

Department of Computer Systems Engineering

University of Engineering & Technology

Peshawar Pakistan



# Circuits and Systems II

## Lab Manual

Prepared By

Engr. Arbab Masood Ahmad

December 2015

## List of Experiments

<b>Lab 1</b>	The Oscilloscope
<b>Lab 2</b>	Capacitive Reactance
<b>Lab 3</b>	Inductive Reactance
<b>Lab 4</b>	Series R, L, C Circuit
<b>Lab 5</b>	Parallel R, L, C Circuit
<b>Lab 6</b>	AC Superposition
<b>Lab 7</b>	Operational Amplifier-Basic Characteristics and Applications-Inverting and Non-Inverting Amplifier
<b>Lab 8</b>	Op-Amp Inverting Summing and Difference Amplifiers
<b>Lab 9</b>	Series Resonance Circuit
<b>Lab 10</b>	Filters-High Pass, Low Pass, Band Pass and Notch
<b>Lab 11</b>	Integrator using Op-Amp 741
<b>Lab 12</b>	Active Low Pass Filter
<b>Lab 13</b>	Active High Pass Filter
<b>Lab 14</b>	Introduction to Pspice
<b>Lab 15a</b>	To Find The Various Node Voltages And Voltage drops In the Given Circuit
<b>Lab 15b</b>	To Find The Various Loop And Branch Currents In the Given Circuit
<b>Lab 16a</b>	To analyze R-C and R-L circuit using PSPICE (Natural Response).
<b>Lab 16b</b>	To analyze R-C and R-L circuit using PSPICE (Step Response)
<b>Lab 17</b>	To analyze low pass, band pass filter and notch filter using PSPICE

## Lab 1

### The Oscilloscope

#### Objective

This exercise is of a particularly practical nature, namely, introducing the use of the oscilloscope. The various input scaling, coupling, and triggering settings are examined along with a few specialty features.

#### Theory Overview

The oscilloscope (or simply scope, for short) is arguably the single most useful piece of test equipment in an electronics laboratory. The primary purpose of the oscilloscope is to plot a voltage versus time, although it can also be used to plot one voltage versus another voltage, and in some cases, to plot voltage versus frequency. Oscilloscopes are capable of measuring both AC and DC waveforms, and unlike typical DMMs, can measure AC waveforms of very high frequency (typically 100 MHz or more versus an upper limit of around 1 kHz for a general purpose DMM). It is also worth noting that a DMM will measure the RMS value of an AC sinusoidal voltage, not its peak value.

While the modern digital oscilloscope on the surface appears much like its analog ancestors, the internal circuitry is far more complicated and the instrument affords much greater flexibility in measurement.

At a minimum, modern oscilloscopes offer two input measurement channels although four and eight channel instruments are increasing in popularity.

Unlike handheld DMMs, most oscilloscopes measure voltages with respect to ground, that is, the inputs are not floating and thus the black, or ground, lead is always connected to the circuit ground or common node. This is an extremely important point as failure to remember this may lead to the inadvertent short circuiting of components during measurement. The standard accepted method of measuring a non-ground referenced potential is to use two probes, one tied to each node of interest, and then setting the oscilloscope to subtract the two channels rather than display each separately. Note that this technique is not required if the oscilloscope has floating inputs (for example, in a handheld oscilloscope). Further, while it is possible to measure non-ground referenced signals by floating the oscilloscope itself through defeating the ground pin on the power cord, this is a safety violation and should not be done.

#### Equipment

1. DC Power Supply
2. AC Function Generator
3. Digital Multimeter
4. Oscilloscope

#### Components

- |                  |               |
|------------------|---------------|
| 1. 10 k $\Omega$ | actual: _____ |
| 2. 33 k $\Omega$ | actual: _____ |

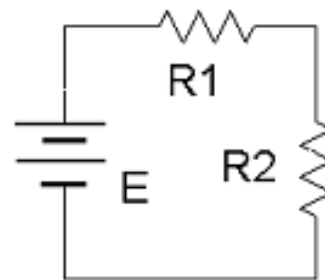


Figure 1

## Procedure

1. Find the following elements on your oscilloscope:
  - Channel-1 and Channel-2 BNC input connectors.
  - Trigger BNC input connector.
  - Channel-1 and Channel-2 select buttons.
  - Horizontal Sensitivity (or Scale) and Position knobs.
  - Vertical Sensitivity (or Scale) and Position knobs.
  - Trigger Level knob.
2. Note that the main display is similar to a sheet of graph paper. Each square will have an appropriate scaling factor or weighting, for example, 1 volt per division vertically or 2 milliseconds per division horizontally. Waveform voltages and timings may be determined directly from the display by using these scales.
3. Select the channel-1 and 2 buttons. There should now be two horizontal lines on the display. They may be moved via the Position knob.
4. One of the more important fundamental settings on an oscilloscope is the Input Coupling. This is controlled via one of the bottom row buttons. There are three choices: Ground removes the input thus showing a zero reference, AC allows only AC signals through thus blocking DC, and DC allows all signals through (it does not prevent AC).
5. Set the channel-1 Vertical Scale to 5 volts per division. Set the channel-2 Scale to 2 volts per division. Set the Time (Horizontal) Scale to 1 millisecond per division. Finally, set the input coupling to Ground for both input channels and align the two lines to the center line of the display via the Vertical Position knob.
6. Build the circuit shown in the figure using  $E=10V$ ,  $R_1=10k\Omega$  and  $R_2= 33k\Omega$ . Connect a probe from the channel-1 input to the power supply (tip to plus, black clip to ground). Connect a second probe from channel-2 to  $R_2$  (again, tip to the high side of the resistor and the black clip to ground).
7. Switch both inputs to DC coupling. The two lines should have deflected upward. Channel-1 should be raised two divisions (2 divisions at 5 volts per division yields the 10 volt source). Using this method, determine the voltage across  $R_2$  (remember, input-2 should have been set for 2 volts per division). Calculate the expected voltage across  $R_2$  using measured resistor values and compare the two in Table 1. Note that it is not possible to achieve extremely high precision using this method (e.g., four or more digits). Indeed, a DMM is often more useful for direct measurement of DC potentials. Double check the results using a DMM and the final column of Table 1.
8. Select AC Coupling for the two inputs. The flat DC lines should drop back to zero. This is because AC Coupling blocks DC. This will be useful for measuring the AC component of a combined AC/DC signal, such as might be seen in an audio amplifier. Set the input coupling for both channels back to DC.
9. Replace the DC power supply with the function generator. Set the function generator for a 1 volt peak sine wave at 1 kHz and apply it to the resistor network. The display should now show two small sine waves. Adjust the Vertical Scale settings for the two

inputs so that the waves take up the majority of the display. If the display is very blurry with the sine waves appearing to jump about side to side, the Trigger Level may need to be adjusted. Also, adjust the Time Scale so that only one or two cycles of the wave may be seen. Using the Scale settings, determine the two voltages (following the method of step 7) as well as the waveform's period and compare them to the values expected via theory, recording the results in Tables 2 and 3. Also crosscheck the results using a DMM to measure the RMS voltages.

10. To find the voltage across R1, the channel-2 voltage (VR2) may be subtracted from channel-1 (E source).
11. One of the more useful aspects of the oscilloscope is the ability to show the actual wave shape. This may be used, for example, as a means of determining distortion in an amplifier. Change the wave shape on the function generator to a square wave, triangle, or other shape and note how the oscilloscope responds. Note that the oscilloscope will also show a DC component, if any, as the AC signal being offset or "riding on the DC". Adjust the function generator to add a DC offset to the signal and note how the oscilloscope display shifts. Return the function generator back to a sine wave and remove any DC offset.

## Data Tables

$V_{R2}$	Scale (V/Div)	Number of Divisions	Voltage Scope	Voltage DMM
Oscilloscope				
Theory	X	X		

**Table 1**

	Scale (V/Div)	Number of Divisions	Voltage Peak	Voltage RMS
E Oscilloscope				
E Theory	X	X		
$V_{R2}$ Oscilloscope				
$V_{R2}$ Theory	X	X		

**Table 2**

	Scale (S/Div)	Number of Divisions	Period	Frequency
E Oscilloscope				
E Theory	X	X		

**Table 3**

## Lab 2

### Capacitive Reactance

#### Objective

Capacitive reactance will be examined in this exercise. In particular, its relationship to capacitance and frequency will be investigated, including a plot of capacitive reactance versus frequency.

#### Theory Overview

The current – voltage characteristic of a capacitor is unlike that of typical resistors. While resistors show a constant resistance value over a wide range of frequencies, the equivalent ohmic value for a capacitor, known as capacitive reactance, is inversely proportional to frequency. The capacitive reactance may be computed via the formula:

$$X_c = -j \frac{1}{2\pi f C}$$

The magnitude of capacitive reactance may be determined experimentally by feeding a capacitor a known current, measuring the resulting voltage, and dividing the two, following Ohm's Law. This process may be repeated across a range of frequencies in order to obtain a plot of capacitive reactance versus frequency. An AC current source may be approximated by placing a large resistance in series with an AC voltage, the resistance being considerably larger than the maximum reactance expected.

#### Equipment

1. AC Function Generator
2. Oscilloscope

#### Components

1. 1  $\mu\text{F}$  actual: \_\_\_\_\_
2. 2.2  $\mu\text{F}$  actual: \_\_\_\_\_
3. 10  $\text{k}\Omega$  actual: \_\_\_\_\_

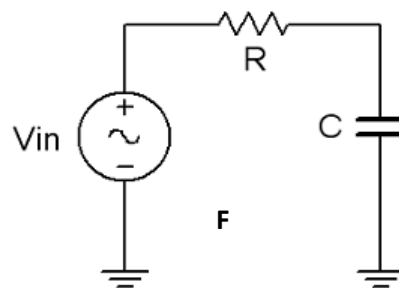


Figure 1

#### Procedure

##### Current Source

1. Using Figure 1 with  $V_{in}=10V_{p-p}$  and  $R=10\text{k}\Omega$ , and assuming that the reactance of the capacitor is much smaller than 10k and can be ignored, determine the circulating current using measured component values and record in Table 1.

## Measuring Reactance

2. Build the circuit of Figure 1 using  $R=10k\Omega$ , and  $C=1\ \mu F$ . Place one probe across the generator and another across the capacitor. Set the generator to a 200 Hz sine wave and  $10V_{p-p}$ . Make sure that the Bandwidth Limit of the oscilloscope is engaged for both channels. This will reduce the signal noise and make for more accurate readings.
3. Calculate the theoretical value of  $X_c$  using the measured capacitor value and record in Table 2.
4. Record the peak-to-peak capacitor voltage and record in Table 2.
5. Using the source current from Table 1 and the measured capacitor voltage, determine the experimental reactance and record it in Table 2. Also compute and record the deviation.
6. Repeat steps three through five for the remaining frequencies of Table 2.
7. Replace the  $1\ \mu F$  capacitor with the  $2.2\ \mu F$  unit and repeat steps two through six, recording results in Table 3.
8. Using the data of Tables 2 and 3, create plots of capacitive reactance versus frequency.

$i_{source}\ (p-p)$	
---------------------	--

**Table 1**

Frequency	$X_c$ Theory	$V_{C(p-p)}$ Exp	$X_c$ Exp	% Dev
200				
400				
600				
800				
1.0 k				
1.2 k				
1.6 k				
2.0 k				

**Table 2**

Frequency	$X_C$ Theory	$V_{C(p-p)}$ Exp	$X_C$ Exp	% Dev
200				
400				
600				
800				
1.0 k				
1.2 k				
1.6 k				
2.0 k				

**Table 3**

### Questions

1. What is the relationship between capacitive reactance and frequency?
2. What is the relationship between capacitive reactance and capacitance?
3. If the experiment had been repeated with frequencies 10 times higher than those in Table 2, what would the resulting plots look like?
4. If the experiment had been repeated with frequencies 10 times lower than that in Table 2, what effect would that have on the experiment?



## Lab 3

### Inductive Reactance

#### Objective

Inductive reactance will be examined in this exercise. In particular, its relationship to inductance and frequency will be investigated, including a plot of inductive reactance versus frequency.

#### Theory Overview

The current – voltage characteristic of an inductor is unlike that of typical resistors. While resistors show a constant resistance value over a wide range of frequencies, the equivalent ohmic value for an inductor, known as inductive reactance, is directly proportional to frequency. The inductive reactance may be computed via the formula:

$$X_L = j 2\pi f L$$

The magnitude of inductive reactance may be determined experimentally by feeding an inductor a known current, measuring the resulting voltage, and dividing the two, following Ohm's Law. This process may be repeated across a range of frequencies in order to obtain a plot of inductive reactance versus frequency. An AC current source may be approximated by placing a large resistance in series with an AC voltage, the resistance being considerably larger than the maximum reactance expected.

#### Equipment

1. AC Function Generator
2. Oscilloscope

DMM

#### Components

1. 1 mH actual: \_\_\_\_\_
2. 10 mH actual: \_\_\_\_\_
3. 10 k $\Omega$  actual: \_\_\_\_\_

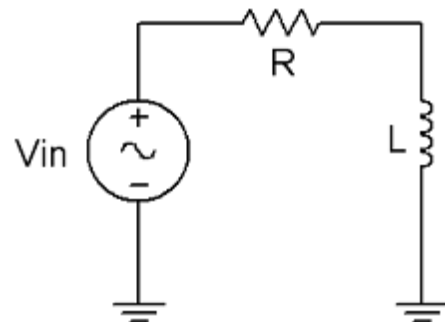


Figure 1

#### Procedure

##### 1. Current Source

Using Figure 1 with  $V_{in}=10$  Vp-p and  $R=10$  k $\Omega$ , and assuming that the reactance of the inductor is much smaller than 10k and can be ignored, determine the circulating current using measured component values and record in Table 1. Also, measure the DC coil resistances of the inductors using an ohmmeter or DMM and record in Table 5.1.

##### 2. Measuring Reactance

Build the circuit of Figure 1 using  $R=10$  k $\Omega$ , and  $L=10$  mH. Place one probe across the generator and another across the inductor. Set the generator to a 1000 Hz sine wave and 10Vp-p. Make sure that the Bandwidth Limit of the oscilloscope is engaged for both channels. This will reduce the signal noise and make for more accurate readings.

3. Calculate the theoretical value of  $X_L$  using the measured inductor value and record in Table2.

4. Record the peak-to-peak inductor voltage and record in Table 2.
5. Using the source current from Table 1 and the measured inductor voltage, determine the experimental reactance and record it in Table 2. Also compute and record the deviation.
6. Repeat steps three through five for the remaining frequencies of Table 2.
7. Replace the 10 mH inductor with the 1mH unit and repeat steps two through six, recording results in Table 3.
8. Using the data of Tables 2 and 3, create plots of inductive reactance versus frequency.

$I_{\text{source(p-p)}}$	
$R_{\text{coil of 10 mH}}$	
$R_{\text{coil of 1 mH}}$	

Table 1

Frequency	$X_L$ Theory	$V_{L(p-p)}$ Exp	$X_L$ Exp	% Dev
1 k				
2 k				
3 k				
4 k				
5 k				
6 k				
8 k				
10 k				

Table 2

Frequency	$X_L$ Theory	$V_{L(p-p)}$ Exp	$X_L$ Exp	% Dev
10 k				
20 k				
30 k				
40 k				
50 k				
60 k				
80 k				
100 k				

Table 3

### Questions

1. What is the relationship between inductive reactance and frequency?
2. What is the relationship between inductive reactance and inductance?
3. If the 10mH trial had been repeated with frequencies 10 times higher than those in Table 2, what effect would that have on the experiment?
4. Do the coil resistances have any effect on the plots?

## Lab 4

### Series R, L, C Circuits

#### Objective

This exercise examines the voltage and current relationships in series R, L, C networks. Of particular importance is the phase of the various components and how Kirchhoff's Voltage Law is extended for AC circuits. Both time domain and phasor plots of the voltages are generated.

#### Theory Overview

Each element has a unique phase response: for resistors, the voltage is always in phase with the current, for capacitors the voltage always lags the current by 90 degrees, and for inductors the voltage always leads the current by 90 degrees. Consequently, a series combination of R, L, and C components will yield a complex impedance with a phase angle between +90 and -90 degrees. Due to the phase response, Kirchhoff's Voltage Law must be computed using vector (phasor) sums rather than simply relying on the magnitudes. Indeed, all computations of this nature, such as a voltage divider, must be computed using vectors.

#### Equipment

1. AC Function Generator
2. Oscilloscope

#### Components

1. 10 nF actual: \_\_\_\_\_
2. 10 mH actual: \_\_\_\_\_
3. 1 k $\Omega$  actual: \_\_\_\_\_

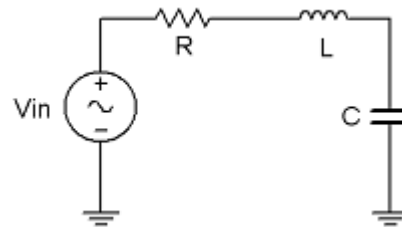


Figure 1

#### Procedure

1. Using Figure 1 with  $V_{in}=2V_{p-p}$  sine at 10 kHz,  $R=1k\Omega$ ,  $L=10mH$  and  $C=10nF$ , determine the theoretical inductive and capacitive reactance and circuit impedance, and record the results in Table 1 (the experimental portion of this table will be filled out in step 4). Using the voltage divider rule, compute the resistor, inductor and capacitor voltages and record them in Table 2.
2. Build the circuit of Figure 1 using  $R=1k\Omega$ ,  $L=10mH$  and  $C=10nF$ . Set the generator to a 10 kHz sine wave and  $2 V_{p-p}$ . Using oscilloscope measure the signals. Unfortunately it is impossible to see the voltages of all the three components simultaneously using only two probes of the oscilloscope. To obtain the proper readings, place one probe on the function generator to see the input signal and the second probe across the last element. This step is repeated three times. The first time the components are so arranged that capacitor is the last component, the second time inductor is connected as the last component and finally resistor is made the last component. The peak to peak voltages and phase angles of each one of the three components, relative to the source are thus determined in turn. Thus  $V_{in}$ ,  $V_C$ ,  $V_L$  and  $V_R$  are measured. The phase angle of the last component relative to the source is determined by measuring the time difference ' $t$ ' between the

zero crossings of the two waveforms. Thus if ' $T$ ' is the time period of the signal then the phase difference can be calculated using the formula:  $\theta^\circ = 360 \times t/T$ . Record this information in Table 2.

3. Compute the deviations between the theoretical and experimental values of Table 2 and record the results in the final columns of this table.
4. Based on the experimental values, determine the experimental  $Z$ ,  $X_L$  and  $X_C$  values via Ohm's Law ( $i = V_R/R$ ,  $X_L = V_L/i$ ,  $X_C = V_C/i$ ,  $Z = V_{in}/i$ ) and record back in Table 1 along with the deviations.
5. Create a phasor plot showing  $V_{in}$ ,  $V_L$ ,  $V_C$ , and  $V_R$ .

## Data Tables

	Theory	Experimental	% Deviation
$X_C$			
$X_L$			
$Z$ Magnitude			
$Z \theta$			

Table 1

	Theory Mag	Theory $\theta$	Exp Mag	Exp Delay	Exp $\theta$	% Dev Mag	% Dev $\theta$
$V_C$							
$V_L$							
$V_R$							

Table 2

## Questions

1. What is the phase relationship between R, L, and C components in a series AC circuit?
2. Based on measurements, does Kirchhoff's Voltage Law apply to the tested circuits?
3. In general, how would the phasor diagram of Figure 1 change if the frequency was raised?
4. In general, how would the phasor diagram of Figure 1 change if the frequency was lowered?

## Lab 5

### Parallel R, L, C Circuits

#### Objective

This exercise examines the voltage and current relationships in parallel R, L, C networks. Of particular importance is the phase of the various components and how Kirchhoff's Current Law is extended for AC circuits. Both time domain and phasor plots of the currents are generated. A technique to measure current using a current sense resistor will also be explored.

#### Theory Overview

Recall that for resistors, the voltage is always in phase with the current, for capacitors the voltage always lags the current by 90 degrees, and for inductors the voltage always leads the current by 90 degrees. Because each element has a unique phase response between +90 and -90 degrees, a parallel combination of R, L and C components will yield a complex impedance with a phase angle between +90 and -90 degrees. Due to the phase response, Kirchhoff's Current law must be computed using vector (phasor) sums rather than simply relying on the magnitudes. Indeed, all computations of this nature, such as a current divider, must be computed using vectors.

#### Equipment

1. AC Function Generator
2. Oscilloscope

#### Components

1. 10 nF actual: \_\_\_\_\_
2. 10 mH actual: \_\_\_\_\_
3. 1 k $\Omega$  actual: \_\_\_\_\_
4. 10  $\Omega$  actual: \_\_\_\_\_

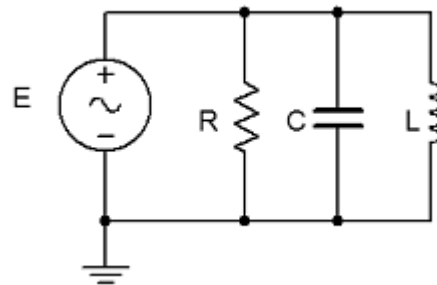


Figure 1

#### Procedure

1. Using Figure 1 with a 10 V p-p 10 kHz source,  $R=1\text{k}\Omega$ ,  $C=10\text{nF}$  and  $L=10\text{mH}$ , determine the theoretical capacitive reactance, inductive reactance and circuit impedance, and record the results in Table 1 (the experimental portion of this table will be filled out in step 5). Using the current divider rule, compute the currents in resistor( $i_R$ ), inductor( $i_L$ ) and capacitor( $i_C$ ) and record them in Table 2.
2. Build the circuit of Figure 1 using  $R=1\text{k}\Omega$ ,  $L=10\text{mH}$  and  $C=10\text{nF}$ . A common method to measure current using the oscilloscope is to place a small current sense resistor in line with the current of interest. If the resistor is much smaller than the surrounding reactance it will have a minimal effect on the current. Because the voltage and current of the resistor are always in phase with each other, the relative phase of the current in question must be the same as that of the sensing resistor's voltage. Each of the three circuit currents will be measured separately and with respect to the source in order to determine relative phase.

To measure the total current, place a 10 $\Omega$  resistor between ground and the bottom connection of the parallel components. Set the generator to a 10 V p-p sine wave at 10 kHz.

- Place probe1 across the generator and probe2 across the sense resistor. Measure the voltage across the sense resistor; calculate the corresponding total current via Ohm's Law and record in Table 2. Along with the magnitude, be sure to record the time deviation between the sense waveform and the input signal (from which the phase may be determined eventually).
- Remove the sense resistor and place one  $10\Omega$  resistor between the capacitor and ground to serve as the capacitor current sense. Place a second  $10\Omega$  resistor between the resistor and ground to sense the resistor current, and a third  $10\Omega$  resistor between the inductor and ground for the inductor current. Leave probe one at the generator and move probe two across the sense resistor in the resistor branch. Repeat the process to obtain its current, recording the magnitude and phase angle in Table 2. In a similar way move probe2 so that it is first across the capacitor's sense resistor and then across the inductor sense resistor. Measure and record the appropriate values in Table 2.
- Compute the deviations between the theoretical and experimental values of Table 2 and record the results in the final columns of Table 2. Based on the experimental values, determine the experimental  $Z$ ,  $X_L$  and  $X_C$  values via Ohm's Law ( $X_C = V_C/I_C$ ,  $X_L = V_L/I_L$  and  $X_Z = V_{in}/i_{in}$ ) and record back in Table 1 along with the deviations.
- Create a phasor plot showing  $i_{in}$ ,  $i_C$ ,  $i_L$  and  $i_R$ . Include both the time domain display and the phasor plot with the technical report.

	Theory	Experimental	% Deviation
$X_C$			
$X_L$			
$Z$ Magnitude			
$Z \theta$			

Table 1

	Theory Mag	Theory $\theta$	Exp Mag	Exp Delay	Exp $\theta$	% Dev Mag	% Dev $\theta$
$i_C$							
$i_L$							
$i_R$							
$i_{in}$							

Table 2

## Lab 6

### AC Superposition

#### Objective

This exercise examines the analysis of multi-source AC circuits using the Superposition Theorem. In particular, sources with differing frequencies will be used to illustrate the contributions of each source to the combined result.

#### Theory Overview

The Superposition Theorem can be used to analyze multi-source AC linear bilateral networks. Each source is considered in turn, with the remaining sources replaced by their internal impedance, and appropriate series-parallel analysis techniques employed. The resulting signals are then summed to produce the combined output signal. To see this process more clearly, the exercise will utilize two sources operating at different frequencies. Note that as each source has a different frequency, the inductor and capacitor appear as different reactances to the two sources.

#### Equipment

1. AC Function Generators
2. Oscilloscope

#### Components

- |                      |               |
|----------------------|---------------|
| 1. 0.1 $\mu\text{F}$ | actual: _____ |
| 2. 10mH              | actual: _____ |
| 3. 1k $\Omega$       | actual: _____ |
| 4. 50 $\Omega$       | actual: _____ |

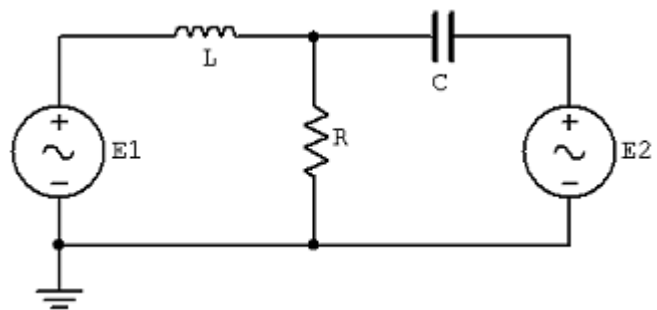


Figure 1

#### Procedure

1. Typical function generators have a 50 $\Omega$  internal impedance. These are not shown in the circuit of Figure 1. To test the Superposition Theorem, sources E1 and E2 will be examined separately and then together.

#### Source One Only

2. Consider the circuit of Figure 1 with  $C=0.1 \mu\text{F}$ ,  $L=10\text{mH}$ ,  $R=1\text{k}\Omega$ , using only source  $E1=2 \text{ V}_{\text{p-p}}$  at 1 kHz and with source E2 replaced by its internal impedance of 50  $\Omega$ . Using standard series-parallel techniques; calculate the voltages across E1, R, and E2. Remember to include the 50  $\Omega$  internal impedances in the calculations. Record the results in Table 10.1.
3. Build the circuit of Figure 10.1 using  $C=0.1 \mu\text{F}$ ,  $L=10\text{mH}$ , and  $R=1\text{k}\Omega$ . Replace E2 with a 50  $\Omega$  resistor to represent its internal impedance. Set E1 to 2V p-p at 1 kHz, unloaded. Place probe one across E1 and probe two across R. Measure the voltages across E1 and R, and record in Table 1. Record a copy of the scope image. Move probe two across E2 (the 50  $\Omega$ ), measure and record this voltage in Table 1.



### Source Two Only

4. Consider the circuit of Figure 10.1 using only source  $E_2=2\text{ V p-p}$  at 10 kHz and with source  $E_1$  replaced by its internal impedance of  $50\ \Omega$ . Using standard series-parallel techniques; calculate the voltages across  $E_1$ ,  $R$ , and  $E_2$ . Remember to include the  $50\ \Omega$  internal impedances in the calculations. Record the results in Table 2.
5. Replace the  $50\ \Omega$  with source  $E_2$  and set it to  $2V_{p-p}$  at 10 kHz, unloaded. Replace  $E_1$  with a  $50\ \Omega$  resistor to represent its internal impedance. Place probe one across  $E_2$  and probe two across  $R$ . Measure the voltages across  $E_2$  and  $R$ , and record in Table 10.2. Record a copy of the scope image. Move probe two across  $E_1$  (the  $50\ \Omega$ ), measure and record this voltage in Table 2.

### Sources One and Two

6. Consider the circuit of Figure 1 using both sources,  $E_1=2V_{p-p}$  at 1 kHz and  $E_2=2V_{p-p}$  at 10 kHz. Add the calculated voltages across  $E_1$ ,  $R$ , and  $E_2$  from Tables 1 and 2. Record the results in Table 3. Make a note of the expected maxima and minima of these waves and sketch how the combination should appear on the scope.
7. Replace the  $50\ \Omega$  with source  $E_1$  and set it to  $2V_{p-p}$  at 1 kHz, unloaded. Both sources should now be active. Place probe one across  $E_1$  and probe two across  $R$ . Measure the voltages across  $E_1$  and  $R$ , and record in Table 3. Record a copy of the scope image. Move probe two across  $E_2$ , measure and record this voltage in Table 3

### Data Tables

#### Source One Only

	Theory	Experimental	% Deviation
$E_1$			
$E_2$			
$V_R$			

Table 1

### Source Two Only

	Theory	Experimental	% Deviation
$E_1$			
$E_2$			
$V_R$			

**Table 2**

### Sources One and Two

	Theory	Experimental	% Deviation
$E_1$			
$E_2$			
$V_R$			

**Table 3**

### Questions

1. Why must the sources be replaced with a  $50\ \Omega$  resistor instead of being shorted?
2. Do the expected maxima and minima from step 6 match what is measured in step 7?
3. Does one source tend to dominate the  $1\text{k}\Omega$  resistor voltage or do both sources contribute in nearly equal amounts? Will this always be the case?

## Lab 7

### Operational Amplifiers – Basic Characteristics and Applications

#### Objective

The objective of this lab experiment is to learn how to use the operational amplifier (op-amp). In this experiment some of the basic characteristics of the op-amp would be examined and then some of its applications like the Inverting amplifier, Non inverting amplifier will be experimented.

#### Theory Overview

The Operational Amplifier (Op Amp) is an extremely useful device, as we will see in this lab. With the addition of a few external components, an extraordinary variety of functions can be implemented. The Op Amp is an active element that needs to be supplied with power to operate. A common way to supply this power is shown in Figure 1(a). Two power supply voltages are used, with equal values denoted by  $V_{CC}$  and  $V_{DD}$  (or  $\pm V_{CC}$ ) (often in the range of 5 V to 15 V). The common node between the supplies is the ground node. The op amp's output voltage is taken between the output terminal and the ground node. The remaining two terminals are the input of the op amp. An interesting property of the op-amp is that the output voltage is only a function of the difference of the two input Terminals. Figure 1(b) shows the top view of widely used OpAmp type known as the 741. It comes in a package, with metal pins.

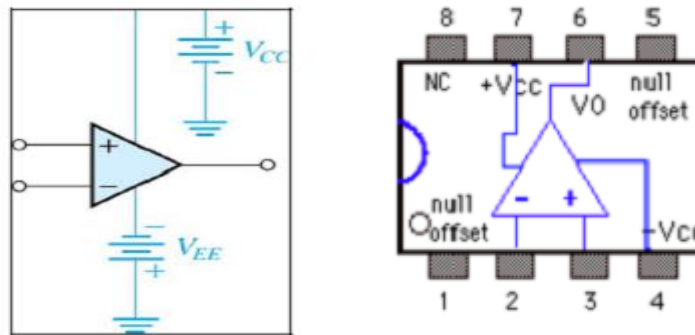
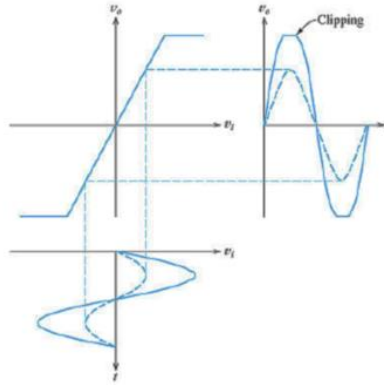


Figure 1 (a) and (b)

The most basic function of the op amp is the voltage amplification. However, the output voltage of a real op amp is limited to the range between certain limits that depend on the internal design of the op amp. As shown in Figure 2, when the output voltage tries to exceed these limits, clipping occurs.



**Figure 2**

### Equipment:

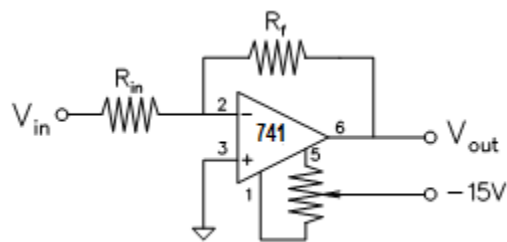
1. Digital multimeter
2. Variable DC power supply
3. Protoboard

### Components

1.  $100\text{k}\Omega$
2.  $10\text{k}\Omega$
3. 741 op-amp

### Procedure

#### Inverting amplifier



**Figure 3**

1. The input offset voltage of op-amps can introduce significant output errors. Many op-amps (351, 741) have additional pins for adjusting the offset to zero. Wire the circuit shown with  $R_f = 100\text{ k}\Omega$  and  $R_{in} = 10\text{ k}\Omega$  (gain  $\approx 10$ ); connect input to common, and adjust the balance potentiometer until the op-amp output is nearly zero ( $\leq 1\text{ mV}$ ). Set DMM to an appropriate scale. Prior to every other experiment in this lab, check in the same manner whether the op-amp remains balanced (it should!)
2. For five or more values of  $V_{in}$ , in the range  $\pm 0.7\text{ V}$  calculate the value of  $V_{out}$  using the following formula for voltage gain of Inverting amplifier and write them in Table 1:

$$A_v = V_{out}/V_{in} = -R_f/R_{in}$$

- Measure the value of  $V_{out}$  for each value of  $V_{in}$  as mentioned above, using a DC voltmeter and write the results in Table 1. Find the % age error.

$V_{in}$	Calculated $V_{out}$	Measured $V_{out}$	% error

**Table 1**

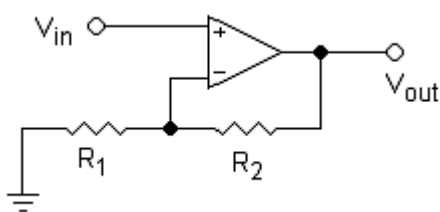
- Set the Function generator at a frequency of 1 kHz and apply as input  $V_{in}$  to the inverting amplifier. Use the two channels of the scope to monitor the inverting input  $V_{in}$  of the op-amp and the output  $V_{out}$ . Slowly increase the amplitude of the input signal, starting near zero. Observe the phase difference between the input and output. Keeping the amplitude of the input low and constant, vary its frequency. Observe the reduction in output amplitude as frequency increases.

#### Non – Inverting Amplifier:

- Set up the non-inverting amplifier circuit of Figure 4 with  $R_1 = 10\text{ k}$ . With a 1 kHz sinusoidal input having different amplitudes, calculate the output with  $R_2 = 100\text{ k}$  and with  $R_2 = 10\text{ k}$  using the formula and write the results in front of each input in Table 2:

$$A_v = V_{out}/V_{in} = 1 + R_2/R_1$$

- Measure the output with an oscilloscope and write them in front of each input in the table. Find the % age error.



$V_{in}$	Calculated $V_{out}$	Measured $V_{out}$	% error

**Table 2**

## Lab 8

### Operational Amplifier Applications-Inverting Summing Amplifier and Difference Amplifier

#### OBJECTIVES:

To demonstrate the use of Operational Amplifier for performing mathematical operations of summation and difference.

#### EQUIPMENT:

1. DC Power Supply
2. Oscilloscope
3. Function Generator

#### Components

1. LM 741 Op-amp
2. 47k $\Omega$
3. 100k $\Omega$

### Part A

#### Inverting Summing Amplifier

#### Theory Overview

Figure 1 shows an example of how an operational amplifier is connected to perform voltage summation. In this figure, an ac and a dc voltage are summed. In general,

$$V_o = -\left(\frac{R_f}{R_1} V_1 + \frac{R_f}{R_2} V_2 + \dots \text{etc.}\right)$$

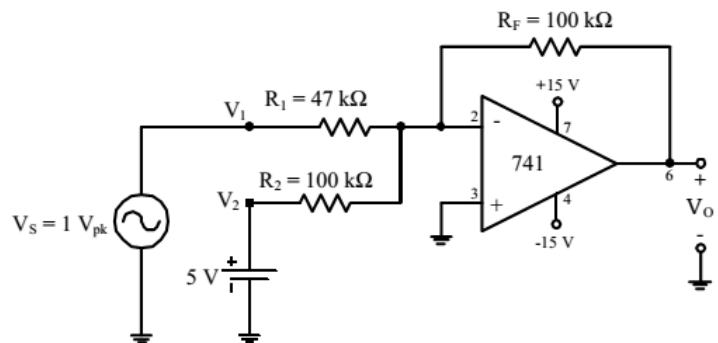


Figure 1

#### Procedure

1. To demonstrate the use of an operational amplifier as a summing amplifier, connect the circuit of Figure 1.

2. With  $V_S$  adjusted to produce a 1 V peak sine wave at 1 kHz, observe the output voltage  $V_O$  (and  $V_S$  to note the phase relationship) on an oscilloscope set to dc input coupling.
3. Sketch the output voltage waveform. Be sure to note the dc level in the output.
4. Interchange the 5 V dc power supply and the 1 V peak signal generator.
5. Repeat procedure step 2 and observe the change in output waveform.

## Part B

### Difference Amplifier

#### Theory Overview

A difference amplifier has two inputs and the output voltage is proportional to the voltage difference of the input voltages. In fact, the (open-loop) Op-Amp itself is a difference amplifier, except that the gain is ideally infinity. Here we want a difference amplifier with finite gain. One such circuit using a single Op-Amp is shown in Figure 4. It can be shown that the gain of the difference amplifier can be calculated using the following:

$$V_O = \left( V_2 \left( 1 + \frac{R_f}{R_1} \right) \left( \frac{R_3}{R_2 + R_3} \right) \right) - \left( \frac{R_f}{R_1} V_1 \right)$$

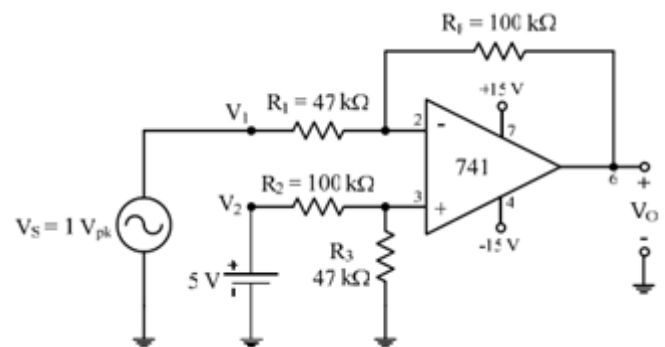


Figure 2

This equation can be simplified by making  $R_3 = R_f = R_1 = R_2$ , yielding a simple differential amplifier with unity gain:

$$V_O = V_2 - V_1$$

#### Procedure

1. To investigate the use of an operational amplifier in a difference amplifier configuration, connect the circuit of Figure 2.
2. With  $V_S$  adjusted to produce a 1 V peak sine wave at 1 kHz, observe the output voltage  $V_O$  (and  $V_S$  to note the phase relationship) on an oscilloscope set to dc input coupling.
3. Sketch the output voltage waveform. Be sure to note the dc level in the output.
4. Interchange the 5 V dc power supply and the 1 V peak signal generator.
5. Repeat procedure step 2 and observe the change in output waveform.

## Lab 9

### Series Resonance Circuit

#### Objectives:

The response of a circuit containing both inductors and capacitors in series or in parallel depends on the frequency of the driving voltage or current. This laboratory will explore one of the more dramatic effects of the interplay of capacitance and inductance, namely, resonance, when the inductive and capacitive reactances cancel each other. Resonance is the fundamental principle upon which most filters are based — filters that allow us to tune radios, televisions, cell phones, and a myriad of other devices deemed essential for modern living.

#### EQUIPMENT:

1. Function generator
2. Oscilloscope
3. Digital Multimeter

#### Components:

1. Resistor, 100  $\Omega$
2. Resistor, 10  $\Omega$
3. Inductor, 100mH
4. Capacitor, 0.01  $\mu\text{F}$

#### Background

The reactance of inductors increases with frequency:  $X_L = 2\pi fL$

The reactance of capacitors decreases with frequency:  $X_C = \frac{1}{2\pi fC}$

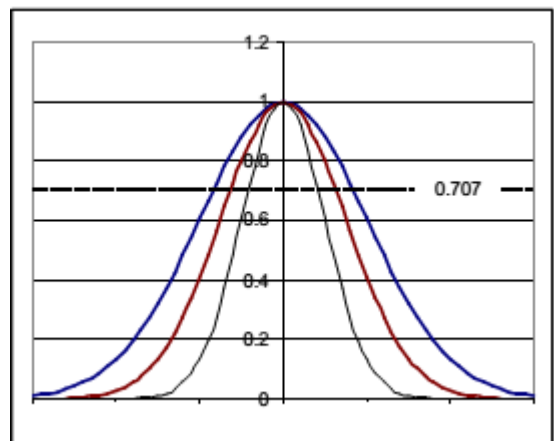
In an LC circuit, whether series or parallel, there is some frequency at which the magnitudes of these two reactances are equal. That point is called resonance. Setting  $X_L = X_C$ , and solving for  $f$ , we find that the resonant frequency  $f_0$  of an LC circuit is:  $f_0 = \frac{1}{2\pi\sqrt{LC}}$

The frequency  $f$  has units cycles/second or  $\text{sec}^{-1}$ . The frequency may also be expressed as angular frequency,  $\omega$ , where  $\omega = 2\pi f$  and has units radians/sec. Thus, the resonant frequency may also be written as:

$$\omega_0 = 2\pi f_0 = \frac{1}{\sqrt{LC}}$$

The resonant frequency is generally the highest point of a peak (or the deepest point of a valley) with bandwidth BW (cycles/sec) or  $\beta$  (radians/sec). The resonant frequency is also called the center frequency, because it is at the mid-point of the peak frequency response.

The lowest frequency ( $f_1$  or  $\omega_1$ ) and the highest frequency ( $f_2$  or  $\omega_2$ ) of the band are the “half-power points” at which the power is  $\frac{1}{2}$  that at the peak frequency. Since power goes like the square of the current, the current at the half-power points is  $1/\sqrt{2}$  ( $= 0.707$ )





times the current at the maximum. Thus, the bandwidth of a resonant circuit is the frequency range over which the current is at least 70.7% of the maximum.

$$BW = f_2 - f_1 \text{ or } \beta = \omega_2 - \omega_1$$

As the bandwidth narrows, the circuit becomes more highly selective, responding to a narrow range of frequencies close to the center frequency. The sharpness (narrowness) of that resonant peak is measured by the quality factor  $Q$ . The quality factor is a unitless quantity that is defined as

$$Q = 2\pi \left[ \frac{\text{maximum energy stored}}{\text{energy dissipated per cycle}} \right]$$

In more practical terms,

$$Q = \frac{f_0}{BW} \text{ or } Q = \frac{\omega_0}{\beta}.$$

### Series Resonance

For a series LC circuit, the current is the same throughout. What about the voltages? To visualize the concept of resonance, consider the simple series RLC circuit in Figure 1 operating at resonance, and its associated reactance diagram.

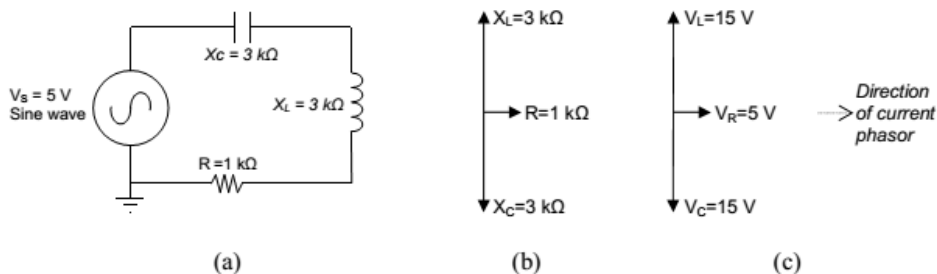


Figure 1

The phase shift caused by the capacitor is directly opposite the phase shift caused by the inductor; that is, they are  $180^\circ$  out of phase. Therefore, in the reactance phasor diagram (b) for the circuit, the two phasors point in opposite directions. At resonance, the magnitudes of the capacitor reactance and the inductor reactance are equal, so the sum of the two phasors is zero, and the only remaining impedance is due to the resistor. Notice in the voltage phasor diagram (c) that the voltage drop across the inductor and the capacitor may be quite large — bigger even than the source voltage — but those voltages are opposite in phase and so cancel each other out as voltages are summed around the circuit. Kirchhoff's voltage law remains valid, and the generator's voltage output is dropped entirely over the resistor  $R$ . Since at resonance the only impedance is the resistance  $R$ , the impedance of the series circuit is at a minimum, and so the current is a maximum. That current is  $V_s/R$ . The source voltage and the current are in phase with each other, so the **power factor** = 1, and maximum power is delivered to the resistor. But what happens at neighboring frequencies? At lower frequencies, the inductor's reactance decreases, and the capacitor has greater effect. At higher frequencies, the inductor dominates, and the circuit will take on inductive characteristics. How sharply defined is the resonance? How selective is it? We have said that for a resonant circuit, the quality factor  $Q$  is the ratio of the resonant frequency to the bandwidth. Thus,  $Q$  gives a measure of the bandwidth normalized to the frequency, thereby describing the shape of the circuit's response independent of the actual resonant frequency.

$$Q = \frac{f_0}{BW}$$

We list here two other useful relationships for  $Q$  in a series resonant circuit. The first relates  $Q$  to the circuit's capacitance, inductance, and total series resistance.

$$Q = \frac{1}{R} \sqrt{\frac{L}{C}}$$

The value of R in this equation is the total equivalent series resistance in the circuit. This form of the equation makes it easy to see ways to optimize the Q for the desired circuit. Decreasing R, increasing inductance, or decreasing capacitance will all tend to make Q larger and increase the circuit's selectivity. The second useful relationship for Q can be derived from the previous equation. Recall that  $X_L = 2\pi fL$  and  $X_C = \frac{1}{2\pi fC}$ . Then the previous equation can be rewritten as  $Q = \frac{1}{R} \sqrt{X_L \cdot X_C}$ .

Since at resonance the inductive and capacitive reactances are equal, this equation can be reduced to  $Q = \frac{X_L}{R}$  or  $Q = \frac{X_C}{R}$

Where R is again the total equivalent series resistance of the circuit. Usually the  $X_L$  form is used because the resistance of the inductor frequently is the dominant resistance in the circuit. An equivalent form of this last equation is  $Q = \frac{2\pi f_0 L}{R}$  or  $Q = \frac{1}{2\pi f_0 CR}$

#### Summary of the Characteristics of RLC Series circuit

Characteristic	Series circuit
Resonant Frequency, $f_o$	$\frac{1}{2\pi\sqrt{LC}}$
Quality factor, Q	$\frac{2\pi f_o L}{R}$ or $\frac{1}{2\pi f_o RC}$
Bandwidth, BW	$\frac{f_o}{Q}$
Half-power frequencies $f_1, f_2$	$f_o \sqrt{1 + \left(\frac{1}{2Q}\right)^2} \pm \frac{f_o}{2Q}$
For $Q \geq 10$ , $f_1, f_2$	$f_o \pm \frac{BW}{2}$

**Table 1**

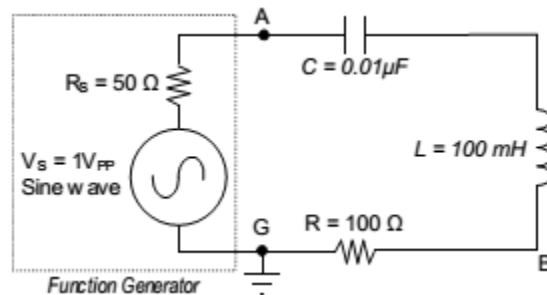
## PROCEDURE

- Using DMM, measure the values of the following components: 0.01 $\mu$ F capacitor; 100  $\Omega$  resistor; 10  $\Omega$  resistor. Also measure the winding resistance  $R_w$  of the 100mH inductor. Record the nominal and measured values in Table 2

	Nominal	Actual
$L$	100 mH	
$C$	0.01 $\mu\text{F}$	
$R_W$		
$R_I$	100 $\Omega$	
$R_2$	10 $\Omega$	

**Table 2**

- For the circuit shown in Figure 2, calculate predictions for  $f_0$ ,  $Q$ ,  $BW$ ,  $f_1$ , and  $f_2$ . Don't forget to include the impedance of the function generator ( $R_S \approx 50\Omega$ ) and  $R_W$  as part of the total resistance in the circuit. Record the results in the first "predicted" column in a table such as Table 3.



**Figure 2**

- Construct the circuit shown in Figure 2. Adjust the function generator to generate a sine wave with voltage 1.0  $V_{pp}$ . Initially set the frequency to 1 kHz.
- Connect oscilloscope CHANNEL 1 across the function generator (FGEN and GND) and confirm that the voltage is 1.0  $V_{pp}$ .
- Connect oscilloscope CHANNEL 2 across the resistor  $R$  and observe the voltage.
- Using your predicted values as a guide, adjust the frequency of the function generator to tune for resonance, as observed on CHANNEL 2 of the oscilloscope. Measure the resonant frequency  $f_0$  on the oscilloscope, and record the value in the first "measured" column of Table 3.

### Experimental Values

$L$				
$C$				
$R_W$				
$R$	100 $\Omega$		10 $\Omega$	
$R_{total}$				
	Predicted	Measured	Predicted	Measured
$f_o$				
$Q$				
$BW$				
$f_1$				
$f_2$				

**Table 3.**

- Confirm that the voltage on CHANNEL 1 of the scope is 1.0 V<sub>pp</sub>, and adjust it if necessary. The current through the circuit and resistor R is proportional to the voltage across R. Record the voltage across resistor R

For steps 8 and 9, DO NOT adjust the voltage output of the function generator.

- Reduce the frequency on the function generator until the voltage across R is 70.7% of the initial value. This is the lower half-power point  $f_1$ . Record the measured frequency  $f_1$  in the first “measured” column of Table 3.
- Increase the frequency through resonance and continue to increase it until the voltage across R is 70.7% of the value at resonance. This is the upper half-power point  $f_2$ . Record the measured frequency  $f_2$  in the first “measured” column of Table 3.
- Calculate the bandwidth  $BW = f_2 - f_1$ . Record the result in the first “measured” column of Table 3.
- Stop the function generator. Remove the 100 $\Omega$  resistor from the circuit and replace it with the 10  $\Omega$  resistor measured earlier.
- Calculate predictions for  $f_o$ ,  $Q$ ,  $BW$ ,  $f_1$ , and  $f_2$  and record the results in the second “predicted” column in Table 3.
- Start the function generator and, as before, adjust the function generator to create a sine wave with voltage 1.0 V<sub>pp</sub>.
- Repeat steps 5 through 10, recording the measured values in the second “measured” column.
- Fill out Table 4, calculating the percent differences between predicted and measured values.

### Percent difference

$\% \text{ difference} \left[ = 100\% \left( \frac{\text{measured} - \text{predicted}}{\text{predicted}} \right) \right]$		
	R = 100 $\Omega$	R = 10 $\Omega$
$f_o$		
$Q$		
$BW$		
$f_1$		
$f_2$		

**Table 4**

## Lab 10

### Filters: High-pass, Low-pass, Band-pass, and Notch

#### INTRODUCTION:

This laboratory studies the use of passive components to create filters to separate portions of time-dependent waveforms. Filters are an essential tool in our complex world of mixed signals — both electronic and otherwise. Passive components (resistors, capacitors, and inductors) have long served as filter components for everything from selecting radio stations to filtering out electrical noise.

#### OBJECTIVES:

1. Learn the four general filter types: High-pass, Low-pass, Band-pass, and Notch
2. Learn to alter filter type by changing contacts for output voltage.
3. Learn phase angle at cutoff for simple RC and RL filters.
4. Learn to draw Bode Plots.

#### Equipment:

1. Function generator
2. Oscilloscope
3. Digital Multimeter

#### Components:

1. Resistors: 10k $\Omega$ , 100  $\Omega$
2. Inductor: 100mH
3. Capacitor: 0.005  $\mu$ F, 0.01  $\mu$ F

#### BACKGROUND

In many circuits, a wide range of different frequencies are present, some of which are desired, while others are not. The frequency response of capacitors and inductors allows us to construct filters that will pass or reject certain ranges of the electrical frequencies that are applied to them.

"Passive filters" created from "passive" components (inductors, capacitors, and resistors) have served us well for a long time for such purposes as selecting radio and television stations and filtering noise out of various signals. Indeed, much of the electronics we take for granted today would not be possible without the use of such filters.

The four typical types of filter behaviors are illustrated in Figure 1, along with schematics of simple filters that exhibit the indicated behavior.

## Types of Filters

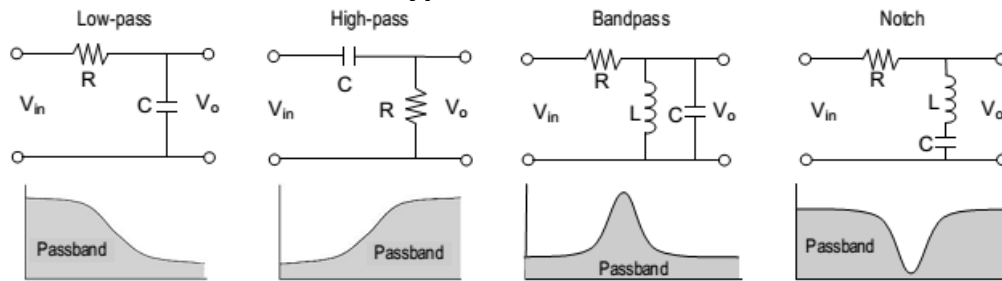


Figure 1

The filter types are low-pass, high-pass, band-pass, and notch (or band-reject) filters. In Figure 1, the grayed area is the pass-band, that is, the part of the signal that is passed to the output of the filter. The rejected portions are called the stop-band. The frequency that separates the pass-band from the stop-band is called the cutoff frequency. The cutoff frequency is equivalent to the half-power points. The cutoff frequency is also sometimes called the corner frequency.

A low-pass filter would allow extracting a low frequency, such as an audio signal, that is mixed with a high frequency radio wave. A high-pass filter would do the opposite. A resonant circuit can be tuned as a band-pass filter to retain signals in a narrow range of frequencies, while rejecting frequencies outside that range. Such is the case with a radio tuner. A notch filter generally keeps all frequencies except those in a narrow band. Notch filters are widely used to block interfering signals from noise sources. Band-pass and notch filters require resonant circuits.

Notice that the components making the low-pass and high-pass filters in Figure 1 are the same. Whether the circuit is low-pass or high-pass depends only upon which voltage we look at: the voltage across the capacitor or the voltage across the resistor. (Equivalent circuits could have been made using an inductor and a resistor.) Similarly, the notch filter is identical to the RLC series resonant circuit. RC and RL filters are simple, inexpensive, and often used effectively as filters. Their major problem is their generally slow (in frequency) transition from pass-band to stop-band. The addition of a few simple components in filter “stages” can increase the transition rate, giving the filter a sharper cutoff. The ratio of an output response to an input signal is referred to as a transfer function. The input signal and the output response do not need to be the same entity type. For example, a transfer function may prescribe an output voltage resulting from an input current. Transfer functions are often used as a tool to characterize the effect of a filter regardless of the details of the filter’s structure. It can make the analysis of complex circuits easier. In this lab, however, we will mostly be studying the filter itself.

### Cutoff Frequency for series RC and RL circuits

As mentioned, the cutoff frequency, sometimes called the corner frequency, is equivalent to the half-power points. Since the power is half that at the peak, the voltage (or current) will be the peak voltage (or current) multiplied by  $1/\sqrt{2} = 0.707$ . For a simple 2-component RC or RL circuit, the half-power point will occur when half the power is dropped on the resistor and half on the capacitor or inductor. Thus, the cutoff frequency will occur when the reactance of the capacitor or inductor equals the total series resistance in the circuit. That is,  $X_C = \frac{1}{2\pi f_c C} = R$  and  $X_L = 2\pi f_c L = R$  and so,

$$f_c = 1/2\pi RC$$

and

$$f_c = R/2\pi L$$

## decibels (dB)

As discussed in your textbook, the decibel (dB) is commonly used for the magnitude of voltages and currents in making Bode plots. Keep in mind that a decibel is a unit created to measure the transfer function for power gain (or loss) through a circuit module or stage:

$$\text{Number of decibels} = 10 \log_{10} \left[ \frac{\text{power output}}{\text{power input}} \right]$$

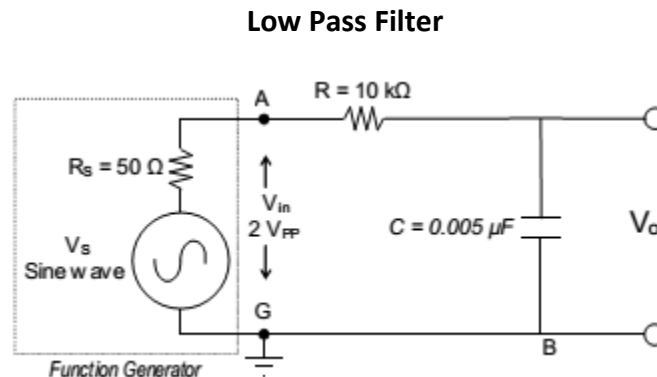
Since power is proportional to the square of the voltage or the current, we have equivalently,

$$\text{Number of decibels} = 20 \log_{10} \left[ \frac{V_{out}}{V_{in}} \right] \quad \text{and} \quad \text{Number of decibels} = 20 \log_{10} \left[ \frac{I_{out}}{I_{in}} \right]$$

## Procedure:

### Low-pass filter.

1. Obtain a 10k $\Omega$  resistor and a 0.005  $\mu$ F capacitor. Measure and record the actual values of the components.
2. Using the measured components, set up the circuit as shown in Figure 2. Use the function generator FGEN for the supply voltage.



**Figure 2**

3. Calculate the cutoff frequency for the circuit, assuming the output is at  $V_o$ . At the Cutoff frequency, what, theoretically, will be the voltage  $V_o$ ?
4. Connect CHANNEL 1 of the oscilloscope to measure the  $V_{in}$  (i.e., FGEN).
5. Connect CHANNEL 2 of the oscilloscope to measure the filter's output voltage  $V_o$ . Vary the frequency from 500 Hz to 10 kHz in steps indicated in Table 1, and record the indicated values.
6. Measure the phase shift  $\Delta t$  and then calculate the phase angle  $\phi$  between  $V_{in}$  and  $V_o$  at 500 Hz, at 10000 Hz, and at the cutoff frequency.
7. Using the data of Table 1, sketch a Bode plot of the of the filter's output voltage.



Freq. kHz	Actual $f$ kHz	$V_{in}$ RMS	$V_{in}$ PP	$V_o$ RMS	$V_o$ PP	$\Delta t$ $\mu s$	$\phi$ degrees
0.500							
1.000							
2.000							
4.000							
6.000							
8.000							
10.000							
Cutoff							

Table 1

### Band-Pass Filter

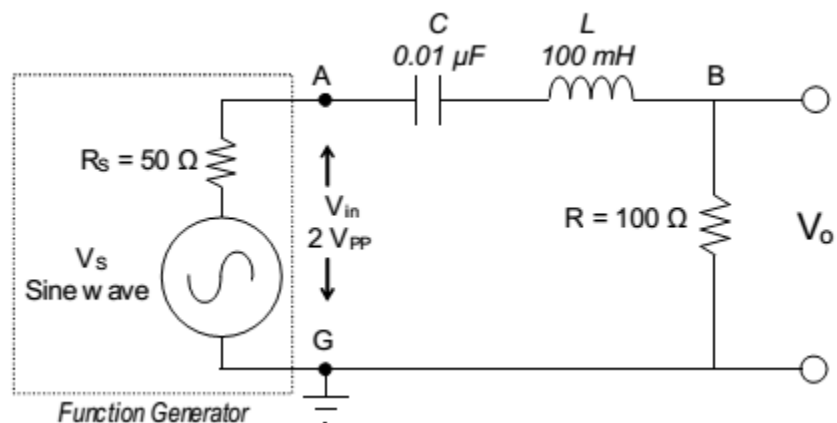
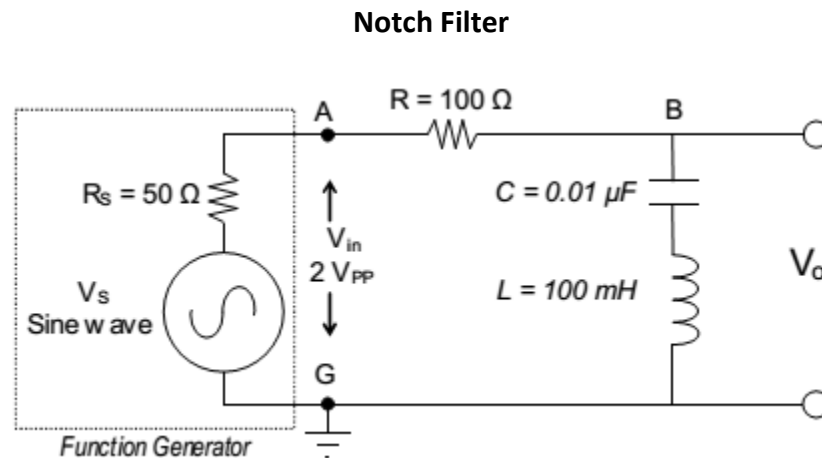


Figure 3

8. Set up the series RLC circuit shown in Figure 3, using the function generator to provide the sinusoidal input voltage.
9. Calculate the resonant frequency  $f_0$  of the circuit. (See Laboratory 9.)
10. Set the "peak amplitude" to 1.0 V (that is, 2.0  $V_{pp}$ ).
11. What is the measured resonant frequency? \_\_\_\_\_
12. What is the measured bandwidth? \_\_\_\_\_
13. What is the phase angle at resonance? \_\_\_\_\_
14. What is the phase angle at the two cutoff frequencies? \_\_\_\_\_
15. Sketch the Magnitude and Phase angle Bode plots, marking key reference points.

### Notch Filter

16. Switch the positions of the resistor with inductor and capacitor to get the series RLC circuit shown in Figure 4.



**Figure 4**

17. Calculate the resonant frequency  $f_0$  of the circuit. \_\_\_\_\_
18. What is the measured resonant frequency  $f_c$ ? \_\_\_\_\_
19. What are the lower and upper half-power points,  $f_1$  \_\_\_\_\_ and  $f_2$ ? \_\_\_\_\_
20. What is the measured bandwidth? \_\_\_\_\_
21. What is the phase angle  $\phi$  at resonance? \_\_\_\_\_
22. Draw the Bode plot.

## Lab 11

### INTEGRATOR USING IC741 OP-AMP

#### Objective

To study the operation of the Integrator using op-amp and trace the output wave forms for sine and square wave inputs.

#### THEORY:

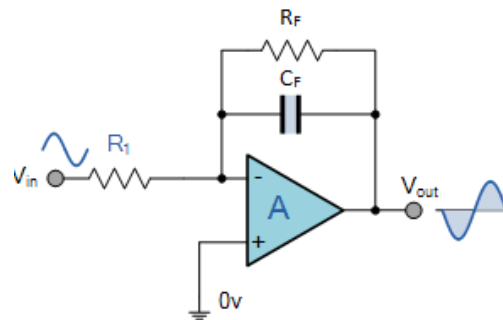


Figure 1

A circuit in which the output voltage is the integration of the input voltage is called an integrator.

$$V_o = -\frac{1}{R_1 C_F} \int V_{in} dt$$

In the practical integrator shown in Figure 1, to reduce the error voltage at the output, a resistor  $R_F$  is connected across the feedback capacitor  $C_F$ . Thus,  $R_F$  limits the low-frequency gain and hence minimizes the variations in the output voltage.

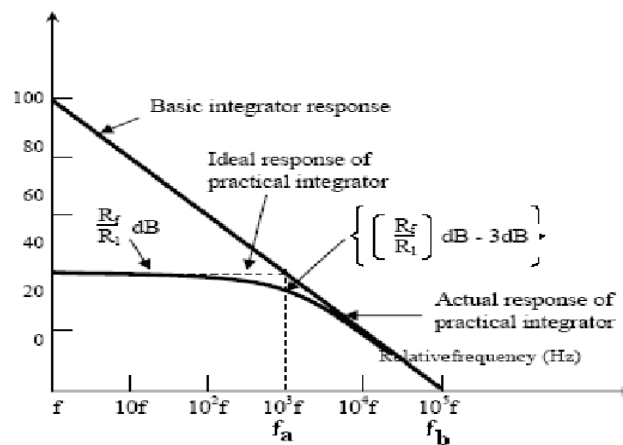


Figure 2

The frequency response of the integrator is shown in Figure 2.  $f_b$  is the frequency at which the gain is 0 dB and is given by:

$$f_b = 1/2\pi R_1 C_F$$

In this figure there is some relative operating frequency, and for frequencies from  $f$  to  $f_a$  the gain  $R_F/R_1$  is constant. However, after  $f_a$  the gain decreases at a rate of 20 dB/decade. In other words, between  $f_a$  and  $f_b$  the circuit of fig. 2.1 acts as an integrator. The gain limiting frequency  $f_a$  is given by

$$f_a = 1/2\pi R_F C_F.$$

Normally  $f_a < f_b$ . From the above equation, we can calculate  $R_F$  by assuming  $f_a$  &  $C_F$ . This is very important frequency. It tells us where the useful integration range starts.

- If  $f_{in} < f_a$  - circuit acts like a simple inverting amplifier and no integration results,
- If  $f_{in} = f_a$  - integration takes place with only 50% accuracy results,
- If  $f_{in} = 10f_a$  - integration takes place with 99% accuracy results.

In the circuit diagram of Integrator, the values are calculated by assuming  $f_a$  as 50 Hz. Hence the input frequency is to be taken as 500Hz to get 99% accuracy results. Integrator has wide applications in

1. Analog computers used for solving differential equations in simulation arrangements.
2. A/D Converters.
3. Signal wave shaping.
4. Function Generators.

### Equipment:

1. Oscilloscope
2. AC Function Generator
3. Digital Multimeter

### Components:

1. Resistors: 10k $\Omega$ , 22k $\Omega$
2. Capacitor 0.1 $\mu$ F
3. Op-amp 741

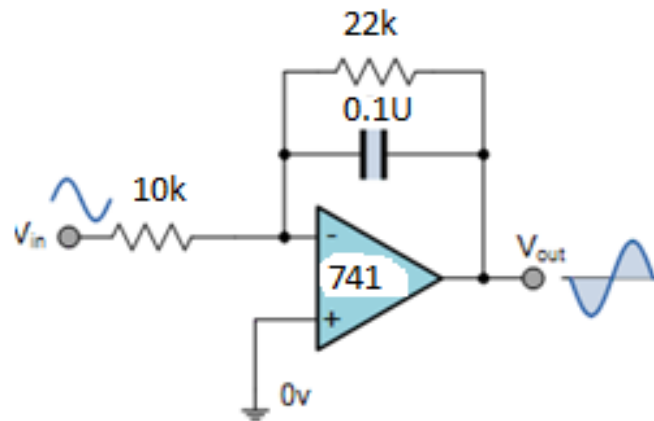


Figure 3

### PROCEDURE:

1. Connect the components/equipment as shown in the circuit diagram Figure 3.
2. Switch ON the power supply.
3. Apply sine wave at the input terminals of the circuit using function Generator.
4. Connect channel-1 of CRO at the input terminals and channel-2 at the output terminals.
5. Observe the output of the circuit on the CRO which is a cosine wave (90° phase shifted from the sine wave input) and note down the position, the amplitude and the time period of  $V_{in}$  &  $V_o$ .
6. Now apply the square wave as input signal.
7. Observe the output of the circuit on the CRO which is a triangular wave and note down the position, the amplitude and the time period of  $V_{in}$  &  $V_o$ .
8. Plot the output voltages corresponding to sine and square wave inputs as shown in the Figure 4 below.

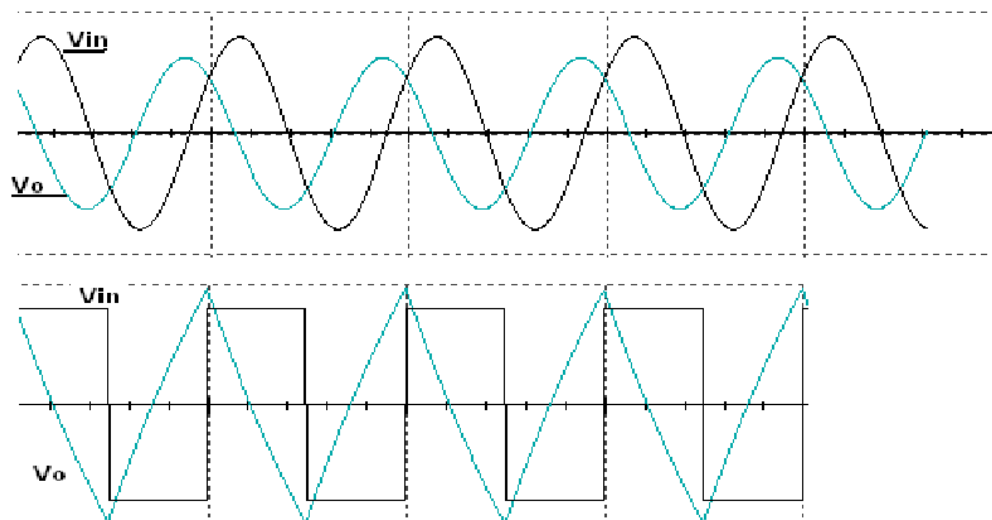


Figure 4

## Active Filters

### Theory

An electric filter is a frequency-selecting circuit designed to pass a specified band of frequencies while attenuating signals of frequencies outside this band. Filters may be either active or passive depending on the type of elements used in their circuitry. Passive filters contain only resistors, capacitors, and inductors. Active filters employ transistors or op-amps in addition to resistors and capacitors. Active filters offer several advantages over passive filters. Since the op-amp is capable of providing a gain, the input signal is not attenuated as it is in a passive filter. Because of the high input and low output resistance of the op-amp, the active filter does not cause loading of the source or load. There are four types of filters: low-pass, high-pass, band-pass, and band-reject filters. A low-pass filter has a constant gain ( $=V_{out}/V_{in}$ ) from 0 Hz to a high cut off frequency  $f_H$ . This cut off frequency is defined as the frequency where the voltage gain is reduced to 0.707, that is at  $f_H$  the gain is down by 3 dB; after that ( $f > f_H$ ) it decreases as  $f$  increases. The frequencies between 0 Hz and  $f_H$  are called pass band frequencies, whereas the frequencies beyond  $f_H$  are the so-called stop band frequencies. A common use of a low-pass filter is to remove noise or other unwanted high-frequency components in a signal for which you are only interested in the dc or low frequency components. Low-pass filters are also used to avoid aliasing in analog-digital conversion correspondingly, a high-pass filter has a stop band for  $0 < f < f_L$  and where  $f_L$  is the low cut off frequency. A common use for a high-pass filter is to remove the dc component of a signal for which you are only interested in the ac components (such as an audio signal). A band pass filter has a pass band between two cut off frequencies  $f_H$  and  $f_L$ , ( $f_H > f_L$ ), and two stop bands  $0 < f < f_L$  and  $f > f_H$ . The bandwidth of a band pass filter is equal to  $f_H - f_L$ . The actual response curves of the filters in the stop band either steadily decrease or increase with increase of frequency. The roll-off rate, measured at [dB/decade] or [dB/octave] is defined as rate change of power at 10 times (decade) or 2 times (octave) change of frequency in the stop band. The “First-order” filters attenuate voltages in the stop band 20 dB/decade (for example, a first-order low pass filter would attenuate a signal at a frequency 100 times (2 decades) higher than  $f_H$  by 40 dB. The second-order filters attenuate by about 40 dB/decade.

## Lab 12

### Low Pass Filter

#### Objectives :

To study the Active Low pass filter and to evaluate :

- High cutoff frequency of Low pass filter.
- Pass band gain of Low pass filter.
- Plot the frequency response of Low pass filter.

#### Equipment:

1. DC power supplies +15V, -15V from external source
2. Function generator
3. Oscilloscope
4. Digital Multimeter

#### Components:

1. Resistance 10k $\Omega$
2. Resistance 22k $\Omega$
3. Capacitor 0.01 $\mu$ F
4. LM 741

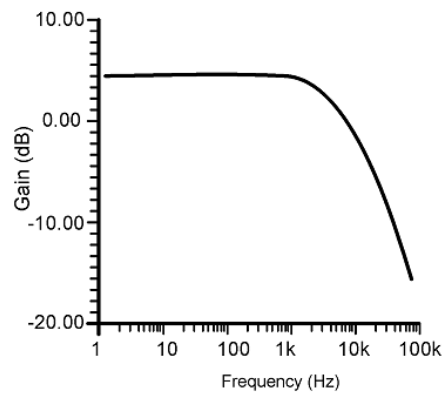
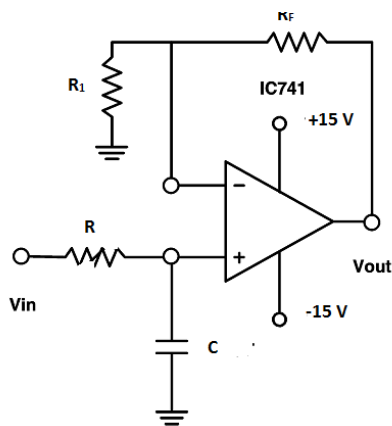


Figure 1

#### Equation of low pass filter

$$\frac{V_{out}}{V_{in}} = \frac{A_F}{1+j(f/f_h)} \quad 1$$

$$\frac{V_{out}}{V_{in}} = \frac{A_F}{\sqrt{1 + \left(\frac{f}{f_h}\right)^2}} \quad 2$$

$V_{in}$  = Input signal Voltage

$V_{out}$  = Output signal Voltage

$|V_{out}/V_{in}|$  = Gain of filter as a function of frequency

$A_F = 1 + R_F/R_1$  = pass band gain of filter

$f$  = frequency of input signal

$f_H = 1/2\pi RC$  = high cut off frequency, 3-dB frequency, corner frequency

Operation of low pass filter using equation 2

- At low frequencies  $f < f_H$ ;  $\left| \frac{V_{out}}{V_{in}} \right| = A_F$
- At  $f = f_H$   $\left| \frac{V_{out}}{V_{in}} \right| = 0.707 * A_F$  (Approx)
- At  $f > f_H$   $\left| \frac{V_{out}}{V_{in}} \right| < A_F$

The ideal low pass filter has a constant gain  $A_F$  from 0 to high cut off frequency ( $f_H$ ) at  $f_H$  the gain is  $0.707 * A_F$ , and after  $f_H$  it decreases at a constant rate with an increase in frequency i.e. when input frequency is increased tenfold (one decade), the voltage gain is divided by 10.  
Gain (db) =  $20 \log |V_{out} / V_{in}|$  i.e. Gain Roll off rate is  $-20\text{db} / \text{decade}$ .

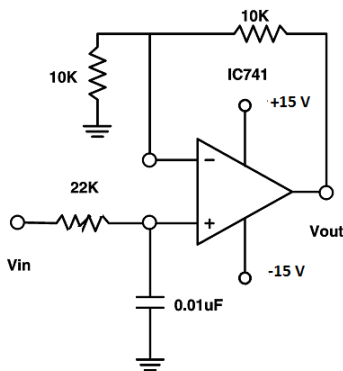


Figure 2

Sr. No.	Input frequency (Hz)	Vout	$ V_{out} / V_{in}  = \text{gain}$	Gain(db) = $20 \log  v_{out} / v_{in} $
1	500			
2	1 K			
3	5 K			
4	10 K ( $f_H$ )			
5	15 K			
6	20 K			
7	30 K			

Table 1



## Procedure :

1. Connect the circuit as shown in Figure 2.
2. Switch ON the power supply
3. Connect a sinusoidal signal of amplitude 1V (p-p) of frequency 1KHz to  $V_{in}$  of Low pass filter from function generator
4. Connect Ch-1 of oscilloscope to the signal source
5. Observe output on Ch-2 of oscilloscope
6. Increase the frequency of input signal step by step and observe the effect on output  $V_{out}$  on oscilloscope
7. Tabulate values of  $V_{out}$ , gain, gain (db) at different values of input frequency shown in observation Table 1.
8. Plot the frequency response of low pass filter using the data obtained at different input frequencies.

## Theoretical Calculations :

Calculate all the following values

1. Pass band gain of Low pass filter  $A_F = 1 + R_F / R_1$
2. Pass band gain (db)  $= 20 \log |V_{out} / V_{in}|$
3. 3 db frequency  $f_H = 1/2\pi RC$
4. Gain at 3 db frequency  $f_H = 0.707 * A_F$
5. Roll off rate  $= -20\text{db/decade}$

## Results :

	Theoretical	Practical
Pass band gain( $A_f$ )		
Pass band gain( $A_f$ ) in db		
3db frequency $f_H$		
Gain at 3db frequency ( $f_H$ ) in db		

## Lab 13

### High Pass Filter

It is a frequency selective circuit, which passes signals of frequencies above its low cut off frequency ( $f_L$ ) and attenuates signals of frequencies below  $f_L$ .

### Objectives :

To study the Active High pass filter and to evaluate :

- Low cutoff frequency of High pass filter.
- Pass band gain of High pass filter.
- Plot the frequency response of High pass filter.

### Equipment:

1. DC power supplies +15V, -15V from external source
2. Function generator
3. Oscilloscope
4. Digital Multimeter

### Components:

5. Resistance 10k $\Omega$
6. Resistance 22k $\Omega$
7. Capacitor 0.01 $\mu$ F
8. LM 741

### Equation of High pass filter

$$\frac{V_{out}}{V_{in}} = \frac{A_F}{1+j(f/f_L)} \quad 1$$

$$\frac{V_{out}}{V_{in}} = \frac{A_F}{\sqrt{1 + \left(\frac{f}{f_L}\right)^2}} \quad 2$$

$V_{in}$  = Input signal Voltage

$V_{out}$  = Output signal Voltage

$|V_{out}/V_{in}|$  = Gain of filter as a function of frequency

$A_F = 1 + R_F/R_1$  = pass band gain of filter

$f$  = frequency of input signal

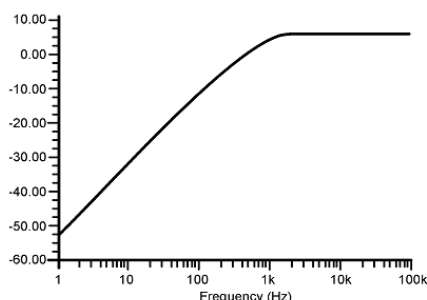
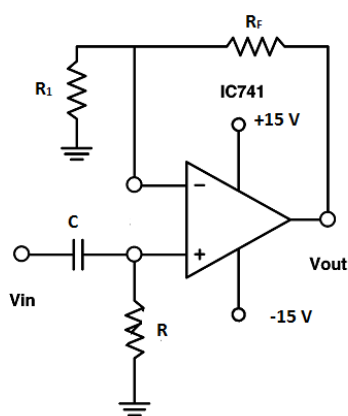
$f_L = 1/2\pi RC$  = Low cut off frequency, 3-dB frequency, corner frequency

Operation of high pass filter using equation 2

d. At low frequencies  $f < f_L$ ;  $\left| \frac{V_{out}}{V_{in}} \right| < A_F$

e. At  $f = f_L$   $\left| \frac{V_{out}}{V_{in}} \right| = 0.707 * A_{F(Approx)}$

f. At  $f > f_L$   $\left| \frac{V_{out}}{V_{in}} \right| = A_F$



In ideal high pass filter, when  $f < f_L$  gain is increased at a constant rate with an increase in frequency. At  $f_L$  the gain is  $0.707 * A_F$ , and above  $f_L$  it has constant gain of  $A_F$ . Below  $f_L$  when input frequency is increased tenfold (one decade), the voltage gain is multiplied by 10.

$$\text{Gain (db)} = 20 \log |V_{out} / V_{in}|$$

i.e. Gain Roll off rate is  $-20\text{db} / \text{decade}$ .

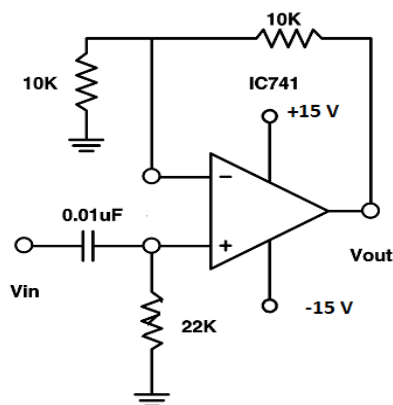


Figure 2

Sr. No.	Input frequency (Hz)	Vout	$ V_{out} / V_{in}  = \text{gain}$	Gain(db) = $20 \log  v_{out} / v_{in} $
1	100			
2	200			
3	500			
4	1K( $f_L$ )			
5	5 K			
6	10 K			
7	15 K			
8	20 K			

Table 1

## Procedure :

9. Connect the circuit as shown in Figure 2.
10. Switch ON the power supply.
11. Connect a sinusoidal signal of amplitude 1V (p-p) of frequency 1KHz to  $V_{in}$  of High pass filter from function generator.
12. Connect Ch-1 of oscilloscope to the signal source.
13. Observe output on Ch-2 of oscilloscope.
14. Increase the frequency of input signal step by step and observe the effect on output  $V_{out}$  on oscilloscope.
15. Tabulate values of  $V_{out}$ , gain, gain (db) at different values of input frequency shown in observation Table 1.
16. Plot the frequency response of High pass filter using the data obtained at different input frequencies.

## Theoretical Calculations :

Calculate all the following values

6. Pass band gain of High pass filter  $A_F = 1 + R_F / R_1$
7. Pass band gain (db)  $= 20 \log |V_{out} / V_{in}|$
8. Low cutoff frequency  $f_L = 1/2\pi RC$
9. Gain at Low cutoff frequency  $f_L = 0.707 * A_F$
10. Roll off rate  $= -20\text{db/decade}$

## Results:

	Theoretical	Practical
Pass band gain( $A_f$ )		
Pass band gain( $A_f$ ) in db		
Low cutoff frequency ( $f_L$ )		
Gain at 3db frequency ( $f_L$ ) in db		

## Lab 14


### Introduction to Pspice

#### **Objectives:**

To become familiar with the use of PSpICE and to learn to use it to assist in the analysis of circuits.


PSpICE stands for “**Personal Computer Simulation Program with Integrated Circuit Emphasis**”.

#### **I. Opening PSpice:**

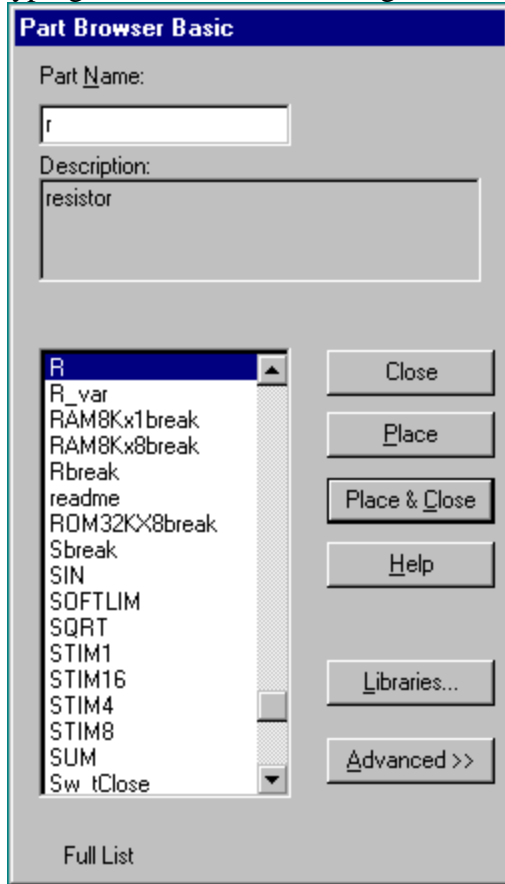
- Find PSpice on the C-Drive. Open Schematics or you can go to PSpice A\_D and then click on the schematic icon .

#### **II. Drawing the circuit:**

##### **A. Getting the Parts:**

- The first thing that you have to do is get some or all of the parts you need.
- This can be done by
  - Clicking on the 'get new parts' button , or
  - Pressing "Control+G", or
  - Going to "Draw" and selecting "Get New Part..."

- Once this box is open, select a part that you want in your circuit. This can be done by typing in the name or scrolling down the list until you find it.




- Some common parts are:
  - o r - resistor
  - o C - capacitor
  - o L - inductor
  - o d - diode
  - o GND\_ANALOG or GND\_EARTH -- this is very important, you MUST have a ground in your circuit
  - o VAC and VDC
- Upon selecting your parts, click on the place button then click where you want it placed (somewhere on the white page with the blue dots). Don't worry about putting it in exactly the right place, it can always be moved later.
- Once you have all the parts you think you need, close that box. You can always open it again later if you need more or different parts.

## B. Placing the Parts:

- You should have most of the parts that you need at this point.
- Now, all you do is put them in the places that make the most sense (usually a rectangle works well for simple circuits). Just select the part and drag it where you want it.

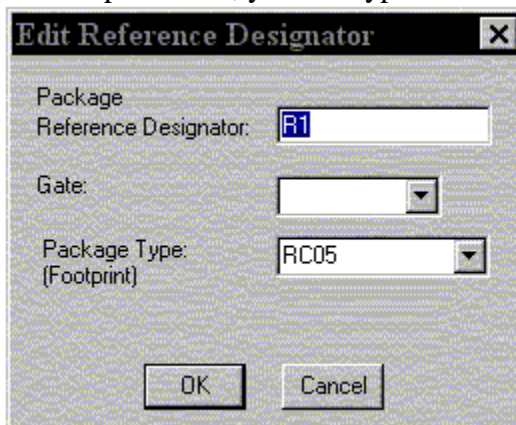
- To rotate parts so that they will fit in you circuit nicely, click on the part and press "Ctrl+R" (or Edit "Rotate"). To flip them, press "Ctrl+F" (or Edit "Flip").
- If you have any parts left over, just select them and press "Delete".

### C. Connecting the Circuit:

- Now that your parts are arranged well, you'll have to attach them with wires.
- Go up to the tool bar and
  - select "Draw Wire"  or
  - "Ctrl+W" or
  - go to "Draw" and select "Wire".
- With the pencil looking pointer, click on one end of a part, when you move your mouse around, you should see dotted lines appear. Attach the other end of your wire to the next part in the circuit.
- Repeat this until your circuit is completely wired.
- If you want to make a node (to make a wire go more then one place), click somewhere on the wire and then click to the part (or the other wire). Or you can go from the part to the wire.
- To get rid of the pencil, right click.
- 21QIf you end up with extra dots near your parts, you probably have an extra wire, select this short wire (it will turn red), then press "Delete".
- If the wire doesn't go the way you want (it doesn't look the way you want), you can make extra bends in it by clicking in different places on the way (each click will form a corner).

### D. Changing the Name of the Part:

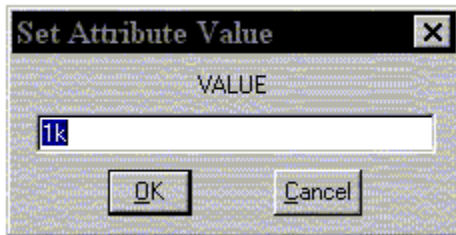
- You probably don't want to keep the names C1, C2 etc., especially if you didn't put the parts in the most logical order. To change the name, double click on the present name (C1, or R1 or whatever your part is), then a box will pop up (Edit Reference Designator). In the top window, you can type in the name you want the part to have.



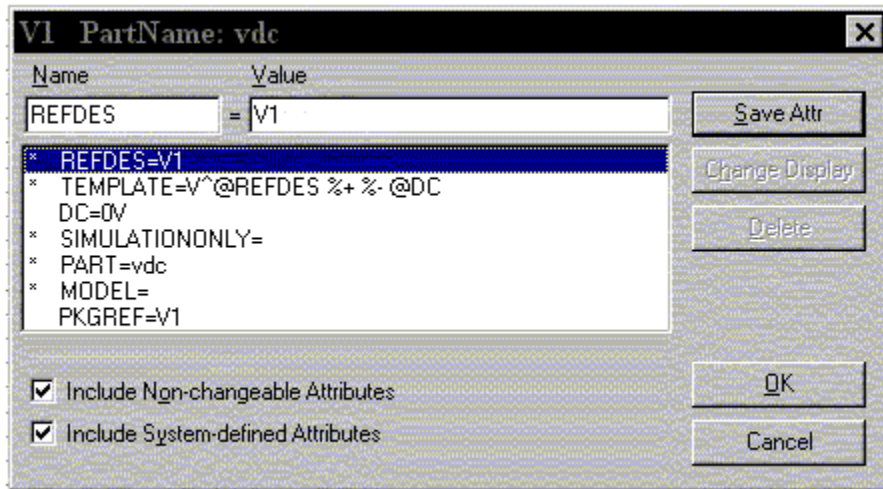
- Please note that if you double click on the part or its value, a different box will appear.

### E. Changing the Value of the Part:

- If you only want to change the value of the part (if you don't want all your resistors to be 1K ohms), you can double click on the present value and a box called "Set Attribute Value" will appear. Type in the new value and press OK. Use u for micro as in uF = microFarad.





- If you double click on the part itself, you can select VALUE and change it in this box.



## F. Making Sure You Have a GND:

- This is very important. You cannot do any simulation on the circuit if you don't have a ground. If you aren't sure where to put it, place it near the negative side of your voltage source.

## G. Voltage and Current Bubbles:


- These are important if you want to measure the voltage at a point or the current going through that point.
- To add voltage or current bubbles, go to the right side of the top tool bar and select "Voltage/Level Marker" (Ctrl+M)  or "Current Marker" . To get either of these, go to "Markers" and either "Voltage/Level Marker" or "Current Marker".

## H. Saving:

- To save the circuit, click on the save button  on the tool bar (or any other way you normally save files).



## **I. Printing:**


- To print, you must first use your mouse to make a rectangle around your circuit, this is the area of the page that will be printed. Then select print as usual. (You can select ).

## **III. Probe:**

### **A. Before you do the Probe:**

- You have to have your circuit properly drawn and saved.
- There must not be any floating parts on your page (i.e. unattached devices).
- You should make sure that all parts have the values that you want.
- There are no extra wires.
- It is very important that you have a ground on your circuit.
- Make sure that you have done the Analysis Setup and that only the things you want are enabled.


### **B. To Start the Probe:**

- Click on the Simulate button on the tool bar  (or Analysis, Simulate, or F11).
- It will check to make sure you don't have any errors. If you do have errors, correct them.
- Then a new window will pop up. Here is where you can do your graphs.

### **C. Graphing:**

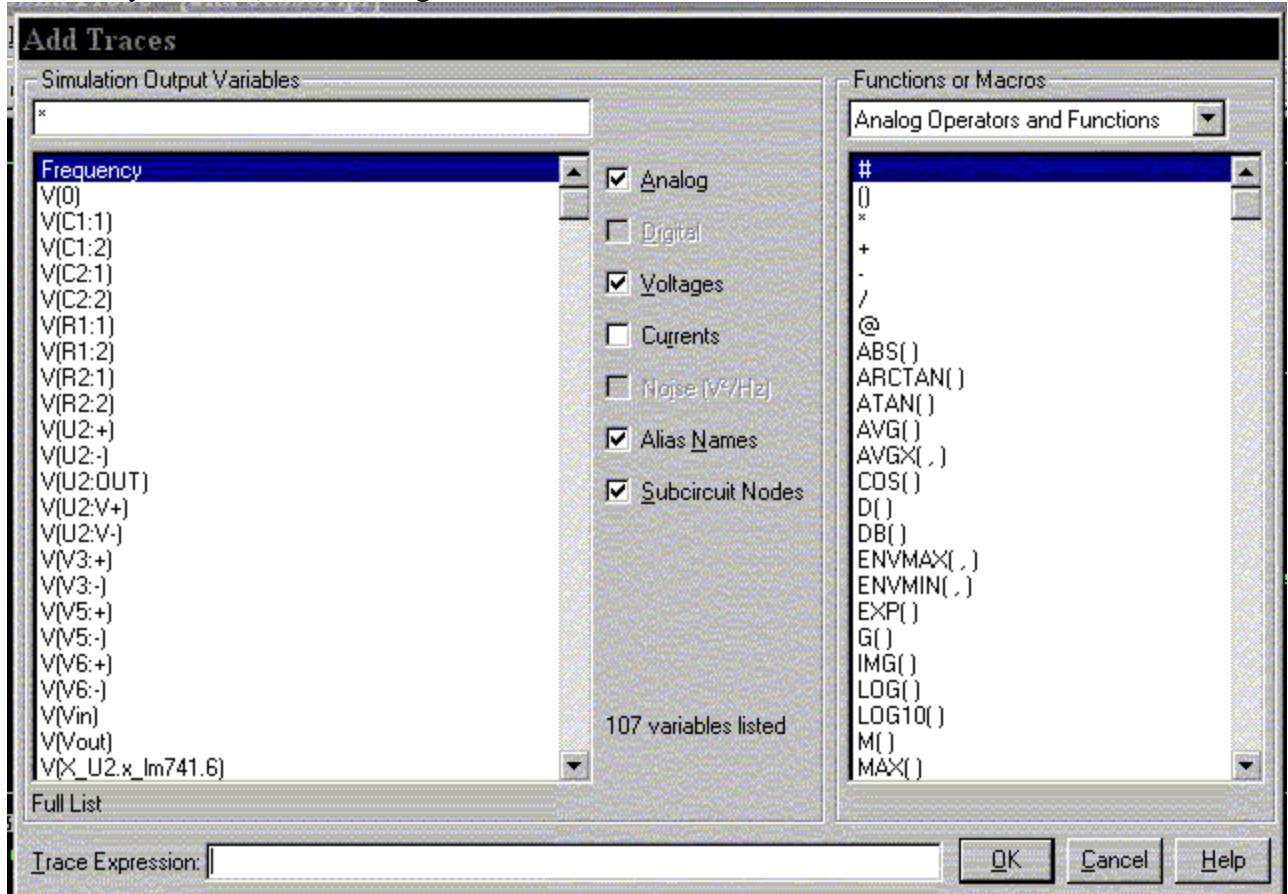
- If you don't have any errors, you should get a window with a black background to pop up.
- If you did have errors, in the bottom, left hand side, it will say what your errors were (these may be difficult to understand, so go To "View - Output File").

### **D. Adding/Deleting Traces:**

- PSpice will automatically put some traces in. You will probably want to change them.
- Go to Trace - Add Trace or  on the toolbar. Then select all the traces you want.
- To delete traces, select them on the bottom of the graph and push Delete.


### **E. Doing Math:**

- In Add Traces, there are functions that can be performed, these will add/subtract (or whatever you chose) the lines together.





- Select the first output then either on your keyboard or on the right side, clicks the function that you wish to perform.
- There are many functions here that may or may not be useful. If you want to know how to use them, you can use PSpice's Help Menu.
- It is interesting to note that you can plot the phase of a value by using IP(xx), where xx is the name of the source you wish to see the phase for.

## F. Labeling:

- Click on Text Label  on top tool bar.
- Type in what you want to write.
- Click OK
- You can move this around by single clicking and dragging.

## G. Finding Points:


- There are Cursor buttons that allow you to find the maximum or minimum or just a point on the line. These are located on the toolbar (to the right).
- Select which curve you want to look at and then select "Toggle Cursor" .

- Then you can find the max, min, the slope, or the relative max or min (  is find relative max).

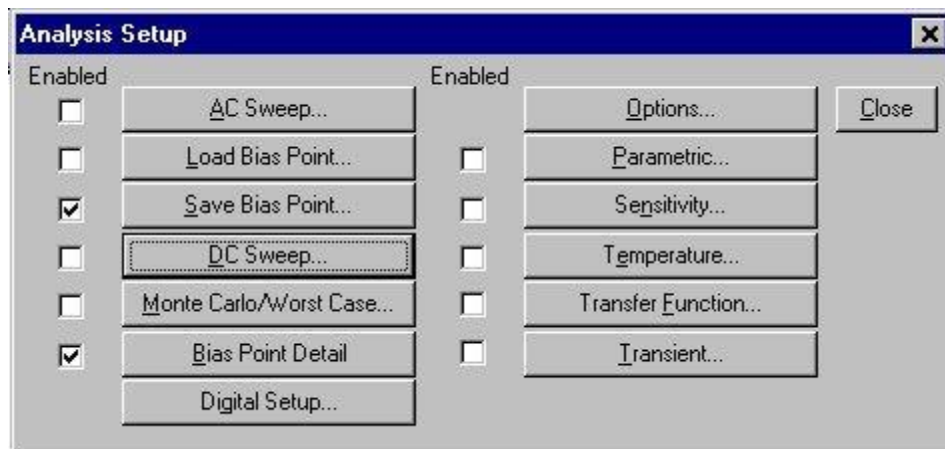
## H. Saving:


- To save your probe you need to go into the tools menu and click display, this will open up a menu which will allow you to name the probe file and choose where to save it. You can also open previously saved plots from here as well.

## I. Printing:

- Select Print in Edit or on the toolbar .
- Print as usual.

## IV. Analysis Menu



To open the analysis menu click on the  button.

### A. AC Sweep

- The AC sweep allows you to plot magnitude versus frequency for different inputs in your circuit.  
In the AC sweep menu you have the choice of three types of analysis:
  - **Linear,**
  - **Octave and**
  - **Decade.**
- These three choices describe the X-axis scaling which will be produced in probe. For example, if you choose decade then a sample of your X-axis might be 10Hz, 1kHz, 100kHz, 10MHz, etc.... Therefore if you want to see how your circuit reacts over a very large range of frequencies choose the decade option.

- You now have to specify at how many points you want PSpice to calculate frequencies, and what the start and end frequency will be. That is, over what range of frequencies do you want to simulate your circuit.

## B. DC Sweep

- The DC sweep allows you to do various different sweeps of your circuit to see how it responds to various conditions.
- For all the possible sweeps,
  - voltage,
  - current,
  - temperature, and
  - parameter and global

you need to specify a start value, an end value, and the number of points you wish to calculate.

- For example you can sweep your circuit over a voltage range from 0 to 12 volts. The main two sweeps that will be most important to us at this stage are the voltage sweep and the current sweep. For these two, you need to indicate to PSpice what component you wish to sweep, for example V1 or V2.
- Another excellent feature of the DC sweep in PSpice, is the ability to do a **nested sweep**.
- A nested sweep allows you to run two simultaneous sweeps to see how changes in two different DC sources will affect your circuit.
- Once you've filled in the main sweep menu, click on the nested sweep button and choose the second type of source to sweep and name it, also specifying the start and end values. (Note: In some versions of PSpice you need to click on **enable nested sweep**). Again you can choose Linear, Octave or Decade, but also you can indicate your own list of values, example: 1V 10V 20V. **DO NOT** separate the values with commas.

## C. Bias Point Detail

- This is a simple, but incredibly useful sweep. It will not launch Probe and so give you nothing to plot. But by clicking on **enable bias current display** or **enable bias voltage display**, this will indicate the voltage and current at certain points within the circuit.

## D. Parametric

- Parametric analysis allows you to run another type of analysis (transient, sweeps) while using a range of component values using the **global parameter** setting. The best way to demonstrate this is with an example, we will use a resistor, but any other standard part would work just as well (capacitor, inductor).
- First, double-click the value label of the resistor that is to be varied. This will open a "Set Attribute Value" dialog box. Enter the name **{RVAL}** (including the curly braces) in place of the component value. This indicates to PSpice that the value of the resistor is a

global parameter called RVAL. In order to define the RVAL parameter it is necessary to place a global parameter list somewhere on the schematic page. To do this, choose "**Get New Part**" from the menu and select the part named **param**.

- Place the box anywhere on the schematic page. Now double-click on the word **PARAMETERS** in the box title to bring up the parameter dialog box. Set the NAME1= value to **RVAL** (no curly braces) and the VALUE1= value to the nominal resistance value. This nominal value is required, but it is only used if the DC bias point detail is computed. Otherwise, the value is ignored by PSpice.
- Finally, go to the "Analysis Setup" menu and enable "Parametric" analysis. Open the Parametric setup dialog box and enter the sweep parameters: Name: **RVAL** Swept variable type: Global Parameter. Make sure the other analysis type(s) are selected in the analysis setup menu (transient, sweeps). PSpice will now automatically perform the simulation over and over, using a new value for **RVAL** during each run.
- This isn't as important for us in the lab, but some day when you are constructing real circuits that need to function under various conditions this will be useful.

## E. Temperature

- The temperature option allows you to specify a temperature, or a list of temperatures (do not include commas between temperature values) for which PSpice will simulate your circuit.
- For a list of temperatures that simulation is done for each specified temperature.

## F. Transient

- The transient analysis is probably the most important analysis you can run in PSpice, and it computes various values of your circuit over time. Two very important parameters in the transient analysis are:
  - **print step**
  - **final time.**
- The ratio of **final time: print step** determines how many calculations PSpice must make to plot a wave form. PSpice always defaults the start time to zero seconds and going until it reaches the user defined final time. It is incredibly important that you think about what print step you should use before running the simulation, if you make the print step too small the probe screen will be cluttered with unnecessary points making it hard to read, and taking extreme amounts of time for PSpice to calculate. However, at the opposite side of that coin is the problem that if you set the print step too high you might miss important phenomenon that are occurring over very short periods of time in the circuit. Therefore play with step time to see what works best for your circuit.
- You can set a step ceiling which will limit the size of each interval, thus increasing calculation speed. Another handy feature is the Fourier analysis, which allows you to specify your fundamental frequency and the number of harmonics you wish to see on the

plot. PSpice defaults to the 9th harmonic unless you specify otherwise, but this still will allow you to decompose a square wave to see its components with sufficient detail.

## V. Types of Sources

### A. Voltage Sources

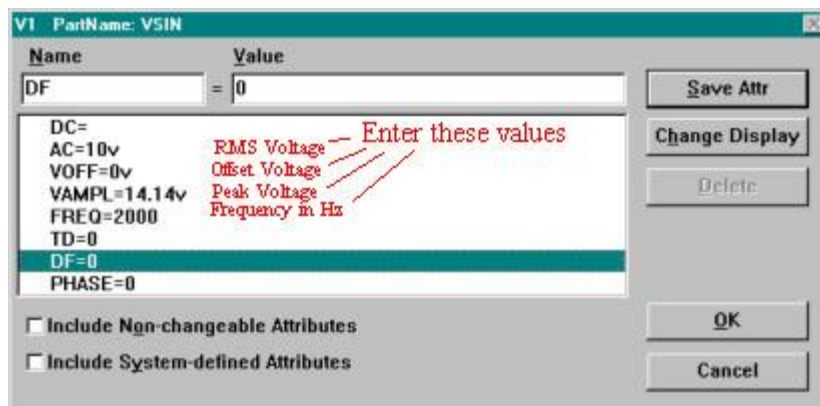
#### i. VDC

- This is your basic direct current voltage source that simulates a simple battery and allows you to specify the voltage value.

#### ii. VAC

- A few things to note about the alternating current source, first PSpice takes it to be a sine source, so if you want to simulate a cosine wave you need to add (or subtract) a  $90^\circ$  phase shift. There are three values which PSpice will allow you to alter, these being:
  - **ACMAG** which is the RMS value of the voltage.
  - **DC** which is the DC offset voltage
  - **ACPHASE** which is the phase angle of the voltage
- Note that the phase angle if left unspecified will be set by default to  $0^\circ$

#### iii. VSIN



- The SIN type of source is actually a damped sine with time delay, phase shift and a DC offset. If you want to run a transient analysis you need to use the VSIN see how AC will effect your circuit over time. Do not use this type of source for a phasor or frequency sweep analysis, VAC would be appropriate for that.
  - **DC** the DC component of the sine wave
  - **AC** the AC value of the sine wave
  - **Voff** is the DC offset value. It should be set to zero if you need a pure sinusoid.

- **Vamplitude** is the undamped amplitude of the sinusoid; i.e., the peak value measured from zero if there were no DC offset value.
  - **FREQ** is the frequency in Hz of the sinusoid.
  - **TD** is the time delay in seconds. Set this to zero for the normal sinusoid.
  - **DF** is the damping factor. Also set this to zero for the normal sinusoid.
  - **PHASE** is the phase advance in degrees. Set this to 90 if you need a cosine wave form.
- Note that the normal usage of this source type is to set **Voff**, **TD** and **DF** to zero as this will give you a 'nice' sine wave.

#### iv. VPULSE

- The VPULSE is often used for a transient simulation of a circuit where we want to make it act like a square wave source. It should never be used in a frequency response study because PSpice assumes it is in the time domain, and therefore your probe plot will give you inaccurate results.
  - **DC** the DC component of the wave.
  - **AC** the AC component of the wave.
  - **V1** is the value when the pulse is not "on." So for a square wave, the value when the wave is 'low'. This can be zero or negative as required. For a pulsed current source, the units would be "amps" instead of "volts."
  - **V2** is the value when the pulse is fully turned 'on'. This can also be zero or negative. (Obviously, V1 and V2 should not be equal.) Again, the units would be "amps" if this were a current pulse.
  - **TD** is the time delay. The default units are seconds. The time delay may be zero, but not negative.
  - **TR** is the rise time of the pulse. PSpice allows this value to be zero, but zero rise time may cause convergence problems in some transient analysis simulations. The default units are seconds.
  - **TF** is the fall time in seconds of the pulse.
  - **TW** is the pulse width. This is the time in seconds that the pulse is fully on.
  - **PER** is the period and is the total time in seconds of the pulse.
- This is a very important source for us because we do a lot of work with the square wave on the wave generator to see how various components and circuits respond to it.

#### B. Current Sources

- For any of the previous discussed voltage sources, there exist the exact source except that they produce current. There is one thing that should be mentioned; current sources in PSpice get a little confusing. For those current sources whose circuit symbol has an arrow, you have to point the arrow in the direction of conventionally flowing current. This applies to all current sources, including AC and DC. Therefore placing the current source in the circuit backwards with seemingly incorrect polarities will give the correct results.

- An interesting little feature under the **markers** menu is the ability to add markers to your circuit so you can see where the current and voltage have imaginary values in the circuit, and the phase of your source at any point in the circuit.

## VI. Digital Simulation

- PSpice can simulate digital circuits and Probe can output a timing diagram showing the relationship between all the signals propagating in the circuit. The following will be a brief introduction to digital analysis using PSpice.
- The evaluation version of PSpice provides many of the common digital parts that we use in the lab exercises these include, but are not limited to,
  - counters,
  - multiplexers,
  - decoders,
  - flip-flops,
  - latches,
  - all common gates,
  - buffers,
  - adders and a lot more.
- To begin, create a digital circuit the same way you would create an analog one by getting and placing the parts (see above for details on creating a circuit).
- Now instead of an analog voltage source you want to place digital stimuli, these are located in the parts menu as **HI** and **LO**.
- Draw wires connecting the parts like usual, but whenever you have an output of a gate that would be an output to your circuit you need to terminate it with a block.
- You can add these blocks to your circuit. On the tool bar there is a button labeled **Add Block**; connect this block to any output signal of your circuit.
- Then run a bias point analysis and click on the button to see the **enable voltage bias display**, to see the logic level on each of your lines.
- You can run a transient analysis at this point and plot the signals, but they will just be straight lines, showing the logic level of each signal. However a digital circuit that remains in a constant state all the time is of little interesting. Eventually, you are going to encounter a circuit that needs to be stimulated with a clock pulse.
- To add a time varying 'clock pulse' to your circuit, enter the parts menu and get a part labeled **STIM1** and place that as the input to a device. Double click on the **STIM1** to open up a dialogue box that looks like this:



Name	Value
COMMAND2	= 1m 1
TIMESTEP=1m COMMAND1=0s 0 <b>COMMAND2=1m 1</b> COMMAND3=2m 0 COMMAND4=3m 1 COMMAND5=4m 0 COMMAND6=5m 1 COMMAND7=6m 0	

☒ Include Non-changeable Attributes  
☒ Include System-defined Attributes

Buttons: Save Attr, Change Display, Delete, OK, Cancel

and fill it in with the **Time Step** that you want your 'clock' to have. This **Time Step** value is the your 'clock' pulse. There are various ways you can fill out the command prompts, one way is to do it as shown in the picture, however here are some simple commands that will allow you to do a range of simulations:

- REPEAT FOREVER, to simulate a real clock
  - REPEAT <n> TIMES
  - END REPEAT
  - <<time>> INCR <<value>>
  - <<time>> DECR <<value>>
  - for <<value>> you can use, 0,1,R(rising), F(falling), X(unknown), or Z(high impedance)
- There are other useful commands that are documented well in the online manual. If you are interesting in seeing the voltage levels attached to these gates to see if you need to buffer any signals or to see if you are exceeding the fan out and/or fan in of any gate, you need to add dropping resistors to the inputs of gates, and a load resistor the outputs of the gates and simulate again. When you are running the transient analysis, don't forget to change your **print step** and the **final time** so you will actually be able to see the results.
  - If your digital circuit contains a counter, or decoder or some other device with many outputs that are all going to a single destination, instead of drawing many individual signal lines from each source to the proper destination you can use a **bus**. In the **draw** menu at the top of the screen is an option for drawing a bus which is a thick wire able to carry many separate signals to a single place. The order in which the signals go on the bus is the order in which they come off at the destination.

## Lab 15

### a) To Find The Various Node Voltages And Voltage drops In the Given Circuit

#### **Objective:**

- To measure the Node voltages using “VPRINT1” components
- To measure the voltage difference using “VPRINT2” components

#### **Theory:**

VPRINT1 and VPRINT2 are use in AC analysis for voltage.VPRINT1 is use to find voltage at node and VPRINT2 is use to find voltage across each component.

#### **Components:**

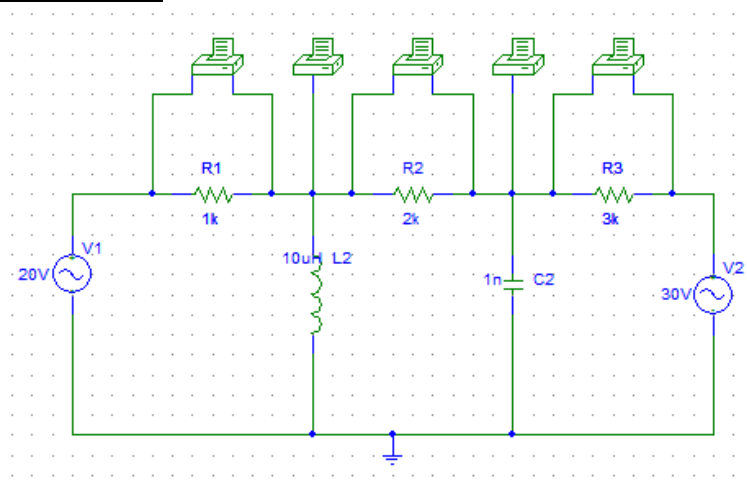
- Capacitor
- Resistors
- Inductor
- AC voltage source
- GND-EARTH

#### **Procedure:**

- Make the circuit in PSPICE simulator using parts from the library (Ctrl+G) or from get new part.
- Use draw wire to connect all the components together.
- Ground the whole circuit.

- Use VPRINT1 to find voltage at node and VPRINT2 to find voltage across each component.
- Set the following attributes of AC voltage source
- ACMAG=20V
- ACPHASE=OK
- Set the desire value for each component.
- Set the following attributes for VPRINT1 and VPRINT2
- AC=OK
- MAG=OK
- PHASE=OK
- Save the whole circuit by a single click on the icon save.
- Go to analysis ,click setup then click AC sweep.
- In AC sweep set following attributes as
- Total pts.:1
- Start freq.:50
- End freq.:50
- Click OK then click close.
- Click analysis ,a new window appear click view there then click output file and get the result.

## **Circuit:**



## Output File:

\*\*\*\* AC ANALYSIS \*\*\*\*

FREQ VM(\$N\_0002,\$N\_0001)VP(\$N\_0002,\$N\_0001)

5.000E+01 2.000E+01 -2.340E-04

\*\*\*\* AC ANALYSIS TEMPERATURE = 27.000 DEG C

\*\*\*\*\*

FREQ VM(\$N\_0001) VP(\$N\_0001)

5.000E+01 8.168E-05 8.999E+01

\*\*\*\* AC ANALYSIS TEMPERATURE = 27.000 DEG C

\*\*\*\*\*

FREQ VM(\$N\_0001,\$N\_0003)VP(\$N\_0001,\$N\_0003)

5.000E+01 1.200E+01 1.800E+02

\*\*\*\* AC ANALYSIS TEMPERATURE = 27.000 DEG C

\*\*\*\*\*

FREQ VM(\$N\_0003) VP(\$N\_0003)

5.000E+01 1.200E+01 -2.137E-02

\*\*\*\* AC ANALYSIS TEMPERATURE = 27.000 DEG C

\*\*\*\*\*

FREQ VM(\$N\_0003,\$N\_0004)VP(\$N\_0003,\$N\_0004)

5.000E+01 1.800E+01 -1.800E+02

## **b) To Find The Various Loop And Branch Current In the Given Circuit**

### **Objective:**

- To find loop and branch current in the given circuit

### **Theory:**

IPRINT is use in AC analysis for current measurement.

### **Components:**

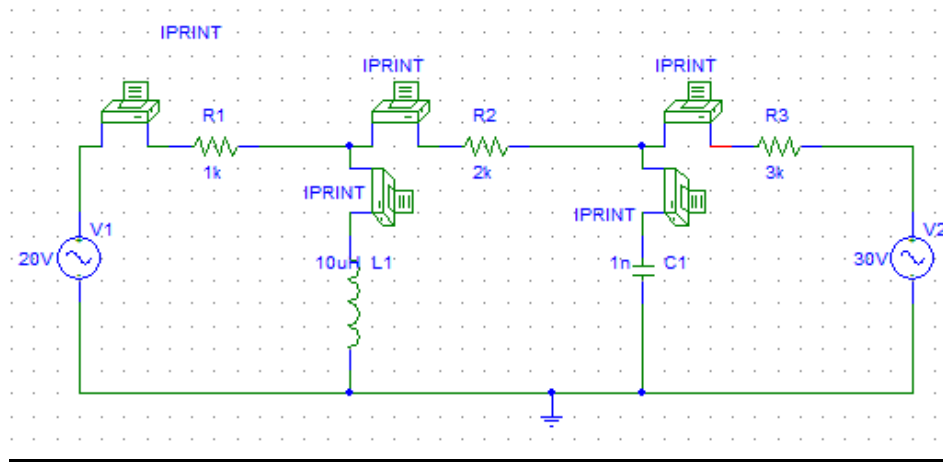
- Capacitor
- Resistors
- Inductor
- AC voltage source
- GND-EARTH

### **Procedure:**

- Make the circuit in PSPICE simulator using parts from the library (Ctrl+G) or from get new part.
- Use draw wire to connect all the components together.
- Ground the whole circuit.
- Use IPRINT to find current .
- Set the following attributes of AC voltage source
- ACMAG=20V
- ACPHASE=OK

- Set the desire value for each component.
- Set the following attributes for IPRINT.
- AC=OK
- MAG=OK
- PHASE=OK
- Save the whole circuit by a single click on the icon save.
- Go to analysis ,click setup then click AC sweep.
- In AC sweep set following attributes as
- Total pts.:1
- Start freq.:50
- End freq.:50
- Click OK then click close.
- Click analysis ,a new window appear click view there then click output file and get the result.

## Circuit:



## Output File:

\*\*\* AC ANALYSIS \*\*\*

FREQ IM(V\_PRINT1)IP(V\_PRINT1)

5.000E+01 2.000E-02 -2.340E-04

\*\*\* AC ANALYSIS TEMPERATURE = 27.000 DEG C

FREQ IM(V\_PRINT6)IP(V\_PRINT6)

5.000E+01 6.000E-03 1.800E+02

\*\*\* AC ANALYSIS TEMPERATURE = 27.000 DEG C

FREQ IM(V\_PRINT2)IP(V\_PRINT2)

5.000E+01 2.600E-02 -5.201E-03

\*\*\* AC ANALYSIS TEMPERATURE = 27.000 DEG C

FREQ IM(V\_PRINT5)IP(V\_PRINT5)

5.000E+01 3.770E-06 8.998E+01

\*\*\* AC ANALYSIS TEMPERATURE = 27.000 DEG C

FREQ IM(V\_PRINT3)IP(V\_PRINT3)

5.000E+01 6.000E-03 -1.800E+02

## Lab 16

a) To analyze R-C and R-L circuit using PSPICE (Natural Response).

### **Objective:**

To analyze R-C and R-L circuit using PSPICE (natural response).

### **Theory:**

#### **Natural Response:**

Sudden resistance is shown by capacitor when it is abruptly connected to DC source. At time  $t > 0$  the switch is close and capacitor is fully charged and at  $t = 0$ , switch is open and capacitor acts as a source. Such a response of a circuit is called the natural response of the circuit.

### **Components:**

- Resistor.
- VDC
- Capacitor
- Close Switch

### **Procedure:**

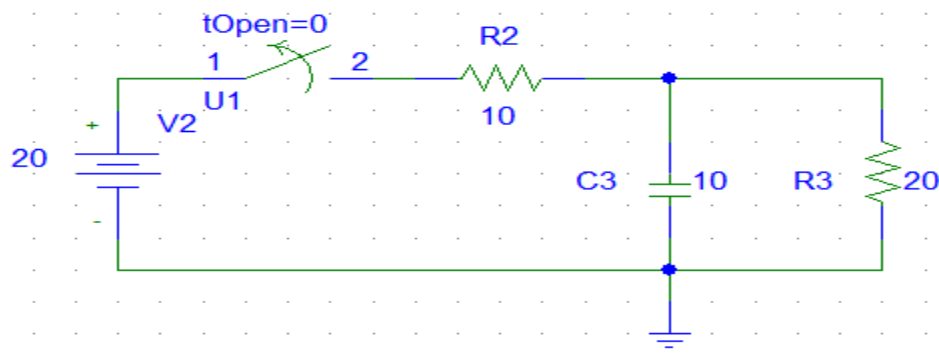
- Make the circuit in PSPICE simulator using the part from library (Ctrl+G).
- Connect Resistor capacitor and switch together by using draw wire.
- Use DC voltage source.
- Connect the switch in series with capacitor.
- Switch becomes open when time reaches to 0.
- Use transient option and give it the following values

Print step

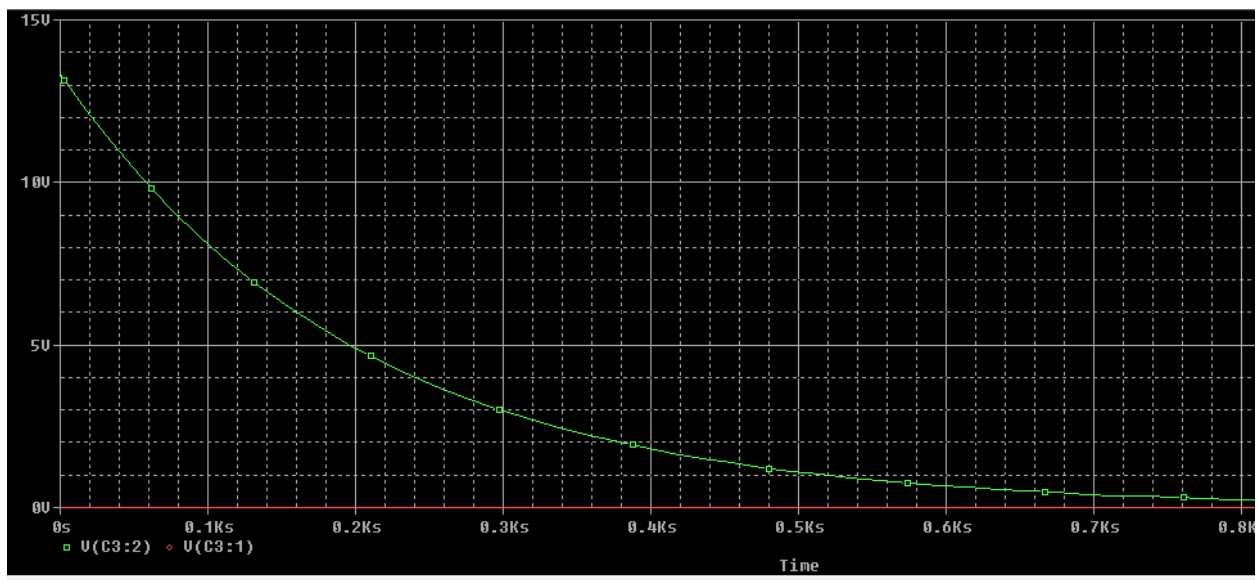
Final step



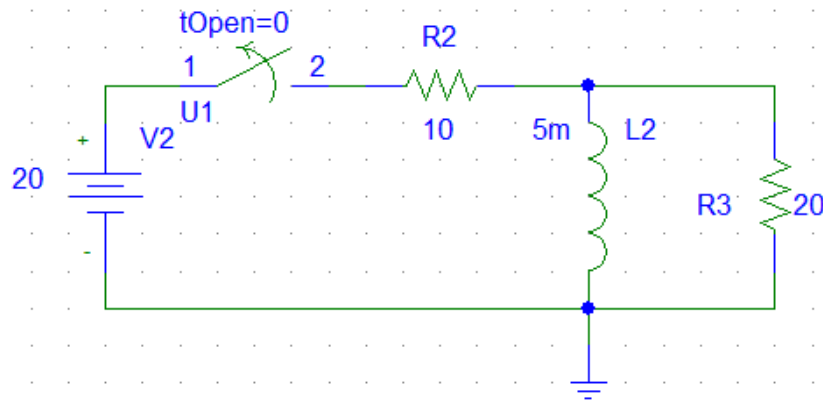
## Part No 1: RC Circuit:



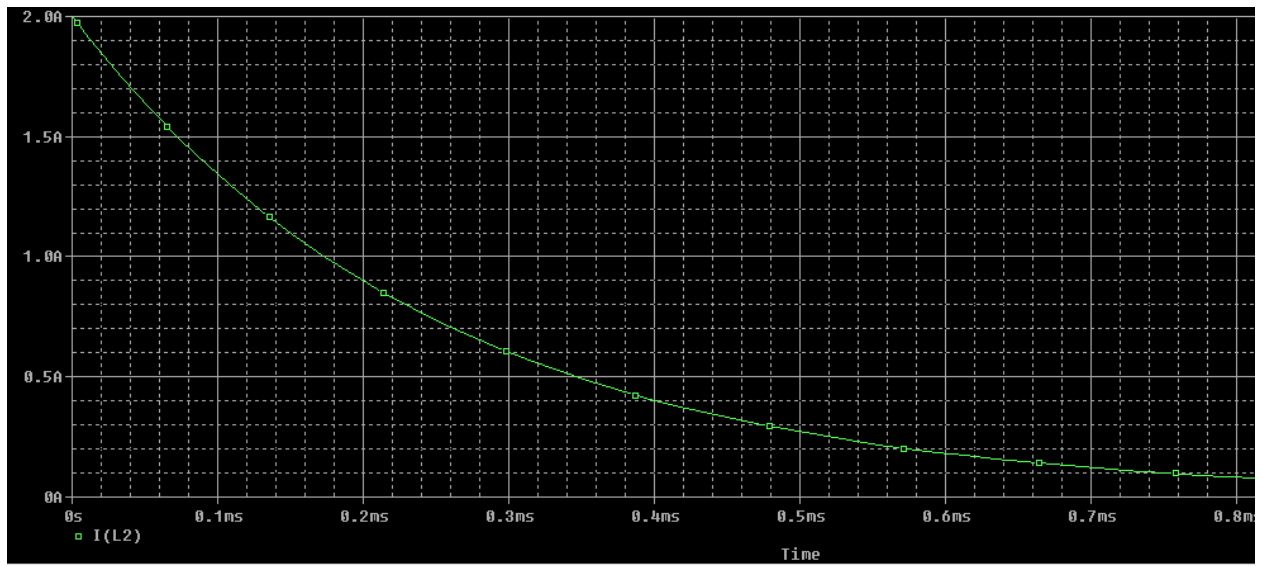
## Graph:



## Part No 2: R-L Circuit:



**Graph:**



b) To analyze R-C and R-L circuit using PSPICE (Step Response)

## **Objective:**

To analyze R-C circuit and R-L using PSPICE (step response).

## **Theory:**

### **Step Response:**

When a dc voltage (current) source is suddenly applied to a circuit, it can be modeled as a step function, and the resulting response is called step response.

## **Components:**

- Resistor.
- VDC
- Capacitor
- Inductor
- Close Switch

## **PART 1: RC circuit:**

## **Procedure:**

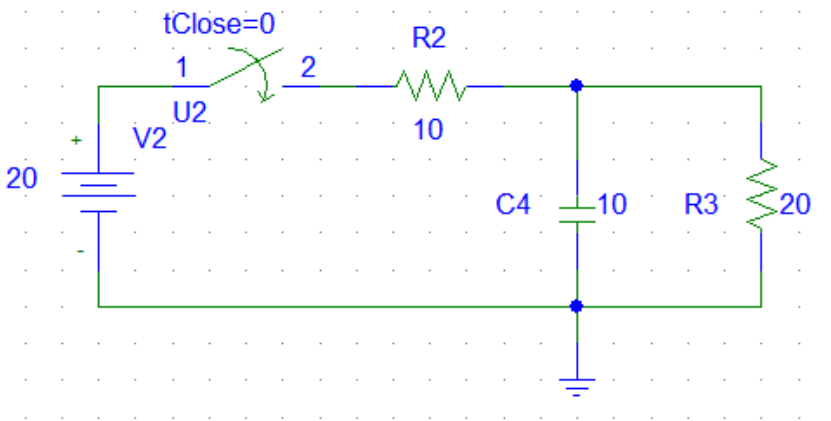
- Make the circuit in PSPICE simulator using the part from library (Ctrl+G).
- Connect Resistor capacitor and switch together by using draw wire.
- Use DC voltage source.
- Connect the switch in series with capacitor.
- Switch becomes close when time reaches to 0.

- Use transient option and give it the following values

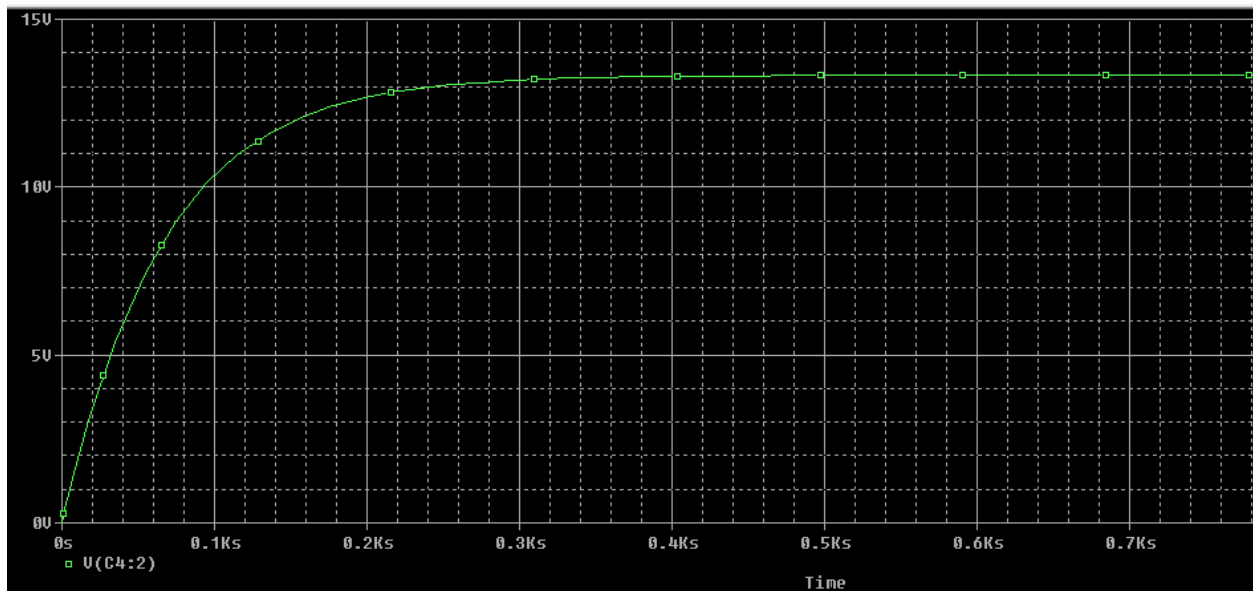
Print step

Final step

## **PART 1: RC circuit:**



## **Graph:**



## **PART 2: RL circuit:**

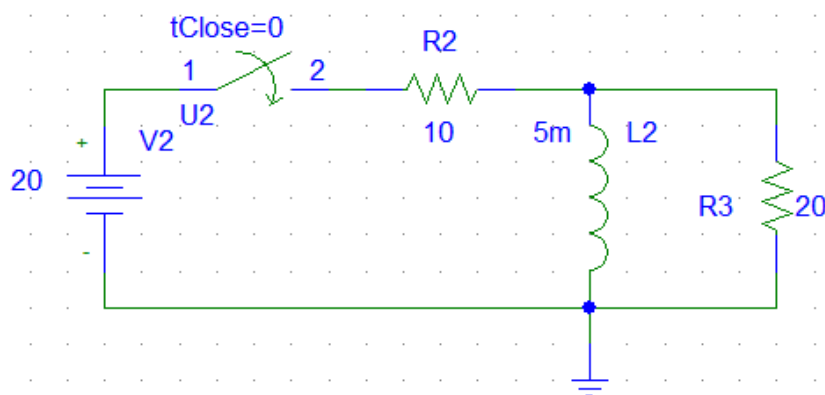
### **Procedure:**

- Make the circuit in PSPICE simulator using the part from library (Ctrl+G).
- Connect Resistor capacitor and switch together by using draw wire.
- Use DC voltage source.
- Connect the switch in series with capacitor.
- Switch becomes close when time reaches to 0.
- Use transient option and give it the following values

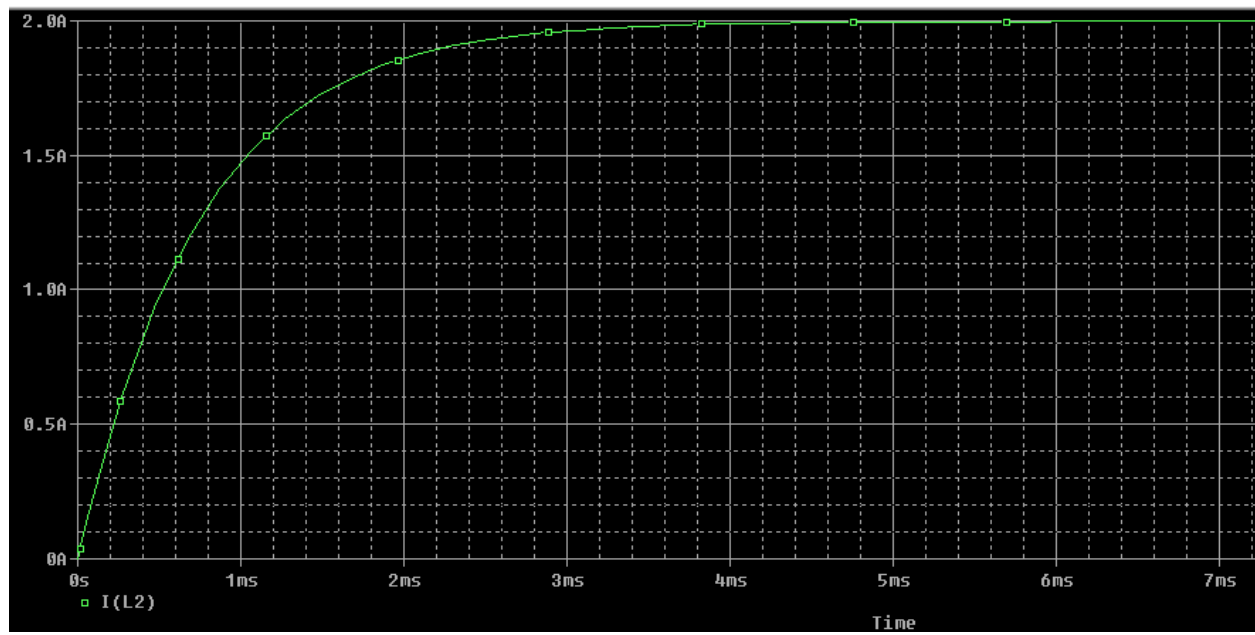
Print step

Final step

## **PART 2: RL circuit:**



## Graph:



## Lab 17

To analyze low pass, band pass filter and notch filter using PSPICE

### Objective:

To analyze low pass, band pass filter and notch filter using PSPICE simulator.

### Theory:

Low Pass Filter:

A **low-pass filter** is a **filter** that **pass** signals with a frequency **lower** than a certain cutoff frequency and attenuates signals with frequencies higher than the cutoff frequency. The amount of attenuation for each frequency depends on the **filter** design.

Band Pass Filter:

A **bandpass filter** is an electronic device or circuit that allows signals between two specific frequencies to **pass**, but that discriminates against signals at other frequencies.

Notch Filter:

In signal processing, a **band-stop filter** or **band-rejection filter** is a **filter** that **passes** most frequencies unaltered, but attenuates those in a specific range to very low levels. It is the opposite of a **band-pass filter**. A **notch filter** is a **band-stop filter** with a narrow stopband (high Q factor).

### Components:

- Resistors
- Inductor
- Capacitor
- AC voltage source
- GND-EARTH

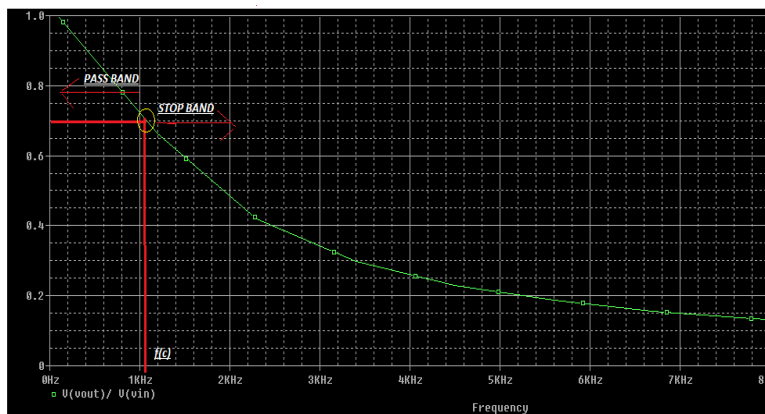
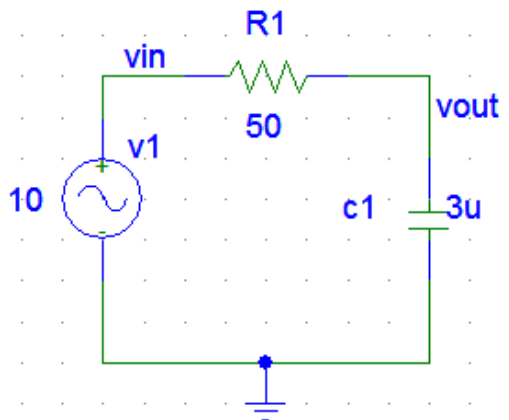
## Procedure:

- Make circuit for low pass, band pass and notch filter one by one in PSPICE simulator using the parts from the library(Ctrl+G).
- Use AC voltage source for each circuit.
- Use Transient option and give it the following values:

Print step

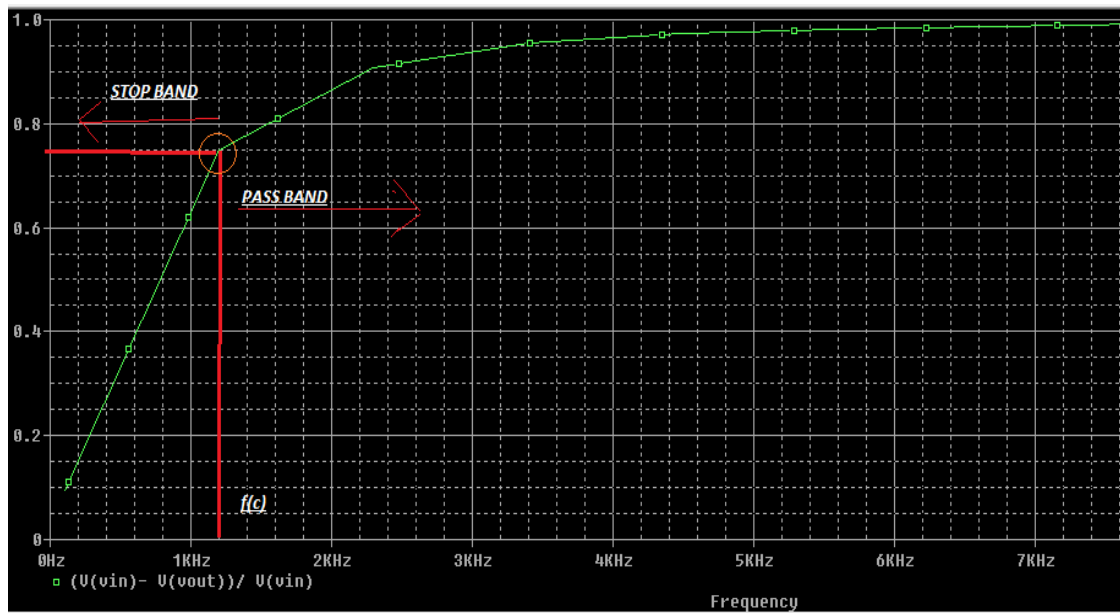
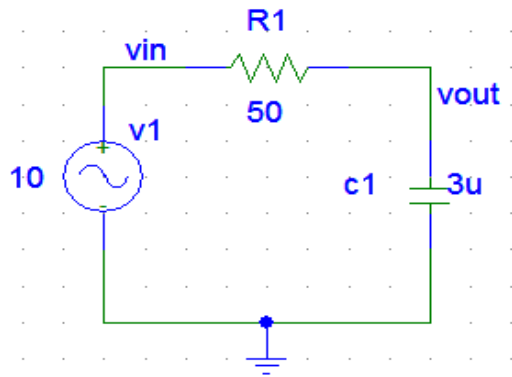
Final step

## Low Pass Filter:

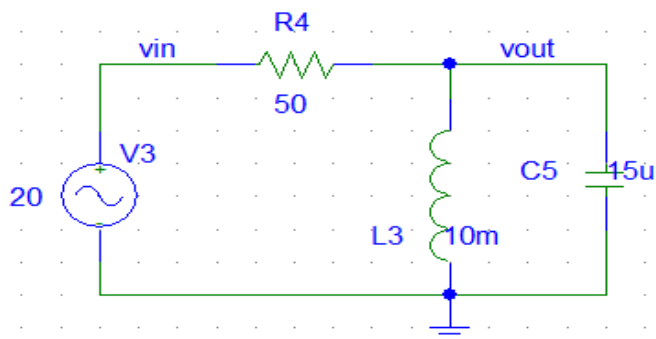


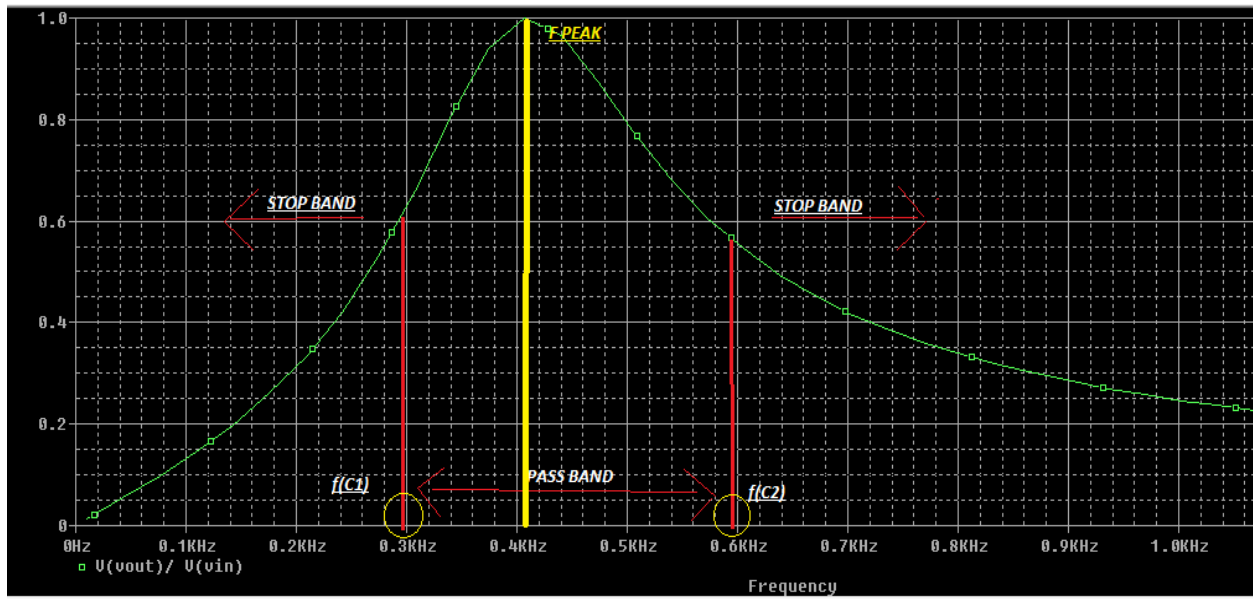
## High Pass Filter:





## Band Pass Filters:





## Notch Filter:

