Ahsanullah University of Science and Technology

Department of Electrical and Electronic Engineering

LABORATORY MANUAL
FOR
ELECTRICAL AND ELECTRONIC SESSIONAL COURSE

Student Name :

Student ID :

Course no : EEE 1202
Course Title : ELECTRICAL CIRCUIT-II LAB

For the students of
Department of Electrical and Electronic Engineering
1st Year, 2nd Semester

Experiment No.	Name of the Experiments
01	Fundamentals of Ac circuit - Familiarization with Wave shape and Measurement of R.M.S. Value, Frequency and Phase Difference
02	Determination of the Bandwidth and Quality Factor of a Series RLC Circuit
03	Study of Resonance Behavior of a Series RLC Circuit with Variable Capacitance
04	Study of Resonance Behavior of a Parallel RLC Circuit with Variable Capacitance
05	Measurement of AC Power Using Wattmeter
06	Study of a 3-Φ System with Balanced Load
07	Determination of Phase Sequence of a 3-Φ System
08	Study of RC Filter Characteristics
09	Open Ended Lab

FUNDAMENTALS OF AC CIRCUIT - FAMILIARIZATION WITH WAVESHAPE AND MEASUREMENT OF R.M.S. VALUE, FREQUENCY AND PHASE DIFFERENCE.

PART A : AC Circuit Fundamentals (Hardware)

OBJECTIVE

In this experiment, we will observe voltage waveshapes across different ac circuit elements, such as resistor, inductor and capacitor and establish the relationships between these waveshapes. We will also measure different parameters associated with these wave shapes, such as peak value, peak-to-peak value, r.m.s value, time period, frequency, and phase difference. Finally, we will make a comparison between the theoretical values and the experimental results.

INTRODUCTION

Any periodic variation of current or voltage where the current (or voltage), when measured along any particular direction goes positive as well as negative, is defined to be an AC quantity. Sinusoidal AC wave shapes are the ones where the variation (current or voltage) is a sine function of time.
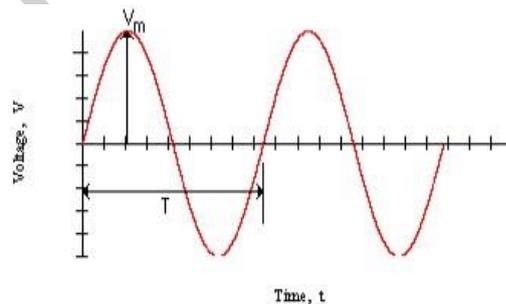

Figure 1: AC voltage waveform

Here, Time period = T, Frequency $f = \dfrac{1}{T}$

$v(t) = V_m \sin(2\pi f\ t)$, V_m or V_p = peak value = $\dfrac{V_{p-p}}{2}$

Effective value

The general equation of rms value of any function (voltage, current or any other physical quantity for which rms calculation is meaningful) is given by the equation

$V = \sqrt{\dfrac{1}{T}\int_0^T v^2\, dt}$.

Now, for sinusoidal functions, using the above equation we get the rms value by dividing the peak value (V_m) by square root of 2. That is

$$V = \sqrt{\frac{1}{T}\int_0^T (V_m \sin(2\pi f.t))^2 \, dt}$$

$$= \sqrt{\frac{1}{2\pi}\int_0^{2\pi} (V_m \sin\theta)^2 \, d\theta} = \frac{V_m}{\sqrt{2}}$$

Similarly, for currents, $I = \dfrac{I_m}{\sqrt{2}}$. These rms values can be used directly for power calculation. The formula for

average power is given by $P_{avg} = \dfrac{1}{T}\int_0^T (vi)dt$. And for sinusoids this leads to $P_{avg} = VI\cos(\theta)$. Here, V and I are rms

values and θ is the phase angle between voltage wave and current wave. The phase angle is explained in the next section.

Phase Angle

Phase difference between two ac sinusoidal waveforms is the difference in electrical angle between two identical points of the two waves. In figure 2, the voltage and current equations are given as:

$$v = V_m Sin(2\pi f\, t)$$
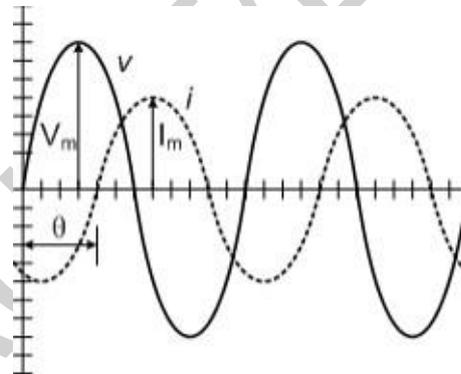$$i = I_m Sin(2\pi f.t - \theta)$$

Figure 2: Phase difference between a voltage and a current waveform.

Impedance

For, ac circuit analysis, impedance plays the same role as resistance plays in dc circuit analysis. It, can be stated fairly safely that, the concept of impedance is the most important thing, that makes the ac analysis, so much popular to the engineers. As you will see in your latter courses, any other periodic forms of time varying voltages or currents, are converted into an equivalent series consisting of sines and cosines (much like any function can be expanded by the power series of the independent variable using the Taylor series), only because the analysis of sinusoidal voltages are very much simple due to the impedance technique.

What is the impedance anyway? Putting it simply, it is just the ratio of rms voltage across the device to the rms current through it. That is

$$Z = \frac{V}{I\angle\theta} = \frac{V_m}{I_m\angle\theta}. \text{ Its unit is ohms}$$
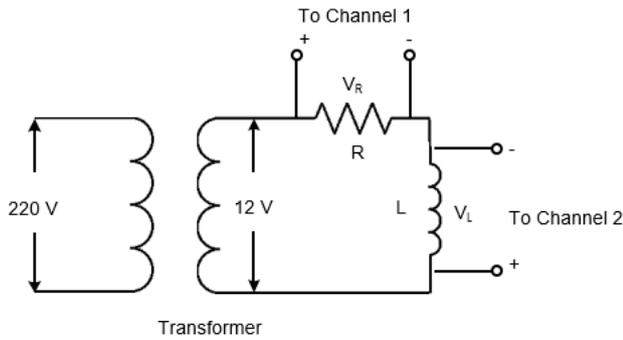
CIRCUIT DIAGRAM

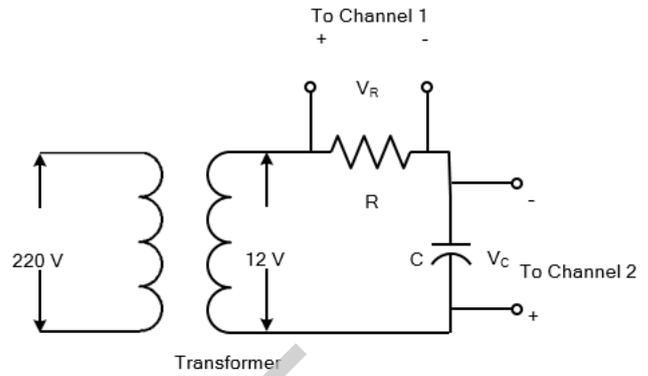
Figure 3: Experimental setup for R-L circuit

Figure 4: Experimental setup for R-C circuit

APPARATUS

- Dual-beam Oscilloscope
- Digital Multimeter
- Bread Board
- Transformer: 220V/12V - 1 Piece
- Resistor: 100Ω - 1 Piece
- Inductor-1 piece
- Capacitor: 100μF - 1 piece

PROCEDURE

1. Make the circuit connection for the R-L circuit, (Fig.3). Connect the oscilloscope probes properly as shown.
2. Observe the voltage waveshapes obtained in different modes of the oscilloscope. Channel 1 shows the voltage waveform across the resistance (V_R), channel 2 shows the voltage waveform across the inductor (V_L), and dual mode shows the above two waves simultaneously. Note that V_R is in phase with the current flowing in the circuit.
3. Sketch the waveshapes approximately in dual mode on a graph paper. For each waveform, indicate the volts/division and time/division used.
4. Measure the peak-to-peak (p-p) values for both voltage wave shapes. Also measure the time period T and the phase difference Δt (in seconds) between the two voltage wave shapes. Note down the lagging/leading voltage. Record these readings in Table 1.
5. Now calculate the peak and r.m.s. (root mean square) values for both voltage wave shapes. Also calculate the frequency and the phase difference θ (in degrees) between the two waveshapes. The phase difference θ is given by (Δt/T)×360 degree, where Δt is the time delay between the two waves.

6. Measure the r.m.s voltages by multimeter. Compare the results with that obtained from the oscilloscope. **Note:** When in ac mode, the multimeter gives the r.m.s. value of the voltage being measured, and when in dc mode, it shows the dc or average value (taken over a complete cycle) of the measured voltage. The above ac voltages are purely sinusoidal and have no dc value; you should verity it by measuring the voltages by a multimeter set in the dc mode.

7. Measure the input voltage with multimeter with AC voltage mode. The ratio between the voltage to the current gives the magnitude of the impedance Z.
8. Make connections for the R-C circuit shown in Fig. 4 and repeat steps 2 to 7 for the R-C circuit. Record the readings in Table

Table 1 Data for R-L circuit

p-p Values		Peak Values		r.m.s. values				T	f	Phase Difference		
				From peak values		From Multimeter				Δt	θ	Lagging / Leading Voltage
V_R	V_L	V_R	V_L	V_R	V_L	V_R	V_L				In deg	

Table 2 Data for R-C circuit

p-p Values		Peak Values		r.m.s. values				T	f	Phase Difference		
				From peak values		From Multimeter				Δt	θ	Lagging / Leading Voltage
V_R	V_C	V_R	V_C	V_R	V_C	V_R	V_C				In deg	

REPORT

1. Sketch neat diagrams of different waveshapes on graph paper for both the R-L and R-C circuits.
2. What will happen if the oscilloscope probes of Fig.3 are connected as shown in Fig.5? Explain.
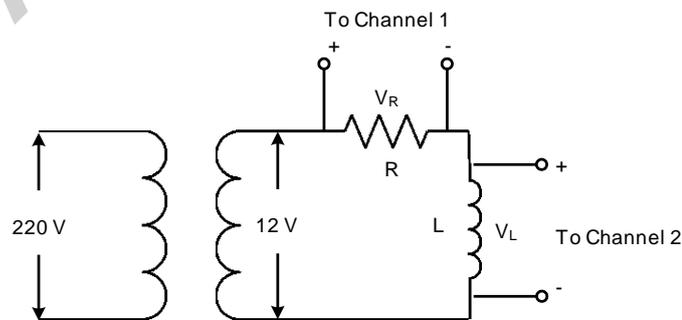
Figure 5

3. To correct the circuit of Fig. 5, the probes arc connected as shown in Fig.3 (with the ground probes at the same node). In this case, we do not obtain the actual waveshapes. Why? What measure should he taken in the oscilloscope to obtain the actual wave shapes?
4. Calculate the magnitude and phase angle of the circuit impedance from the readings taken in step 5, 6 and 7.
5. Can the oscilloscope be used directly to observe current waveshapes?

PART B : AC ANALYSIS (Simulation)

Using AC sweep, *PSpice* can perform AC analysis of a circuit for a single frequency or over a range of frequencies in increments that can vary linearly, by decade, or by octave. In AC sweep, one or more sources are swept over a range of frequencies while the voltages and currents of the circuit are calculated. Thus, we use AC sweep both for phasor analysis and for frequency response analysis: it will output Bode gain and phase plots. (Keep in mind that a phasor is a complex quantity with real and imaginary parts or with magnitude and phase.) While transient analysis is done in the time domain, AC analysis is performed in the frequency domain. For example, if $vs = 10\cos(377t + 40^0)$ transient analysis can be used to display v as a function of time, whereas AC sweep will give the magnitude as 10 and phase as 40^0. To perform AC sweep requires taking three steps similar to those for transient analysis:

          (1) Drawing the circuit
          (2) Providing specifications
          (3) Simulating the circuit

1. Drawing the Circuit

We first draw the circuit using *Schematics* and specify the source(s). Sources used in AC sweep are AC sources **VAC** and **IAC**. The sources and attributes are entered into the *Schematics* as stated in the previous section. For each independent source, we must specify its **magnitude and phase**.

2. Providing Specifications

Before simulating the circuit, we need to add some specifications for AC sweep. For example, suppose we want a linear sweep at frequencies 50, 100, and 150 Hz. We enter these parameters as follows:

1. Select **Analysis/Setup/AC Sweep** to open up the dialog box for AC Sweep.
2. **CLICKL** *Linear* for the *X* axis to have a linear scale.
3. Type 3 in the *Total Pts* box.
4. Type 50 in the *Start Freq* box.
5. Type 150 in the *End Freq* box.
6. **CLICKL OK/Close** to accept specifications.

A linear sweep implies that simulation points are spread uniformly between the starting and ending frequencies. **Note that the *Start Freq* cannot be zero because 0 Hz corresponds to DC analysis**. If we want the simulation to be done at a single frequency, we enter 1 in step 3 and the same frequency in steps 4 and 5. If we want the AC sweep to simulate the circuit from 1 Hz to 10 MHz at 10 points per decade, we **CLICKL** on *Decade* in step 2 to make the *X* axis logarithmic, enter 10 in the *Total Pts* box in step 3, enter 1 in the *Start Freq* box, and enter 10Meg in the *End Freq* box. Keep in mind that a decade is a factor of 10. In this case, a decade is from 1 Hz to 10 Hz, from 10 Hz to 100, from 100 to 1 kHz, and so forth.

3. Simulating the Circuit

After providing the necessary specifications and saving the circuit, we perform the AC sweep by selecting **Analysis/Simulate**. If no errors are encountered, the circuit is simulated. At the end of the simulation, the system displays **AC analysis finished** and creates an output file with extension .out. Also, the *Orcad PSpice* program will automatically run if there are no errors. The frequency axis (or *X* axis) is drawn but no curves are shown yet. Select **Trace/Add** from the *Orcad Pspice* menu bar and click on the variables to be displayed. We may also use current or voltage markers to display the traces as explained in the previous section. To use advanced markers such as *vdb*, *idb*, *vphase*, *iphase*, *vreal*, and *ireal*, select **Markers/Mark Advanced**. In case the resolution of the trace is not good enough, we may need to check the data points to see if they are enough. To do so, select **Tools/Options/Mark Data Points/OK** in the *Orcad PSpice* menu and the data points will be displayed. If necessary, we can improve the resolution by increasing the value of the entry in the **Total Pts** box in the **Analysis/Setup/AC Sweep and Noise Analysis** dialog box for AC sweep.

Bode plots are separate plots of magnitude and phase versus frequency. To obtain Bode plots, it is common to use an AC source, say V1, with 1 volt magnitude and zero phase. After we have selected **Analysis/Simulate** and have the *Orcad PSpice* program running, we can display the magnitude and phase plots as mentioned above. Suppose we want to display a Bode magnitude plot of V(4). We select **Trace/Add** and type **dB(V(4))** in **Trace Command** box. dB(V(4)) is equivalent to 20log(V(4)), and because the magnitude of V1 or V(R1:1) is unity, dB(V(4)) actually corresponds to dB(V(4)/V(R1:1)), which is the gain. Adding the trace dB(V(4)) will give a Bode magnitude/gain plot with the axis in Y dB.

Once a plot is obtained in the *Orcad PSpice* window, we can add labels to it for documentation purposes. To add a title to the plot, select **Edit/Modify Title** in the *Orcad PSpice* menu and type the title in the dialog box. To add a label to the *Y* axis, select **Plot/Y Axis Settings**, type the label, and **CLICKL OK**. Add a label to the *X* axis in the same manner.

As an alternative approach, we can avoid running the *Orcad PSpice* program by using *pseudocomponents* to send results to the output file. Pseudocomponents are like parts that can be inserted into a schematic as if they were circuit elements, but they do not correspond to circuit elements. We can add them to the circuit for specifying initial conditions or for output control. In fact, we have already used two pseudocomponents, VIEWPOINT and IPROBE, for DC analysis. Other important pseudocomponents and their usage are shown in Fig. 1 and listed in Table. The pseudo components are added to the schematic. To add a pseudocomponent, select **Draw/Get New Parts** in the *Schematics* window, select the pseudocomponent, place it at the desired location, and add the appropriate attributes as usual. Once the pseudocomponents are added to the schematic, we select **Analysis/Setup/AC Sweep** and enter the specifications for the AC sweep, and finally, select **Analysis/Simulate** to perform AC sweep. If no errors are encountered, the voltages and currents specified in the pseudocomponents will be saved in the output file. We obtain the output file by selecting **Analysis/Examine Output** in the *Schematics* window.

Figure 1: Print and plot pseudocomponents.

| Symbol	Description
IPLOT	Plot showing branch current; symbol must be placed in series IPRINT
	Table showing branch current; symbol must be placed in series VPLOT1
	Plot showing voltages at the node to which the symbol is connected
VPLOT2	Plot showing voltage differentials between two points to which the symbol is connected
VPRINT1	Table showing voltages at the node to which the symbol is connected VPRINT2
	Table showing voltage differentials between two points to which the symbol is Connected

EXAMPLE 1: Find current *i* in the circuit in Fig. 2.
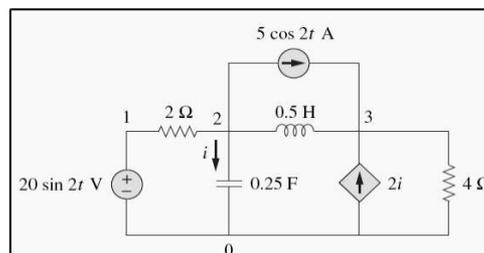
Figure: 2

Solution:
Recall that **20 sin 2t = 20 cos(2t — 90°)** and that $f = \omega/2\pi = 2/2\pi = 0.31831$. The schematic is shown in Fig. 3. The attributes of V1 are set as *ACMAG* = 20, *ACPHASE* = —90; while the attributes of IAC are set as AC = 5. The current-controlled current source is connected in such a way as to conform with the original circuit in Fig.2; its gain is set equal to 2. The attributes of the pseudocomponent IPRINT are set as *AC* = yes, *MAG* = yes, *PHASE* = ok, *REAL* =, and *IMAG* =. Since this is a single-frequency ac analysis, we select **Analysis/ Setup/AC Sweep** and enter *Total Pts* = 1, *Start Freq* = 0.31831, and *Final Freq* = 0.31831. We save the circuit and select **Analysis/Simulate for simulation**. The output file includes

FREQ	IM(V_PRINT3)	IP(V_PRINT3)
3.183E-01	7.906E+00	4.349E+01

From the output file, we obtain $I = 7.906/43.49°$ A or $i(t) = 7.906 \cos(2t + 43.49°)$ A. This example is for a single-frequency ac analysis; Example 3 is for AC sweep over a range of frequencies.
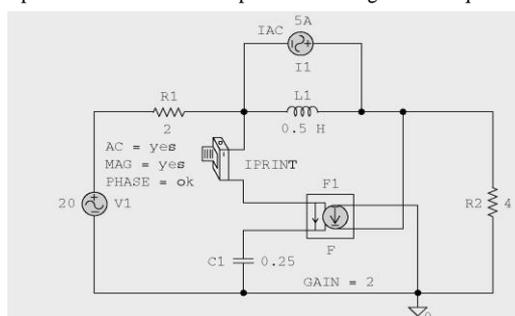

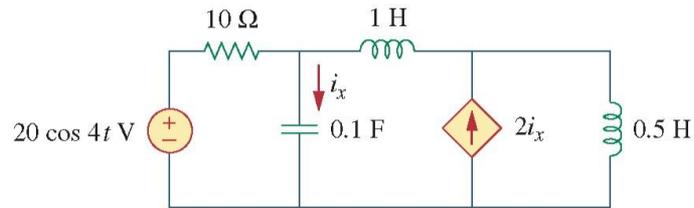
Fig: 3

Example 2: Find $i_x(t)$ in the circuit in Fig.3.

Figure :4

Answer: From the output file, $I_x = 7.59\underline{/108.43°}$ or $i_x = 7.59 \cos(4t + 108.43°)$ A.

For the *RC* circuit shown in Fig.4, obtain the magnitude plot of the output voltage v_o for frequencies from 1 Hz to 10 kHz. Let $R = 1$ kΩ and $C = 4$ **uF**.

Solution:
The schematic is shown in Fig.5. We assume that the magnitude of V1 is 1 and its phase is zero; we enter these as the attributes of V1. The schematic of the circuit in Fig.6.
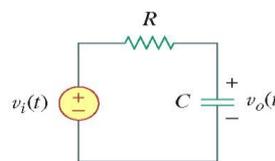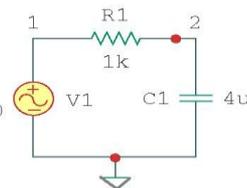

Figure.5   Figure. 6

We also assume 10 points per decade. For the AC sweep specifications, we select **Analysis/Setup/AC Sweep** and enter 10 in the *Total Pts* box, 1 in the *Start Freq* box, and 10k in the *Final Freq* box. After saving the circuit, we select **Analysis/Simulate**. From the *Orcad PSpice* menu, we obtain the plot in Fig. 7(a) by selecting **Traces/Add** and clicking V(2). Also, by selecting **Trace/Add** and typing dB(V(2)) in the *Trace Command* box, we obtain the Bode plot in Fig. D.7(b). The two plots in Fig.7 indicate that the circuit is a lowpass filter: low frequencies are passed while high frequencies are blocked by the circuit.
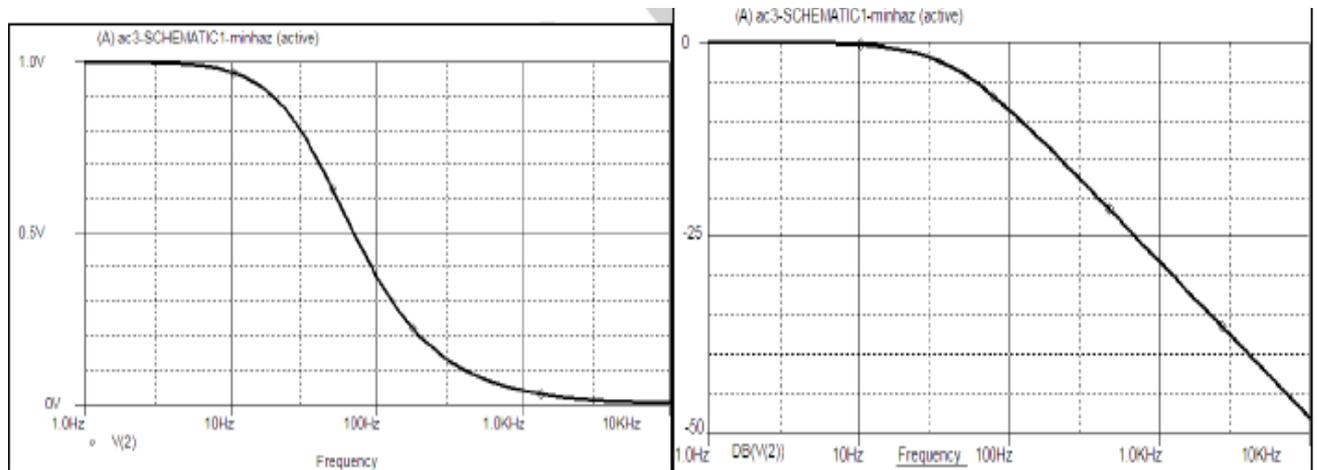

Figure 7

Result of Example 2: (a) linear, (b) Bode plot.

Practice Problem: For the circuit in Fig. 5, replace the capacitor C with an inductor $L = 4$ mH and obtain the magnitude plot (both linear and Bode) for v_o for $10 < f < 100$ MHz.

Answer: See the plots in Fig.8.
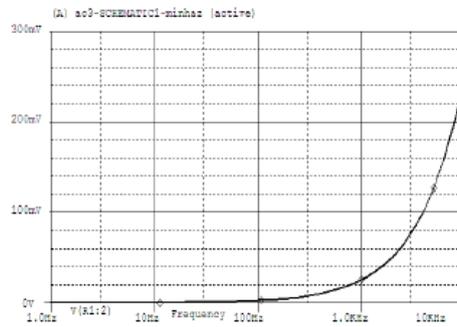
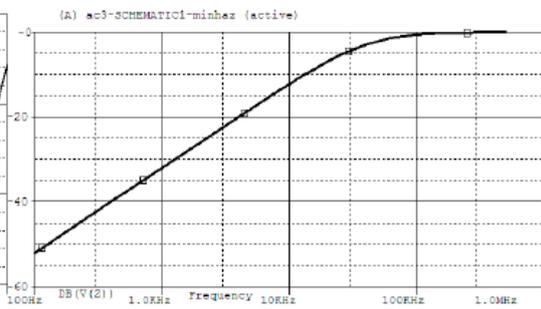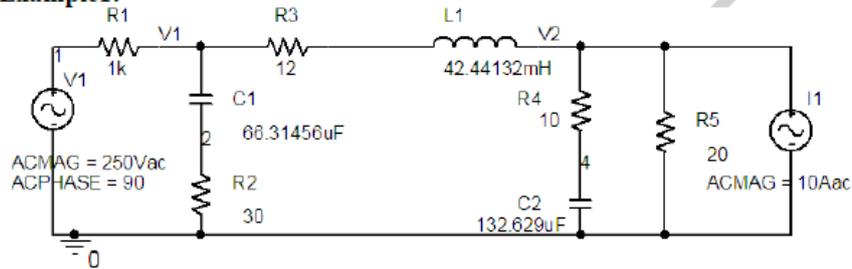Fig: 8(a)   Fig: 8(b)

Result of Practice Prob.: (a) linear plot, (b) Bode plot.

Example1:

(a) Use node-voltage analysis to find the magnitude and phase angle of the node voltages V1 and V2, then find magnitude and phase angle of current in R3.

$$\begin{bmatrix} 0.043 - j0.024 & -0.03 + j0.04 \\ -0.03 + j0.04 & 0.1 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} j0.25 \\ 10 \end{bmatrix}$$

Transforming the voltage source to the current source and writing the node-voltage equation results in

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} - \begin{bmatrix} 20 & 6 - j8 \\ 6 - j8 & 8.6 - j8.4 \end{bmatrix} \begin{bmatrix} j0.25 \\ 10 \end{bmatrix} - \begin{bmatrix} 96.0469\angle 51.34° \\ 99.5301\angle -27.85° \end{bmatrix} \quad \text{Volts}$$

or Current in R3 is

$$I = \frac{96.0469\angle -51.34° - 99.5301\angle -27.85°}{12 + j16} = 1.9977\angle 172.3769° \quad \text{A}$$

(b) The voltage source and the current source frequency are 60 Hz. Use PSpice and AC Analysis for a single point 60 Hz to obtain the node voltages V1 and V2, and current in R3. Use VPRINT, and IPRINT to send the node voltages and current to the output file. In the VPRINT and IPRINT dialog box set the attributes AC, MAG, and PHASE to **yes.**

10

DETERMINATION OF THE *BANDWIDTH* AND *QUALITY FACTOR* OF A SERIES RLC CIRCUIT

**PART A : Series Resonance (Hardware)**

INTRODUCTION

Resonance is a particular situation that may occur in an electric circuit containing both inductive and capacitive elements. In a series RLC circuit this resonance occurs when,

$$Inductive\ Impedance\ =\ Capacitive\ Impedance$$
$$\Rightarrow\ X_L\ =\ X_C$$
$$\Rightarrow\ 2\pi f L\ =\ 1/2\pi f C \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots(Equation\ 1)$$

Thus it is possible to have resonance by varying the frequency 'f' while keeping 'L' and 'C' constant. For a series circuit we should observe the followings while we vary 'f' in order to have resonance.

- The frequency at which the resonance occur is known as resonant frequency f_r. At this frequency the current is maximum and the voltage drop across 'R' is also maximum. Beyond this frequency the current as well as the voltage drop, V_R give rise to a profile shown below:
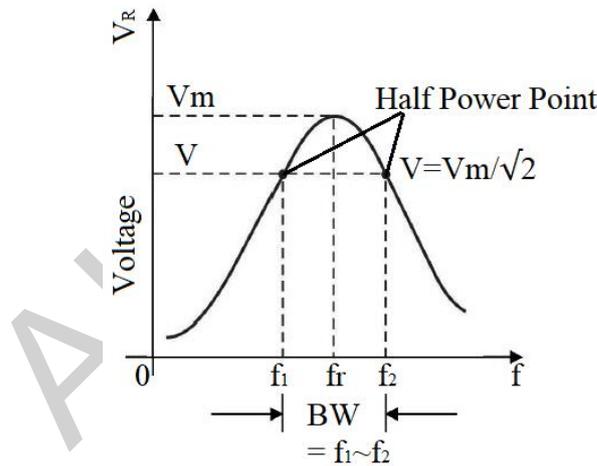
- From the above figure, notice that the maximum value of voltage (representing the maximum current value) occurs at resonant frequency, fr.
- The Half power point corresponds to the points at which voltage is equal to $V_m/\sqrt{2} = 0.707\ V_m$.
- The frequencies (f_1 and f_2) corresponding to the half power points represents the cut-off frequencies of the circuit.
- The Bandwidth of the circuit is defined as BW = $f_1 \sim f_2$

In our experiment we shall try to find the frequency response of the series RLC circuit and find its bandwidth. We shall conduct the following studies on quality factor of the series circuit.

- Selectivity of a coil (a resistance and pure inductance) or series RLC circuit is represented by the Quality Factor 'Q'. The quality factor is defined as:
  Q = center frequency/Bandwidth = f_0 / ($f_1 \sim f_2$) ....(Equation 2)
  Here the center frequency f_0 = fr
- From the definition of Q, it is clear that for a particular fr the smaller the value of ($f_1 \sim f_2$) the larger the value of Q. Hence larger value of Q represents sharp peak in the resonance profile signifying the circuit/coil is more selective to the particular frequency fr.
- For a series circuit, it can be shown that Q = $X_L/R = X_C/R$ ....(Equation 3)
  At resonance, Q = $Ir \times X_C/Ir \times R = V_C/V_S$ ....(Equation 4)
  Similarly it can be shown that, Q = V_L/V_S ....(Equation 5)

11

Since for most of the coil, Q>1 then we get V_L>V_S and V_C>V_S (Typical value of Q of a coil is 100). We shall try to match our findings using equations 2, 3, 4 and 5.

CIRCUIT DIAGRAM
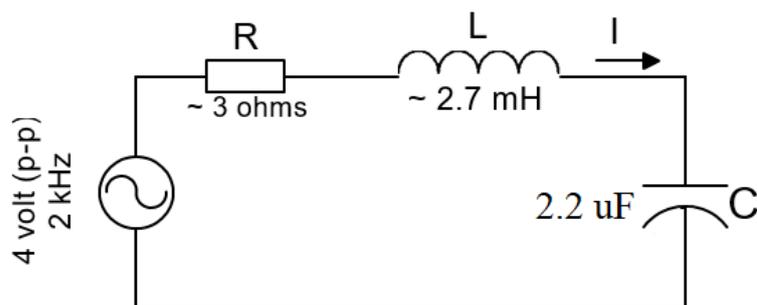
APPARATUS

1. Resistance : 2.2 Ω   (or any in the range of 2 ~ 4 Ω) ……….1 *pc*
2. Inductor   : 2.7 mH (typical resistance 8 Ω)……………….1 *pc*
3. Capacitor : 0.1uf (104K), 0.22uf, 1uf, 2.2uf, 4.7uf, 10uf…….1 *pc* each
4. Oscilloscope
5. Oscilloscope Probe …………………..………………………..2 *pc*
6. Signal Generator ………………….………………………1 *pc*
7. Signal Generator Probe …………….………………………1 *pc*
8. Multi-meter …………………………………………………1 *pc*
9. Bread Board …………………..………………………………1 *pc*
10. Wires for connection

DATA SHEET

Step : 1
Measure and note the following values:

R = [   ] R_L = [   ]   L = [   ]

Step : 2
Setup the circuit

Step : 3
Adjust the signal generator at 4 volt (p-p) & 1 2 0 0 Hz (*f*). Adjust the voltage gain after connecting to the circuit.

Step : 4
Measure / Determine the following quantities and fill up the table below :

1. Measure V_R using multi-meter in AC voltage mode.
2. Measure V_L & V_C using multi-meter.

Repeat step 4 for each of the frequencies shown in the table.
Note : Adjust the amplitude of source each time to 4 volt (p-p).

Table :

No	f (Hz)	Vin (V)	V_R (mV)	V_L (V)	V_C (V)
1					
2					
3					
4					
5		0.707			
6					
7					
8					
9					
10					

REPORT

Plot the V_R vs frequency and fill in the boxes below:

1. Maximum value of V_R = [＿＿＿＿＿] Corresponding frequency, fr = [＿＿＿＿＿]

2. Values of V_R at half power point V = [＿＿＿＿＿]

3. Values of cut off frequencies, f_1 = [＿＿＿＿＿] and f_2 = [＿＿＿＿＿]

4. Calculated Bandwidth = [＿＿＿＿＿]

5. Calculated Quality Factor, Q = [＿＿＿＿＿] = [＿＿＿＿＿]

6. Verify equation 3, 4 and 5 with the value of Q calculated in step 5.

PART B : Series Resonance (Simulation)

Example 1:

For the series RLC circuit shown, use PSpice, and AC Analysis to sweep the frequency from 10Hz to 100KHz over a decade variation with 200 points per decade.
(a) Use Probe to plot the current magnitude and phase angle. Determine the resonant frequency. Also draw the 0.707(V0(max)) line and estimate the circuit bandwidth b, and the Q-factor.

(b) On a separate graph plot the real and reactive power delivered to the circuit as a function of frequency.

$$\cos(VP(1) * pi/180 - IP(R1) * pi/180))$$
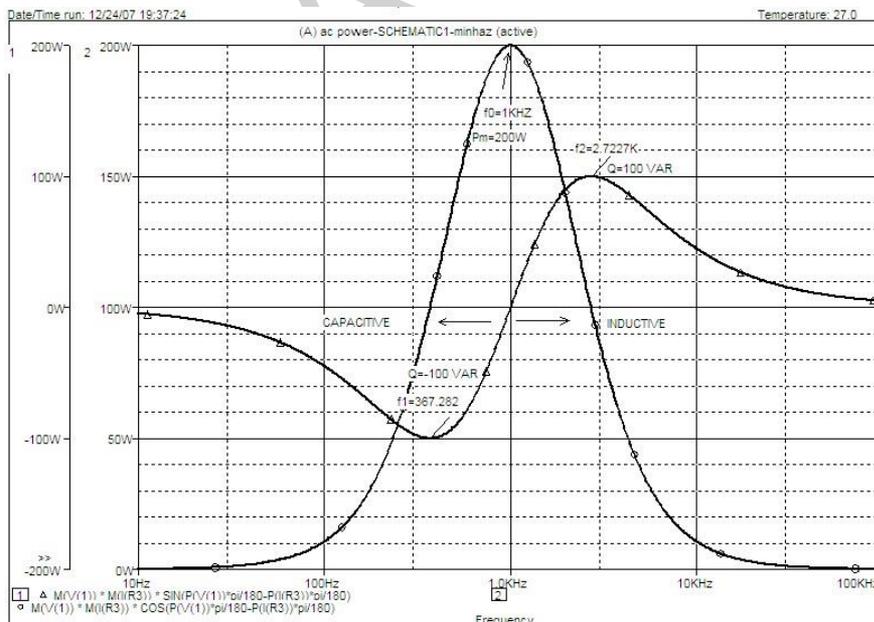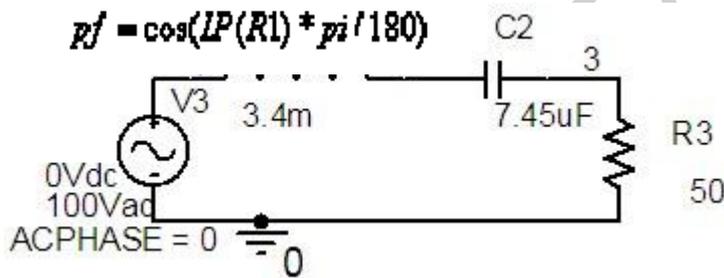
$$P = V(1) * I(R1) \cos(VP(1) * pi/180 - IP(R1) * pi/180))$$
$$Q = V(1) * I(R1) \sin(VP(1) * pi/180 - IP(R1) * pi/180))$$

(c) On a separate graph, plot the power factor as a function of frequency.

$$pf = \cos(IP(R1) * pi/180)$$

Hints : M(V(3))* M(I(V3))*SIN(P(V(3))*pi/180 -P(I(V3))*pi/180)

= Magnitude(voltage) * Magnitude (current) * SIN(Phase Differernce)

Date/Time run: 12/14/07 00:36:36   Temperature: 27.0

(A) ac power-SCHEMATIC1-minhaz (active)

1.0

UNITY POWER FACTOR

0.5

EADING POWER FACTOR   LAGGING POWER FACTOR

0

10Hz   100Hz   1.0KHz   10KHz   100KHz

□ COS(P(I(R3))*pi/180)

Frequency

Date/Time run: 12/14/07 00:36:36   Temperature: 27.0

(A) ac power-SCHEMATIC1-minhaz (active)

90 DEGREE

LEADING POWER FACTOR   LAGGING POWER FACTOR

UNITY POWER FACTOR   0 DEGREE

-90 DEGREE

0Hz   100Hz   1.0KHz   10KHz   100KHz

1 □ P(I(R3))*pi/180

Frequency

Example 2:
(i)   Determine the real power in the circuit shown.
(ii) Use PSpice and Transient Analysis to obtain a plot of the instantaneous voltage vl(t), i(t),
and the instantaneous power p(t)

STUDY OF RESONANCE BEHAVIOUR OF A SERIES RLC CIRCUIT WITH VARIABLE CAPACITANCE

INTRODUCTION

Resonance is a particular situation that may occur in an electric circuit containing both inductive and capacitive elements. In a series RLC circuit this resonance occurs when,

$$Inductive\ Impedance\ =\ Capacitive\ Impedance$$

$$=>\ X_L\ =\ X_C$$

$$=>\ 2\pi f L\ =\ 1/2\pi f C \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots(Equation\ 1)$$

At resonant condition for a series circuit the following should be observed :
1. The resultant impedance of the circuit is purely resistive in nature :
$$Z_T = R\ +\ j\,(\,X_L\ -\ X_C\,) = R\ +\ j\,.\,0\ = R$$
2. Since the impedance is minimum the current in the series circuit will be maximum and given as :
$$I_{max}\ =\ V_S\,/\,R$$
3. The circuit yields an unity power factor since the impedance is purely resistive
4. The average power absorbed by the circuit is maximum at this resonance ; $P_{max}\ =\ I_{max}{}^2\ R$

How can resonance be achieved?
We can obtain the resonance situation by satisfying equation 1 for the series RLC circuit. Our options are to vary any of the three parameters – L,C or f or any combination of them. In this experiment we shall try to achieve resonance point by varying 'C' keeping 'L' and 'f' constant.

Practical Considerations to be taken for the experiment :

- Practical inductor has some resistance of its own. So make sure that this resistance R_L doesn't hamper your calculation significantly.
- It is always better to predict or calculate the probable resonant point in advance using the formula $C = 1/(\,4\,\pi 2 f L\,)$.
- Due to loading effect the amplitude of the source voltage changes when we use different capacitors. So one should check the amplitude every time a new capacitor is used and make adjustment if necessary. Remember that we should keep same amplitude value throughout the experiment.
- Since at resonance point the circuit current is maximum , the watt dissipation of the resistor R must be checked against its wattage rating. It should be applicable also for the inductor current rating. Besides, the signal generator we are using must be capable of delivering the maximum current.
- In a series circuit containing R, L and C the voltage across the C can raise beyond the supply voltage. So other circuit equipment those are subjected to V_C must be able to withstand that voltage.
- When measuring the phase angle between voltage and current we should check the connection of the oscilloscope probes carefully. Each of the black crocodile clip represents the ground of respective oscilloscope channel. But inside the oscilloscope these two grounds are shorted. So when we use

both the probes there is a great chance that we short-circuit some parts of the circuit. We should always avoid this situation.

CIRCUIT DIAGRAM
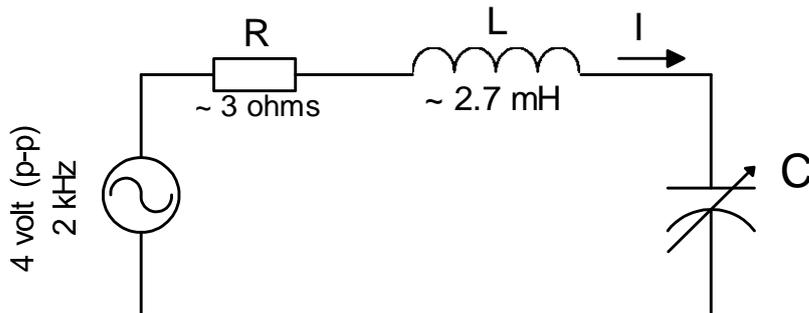
APPARATUS

1.   Resistance : 2.2 Ω   (or any in the range of 2 ~ 4 Ω) ………………1 *pc*
2.   Inductor   : 2.7 mH (typical resistance 8 Ω)…………………………..1 *pc*
3.   Capacitor : 0.1uf (104K), 0.22uf, 1uf, 2.2uf, 4.7uf, 10uf.……………1 *pc* each
4.   Oscilloscope
5.   Oscilloscope Probe ………………...………………………………2 *pc*
6.   Signal Generator …………………………………………….…..1 *pc*
7.   Signal Generator Probe ……………………………………………1 *pc*
8.   Multi-meter …………………………………………………………1 *pc*
9.   Bread Board …………………………..……………………………1 *pc*
10. Wires for connection

PROCEDURE

Step : 1

Measure and note the following values :

R = [   ] R_L = [   ]   L = [   ]

Step : 2
Setup the circuit with C = 0.1 µf

Step : 3
Adjust the signal generator at 4 volt (p-p) & 2 kHz (f).
Calculate the value of C for which resonance is expected : $C = 1 / (4 \pi^2 f L) = $ [   ]

Step : 4

Measure / Determine the following quantities and fill up the table below :

1. Find I : Measure V_R using multi-meter in AC voltage mode. Then I = V_R / R.
2. Measure V_L & V_C using multi-meter.
3. Measure the phase difference θ between Vs and I using Oscilloscope in dual mode (connect the oscilloscope probes as shown below). Note down the relative lag/lead position of the voltage and current wave.
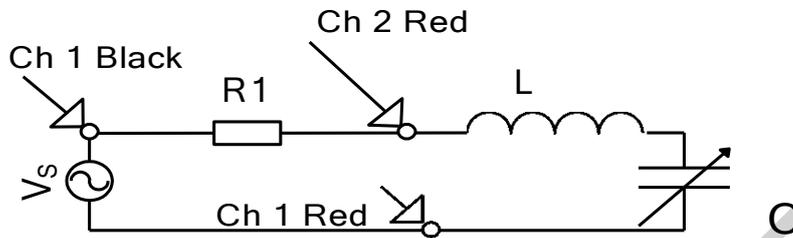
Step : 5

Repeat step 4 for each of the capacitors.

Note : Adjust the amplitude of source each time to 4 volt (p-p).

Table :

No	C (μf)	V_R (mV)	I (mA)	V_L (volt)	V_C (volt)	θ (deg.)	Power Factor	Lag/Lead
1	0.1							
2	0.22							
3	1.0							
4	1.5 (2.2 + 4.7)							
5	2.2							
6	3.2 (4.7+ 10)							
7	4.7							
8	6.9 (4.7 ∥ 2.2)							
9	10							

REPORT

Plot the following curves:

    1. I versus C
    2. P.f. versus C
    3. V_L and V_C versus C

- Plot #1
  1. Maximum Current Value I_{max} =
  2. Resonance Point   C   =
  3. Mark the resonance point in the graph.

- Plot #2
  1. Mark the resonance point in the graph.
  2. Mark the lagging p.f. region on the graph.
  3. Mark the leading p.f region graph.

- Plot #3
  1. Mark resonance point on the graph.
 2. Observe the peak of V_L & V_C with respect to resonance point.
  3. Note that $(V_C)_{max}$ > supply voltage.

- Draw the vector diagram at resonance.

STUDY OF RESONANCE BEHAVIOUR OF A PARALLEL RLC CIRCUIT WITH VARIABLE CAPACITANCE.

PART A : Parallel Resonance (Hardware)

INTRODUCTION

Resonance is a particular situation that occurs in an electric circuit when the resultant impedance is resistive. For the parallel RLC circuit, resonance occurs when susceptance of the inductive branch is equal to the susceptance of the capacitive branch, i.e.

$$b_L = b_C$$

In this experiment, we will vary C to obtain the resonance condition.

At resonant condition for a parallel circuit the following should be observed:

1. The resultant impedance of the circuit is purely resistive in nature and is maximum. Since the impedance is maximum the current from the source is minimum at resonance.

2. The circuit yields unity power factor since the impedance is purely resistive.

CIRCUIT DIAGRAM
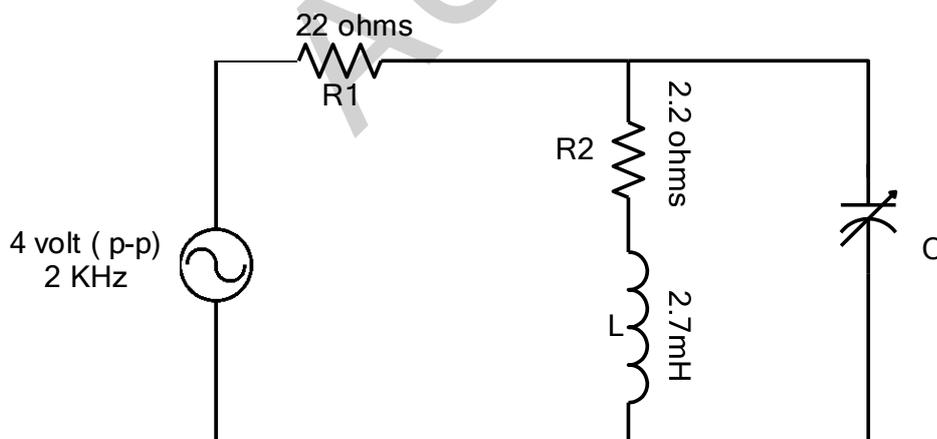
Figure 1

APPARATUS

1. Resistance : 2.2 Ω , 22 Ω   ……………………………………………1 *pc* each
2. Inductor   : 2.7 mH   (typical resistance 8 Ω)………………………..1 *pc*
3. Capacitor   : 0.1uf (104K), 0.22uf, 1uf, 2.2uf, 4.7uf, 10uf.……………1 *pc* each
4. Oscilloscope
5. Oscilloscope Probe …...…………………………………………….2 *pc*
6. Signal Generator ………………………………………………..…1 *pc*
7. Signal Generator Probe ………………………………………………1 *pc*
8. Multi-meter ………………..…………………………………………1 *pc*
9. Bread Board ……………………………………………………………1 *pc*
10. Wires for connection

PROCEDURE

Step : 1

Measure and note the following values :

R₁ = [   ]   R₂ = [   ]   R_L = [   ]   L = [   ]

Step : 2
Setup the circuit with C = 0.1 µf
Step : 3
Adjust the signal generator at 4 volt (p-p) & 2 kHz (*f*).

Step : 4
Measure / determine the following quantities and fill up the table below:

1. Find I : Measure V_{R1} , V_{R2} using multi-meter in AC voltage mode. Then I = V_{R1} / R1 & I_L = V_{R2} / R2.
2. Measure V_L & V_C using multi-meter. Then I_C = V_C / X_C ; $X_C = 1/2\pi fC$.
3. Measure the phase angle θ of the current wave using Oscilloscope in dual mode (connect the oscilloscope probes as shown below). Note down the relative lag/lead position of the voltage and current wave.
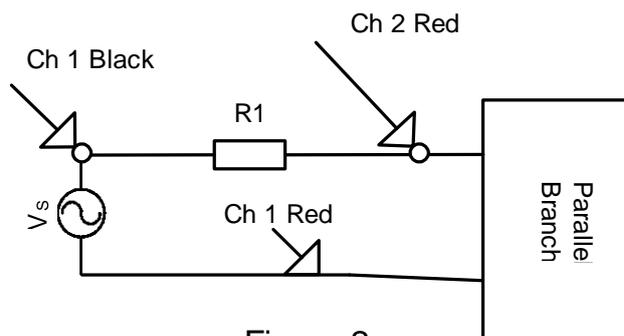
Figure 2

Step : 5
Repeat step 4 for each of the capacitors.
Note : Adjust the amplitude of source each time to 4 volt (p-p).

Table :

C (μf)	V$_{R1}$ (mV)	I (mA)	V$_{R2}$ (mV)	I$_L$ (mA)	V$_L$ (volt)	V$_C$ (volt)	I$_C$ (mA)	θ deg	*Pf*	Lag / Lead
0.1										
0.22										
1.0										
1.5 (2.2 + 4.7)										
2.2										
3.2 (4.7+ 10)										
4.7										
6.9 (4.7 ∥ 2.2)										
10										

REPORT

- Plot the following curves :

  a.   Plot # 1   I vs. C

    1. Minimum Current Value I $_{min}$ =
    2. Resonance Point   C   =
    3. Mark the resonance point in the graph.

  b.   Plot # 2   power factor vs. C

    1. Mark the resonance point in the graph.
    2. Resonance Point   C   =
    3. Mark the lagging p.f. region on the graph.
    4. Mark the leading p.f region graph.

- Draw the vector diagram of all currents at resonance.

PART B : Parallel Resonance (Simulation)

Example

For the parallel RLC circuit shown, use PSpice and Probe to plot voltage magnitude and phase angle over a range of 30 to 300 Hz in increment of 1 Hz (Total point =271). Determine the resonant frequency. Also draw the 0.707 V0(max) line and estimate the circuit bandwidth and the Q-factor.
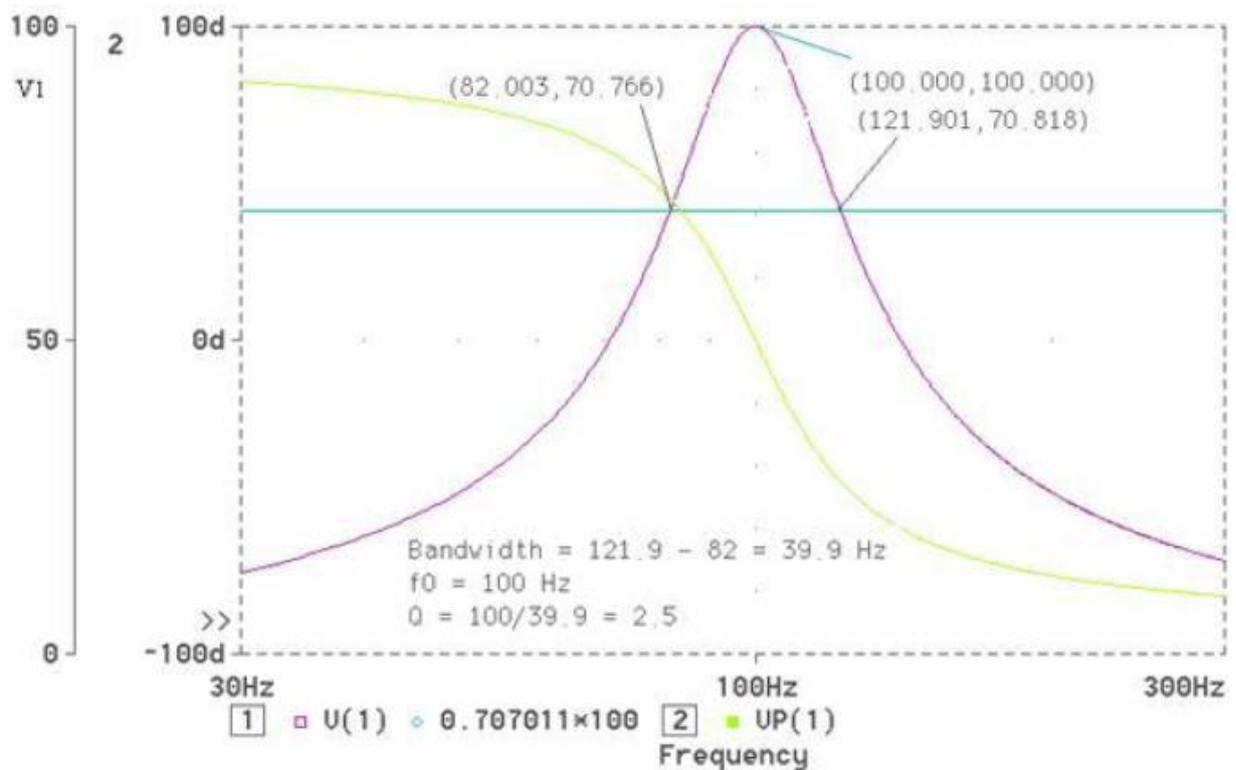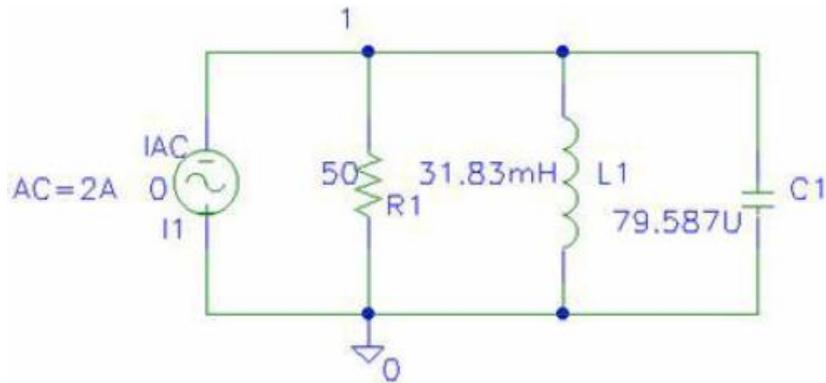
Exercise:

As instructed by teacher.

MEASUREMENT OF AC POWER USING WATTMETER

Part A: AC Power measurement using wattmeter (Hardware)

OBJECTIVE

The objective of this experiment is to learn the use of wattmeter for measuring power in ac circuits.

APPARATUS

1. Lamp board (2)
2. Ballast (1)
3. Multimeter (1)
4. AC ammeter (1)
5. Wattmeter (1)
6. Connecting wires

Circuit Diagram
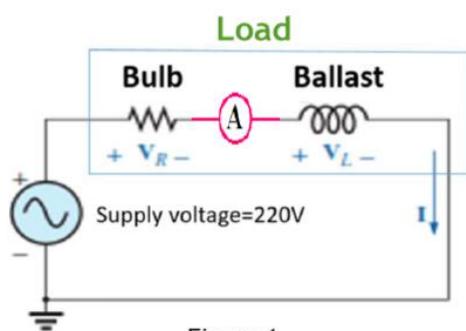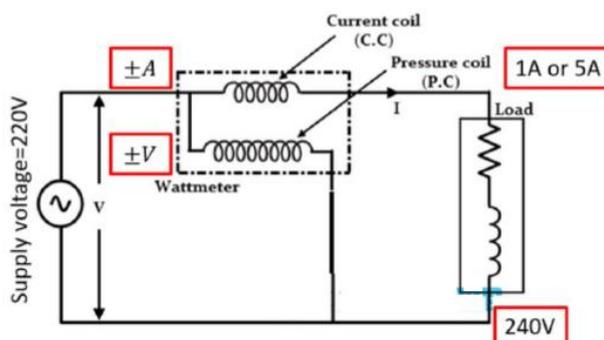
Figure 1

Figure 2

PROCEDURE

1. Connect the circuit in Figure 1. Measure supply voltage, V and I.

    Average Power/Real Power = $VI\cos\theta$,

    Here, $\tan\theta = X_L/R = IX_L/IR = V_L/V_R$ and $\theta = \tan^{-1}(V_L/V_R)$

2. Connect the Multimeter across the Lamp and Ballast and calculate V_L and V_R keeping the meter in ac mode.

3. Calculate the power from step 1.

4. Connect the wattmeter as shown and measure the power W.

5. Repeat steps 2, 3 and 4 for a different combination of lamp.
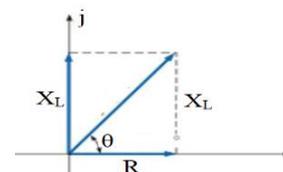
REPORT

1. Compare/relate the wattmeter reading, W and real power observed from first circuit. Give your comments.

2. Use your results to find out of individual circuit components and also the power factors of each circuit.

Part B: AC Power measurement using wattmeter (Simulation)

Example 01:
(i) Determine the real power in the circuit shown.
(ii) Use PSpice and Transient Analysis to obtain a plot of the instantaneous voltage v1(t), i(t), and the instantaneous power p(t).


VOFF = 0
PHASE = 0
FREQ = 60
VAMPL = 169.705

L1 28.715m

R3 6.25
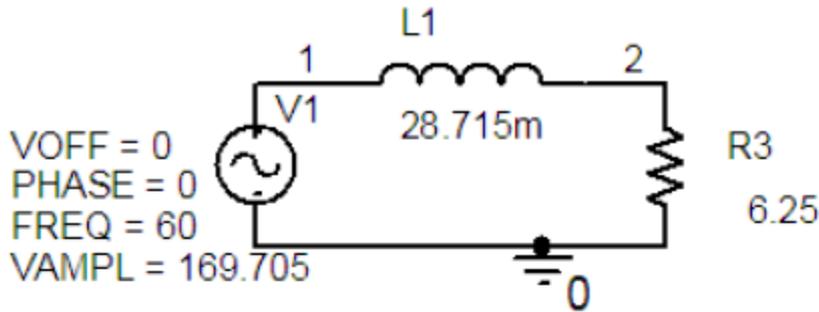
$v_1(t) = 169.705 \sin(2\pi 60 t)$ Volt

$Z = 6.25 + 2\pi 60 L = 12.5\angle 60 \ \Omega$

$I = \frac{120\angle 0}{12.5\angle 60} = 9.6\angle -60 \ A$

$P = VI \cos\theta = (120)(9.6)(\cos 60) = 576 \ W$

(ii) The Final Time in the Transient Analysis is set to T = 1/60 = 16.667ms. The following instantaneous quantities and the average power are plotted

25
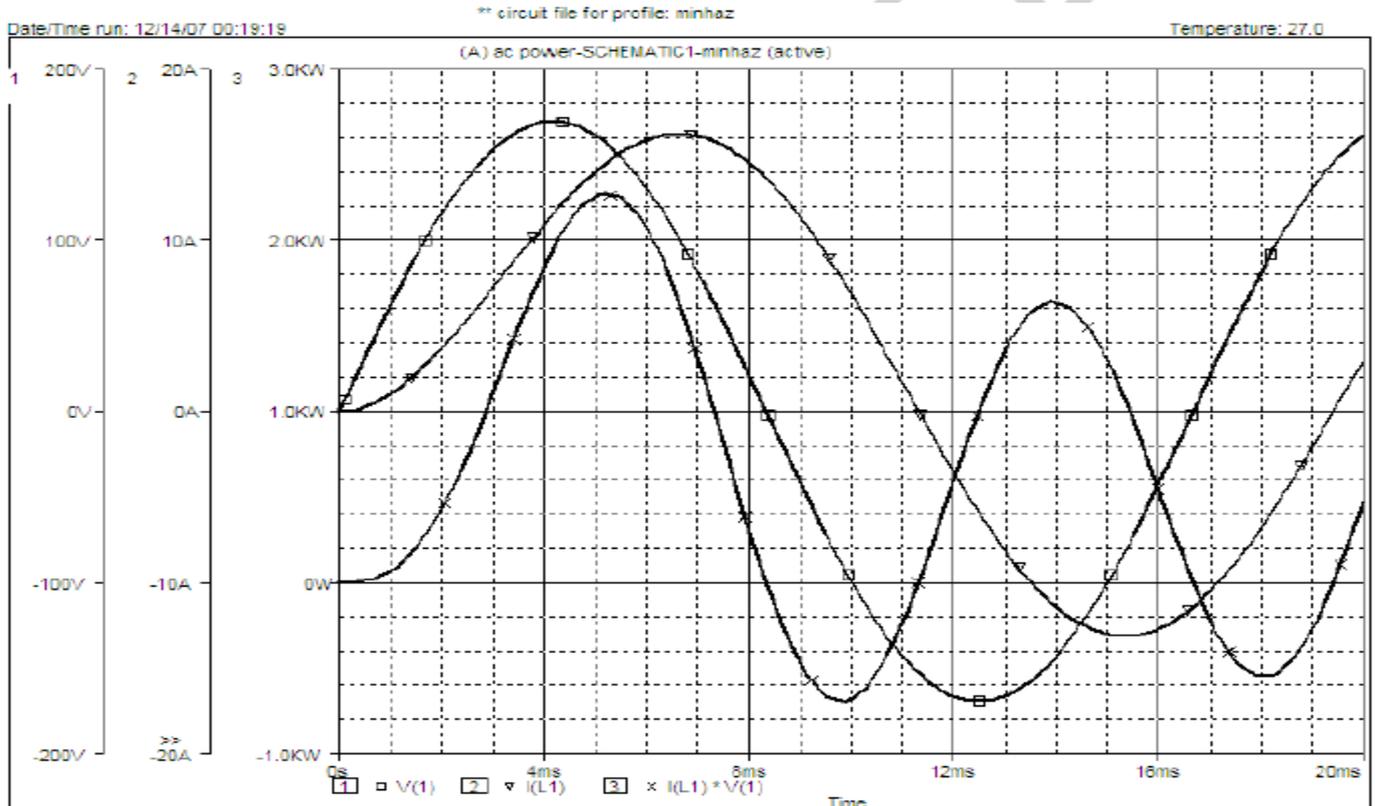
Example 02:

(i) Determine the real power in the circuit shown.

$$v_1(t) = 169.705 \sin(2\pi 60t) \quad \text{Volt}$$

(ii) Use PSpice and Transient Analysis to obtain a plot of the instantaneous voltage v1(t), i(t), and the instantaneous power p(t).

IC = -73.48469
C1
1   2
245.035u

VOFF = 0   V2   R1
FREQ = 60   18.75
PHASE = 0
VAMPL = 169.705   0
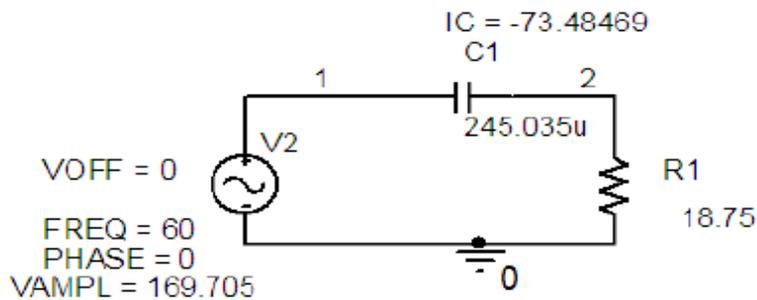
$$I = \frac{120\angle 0}{21.65\angle - 30} = 5.543\angle - 30 \quad A$$

$$Z = 18.75 - j\frac{1}{2\pi 60C} = 21.65\angle - 30\,\Omega$$

$$I = \frac{120\angle 0}{21.65\angle - 30} = 5.543\angle - 30 \quad A$$

$$P = VI\cos\theta = (120)(5.543)(\cos 30) = 576 \quad W$$

(ii) The Final Time in the Transient Analysis is set to T = 1/60 = 16.667ms. The following instantaneous quantities and the average power are plotted
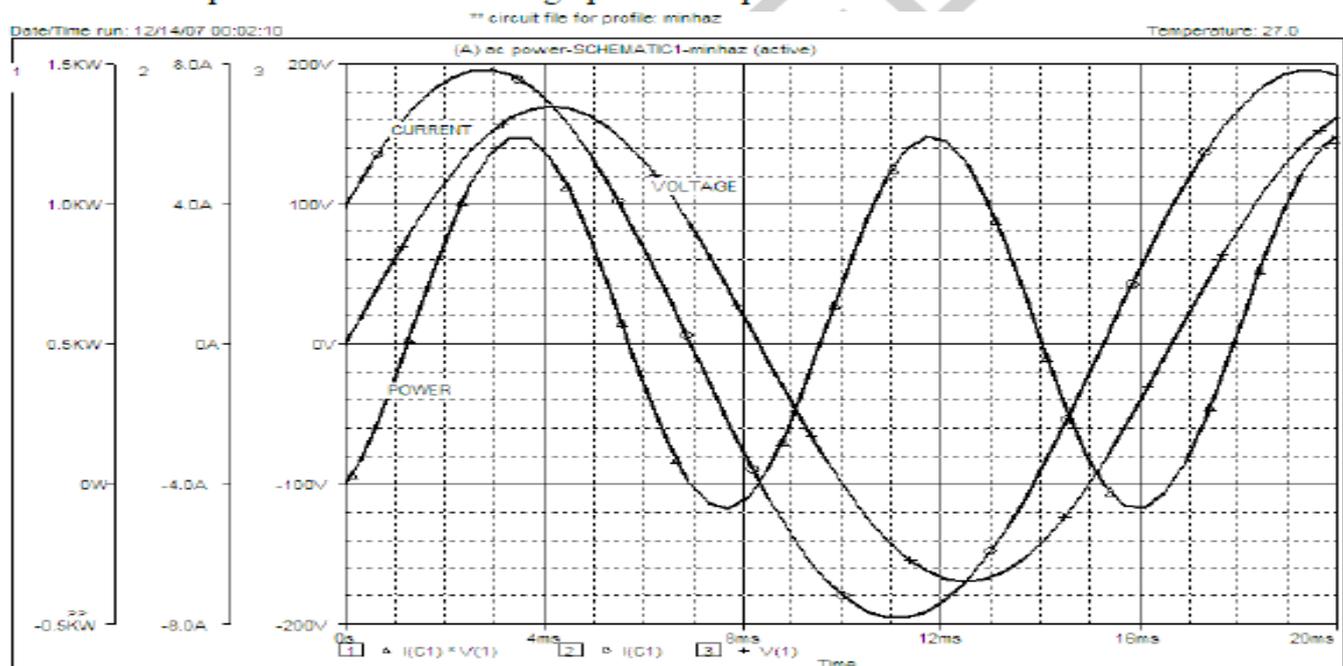
Exercice:
As instructed by the lab teacher.

STUDY OF A 3-Φ SYSTEM WITH BALANCED LOAD

THEORY

A three phase system can be viewed as a combination of three individual sin wave sources those are separated 120 degree in electrical phase. Our distribution system is of this kind where 3 wires are dedicated to three individual phases and the fourth one is the neutral. As for most of the appliances we use single phase 220 volt supply which is in fact one of the three phases those come from distribution lines. We call it the 'phase voltage' or 'line to neutral voltage'. Such a system is schematically drawn in the following figure together with the phasor diagram:
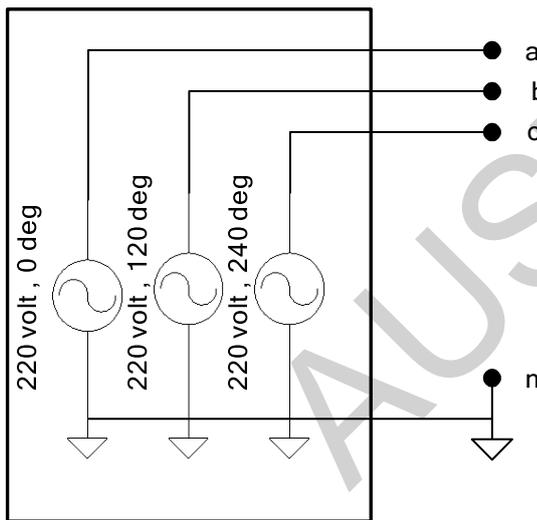
Figure : A 3 - phase system
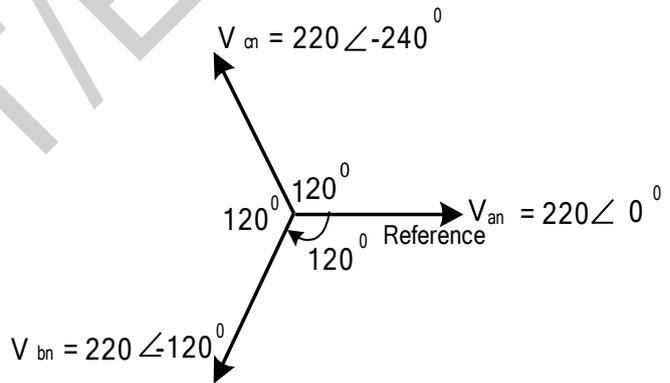
Figure : Phasor diagram of 3-phase system

Three phase system is also represented by the 'line-to-line voltage'. Consider the above system (with sequence 'a-b-c'). The line to line voltage V_{ab} can be expressed as :

$$V_{ab} = V_{an} + V_{nb} = V_{an} - V_{bn} \text{ ………………………………………. Eq. 1}$$

Here , the quantities should be treated as vector. As shown in the following figure the summation of 'V_{an}' and '-V_{bn}' gives 'V_{ab}'. Now it should be observed that 'V_{ab}' makes an angle of 30 degree with 'V_{an}' and its magnitude is $\sqrt{3} | V_{an} |$. In similar way we can determine the other two line-to-line voltages V_{bc} and V_{ca}. These three line to line voltages are also 120 degree separated. Thus we can define 3 phase system by line to line voltages and chose one of them as reference. The second figure shows the relation

of phase and line-to-line quantities taking V_{ab} as reference. Using this representation we can restate the sequence as 'ab-bc-ca'.
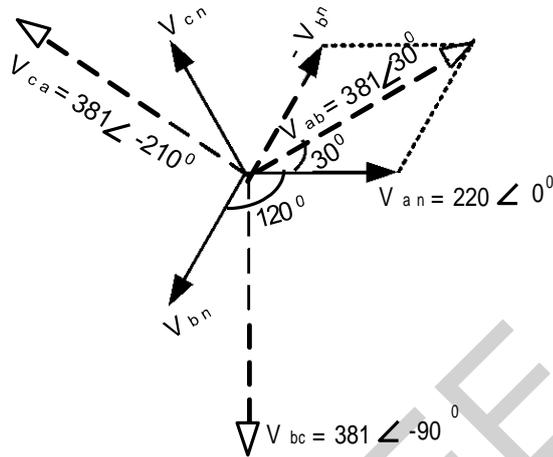

Figure: Line and phase voltages for a-b-c sequence.

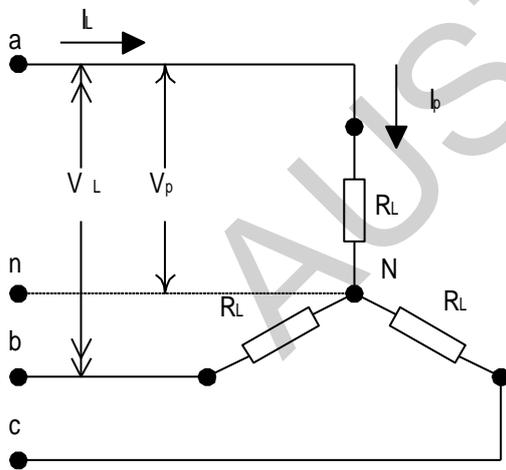## Balanced Y- connected and Δ - connected load :
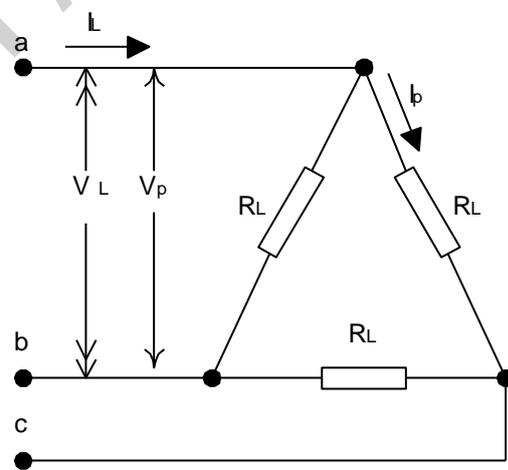

Fig.   Balanced Y- connected Load

Fig.   Balanced △- connected load

Here each of the loads R_L are equal. So the system is balanced.

The following are true for a balanced Y- connected load :

- Each of the loads are subjected to equal phase voltages ; generally 220 volt . The quantities differ only by phase angle.

- The vector summation of line-to-line voltages are zero :

  $V_{ab} + V_{bc} + V_{ca} = 0$

- The vector summation of the phase currents are zero :
  $I_{an} + I_{bn} + I_{cn} = 0$

  This implies that their should be no current flow from node 'n' and 'N'. So for a balanced system there is no need to use the neutral wire.

- The relation between the phase and line quantities are :

  Line current , I_L = phase current , I_P
  Line-to-Line voltage $V_L = \sqrt{3} \, V_P$

The following are true for a balanced Δ- connected load :

- Each of the loads are subjected to equal line-to-line voltages ; generally 381 volt . The quantities differ only by phase angle.

- The vector summation of line-to-line voltages are zero :
  $V_{ab} + V_{bc} + V_{ca} = 0$

- The vector summation of the phase currents are zero :
  $I_{ab} + I_{bc} + I_{ca} = 0$

- The relation between the phase and line quantities are :

  Line current , $I_L = \sqrt{3} \times$ phase current $= \sqrt{3} \, I_P$
  Line-to-Line voltage $V_L =$ phase voltage $= V_P$

Objective of our experiment :

We shall construct a balanced system consisting of Y connected load and Δ connected load . Then we shall measure the line and phase quantities in both cases. We shall verify the above theoretical facts with our findings. In this experiment we shall consider only for resistive loads.

APPARATUS

1. Lamp Holder : …………………………………………...……………6 *pc*
2. Bulb   : 60 Watt ………………………… . ……………………..3 *pc*
                    100 Watt ………………………… . ………………….....3 *pc*

3. Oscilloscope ….. …………………………………………….…..1 *pc*
4. Oscilloscope Probe …………………………………………….. 2 *pc*
5. Multi-meter …………………………………………………………1 *pc*

6. Wires for connection
7. Power Supply Cord ……………………………………………..1 *pc*

PROCEDURE

<u>**Step : 1**</u>

Connect the bulbs (60 Watt) in Y- connection as shown in the following figure :
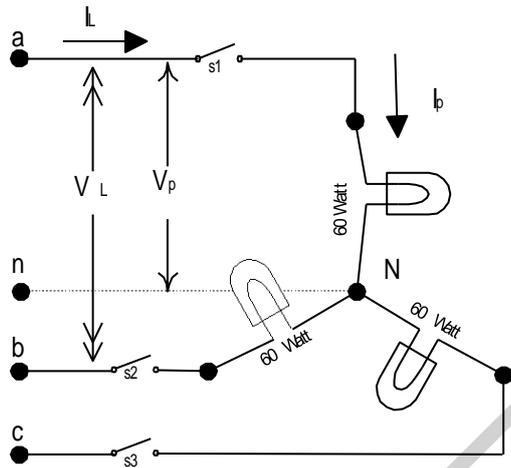

Fig. Balanced Y- connected Load

Measure and note the following values :

Line to Line Voltages   : V_{ab} = ……………. , V_{bc} = ………………. , V_{ca} = …………………

Line to Neutral Voltages : V_{an} = …………..… , V_{bn} = ………………. , V_{cn} = …………………

Phase Currents   : I_{an} = ……………. , I_{bn} = .………………. , I_{cn} = …………………

<u>**Step : 2**</u>

Connect the bulbs (60 Watt + 100 Watt) in Δ - connection as shown in the following figure :
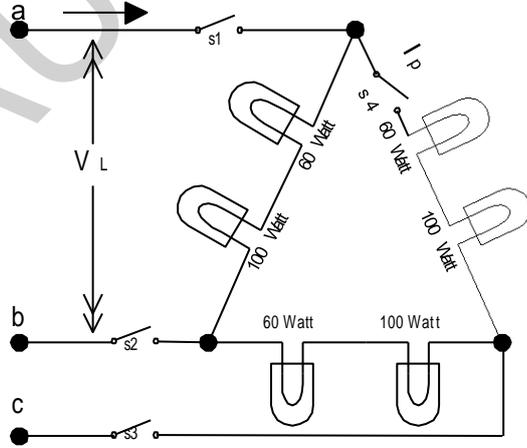
Fig. Balanced delta connected Load

Measure and note the following values :

Line to Line Voltages   : V_{ab} = ……………. , V_{bc} = ………………. , V_{ca} = …………………

Line Currents   : I_a = ……………. , I_b = ………………. , I_c = …………………

Phase Currents   : I_{ab} = ……………. , I_{bc} = ………………. , I_{ca} = …………………

Step : 3

Connect the bulbs (60 Watt + 100 Watt) in Δ - connection as shown in the following figure. In one of the branches connect a rheostat (100 ohms ; Set its resistance to 1~3 ohms) as shown.
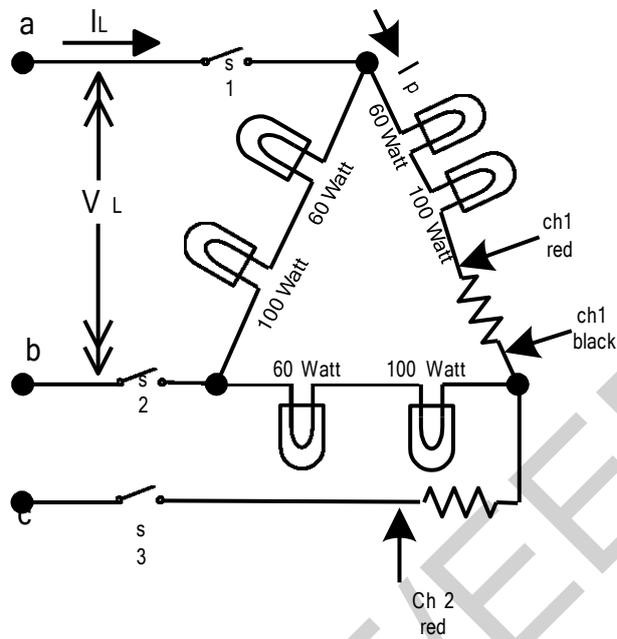
Observe the wave shapes of I_L and I_p . Take measurement of their phase difference: _____

Which wave is lagging   ? _____

REPORT

1. From data found in Step 1 & Step 2, draw the phasor diagrams showing the relations between phase and line quantities for Y- and Δ-load.

2. For a Balanced Y connected load the neutral wire usually not been used . --- why ?

DETERMINATION OF PHASE SEQUENCE OF A 3-Φ SYSTEM

THEORY

A three phase system can be viewed as a combination of three individual sin wave sources those are separated 120 degree in electrical phase. Such a system is schematically drawn in the following figure :
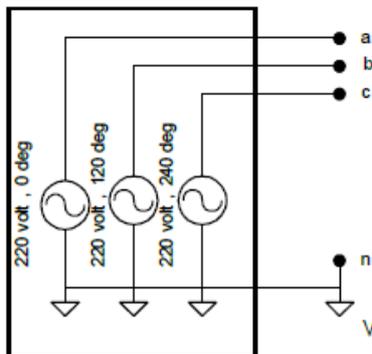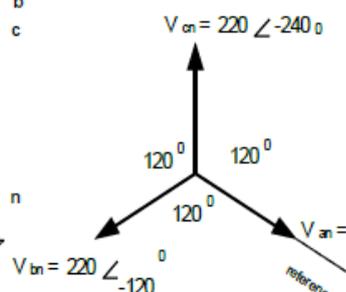

Fig : A 3 - phase system

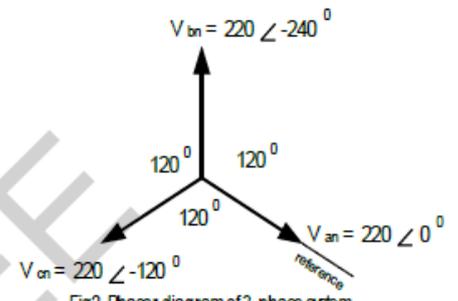Fig1. Phasor diagram of 3-phase system ('a-b-c') sequence

Fig2. Phasor diagram of 3-phase system ('a-c-b') sequence

As from the consumer side we see the four wires only. For our advantage we mark the three phase wires as 'a' , 'b' and 'c' (or sometimes as 'R' , 'Y' and 'B') and the neutral wire as 'n'. We take one of the wires as reference phase and then find the phases of the other two wires with respect to it. In the above 'Fig:1' we take phase 'a' as our reference. As shown in the figure phase 'b' leads phase 'a' by 120 deg and phase 'c' leads phase 'b' by another 120 degree. This sequence is called 'sequence a-b-c' . For three phase system another sequence is possible that is 'sequence a-c-b'. It that case, phase 'c' leads phase 'a' by 120 deg and phase 'b' leads phase 'c' by another 120 degree. The sequence is shown in 'Fig:2'.

Three phase system is also represented by the 'line-to-line voltage'. These three line to line voltages are also 120 degree separated. Using this representation we can restate the sequence 'a-b-c' as 'ab-bc-ca' and 'a-c-b' as 'ab-ca-bc'. The phasor diagram for these two sequences are shown below :
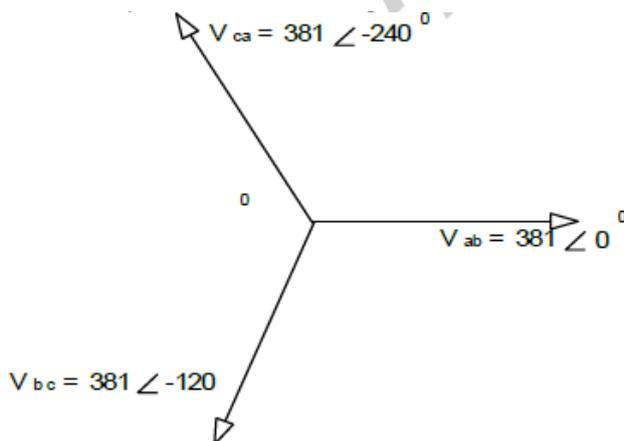

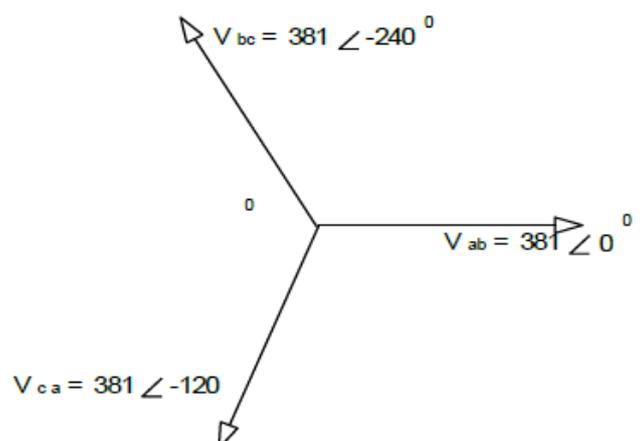Fig: 4 Line to Line voltages for 'ab-bc-ca' sequence

Fig:3 Line to Line voltages for 'ab-ca-bc' sequence

As mentioned earlier, from the consumer side we see the four wires only. For some 3 phase loads we need to know the phases sequences before we connect it to the supply. Taking one of the wires as reference we need to know which of the other two wires leads 120 degree from the reference wire (or which of the other two wires lags 240 degree from the reference wire). If we mark the reference wire as 'a' and find the wire that matches this criterion and mark this wire as 'b' and the remaining wire as 'c' then we say that we are working with 'a-b-c' sequence. It should be observed that when we say about a sequence as 'a-b-c' or 'a-c-b' we are referring to the three wires that we have already marked. Without this marking the phrase 'a-b-c sequence' bears no meaning.

Objective of our experiment :

We are given the three wires of a 3-phase system with neutral connection. We have no idea about the phase sequence of the three wires. We want to mark the three wires as 'a' , 'b' and 'c' so that they represent 'a-b-c' sequence (or we may want to mark the three wires as 'a', 'b' and 'c' so that they represent 'a-c-b' sequence) .

Methods of determining Phase Sequence :

Method 1 :

Let us connect an unbalanced load to the three wires as show in the figure :
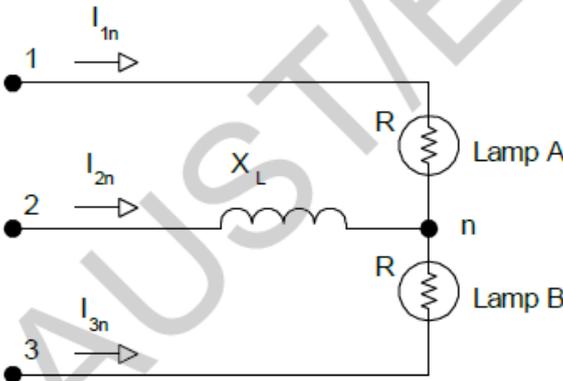
Now it can be shown that the voltage across the Lamp A is equal to ,

$$V_{Lamp\,A} = Z_{1n} \cdot \frac{V_{12}(Z_{1n} + Z_{3n}) + V_{23}Z_{2n}}{Z_{1n}(Z_{2n} + Z_{3n}) + Z_{3n}.Z_{2n}} \quad \ldots\ldots\ldots\ldots eq4$$

The voltage across the Lamp B is ,

$$V_{Lamp\,B} = V_{31} + Z_{1n}.I_{1n} \quad \ldots\ldots\ldots\ldots eq5$$

Here Z's are the impedances of respective phases. For our cases ,
$Z_{1n} = R_1$, the resistance of the bulb connected to line 'a'
$Z_{3n} = R_2$, the resistance of the bulb connected to line 'c'

$Z_{2n} = R_L + j X_L$, $\qquad$ R_L the inductor resistance and X_L reactance of the inductor .

Now, *let us assume* that according to the wire marks, *'1-2-3' represents the 'a-b-c' sequence*. Therefore line to line voltages are,

$$V_{12} = V \angle 0° \qquad\qquad V_{23} = V \angle -120° \qquad\qquad V_{31} = V \angle -240°$$

where 'V' is the rms value of the line to line voltages. If we calculate the voltage across 'Lamp A' and 'Lamp B' using equations 4 and 5 then we find that ,

$$V_{Lamp\ B} < V_{Lamp\ A} \qquad \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots.eq\ 6$$

Thus, Lamp A will glow brighter than Lamp B.

But if we had assumed that '1-2-3' represented the 'a-c-b' sequence then our voltage references would be,

$$V_{12} = V \angle 0° \qquad\qquad V_{23} = V \angle -240° \qquad\qquad V_{31} = V \angle -120°$$

Then we would find that,

$$V_{Lamp\ B} > V_{Lamp\ A} \qquad \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots.eq\ 7$$

In this case Lamp B would glow brighter than Lamp A.

In practice we first arbitrarily mark the three wires as '1' , '2' and '3'. Then we will connect the lamps and the inductor according to the figure. The lamp which glows brighter must be connected to line 'a' the other is connected to 'c' and the middle wire to 'b'. Thus we can mark the three wires as 'a', 'b' and 'c' that will give us the sequence of 'a-b-c'. After determining this sequence of the supply we could either connect a load in 'a-b-c' sequence or in 'a-c-b' sequence as necessary.

Method 2 :

Let us connect an unbalanced load to the three wires as show in the figure :
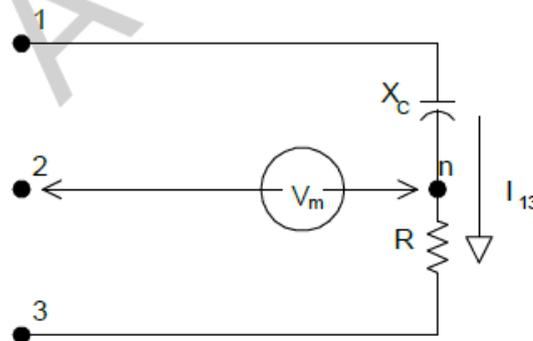
For this setup it can be shown that , for 'a-b-c' sequence ,

$\qquad$ $V_m >$ line-to-line voltage

And for 'a-c-b' sequence ,

$\qquad$ $V_m <$ line-to-line voltage

This result can be realized from the phasor diagram shown below :
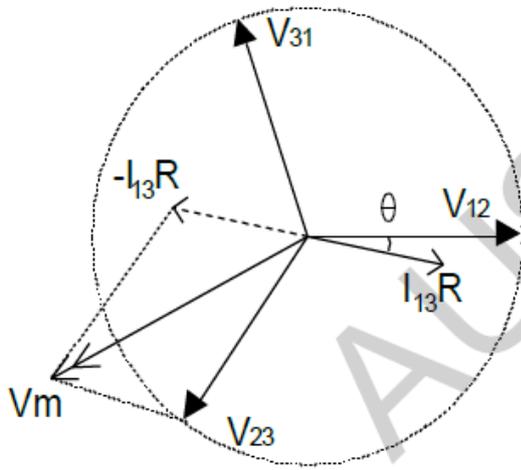
$Z_T = R - j X_C$; R = the resistance used in phase connected to line '3'
                    X_C = impedance of the capacitor connected to line '1'

$$I_{13} = \frac{V_{13}}{Z_T} = \frac{-V_{31}}{R - j X_C} = |I| \angle \theta \quad \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots eq\ 8$$

$$V_m = V_{23} - I_{13}.\ R \quad \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots eq\ 9$$

For 'a-b-c' sequence :

$V_{12} = V \angle 0°$
$V_{23} = V \angle -120°$
$V_{31} = V \angle -240°$

For 'a-c-b' sequence :

$V_{12} = V \angle 0°$
$V_{23} = V \angle -240°$
$V_{31} = V \angle -120°$

Vm > Line-to-Line voltage
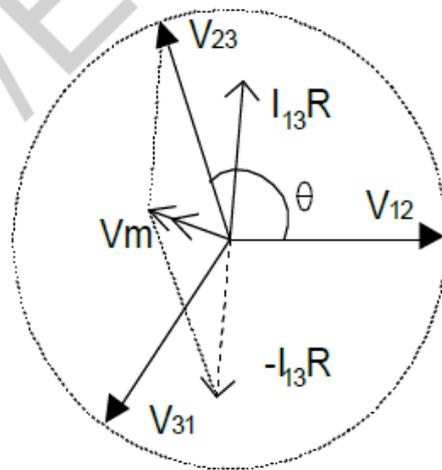
Vm < Line-to-Line voltage

Therefore this setup can also be used for determining phase sequence.

EQUIPMENT

1. Lamp Holder : ....2 pc
2. Bulb   : 60 Watt ....2 pc
3. Rheostat   : 500 ohms (3 Amp.) .... 1 pc
4. Inductor   : Ballast .... 1 pc
5. Capacitor : 3.4uF , 380 V (fan / motor capacitor) .... 1 pc
6. SPST switch : .... .3 pc

7. Multi-meter .... 1 pc

8. Wires for connection
9. 3 phase supply cord .... .3 pc
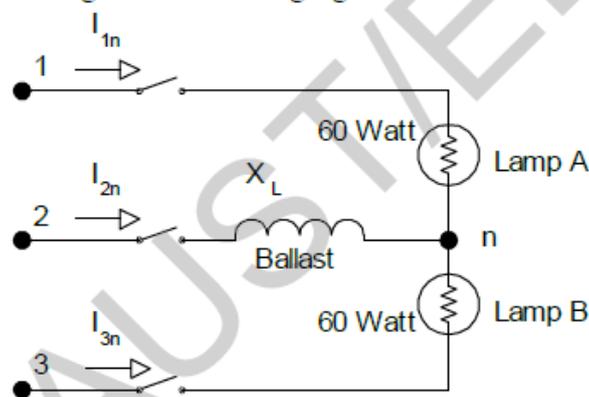
PROCEDURE

Step : 1

First mark the three wires arbitrarily by the notations '1' , '2' and '3'.

Step : 2

Setup the equipments according to the following figure :

Take the following readings :

R_L = [ ]

V_{12} = [ ] V_{23} = [ ] V_{31} = [ ]

V_{1n} = [ ] V_{2n} = [ ] V_{3n} = [ ]

I_{1n} = [ ] I_{2n} = [ ] I_{3n} = [ ]

Note which Lamp glows brighter ? [ ]

Calculate the following :

$Z_{1n} = R_1 = V_{1n} / I_{1n} =$ [ ]

$Z_{3n} = R_2 = V_{3n} / I_{3n} =$ [ ]

$Z_{2n} = R_L + j X_L = Z_T \angle \theta = (V_{2n} / I_{2n}) \angle (\tan^{-1} X_L / R_L) =$ [ ]
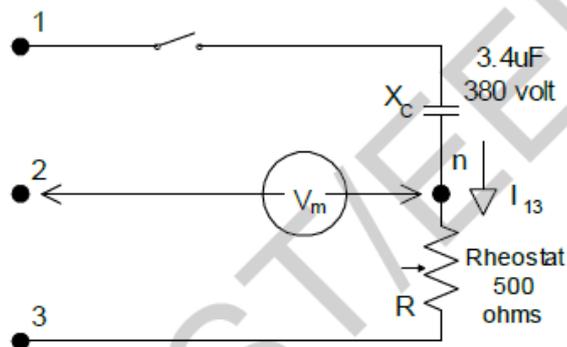
Step : 3

Now interchange the wires at the phases '1' and '3'.

Note which Lamp glows brighter ? [ ]

Step : 4

Adjust the rheostat to its full resistance range and measure its value. Setup the equipments according to the following figure :

Take the following readings :

R = [ ]

$V_{12} =$ [ ] $V_{23} =$ [ ] $V_{31} =$ [ ]

$V_{1n} =$ [ ] $V_m =$ [ ] $V_{3n} =$ [ ]

$I_{1n} =$ [ ]

Note whether the voltage V_m is greater than line voltage or not ? [ ]

Calculate the followings :

$Z_T = R - j X_C =$ [ ] $= Z \angle \theta =$ [ ]

Step : 5

Now interchange the wires at the phases '1' and '3'.
Take the following readings :

$V_{1n} =$ [   ] $V_m =$ [   ] $V_{3n} =$ [   ]

$I_{1n} =$ [   ]

Note whether the voltage V_m is greater than line voltage or not ? [   ]

REPORT

1. From data obtained form step 2 calculate the voltage across the Lamps 'A' and 'B' (use equation 4 and 5). Check your results against the phase sequence found.

2. Using the data found in step 4 and step 5, draw the phasor diagrams for the voltmeter method for both squences.

N.B Discharge capacitor after you switch of the supply by proper method.

STUDY OF RC FILTER CHARACTERISTICS

INTRODUCTION

Many electrical circuits contain filters, which separate electric signals on the basis of their frequency contents. Filters are characterized by their frequency response as well as phase relation between input and output signals. In this experiment we will study low pass, high pass and band pass filters.

THEORY

An ideal low pass filter passes all signals below a certain frequency which is termed as the *cutoff frequency* for that filter. In other words the output signal amplitude will be zero if we feed the filter with an input signal that has frequency greater than its cutoff frequency. For signals with lower frequency the output signal amplitude will remain unaffected. This is true only for „ideal filter" and the frequency response curve for ideal filter looks like Fig 1 (a). But for a practical filter the amplitude near the cutoff does not change instantly rather it follows a decreasing nature as shown in Fig 1 (b).Thus for a practical filter we need to define cutoff frequency in different way. Some of the definitions for cutoff frequencies are given below:
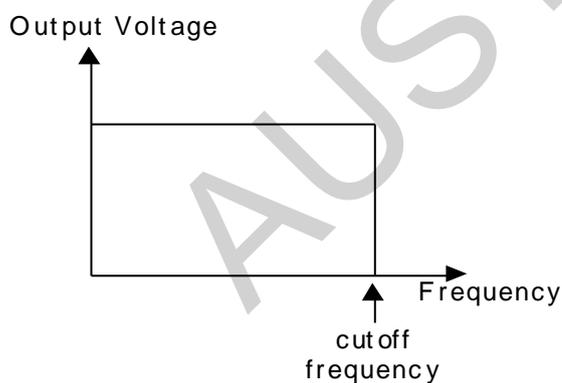

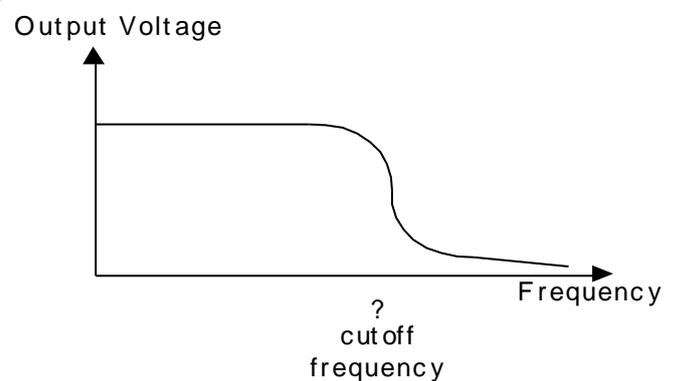Fig 1. (a) : Ideal Low pass Filter   Fig 1. (b) : Practical Low pass Filter

What is cutoff frequency ?

1. When our frequency response curve is given in terms of „voltage vs. frequency" :

   "cutoff frequency is the point at which the voltage level of the signal falls "1 / √2 or 0.707" times from its maximum value provided that the input signal amplitude remains same for all frequencies."

2. When our frequency response curve is given in terms of „signal power vs. frequency" :

> "cutoff frequency is the point at which the power level of the signal falls "1/2 or 0.5" times from its maximum value."

3. When our frequency response curve is given in terms of „gain vs. frequency" :

> "cutoff frequency is the point at which the gain of the signal falls "1 / √2 or 0.707" times from its maximum value."

4. When our frequency response curve is given in terms of „gain(in db) vs. frequency" :

> "cutoff frequency is the point at which the gain(in db) of the signal falls "-3db" from its maximum value."

A simple first order RC low pass filter is shown in figure 2. Gain of this filter is given by:

$$A_v = \frac{V_o}{V_i} = \frac{1}{j\omega C(R + j\omega C)}$$

where ω_C is the corner or cutoff frequency . For our low pass filter, cut off frequency, $\omega_C = \frac{1}{RC}$.

An ideal high pass filter – blocks all signals below its cutoff frequency but passes all signals above cut off with no attenuation. A simple first order RC high pass filter is shown in figure 3. The gain of this filter is:

$$A_v = \frac{R}{(R + j\omega C)}$$

And its corner frequency ω_C is also $\frac{1}{RC}$. Figure 4 shows a second order band pass filter, which is designed to pass all signals whose frequency lie within a specific band. Therefore it has two cutoff frequencies ω_{CL} below which the filter blocks signals and ω_{CH} above which it again blocks signals. Thus the filter has a pass band between these two frequencies and the frequency ω_O is the mid band frequency where the gain is maximum. The gain of this filter is given by

$$A_v = \frac{\dfrac{1}{j\omega R_1 C_1}}{1 + \dfrac{1}{j\omega}\left(\dfrac{1}{R_1 C_1} + \dfrac{1}{R_2 C_2} + \dfrac{1}{R_1 C_2}\right) + \dfrac{R_1 C_1 R_2 C_2}{(j\omega)^2}}$$
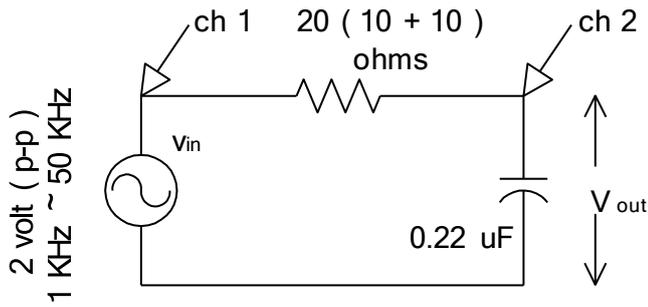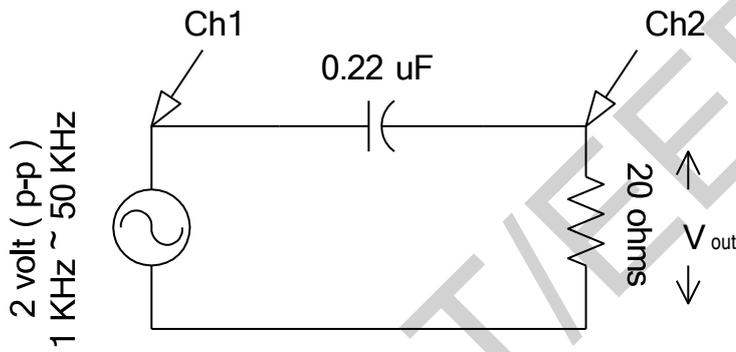
CIRCUIT DIAGRAM

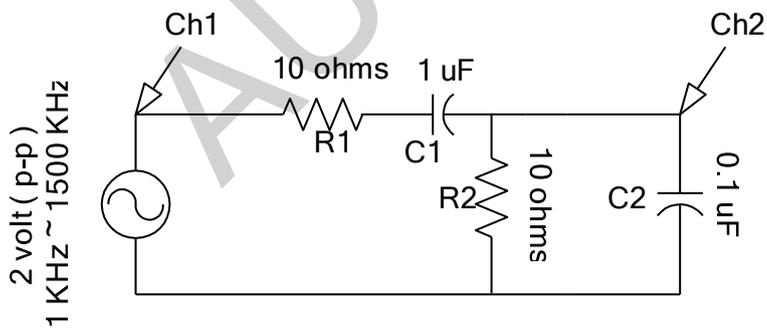
Figure 2 : RC Lowpass Filter

Figure 3: RC High pass Filter

Figure 4 : RC Bandpass Filter

APPARATUS

1. Resistance : 10 Ω ....2 pc
2. Capacitor : 0.22uf ....1 pc
   0.1 uf ....1 pc
   1uf ....1 pc
   2.2 uf ....1 pc

3. Oscilloscope
4. Oscilloscope Probe ....2 pc
5. Signal Generator ....1 pc
6. Signal Generator Probe ....1 pc
7. Multi-meter ....1 pc
8. Bread Board ....1 pc
9. Wires for connection

PROCEDURE

1. Construct the circuit of figure 2. Connect V_{in} to channel 1 and V_{out} to channel 2 of the oscilloscope. Adjust the signal generator to 2 volt (p-p) & 100 Hz.
2. Measure V_{in} , V_{out} and phase difference between them using oscilloscope and fill up the table .Repeat for each of the frequencies shown in the table.

Note : Adjust the amplitude of source (channel 1) each time to 2 volt (p-p).

Table : 1

Frequency, f (Hz)	Output Voltage, V_{out}	Gain , V_{out} / V_{in}	Phase difference, θ
50 (20ms)			
333 (3ms)			
500 (2ms)			
1000 (1ms)			
5000 (0.2ms)			
20000 (50 us)			
33333 (30 us)			
50000 (20 us)			
100000 (10 us)			

V_o lags V_{in} or V_o leads V_{in}

Table 2:

Frequency, f (Hz)	Output Voltage, V_{out}	Gain, V_{out} / V_{in}	Phase difference, θ
10000 (0.1ms)			
20000 (50 us)			
40000 (25 us)			
50000 (20 us)			
66666 (15 us)			
400000 (2.5 us)			
666666 (1.5 us)			
1000000 (1 us)			

V_o lags V_{in} or V_o leads V_{in}

4.   Now construct the circuit of figure 4 and repeat steps 1 and 2 and fill up table 3.

Table 3:

Frequency, f (Hz)	Output Voltage, V_{out}	Gain, V_{out} / V_{in}	Gain in dB, $20 \log_{10} (V_{out} / V_{in})$
1000 (1ms)			
3333 (0.3 ms)			
8000 (125 us)			
16666 (60 us)			
25000 (40 us)			
50000 (20 us)			
100000 (10 us)			
166666 (6 us)			
250000 (4 us)			
500000 (2 us)			

V_o lags V_{in} or V_o leads V_{in}

REPORT

1. Determine the theoretical corner frequencies of filter 1, 2 and 3 and also the frequency ω_o, at which the gain of filter 4 is maximum.
2. Plot the gain, A_V vs frequency curves for the three filters on separate sheets. Use semi-logarithm graph paper. For the bandpass filter plot the gain (in dB) vs. frequency.
3. Determine the cut-off frequencies from the graphs and compare these with the theoretical values.
4. For filters 1 and 2, plot the phase difference vs. frequency on semi-log graph paper and determine the cut-off frequency.