

$$\big(\omega_R L - \frac{1}{\omega_R C}\big) = 0 \tag{6}$$

$${\omega_R}^2 = \frac{1}{LC} \tag{7}$$

$$\omega_R = \frac{1}{\sqrt{LC}} \tag{8}$$

$$f_R = \frac{1}{2\pi\sqrt{LC}} \tag{9}$$

If L is in henry and C is in farads then f_R is in Hz.

Task 1:

1. For the circuits inductance and capacitance values calculate the resonant frequency and connect the circuit as shown in Fig. 1.

2. Apply sinusoidal signal from the signal generator of 5V peak to the network and set the frequency to a value less than resonance frequency f_R. Phasor notation of V_{in} becomes $5\angle 0$, therefore calculate phasor form of X_L, X_C, Z, I, V_L and V_C at this frequency and note on Table 1. Calculate the above six quantities at f_R and another frequency greater than f_R. Note all of them in the table.

3. Now measure V_C, V_L and V_R at f_R, one frequency below f_R and one above f_R. Insert all values in Table 2. Make sure you measure both the peak magnitude and phase of V_C, V_L and V_R at these frequencies as done in Experiment 2. Complete Table 2 by calculating I, X_L and X_C from the above measured V_C, V_L and V_R.

Observations & Calculations

Calculated resonant frequency = ____________________

Table 1: Calculated values.

No	Frequency (Hz)	jX_C (Ω)	jX_L (Ω)	Z (Ω)	$I = \frac{V_{in}}{Z}$ (amp)	$V_L = jIX_L$ (V)	$V_C = jIX_C$ (V)

Table 2: Measured values.

No	Frequency (Hz)	jX_C (Ω)	jX_L (Ω)	Z (Ω)	I (amp)	V_L (V)	V_C (V)

Resonance in a parallel RLC circuit

As we already know that if the capacitive reactance of a circuit is equal to the inductive reactance then the circuit is in resonance condition. Fig. 2 shows a parallel RLC circuit where $i_R(t)$, $i_L(t)$, and $i_C(t)$, are the currents flowing through R, L and C respectively. The voltage across all these three components is same and is equal to v_{in}.

V_{in}, I_{in}, I_R, I_L, I_C represent phasor representation of input voltage, total input current, resistor current, inductor current and capacitor current respectively. V_{in} and I_R are in phase and total current I can be calculated as

$$\begin{aligned} I &= I_R + I_L + I_C \qquad (10) \\ &= \frac{V_{in}}{R} + \frac{V_{in}}{X_L} + \frac{V_{in}}{X_C} \end{aligned}$$

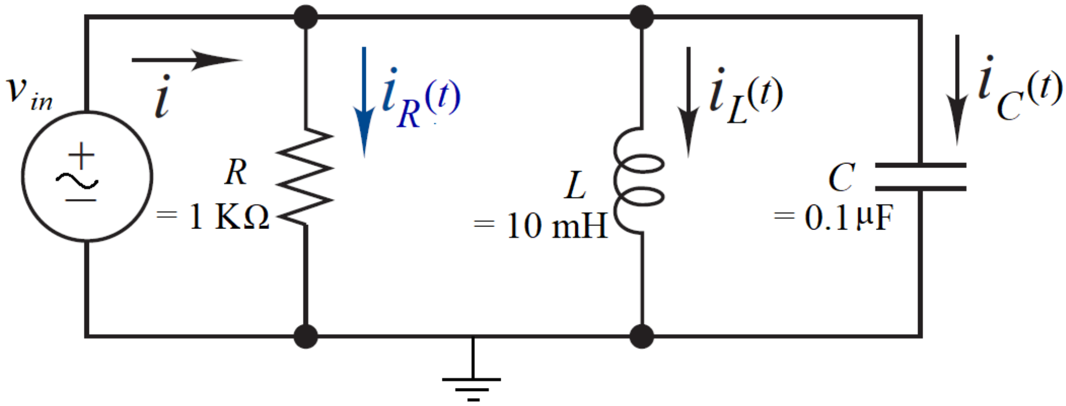


Figure 2: A parallel RLC Circuit.

where X_L and X_C represent inductive and capacitive reactances respectively.

$$\begin{aligned} I &= V_{in}\left(\frac{1}{R} + \frac{1}{j\omega L} + j\omega C\right) \\ &= V_{in}\left(\frac{1}{R} + j\left(\omega C - \frac{1}{\omega L}\right)\right) \\ &= V_{in} Y \end{aligned} \tag{11}$$

where Y is the admittance of the parallel RLC circuit. At resonance condition,

$$\begin{aligned} Z_L &= Z_C \\ j\omega_R L &= \frac{1}{j\omega_R C} \\ {\omega_R}^2 &= \frac{1}{LC} \\ \omega_R &= \frac{1}{\sqrt{LC}} \qquad \text{(rad/sec)} \\ f_R &= \frac{1}{2\pi\sqrt{LC}} \qquad \text{(Hz)} \end{aligned} \tag{12}$$

Thus at resonance condition, total impedance of the RLC circuit will be equal to R (Refer to (12) for better understanding.

Task 2:

1. For the RLC Parallel circuit, calculate the resonant frequency and connect the circuit as shown in Fig. 2.
2. Connect small resistors (like 1 Ω) in series with inductor and capacitor to measure I_c and I_L.

3. Apply sinusoidal signal from the signal generator of 5V peak to the network and set the frequency to a value less than resonance frequency f_R. Phasor notation of V_{in} becomes $5\angle 0$, therefore calculate phasor form of X_L, X_C, Y, I, I_R, I_L and I_C at this frequency and note on Table 3. Calculate the above six quantities at f_R and another frequency greater than f_R. Note all of them in the table.

4. Now measure I_L, I_c and I_R at f_R, one frequency below f_R and one above f_R. Insert all values in Table 4. Make sure you measure both the peak magnitude and phase of I at these frequencies. Complete Table 2 by calculating the remaining parameters.

Observations & Calculations

Calculated resonant frequency = ____________________

Table 3: Calculated values.

No	f (Hz)	Z_C (Ω)	Z_L (Ω)	Y_{Total} (Ω^{-1})	$I = V_{in}Y$ (A)	$I_L = \frac{V_{in}}{Z_L}$ (A)	$I_C = \frac{V_{in}}{Z_C}$ (A)

Table 4: Measured values.

No	f (Hz)	Z_C (Ω)	Z_L (Ω)	Y_{Total} (Ω^{-1})	I_R (A)	I_L (A)	I_C (A)

Discussion

Q1: Please discuss the how the total impedance of series RLC circuit varies by varying frequency of input signal. You can draw a graph of Z versus frequency ($0 \rightarrow \inf$).

Q2: Please discuss the how the total impedance of parallel RLC circuit varies by varying frequency of input signal. You can draw a graph of Z versus frequency ($0 \rightarrow \inf$).

Name: ________________________ Registration #: ________________________

EXPERIMENT NO 4

Analysis of Circuits Using MATLAB

Objectives:

The objective of this lab is to get familiar with MATLAB and utilize it in context of solving basic circuits.

Basic Concept

Double click on MATLAB icon and a window will open like the one shown in Fig. 1. The display can be slightly different from the one shown in this Fig. due to different versions of MATLAB.

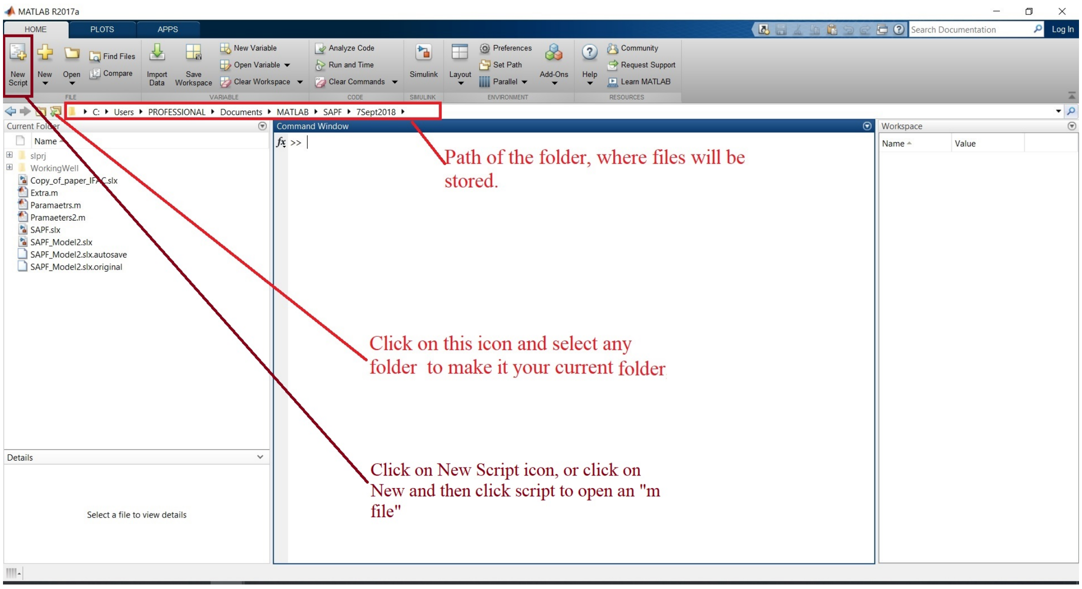


Figure 1: MATLAB window.

PLOTTING FUNCTIONS

MATLAB allows you to create plots of functions easily. We will demonstrate this by the following example. Suppose we want to plot the following three functions:

$$v_1(t) = 5cos(2t + 45^o)$$
$$v_2(t) = 2exp(-t/2)$$
$$v_3(t) = 10exp(-t/2)cos(2t + 45^o)$$

Further more we want to use red for v_1, green for v_2 and blue for v_3. In addition we want to label the figure, the horizontal and the vertical axes as well as each on of the the curves. The following is a sequence of MATLAB commands that will allow us to do this. This is not a unique set of commands. Write the following code in m file. Save it and run it.

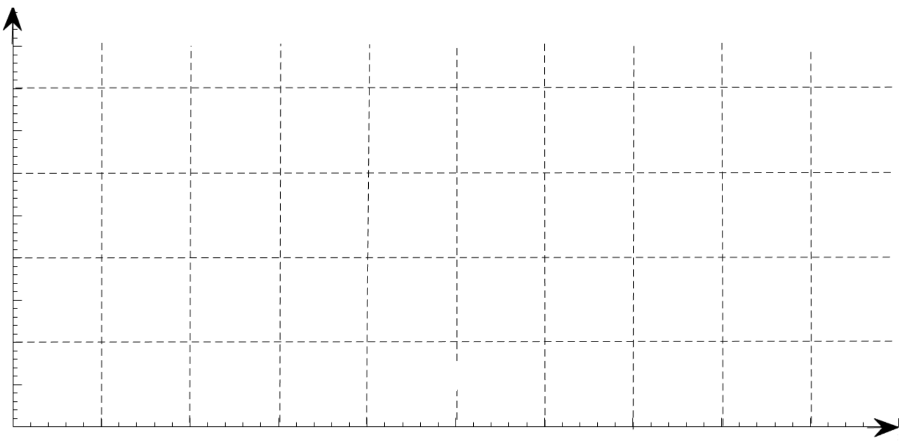
```
clc
clear
t=0:0.1:10; % t is the time varying from 0 to 10 in steps of 0.1s
v1=5*cos(2*t+0.7854);
taxis=0.000000001*t;
plot(t,taxis,'w',t,v1,'r')
grid
hold on % For holding the previous figure and new plot will be drawn over the previous one
v2=2*exp(-t/2);
plot (t,v2,'g')
v3=10*exp(-t/2).*cos(2*t+0.7854);
plot (t,v3,'b')
title('Example 1 -- Plot of v1(t), v2(t) and v3(t)')
xlabel ('Time in seconds')
ylabel ('Voltage in volts')
text (6,6,'v1(t)')
text (4.25,-1.25,'v2(t)')
text (1,1.75,'v3(t)')
```

Please note the following important point:

- ; at the end of a line defines the command but does not immediately execute it.
- The symbol * is used to multiply two numbers or a number and a function.
- he combination of symbols .* is used to multiply two functions.

- The command “hold on” keeps the existing graph and adds the next plot on to it. The command “hold off” undoes the effect of “hold on”.
- In the command “text”, the first two numbers give the X and Y coordinate of where the test willo appear in the figure.
- The command “plot” can plot more than one function simultaneously. In fact, in this example we could get away with only one plot command.
- Comments can be included after the % symbol.
- In the plot command, one can specify the color of the line as well as the symbol: ‘b’ stands for blue, ‘g’ for green, ‘r’ for red, ‘y’ for yellow, ‘k’ for black; ‘o’ for circle, ‘x’ for x-mark, ‘+’ for plus, etc. For more information type help plot in MATLAB command prompt.

Save the plot as a jpg file. Draw it or paste in the space given below.



COMPLEX NUMBERS

Working with complex numbers in MATLAB is easy. MATLAB works with the rectangular representation. To enter a complex number, type:
`a + bj or a + bi` with a and b being numerical values in command prompt.

example: z = 5 - 3j

To find the magnitude and angle of z, use the `abs()` and `angle()` function.

```
z = 5 - 3j
Mag = abs(z)
Angle = angle(z)
```

The angle function gives the angle in radians. To convert to degrees you can use:
`AngleDeg = angle(z)*180/pi`

Example: $V = \frac{(5+j9)(7+j)}{3-j2}$. Now type in command prompt:

```
>>V = (5+9j)*(7+j)/(3-2j);
>>MagnV = abs(V);
```

To find the real and imaginary part of a complex number z, type:

```
>>realZ=real(z);
>>ImagZ=imag(z);
```

SOLVING LINEAR EQUATIONS AND MATRICES

Assume you have the following two linear equations with unknown I1 and I2:

$$(600 + 1250j)I1 + 100j.I2 = 25$$

$$100j.I1 + (60 - 150j).I2 = 0$$

This can be written in matrix form: A.I = B. To solve this in MATLAB one can use the matrix left division operator as `I = A\B`. Or one can also use the following command: `I = inv(A)*B`

The MATLAB code is as follows:

```
>>A=[600+1250j 100j;100j 60-150j];
>>B=[25;0];
>>I=A\B
I =

0.0074 - 0.0156i
0.0007 - 0.0107i

>>MAGN=abs(I)

MAGN =

0.0173
0.0107

>>ANGLE=angle(I)*180/pi

ANGLE =
```

```
-64.5230
-86.3244
```

Type the above given code in command prompt. When your write a code in command prompt results of previous line are calculated and can also be printed when you move to next line. One uses the `abs()` operator to find the magnitude of the complex number and the `angle()` operator to find the angle (in radians). To get the result in degree we have multiplied the angle by 180/pi as shown above.

FINDING THE ROOTS OF A POLYNOMIAL

To find the roots of a polynomial of the form

$$A = a_m s^m + a_{m-1} s^{m-1} + a_{m-2} s^{m-2} + \cdots + a_1 s^1 + a_0$$

Define the polynomial as follows: `A = [am am-1 am-2 ... a1 a0];`

The command to for finding the roots is `roots(A)`. As an example consider the following function:

$$A = 4s^2 + 12s + 1$$

Write in command prompt:

```
>> A=[4 12 1];
>> roots(A)

ans =
-2.9142
-0.0858
```

This works also for complex roots. As an example consider the function:

Write in command prompt:

```
>> A=[5 3 2];
>> roots(A)

ans =
-0.3000 + 0.5568i
-0.3000 - 0.5568i
```

FINDING THE POLYNOMIAL WHEN THE ROOTS ARE KNOWN

Suppose that you have the following expression F(s) and would like to find the coefficient of the corresponding polynomial:

$$F(s) = (s - a_1)(s - a_2)(s - a_3)$$

This is defined in MATLAB by a column vector of the roots:

```
roots = [a1; a2; a3 ];
```

One finds than the coefficient of the polynomial, using the `poly` command:

```
 poly(roots);
```

As example consider the function $F(s) = s(s + 50)(s + 250)(s + 1000)$. To find the coefficient of the coresonding polynomials, one first define the column vector of the roots:

```
roots=[0; -50; -250; -1000 ];
```

The coefficients are then found from the poly command:

```
coeff = poly(roots)
```

which will give:

```
coeff = 1 1300 312500 12500000 0
```

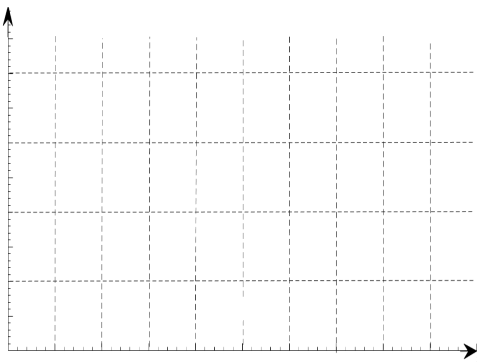
corresponding to the polynomial,

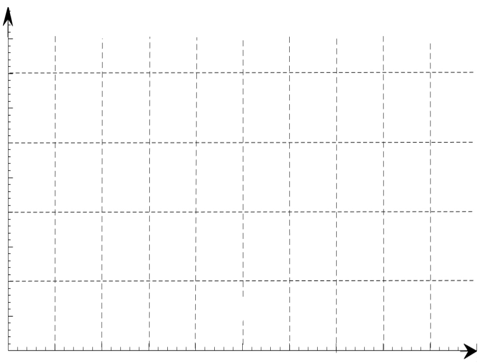
$$F(s) = s^4 + 1300s^3 + 312500s^2 + 12500000s$$

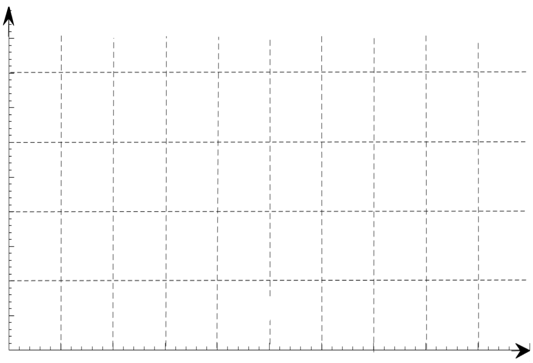
Exercise

Write down the code for a series RLC circuit having following parameter values.
$R = 1K\Omega, L = 10mH, C = 0.1\mu F, V_{in} = 5V$. Calculate and plot following waveforms $v_r(t), v_l(t), v_c(t)$ at Three different frequencies 2 kHz, 5 kHz, and 8 kHz.

Note: There should be total three graphs, one for V_R, second for V_L and third for V_c. Each plot containing three waveforms at three different frequencies (2kHz, 5kHz, and 8 kHz). Plots should be labeled appropriately.







Name: ______________________ Registration #: ______________________

EXPERIMENT NO 5

S Domain Analysis of Circuits Using MATLAB

Objectives:

The objective of this lab is to get familiar with MATLAB and utilize it in context of working with Laplace transforms. MATLAB is a powerful numerical analysis tool, which can be utilized to find, for example the inverse Laplace transform, or the solution of equations arising from the analysis of circuits with time-varying excitations, etc.

Basic Concepts

In particular, MATLAB has the functions *laplace* and *ilaplace*, which operate on symbolic expressions to take the Laplace transform and inverse Laplace transform, respectively.

$$F(s) = \frac{4}{5s^3 + 10s^2 + 10s + 5}$$

```
>> num=[4];
```

% Denominator polynomial, D(s)

```
>> den=[5 10 10 5];
```

% Find the zeros of H(s) by N(s)=0

```
>> roots(num)
ans =
Empty matrix: 0-by-1
```

% Find the poles of H(s) by D(s)=0

```
>> roots(den)
ans =
-1.0000
-0.5000 + 0.8660i
-0.5000 - 0.8660i
```

Matlab output shows that there are no finite zeros; however, there are three poles: 1 and 0.5 ± j0.866. The `polyval(p,x)` function evaluates a polynomial at some specified value of the independent variable x.

Other MATLAB functions used with polynomials are the following:

`conv(a,b)` multiplies two polynomials a and b.

`[q,r]=deconv(c,d)` divides polynomial c by polynomial d and displays the quotient q and remainder r.

`polyder(p)` produces the coefficients of the derivative of a polynomial p.

Example:

$$P = 2x^6 - 8x^4 + 4x^2 + 10x + 12$$

Calculate roots of polynomial P, evaluate this polynomial at x = 5, also compute the derivative of the given polynomial.

MATLAB Code:

```
>> P = [2 0 -8 0 4 10 12];
>> PValueAt5 = polyval(P,5)
PValueAt5 =
26412
>> der_p=polyder(p) % Compute the coefficients of the derivative of P
>> der_p =
12 0 -32 0 8 10
```

Therefore,

$$\frac{dP}{dx} = 12x^5 - 32x^3 + 8x + 10$$

Rational Polynomials

Rational Polynomials are those which can be expressed in ratio form, that is, as

$$R(x) = \frac{Num(x)}{Den(x)} = \frac{b_n x^n + b_{n-1}x^{n-1} + b_{n-2}x^{n-2} + \cdots + b_1 x^1 + b0}{a_m x^m + a_{m-1}x^{m-1} + a_{m-2}x^{m-2} + \cdots + a_1 x^1 + a0}$$

where some of the terms in the numerator and/or denominator may be zero. We can find the roots of the numerator and denominator with the `roots(p)` function as before. Also note, we can write MATLAB statements in one line, if we separate them by commas or semicolons. Commas will display the results whereas semicolons will suppress the display.

Example

$$R(x) = \frac{P_{Num}}{P_{Den}} = \frac{x^5 - 3x^4 + 5x^2 + 7x + 9}{x^6 - 4x^4 + 2x^2 + 5x + 6}$$

Express the numerator and denominator in factored form, using the `roots(p)` function.

MATLAB Code:

```
>> num=[1 -3 0 5 7 9]; den=[1 0 -4 0 2 5 6];
>> roots_num=roots(num), roots_den=roots(den)
roots_num =
2.4186+ 1.0712i 2.4186- 1.0712i -1.1633
-0.3370+ 0.9961i -0.3370- 0.9961i
roots_den =
1.6760+0.4922i 1.6760-0.4922i -1.9304
-0.2108+0.9870i -0.2108-0.9870i -1.0000
```

As expected, the complex roots occur in complex conjugate pairs. For the numerator, we have the factored form.

$P_{Num} = (x-2.4186-j1.0712)(x-2.4186+j1.0712)(x+1.1633)(x+0.3370-j0.9961)(x+0.3370+j0.9961)$

Similarly for denominator,

$P_{Den} = (x-1.6760-j0.4922)(x-1.6760+j0.4922)(x+1.9304)(x+0.2108-j0.9870)(x+0.2108j+0.9870)(x+1)$

We can check this result with MATLABs *Symbolic Math Toolbox* which is a collection of tools (functions) used in solving symbolic expressions. For the present, our interest is in using the `collect(s)` function that is used to multiply two or more symbolic expressions to obtain the result in polynomial form. We must remember that the `conv(p,q)` function is used with numeric expressions only, that is, polynomial coefficients. Before using a symbolic expression, we must create one or more symbolic variables such as x, y, t, and so on. For our example, we use the following code:

MATLAB Code:

```
syms x % Define a symbolic variable and use collect(s) to express numerator in polynomial
form
collect((x-2.4186-j*1.0712)*(x-2.4186+j*1.0712)*(x+1.1633)*(x + 0.3370-j*0.9961)*(x + 0.33
ans =

x^5 - (29999*x^4)/10000 - (71*x^3)/160000 + (5000547421061*x^2)/1000000000000 + (109391796
```

`double()` or `vpa()` commands can be used to convert fractional values in decimals in symbolic toolbox.

Laplace & Inverse Laplace Transform

Example: Let $f(t) = sin2t$, Calculate its Laplace and inverse Laplace transform using MATLAB.

MATLAB Code:

```
>> syms t
>> f = sin (2*t)
f =
sin(2*t)
>> F = laplace(f)
F =
2/(s^2 + 4)
```

For converting F back to time domain from s domain, `ilaplace()` function can be used

```
>> ilaplace(F)
ans =
sin(2*t)
```

Pole-zero map

Let $f(t) = cos5000t$ Calculate its Laplace and inverse Laplace transform usin MATLAB and draw its pole-zero diagram.

MATLAB Code:

```
>> syms t
>> f = cos (5000*t)
f =
cos(5000*t)
>> F = laplace(f)
F =
s/(s^2 + 25000000)
>> sys = tf([1, 0],[1,0,25000000])
sys =
s
------------
s^2 + 2.5e07
Continuous-time transfer function.
>> pzmap(sys) %
```

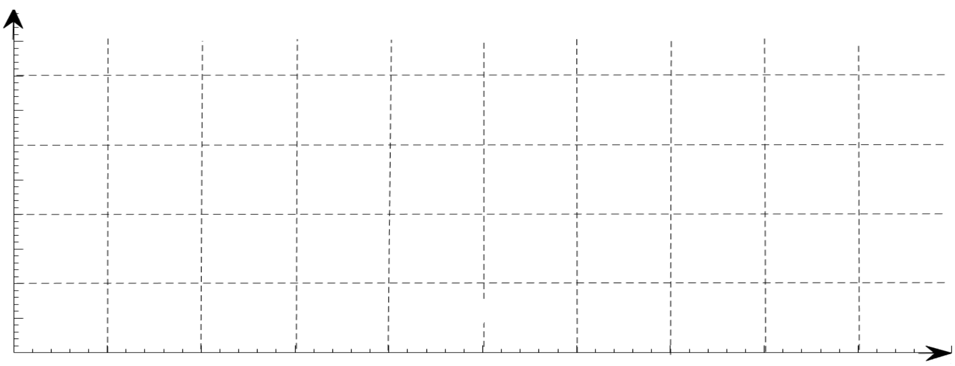
NOTE: Please check the use of `tf()` and `pzmap()` from MATLAB help.

Exercise

End Problem 9.18 from text book. For the following waveform:
$f(t) = [500 + 100e^{-500t} t sin(1000t)]u(t)$.

(a) Find the Laplace transform of the waveform. Locate the poles and zeros of F(s).

(b) Validate your result using MATLAB and draw pole zero diagram.



Name: ______________________ Registration #: ______________________

EXPERIMENT NO 6

Circuit Analysis With Laplace Transform

Objectives:

The objective of this lab is to utilize MATLAB in context of working with Laplace transforms. MATLAB is a powerful numerical analysis tool, which can be utilized to find, for example the inverse Laplace transform, or the solution of equations arising from the analysis of circuits with time-varying excitations, etc.

Basic Concepts

Circuit Transformation from Time to Complex Frequency

The voltage-current relationships for the three elementary circuit devices, resistors, inductors, and capacitors in the complex frequency domain are as follow.

Resistor

The time and complex frequency domains for purely resistive circuits are shown in Fig. 1

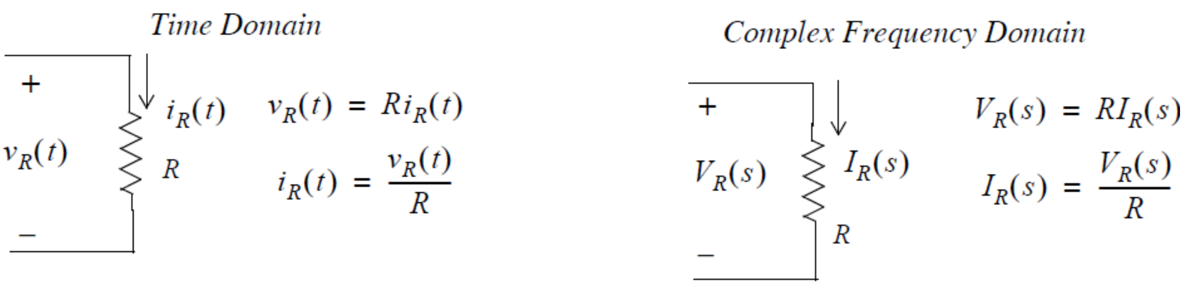


Figure 1: Resistive circuit in time domain and complex frequency domain.

Inductor

The time and complex frequency domains for purely inductive circuits is shown in Fig. 2

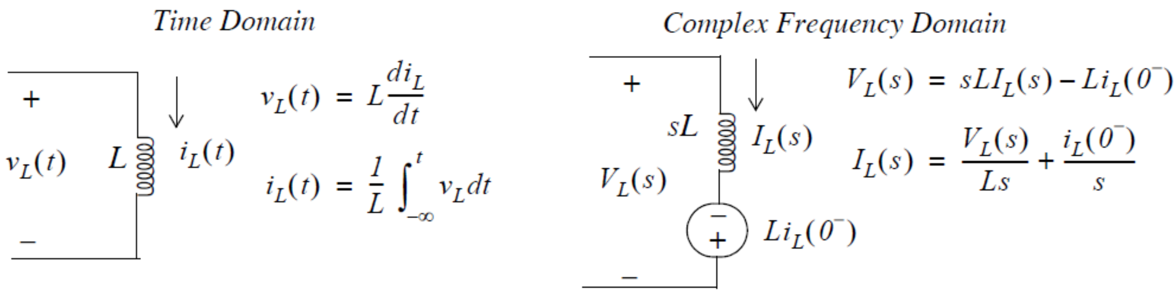


Figure 2: Inductive circuit in time domain and complex frequency domain.

Capacitor

The time and complex frequency domains for purely capacitive circuits is shown in Fig. 3

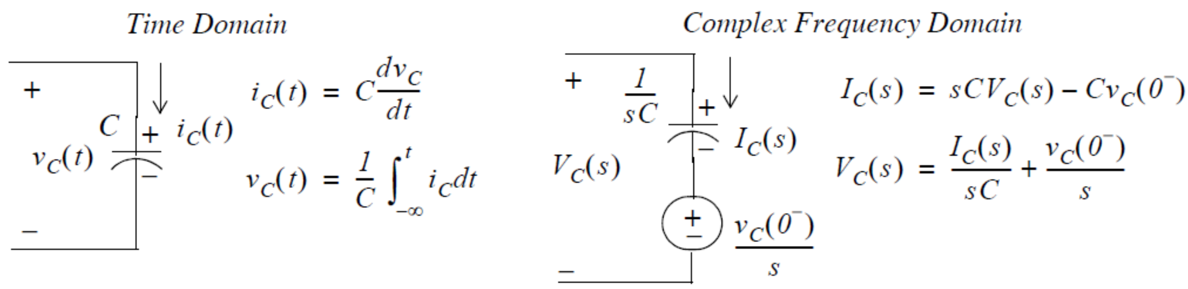


Figure 3: Capacitive circuit in time domain and complex frequency domain

Example 1: Compute $Z(s)$ and $Y(s)$ for the circuit of Fig. 4. All values are in Ω. Verify your answers with MATLAB.

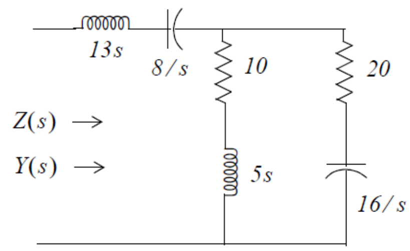


Figure 4: Linear circuit of Example 1

It is convenient to represent the given circuit as shown in Fig. 5. *Solution*

$$Z_1 = 13s + \frac{8}{s} = \frac{13s^2 + 8}{s}$$

$$Z_2 = 10 + 5s$$

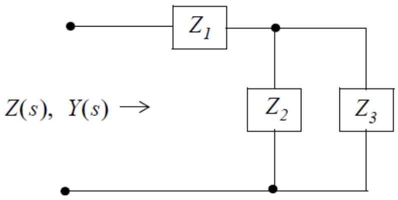


Figure 5: Simplified Circuit

$$Z_3 = 20 + \frac{16}{s} = \frac{4(5s+4)}{s}$$

$$Z_s = Z_1 + Z_2||Z_3 = Z_1 + \frac{Z_2 Z_3}{Z_2 + Z_4} = \frac{13s^2+8}{s} + \frac{(10+5s)\left(\frac{4(5s+4)}{s}\right)}{(10+5s) + \left(\frac{4(5s+4)}{s}\right)}$$

$$Z_s = \frac{65s^4 + 490s^3 + 528s^2 + 400s + 128}{s(5s^2+30s+16)}$$

Checking With MATLAB

```
syms s; z1 = 13*s + 8/s; z2 = 5*s + 10; z3 = 20 + 16/s; z = z1 + z2 * z3 / (z2+z3)
z10 = simplify(z), pretty(z10), Y10 = pretty(1/z10)
```

Exercise

Task1: The switch in Fig. 6 has been in position A for a long time and is moved to position B at t= 0.
(a) Transform the circuit into the s domain and solve for $I_L(s)$ in symbolic form.
(b) Repeat part (a) using MATLAB. *Hint: Use syms s R1 R2 R3 L C*
(c) Find $i_L(t)$ for R1= R2 = 500 Ω, R3 = 1 kΩ, L = 500 mH, C = 0.2 μF, and VA = 15 V.

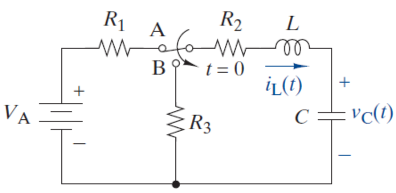


Figure 6: Circuit of Task 1

Task2: There is no external input in the circuit in Fig. 7.
(a) Find the zero-input node voltages $v_A(t)$ and $V_B(t)$, and the voltage across the capacitor $v_C(t)$ when $v_C(0) = -5$ V and $i_L(0) = 0$ A.
(b) Use MATLAB to calculate and plot your results in (a).

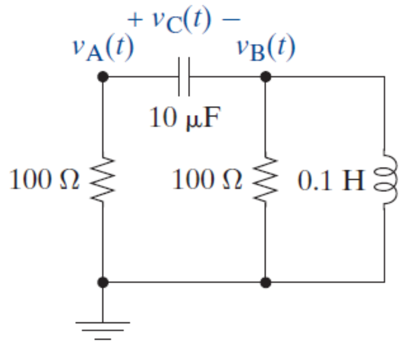


Figure 7: Circuit of Task 2

Task3: The switch in Fig. 8 has been in position A for a long time and is moved to position B at $t = 0$.
Use MATLAB to solve for $V_C(s)$ and $v_C(t)$. Also using MATLAB, plot $v_C(t)$ and the exponential source on the same axes.

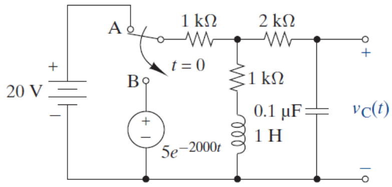


Figure 8: Circuit of Task 1

Name: ____________________ Registration #: ____________________

EXPERIMENT NO 7

Analysis of first and second order circuits (Frequency Response)

Objectives:

The objective of this lab is to utilize MATLAB to analyze the frequency response of first and second order circuits (with real poles).

Basic Concepts

Step Response

The time domain response of a circuit when a unit step is applied at its input. MATLAB has a built-in function `step()` which plots the step response of the system.

Impulse Response

The time domain response of a circuit when an impulse is applied at its input. MATLAB has a built-in function `impulse()` which plots the step response of the system.

Bode Plot

A bode plot is a graph of the frequency response of the system. The Bode magnitude plot consists of the magnitude (in dB) vs the frequency (Hz or radian per sec.) in logarithmic scale. The Bode phase plot expresses the phase or the phase shift (in radians or degrees) vs the frequency (Hz or radian per sec.) in logarithmic scale. Bode plots can be evaluated from the transfer function by substituting $s = jw$.

$$Frequencey \quad Response = H(jw) = H(s) = \frac{V_{out}(s)}{V_{in}(s)} \tag{1}$$

Cutoff frequency

The cutoff frequency is the frequency at which the gain of the circuit is 3dB less than the maximum gain.

MATLAB built-in functions

Few recommended builtin functions for this lab are as follows:

```
bode(sys); tf(num,den); bodemag(sys,w); bode(sys,w)
logspace(first_exponent,last_exponent ,number_of_values)  %logarithmically spaced vector
of specified number of values from 10^{first_exponent} to 10^{last_exponent}
solve(equ)          %solves the equation by equating it to zero.
semilogx(x,y)       %plots y vs x with x axis in log scale.
semilogy(x,y)       %plots y vs x with y axis in log scale.
roots(p)           %finds the roots of a polynomial
impulse(sys); fprintf(' \n'); disp(); pzmap()
[y,t] = step(sys)  %stores step response data in arrays
```

Please take some time to check the help of above mentioned functions. Please make separate m files for each of the following examples and tasks.

Example 1:

Draw the bode plot of the given transfer function $G(s) = \frac{1100s}{s^2+1100s+10^5}$

```
num=[0 1100 0]; den=[1 1100 10^5]; sys=tf(num,den);
w=logspace(0,5,100); bodemag(sys,w); grid
```

Example 2:

The following script shows you how to create a magnitude and a phase Bode plot for the transfer function $H(s) = \frac{s}{s+1000}$ using 30 points from 5 rad/s to 10,000 rad/s:

```
num = [1 0]; % Numerator
den = [1 1000]; % Denominator
w = logspace(log10(5), 4, 30);
[mag, phase] = bode(num, den, w);
magdb = 20 * log10(mag);
subplot(2,1,1);
semilogx(w, magdb);
grid;
axis([5 10000 -50 0]);
```

```
title('Bode Plots for H(s) = s / s + 1000');
xlabel('Frequency, rad/s');
ylabel('Magnitude, dB');
subplot(2,1,2);
semilogx(w, phase);
grid;
axis([5 10000 0 100]);
xlabel('Frequency, rad/s');
ylabel('Phase, deg');
```

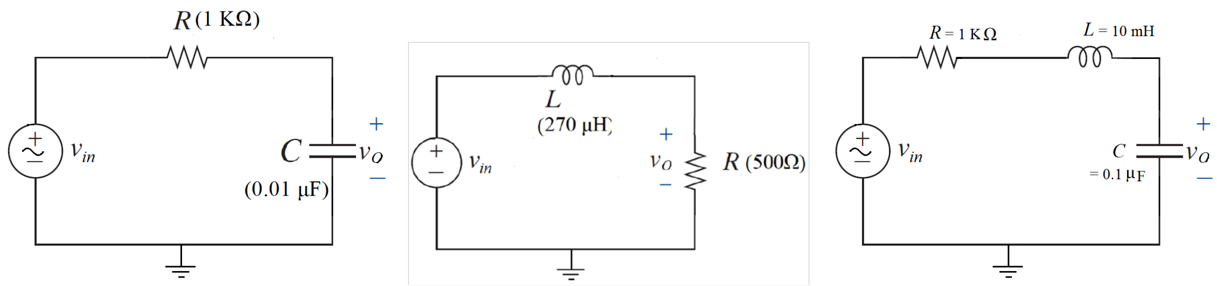
Example 3:

Solve the given equation and write the value of w : $atan(w/100) + atan(w/1000) - /2 = 0$

```
syms w; x=solve(atan(w/100)+atan(w/1000)-pi/2);
combine(x)
```

Exercise:

Task1:



(a) RC circuit *(b) RL circuit* *(c) RLC circuit*

Figure 1: Various circuits for Task 1

a) Determine the transfer function of the three circuits given in Fig. 1.

Transfer function of RC circuit = ______________________

Transfer function of RL circuit = ______________________

Transfer function of RLC circuit = ______________________

b) Plot in Fig. 2, the step response of RC, RL and RLC circuit given in Fig. 1.

c) Plot in Fig. 3, the impulse response.of RC, RL and RLC circuits given in Fig. 1.

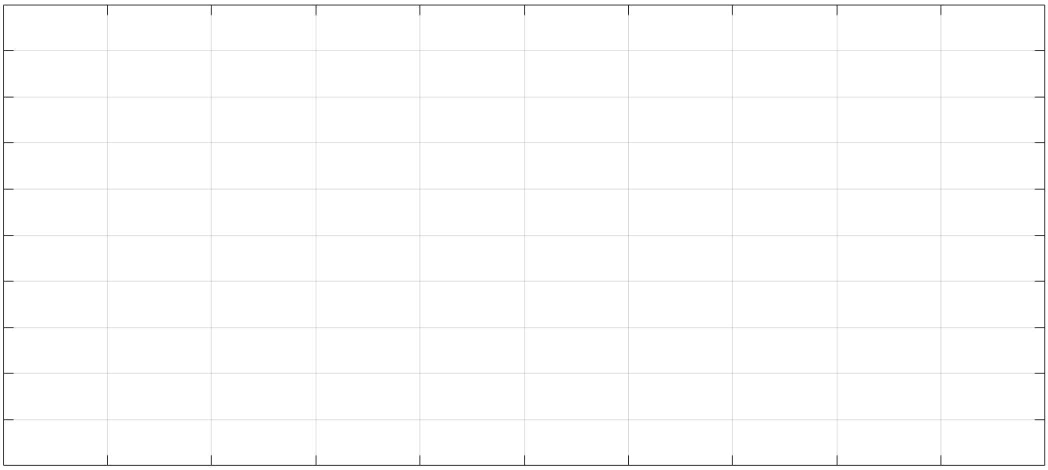


Figure 2: Step response of RC, RL and RLC circuit.

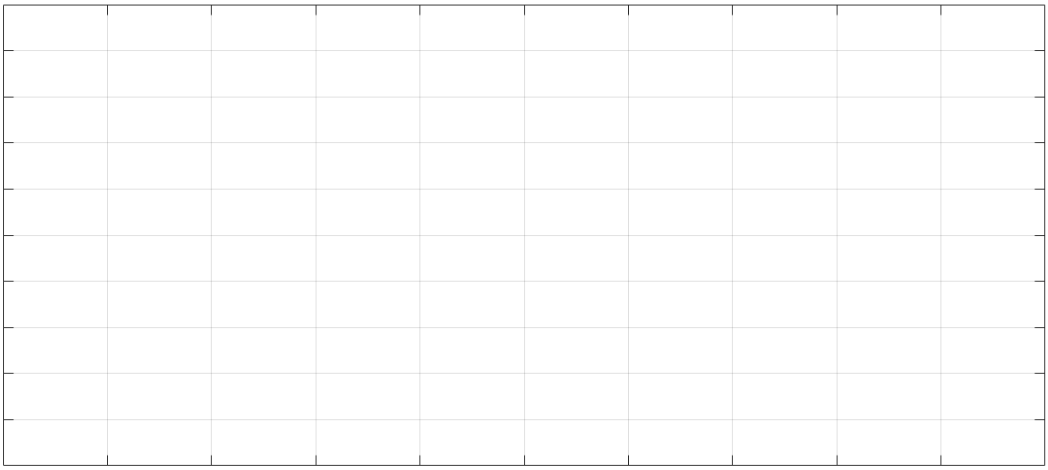


Figure 3: Impulse response of RC, RL and RLC circuit.

d) Plot the bode plots, of the three circuits, in Fig. 4.

e) Determine the cut off frequency and 3dB bandwidths using your answers to part d.

f_{cutoff} of RC circuit = ________________- Bandwidth of RC circuit = ________________-

f_{cutoff} of RL circuit = ________________- Bandwidth of RL circuit = ________________-

f_{cutoff} of RLC circuit = ________________- Bandwidth of RLC circuit = ________________-

f) Draw the pole zero diagram of the three circuits in Fig. 5.

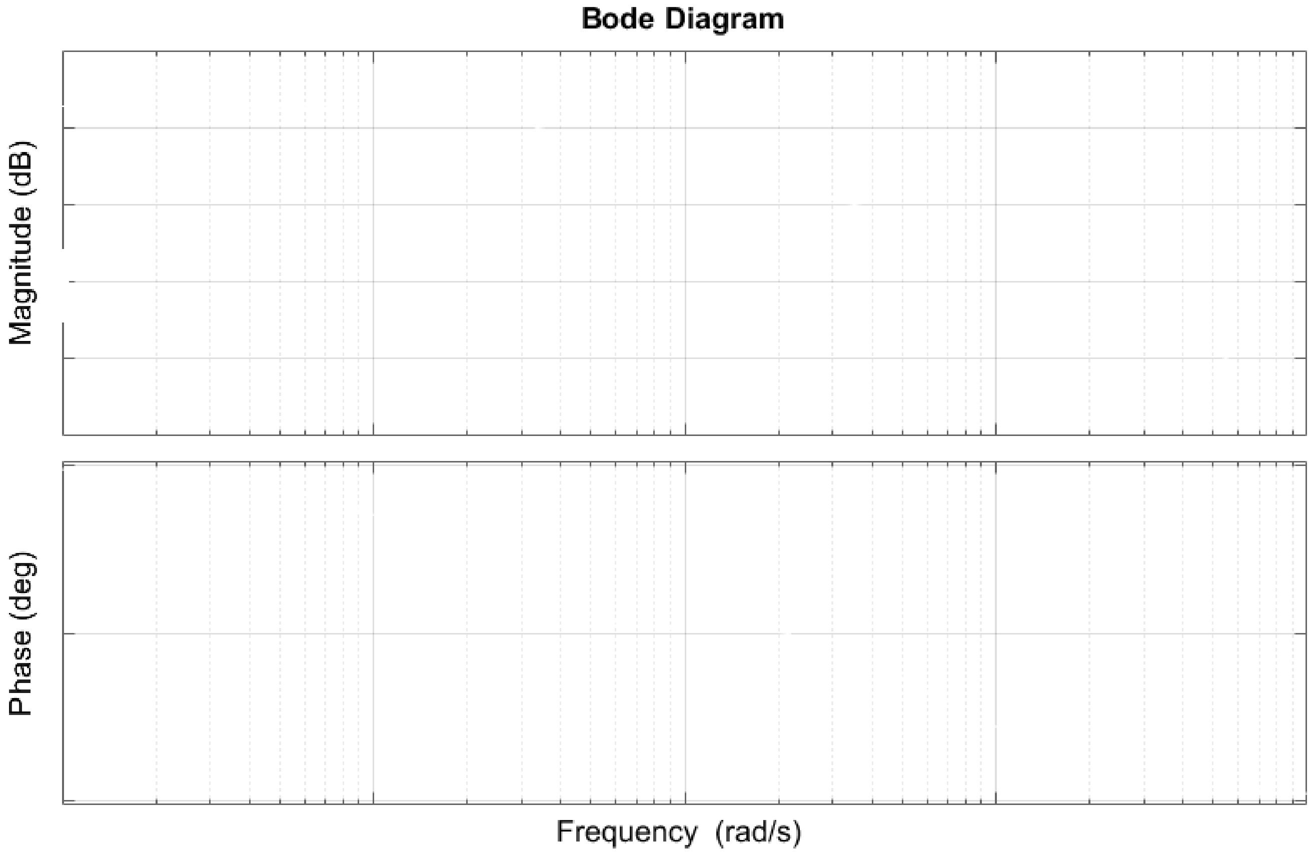


Figure 4: Bode plot of RC, RL and RLC circuit.

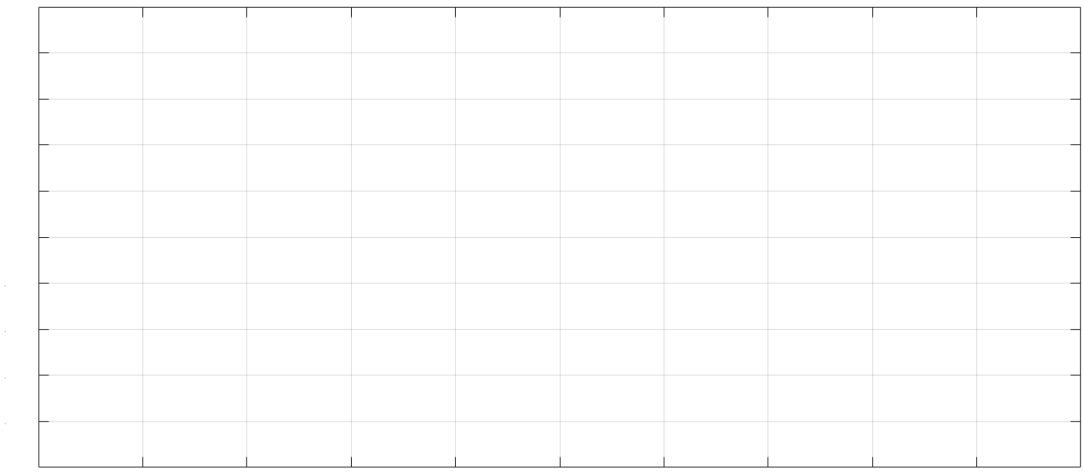


Figure 5: Pole zero plot of RC, RL and RLC circuit.

Task 2: For the series RLC circuit, given in Task 1, use your Bode plot to estimate the frequency for which the phase is zero radians. Also compute the actual frequency where the phase is zero.

(*Hint: The phase will be zero when the numerator and denominator phases are equal. You can use the MATLAB solve function). Use your Bode plot to determine the output $v_{out}(t)$ when the input $v_{in}(t) = 5cos(200\pi t + \pi/6)$

Task 3: Determine the transfer function and Bode plot for the circuit shown in Fig. 6 using your answers of Task1 (a,b). *(Hint: You can use convolution function).*

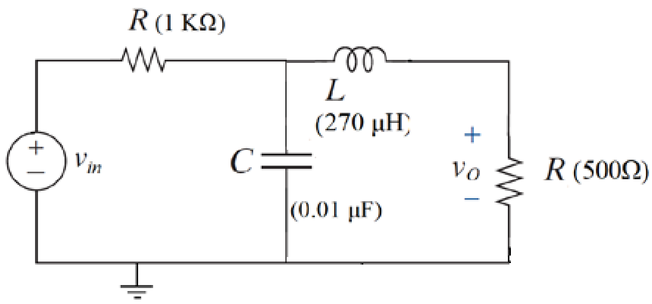


Figure 6: Task 3

Name: —————————— Registration #: ——————————

EXPERIMENT NO 8

To plot the frequency (consisting of magnitude and phase) response of passive filters.

Objectives:

The objective of this lab is to observe and plot the magnitude & phase response of first and second order filters.

Apparatus

- Oscilloscope
- Signal Generator
- Capacitor (0.1 μF, 1 nF or any capacitor of standard value in the Lab)
- Resistors(10 KΩ /20 KΩ /Use resistor of standard value available in the Lab)
- Inductor (270 μ H/ similar value available in the Lab)

Theoretical Background

Using various combinations of resistors capacitors and inductors, circuits can be built that have the property of passing or rejecting either low or high frequencies or band of frequencies. These frequency selective networks, which alter the amplitude and phase characteristics of the input ac signal, are called filters. Thus filter is an AC circuit that separates some frequencies from other frequencies within a composite signal of various frequencies.

Electronic filters can be passive or active. Passive implementations of filters are based on combinations of resistors, capacitors and inductors. These types are collectively known as passive filters, because they do not depend upon an external power supply and/or they do not contain active components such as transistors. Active filters are implemented using a combination of passive and active (amplifying) components, and require an outside power source.

Critical Frequency:

The critical frequency is the frequency at which the filter's output voltage is 70.7 % of the maximum. The filter's critical frequency is also called the cutoff frequency, break frequency, or -3 dB frequency because the output voltage at this frequency is 3 dB less than its maximum value. The term dB (decibel) is a commonly used in filter measurements and is explained below.

Decibels

The basis for the decibel unit stems from the logarithmic response of the human ear to the intensity of sound. The decibel is a logarithmic measurement of the ratio of one power to another. For example for filters this ratio can be expressed as output power from a filter to input power given to a filter.

$$decibel = 10log_{10}\frac{P_{output}}{P_{input}}$$

As electric power across a resistor can be expressed as

$$Power = P = \frac{V^2}{R}$$

Assuming that input and output power are measured in terms of voltages across same resistor

$$decibel = 10log_{10}\frac{{V_{output}}^2/R}{{V_{input}}^2/R} \tag{1}$$

$$decibel = 20log_{10}\frac{V_{output}}{V_{input}} \tag{2}$$

Passive first order filters

Low Pass RC Filter

A Low Pass Filter (LPF) allows signals of frequencies lower than cutoff frequency to pass from input to output while rejecting higher frequencies.

Task1

1. Patch the RC circuit given in Fig. 1 on the breadboard and apply sinusoidal signal from the signal generator. Adjust the magnitude of the sinusoidal signal to 10 V peak and 100 Hz.

2. Display simultaneously voltage v_{in} across the function generator (on CH 1) and v_{out} across the capacitor C (on CH 2). Ensure common ground for both channels and the signal generator. Measure magnitude of v_{out} and its phase with respect to v_{in}, and note in Table 1.

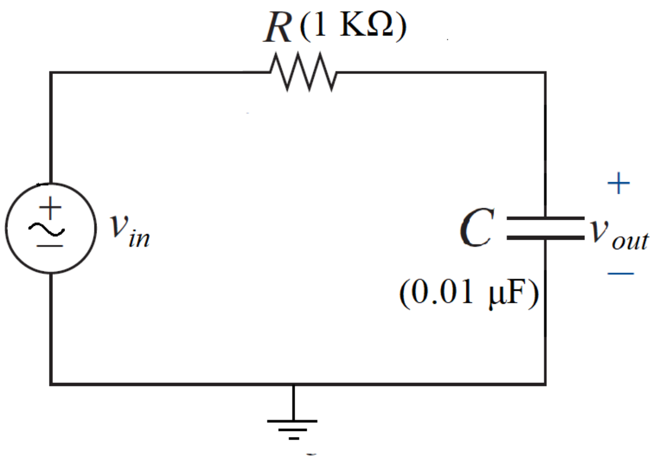


Figure 1: Low Pass RC filter

3. Vary the frequency of input signal, measure and calculate the gain and phase and note in Table 1.

4. Plot the magnitude and phase of RC Low pass filter based on the parameters noted in Table 1.

Table 1: Low Pass RC Filter

No.	f_{in} (Hz)	ω_{in} (rad/s)	v_{in} (V)	v_{out} (V)	Gain $= \frac{v_{out}}{v_{in}}$ (dB)	Phase (degrees)
1						
2						
3						
4						
5						
6						
7						
8						

High Pass RC Filter A high pass filter allows signals with higher frequencies greater than cutoff frequency, to pass from input to output while rejecting lower frequencies.

Task2

1. Patch the RC circuit given in Fig. 1 by interchanging the resistor and capacitor positions and apply sinusoidal signal from the signal generator. Adjust the magnitude of the sinusoidal signal to 10 V peak and 100 Hz.

2. Display simultaneously voltage v_{in} across the function generator (on CH 1) and v_{out} across the resistor R (on CH 2). Ensure common ground for both channels and the signal generator. Measure magnitude of v_{out} and its phase with respect to v_{in}, and note in Table 2.

3. Vary the frequency of input signal, measure and calculate the gain & phase, and note in Table 2.

4. Plot the magnitude and phase of RC Low pass filter based on the parameters noted in Table 2.

Table 2: High Pass RC Filter

No.	f_{in} (Hz)	ω_{in} (rad/s)	v_{in} (V)	v_{out} (V)	Gain $= \frac{v_{out}}{v_{in}}$ (dB)	Phase (degrees)
1						
2						
3						
4						
5						
6						
7						
8						

High Pass RL Filter

Task3

1. Patch the RL circuit given in Fig. 2 on the breadboard and apply sinusoidal signal from the signal generator. Adjust the magnitude of the sinusoidal signal to 10 V peak and 100 Hz.

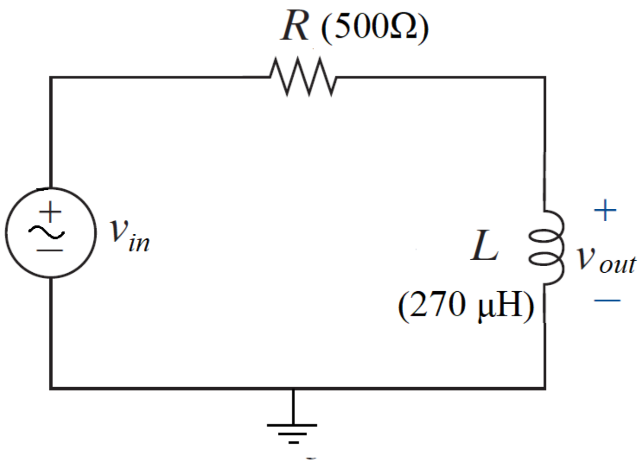


Figure 2: High Pass RL filter

2. Display simultaneously voltage v_{in} across the function generator (on CH 1) and v_{out} across the inductor L (on CH 2). Ensure common ground for both channels and the signal generator. Measure magnitude of v_{out} and its phase with respect to v_{in}, and note in Table 3.

3. Vary the frequency of input signal, measure and calculate the gain and phase and note in Table 3.

4. Plot the magnitude and phase of RL Low pass filter based on the parameters noted in Table 3.

<u>*Low Pass RC Filter*</u>

Task4

1. Patch the RL circuit given in Fig. 1 by interchanging the resistor and inductor positions and apply sinusoidal signal from the signal generator. Adjust the magnitude of the sinusoidal signal to 10 V peak and 100 Hz.

2. Display simultaneously voltage v_{in} across the function generator (on CH 1) and v_{out} across the resistor R (on CH 2). Ensure common ground for both channels and the signal generator. Measure magnitude of v_{out} and its phase with respect to v_{in}, and note in Table 4.

3. Vary the frequency of input signal, measure and calculate the gain & phase, and note in Table 4.

4. Plot the magnitude and phase of RC Low pass filter based on the parameters noted in Table 4.

Table 3: High Pass LC Filter

No.	f_{in} (Hz)	ω_{in} (rad/s)	v_{in} (V)	v_{out} (V)	Gain $= \frac{v_{out}}{v_{in}}$ (dB)	Phase (degrees)
1						
2						
3						
4						
5						
6						
7						
8						

Table 4: Low Pass LC Filter

No.	f_{in} (Hz)	ω_{in} (rad/s)	v_{in} (V)	v_{out} (V)	Gain $= \frac{v_{out}}{v_{in}}$ (dB)	Phase (degrees)
1						
2						
3						
4						
5						
6						
7						
8						

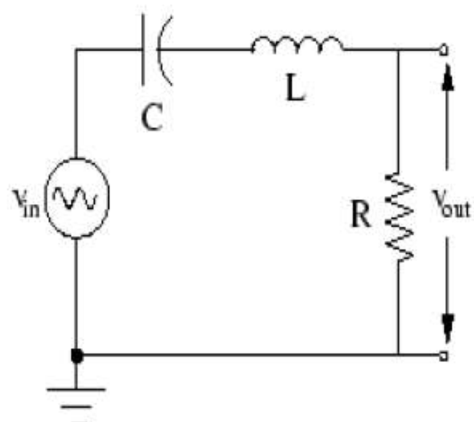
University of Engineering and Technology
Department of Electrical Engineering
EE 110 Circuit Analysis and Design Laboratory

Experiment 9: To plot the magnitude and phase response of a series resonant band-pass filter.

Equipment and Components required:

- Signal Generator
- Oscilloscope
- Multimeter
- Capacitor: 0.01 μ F (use capacitor of any value available in the lab)
- Inductor 100-200 mH (use inductor of any value available in the lab)
- Resistors (1/4W): 1KΩ, 2KΩ ±5 percent

Circuit to be patched on breadboard:



Theory:

Band pass Filter

It allows a certain band of frequencies to pass and attenuates or rejects all frequencies below and above the pass band. A combination of low-pass and high-pass filter can be used to form band pass filters.

Low-Pass and High-Pass Filters used to form a band-pass filter

Operation of Series Resonant Band Pass Filter

A series resonant filter has minimum input impedance. At critical frequency the inductor and the capacitor in series behave as simple resistor. Hence making maximum output across the load resistor.
At the frequencies other than resonant frequencies, the reactance offered by the inductor or capacitor is very large, hence output voltage will be very small at high as well as at low frequencies.

Bandwidth

The bandwidth of a band-pass filter is the range of frequencies for which the current, and therefore the output voltage, is equal or greater than 70.7% of its value at the resonant frequency.

Mathematically, $\text{Bandwidth} = \dfrac{\text{Resonant Frequency } f_r}{\text{Quality Factor } Q}$

Quality factor

Quality Factor is the ratio of reactive power developed in inductor or capacitor to average power dissipated in resistor.

$$\text{Quality Factor} = \frac{\text{Reactive power developed in inductor or capacitor}}{\text{Average power dissipated in resistor}}$$

Quality Factor indicates the selectivity of the filter and can be expressed as,

$$\text{Quality Factor} = \frac{\omega L}{R} = \frac{2\pi f_r L}{R}$$

Laboratory Tasks:

1]. For the components used in the circuit, calculate and record the resonant frequencies for the circuit in the fig. Calculate, also, the circuit-Q and bandwidth of the circuit.

2]. Construct the circuit shown in fig on the breadboard.

3]. At a frequency of 500Hz, adjust V_{in} to some convenient value, such as 5V rms.

4]. Use multimeter to measure V_o and record it in table.

5]. Vary the frequency, measure and record V_o while maintaining Vin constant.

6]. Complete the decibel gain row of the table.

Plot the decibel voltage gain ratio versus log frequency.

OBSERVATIONS & CALCULATIONS

Resonant Frequency $f_r = 1/\ 2\pi\sqrt{LC}$
Quality Factor $Q = \omega L\ /\ R$
Bandwidth = fr / Q

No.	Input Frequency f (Hz)	Input Voltage Vin (volts)	Output Voltage Vo (volts)	Vo/ Vin (volts)	db = 20 log (Vo/Vin)	$\theta = \tan^{-1}(X_L$ - Xc/R) (degrees)

Laboratory Report:

1]. Plot on semilog paper:

I. db against frequency

II. Phase angle against frequency

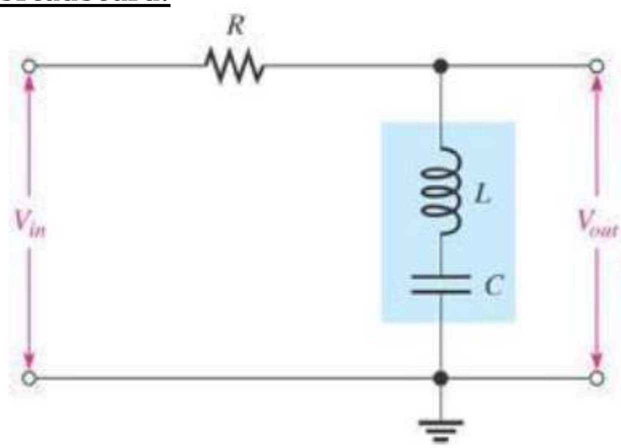
University of Engineering and Technology
Department of Electrical Engineering
EE 110 Circuit Analysis and Design Laboratory

Experiment 10: To plot the magnitude and phase response of a series resonant band-stop filter.

Equipment and Components required:

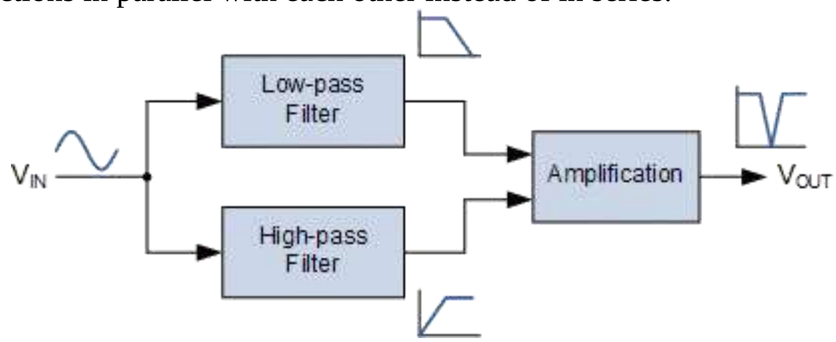
- Signal Generator
- Oscilloscope
- Multimeter
- Capacitor: 0.01 µ F (use capacitor of any value available in the lab)
- Inductor 100-200 mH (use inductor of any value available in the lab)
- Resistors (1/4W): 1KΩ, 2KΩ ±5 percent

Circuit to be patched on breadboard:



Band Stop Filter

It is a filter that rejects a certain band or range of frequencies while passing all frequencies below and above the rejected band. Band stop filters block signals occurring between two given frequencies, FL and FH . It can be made out of a low-pass and a high pass filter by connecting the two filter sections in parallel with each other instead of in series.



Low-Pass and High-Pass Filters used to form a band-stop filter

Operation of Bandstop filter

When the series LC combination reaches resonance, its very low impedance shorts out the signal, dropping it across resistor R1 and preventing its passage on to the load. Thus, within the band at which the resonant frequency occurs, there is a relatively less output and that set of frequencies are attenuated.
At frequencies other than resonant frequencies, the reactance offered by inductor and capacitor is very large, thus outside the band at which resonant frequency occurs, there is large output and that set of frequencies are passed to the output.

Corner Frequency

Because a real filter rolls off gradually, you usually specify the corner frequency as the frequency at which the response is 1 / 2 (0.707) of that in the pass band. Because electronic engineers traditionally describe relative signal strengths in decibels, the frequency is also referred to as 3-dB point.

Laboratory Tasks:

1]. For the components used in the circuit, calculate and record the resonant frequencies for the circuit in the fig. Calculate, also, the circuit-Q and bandwidth of the circuit.

2]. Construct the circuit shown in fig on the breadboard.

3]. At a frequency of 500Hz, adjust V_{in} (sinusoidal) to some convenient value, such as 5V rms.

4]. Use oscilloscope to measure V_o and record it in table.

5]. Vary the frequency, measure and record V_o while maintaining Vin constant.

6]. Complete the decibel gain row of the table.

Plot the decibel voltage gain ratio versus log frequency.

OBSERVATIONS & CALCULATIONS

Resonant Frequency $f_r = 1/ 2\pi\sqrt{LC}$
Quality Factor $Q = \omega L / R$
Bandwidth = fr / Q

No.	Input Frequency f (Hz)	Input Voltage Vin (volts)	Output Voltage Vo (volts)	Vo/ Vin (volts)	db = 20 log (Vo/Vin)	$\theta = \tan^{-1}(X_L - Xc/R)$ (degrees)

Laboratory Report:

1]. Plot on semilog paper:

I. db against frequency

II. Phase angle against frequency

University of Engineering and Technology
Department of Electrical Engineering
EE 110 Circuit Analysis and Design Laboratory

Experiment 11: Implementation of 1st order Active LP and HP filters.

When a filter contains a device like an Op Amp they are called active filters. These active filters differ from passive filters (simple RC circuits) by the fact that there is the ability for gain depending on the configuration of the elements in the circuit. There are some problems encountered in active filters that need to be overcome. The first is that there is still a gain bandwidth limitation that arises. The second is the bandwidth in general. In a high pass filter there is going to be high frequency roll off due to the limitations of the Op Amp used. This is very hard to overcome with conventional op amps. However, active filters have three main advantages over passive filters: Inductors can be avoided. Without them, passive filters cannot obtain a high Q (low damping), but inductors are often large and expensive (at low frequencies), may have significant internal resistance, and may pick up surrounding electromagnetic signals. The shape of the response and the tuned frequency can often be set easily by varying resistors, in some filters one parameter can be adjusted without affecting the others. Variable inductances for low-frequency filters are not practical. The amplifier powering the filter can be used to buffer the filter from the electronic components it drives or is fed from, variations in which could otherwise significantly affect the shape of frequency response.

IMPLEMENTATION OF LOW PASS FILTER:

The low pass filter is one that allows low frequencies and stops (attenuates) higher frequencies, hence the name. The design of a low pass filter needs to take into consideration the maximum frequency that would need to be allowed through. This is called the cut off frequency (or the 3 dB down frequency). Based on the type of filter that is used (e.g. Butterworth, Bessel, Chebyscheff) the attenuation of the higher frequencies can be greater. This attenuation is also based on the order (e.g. 1st, 2nd, 3rd…) of the filter that is used. Based on the order of the filter the roll-off of the filter can be calculated using the formula –n*20 dB/decade. This means that a first order low pass filter has an attenuation of -20 dB/decade, while a second order filter should have -40 dB/decade roll-off and on down the list for higher orders. Shown in Figure 1 is the basic active 1st order low pass filter (in the non-inverting configuration) with unity gain.

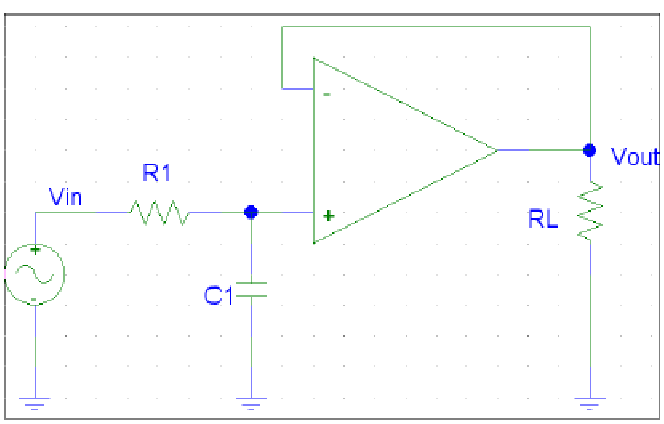


Figure 1: Basic 1st Order Bessel LP Filter in the Non-Inverting Configuration with Unity Gain

The equation in (1) is used to calculate the value of the capacitor needed based on a chosen value for cutoff frequency and R_1 (or vice versa if a value for C_1 and a cutoff frequency are chosen then the value of R_1 can be found). There is unity gain in this configuration because of the non-inverting properties of the Op Amp. To change the gain, the feedback network must be changed to include two other resistors (R_2 and R_3). The gain is then found to be $1 + R_3/R_2$ because of the non-inverting configuration. The circuit with a non-unity gain is shown in Figure 2.

$$fc = \frac{1}{2\pi R_1 C_1} \qquad (1)$$

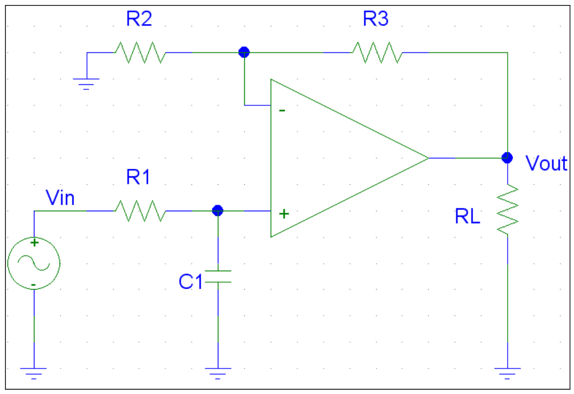


Figure 2: Basic 1st Order Bessel LP Filter in the Non-Inverting Configuration with Non-unity Gain

Equipment and Components required:

- **Oscilloscope**
- **Signal Generator**
- **Capacitor**
- **Resistors(Use resistor of standard value available in the Lab)**

Circuit to be patched on breadboard:

- **Patch both the above circuits and verify the output of Low pass filter with unity and no unity gain.**

Laboratory Tasks:

1. Choose appropriate value of R1 and C1. R1= 70mV/ 500nA, so R1 comes out to be 140kohms. You can use standard value of 120kohms. Similarly use appropriate value of C1 (0.001micro) so that cut-off frequency is not very small. For the filter with non-unity gain set the Gain=5 and then implement it.

2. Apply a 1 Vpp 100 Hz signal as input to the network of LPF and measure the corresponding output voltage level. Determine the decibel gain of the filter.

G (dB) = 201og [Vo/Vin]

3. Repeat step 1 and 2 for three frequencies lesser than the critical frequency, one at critical frequency and at three frequencies greater than the critical frequency.

Student Name: __________________ Student Reg. & Section: _______________

Observations & Calculations:

Cutoff Frequency: f_c = ______________

No.	Input Frequency f (Hz)	Input Voltage Vin (volts)	Output Voltage Vo (volts)	Vo/ Vin (volts)	db = 20 log (Vo/Vin)

IMPLEMENTATION OF HIGH PASS FILTER:

High Pass Filter: A high pass filter allows signals with higher frequencies to pass from input to output while rejecting lower frequencies. If creating a low pass filter was easy, then creating a high pass filter is even easier. In the case of the 1st order Bessel LP filter the capacitor and resistor only need to be interchanged with each other and the result is a high pass filter. The same equation holds for finding the cutoff frequency and is shown in (1). The circuit shown in Figure 4 is that of a 1st order Bessel HP filter with unity gain. The gain can be adjusted to the non-unity case by adding the feedback network resistors in the same location as the LP circuit of Figure 2.

Circuit to be patched on breadboard:

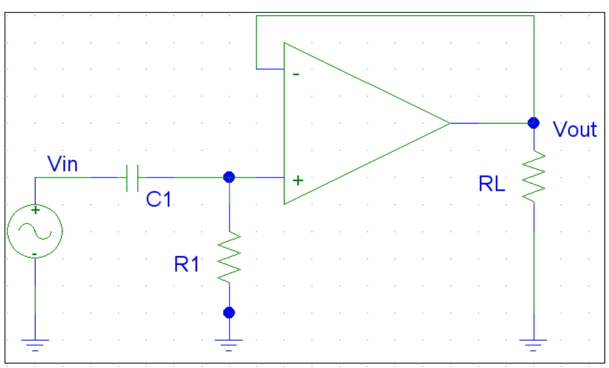


Figure 3: 1st Order Bessel HP Filter with unity gain

Student Name: ___________________ Student Reg. & Section: ________________

Observations & Calculations:

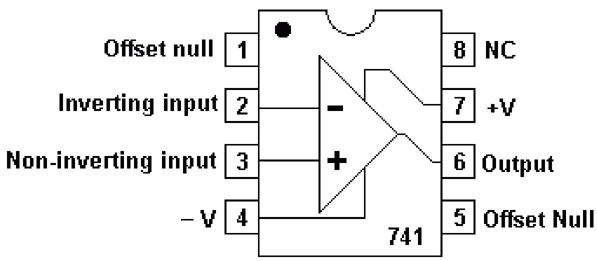
(Choose C1=0.001micro Farads and find the value of R1 for any value of f_c (say f_c=5 kHz))

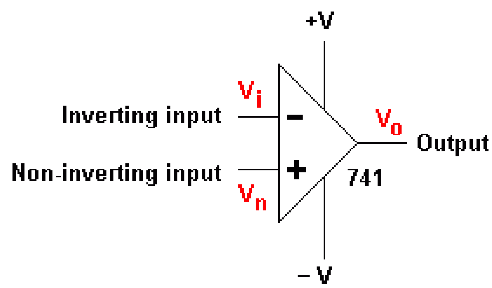
Cutoff Frequency: f_c = ______________

No.	Input Frequency f (Hz)	Input Voltage Vin (volts)	Output Voltage Vo (volts)	Vo/ Vin (volts)	db = 20 log (Vo/Vin)

Pin Layout of OP-AMP:

Biasing Voltage = +15V





Laboratory Report:

1. Plot the graph between gain against frequency for low pass filters?
2. What is the gain of the First Order LP Filter in the pass band? How does this compare to the theoretical value? What is the roll off rate of the filter in the stop band?
3. Plot the graph between gain against frequency for high pass filter?
4. Find the cut-off frequency from graphs and compare it with calculated value?