COMSATS University Islamabad (CUI), Lahore Campus
Department of Electrical & Computer Engineering

CPE-222 – Electric Circuit Analysis II

Lab Manual for Spring 2023

Lab Resource Person

Sarmad Hassan

Theory Resource Person

Dr. Mirza Tariq Hamayun

Name: ____________________________ **Registration Number:** *CIIT/ - - /LHR*

Program: _________________________ **Batch:** ____

Semester _____________________________

Revision History

Sr. #.	Activity	Date	Performed by
1	Lab Manual Preparation	02/09/2013	Engr. Inam Khan
2	Lab Manual Review	23/09/2013	Dr. Mujtaba Hussain Jaffery
3	Layout modifications	03/09/2015	Engr. Masood Ahmad
4	Lab Manual Review	29/09/2015	Dr. Mujtaba Hussain Jaffery
5	Lab Manual Review	25/08/2016	Dr. Mirza Tariq Hamayun
6	Lab Manual Modifications	26/08/2021	Engr. Arfa Tariq
7	Lab Manual Update and Modifications	30/09/2022	Ameer Hamza Ghazala Mushtaq Arfa Tariq Talha Raheem

Preface

This course takes the students beyond simply analyzing circuits and helps them to develop the skills needed to solve problems, choosing the best design from several competing solutions. Understanding of AC circuits, ability to analyze the poly-phase circuits and measuring the AC power is the main objective of this course. It also helps students to analyze the two port circuits which will be useful to all Engineers.

Following are the major topics covered in this course.

- Natural and Step Response of a Parallel RLC circuit
- Steady State Response of RC and RL circuits
- Network Theorems (KCL, KVL and Thevenin Theorem) in Phasor Domain
- Power Measurements (Real, Reactive and Apparent Power) and Power Factor Correction
- Measurement of voltage, current, and power in a balanced three phase wye-connected circuit
- Measurement of voltage, current, and power in a balanced three phase delta-connected circuit
- Design of RC low pass & high pass Filter
- Design of RL low pass & high pass Filter
- Resonance in series RLC circuit
- Resonance in parallel RLC circuit
- Design of passive Band Pass and Band Stop Filter
- Two port networks
- Design of second order and third order RC low pass Filter

Books

Text Books

1. Fundamentals of Electric Circuits (Fifth Edition), C. K. Alexander, M. N. O. Sadiku, McGraw Hill, 2006.
2. Hayt, Kemmerly, Durbin, Engineering Circuit Analysis, McGraw Hill, 2006 (seventh edition)

Reference Books

1. Electric Circuits, J. W. Nilsson, Addison-Wesley 1996 (fifth edition)
2. Basic Engineering Circuit Analysis, J. D. Irwin, Prentice Hall 1999 (sixth edition)
3. Introduction to Electric Circuits, R. C. Dorf, Wiley 1993 (second edition)

Learning Outcomes

Theory CLOs

After successfully completing this course, the students will be able to:

1. Solve the single and three phase ac circuits using the concepts of sinusoids, phasors, and basic circuit analysis theorems. (PLO1, C3)
2. Analyse the two-port networks and LTI systems using the concepts of frequency response and Laplace transformation. (PLO2, C4)

Lab CLOs

After successfully completing this course, the students will be able to:

3. To design and compute the parameters for single and three phase electric circuits using standard circuit analysis techniques. (C5 – PLO3)
4. To reproduce the single and three phase electric circuits and display the required parameters using simulation tool (LTspice), hardware platforms (breadboard, digital multi-meter (DMM) and digital storage oscilloscope (DSO)). (P3 – PLO5)

CLOs – PLOs Mapping

PLO / CLO	PLO1	PLO2	PLO3	PLO5	PLO10	Cognitive Domain	Affective Domain	Psychomotor Domain
Lab CLO1			x			C5		
Lab CLO2				x				P3

Lab CLOs – Lab Experiment Mapping

Lab / CLO	Lab 1	Lab 2	Lab 3	Lab 4	Lab 5	Lab 6	Lab 7	Lab 8	Lab 9	Lab 10	Lab 11	Lab 12
Lab CLO2	P2	P2	P2	P2, P3	P3	P3	P3	P3	P3	P3	P3	P3

* Lab CLO1 will be evaluated in Mid-term and Final lab exam

Grading Policy

The final marks for lab would comprise of Lab Assessment (25%), Lab Midterm (25%), and Lab Terminal (50%).

Lab Assignments:

i. Lab Assignment 1 Marks (Lab marks from Labs 1-3)
ii. Lab Assignment 2 Marks (Lab marks from Labs 4-6)
iii. Lab Assignment 3 Marks (Lab marks from Labs 7-9)
iv. Lab Assignment 4 Marks (Lab marks from Labs 10-12)

Lab Mid Term = 0.5*(Lab Mid Term exam) + 0.5*(average of lab evaluation of Lab 1-6)

Lab Terminal = 0.5*(Lab Terminal Exam) +0.375*(average of lab evaluation of Lab 7-12) + 0.125*(average of lab evaluation of Lab 1-6)

The minimum pass marks for both lab and theory shall be 50%. Students obtaining less than 50% marks (in either theory or lab, or both) shall be deemed to have failed in the course. The final marks would be computed with 75% weight to theory and 25% to lab final marks.

List of Equipment

- Oscilloscope
- Bread Board Panel
- Function Generator
- Digital Multi-meter
- Resistors
- Capacitors
- Inductors
- Three Phase circuit Board

Software Resources

- LT Spice

Lab Instructions

- This lab activity comprises of three parts: Pre-lab, Lab Tasks and Viva session.
- The students should perform and demonstrate each lab task separately for step-wise evaluation.
- Only those tasks that are completed during the allocated lab time will be credited to the students.
- Students are however encouraged to practice on their own in spare time for enhancing their skills.

Safety Instructions

The following general rules and precautions are to be observed at all times in the laboratory. These rules are for the benefit of the experimenter as well as those around him/her. Additional rules and precautions may apply to a particular laboratory.

1. There must be at least two (2) people in the laboratory while working on live circuits.
2. Remove all loose conductive jewelry and trinkets, including rings, which may come in contact with exposed circuits.
3. When making measurements, form the habit of using only one hand at a time. No part of a live circuit should be touched by the bare hand.
4. Keep the body, or any part of it, out of the circuit. Where interconnecting wires and cables are involved, they should be arranged so people will not trip over them.
5. Be as neat as possible. Keep the work area and workbench clear of items not used in the experiment.
6. Always check to see that the power switch is OFF before plugging into the outlet. Also, turn the instrument or equipment OFF before unplugging from the outlet.
7. When unplugging a power cord, pull on the plug, not on the cable.
8. When disassembling a circuit, first remove the source of power.
9. "Cheater" cords and 3-to-2 prong adapters are prohibited unless an adequate separate ground lead is provided, the equipment or device is double insulated, or the laboratory ground return is known to be floating.
10. No ungrounded electrical or electronic apparatus is to be used in the laboratory unless it is double insulated or battery operated.
11. Keep fluids, chemicals, and beat away from instruments and circuits.
12. Report any damage to equipment, hazards, and potential hazards to the laboratory instructor.
13. If in doubt about electrical safety, see the laboratory instructor. Regarding specific equipment, consult the instruction manual provided by the manufacturer of the equipment. Information regarding safe use and possible- hazards should be studied carefully.

Table of Contents

Revision History ... i
Preface ... ii
Books ... iii
Learning Outcomes ... iii
CLOs – PLOs Mapping ... iii
Lab CLOs – Lab Experiment Mapping ... iii
Grading Policy ... iv
List of Equipment ... iv
Software Resources ... iv
Lab Instructions ... iv
Safety Instructions ... v
Table of Contents ... vi
LAB # 1: To explain the procedure for generation and display of AC signal using hardware tool ... 1
- Objectives ... 1
- Pre-Lab ... 1
- In Lab Task ... 6
- Post Lab Task ... 8

LAB # 2: To explain natural and step response of parallel RLC circuits using software tool ... 9
- Objectives ... 9
- Pre-Lab ... 9
- Pre-Lab Task ... 12
- In Lab Tasks ... 12
- Post lab Task ... 17

LAB # 3: To Display the Sinusoidal Steady State Response of RC and RL circuits using hardware & software tools ... 19
- Objectives ... 19
- Pre-Lab ... 19
- Pre-Lab Exercise ... 21
- In Lab Task ... 23
- Post lab Task ... 24

LAB # 4: To Reproduce the circuits in Ltspice and Show the Network theorems in phasor domain using software tool ... 27
- Objectives ... 27
- Pre-Lab ... 27
- In Lab Task ... 27
- Post Lab Task ... 30

LAB # 5: To Reproduce the circuit and show the Power (Active, Reactive and Apparent) and Power Factor Correction using software tool 32

Objectives 32

Pre-Lab 32

Pre Lab Exercise 34

In Lab Task 35

Post Lab Task 36

LAB # 6: To reproduce three phase circuits using software tool 38

Objectives 38

Pre-Lab 38

In Lab Task 41

Post Lab Task 45

LAB # 7: To reproduce RC Low and High pass Filter by selecting appropriate components using hardware & software tools 47

Objectives 47

Pre-Lab 47

Pre-Lab Exercise 49

In Lab Task 49

Post Lab Task 51

LAB # 8: To reproduce RL Low and High pass Filter by selecting appropriate components using hardware & software tools 53

Objectives 53

Pre-Lab 53

Pre-Lab Exercise 55

In Lab Tasks 55

Post Lab Task 57

LAB # 9:To reproduce Passive filters (Band pass, Band stop) by selecting appropriate components using software tool 59

Objective 59

Pre-Lab 59

Pre lab Exercise 60

In Lab Task 60

Post Lab Task 61

LAB # 10: To reproduce series and parallel resonance RLC circuit using software tool 63

Objectives 63

Pre Lab 63

Pre Lab Exercise 67

In Lab Task.. 68

Post Lab Task.. 69

LAB # 11: To reproduce the circuit and show the impedance, admittance and hybrid parameter for unknown two port network using software tool 71

Objectives 71

Pre Lab 71

Pre Lab Exercise 73

In Lab Task.. 73

Post Lab Task.. 75

LAB # 12: To reproduce second and third order RC low pass filter circuits using hardware & software tool 77

Objectives 77

Pre Lab 77

Pre Lab-Exercise.. 78

In Lab Task.. 78

Post Lab Task.. 79

LAB # 1: To explain the procedure for generation and display of AC signal using hardware tool

Objectives

- To explain the procedure for generating the arbitrary waveforms using function generator.
- To explain the basic understanding of viewing a time varying voltage waveform and various control knobs using digital oscilloscope.

Pre-Lab

Oscilloscope

An oscilloscope is a graph displaying device. It is used to visualize time-varying electronic signals on a screen. The signals are graphed using an analogue circuitry or a digital apparatus.

Analog Oscilloscope

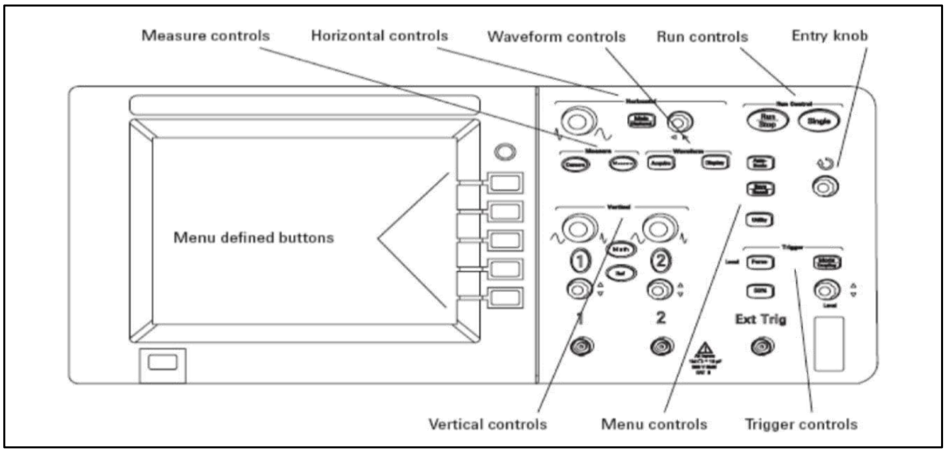
It works on the functionality of Cathode Ray Tube (CRT). A beam of electrons is made to fall on a screen where it becomes visible as a bright blue dot. The beam is then moved along a horizontal line using a sawtooth voltage applied along the horizontal axis. The fast moving dot gives the appearance of a blue line. Then the signal to be graphed on the screen is applied vertically so that the beam of electrons moves in a vertical access accordingly. The result is a plot of the time varying applied signal on the oscilloscope screen.

Digital Oscilloscope

It works on the functionality of Analog to-Digital (A/D) converter. The applied input analog signal is sampled at a high rate; the received samples are then plotted on the screen.

Digital Oscilloscopes have some obvious benefits over analog counterparts. The advent in digital circuitry has made the oscilloscopes cheaper. Moreover, the sampled signal in a digital oscilloscope can be stored in memory, can be easily modified or transferred to a computer for further analysis. In this lab we shall conduct all experiments using digital oscilloscopes. However, it is advised that engineering students should get hands-on experience on both types of +oscilloscopes.

In our lab (i.e. Electronics Lab: CIIT Lahore) we shall work with Agilent 3000 Series oscilloscopes.



. *Figure1- 1:Agilent 3000 Series Oscilloscope (Panel Controls).*

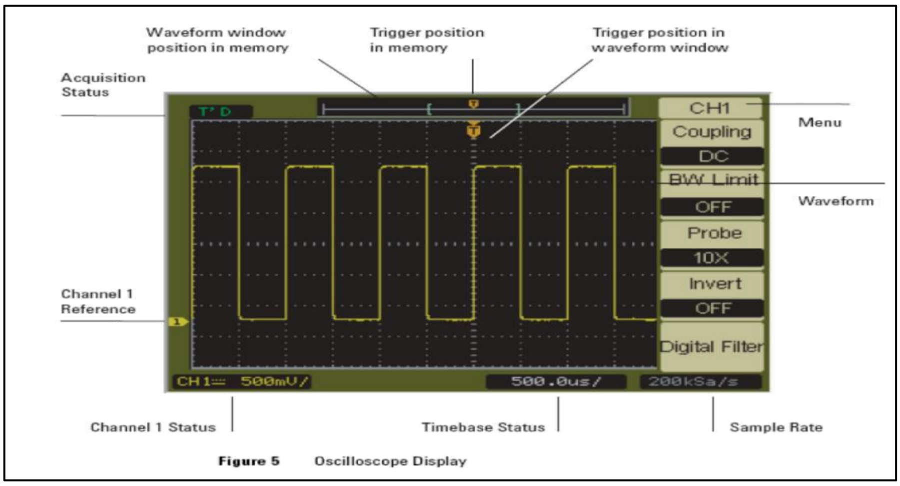


Figure1- 2:Oscilloscope Display.

Using the Oscilloscope

Auto –Scale Feature:

The oscilloscope has a very useful auto-scale feature that sets the various display scales automatically according to the input waveform. It is a handy tool to start with until the students get a better grasp at the control knobs. The students are advised not to rely on this feature completely but try to learn to set the scale parameters themselves.

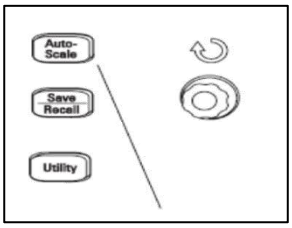


Figure1- 3: Auto Scale Button.

This feature requires an input frequency at least 50Hz and a duty cycle at least 1%.

Input a Signal:

Use one of the supplied passive probes to input the signal into one of the channels of the oscilloscope.

Using the Run Control Buttons:

There are two buttons for starting and stopping the oscilloscope's acquisition system: Run/Stop and Single.

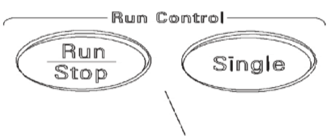


Figure1- 4:Run Control Buttons.

- When the Run/Stop button is green, the oscilloscope is acquiring data.
- To stop acquiring data, press Run/Stop. When stopped, the last acquired waveform is displayed.
- When the Run/Stop button is red, data acquisition is stopped. To start acquiring data, press Run/Stop.

- To capture and display a single acquisition (whether the oscilloscope is running or stopped), press Single. After capturing and displaying a single acquisition, the Run/Stop button is red.

Horizontal Scale:

The horizontal controls consist of:

- The horizontal scale knob — changes the oscilloscope's time per division setting using the center of the screen as a reference.
- The horizontal position knob — changes the position of the trigger point (trigger is explained in the subsequent section) relative to the centre of the screen.
- The Main/Delayed button (We shall not use this button in this lab: For details of this feature refer to the user manual of the device)

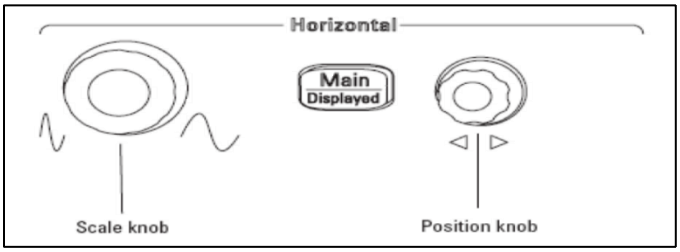


Figure1- 5:Horizontal Controls.

- Turn the horizontal scale knob to change the horizontal time per division (time/div) setting. The time/div setting changes in a 1- 2- 5 step sequence.
- Push the horizontal scale knob to toggle between Vernier (fine scale) adjustment and normal adjustment. With Vernier ak2djustment, the time/div setting changes in small steps between the normal (coarse scale) settings.
- The time/div setting is displayed in the status bar at the bottom of the screen.

Triggering:

The trigger determines when captured data should be stored and displayed. When a trigger is set up properly, it can convert unstable displays or blank screens into meaningful waveforms. When the oscilloscope starts to acquire a waveform, it collects enough data so that it can draw the waveform to the left of the trigger point. The oscilloscope continues to acquire data while waiting for the trigger condition to occur. After it detects a trigger, the oscilloscope continues to acquire enough data so that it can draw the waveform to the right of the trigger point.

The oscilloscope provides these trigger modes:

- Edge — can be used with analog and digital circuits. An edge trigger occurs when the trigger input passes through a specified voltage level with the specified slope.
- Pulse — is used to find pulses with certain widths.
- Video — is used to trigger on fields or lines for standard video waveforms.

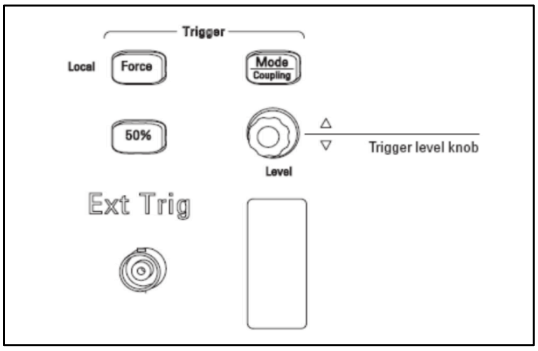


Figure1- 6:Trigger Controls.

- To adjust the trigger level, turn the trigger Level knob. Two things happen: The trigger level value is displayed at the lower left- hand corner of the screen and a line is displayed showing the location of the trigger level with respect to the waveform (except when using AC coupling or LF reject coupling modes).
- Push 50% to set the level at 50% of the signal's vertical amplitude.
- To make an acquisition even if no valid trigger has been found: Press Force.
- Forcing a trigger is useful, for example, when you want to display the DC voltage of a level signal.

Vertical Controls

The vertical controls consist of:

- The channel (1, 2) Math, and Ref buttons —turn waveforms on or off (display or hide their menus).
- The vertical scale knobs — change the amplitude per division setting for a waveform using ground as a reference.
- The vertical position knobs — change the vertical position of the waveform on the screen.

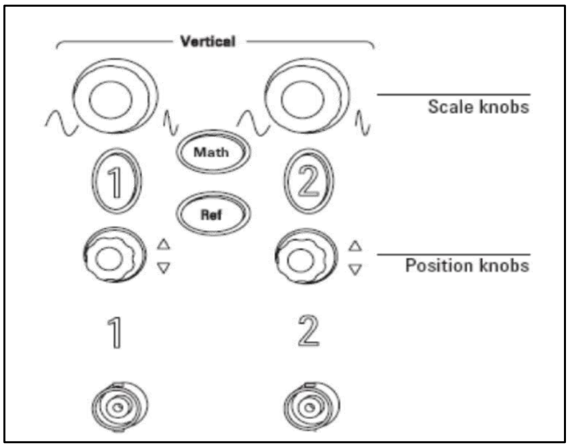


Figure1- 7:Vertical Controls.

Pressing the channel (1, 2), Math, or Ref buttons have the following effect: If the waveform is off, the waveform is turned on and its menu is displayed. If the waveform is on and its menu is not displayed, its menu will be displayed. If the waveform is on and its menu is displayed, the waveform is turned off and its menu goes away.

- Turn its vertical scale knob to change the amplitude per division setting. The amplitude/div setting changes in a 1- 2- 5 step sequence from 2 mV/div to 10 V/div (with “1X” probe attenuation). Ground is used as a reference.
- Push its vertical scale knob to toggle between Vernier (fine scale) adjustment and normal adjustment. With Vernier adjustment, the amplitude/div setting changes in small steps between the normal (coarse scale) settings.
- The amplitude/div setting is displayed in the status bar at the bottom of the screen.
- Adjusting their vertical position lets you compare waveforms by aligning them above one another or on top of each other. When an input channel waveform is on: Turn the vertical position knob to change the vertical position of the waveform on the screen. Notice that the ground reference symbol on the left side of the display moves with the waveform.
- Notice that, as you adjust the vertical position, a message showing the position of the ground reference relative to the center of the screen is temporarily displayed in the lower left- hand corner of the screen.
- To specify channel coupling, if the channel’s menu is not currently displayed, press the channel button (1, 2). In the Channel menu, press Coupling to select between:
- DC — passes both DC and AC components of the input waveform to the oscilloscope.
- AC — blocks the DC component of the input waveform and passes the AC component.
- GND — the waveform is disconnected from the oscilloscope input.

Function Generator

A function generator is a device to generate arbitrary time varying waveforms. It is used for testing and designing circuits in a lab environment. The function generator used in this lab is Agilent 3320A. Using the Function Generator

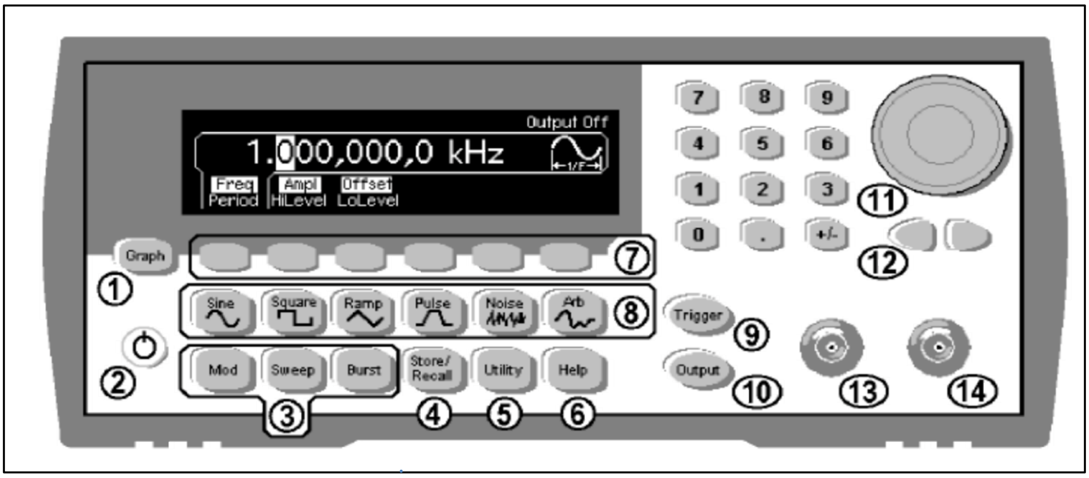


Figure1- 8: Agilent 3320A Function Generator.

Generating a Signal

- Turn the power on and press the output key.
- Press the sine key, if not already active.

- Enter a value of 2 using numeric keypad and then choose units to be kHz. We can also specify time period instead of frequency if we press the “Freq” soft key and then specify the time period.
- Similarly press the Amplitude soft key to enter amplitude and offset soft key to enter DC offset.
- The units can be changed by pressing first the +/- key and then entering new units.
- Similarly by pressing the square, ramp, pulse etc keys we can generate arbitrary waveforms of different characteristics.

In Lab Task

Task 1

- Turn on the oscilloscope and function generator.
- Note down the values of Channel 1 voltage axis and Time axis on the oscilloscope screen. Write them in the table 1.
- Generate a sinusoidal wave of 2 KHZ and 5Vp-p. Connect the function generator using probes to the oscilloscope. Press Auto Scale.
- Press channel (1) button, make sure from the menu that coupling is DC, bandwidth limit is off and probe is set at (1X).
- Play with the horizontal and vertical position and scale knobs and try to understand their effect. Finally press auto-scale again.
- Note down the new values of Channel 1 voltage axis and Time base axis. Interpret the graph displayed using these values.
- Change the offset from -1V to 2V with a difference of 0.5V. Observe the change in waveform. Change the vertical scale i.e., the whole waveform is again at the center of screen. Note down the new values of voltage and Time of Channel 1.
- Press “measure” button, press “Voltage” and then press soft keys to determine following values , , , . Similarly press “Time” and determine the values of frequency, time period etc. Fill the table 2.
- Change the coupling to AC. What do you observe?

Task 2

- Generate a square wave pulse between 0-5V. Let the frequency be 4KHZ. Observe the time period of a wave.
- Change the duty cycle to 25%.
- Observe the on time +5V and off time 0V. Find the ratio of On-time and the time period of the square wave.
- To measure the time (off or on) note down the time base status. It represents how much time one division (box) on the horizontal axis represents. Using this information calculate the time i.e.
- divisions for which the wave is +5V and for which it is 0volts.
- Change the duty cycle to 70% and repeat the experiment.

Task 3

- Generate a Sine wave of 1 KHz (note down its value in radians/sec using w = 2πf), 10Vp-p.
- Press “Ref” and press “save” to save this waveform as reference.
- Now change the horizontal position knob, the reference wave remains static but the live voltage waveform will change position. On lower left corner of the screen the time delay will be displayed.
- Delay the wave using horizontal position knob until the live waveform is at 180 degrees out of phase with the reference (π radians). 180 degrees means the wave becomes exactly the inverted version of the reference waveform. Note the numerical value of the time delay.
- Change the frequency to 3 KHz and 3.5 KHz and repeat the experiment. Fill the table 3.

Observation Tables

Task 1

Table 1 (a)

CH-1 Voltage status	Time Base Status	CH-1 status (After offset)	Time Base Status (After offset)

Table 1 (b)

Voltage Measurements	
Time Measurements	

Task 2

Table 2

Time Period	Duty Cycle	Time Base Status	On-Time	Off-Time	Ratio

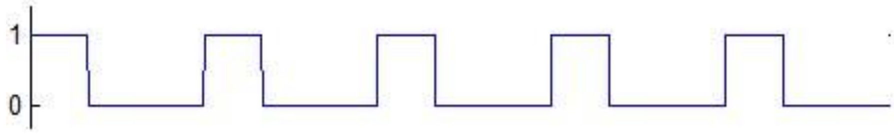
Task 3

Table 3

Frequency ( )	Frequency (/)	Time delay ( )	

Post Lab Task

1. Generate the voltage waveform shown below on oscilloscope by using function generator. The on time of the wave is 50% of the off time and only five complete cycles should be visible on the screen. The voltage should vary from 0 to 1V. (Choose frequency of your own choice, show the output to the instructor)



2. Observe the relationship of frequency and time.
3. Observe the relationship between in phase and out of phase waveforms.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ______________________________ Date: ____________________

LAB # 2: To explain natural and step response of parallel RLC circuits using software tool

Objectives

- To explain the basic understanding of LTspice software
- To explain the DC bias point and Transient Analysis using LTspice.
- To explain the Natural and Step response of RLC circuits using LTspice.
- To explain the concepts of overshoot, settling time and rise time using LTspice.

Pre-Lab

LTSpice

SPICE (Simulation Program with Integrated Circuit Emphasis) is a general-purpose electronic circuit simulator used to predict circuit behaviour. LTSPICE is a SPICE simulator with graphical interface (schematic capture) and waveform viewer.

Downloading and Installing LTSPICE:

LT Spice can be downloaded from http://www.linear.com/designtools/software/ltspice.jsp.The downloaded file is .exe file which directly installs LT Spice. Creating a simple circuit:

1. Open the LT Spice software.
2. Choose File -> New Schematic.
3. From Tools menu the color preferences can be changed, the grid can be turned on or off from the view menu.
4. The toolbar is explained below

Figure 2- 1:LTspice Toolbar.

5. The component button can be used to put any circuit component on the schematic diagram. The wire button can be used to connect different components.
6. The label button can be used to give labels to different nodes. Otherwise, a default name is given to each node.
7. To delete a component from the diagram either use F5 or click the scissors button and click on the component to be deleted.
8. To make a simple circuit as shown below click on the component button.

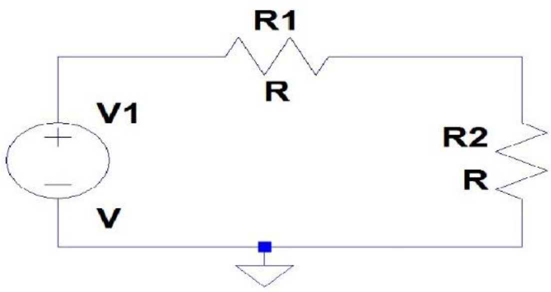


Figure 2- 2:A simple Resistive Circuit.

9. The following window appears.

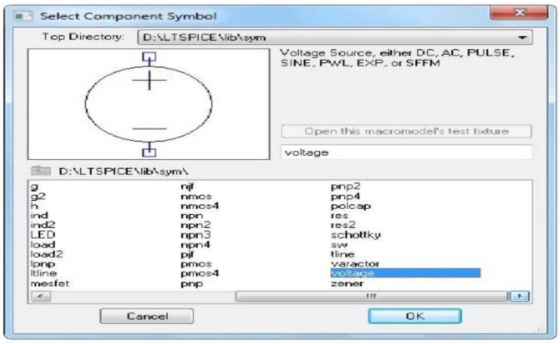


Figure 2- 3:Component Selection Window.

10. This window contains a collection of basic components to draw the circuit as shown above. Choose the voltage source and place it on the schematic diagram.
11. Place resistors on the schematic diagram and join those using wires to make the complete circuit. To rotate a resistor so that it can be placed as in the given circuit, select the resistor and press ctrl+r. Similarly ctrl+e are used to mirror a resistor. Place the ground at the lower node.
12. The circuit is complete. To set the values of components, observe that each component has two labels attached to it. One represents the name and other represents the value of the component. To change the name or the value of any component left click on the corresponding label e.g. each resistor comes with a label R1, R2 etc that represents its name. Each resistor is also accompanied by a label R that represents its value. To change the value of the resistor use left click on the label. The following window appears.

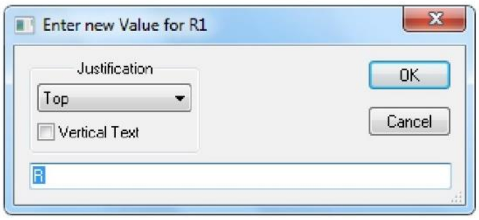


Figure 2- 4:Component Values Window.

13. Enter the value in the text field and click ok.
14. Another way setting different properties of a component is by using left click on the component itself e.g., if we use left click on the voltage source the following window appears.

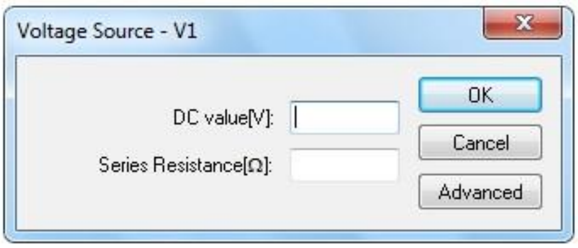


Figure 2- 4:Source Value Window.

15. Now the DC value and the source internal resistance can be set from this window. The advanced button can be used to change the voltage source from DC to other types which shall be explored in other tasks.

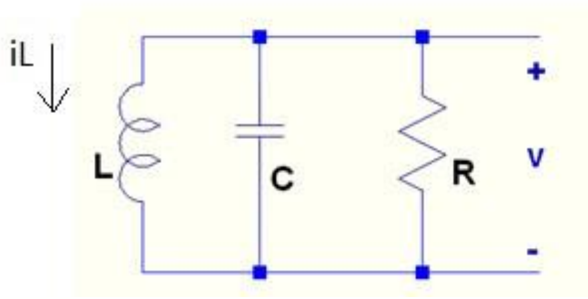


Figure 2- 5:Parallel RLC Circuit.

The parallel RLC circuit shown is described by the following differential equation:

$$\frac{d^2v}{dt^2} + \frac{1}{RC}\frac{dv}{dt} + \frac{v}{LC} = 0$$

The characteristic equation thus obtained by using $s = d/dt$ is

$$s^2 + \frac{s}{RC} + \frac{1}{LC} = 0$$

The roots of the characteristics equations are:

$s_{1,2} = -\frac{1}{2RC} \pm \sqrt{\left(\frac{1}{2RC}\right)^2 - \frac{1}{LC}}$ or $s_{1,2} = -\alpha \pm \sqrt{(\alpha)^2 - (\omega_\circ)^2}$

Where, $\alpha = \frac{1}{2RC}$, $\omega_o = \frac{1}{\sqrt{LC}}$ and $\omega_d = \sqrt{{\omega_o}^2 - \alpha^2}$

$s_1 = -\alpha + \sqrt{\alpha^2 - {\omega_o}^2}$ and $s_2 = -\alpha - \sqrt{\alpha^2 - {\omega_o}^2}$

Condition	Response	General Formula
If ${\omega_o}^2 < \alpha^2$	Over damped	$A_1 e^{s_1 t} + A_2 e^{s_2 t}$
If ${\omega_o}^2 > \alpha^2$	Under damped	$B_1 e^{-\alpha t} \cos \omega_d t + B_2 e^{-\alpha t} \sin \omega_d t$
If ${\omega_o}^2 = \alpha^2$	Critically damped	$D_1 t e^{-\alpha t} + D_2 e^{-\alpha t}$
If $\alpha = 0, \omega_\circ = \frac{1}{\sqrt{LC}}$	Undamped/ Oscillatory	$A \cos(\omega t + \varphi)$

The response is overdamped when the roots of the circuit's characteristic equation are unequal and real, critically damped when the roots are equal and real, underdamped when the roots are complex and undamped or oscillatory when the circuit is resistance free and LC circuit behaves as an oscillator.

Pre-Lab Task

Solve the parallel RLC circuit for the following conditions.

Bring the solution and final result with you in the lab.

1. Let $R = 2k\Omega$, $L = 250mH$ and $C = 10nF$.The initial current in the inductor at $t = 0$ is $-4A$. The initial voltage across the capacitor is zero. Find the expression of$v(t)$for $t > 0$.
2. Let $R = 62.5\Omega$, $L = 10mH$and $C = 1uF$. The initial current in the inductor at $t = 0$ is $80mA$. The initial voltage across the capacitor is $10V$.Find the expression of $v(t)$for $t > 0$.

In Lab Tasks

Task 1: Simulating Bias Point

1. Create the simple circuit as described in the introduction section.
2. Set the DC voltage source equal to 5V and both resistors are set equal to 1K. (The symbol for prefixes such as “kilo” “milli” and “mega” are case insensitive e.g. the symbol for kilo is K or k, for milli it is M or m and for mega it is “MEG” or “meg”.
3. Now click Simulate->run from the top menu or click the run button on the toolbar. The following window appears.

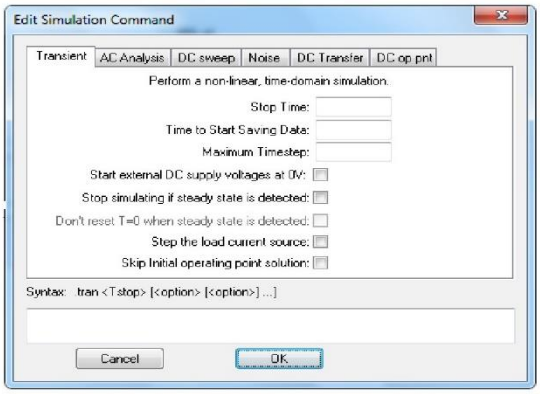


Figure 2- 6:Simulation command window.

It shows the possible type of analyses LT Spice can perform. At the moment we are only interested in the DC bias point so click the DC op point button on the top menu of this window and click ok.

4. The operating point is calculated and the following results appear.

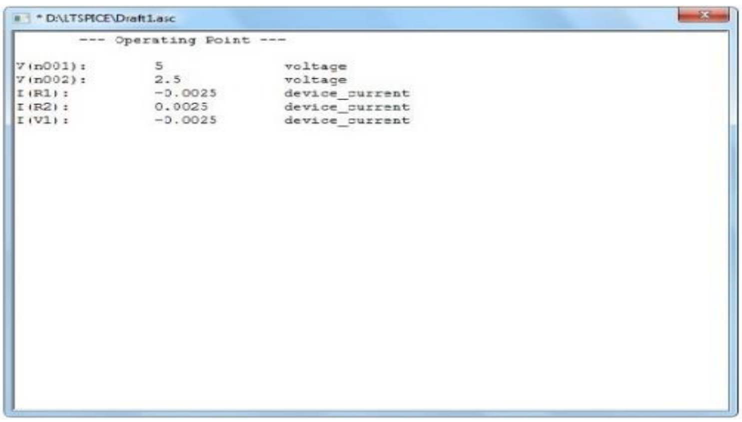


Figure 2- 7:Bias point result window.

5. Since we placed no label on the nodes so they are given names n001 and n002. The node with ground connected is named 0.
6. Now we place our own labels on the nodes by using the label net button on the toolbar and run the simulation again

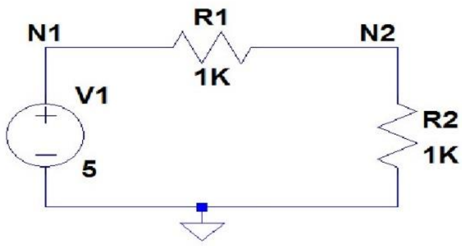
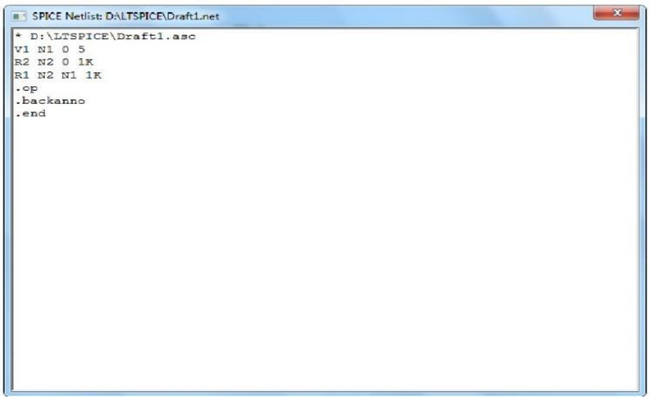


Figure 2- 8:Simple circuit with nodes labelled.

7. So the node voltages and current through each component are listed. Note that the current through the resistor is negative. The reason is that it was rotated before being placed in the circuit. LTspice defines a predetermined direction of current through each resistor. A negative value shows that the actual direction of current is opposite to the predetermined assumed direction. To check what direction LTspice has assumed click View-> Spice Net-list from the top menu. A net-list is a text version of the schematic diagram. The following window appears


SPICE Netlist: D:\LTSPICE\Draft1.net

```
* D:\LTSPICE\Draft1.asc
V1 N1 0 5
R2 N2 0 1K
R1 N2 N1 1K
.op
.backanno
.end
```

Figure 2- 9:Net-list window.

8. It shows that R1 is connected between nodes N2 and N1 and hence the assumed direction of current is from N2 to N1. Whereas the actual current flows from N1 to N2 and hence the output generated a negative sign.
9. To connect R1 in the assumed direction e.g. from N1 to N2 select the resistor by using the move or drag button (the buttons with the symbol of open or closed hand) from the toolbar and press ctrl + e to mirror the resistor. Now run the simulation and view the Spice Net-list.
10. The current through Voltage source is negative as it should be by passive sign convention. Fill the table given below. Show your results to the instructor.

Task 2: Transient Analysis

1. Create the parallel RLC circuit as shown in the introduction section.
2. Assign the values of $R = 200\Omega$, $L = 50mH$ and $C = 0.2uF$.   .

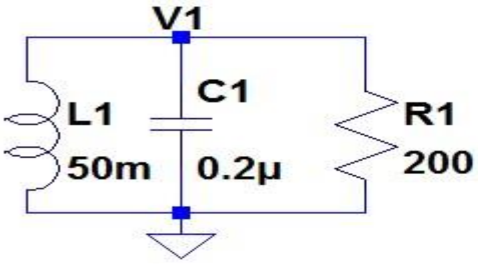


Figure 2- 10:Parallel RLC circuit with node labelled.

3. To assign initial values of currents and voltages in capacitor and inductor, press the .op button on the toll bar (circled in the figure below). The following window appears

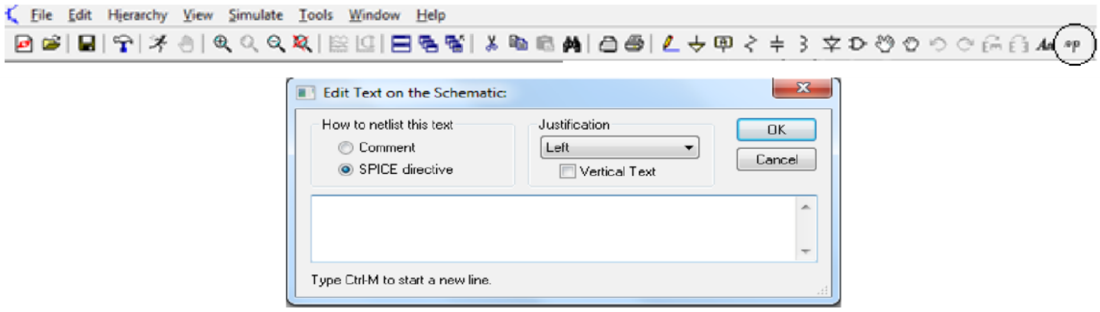


Figure 2- 11:Spice directive window.

4. Write the following statement".$IC\ I(L1) = 30m\ V(V1) = 12$". This sets the initial current of inductor as 30mA and initial capacitor voltage as 12 $volts$.
5. Check net-list to see if you have connected the resistor, capacitor and inductor in the correct direction. Now click Simulate->Edit Simulation cmd. The simulation command window will appear.
6. Choose Transient and set Stop Time equal to 300 $usec$.
7. Run the simulation. A graphical black window would appear. Maximize this window.

Graphical Analysis (TRACE)

1. Since the results of the DC Sweep are best viewed using a graphical utility so shall use the graphical analysis of LTSPICE also called TRACE. Take the mouse cursor over the horizontal axis, the mouse cursor changes into a scale icon. Use the 'right-click' button and a window would appear.

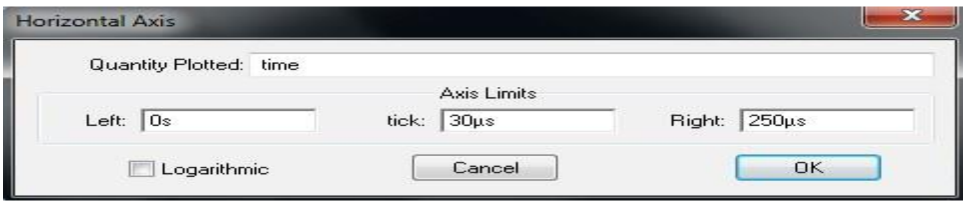


Figure 2- 12:Horizontal axis control window.

2. This window tells us that the quantity plotted on the horizontal-axis is time. It also tells what the maximum and minimum value on the axis is and where the ticks are placed.

3. Now move the mouse cursor somewhere on the graphical screen and use 'left-click', from the dropdown menu that appears click 'Add Trace'. The following window appears

Figure 2- 13:Add trace window.

4. It lists all the voltages and current which have been calculated and can be plotted. Choose$V(V1)$.
5. A number of mathematical operations can be performed on the graphs. A constant may be added, subtracted, multiplied or divided from the graph. Two or more graphs may be added, subtracted, multiplied or divided. Similarly, the logarithm or some trigonometric function of the graph may be plotted as well. To apply a mathematical operation on the graph use left click on the title of the graph ($(V1)$ in this case). The following window appears

Figure 2- 14:Waveform expression editor.

6. In this window any algebraic expression may be written.
7. By using right click on the graph, the numerical values at different points can be observed.
8. Observe the waveform. What type of response is it? The time after which the waveform becomes in between +1% and -1% of the final value (zero volts in case of natural response). This time is called settling time of the response. Increase the simulation time and note down the settling time.
9. Change the value of resistor to 312.5. Again run the simulation. Observe the response and note down the settling time.
10. Change the value of resistor to 200 and repeat the experiment. Save the waveforms for lab report. To save a graph click Tools->Copy bitmap to clipboard and paste the graph in paint or in word file.
11. Fill up all the tables given below. Show the graphs to the instructor. Save graphs for lab report.

Task 3: Step Response

1. Create the following circuit on LT Spice. Do not set any initial conditions.

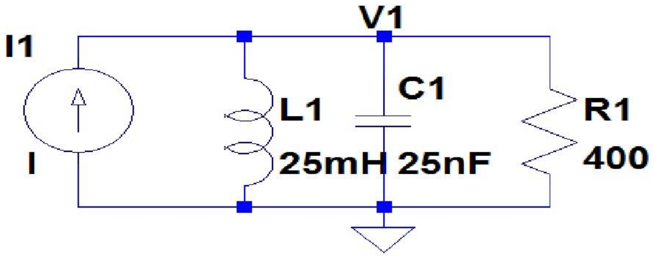


Figure 2- 15:Parallel RLC circuit with step source.

2. To set the current source as a step source of 24mA, take the cursor on the current source and left click. Click "Advanced", the following window appears.

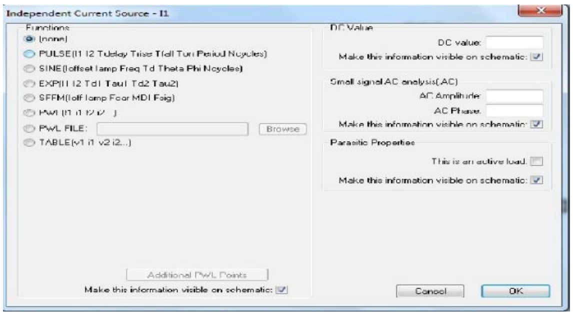


Figure 2- 16:Advanced controls of a source .

3. Click PWL radio button. This button is used to generate an arbitrary waveform by specifying different points of the waveform. The remaining waveform is calculated by linear interpolation. Specify the following points to generate a step source of 24mA.

Table 2 : Values for a step current source

Time1	0
Value1	0
Time2	1nsec
Value2	24mA
Time3	1msec
Value3	24mA

4. Simulate the circuit for $200usec$ with the step of $100usec$; plot the inductor current. Observe the type of response and measure the values of overshoot, rise time and settling time. The definitions are given below
   i. **Over-Shoot**: The maximum value of waveform greater than the steady state value (steady state value is $24mA$in this case).
   ii. **Rise Time**: The time in which the waveform reaches 90% of the steady state value.
   iii. **Settling Time**: The time after which the waveform becomes in between $+1\%\ and\ -1\%$ of the steady state value. (Note: Some definitions might define it as in between 5% or some similar value).
5. Change the value of resistor to 625 and 500. Repeat the experiment.

6. Fill up all the tables given below. Show the graphs to the instructor. Save graphs for lab report.

Table 3

R	${w_o}^2$	α^2	Type of response	Overshoot	Rise Time	Settling Time

Post lab Task

Create the following circuit.

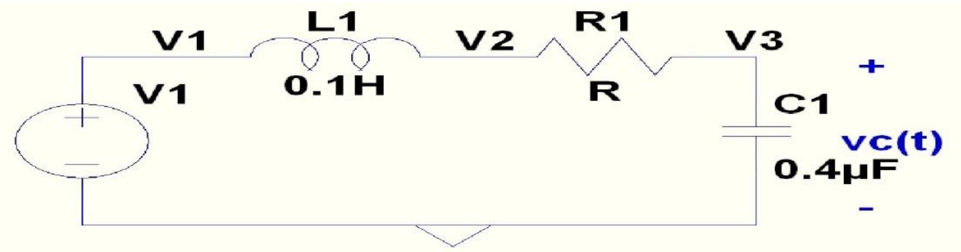


Figure 2- 17:A series RLC circuit.

Make sure that components are connected in right direction using Net-list. Voltage source V_1 is a step source of 48V.

1. Design the value of R i.e. the rise time of the voltage $vc(t)$ is less than$400usec$. Determine the type of response?
2. Now re-design the value of R i.e. there is no over-shoot in the waveform now. Determine the type of response and the value of rise time.
3. Attach your results with lab report.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ________________________________ Date: _____________________

LAB # 3: To Display the Sinusoidal Steady State Response of RC and RL circuits using hardware & software tools

Objectives

- To Display the behavior of capacitor and inductor under time varying signal.
- To Explain the experimental verification of capacitor and inductor's impedance expression.

Pre-Lab

Introduction

Phasor analysis is a useful tool to analyze the sinusoidal steady state behavior of circuits containing capacitors and inductors. In phasor analysis the information about the frequency of the applied signal is suppressed and only magnitude and phase are analyzed. This makes sense since linear circuits (i.e. circuits containing resistors(R), capacitors(C) and inductors (L)) can only affect the magnitude and phase of the input signal and the frequency of the sinusoid remains the same throughout the circuit. Using phasor technique we transform the time domain V-I relations into phasor domain relations in the complex domain. This converts all the differential equations that describe the circuit in time domain to phasor equivalent linear equations in complex frequency domain. In phasor domain all passive elements (R, L and C) are converted into respective impedances Z. All impedances in phasor domain follow the simple relation.

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

Where $\tilde{V}$ and $\tilde{I}$ are complex values of phasor voltage and current. The impedance Z is a complex number defined as the ratio between the voltage and current in phasor domain. Since all elements obey this form of Ohm `s law in phasor domain hence all the techniques of resistive circuit analysis are valid for $R, LandC$ circuits in phasor domain. The impedance of resistor is purely real whereas impedance of capacitors and inductors are purely imaginary. The equivalent impedance of a network of RLC components is a complex number, the real part of such impedance is called "Resistance" and imaginary part is called "Reactance".

$$Z = R + jX$$

Table 1

Component	Impedance (Z)	Resistance (R)	Reactance (X)
R	R	R	0
C	$-j/wC$	0	$-1/wC$
L	jwL	0	wL

Where 'w' is the frequency of the sinusoidal signal in rad/sec.

Capacitor

As shown in the above table the impedance of capacitor is given as

$$Z_c = \frac{\tilde{V}}{\tilde{I}} = -j/wC$$

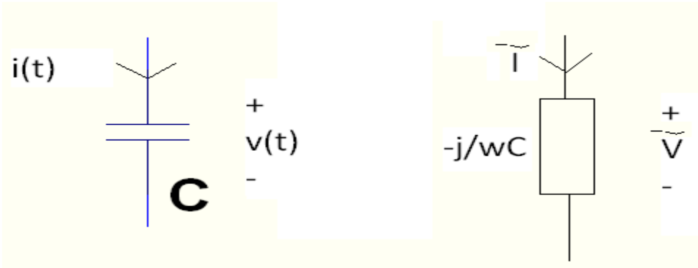


Figure 3- 1:Capacitor and its Impedance.

There are two important inferences that can be drawn from the impedance relationship.

1. The impedance is inversely proportional to the applied frequency. Hence capacitor behaves as short circuit for high frequencies and it behaves as open for low frequencies (i.e. DC signal which has zero frequency).
2. The '-j' in the relationship describes that phasor voltage and current of a Capacitor are at a phase angle of -90 degrees from each other.

Inductor

As shown in the above table the impedance of inductor is given as

$$Z_L = \frac{\tilde{V}}{\tilde{I}} = jwL$$

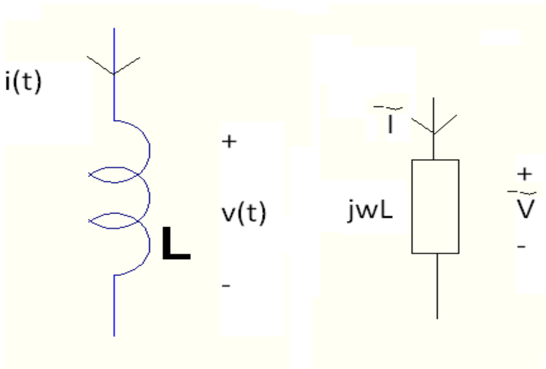


Figure 3- 2:Inductor and its Impedance.

There are two important inferences that can be drawn from the impedance relationship.

1. The impedance is directly proportional to the applied frequency. Hence inductor behaves as open circuit for high frequencies and it behaves as short for low frequencies (i.e. DC signal which has zero frequency).
2. The 'j' in the relationship describes that phasor voltage and current of an Inductor are at a phase angle of 90 degrees from each other.

Pre-Lab Exercise

In this assignment student will learn how to find the sinusoidal steady state response of simple RL and RC circuits on LT spice.

1. To generate a sinusoidal source, first make the following circuit on LT spice.

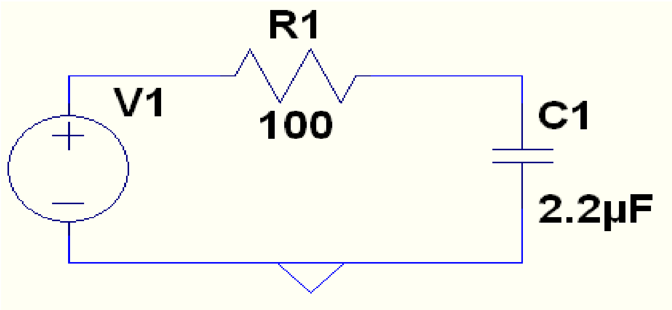
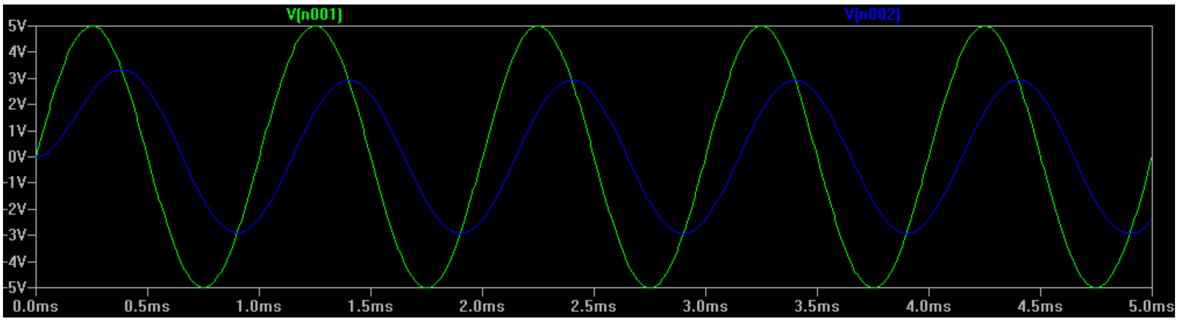


Figure 3- 3:A simple RC circuit..

2. Left-Click the voltage source, go to advance menu and select 'SINE' radio button.
3. Specify the value of offset as zero, Amplitude as 5 and Frequency as $1KHZ$. Leave the rest blank.
4. Now run transient analysis for 5 $msec$. Plot input source voltage and voltage across capacitor on the same screen.



5. To determine the phase difference between the two waveforms. Note down the time difference between the peaks of the two waves. For example let's say any one peak on the green wave (the input signal) is at $2.25msec$ $(t1)$. For the same cycle, the peak on the blue wave (voltage across capacitor) is at$2.4msec$ $(t2)$. Now the phase difference is given as

$$\Delta\theta = \theta_2 - \theta_1 = w(t1 - t2) = w\Delta t$$

Where w is the frequency is rad/sec and θ is in radians. The same relation may be written as

$$\Delta\theta^o = {\theta_2}^0 - {\theta_1}^0 = 360f(t1 - t2) = 360f\Delta t$$

Where phase is in degrees and the frequency$f$in H_Z.

Task 1

A phasor quantity (Voltage or Current) is a complex value which consists of two parts; Magnitude and Phase. The magnitude is equal to the amplitude of the sinusoidal wave and phase is equal to the phase shift of the sinusoidal wave from the origin/reference point. In the above section it is discussed how to calculate the phase difference between two waveforms. Since we had set the phase shift of the input voltage source equal to zero in the previous section (we left the value blank which by default means a zero value) hence the phase difference is in fact equal to phase shift of the capacitor voltage waveform. Hence we can calculate the phasor magnitude and phase from the waveform generated.

1. Make the circuit shown in figure 3, using LT SPICE.
2. Simulate the circuit for 10 different frequencies of your own choice in the range $0 - 10KHz$.
3. For each simulation, determine the phasor magnitude and phase of the capacitor voltage and current (i.e. $|Vc|, |Ic|, \theta_{vc}, \theta_{ic}$).
4. Fill the table for 10 different frequency values; plot all the values of the table as a function on frequency. Bring the table and the plots with you in lab session and get them signed from the instructor.

Table 2

Frequency(KHz)	$\|Vc\|$	$\|Ic\|$	θ_{vc}	θ_{ic}	$\theta_{vc} - \theta_{ic}$

Task 2

Construct the circuit of Fig. 4 in LT Spice and repeat task 1. Record the data into the given table and keep the table and plots with you in lab.

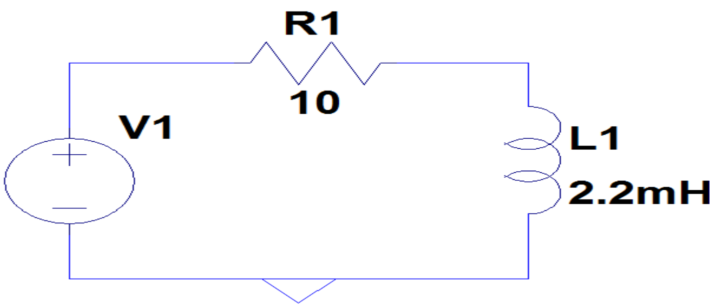


Figure 3- 4:simple RL circuit.

Record the data into the given table and keep the table and plots with you in lab.

Table 3

Frequency(KHz)	$\|VL\|$	$\|IL\|$	θ_{vL}	θ_{iL}	$\theta_{vL} - \theta_{iL}$

In Lab Task

Task 1: Response of C and L for DC and High Frequency

1. Create the simple RC circuit as shown in Fig.3 on a LT Spice.
2. Apply $5V$ DC as V_1 using a DC power supply.
3. Measure the voltage and current through the capacitor.
4. Now apply a sinusoidal signal of very high frequency e.g. $10KHz$ or more as V_1.
5. Examine the voltage and current waveform of a capacitor.
6. Now create the simple RL circuit as shown in Fig.4. Repeat the above experiment for it.

Table 4

Frequency	Capacitor Voltage	Capacitor Current	Inductor Voltage	Inductor Current

Task 2: Phasor Impedance of a Capacitor

1. Create the simple RC circuit as shown in Fig No.3.
2. Apply a sinusoidal signal of 5V peak (10 V_{p-p}) of $1KHz$ frequency as V_1.

Voltage Measurements

3. Examine the magnitude (i.e. peak) of the output voltage across the capacitor.
4. Compare it with the theoretical value. Theoretically, the phasor of the output voltage is:

$$\tilde{V_c} = \frac{-\frac{j}{wC}}{(R - \frac{j}{wC})}\widetilde{V1}$$

The phasor current is

$$\tilde{I_C} = \frac{\tilde{V_c}}{-j/wC}$$

5. From the above equation the theoretical values of the magnitude and phase of the output voltage can be calculated.
6. Repeat the entire experiment for $200Hz$, $500Hz$, $1.5\ KHz$and $2\ KHz$.

Phase Measurements

7. To determine the phase value of the output voltage. We consider the input wave as reference and hence its phase is zero. To calculate the phase difference we need to calculate the time difference between the two waves. Then phase is calculated using the formula
$$\Delta\theta^o = \theta_2{}^0 - \theta_1{}^0 = 360f(t1 - t2) = 360f\Delta t$$
8. If the output wave is ahead in time than the reference, we call it leading. In such a case Δt is negative and so is the phase. If the output wave is behind in time than the reference, we call it lagging. In such a case Δt is positive and so is the phase.
9. Find the phase for $200Hz$, $500Hz$, $1.5\ KHz$and $2\ KHz$.

Current Measurements (magnitude and Phase)

10. To measure the phasor of capacitor current, re-connect the circuit as follows:

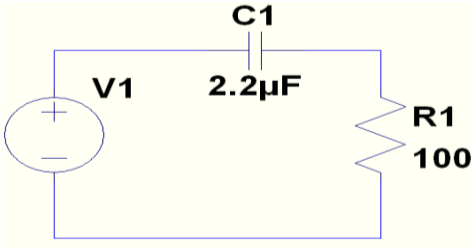


Figure 3- 5:A simple RC circuit.

11. Now connect the input source and measure the output voltage across the resistor. Capacitor current is equal to the voltage across resistor divided by R_1.
12. Determine the magnitude and phase of the current for $200Hz, 500Hz, 1.5\ KHz$and $2\ kHz$.

Table 5

Freq (Hz)	$\lvert\widetilde{V_c}\rvert_{theory}$	$\lvert\widetilde{V_c}\rvert_{exp}$	$\lvert\widetilde{I_c}\rvert_{theory}$	$\lvert\widetilde{I_c}\rvert_{exp}$
200				
500				
1.5K				
2K				

Post lab Task

Phasor Impedance of an Inductor

1. Create the simple RL circuit as shown in Fig.4 on a LT Spice.
2. Measure the phasor voltage and current across the inductor using the same procedure as that was in task 1 for frequencies $100Hz, 200HZ, 500Hz, 1\ KHz and\ 2\ KHz$.
3. Compare it with the theoretical value. Theoretically the phasor of the output voltage will be calculated as

$$\widetilde{V_L} = \frac{jwL}{(R + jwL)}\widetilde{V1}$$

4. And phasor current as

$$\widetilde{I_L} = \frac{\widetilde{V_L}}{jwL}$$

Table 6

Freq (Hz)	$\lvert\widetilde{V_c}\rvert_{theory}$	$\lvert\widetilde{V_c}\rvert_{exp}$	$\lvert\widetilde{I_c}\rvert_{theory}$	$\lvert\widetilde{I_c}\rvert_{exp}$
100				
200				
500				
1K				
2K				

Express your ideas

1. In case of resistor, Is there any phase difference between voltage and current.

2. In an Inductor, voltage waveform leads the current waveform. Elaborate this point on the basic understanding of this lab.

3. Let the phase difference between two cosine waves of frequency 50Hz is 30 degrees. What would be the corresponding time delay?

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ________________________________ Date: _____________________

LAB # 4: To Reproduce the circuits in Ltspice and Show the Network theorems in phasor domain using software tool

Objectives

- To Show the network theorems of electric circuits in Phasor domain using Ltspice

Pre-Lab

We cannot do circuit analysis in the frequency domain without Kirchhoff's current and voltage laws. Therefore, we need to express them in the frequency domain.

For KVL, let v_1, v_2, v_3 be the voltages around a closed loop.

$$v_1 + v_2 + v_3 = 0$$

By following a similar procedure, we can show that Kirchhoff's current law holds for phasors. If we let i_1, i_2, i_3 be the current leaving or entering a closed surface in a network at time t, then

$$i_1 + i_2 + i_3 = 0$$

Pre-Lab Exercise

1. Determine $V_o(t)$in the circuit of Figure.1

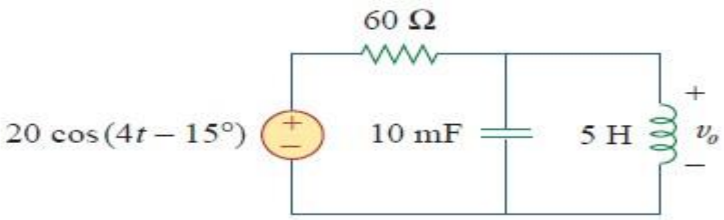


Figure 4- 1: Prelab circuit.

In Lab Task

Lab Task 1: Determine the magnitude and phase of the Nodes

1. Create the circuit as shown in Fig.2.

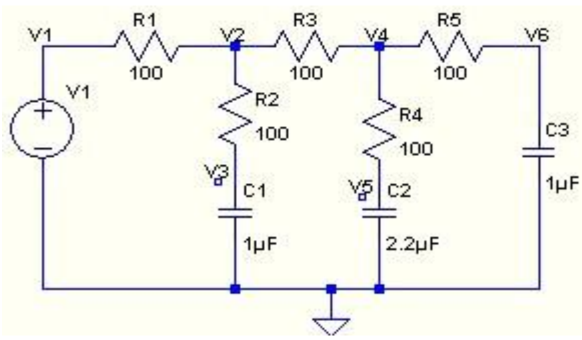


Figure 4- 2:Circuit diagram for the lab experiment.

2. Determine the magnitude and phase (i.e. Phasor Value) of all the node voltages from V_1 to V_6
3. Apply v_{in}=6<0°

Fill in the table and blanks in the measurement section. Verify KCL and KVL using your results for each node and each loop.

Lab Task 2: Verification of Thevenin Theorem

1. For the circuit in Fig. 2. Assume C_3 to be the load capacitor.
2. Disconnect the capacitor form the circuit and determine the magnitude and phase (i.e. Phasor Value) of the open circuit voltage (i.e. V_4).
3. Now replace C_3 with a short circuit and determine the Phasor value of the short circuit current i.e. the current through resistorR_5. In order to determine the current, determine the phasor voltage $V_4$and divide it by the value of R_5 .
4. Compute Thevenin equivalent impedance by the help of given formula
$$Z_{th} = \frac{V_{\text{open circuit}}}{I_{\text{Short Circuit}}}$$
5. Determine the output voltage across the output capacitor C_3 using the Thevenin equivalent.

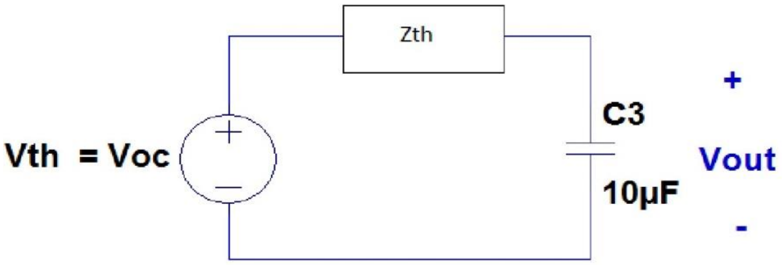


Figure 4- 3:Thevenin Equivalent Circuit.

6. Compare it with the actual voltage across C_3 measured in the previous task (i.e. V6).
7. If the results are the same Thevenin theorem is verified.

Measurement Tables

Task -1

(Fill the table with Phasor values (magnitude and phase) of the node voltages)

Table 1

V_1	V_2	V_3	V_4	V_5	V_6

$I_{R1} = (V_1 - V_2)/R_1$ = ____________________

$I_{R2} = (V_2 - V_3)/R_2$ = ____________________

$I_{R3} = (V_2 - V_4)/R_3$ = ____________________

$I_{R4} = (V_4 - V_5)/R_4$ = ____________________

$I_{R5} = (V_4 - V_6)/R_5$ = ____________________

$I_{c1} = I_{R2}$ = ____________________

$I_{c2} = I_{R4}$ = ____________________

$I_{c3} = I_{R5}$ = ____________________

$V_{R1} = (V_1 - V_2) \quad =$ ____

$V_{R2} = (V_2 - V_3) \quad =$ ____

$V_{R3} = (V_2 - V_4) \quad =$ ____

$V_{R5} = (V_4 - V_6) \quad =$ ____

$V_{c1} = (V_3) \quad =$ ____

$V_{c2} = (V_5) \quad =$ ____

$V_{c3} = (V_6) \quad =$ ____

Verification of KCL

Node V2:

The currents entering the node are: ____

The currents leaving the node are: ____

Sum of all currents entering: ____

Sum of all currents leaving: ____

Node V4:

The currents entering the node are: ____

The currents leaving the node are: ____

Sum of all currents entering: ____

Sum of all currents leaving: ____

NodeV6:

The currents entering the node are: ____

The currents leaving the node are: ____

Sum of all currents entering: ____

Sum of all currents leaving: ____

Verification of KVL

Loop 1: V1->V2->V3->V1

The voltage rise in the loop: ____

The voltage drops in the loop: ____

Sum of all voltage rises: ____

Sum of all voltage drops: ____

Loop 2: V2->V4->V5->V3->V2

The voltage rise in the loop: ____

The voltage drops in the loop: ____

Sum of all voltage rises: ____

Sum of all voltage drops: ____

Loop 3: V4->V6->V5->V4

The voltage rise in the loop: ____

The voltage drops in the loop: ____

Sum of all voltage rises: ____

Sum of all voltage drops: ____

Loop 4: V1->V2->V4->V5->V1

The voltage rise in the loop: ____

The voltage drops in the loop: ____

Sum of all voltage rises: ____

Sum of all voltage drops: ____

Loop 5: V2->V4->V6->V3->V2

The voltage rise in the loop: ____

The voltage drops in the loop: ____

Sum of all voltage rises: ____

Sum of all voltage drops: ____

Loop 6: V1->V2->V4->V6->V1

The voltage rise in the loop: ____

The voltage drops in the loop: ____

Sum of all voltage rises: ____

Sum of all voltage drops: ____

Task 2

$V_{oc} =$ ____________________

$I_{sc} =$ ____________________

$Z_{TH} =$ ____________________

$V_{out}\ (using\ Thevenin\ equivalent) =$

$V_{out}(as\ measured\ in\ Task\ 1) =$

Post Lab Task

1. Verify all the experimental results with analytical method and attach the result with the experiment.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ________________________________ Date: _____________________

LAB # 5: To Reproduce the circuit and show the Power (Active, Reactive and Apparent) and Power Factor Correction using software tool

Objectives

- To show the parameters for AC power analysis using LTspice
- To compute active, reactive and apparent power using standard circuit formulas
- To reproduce the circuit for computing power factor and its correction using LTspice

Pre-Lab

Power: Electrical Power is the rate of doing electrical work by transferring the electrical charge from one point to the other within the circuit.

$$w = \frac{dq}{dt}$$

Real Power or Active Power: The power dissipated in the resistor is the active power. It is also called real power: This power is unidirectional (source to load). It does not flow back to the circuit. In AC, the time averaging of instantaneous power is done over a complete cycle. It is usually denoted by “$\boldsymbol{P}$”. It is calculated as

$$P = VI\cos\theta$$

It is measured in Watts.

Reactive Power: The power dissipated in the inductance or capacitance is known as the reactive power.

This power is bidirectional (source to load and back from load to source). It moves back and forth. It is usually denoted by “Q” .It is calculated as

$$Q = VI\sin\theta$$

It is measured in VAR.

Complex Power:

The phasor sum of the active and reactive power is the complex power. Its magnitude is known as the apparent power. It is usually denoted by “S” .It is a phasor quantity, so we need to evaluate its magnitude and phase. It is calculated as

$$[S] = \sqrt{\{Re(s)\}^2 + \{Im(s)\}^2}, \theta = \tan^{-1}([{}^{Im(S)}/_{Re(S)}])$$

It is measured in VA.

The phasor diagram for the relation between the active and reactive power is shown below. The mathematical relationship between the active and reactive power is given by the following equation:

$$S = P + jQ$$

As we can see from the phasor relationship of P and Q, both are at 90^0 or at quadrature in reference to each other.

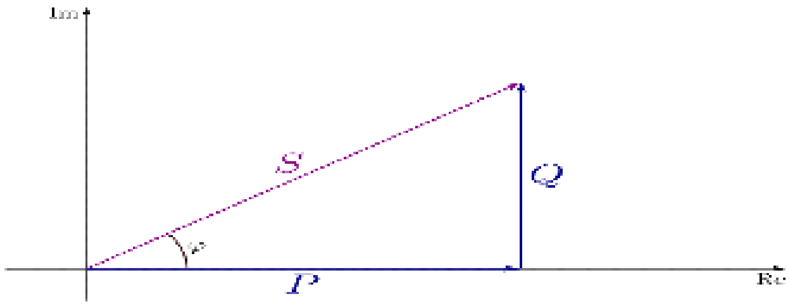


Figure 5- 1:Phasor diagram showing the relationship between P, Q and S.

Practical Power Factor Correction

When the need arises to correct for poor power factor in an AC power system, you probably won't have the luxury of knowing the load's exact inductance in henrys to use for your calculations. You may be fortunate enough to have an instrument called a power factor meter to tell you what the power factor is (a number between 0 and 1), and the apparent power (which can be figured out by taking a voltmeter reading in volts and multiplying by an ammeter reading in amps). In less favorable circumstances you may have to use an oscilloscope to compare voltage and current waveforms, measuring phase shift in degrees and calculating the power factor by the cosine of that phase shift.

Most likely, you will have access to a wattmeter for measuring true power, whose reading you can compare against a calculation of apparent power (from multiplying total voltage and total current measurements). From the values of true and apparent power, you can determine reactive power and power factor. Let's do an example problem to see how this works:

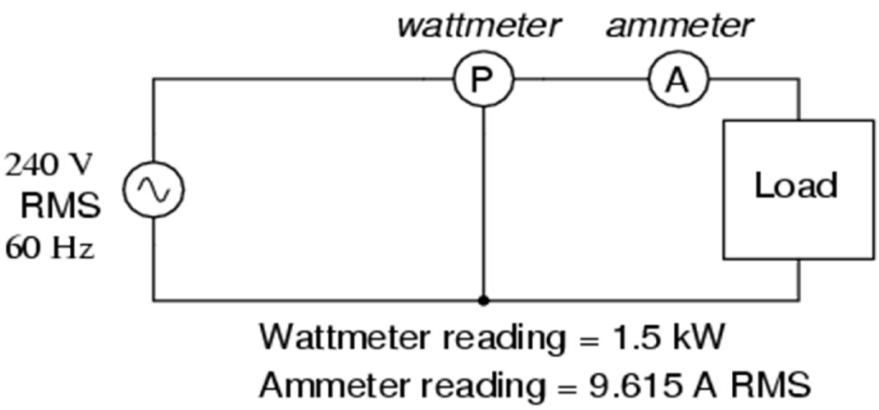


Figure 5- 2: Wattmeter readings.

Wattmeter reads true power, product of voltmeter and ammeter readings yield apparent power. First, we need to calculate the apparent power in KVA. We can do this by multiplying load voltage by load current:

$$S = IV$$

$$S = (9.615A).(240V) = 2.308kVA$$

As we can see, 2.308kVA is a much larger figure than1.5kW, which tells us that the power factor in this circuit is rather poor (substantially less than 1). Now, we figure the power factor of this load by dividing the true power by the apparent power:

$$\text{Power factor} = \frac{\text{P}}{\text{S}}$$

$$\text{Power factor} = \frac{1.5\text{kW}}{2.308\text{kVA}} = 0.65$$

Using this value of power factor, we can draw a power triangle and from that determine the reactive power of this load.

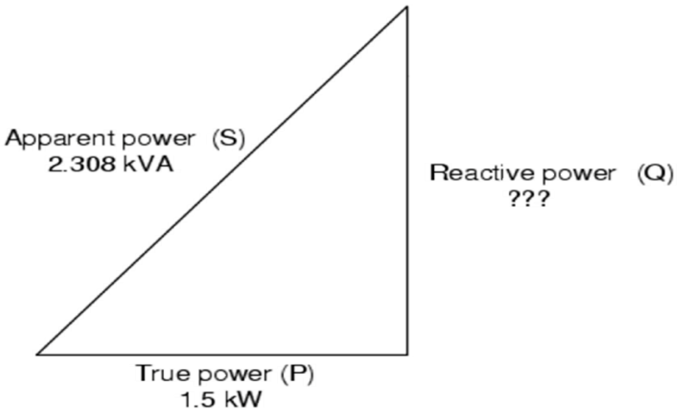


Figure 5- 3: Power triangle.

Reactive power may be calculated from true power and apparent power. To determine the unknown (reactive power) triangle quantity, we use the Pythagorean Theorem.

$$\text{Reactive power} = \sqrt{(\text{Apparent power})^2 - (\text{True power})^2}$$

$$Q = 1.754\ \text{kVAR}$$

Pre Lab Exercise

1. A load draws $12kVA$ at a power factor of 0.856 from 120 rms sinusoidal source.

Determine:

i. Active and reactive power to the load

ii. Peak current and load impedance.

2. When connected to a $120Vrms, 60Hz$ power line, a load absorbs 4kW at a lagging power factor of 0.8. Compute the value of capacitance necessary to raise the power factor to 1.

In Lab Task

1. First of all we will draw the circuit on LTspice according to the circuit diagram in Figure2.

Param	Vs	Rs	R	R1	L	C
Units	V	Ohm			mH	uF
Theor	10	50	150	5.1	100 - 300	0.1 - 3
Exper	-	-				

2. Measure the current (I).
3. Measure the voltage (V).
4. Determine the apparent power (S) by multiplying the voltage and current
5. Determine the Power Factor from the formula $\cos\theta = \frac{P}{S}$. If its value is not near to unity 1, we will then try to improve it.
6. We can correct this power factor by inserting a capacitor of appropriate value in parallel to inductive load by using the following formula:

$$C = \frac{P(\tan\theta_1 - \tan\theta_2)}{\omega\, {V_{rms}}^2}$$

Where P is the real power without C, θ_1 is initial angle, θ_2 is the final angle, ω is the angular frequency and V_{rms} is value of voltage.

7. We will then calculate the value of Q by using the Pythagorean Theorem. We can determine the value of Q by measuring the length of the side of the power triangle.

Table 1

Parameter	Unit	Without C	With C
$\|V\|$	V		
$\|I\|$	*mA*		
$\theta_v - \theta_i$	O		
Pf	-		
P	*W*		
Q	*VAR*		
S	*VA*		

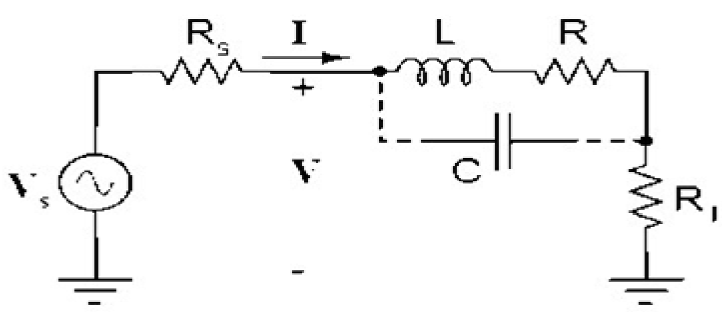


Figure 5- 4:Circuit Diagram for Power Calculations.

Post Lab Task

1. On the basic understanding of this lab, discuss the various methods of power factor improvement.
2. After doing the lab tasks, elaborate the effect of adding a capacitor in parallel to inductive load.
3. In RL series circuit, $V = 480V, R = 1\Omega, L = 1mH$. Determine the leading reactive power that is required to get unity power factor. Also compute the current power factor.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: _______________________________ Date: ____________________

LAB # 6: To reproduce three phase circuits using software tool

Objectives

- To reproduce the three phase circuits with wye and delta configuration using LTspice
- To display the output waveform for three phase circuits with wye and delta configuration using LTspice
- To show the line currents and compute the active power dissipated in the delta connected circuit using LTspice.

Pre-Lab

A poly phase system is basically an ac system composed of a certain number of single-phase ac systems having the same frequency and operating in sequence. Each phase of a poly phase system (i.e., the phase of each single-phase ac system) is displaced from the next by a certain angular interval. In any poly phase system, the value of the angular interval between each phase depends on the number of phases in the system. We will cover the most common type of poly phase system, the three-phase system. Three-phase systems, also referred to as three-phase circuits, are poly phase systems that have three phases, as their name implies. In the majority of cases, three phase circuits are symmetrical and have identical impedances in each of the circuit's three branches (phases). Each branch can be treated exactly as a single-phase circuit, because a balanced three-phase circuit is simply a combination of three single-phase circuits. Therefore, voltage, current, and power relationships for three-phase circuits can be determined using the same basic equations and methods developed for single-phase circuits. Non-symmetrical, or unbalanced, three-phase circuits represent a special condition and their analysis is more complex.

A three-phase ac circuit is powered by three voltage sine waves having the same frequency and magnitude and which are displaced from each other by 120°. The phase shift between each voltage waveform of a three-phase ac power source is therefore 120° (360°/3 phases). Figure 1 shows an example of a simplified three-phase generator (alternator) producing three-phase ac power.

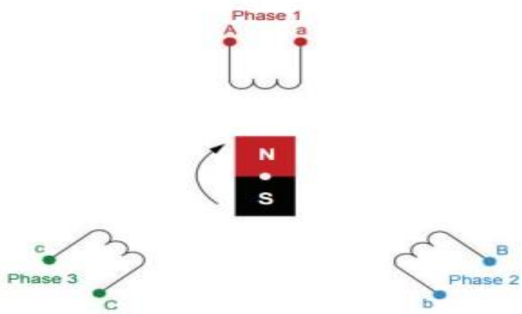


Figure 6- 1: A simple three phase generator.

The phase sequence of the voltage waveforms of a three-phase ac power source indicates the order in which they follow each other and attain the maximal voltage value. Figure 2 shows an example of the voltage waveforms produced in a three-phase ac power source, as well as the phasor diagram related to the voltage waveforms. The voltage waveforms and voltage phasors in Figure 2 follow the phase sequence $(\mathrm{E_A}, \mathrm{E_B}, \mathrm{E_C})$ which, when written in shorthand form, is the sequence A-B-C. This phase sequence is obtained when the magnet in the three-phase generator of Figure 1 rotates clockwise. The phase sequence of a three-phase ac power source is important because it determines the direction of rotation of any three-phase motor connected to the power source. If the phases are connected out of sequence, the motor will turn in the opposite direction, and the consequences could be serious. For example, if a three-phase motor rotating in the clockwise direction causes an elevator to go up, connecting the phase wires incorrectly to the motor would cause the elevator to go down when it is supposed to go up, and vice-versa, which could result in a serious accident.

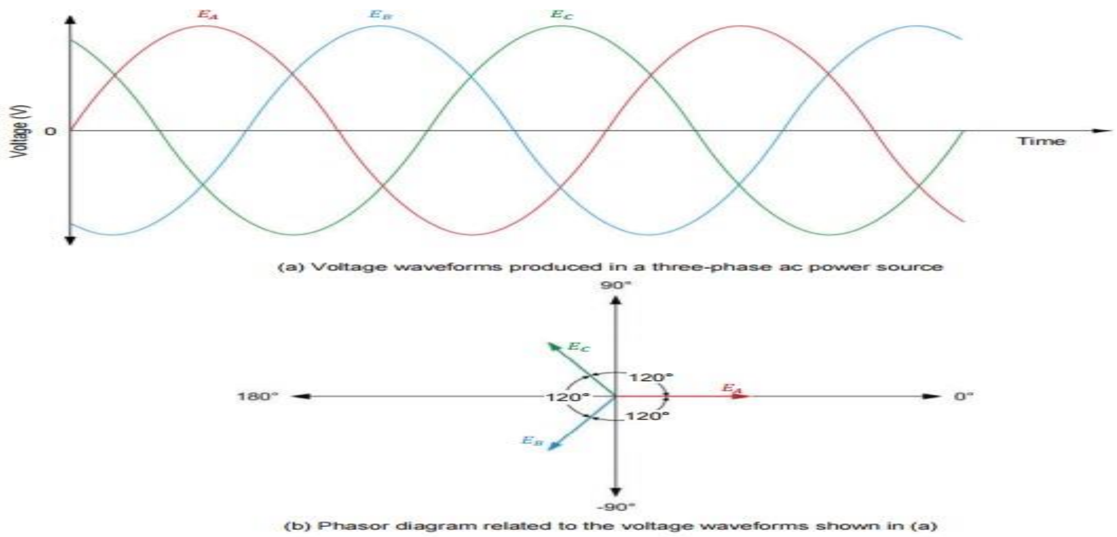
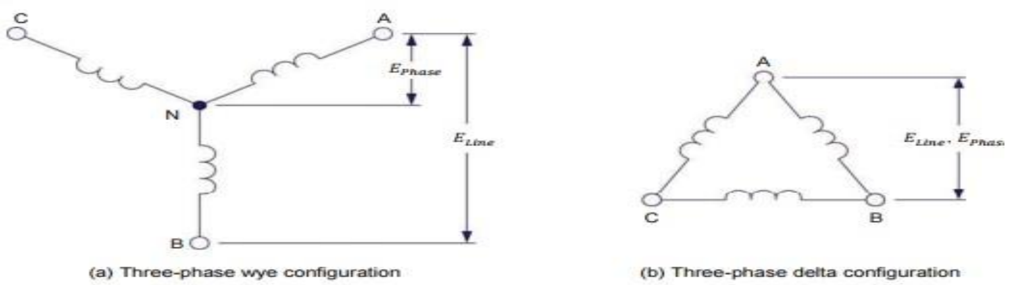


Figure 6- 2: A B C sequence of three phase AC source.

Wye and delta configurations

The windings of a three-phase ac power source (e.g., the generator in Figure 1) can be connected in either a wye configuration, or a delta configuration. The configuration names are derived from the appearance of the circuit drawings representing the configurations, i.e., the letter Y for the wye configuration and the Greek letter delta (ẩ) for the delta configuration. The connections for each configuration are shown in Figure 3. Each type of configuration has definite electrical characteristics. As Figure 3a shows, in a wye-connected circuit, one end of each of the three windings (or phases) of the three-phase ac power source is connected to a common point called the neutral. No current flows in the neutral because the currents flowing in the three windings (i.e., the phase currents) cancel out each other when the system is balanced.

Figure 3b shows that, in a delta-connected circuit, the three windings of the three-phase ac power source are connected one to another, forming a triangle. The three line wires are connected to the three junction points of the circuit (points A, B, and C in Figure 3b). There is no point to which a neutral wire can be connected in a three-phase delta-connected circuit. Thus, delta connected systems are typically three-wire systems.



Distinction between line and phase voltages, and line and phase currents

The voltage produced by a single winding of a three-phase circuit is called the line-to-neutral voltage, or simply the phase voltage ($\mathrm{E_{Phase}}$). In a wye - connected three-phase ac power source, the phase voltage is measured between the neutral line and any one of points A, B, and C as shown in Figure 3a. This result in the following three distinct phase voltages: $\mathrm{E_{A-N}, E_{B-N}, E_{C-N}}$

The voltage between any two windings of a three-phase circuit is called the line-to-line voltage, or simply the line voltage $\mathrm{E_{Line}}$. In a wye - connected three-phase ac power source, the line voltage is √3, (approximately 1.73) times greater than the phase voltage, $\mathrm{E_L} = \sqrt{3} \times \text{Phase voltage}$. In a delta connected three-phase ac power source, the voltage between any two windings is the same as the voltage across the third winding of the source (i.e. $\mathrm{E_{Line} = E_{Phase}}$), as shows Figure 3b. In both cases, this results in the following three distinct line voltages: $\mathrm{E_{A-B}, E_{B-C}, E_{C-A}}$

The three-line wires (wires connected to points A, B, and C) and the neutral wire of a three-phase power system are usually available for connection to the load, which can be connected in either a wye configuration or a delta configuration. The two types of circuit connections are illustrated in Figure 4. Circuit analysis demonstrates that the voltage (line voltage) between any two-line wires, or lines, in a wye connected load is √3 times greater than the voltage (phase voltage) across each load resistor. Furthermore, the line current flowing in each line of the power source is equal to the phase current flowing in each load resistor. On the other hand, in a delta-connected load, the voltage (phase voltage) across each load resistor is equal to the line voltage of the source. Also, the line current is √3 times greater than the current (phase current) in each load resistor. The phase current in a delta-connected load is therefore√3 times smaller than the line current.

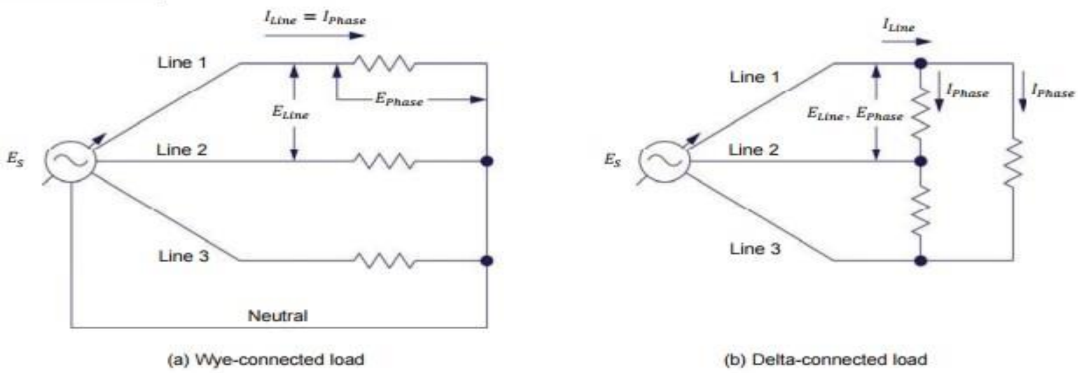


Figure 6- 3: Types of load connections.

The relationships between the line and phase voltages and the line and phase currents simplify the analysis of balanced three-phase circuits. A shorthand way of writing these relationships is given below. In Y-connection:

a) Line voltage (VL) = √3 × Phase voltage
b) Line voltages are 120° apart.
c) Line voltage are 30° ahead of respective voltages (for positive sequence)
d) Line current = Phase current

In a Delta connection:

a) Line voltage (VL) = Phase voltage
b) Line currents are 120° apart.
c) Line current are 30° behind of respective phase current (for positive sequence)
d) Line current = = √3 × Phase current

Pre-Lab Exercise

A balanced abc-sequence Y-connected source with V_{an}=100<10°Vis connected to a delta connected balanced load (8+ j4)Ω per phase. Calculate the phase and line currents.

In Lab Task

The Procedure is divided into the following sections:

- Setup and connections
- Phase and line voltage measurements in the Power Supply
- Voltage and current measurements in a wye - connected circuit
- Voltage and current measurements in a delta - connected circuit

Wye Connection

Task -1

You will set up the circuit to measure the line-to neutral (phase) and line-to-line (line) voltages of the three-phase ac power source.

1. Set up the circuit shown in Figure 5.

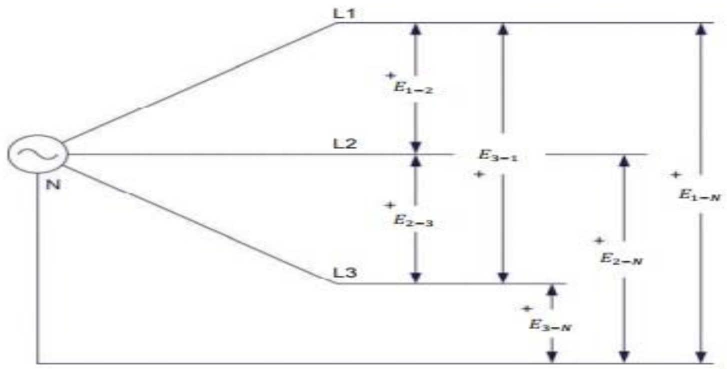


Figure 6- 4:Line and phase voltage measurements.

2. Measure and record below the phase voltages of the three-phase ac power source.

Table 1

E_{1-N}	E_{2-N}	E_{3-N}

3. Determine the average value of the phase voltages

$$\text{Average } E_{phase} = \frac{E_{1-N}+E_{2-N}+E_{3-N}}{3} = ------- V$$

4. Make the appropriate settings in order to observe the phase voltage waveforms related to inputs E1, E2, and E3.
   What is the phase shift between each voltage sine wave of the three-phase ac power source?

   Phase Shift: ………………………………………….

5. Draw the resulting waveforms and also draw the phasor diagram of these waveforms.

6. Modify the connections to the voltage inputs to measure the line voltages of the three-phase ac power source. Measure and record below the line voltages of the three-phase ac power source.

Table 2

E_{1-2}	E_{2-3}	E_{3-1}

7. Determine the average value of the line voltages.

$$\text{Average } E_{\text{Line}} = \frac{E_{1-2} + E_{2-3} + E_{3-1}}{3} = - - - - - - - \text{ V}$$

8. Calculate the ratio of the average line voltageE_{Line} to the average phase voltage.
9. Calculate the ratio of the average line voltage $E_{\text{Line}}$to the average phase voltage E_{phase}

$$\frac{Average\ E_{Line}}{Average\ E_{Phase}} = - - - - - - - -$$

10. Calculate the ratio of the average line voltage to the average phase voltage calculated in the previous step.

Ratio: ……………………………………………..

Task -2

Voltage and current measurement in a wye connected circuit

You will set up a wye - connected, three-phase circuit using three load resistors. You will measure the phase voltages and currents in the circuit, as well as the circuit line voltage and neutral line current. You will confirm that the load is balanced and that the ratio between the line voltage and the average phase voltage in the circuit is equal to √3. You will verify that the current flowing in the neutral line is equal to zero and that removing the neutral line does not affect the measured voltages and currents.

1. Set up the wye - connected, resistive, three-phase circuit shown in Figure 6.

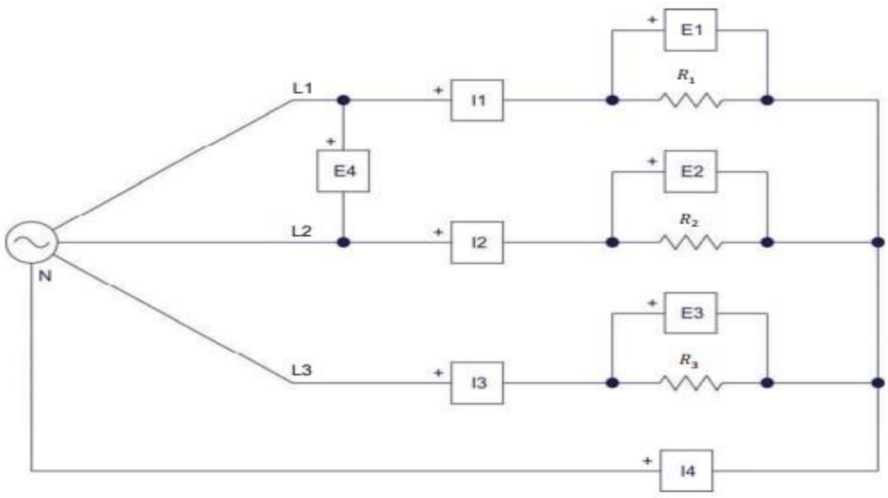


Figure 6- 5:Wye-connected, three-phase circuit.

2. Use the appropriate values of resistor given in Table. 3

Table 3: Three Phase resistive load

Local ac power network		R_1 (Ω)	R_2 (Ω)	R_3 (Ω)
Voltage (V)	**Frequency (Hz)**			
120	60	300	300	300
220	50	1100	1100	1100
240	50	1200	1200	1200
220	60	1100	1100	1100

3. Measure and record the below voltages and currents in the circuit of Figure 6.

Table 4

E_{R1}	E_{R2}	E_{R3}	E_{Line}	I_{R1}	I_{R2}	I_{R3}	I_{Line}

4. Compare the individual voltages E_{R1}, E_{R2}, E_{R3} measured in the previous step. Are they approximately equal?

☐ **Yes** ☐ **No**

5. Compare the individual voltages E_{R1}, E_{R2}, E_{R3} measured in the previous step. Are they approximately equal?

☐ **Yes** ☐ **No**

6. Is the current I_N flowing in the neutral line is approximately equal to zero?

☐ **Yes** ☐ **No**

7. Does this mean that load is balanced?

☐ **Yes** ☐ **No**

Delta Connection

Lab Task 3:

1. Set up the delta-connected, resistive, three-phase circuit shown in Figure 7.

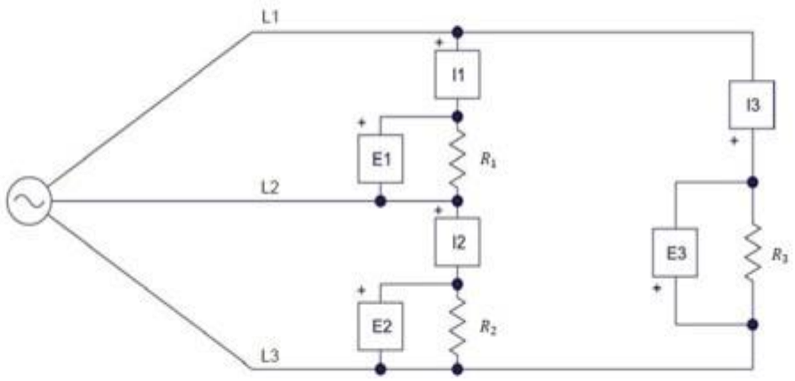


Figure 6- 6:Delta connected three phase resistive.

2. Use the appropriate values of resistor given in Table 5.

Table 5: Three Phase resistive load

Local ac power network		R_1 (Ω)	R_2 (Ω)	R_3 (Ω)
Voltage (V)	**Frequency (Hz)**			
120	60	300	300	300
220	50	1100	1100	1100
240	50	1200	1200	1200
220	60	1100	1100	1100

Turn the three-phase ac power source in the Power Supply on.

3. Measure and record below the voltages and currents in the circuit of Figure 7 in Table 6

. Table 6

E_{R1}	E_{R2}	E_{R3}	I_{R1}	I_{R2}	I_{R3}

4. Compare the individual load voltages E_{R1}, E_{R2}, E_{R3} measured in the previous step. Are they approximately equal?

☐ **Yes** ☐ **No**

5. Compare the individual load currents I_{R1}, I_{R2}, I_{R3} measured in the previous step. Are they approximately equal?

☐ **Yes** ☐ **No**

6. Does this mean that the load is balanced?

☐ **Yes** ☐ **No**

7. Calculate the average phase current using the phase current values recorded in step 3.

$$AverageIphase = \frac{(I_{R1} + I_{R2} + I_{R3})}{3} = \_\_\_\_\_\_\_\_ A$$

8. Reconnect the ammeter inputs I_1, I_2, and I_3 as shown in Figure 2 to measure the line currents in the delta-connected, three-phase circuit.

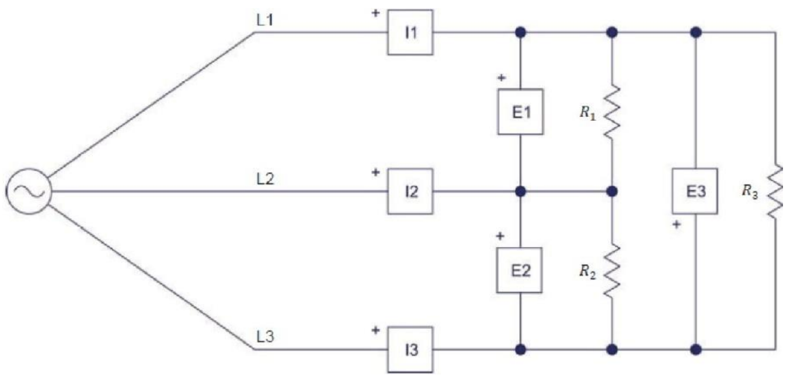


Figure 6- 7:Line current measurements in the delta-connected three-phase circuit.

9. Measure and record below the line currents in the circuit of Figure 2 and determine the average value of the line currents.

Table 1

I_{Line1}	I_{Line2}	I_{Line3}

$$AverageILine = \frac{(I_{Line1} + I_{Line2} + I_{Line3}}{3} = \_\_\_\_\_\_\_\_\_A$$

10. Calculate the active power dissipated in each phase of the circuit and the total active power dissipated in the circuit using the voltages and currents recorded in step 3.

$$P_{R1} = E_{R1} * \mathrm{I}_{R1} = \_\_\_\_\_\_\_\_\_\_\_\_\_W$$

$$P_{R2} = E_{R2} * \mathrm{I}_{R2} = \_\_\_\_\_\_\_\_\_\_\_\_\_W$$

$$P_{R3} = E_{R3} * \mathrm{I}_{R3} = \_\_\_\_\_\_\_\_\_\_\_\_\_W$$

11. Calculate the average phase voltage using the phase voltages recorded in step 3.

$$AverageEphase = \frac{E_{R1} + E_{R2} + E_{R3}}{3} = \_\_\_\_\_\_\_\_\_V$$

12. Calculate the total active power dissipated in the circuit using the average phase voltage recorded in the previous step and average phase current recorded in step 9.

$$P_T = 3\,(E_{phase} * I_{phase} = \_\_\_\_\_\_\_\_\_\_\_\_\_W$$

13. Compare the result with the total active power calculated in step 11. Are both values approximately equal?

☐ **Yes** ☐ **No**

Post Lab Task

1. Explain the difference between the phase voltage and the line voltage in a three-phase circuit
2. What is the ratio between the line and phase voltages and the ratio between the line and phase currents in a wye-connected, three-phase circuit?
3. Calculate the phase voltage, phase current, line voltage and line current for delta connected unbalanced load. Attach the results with the experiment.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ______________________________ Date: ____________________

LAB # 7: To reproduce RC Low and High pass Filter by selecting appropriate components using hardware & software tools

Objectives

- To reproduce the series RC low pass and high pass filters to display its cutoff frequency using Ltspice.
- To design a simple RC low and high pass filter for cut off frequency of 4.8kHz using Ltspice.
  .
- To trace the Bode plot for low and high pass RL filters using graphical techniques.

Pre-Lab

Filter: A filter is a circuit that is designed to pass signals with desired frequencies and reject or attenuate others.

Passive Filters: A filter that only perform the filtering (it does not provide ant gain to the system). The elements used in passive filters are resistors, capacitors and inductors. The gain of passive filters is unity.

Low Pass Filter: A low pass filter passes low frequencies and rejects high frequencies. The simplest filters can be implemented with RC circuits. The key to analysing them by inspection is to remember the behaviour of capacitors. Remember that capacitor blocks low frequencies and pass high frequencies.

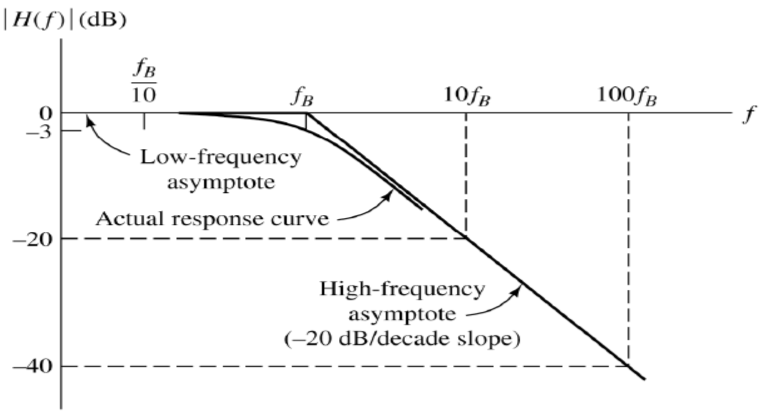


Figure 7- 1:Magnitude plot for First-Order Low Pass Filter.

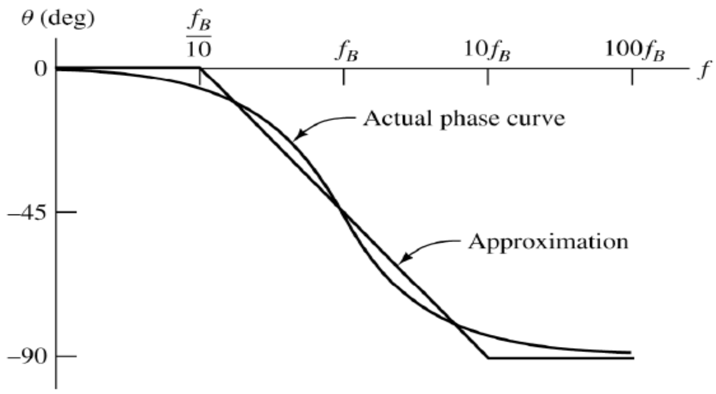


Figure 7- 2:Phase plot for First-Order Low Pass Filter.

High Pass Filter

A high pass filter passes high frequencies and rejects low frequencies.

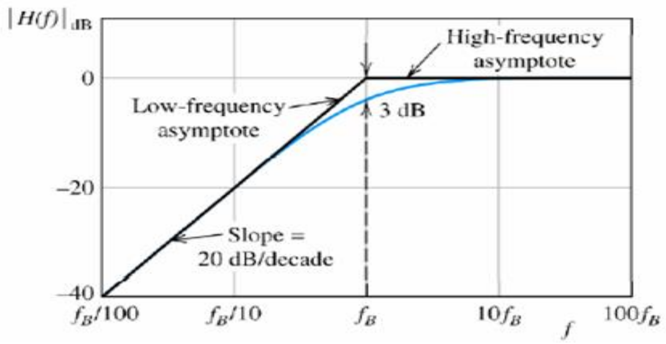


Figure 7- 3:Magnitude plot for First-Order High Pass Filter.

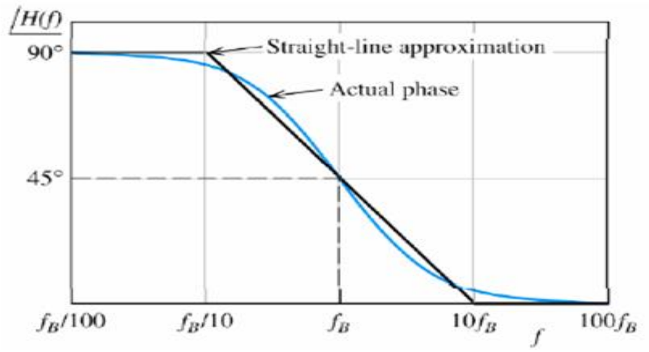


Figure 7- 4:Phase plot for First-Order High Pass Filter.

Frequency Response: The frequency response of a circuit is the variation in its behaviour with change in signal frequency.

Transfer Function: The transfer function $H(\omega)$ of a circuit is the frequency dependent ratio of a phasor output $Y(\omega)$ to a phasor input $X(\omega)$.

$$H(\omega) = \frac{Y(\omega)}{X(\omega)}$$

Bode Plots: Bode plots are semi-log plots of the magnitude (in decibels) and phase (in degrees) of a transfer function versus frequency.

Cut off Frequency: The frequency at which gain becomes 0.707 times of the maximum input is called cut off frequency. It is also called the half power frequency because at this frequency the average power delivered by the circuit is one half the maximum average powers.

$$|H(\omega_C)| = 0.707\ |H(\omega)|_{MAX}$$

$$P(\omega_c) = \frac{P_{max}}{2}$$

$$|H(\omega)| = \frac{1}{\sqrt{(1)^2 + \left(\frac{\omega}{\omega_c}\right)^2}}$$

$$\phi = -\tan^{-1}\left(\frac{\omega}{\omega_c}\right)$$

It can be seen from above equations that at$\omega = 0,\ |H(\omega)| = 1\ and\phi = 0$

It can be seen from equations that at$\omega = \infty,\ |H(\omega)| = 0\ and\phi = -90^0$

Pre-Lab Exercise

Read the pre-lab before coming to lab

In Lab Task

Lab Task 1: Design of RC Low Pass Filter

The cut off frequency for RC circuit is given below:

$$\omega_c = \frac{1}{RC}\ , f_c = \frac{1}{2\pi RC}$$

$$Let\ \mathrm{C} = 10\ \mathrm{nF}\ , \mathrm{C} = 9.08\ \mathrm{nF}$$

$$f_c = 5KHz$$

$$R = \frac{1}{(2\pi)(5 * 10^3)(9.08 * 10^{-9})}$$

$$H(\omega) = \frac{1}{1 + j\omega RC}$$

$$\omega_c = \frac{1}{RC}$$

$$H(\omega) = \frac{1}{1 + \frac{j\omega}{\omega_c}}$$

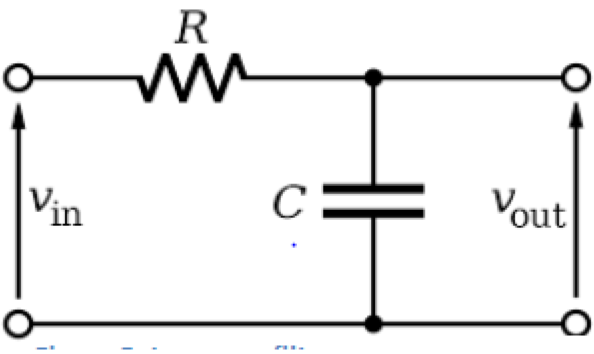


Figure 7- 5:RC Low Pass Filter.

Table 1: RC Low Pass Filter

Frequency (KHz)	ω (rad/s)	$\lvert H(\omega)\rvert = \frac{1}{\sqrt{(1)^2 + \left(\frac{\omega}{\omega_c}\right)^2}}$	$\phi = -\tan^{-1}\left(\frac{\omega}{\omega_c}\right)$	$\frac{V_o}{V_{in}}$	$H(\omega) = 20log\frac{V_o}{V_{in}}$ (dB)	$\phi\ (\omega)$
$0.01f_c$=						
$0.1f_c$ =						
$0.5f_c$ =						
f_c =						
$2f_c$ =						
$4f_c$ =						
$6f_c$ =						
$8f_c$ =						
$10f_c$ =						
$100f_c$ =						

Lab Task 2: Design of RC High Pass Filter

The transfer function of circuit

$H(\omega) = \frac{1}{1+1/j\omega RC}$

$\omega_c = \dfrac{1}{RC}$

$RC = \dfrac{1}{\omega_c}$

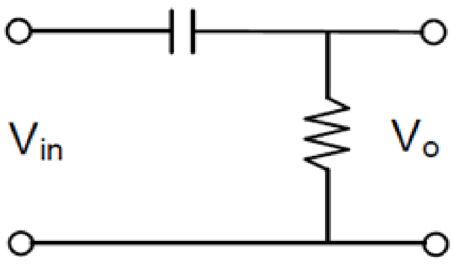


Figure 7- 6:RC High Pass filter.

$Let\ \mathrm{C} = 10\ \mathrm{nF}\ , \mathrm{C} = 9.08\ \mathrm{nF}$

$f_c = 5KHz$

$H(\omega) = \frac{1}{1+1/j\omega RC} = \frac{1}{1-j\frac{\omega_c}{\omega}}$

$$|H(\omega)| = \frac{1}{\sqrt{(1)^2 + \left(-\frac{\omega_c}{\omega}\right)^2}}$$

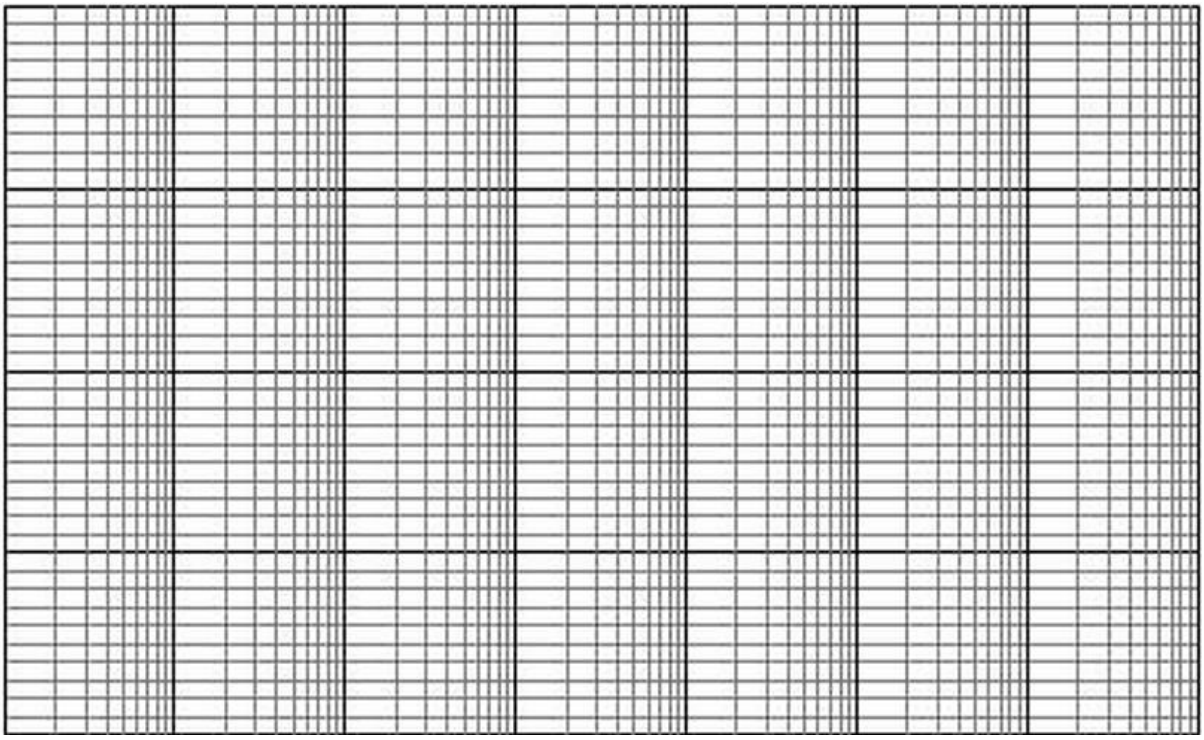
$$\phi = -\tan^{-1}\left(-\frac{\omega_c}{\omega}\right) = \tan^{-1}\left(\frac{\omega_c}{\omega}\right)$$

It can be seen from above equations that at$\omega = 0, |H(\omega)| = 0\ and \phi = 90^0 and at \omega = \infty, |H(\omega)| = 1\ and \phi = 0^0$.

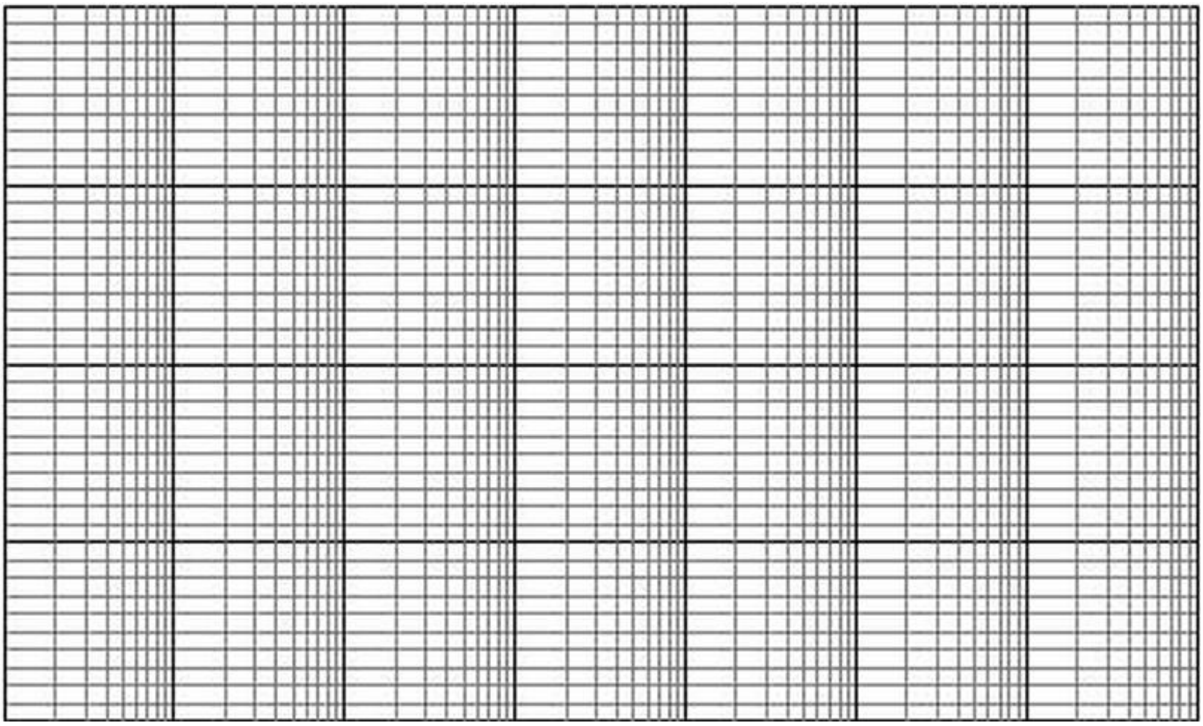
Table 2: RC High Pass Filter

Frequency (KHz)	ω (rad/s)	$\lvert H(\omega)\rvert = \frac{1}{\sqrt{(1)^2+\left(-\frac{\omega_c}{\omega}\right)^2}}$	$\phi = -\tan^{-1}\left(\frac{\omega_c}{\omega}\right)$	$\frac{V_o}{V_{in}}$	$H(\omega) = 20log\frac{V_o}{V_{in}}$ (dB)	$\phi\ (\omega)$
$0.01f_c$=						
$0.1f_c$ =						
$0.5f_c$ =						
f_c =						
$2f_c$ =						
$4f_c$ =						
$6f_c$ =						
$8f_c$ =						
$10f_c$ =						
$100f_c$ =						

Bode Plot for RC Low Pass Filter



Bode Plot for RC High Pass Filter:



Post Lab Task

1. On the bases of measured data, how cut off frequency is determined.
2. “It is preferred that filters are analyzed in frequency domain rather than time domain”, elaborate this point.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ______________________________ Date: ____________________

LAB # 8: To reproduce RL Low and High pass Filter by selecting appropriate components using hardware & software tools

Objectives

- To reproduce the series RL low pass and high pass filters to display its cutoff frequency.
- To design a simple RL low and high pass filter for cut off frequency of 4.8kHz
- To trace the Bode plot for low and high pass RL filters using graphical techniques.

Pre-Lab

Filter: A filter is a circuit that is designed to pass the certain range of frequency signals and reject or attenuate others.

Passive Filters: : A filter that only perform the filtering (it does not provide ant gain to the system). The elements used in passive filters are resistors, capacitors and inductors. The gain of passive filters is unity

Low Pass Filter:A low pass filter passes low frequencies and rejects high frequencies. The simplest filters can be implemented with RC and RL circuits. The key to analyzing them by inspection is to remember the behaviour of inductors and capacitors. Remember that inductors pass low frequencies and block high frequencies, while capacitors do the opposite.

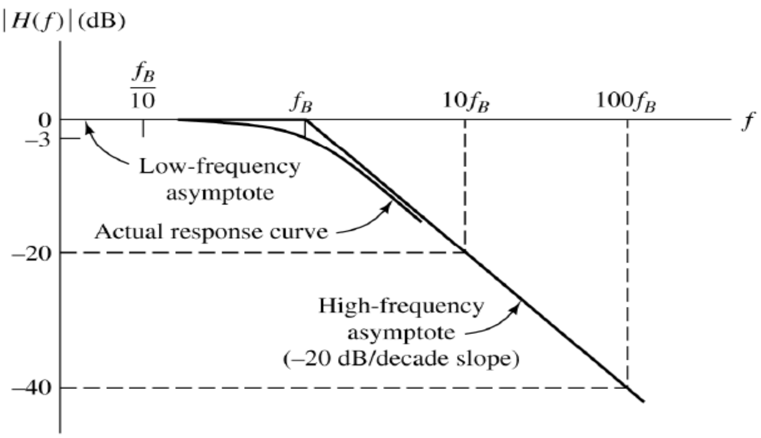


Figure 8- 1:Magnitude plot for First-Order Low Pass Filter.

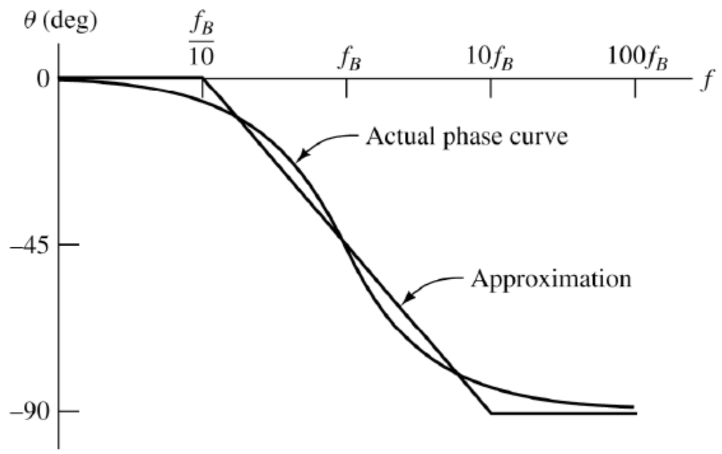


Figure 8- 2:Phase plot for First-Order Low Pass Filter.

High Pass Filter

A high pass filter passes high frequencies and rejects low frequencies. The Bode plots are shown below in Figure 3 and 4. Notice that the effect is opposite to that of the low pass filter. High frequencies are largely untouched, in both magnitude and phase, while low frequencies are increasingly attenuated and phase-shifted

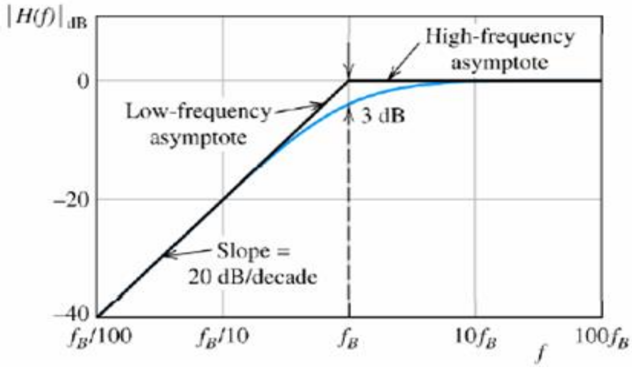


Figure 8- 3:Magnitude plot for First-Order High Pass Filter.

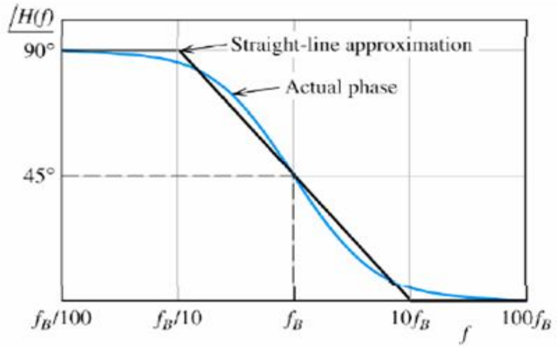


Figure 8- 4:Phase plot for First-Order High Pass Filter.

Frequency Response: The frequency response of a circuit is the variation in its behaviour with change in signal frequency.

Transfer Function: The transfer function $H(\omega)$ of a circuit is the frequency dependent ratio of a phasor output $Y(\omega)$ to a phasor input $X(\omega)$.

$$H(\omega) = \frac{Y(\omega)}{X(\omega)}$$

Bode Plots: Bode plots are semi-log plots of the magnitude (in decibels) and phase (in degrees) of a transfer function versus frequency.

Cut off Frequency: The frequency at which gain becomes 0.707 times of the maximum input is called cut off frequency. It is also called the half power frequency because at this frequency the average power delivered by the circuit is one half the maximum average powers.

$$|H(\omega_C)| = 0.707\ |H(\omega)|_{MAX}$$

$$P(\omega_c) = \frac{P_{max}}{2}$$

$$|H(\omega)| = \frac{1}{\sqrt{(1)^2 + \left(\frac{\omega}{\omega_c}\right)^2}}$$

$$\phi = -\tan^{-1}\left(\frac{\omega}{\omega_c}\right)$$

It can be seen from above equations that at$\omega = 0,\ |H(\omega)| = 1\ and \phi = 0$

It can be seen from equations that at$\omega = \infty,\ |H(\omega)| = 0\ and \phi = -90^0$

Pre-Lab Exercise

Read the prelab part before coming to lab

In Lab Tasks

Lab Task 1: Design a RL Low Pass filter

The cut off frequency for RL circuit is given below

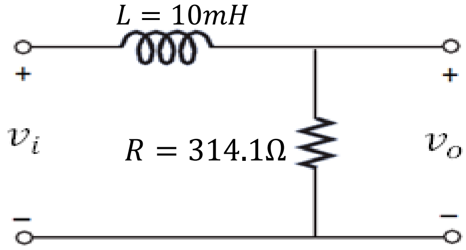


Figure 8- 5: RL Low Pass Filter .

$$\omega_c = \frac{R}{L} \quad , f_c = \frac{R}{2\pi L}$$

$Let L = 10\ mH and f_c = 5KHz$
$R = 2\pi f_c L$
$R = 2\pi * 5 * 10^3 * 10 * 10^{-3}$
$R = 314.1\Omega$

$$H(\omega) = \frac{1}{1 + \frac{j\omega L}{R}}$$

$$\omega_c = \frac{R}{L}$$

$$Put \frac{L}{R} = \frac{1}{\omega_c}$$

$$H(\omega) = \frac{1}{1 + \frac{j\omega}{\omega_c}}$$

$$|H(\omega)| = \frac{1}{\sqrt{(1)^2 + \left(\frac{\omega}{\omega_c}\right)^2}}$$

$$\phi = -\tan^{-1}\left(\frac{\omega}{\omega_c}\right)$$

It can be seen above equations that at $\omega = 0, |H(\omega)| = 1$ and $\phi = 0$ and at $\omega = \infty$, $|H(\omega)| = 0$ and $\phi = -90^0$.

Table 1: RL Low Pass Filter

Frequency (KHz)	ω (rad/s)	$\lvert H(\omega)\rvert = \frac{1}{\sqrt{(1)^2 + \left(\frac{\omega}{\omega_c}\right)^2}}$	$\phi = -\tan^{-1}\left(\frac{\omega}{\omega_c}\right)$	$\frac{V_o}{V_{in}}$	$H(\omega) = 20log\frac{V_o}{V_{in}}$ (dB)	$\phi\ (\omega)$
$0.01f_c$=						
$0.1f_c$ =						
$0.5f_c$ =						
f_c =						
$2f_c$ =						
$4f_c$ =						
$6f_c$ =						
$8f_c$ =						
$10f_c$ =						
$100f_c$ =						

Lab Task 2: Design a RL High Pass filter

The values of R and L is same as found

in RL low pass filter.

Transfer function of the circuit shown in Figure. 6

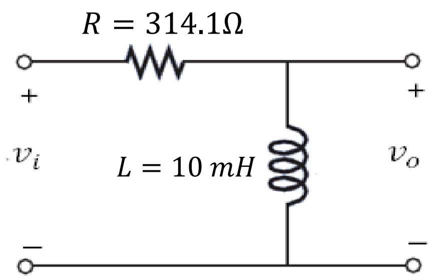


Figure 8- 6:RL High Pass Filter.

$$H(\omega) = \frac{1}{1 + {R}/{j\omega L}}$$

$$\omega_c = \frac{R}{L}$$

$$Put\,\frac{R}{L} = \omega_c$$

$$H(\omega) = \frac{1}{1 + {R}/{j\omega L}} = \frac{1}{1 - j\frac{\omega_c}{\omega}}$$

$$|H(\omega)| = \frac{1}{\sqrt{(1)^2 + \left(-\frac{\omega_c}{\omega}\right)^2}}$$

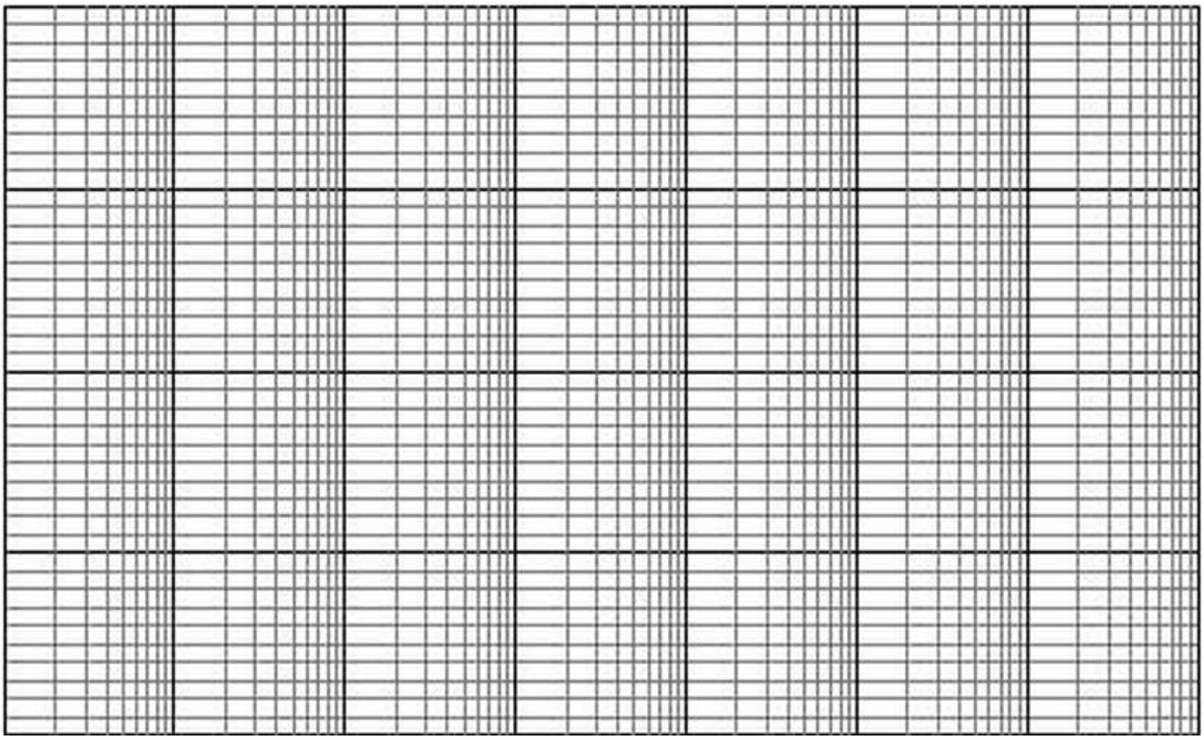
$$\phi = -\tan^{-1}\left(-\frac{\omega_c}{\omega}\right) = \tan^{-1}\left(\frac{\omega_c}{\omega}\right)$$

It can be seen above equations that at$\omega = 0, |H(\omega)| = 0\ and \phi = 90\ and at \omega = \infty,\ |H(\omega)| = 1\ and \phi = 0^0$.

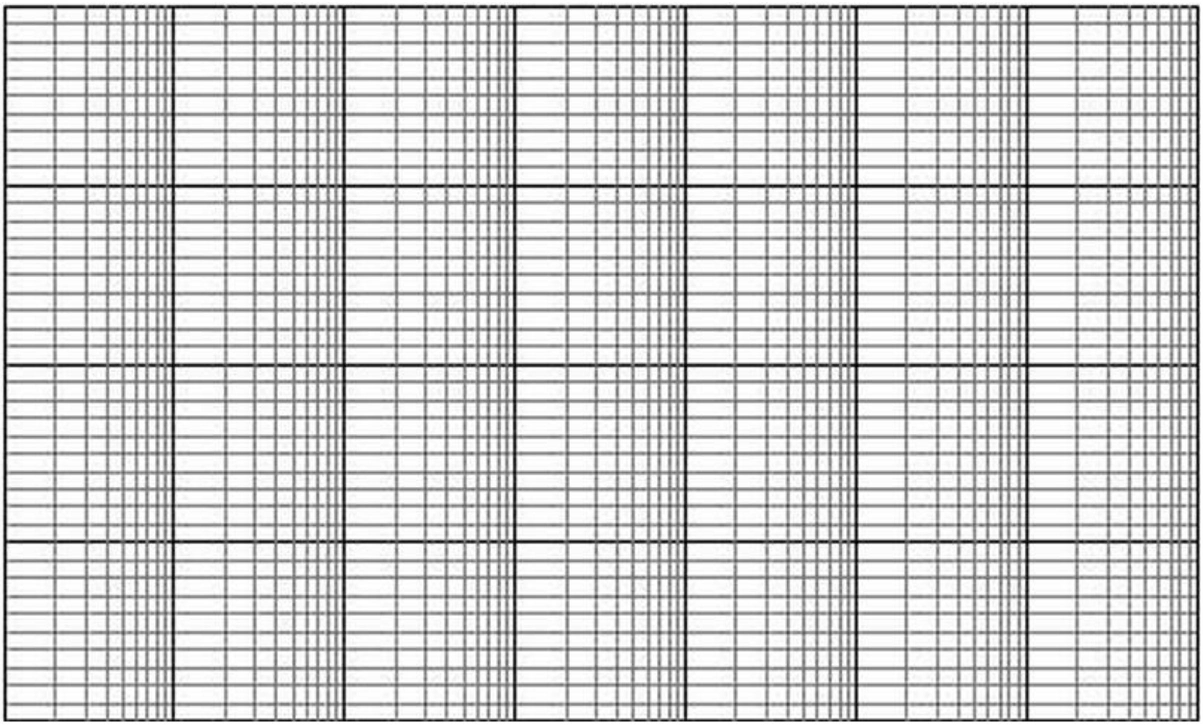
Table 3: RL High Pass Filter

Frequency (KHz)	ω (rad/s)	$\lvert H(\omega)\rvert = \frac{1}{\sqrt{(1)^2 + \left(-\frac{\omega_c}{\omega}\right)^2}}$	$\phi = -\tan^{-1}\left(\frac{\omega_c}{\omega}\right)$	$\frac{V_o}{V_{in}}$	$H(\omega) = 20log\frac{V_o}{V_{in}}$ (dB)	$\phi\ (\omega)$
$0.01f_c$=						
$0.1f_c$ =						
$0.5f_c$ =						
f_c =						
$2f_c$ =						
$4f_c$ =						
$6f_c$ =						
$8f_c$ =						
$10f_c$ =						
$100f_c$ =						

Bode Plot for RL Low Pass Filter



Bode Plot for RL High Pass Filter:



Post Lab Task

- After designing RC and RL filters, how will you differentiate between RC and RL filters
- Give suggestions to improve the performance of low pass and high pass filters.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ________________________________ Date: _____________________

LAB # 9:To reproduce Passive filters (Band pass, Band stop) by selecting appropriate components using software tool

Objective

- To design band-pass and band-stop filters for given specified center frequency using standard design criteria.
- To reproduce the designed circuit and show the parameters for computing center frequency using LTspice.
- To show the difference of measured center frequency with calculated value using LTspice and sketch its frequency response.

Pre-Lab

Band Pass Filter

The *RLC* series resonant circuit provides a bandpass filter when the output is taken off the resistor as shown in Figure.1. The transfer function is:

$$H(w) = \frac{V_o}{V_i} = \frac{R}{R + j(wL - {}^{1}/_{wC})}$$

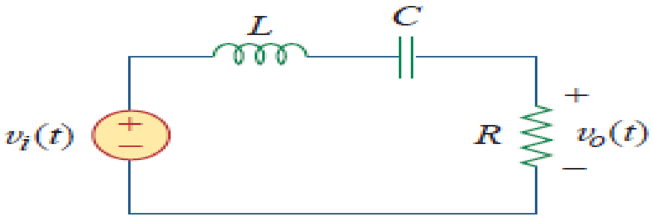


Figure 9- 1:RLC series resonant circuit.

We observe $H(0) = 0$ and $H(\infty) = 0$

Figure 2 shows the plot of $|H(w)|$. The band pass filter passes a band of frequencies ($w_1 < w < w_2$) centered on w_o.The center frequency is given by:

$$w_o = \frac{1}{\sqrt{LC}}$$

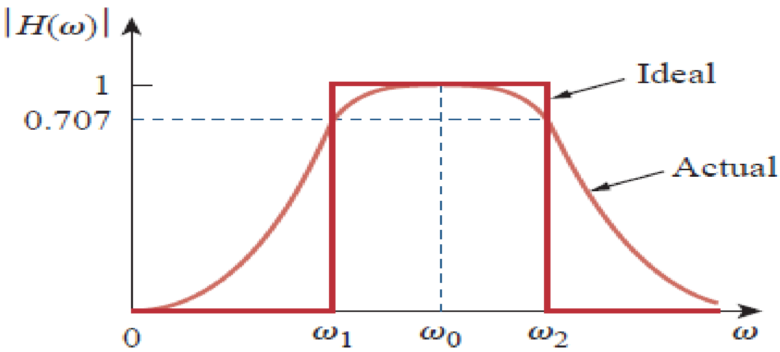


Figure 9- 2:Ideal and actual frequency response of a band pass filter.

A bandpass filter can also be formed by cascading the low pass filter ($w_2 = w_c$) with the high pass filter ($w_1 = w_c$). However, the result would not be the same as just adding the output of the low pass filter to the input of the high pass filter, because one circuit loads the other and alters the desired transfer function.

Band stop Filter

A filter that prevents a band of frequencies between two designated values ($w_1 and w_2$) and passing all other frequencies is known as a band stop, band reject, or notch filter. Frequency response of band stop filter is shown in Figure 3. Band stop filter is formed when the output is taken off the *LC* series combination of series *RLC* resonant circuit as shown in Figure 4. The transfer function is:

$$H(w) = \frac{V_o}{V_i} = \frac{j(wL - {}^1/_{wC})}{R + j(wL - {}^1/_{wC})}$$

We observe $H(0) = 1$ and $H(\infty) = 1$

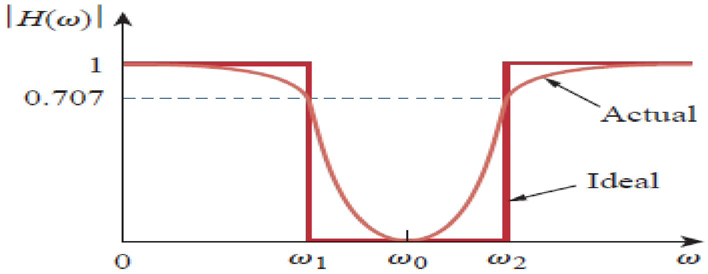


Figure 9- 3:Ideal and actual frequency response of a band stop filter.

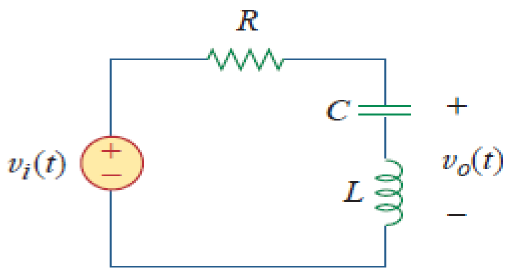


Figure 9- 4:A band stop filter.

The center frequency is given by:

$$w_o = \frac{1}{\sqrt{LC}}$$

Pre lab Exercise

1. If the band stop filter in Figure 4 is to reject a 200-Hz sinusoid while passing all other frequencies, calculate the values of *L* and *C.* Take R = 150Ω and the bandwidth as 100 Hz.
2. Design a series *RLC* type band pass filter with cutoff frequencies of 10 kHz and 11 kHz. Assuming *C* = 80 pF, find *R*, *L*, and *Q*.

In Lab Task

Task 1: RLC Band Pass Filter

1. Construct the circuit shown in Figure 1. Use the supply voltage of $V_{in} = 10\ V_{pp}$.
2. Use values of R, L, C which you have found in question 2 of pre lab section.
3. Connect channels 1 and 2 of the oscilloscope to measure V_{in} and V_{out} simultaneously.
4. Vary the frequency from $10\ Hz\ \ to\ 100\ KHz$ and record the indicated value in Table 1. With each frequency change, make sure that V_{in} is still $10\ V_{pp}$.

   Note: Remember input voltage remains constant.
5. Using the data of Table 1, sketch a Bode plot of the filter's output voltage.

Table 1 Band Pass Filter

Frequency (KHz)	$\omega\ (rad/s)$	$H(\omega) = \dfrac{R}{R + j(\omega L - \frac{1}{\omega C})}$
$0.01f_r$=		
$0.1f_r$ =		
$0.5f_r$ =		
f_r =		
$2f_r$ =		
$4f_r$ =		
$6f_r$ =		
$8f_r$ =		
$10f_r$ =		
$100f_r$ =		

Task 2: RLC Band Stop Filter

1. Construct the circuit shown in Figure 4. Use the supply voltage of $V_{in} = 10\ V_{pp}$.
2. Use values of R, L, C which you have found in question 1 of pre lab section.
3. Connect channels 1 and 2 of the oscilloscope to measure V_{in} and V_{out} simultaneously.
4. Vary the frequency from $10\ Hz\ to\ 10\ KHz$ and record the indicated value in Table 2. With each frequency change, make sure that V_{in} is still $10\ V_{pp}$.
   Note: Remember input voltage remains constant.
5. Using the data of Table 2, sketch a Bode plot of the filter's output voltage.

Table 2 Band Stop Filter

Frequency(KHz)	$\omega\ (rad/s)$	$H(\omega) = \dfrac{j(\omega L - {}^{1}/_{\omega C})}{R + j(\omega L - \frac{1}{\omega C})}$
$0.01f_r$=		
$0.1f_r$ =		
$0.5f_r$ =		
f_r =		
$2f_r$ =		
$4f_r$ =		
$6f_r$ =		
$8f_r$ =		
$10f_r$ =		
$100f_r$ =		

Post Lab Task

1. Design a passive band stop filter with $w_o = 10$ rad/sec and Q = 20.
2. Construct the Bode plot of above designed filter.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ________________________________ Date: ____________________

LAB # 10: To reproduce series and parallel resonance RLC circuit using software tool

Objectives

- To compute the resonant frequency of series and parallel RLC circuit using standard circuit technique.
- To show the resonant frequency of series and parallel RLC circuit and LTSpice
- To design a series and parallel resonance circuit for desired resonant frequency using standard circuit technique.
- To reproduce the response of a series and parallel resonance circuit for desired resonant frequency using LTSpice
- To show the difference of the measured resonant frequency with calculated value of a series and parallel resonance circuit using LTSpice and sketch its frequency response.

Pre Lab

Introduction

Resonance occurs in a circuit when the voltage and current are in phase and the input impedance of the circuit is purely resistive. When plotted on a Bode plot, the response starts at zero, reaches a maximum value in the vicinity of the natural resonant frequency, and then drops again to zero as ω becomes infinite.

So far, we have analysed the behavior of series RLC circuit whose source voltage has fixed frequency. What would happen to the characteristics of the circuit if a supply voltage of fixed amplitude but of different frequencies was applied to the circuit? Also what would the circuits "frequency response" due to this varying frequency.

In a series RLC circuit, there is frequency point where the inductive reactance becomes equal in value to the capacitive reactance ($X_L = X_c$). The point at which this occurs is called the Resonant Frequency (f_r)of the circuit, and as we are analysing a series RLC circuit this resonance frequency produces a Series Resonance.

Series Resonance circuits are one of the important circuits used in electrical and electronic circuits. They can be found in various forms such as in AC mains filters, noise filters and also in radio and television tuning circuits producing a very selective tuning circuit for the receiving of the different frequency channels. Consider the series RLC circuit in Figure 1.

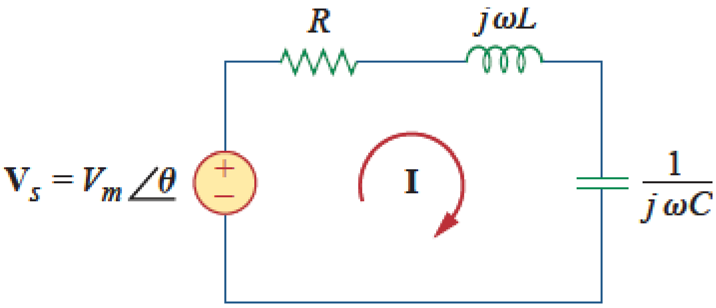


Figure 10- 1:Series RLC circuit.

Inductive reactance: $X_L = 2\pi f L = wL$
Capacitive reactance: $X_c = \frac{1}{2\pi f C} = \frac{1}{wC}$
When $X_L > X_c$ the circuit is Inductive
When $X_c > X_L$ the circuit is Capacitive
Total circuit reactance= $X_T = X_L - X_c$
Total circuit impedance= $Z = \sqrt{R^2 + X^2{}_T} = R + jX$
From the above equations, we notice that inductive reactance increases if frequency or the inductance increases. As the frequency approaches infinity the inductive reactance would also increase towards infinity with the circuit element acting like an open circuit. However, as the frequency approaches zero or DC, the inductive reactance would decrease to zero, causing the opposite effect acting like a short circuit. This means then that inductive reactance is proportional to frequency and is small at low frequencies and high at higher frequencies and this is demonstrated in the Figure 2:

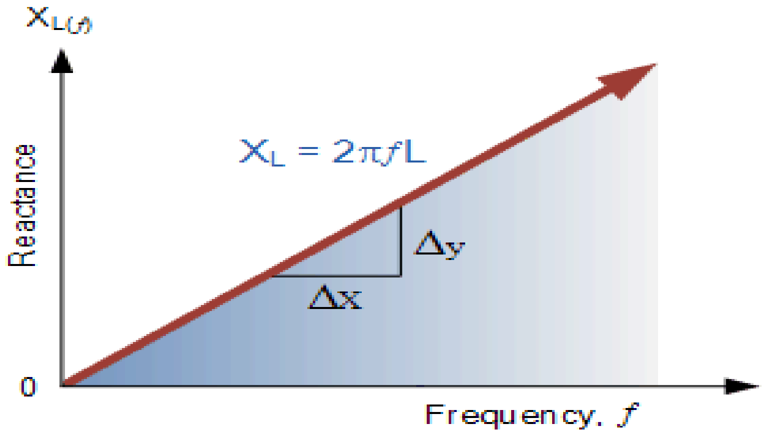


Figure 10- 2:Inductive Reactance variation with frequency.

If "frequency or the capacitance is increased the overall capacitive reactance would decrease. As the frequency approaches infinity, the capacitors reactance would reduce to zero causing the circuit element to act like a perfect conductor.
But as the frequency approaches zero or DC level, the capacitors reactance would rapidly increase up to infinity causing it to act like very large resistance acting like an open circuit condition. This means then that capacitive reactance is inversely proportional to frequency for any given value of capacitance and this is shown in Figure 3:

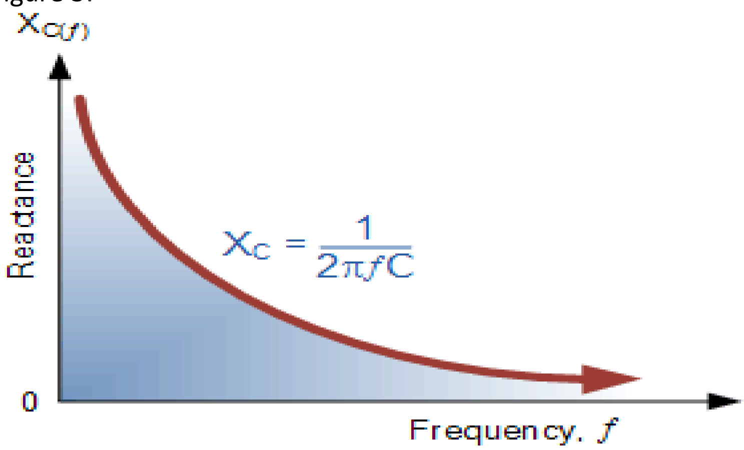


Figure 10- 3:Capacitive Reactance variation with frequency.

We have seen that the value of the reactance depends upon the frequency of the input supply. At a higher frequency X_L is high and at a low frequency X_C is high. Then there must be a frequency point were the value of X_L is the same as the value of X_c. If we now place the curve for inductive reactance on top of the curve for capacitive reactance so that both curves are on the same axes, the point of intersection will give us the series resonance frequency point, (f_r) as shown below in Figure 4.

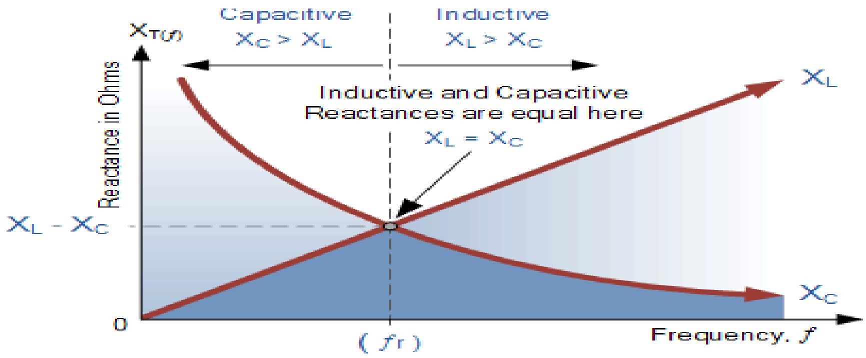


Figure 10- 4:Resonance curve.

Where f_r is in Hertz, L is in Henry and C is in Farad.
The resonant frequency, $f_r$point can be calculated as follows.

$$X_L = X_C \longrightarrow 2\pi f L = \frac{1}{2\pi f C}$$

$$f^2 = \frac{1}{2\pi L \times 2\pi C} = \frac{1}{4\pi^2 LC}$$

$$f = \sqrt{\frac{1}{4\pi^2 LC}}$$

$$f_r = \frac{1}{2\pi\sqrt{LC}}\,(Hz) \qquad or \qquad w_r = \frac{1}{\sqrt{LC}}\,(rad)$$

We can see then that at resonance, the inductive reactance and capacitive reactance cancel each other and thereby making a series LC combination a short circuit with the only opposition to current flow being the resistance, R. In complex form, the resonant frequency is the frequency at which the total impedance of a series RLC circuit becomes purely *"real"*, that is no imaginary impedances exist ($Z = R$).The circuit impedance at resonance is called the "dynamic impedance" of the circuit and depending upon the frequency as shown in Figure 5

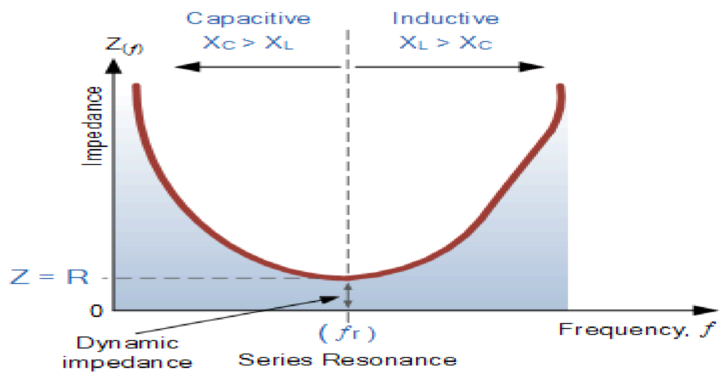


Figure 10- 5:Series resonance circuit impedance variation with frequency.

Series RLC circuit, the voltage across a series combination of RLC is the phasor sum of V_R, V_L and V_C. At resonance the two reactance's are equal and cancelling, the two voltages V_L and V_C must be opposite and equal in value thereby cancelling each other. Thus, in series resonance circuit as V_L=$-V_C$ the resulting reactive voltages are zero and supply voltage is drop across the resistor. Therefore$V_R = V_{supply}$, so series resonance circuits are known as voltage resonance circuits.

Series RLC Circuit at Resonance

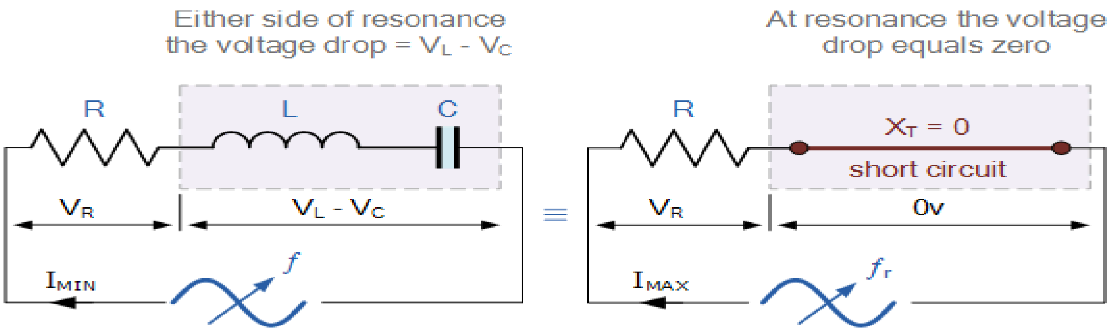


Figure 10- 6:Series RLC circuit at resonance

Series Circuit Current at Resonance

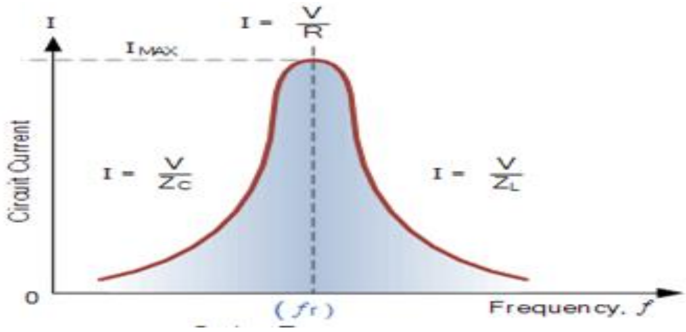


Figure 10- 7:Series RLC circuit current variation with frequency.

You may also notice that as the maximum current through the circuit at resonance is limited only by the value of the resistance (a pure and real value), the source voltage and circuit current must therefore be in phase with each other at this frequency. Then the phase angle between the voltage and current of a series resonance circuit is also a function of frequency for a fixed supply voltage and which is zero at the resonant frequency point when: V, I and V_R are all in phase with each other. Consequently, if the phase angle is zero then the power factor must therefore be unity.

Parallel resonance also has importance in filter circuits, use in electrical and electronic instruments. They can be found in various forms such as in AC mains filters, noise filters and also in radio and television tuning circuits producing a very selective tuning circuit for the receiving of the different frequency channels. Consider the parallel RLC circuit in Figure 8

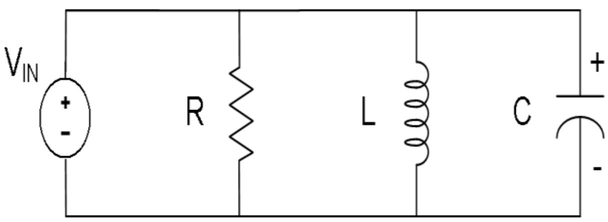


Figure 10- 8:Parallel Resonance Circuit.

The steady-state admittance offered by the circuit is:

$$Y = \frac{1}{R} + j(\omega C - \frac{1}{wL})$$

Resonance occurs when the voltage and current at the input are in phase. This corresponds to a purely real admittance, so that the necessary condition is given by

$$\omega C - \frac{1}{wL} = 0$$

The resonant condition may be achieved by adjusting $L, C, or\ \omega$. Keeping L and C constant, the resonant frequency $\omega_o$is given by:

$$\omega_o = \frac{1}{\sqrt{LC}}$$

$$f_o = \frac{1}{2\pi\sqrt{LC}}$$

Frequency Response

It is a plot of output voltage of a resonance circuit as function of frequency. The response of course starts at zero, reaches a maximum value in the vicinity of the natural resonant frequency, and then drops again to zero as ω becomes infinite. The frequency response is shown in Figure 9.

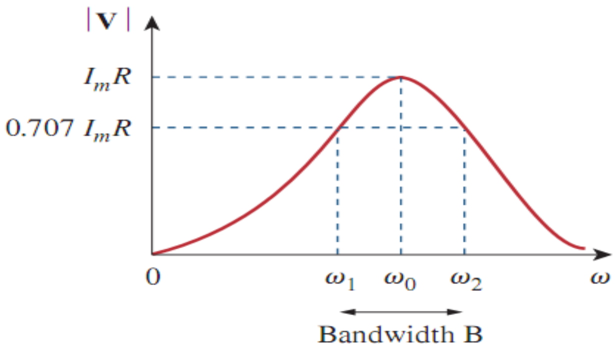


Figure 10- 9:Frequency Response of parallel resonance circuit.

The frequencies $\omega_1$and ω_2 are indicated as half-power frequencies. These frequencies locate those points on the curve at which the voltage response is $\frac{1}{\sqrt{2}}$ or 0.707 times the maximum value. They are used to measure the band-width of the response curve. This is called the half-power bandwidth of the resonant circuit and is defined as:

$$\mathrm{BW} = \omega_2 - \omega_1 = \frac{1}{RC}$$

Pre Lab Exercise

1. In the circuit of Figure 10, $R = 2\Omega, L = 1mHandC = 0.4\mu F$.Find the followings:
    a. Resonant frequency and the half-power frequencies
    b. Calculate the quality factor and bandwidth.
    c. The amplitude of the current at $w_o, w_1 and w_2$

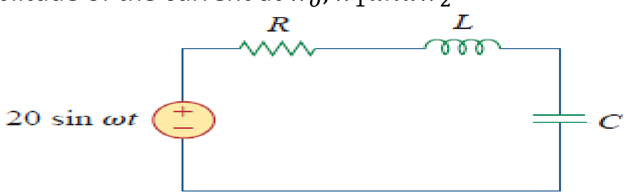


Figure 10- 10:Series RLC circuit.

2. A series-connected circuit shown in Figure 1 has $R = 4\Omega, L = 25mH$
   a. Calculate the value of Cthat will produce a quality factor of 50.
   b. Find $w_1, w_2 and B$
   c. Determine the average power dissipated at $w_o, w_1 and w_2$. Now take $V_{in} = 100V$

3. In the parallel RLC circuit of Figure 11, $R = 8K\Omega, L = 0.2mH and C = 8\mu F$.Find the followings:
   a. Resonant frequency and the half-power frequencies
   b. Inductive and capacitive reactance at resonant frequencies
   c. Current through all three parallel components
   d. Determine power dissipated at$w_o, w_1 and w_2$

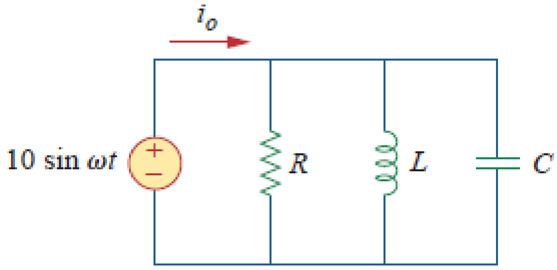


Figure 10- 11: Parallel RLC circuit.

In Lab Task

Task1: Resonant Circuit

1. Implement the circuit of Figure 10 in LTspice and complete the given Table 1.
2. Create Bode plot of the magnitude transfer functions of current.

Table 1 Series Resonance circuit

Frequency (KHz)	$V(in)_{pp}$	$I(out)_{pp}$

Task 2:

1. Design a series resonant circuit in which resonant frequency is of your own choice.

Task 3:

1. Set up the circuit shown in Figure 11. using component values

$$R = 1\,K\Omega, \qquad C = 1\,\mu F and\ L = 33\,mH$$

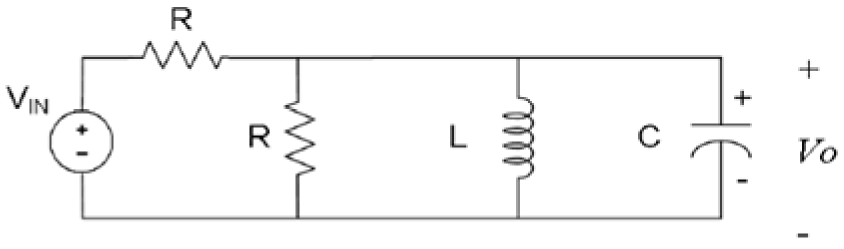


Figure 10- 12:Parallel Resonance circuit.

2. Use $v_{in} = 2V_{pp}$sinusoidal input to the circuit
3. Connect channels 1 and 2 of the oscilloscope to measure V_{in} and V_{out} simultaneously
4. Vary the frequency from $500\,Hz to\,2\,KHz$ in small steps, until at a certain frequency is reached the output of the circuit on Channel B, is maximum. This is the resonant frequency of the circuit.
5. Repeat the task 1 in Figure 4 and use$L = 33mH\ and C = 0.01\mu F\ and\ R = 1\,K\Omega$. The V_o across the resistor is proportional to the series RLC circuit current.
6. Using the data of Table 2, draw the magnitude and phase plots of the of the circuit's output voltage

Table 2 Parallel Resonance circuit

Frequency (KHz)	$v_{in(p-p)}$	$v_{out(p-p)}$

Post Lab Task

1. A 12Ω resistor, your registration number in (μF) capacitor and $8\,mH$ coil is connected in series across an AC source. What is the resonant frequency and draw the magnitude Bode plot.
2. Design the parallel RLC resonant circuit in which resonant frequency is your registration number, assume $R = 1K\Omega$.
3. Draw the magnitude and phase plots for above designed circuit

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ________________________________ Date: ______________________

LAB # 11: To reproduce the circuit and show the impedance, admittance and hybrid parameter for unknown two port network using software tool

Objectives

- To show the impedance, admittance and hybrid parameter for unknown two port network using LTspice

Pre Lab

The aim of this lab is to develop the set of parameters that may be used to relate the output variables of voltage and current, typically designated as $V_2$and I_2, respectively, to the input variables of voltage and current, typically designated as $V_1$and I_1, respectively. This approach is used to characterize a variety of components and circuit element combinations from filters, through transistors, to microwave circuits.

A port consists of a pair of terminals; current enters through one of the terminals and the same current leaves the other terminal. A resistor is a one-port network. In this lab we will study *two-port networks* with one input port and one output port. Such networks are often treated as "black boxes" or modules that may be plugged into a circuit to accomplish some tasks, such as filtering the signal or providing a controlled voltage.

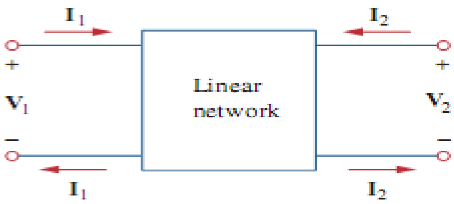


Figure 11- 1: Two port network .

Engineers need a way to characterize the behavior of such a network and have developed several sets of parameters to do that. Each of these parameter sets relates the input (side 1) and output (side 2) voltages and currents. Impedance and admittance parameters are commonly used to characterize filters, and are often useful in designing and characterizing impedance matching and power distribution networks. The term *admittance* is often applied to the use of either impedance or admittance parameters. Be aware that because the voltages and currents are phasors with magnitude and phase angle, the parameters also have magnitude and phase angle. A simple RMS measurement will not suffice.

In Z parameters of a two-port network, the input and output voltages $V_1$and V_2 can be expressed in terms of input and output currents I_1 and I_2. Out of four variables i.e. (V_1, V_2,I_1, I_2) V_1 and V_2 are dependent variables whereas I_1 and I_2 are independent variables. The impedance parameters (***z** parameters*) relate the input and output voltages to the input and output currents by the following two equations:

$$V_1 = z_{11}I_1 + z_{12}I_2$$
$$V_2 = z_{21}I_1 + z_{22}I_2$$

or in matrix notation:

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$

The **z** parameters have units of ohms and are most easily found by applying a set of open-circuit tests on the circuit. When we apply a voltage to the input with the output open-circuited, we can measure the input current and output voltage and find the first two z parameters as follows:

$$z_{11} = \left.\frac{V_1}{I_1}\right]_{I2=0} \quad z_{21} = \left.\frac{V_2}{I_1}\right]_{I2=0}$$

We can determine the other two z parameters by applying a similar test to the output with the input open-circuited:

$$z_{12} = \left.\frac{V_1}{I_2}\right]_{I1=0} \quad z_{22} = \left.\frac{V_2}{I_2}\right]_{I1=0}$$

Sometimes the impedance parameters do not exist because the voltages cannot be described. Therefore, we need alternatives, such as the admittance parameters.

The admittance parameters *(*y parameters*)* relate the input and output currents to the input and output *voltages* by the following two equations:

$$I_1 = y_{11}V_1 + y_{12}V_2$$
$$I_2 = y_{21}V_1 + y_{22}V_2$$

or in matrix notation:

$$\begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix}$$

The y parameters have units of Siemens (or mhos) and are most easily found by applying a set of short circuit tests on the circuit. When we apply a voltage to the inputs with the output short circuited, we can measure the input current and output current to find the first two y parameters:

$$y_{11} = \left.\frac{I_1}{V_1}\right]_{V2=0} \quad y_{21} = \left.\frac{I_2}{V_1}\right]_{V2=0}$$

We can determine the other two y parameters by applying a similar test to the output with the input short-circuited:

$$y_{12} = \left.\frac{I_1}{V_2}\right]_{V1=0} \quad y_{22} = \left.\frac{I_2}{V_2}\right]_{V1=0}$$

There are occasions where neither the impedance nor the admittance parameters exist, so there is need for still another set of parameters.

The hybrid parameters (h parameters) are based on making $V_1$and $I_2$the dependent variables, and relating them to cross-variables V_2 and I_1. The h parameters satisfy the equations

$$V_1 = h_{11}I_1 + h_{12}V_2$$
$$I_2 = h_{21}I_1 + h_{22}V_2$$

or in matrix notation:

$$\begin{bmatrix} V_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} \begin{bmatrix} \mathrm{I}_1 \\ V_2 \end{bmatrix}$$

The h parameters are found using a mix of short- and open-circuit tests as follows:

Short-circuit tests

$$h_{11} = \left.\frac{V_1}{I_1}\right]_{V2=0} \quad h_{21} = \left.\frac{I_2}{I_1}\right]_{V2=0}$$

Open-circuit tests

$$h_{12} = \left.\frac{V_1}{V_2}\right]_{I1=0} \quad h_{22} = \left.\frac{I_2}{V_2}\right]_{I1=0}$$

PROCEDURE

Warning: Unless you are careful in, you're planning, in carrying out the experiment, and in recording your data, you are very likely to make errors. In this lab you must figure out how to do most steps on your own.

Hint: Performing this lab is actually very straightforward, if you take it a step at a time. Think about what you have learned during this semester. If you need to measure the phase angle of a voltage or current, you will need to use the oscilloscope. When necessary, insert a sense resistor of about 10 ohms and measure the voltage across it, including its phase shift, then calculate the magnitude of the current flowing through the sense resistor. Be sure that you have accurately measured the resistor value, though, and not just relied on the nominal value. Use only one resistor decade box (it will make your life easier), and be careful where you put the resistor, so you don't make a grounding error on the oscilloscope.

Carefully record each of the measurements and calculations in table. Think about what you are doing, sketch the circuit, work meticulously and methodically, and record your data carefully in tables.

Pre Lab Exercise

1. Find the hybrid parameters for the two-port network of Figure. 1

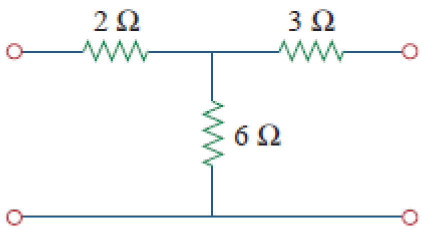


Figure 11- 2:Two port network for hybrid parameters.

In Lab Task

Task 1:

1. Use the function generator to apply a $1\ kHz$ sine wave of about $10\ V_{pp}$ to the input terminals of the circuit in Figure 3.
2. Make the current and voltage measurements necessary to compute the parameters z_{11} and $z_{21}$for the z-parameter set. Note that you must determine not only the magnitude but also the phase of the voltages and currents involved.
3. Now apply the $10\ V_{pp}$, $1\ kHz$ sine wave to the output terminals.
4. Make the current and voltage measurements necessary to calculate the parameters z_{12} and z_{22}.
5. Describe your procedure in your laboratory notebook and record the data necessary to calculate the values for these parameters, including magnitude and phase.

Task 2:

1. Repeat Part 1 to determine the two-port parameter values for the y-parameter set

Task 3:

1. Repeat Part 1, as needed, to determine the two-port parameter values for the h-parameter set.

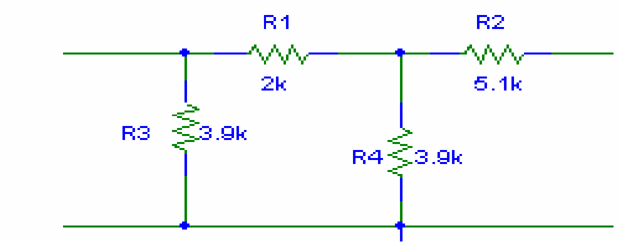


Figure 11- 3:Two port network to acquire impedance and admittance parameters.

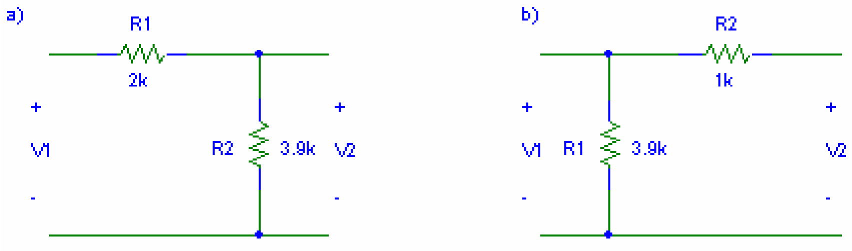


Figure 11- 4:Two port network to acquire hybrid parameters.

Z Parameters

Table 1 Impedance parameters calculation

Sr.NO	When Input port is open circuited ($I_1 = 0$)			When output port is open circuited ($I_2 = 0$)		
	V_2	V_1	I_2	V_2	V_1	V_1

$z_{11} =$ ________________

$z_{21} =$ ________________

$z_{12} =$ ________________

$z_{22} =$ ________________

Y Parameters

Table 2 Admittance parameters calculation

Sr.NO	When Input port is short circuited ($V_1 = 0$)			When output port is short circuited ($V_2 = 0$)		
	V_2	I_1	I_2	V_1	I_1	I_2

$$y_{11} = \_\_\_\_\_\_\_\_\_\_\_\_\_\_$$
$$y_{12} = \_\_\_\_\_\_\_\_\_\_\_\_\_\_$$
$$y_{21} = \_\_\_\_\_\_\_\_\_\_\_\_\_\_$$
$$y_{21} = \_\_\_\_\_\_\_\_\_\_\_\_\_\_$$

h Parameters

Table 3 hybrid parameters calculation

Sr.NO	When output port is short circuited			When input port is open circuited		
	V_1	I_1	I_2	V_2	V_1	I_2

$$h_{11} = \_\_\_\_\_\_\_\_\_\_\_\_\_\_$$
$$h_{12} = \_\_\_\_\_\_\_\_\_\_\_\_\_\_$$
$$h_{21} = \_\_\_\_\_\_\_\_\_\_\_\_\_\_$$
$$h_{22} = \_\_\_\_\_\_\_\_\_\_\_\_\_\_$$

Post Lab Task

1. Prove that the parameter values, which you have calculated for the various two-port network representations (impedance, admittance and hybrid) in last exercise is valid for frequency, value of frequency is your registration number.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ________________________________ Date: _____________________

LAB # 12: To reproduce second and third order RC low pass filter circuits using hardware & software tool

Objectives

- To design second and third order low pass RC filters for desired cut-off frequency using standard design criteria
- To reproduce the designed circuit and show the parameters for computing cut-off frequency using trainer board, digital oscilloscope, function generator and LTspice
- To show the difference of the measured cut-off frequency with calculated value.

Pre Lab

As more stages are added, the filter becomes able to better reject high frequency noise. When plotted on a Bode plot, the gain approaches two asymptotes: the low frequency gain approaches a constant gain of 0dB while the high-frequency gain drops as 20N dB/decade where N is the number of stages.

The single stage RC filter is a low pass filter, low frequencies are passed (have a gain of one) while high frequencies are rejected (the gain goes to zero). This is a useful filter to remove noise from an original signal. Many types of signals are predominantly low-frequency in nature - meaning they change slowly. This includes measurements of temperature, pressure, volume, position, speed, etc. Noise, however, tends to be at all frequencies, and is seen as the “fuzzy” line on your oscilloscope when you amplify the signal.

The trick when designing a low-pass filter is to select the RC time constant so that the gain is one over the frequency range of your signal (so it is passed unchanged) but zero outside this range (to reject the noise).

Second Order Low Pass Filter

Second order low-pass filter (LPF) is shown in Figure.1. It is called “second order" because it contains two resistors, and two capacitors. A 2^{nd}order LPF consists of a chain of two 1st order LPFs.

One problem with adding stages to an RC filter is that each following stage 'loads' on the previous stage. This loading bleeds some current from the previous capacitor, changing the circuit output voltage. If you make the impedance of each stage 10 times the previous stage, this loading is less than 10%, meaning that the transfer function will be close to that from lab 8. The value of R2 has purposely been chosen to be 10 times larger than R1 and the value of C2 has purposely been chosen to be 10 times smaller than R1.

It also means that every RiCi is the same, which makes the situation computationally easy. Every time you add on an RC into chain, you should continue the pattern of increasing R by a factor of10 and decreasing C by a factor of 10. This rule of thumb is a very good one to keep in mind.

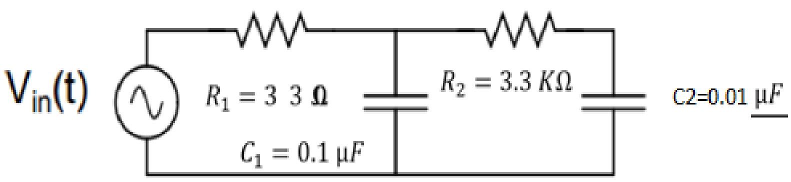


Figure 12- 1:Second Order RC Low Pass Filter.

Third Order Low Pass Filter

Third order low-pass filter (LPF) is shown in Figure. 2. It is called “third order" because it contains three resistors, and three capacitors. A third order LPF consists of a chain of three first order LPFs.

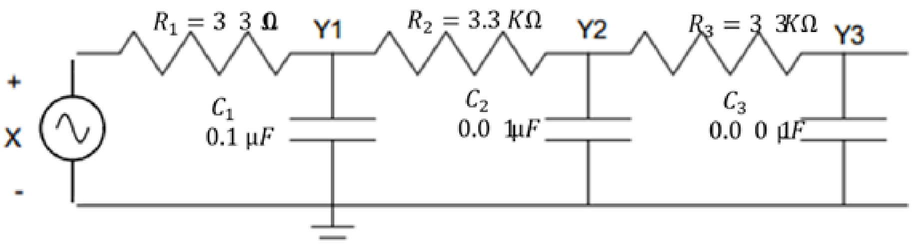


Figure 12- 2:Third Order RC Low Pass Filter.

The gain of this filter is approximately the gain of each stage analyzed separately (i.e., the loading effects are ignored).

$$Y_1 = \left(\frac{1}{jwR_1C_1 + 1}\right)X$$

$$Y_2 = \left(\frac{1}{jwR_2C_2 + 1}\right)Y_1$$

$$Y_3 = \left(\frac{1}{jwR_3C_3 + 1}\right)Y_2$$

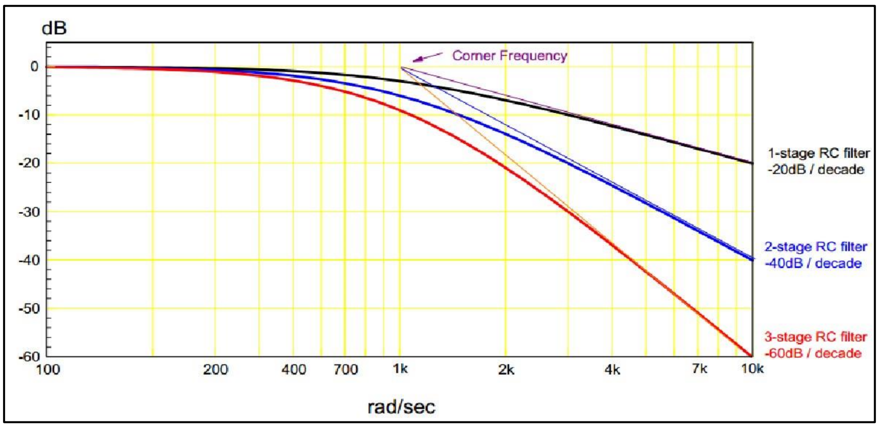


Figure 12- 3:Third order low pass filter graph.

Pre Lab-Exercise

Design a first order low pass filter for frequency of 4.8KHz

In Lab Task

Task 1: Second order RC Low Pass Filter

1. Construct the circuit of low pass filter in Figure 1. Use supply voltage of V_{in}=10 V
2. Connect channels 1 and 2 of the oscilloscope to measure and simultaneously.
3. Vary the frequency from 100 Hz 100KHz and record the indicated values in Table 1. For each frequency change, make sure that Vin is still $10V_{pp}$.
4. Using Table 1, sketch a magnitude and phase response of the filter’s output voltage.

Table 1 Second order RC low pass filter

Frequency(KHz)	$\omega\ (rad/s)$	$\lvert H(\omega)\rvert = \frac{1}{\sqrt{(1)^2 + \left(\frac{\omega}{\omega_c}\right)^2}}$	$H(\omega) = 20log\frac{V_o}{V_{in}}\ (dB)$	$\phi = -\tan^{-1}\left(\frac{\omega}{\omega_c}\right)$
$0.01f_c$=				
$0.1f_c$ =				
$0.5f_c$ =				
1 f_c =				
$2f_c$=				
$4f_c$ =				
$6f_c$ =				
$8f_c$ =				
$10f_c$=				
$100f_c$ =				

Task 2: Third Order RC Low Pass Filter

1. Construct the circuit of low pass filter in Figure 2. Use supply voltage of V_{in}=10 V_{pp}
2. Connect channels 1 and 2 of the oscilloscope to measure V_{in} and V_{out} simultaneously.
3. Vary the frequency from 100 Hz to 100KHz and record the indicated values in Table 2. For each frequency change, make sure that V_{in} is still $10V_{pp}$.
4. Using Table 2, sketch a magnitude and phase response of the filter's output voltage.

Table 2 Third order RC low pass filter

Frequency(KHz)	$\omega\ (rad/s)$	$\lvert H(\omega)\rvert = \frac{1}{\sqrt{(1)^2 + \left(\frac{\omega}{\omega_c}\right)^2}}$	$H(\omega) = 20log\frac{V_o}{V_{in}}\ (dB)$	$\phi = -\tan^{-1}\left(\frac{\omega}{\omega_c}\right)$
0.01fc=				
$0.1f_C$ =				
0.5fc =				
fc =				
2fc =				
4fc =				
6fc =				
8fc =				
10fc =				
100fc =				

Post Lab Task

1. From your understanding, explain why third order has the better efficiency than the other first order and second order filters.
2. Simulate the second and third order RC low pass filter using LT Spice and compare the results with hardware observations.

Rubric for Lab Assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed assigned tasks without any help from the instructor and showed the results appropriately.	4	
Good	The student completed assigned tasks with minimal help from the instructor and showed the results appropriately.	3	
Average	The student could not complete all assigned tasks and showed partial results.	2	
Worst	The student did not complete assigned tasks.	1	

Instructor Signature: ___________________________________ Date: ________________________