Department of Electronics and Communication Engineering

Sub Code/Name: **BEC5L3-COMMUNICATION ENGINEERING LAB-I**

Name : ………………………………………

Reg No : ………………………………………

Branch : ………………………………………

Year & Semester : ………………………………………

LIST OF EXPERIMENTS

INDEX

Expt.	Date	Name of the Experiment	Marks	Staff SIGN

Ex No:1

Date:

AMPLITUDE MODULATION AND DEMODULATION

Aim: 1. To generate amplitude modulated wave and determine the percentage modulation.

2. To Demodulate the modulated wave using envelope detector.

Apparatus Required:

Name of the Component/Equipment	Specifications/Range	Quantity
Transistor(BC 107)	BC107	1
Diode(0A79)	Max Current 35mA	1
Resistors	1KΩ, 22KΩ, 6.8KΩ, 10KΩ,100KΩ	1 each
Capacitor	0.01μF,2 μF	1
Inductor	130mH	1
CRO	20MHz	1
Function Generator	1MHz	2
Regulated Power Supply	0-30V, 1A	1

Theory:

Amplitude Modulation is defined as a process in which the amplitude of the carrier wave c(t) is varied linearly with the instantaneous amplitude of the message signal m(t).The standard form of an amplitude modulated (AM) wave is defined by

$$s(t) = A_c[1 + K_a m(t)]\cos(2\pi f_c t)$$

Where K_a is a constant called the amplitude sensitivity of the modulator.

The demodulation circuit is used to recover the message signal from the incoming AM wave at the receiver. An envelope detector is a simple and yet highly effective device that is well suited for the demodulation of AM wave, for which the percentage modulation is less than 100%.Ideally, an envelop detector produces an output signal that follows the envelop of the input signal wave form exactly; hence, the name. Some version of this circuit is used in almost all commercial AM radio receivers.

The Modulation Index is defined as, $m = \frac{(E_{max} - E_{min})}{(E_{max} + E_{min})}$

Where E_{max} and E_{min} are the maximum and minimum amplitudes of the modulated wave.

Circuit Diagrams:

For modulation:

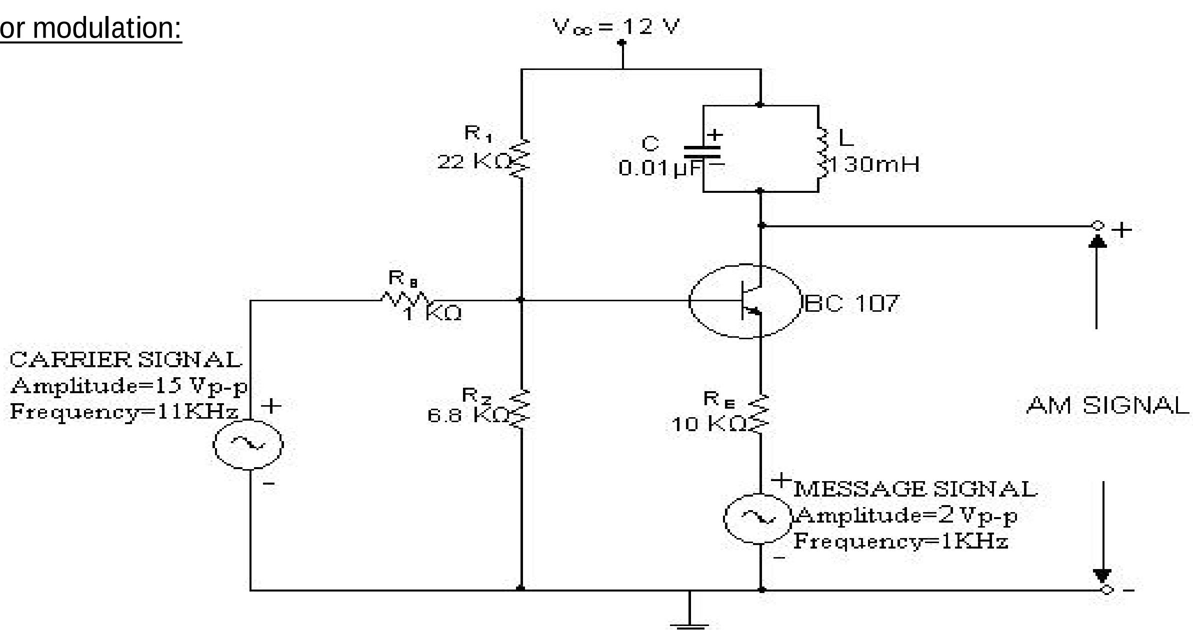


Fig.1. AM modulator

For demodulation:

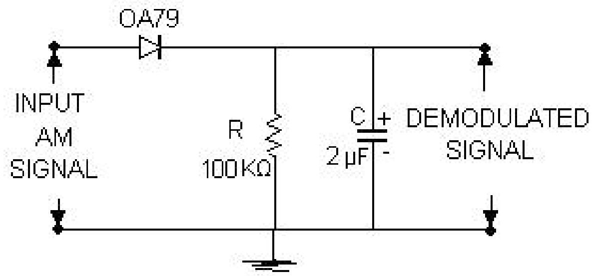


Fig.2. AM demodulator

Procedure:

1. The circuit is connected as per the circuit diagram shown in Fig.1.
2. Switch on + 12 volts V_{CC} supply.
3. Apply sinusoidal signal of 1 KHz frequency and amplitude 2 Vp-p as modulating signal, and carrier signal of frequency 11 KHz and amplitude 15 Vp-p.
4. Now slowly increase the amplitude of the modulating signal up to 7V and note down values of E_{max} and E_{min}.
5. Calculate modulation index using equation
6. Repeat step 5 by varying frequency of the modulating signal.
7. Plot the graphs: Modulation index vs Amplitude & Frequency
8. Find the value of R from $f_m = \frac{1}{2\pi RC}$ taking C = 0.01µF
9. Connect the circuit diagram as shown in Fig.2.
10. Feed the AM wave to the demodulator circuit and observe the output
11. Note down frequency and amplitude of the demodulated output waveform.
12. Draw the demodulated wave form .,m=1

Observations

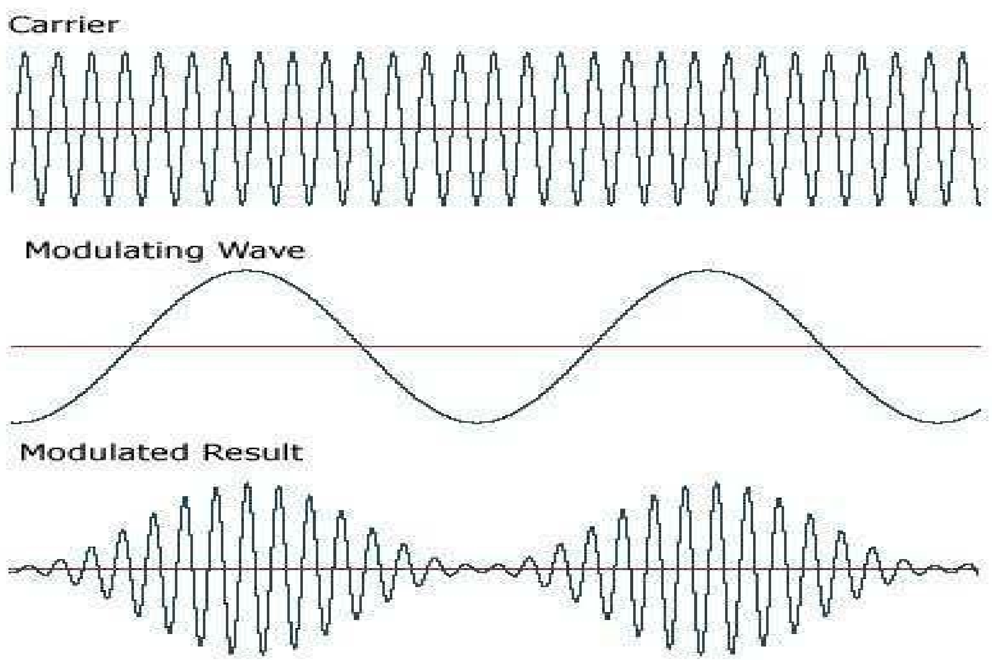
Table 1: f_m= 1KHz, f_c=11KHz, A_c=15 V p-p.

S.No.	V_m(Volts)	E_{max}(volts)	E_{min} (Volts)	m	%m (m x100)

Table 2: A_m= 4 Vp-p f_c =11KHz, A_c=15 V p-p.

S.No.	f_m(KHz)	E_{max}(volts)	E_{min}(Volts)	m	%m (m x100)

Waveforms and graphs:



Precautions:

1. Check the connections before giving the power supply
2. Observations should be done carefully.

RESULT:

1. Thus the amplitude modulated waves were generated and also the percentage modulation calculated.

2. Demodulation of the modulated waves were done using envelope detector.

Ex No:2

Date:

DSB-SC MODULATION AND DEMODULATION

Aim: To generate AM-Double Side Band Suppressed Carrier (DSB-SC) signal.

Apparatus Required:

Name of the Component/Equipment	Specification/Range	Quantity
IC 1496	Internal power dissipation – 500mw(MAX) Wide frequency response up to 100 MHz	1
Resistors	10 K, 3.9 K,1K,51K,6.8K	2,3,1 each
Capacitors	0.1 µF	4
Variable Resistor (Linear Pot)	0-50K	1
CRO	100MHz	1
Function Generator	1MHz	2
Regulated Power Supply	0-30 v, 1A	1

Theory:

Balanced modulator is used for generating DSB-SC signal. A balanced modulator consists of two standard amplitude modulators arranged in a balanced configuration so as to suppress the carrier wave. The two modulators are identical except the reversal of sign of the modulating signal applied to them.

Circuit Diagram:

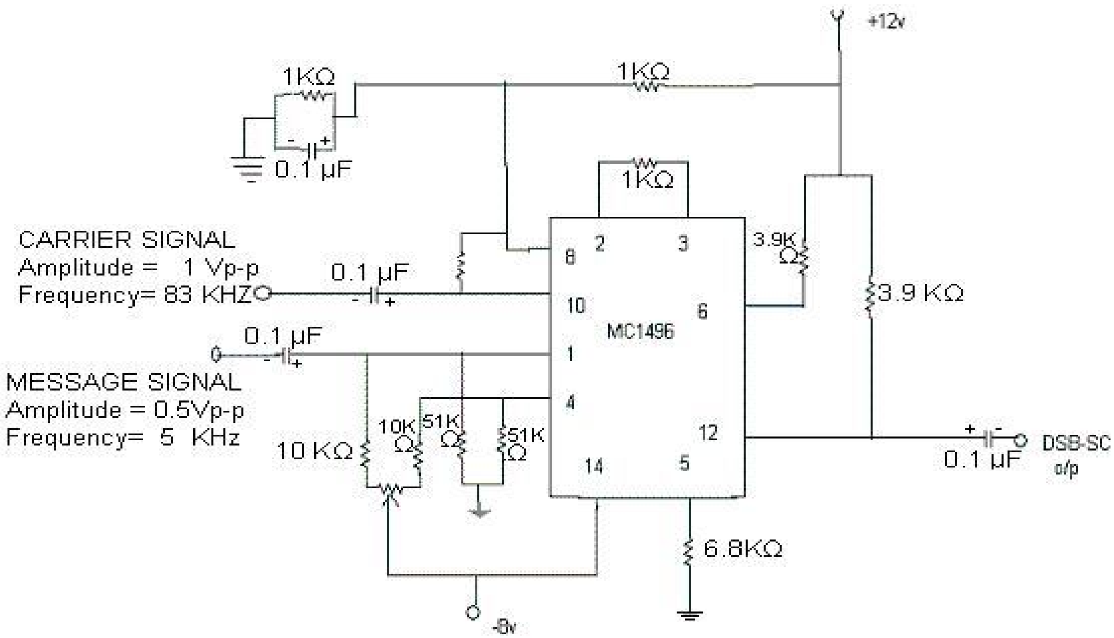


Fig.1. Balanced Modulator Circuit

Procedure:

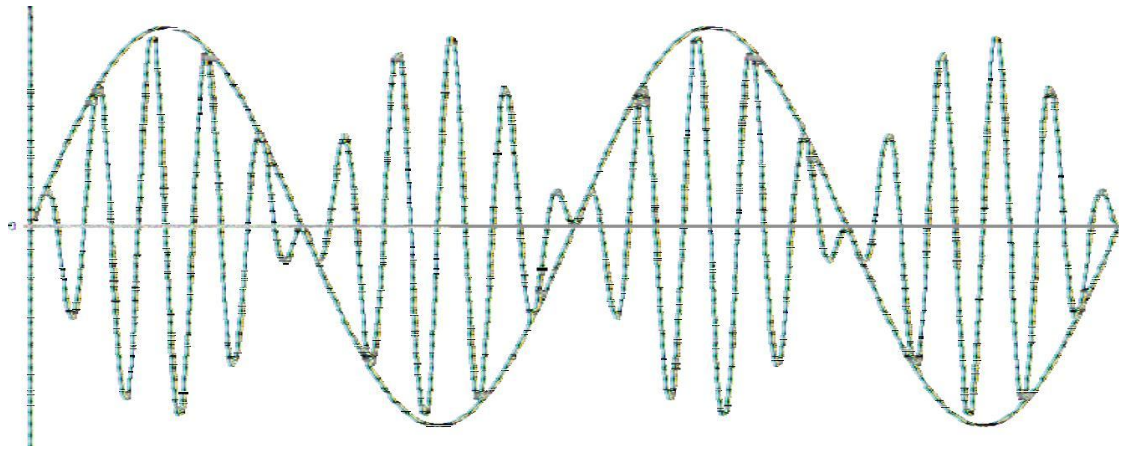
1. Connect the circuit diagram as shown in Fig.1.

2. An Carrier signal of 1Vp-p amplitude and frequency of 83 KHz is applied as carrier to pin no.10.

3. An AF signal of 0.5Vp-p amplitude and frequency of 5 KHz is given as message signal to pin no.1.

4. Observe the DSB-SC waveform at pin no.12.

Waveforms:



RESULT

The AM-Double Side Band Suppressed Carrier (DSB-SC) signal was generated.

Ex No:3

Date:

<u>SSB MODULATOR AND DEMODULATOR</u>.

Aim:

To generate the SSB modulated wave.

Apparatus Required:

Name of the Component/Equipment	Specifications	Quantity
SSB system trainer board	---	1
CRO	30MHz	1

Theory:

An SSB signal is produced by passing the DSB signal through a highly selective band pass filter. This filter selects either the upper or the lower sideband. Hence transmission bandwidth can be cut by half if one sideband is entirely suppressed. This leads to single-sideband modulation (SSB). In SSB modulation bandwidth saving is accompanied by a considerable increase in equipment complexity.

Circuit Diagram:

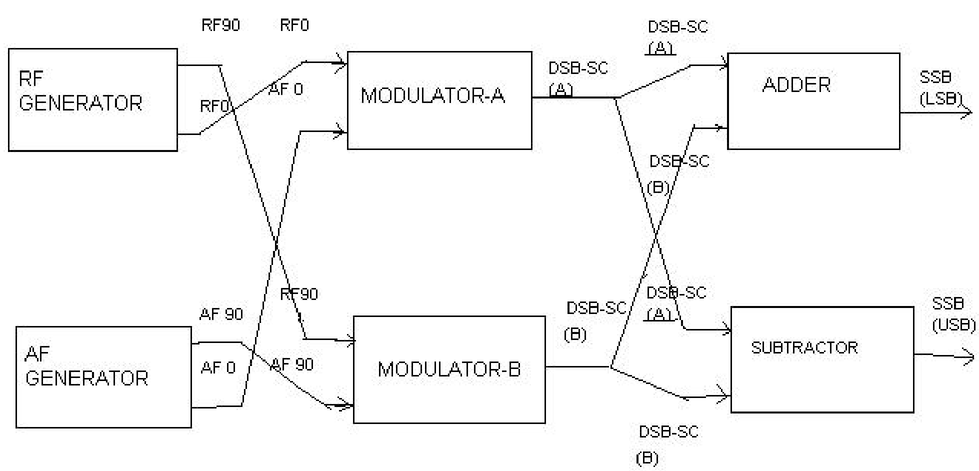


Fig. 1 Single Side Band system

Procedure:

1. Switch on the trainer and measure the output of the regulated power supply i.e., ±12V and -8V.

2. Observe the output of the RF generator using CRO. There are 2 outputs from the RF generator, one is direct output and another is 90^o out of phase with the direct output. The output frequency is 100 KHz and the amplitude is ≥ $0.2V_{PP}$. (Potentiometers are provided to vary the output amplitude).

3. Observe the output of the AF generator, using CRO. There are 2 outputs from the AF generator, one is direct output and another is 90^o out of phase with the direct output. A switch is provided to select the required frequency (2 KHz, 4KHz or 6 KHz). AGC potentiometer is provided to adjust the gain of the oscillator (or to set the output to good shape). The oscillator output has amplitude ⍰ $10V_{PP}$. This amplitude can be varied using the potentiometers provided.

4. Measure and record the RF signal frequency using frequency counter. (or CRO).

5. Set the amplitudes of the RF signals to 0.1 V_{p-p} and connect direct signal to one balanced modulator and 90^o phase shift signal to another balanced modulator.

6. Select the required frequency (2KHz, 4KHz or 6KHz) of the AF generator with the help of switch and adjust the AGC potentiometer until the output amplitude is ⍰ 10 V_{PP} (when amplitude controls are in maximum condition).

7. Measure and record the AF signal frequency using frequency counter (or CRO).

8. Set the AF signal amplitudes to 8 V_{p-p} using amplitude control and connect to the balanced modulators.

9. Observe the outputs of both the balanced modulators simultaneously using Dual trace oscilloscope and adjust the balance control until desired output wave forms (DSB-SC).

10. To get SSB lower side band signal, connect balanced modulator output (DSB_SC) signals to subtract or.

11. Measure and record the SSB signal frequency.

12. Calculate theoretical frequency of SSB (LSB) and compare it with the practical value.

    LSB frequency = RF frequency – AF frequency

13. To get SSB upper side band signal, connect the output of the balanced modulator to the summer circuit.

14. Measure and record the SSB upper side band signal frequency.

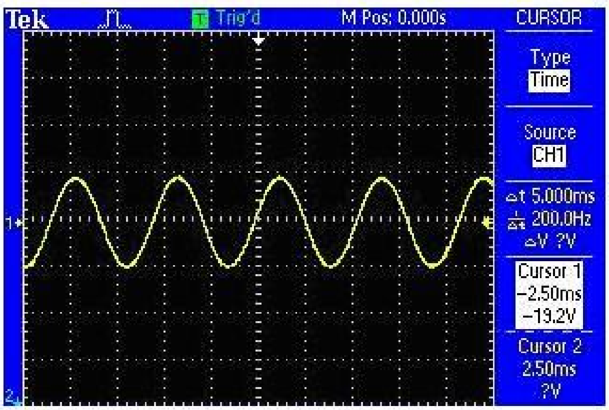
15. Calculate theoretical value of the SSB(USB) frequency and compare it with practical value.

USB frequency = RF frequency + AF frequency

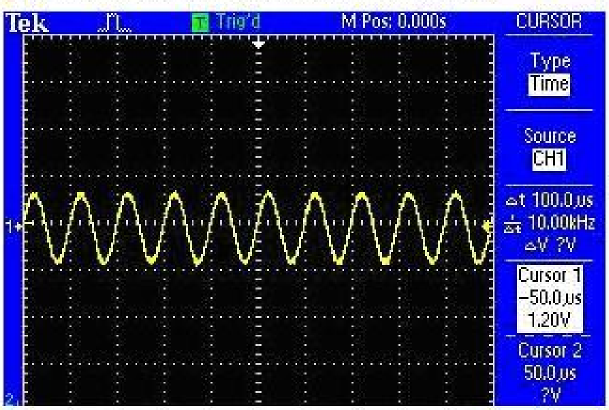
Observations:

signal	Amplitude(volts)	Frequency(KHz)
Message signal		
Carrier signal		
SSB(LSB)		
SSB(USB)		

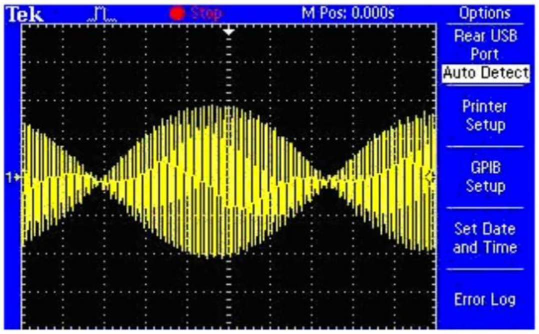
Waveforms:



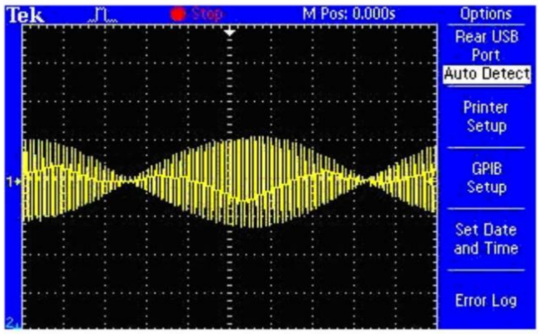
MESSAGE SIGNAL



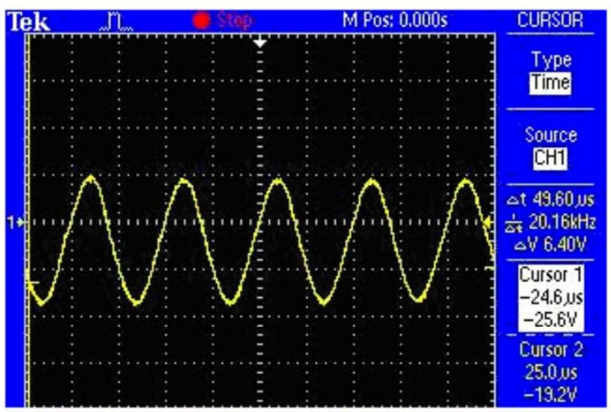
CARRIER SIGNAL



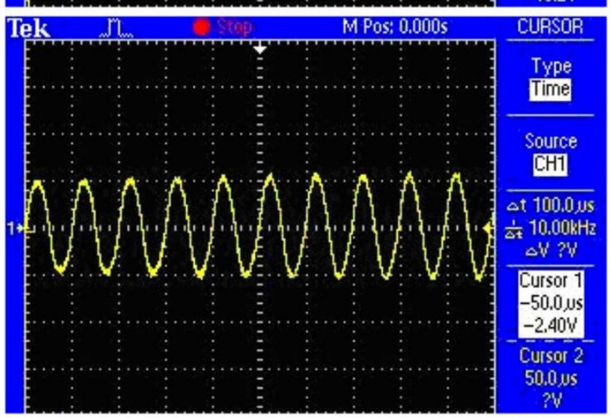
DSB-SC
SIGNAL(A)



DSB-SC
SIGNAL(B)



LSB SIGNAI



USB SIGNAI

Precautions:

1. Check the connections before giving the power supply
2. Observations should be done carefully.

Results:

Thus the SSB signal is generated and demodulated

Questions:

1. What are difficulties in practical implementation of SSB-C system?
2. Why SSB-SC is not used in broadcasting?

Ex No:4

Date:

FREQUENCY MODULATION AND DEMODULATION

Aim:

1.To generate frequency modulated signal and determine the modulation index and bandwidth for various values of amplitude and frequency of modulating signal.

2.To demodulate a Frequency Modulated signal using FM detector.

Apparatus required:

Name of the Component/Equipment	Specifications/Range	Quantity
IC 566	Operating voltage –Max-24 Volts Operating current-Max.12.5 mA	1
IC 8038	Power dissipation – 750mW Supply voltage - ±18V or 36V total	1
IC 565	Power dissipation -1400mw Supply voltage - ±12V	1
Resistors	15 K, 10 K, 1.8 K, 39 K, 560	1,2,1 2,2
Capacitors	470 pF, 0.1µF 100pF , 0.001µF	2,1 1,1 each
CRO	100MHz	1
Function Generator	1MHz	2
Regulated Power Supply	0-30 v, 1A	1

Theory:

The process, in which the frequency of the carrier is varied in accordance with the instantaneous amplitude of the modulating signal, is called "Frequency Modulation". The FM signal is expressed as

$$s(t) = A_c \cos(2\pi f_c t + \beta \sin(2\pi f_m t))$$

Where A_c is amplitude of the carrier signal, f_c is the carrier frequency

β is the modulation index of the FM wave

Circuit Diagrams:

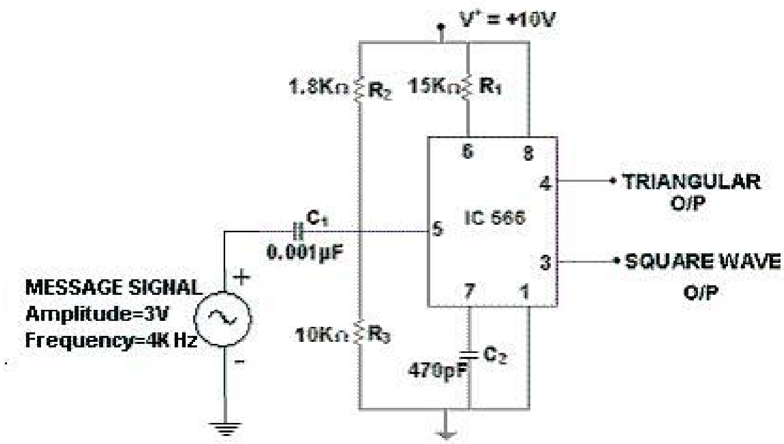


Fig.1. FM Modulator Using IC 566

By using IC 8038:

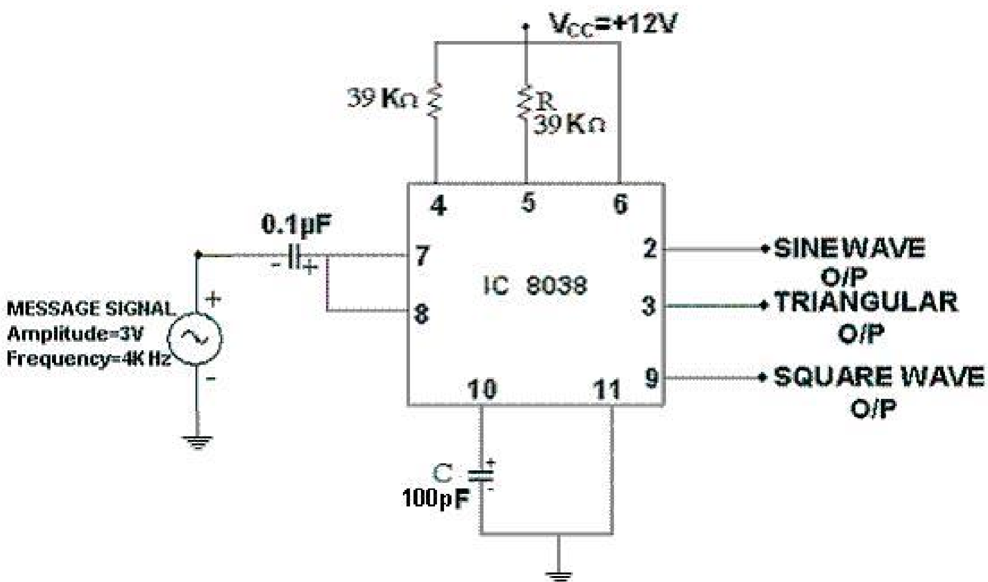


Fig.2. FM Modulator Circuit

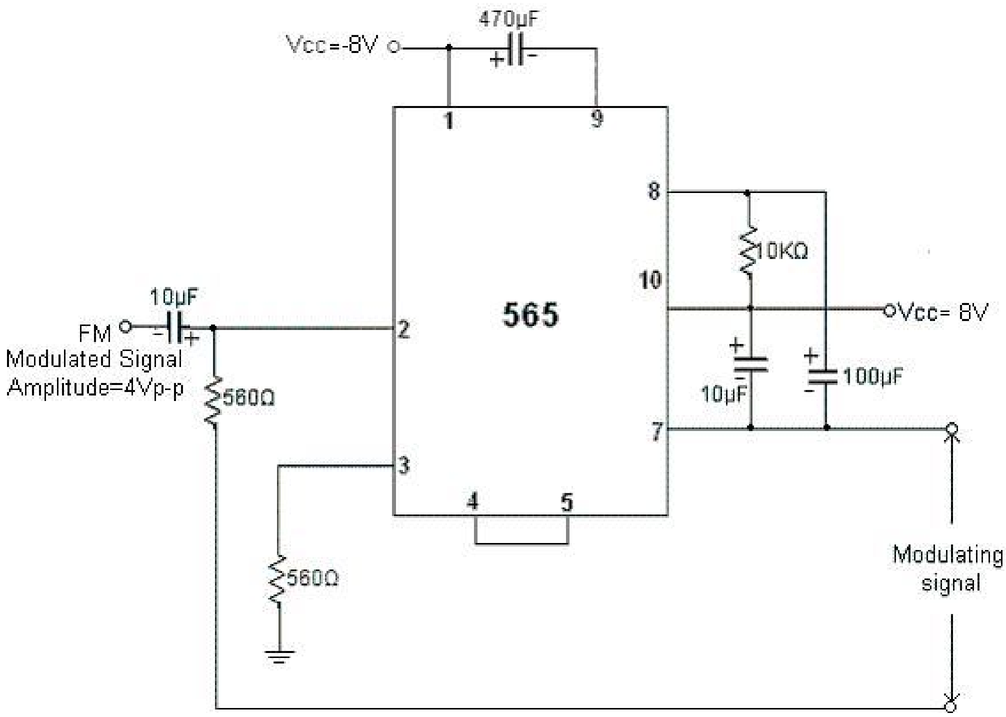


Fig.3. FM Demodulator Circuit

Procedure:

Modulation:

1. The circuit is connected as per the circuit diagram shown in Fig.2(Fig.1 for IC 566)

2. Without giving modulating signal observe the carrier signal at pin no.2 (at pin no.3 for IC 566). Measure amplitude and frequency of the carrier signal. To obtain carrier signal of desired frequency, find value of R from f = 1/ (2ΠRC) taking C=100pF.

3. Apply the sinusoidal modulating signal of frequency 4KHz and amplitude 3Vp-p at

   pin no.7. (pin no.5 for IC 566)

   Now slowly increase the amplitude of modulating signal and measure f_{min} and maximum frequency deviation Δf at each step. Evaluate the modulating index ($m_f = \beta$) using $\Delta f / f_m$ where $\Delta f = |f_c - f_{min}|$.

   Calculate Band width. $BW = 2(\beta + 1)f_m = 2(\Delta f + f_m)$

4. Repeat step 4 by varying frequency of the modulating signal.

Demodulation:

1. Connections are made as per circuit diagram shown in Fig.3
2. Check the functioning of PLL (IC 565) by giving square wave to input and observing the output
3. Frequency of input signal is varied till input and output are locked.
4. Now modulated signal is fed as input and observe the demodulated signal (output) on CRO.
5. Draw the demodulated wave form.

Observation Table:

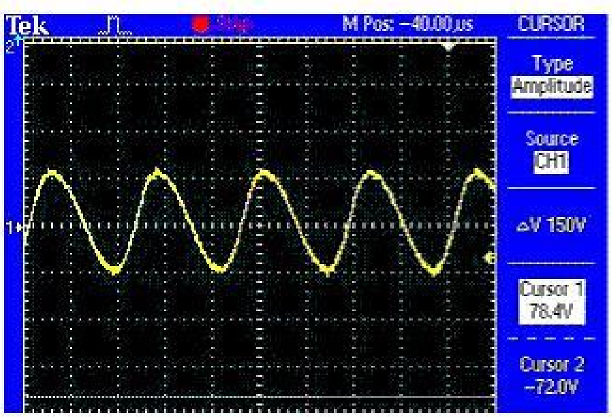
Table: 1 f_c =

S.No.	f_m(KHz)	T_{max} (μsec)	f_{min}(KHz)	f(KHz)	β	BW (KHz)
1						
2						

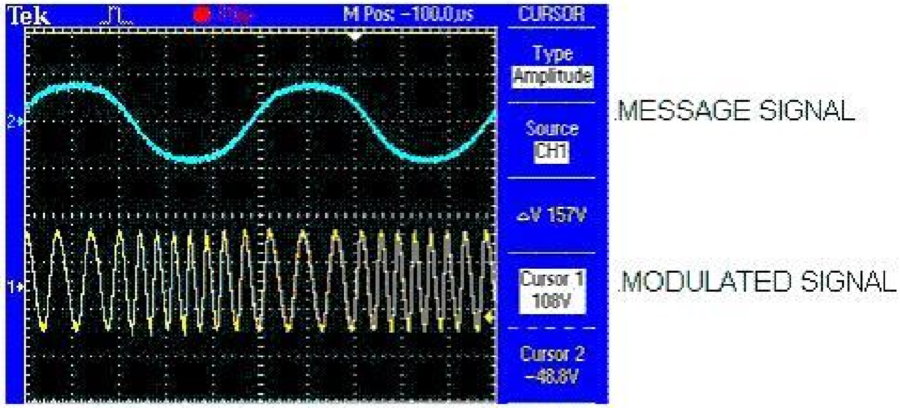
Table 2: f_m = f_c =

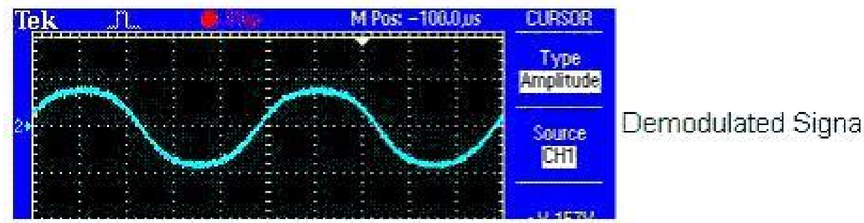
S.No.	A_m (Volts)	T (μsec)	f_{min}(KHz)	Δf (KHz)	β	BW(KHZ)
01						
02						

Model Waveforms:



CARRIER SIGNAL





Precautions:

1. Check the connections before giving the power supply
2. observations should be done carefully

Result:

1.The FM signal is generated and Modulation Index were calculated

2.Demodulation of FM were done by FM Detector.

Questions:

1. Effect of the modulation index on FM signal?

2. In commercial FM broadcasting, what is highest value of frequency deviation and audio frequency to be transmitted?

Ex No:5

Date:

PULSE AMPLITUDE MODULATION & DEMODULATION

Aim: To generate the Pulse Amplitude modulated and demodulated signals.

Apparatus required:

Name of the Apparatus	Specifications/Range	Quantity
Resistors	1K, 10K, 100K, 5.8K, 2.2K,	Each one
Transistor	BC 107	2
Capacitor	10µF, 0.001µF	each one
CRO	30MHz	1
Function generator	1MHz	1
Regulated Power Supply	0-30V,1A	1
CRO Probes	---	1

Theory:

PAM is the simplest form of data modulation .The amplitude of uniformly spaced pulses is varied in proportion to the corresponding sample values of a continuous message m (t).

A PAM waveform consists of a sequence of flat-topped pulses. The amplitude of each pulse corresponds to the value of the message signal x (t) at the leading edge of the pulse.

The pulse amplitude modulation is the process in which the amplitudes of regularity spaced rectangular pulses vary with the instantaneous sample values of a continuous message signal in a one-one fashion. A PAM wave is represented mathematically as,

$$S(t) = \sum_{N=-\infty}^{\infty} [1+K_a\, x(nT_s)]\, P(t-nT_s)$$

Where

x (nT_s) ==> represents the n^{th} sample of the message signal x(t)

K= ==> is the sampling period.

K_a ==> a constant called amplitude sensitivity

P (t) ==>denotes a pulse

PAM is of two types

1) Double polarity PAM ==> This is the PAM wave which consists of both positive and negative pulses shown as

2) Single polarity PAM ==> This consists of PAM wave of only either negative (or)

Positive pulses. In this the fixed dc level is added to the signal to ensure single polarity signal. It is represented as

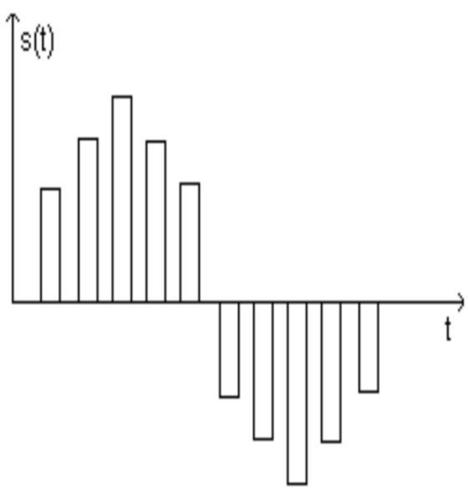


Fig: 1 Bipolar PAM signal

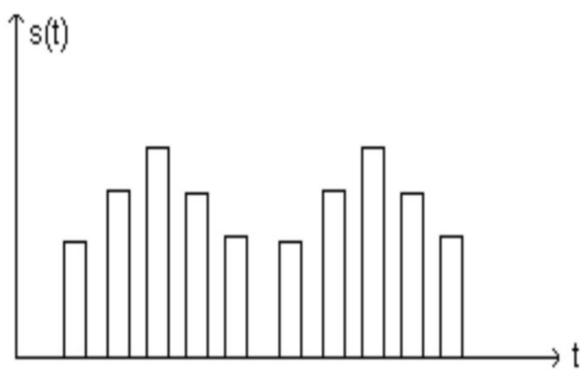


Fig: 2 Single polarity PAM

Circuit Diagram:

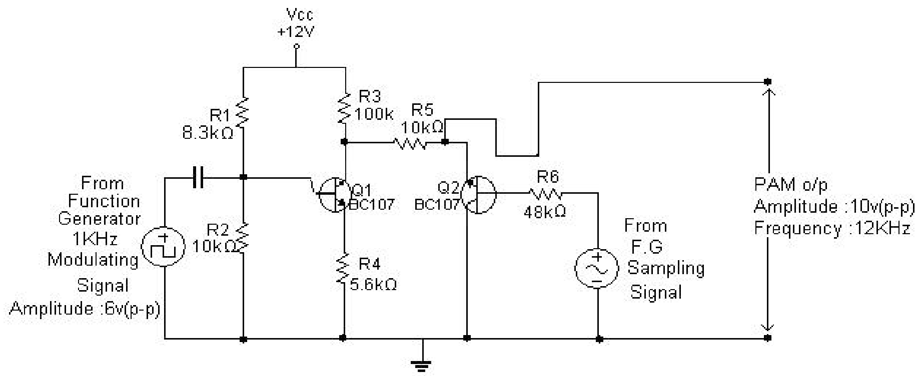


Fig: 3 Pulse Amplitude Modulation Circuit

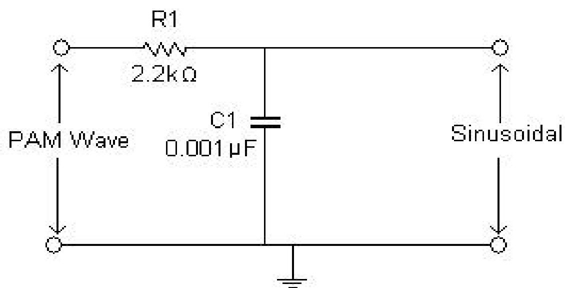


Fig: 4 Demodulation Circuit

Procedure:

1. Connect the circuit as per the circuit diagram shown in the fig 3
2. Set the modulating frequency to 1KHz and sampling frequency to 12KHz
3. Observe the o/p on CRO i.e. PAM wave.
4. Measure the levels of E_{max} & E_{min}.
5. Feed the modulated wave to the low pass filter as in fig 4.
6. The output observed on CRO will be the demodulated wave.
7. Note down the amplitude (p-p) and time period of the demodulated wave. Vary the amplitude and frequency of modulating signal. Observe and note down the changes in output.
8. Plot the wave forms on graph sheet.

Result:

Thus the amplitude modulation and the demodulation output was taken, graph was plotted and output verified.

Ex No:6

Date:

TDM MULTIPLEXER AND DEMULTIPLEXER

Aim:

To study the Time Division Multiplexing (TDM) and draw its waveforms.

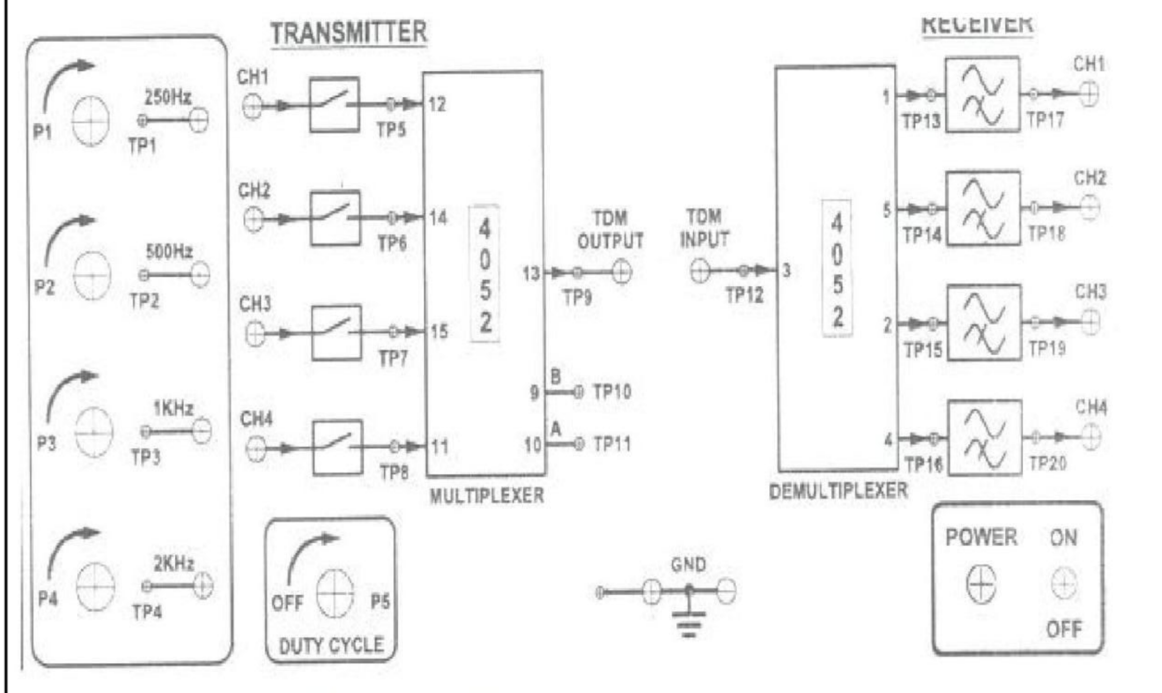
EQUIPMENT:

1. Time Division Multiplexing and Demultiplexing Trainer kit.
2. Oscilloscope-30MHz dual channel.
3. Patch chords.
4. Power supply.

THEORY:

The TDM is used for transmitting several analog message signals over a communication channel by dividing the time frame into slots, one slot for each message signal. The four input signals, all band limited by the input filters are sequentially sampled, the output of which is a PAM waveform containing samples of the input signals periodically interlaced in time. The samples from adjacent input message channels are separated by Ts/M, where M is number of input channels. A set of M pulses consisting of one sample from each of the input M-input channels is called a frame.

At the receiver the samples from individual channels are separated by carefully synchronizing and is critical part of TDM. The samples from each channel are filtered to reproduce the original message signal. There are two levels of synchronization. Frame synchronization is necessary to establish when each group of samples begin and word synchronization is necessary to properly separate samples within each frame. Besides the space diversity & frequency diversity there is a method of sending multiple analog signals on a channel using “TIME DIVISION MULTIPLEXING & DEMULTIPLEXING” technique.



BLOCK DIAGRAM OF TIME DIVISION MULTIPLEXING AND DEMULTIPLEXING

PROCEDURE:

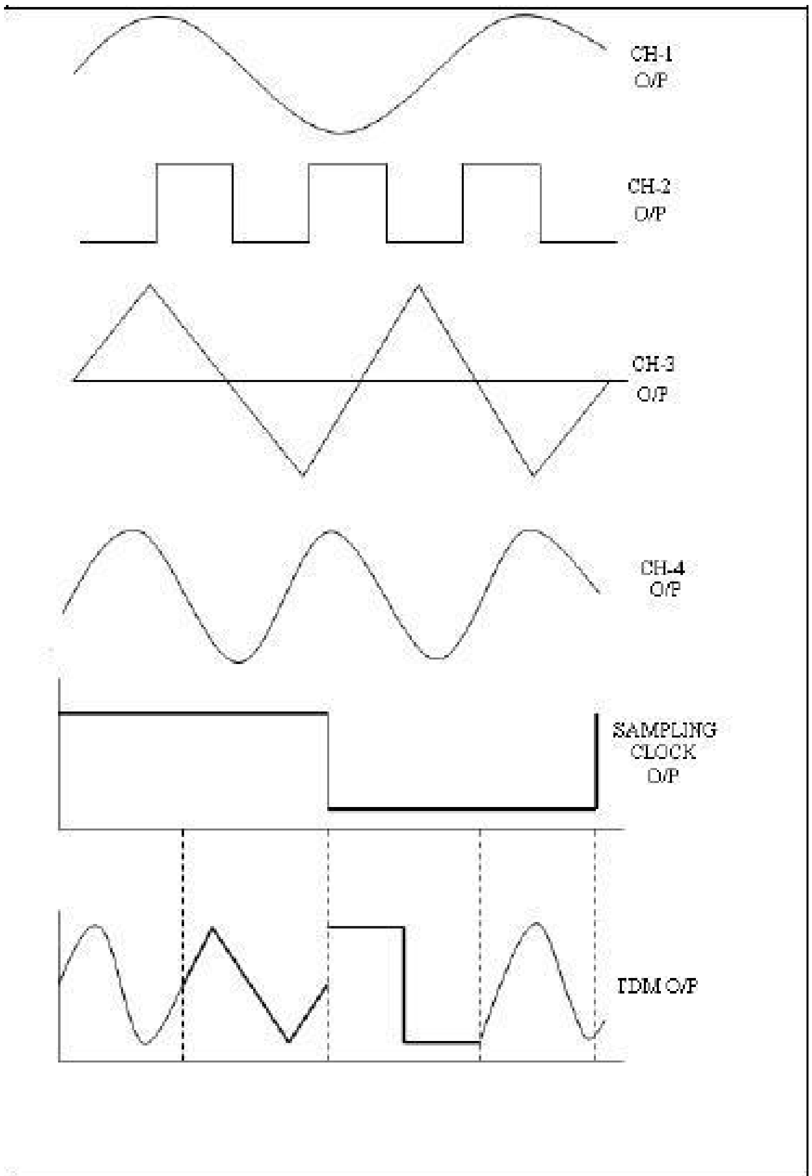
Multiplexing:

1. Connect the circuit as shown in diagram.
2. Switch ON the power supply.
3. Set the amplitude of each modulating signal as 5Vp-p.
4. Monitor the outputs at test points 5, 6, 7, 8. These are natural sampling PAM outputs.
5. Observe the outputs by varying the duty cycle pot(P5). The PAM outputs will varying with 10% to 50% duty cycle.
6. Try varying the amplitude of modulating signal corresponding each channel by using amplitude pots P1, P2, P3, P4. Observe the effect on all outputs.
7. Observe the TDM output at pin no. 13(at TP9) of 4052. All the multiplexer channels are observed during the full period of the clock (1/32 KHz).

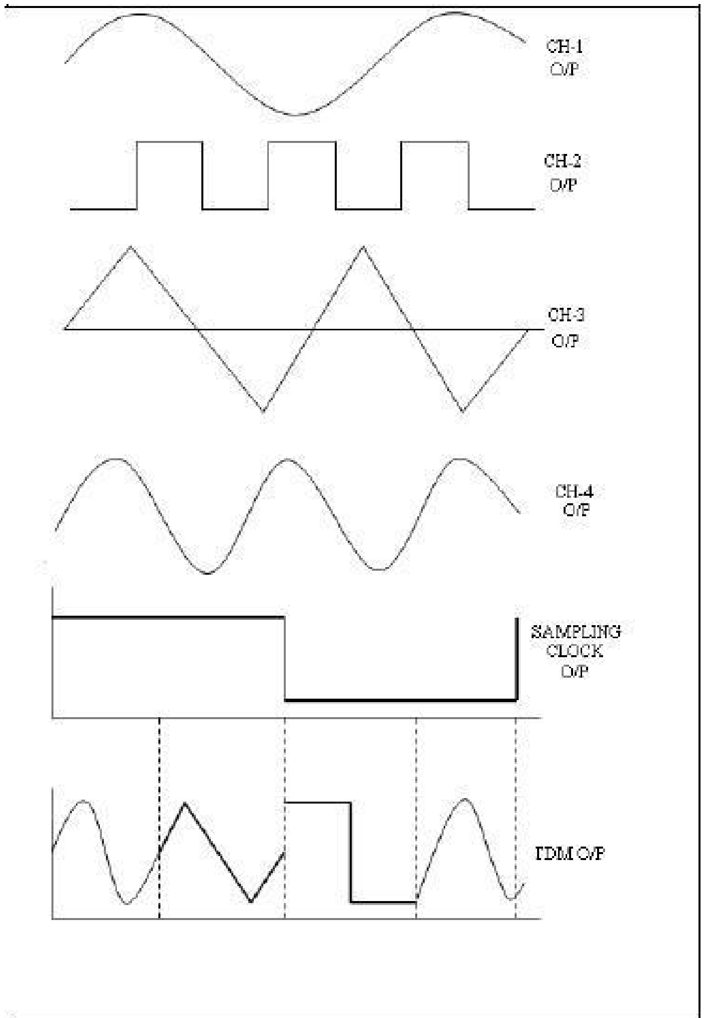
Demultiplexing & low pass filter:

1. Connect the circuit as shown in diagram.
2. Observe the demultiplexed outputs at test points 13, 14, 15, 16 respectively.
3. Observe the effect on the outputs by varying the duty cycle pot P5.
4. Observe the low pass filter outputs for each channel at test points 17, 18, 19, 20 and at sockets channels CH1, CH2, CH3, CH4. These signals are true replica of the inputs. These signals have lower amplitude.

MODEL GRAPHS



MODEL GRAPHS



OBSERVATIONS:

S NO	Amplitude(Vp-p)	Frequency	Time Period
Message signals			
CH1			
CH2			
CH3			
CH4			
TDM signal			
Demultiplexed signal			
CH1			
CH2			
CH3			
CH4			

PRECAUTIONS:

1. Connect power supply with proper polarity.
2. Do not make any interconnections when power supply is ON.

RESULT

Ex No:7

Date:

FDM MULTIPLEXER AND DEMULTIPLEXER

Aim:

To study the Frequency Division Multiplexing (FDM) and draw its waveforms.

Apparatus Required:

1. ACL 06 FDM kit
2. Digital Storage Oscilloscope (DSO)
3. Power supply
4. Patch cords

Procedure:

1.The connections are given as per the block diagram.
2.Connect the power supply in proper polarity to the kit and & switch it on.
Observe the following waveforms at the
a.Input Channel
b.Multiplexer Output (TXD)
c.Reconstructed Signal (OUT0, OUT1,OUT2,OUT3)
and plot it on graph paper

Block Diagram:

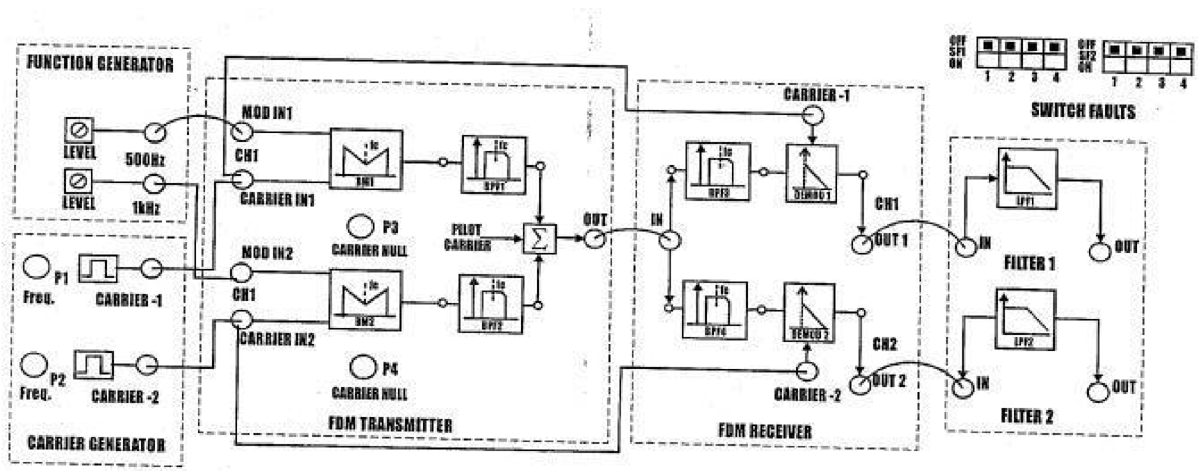
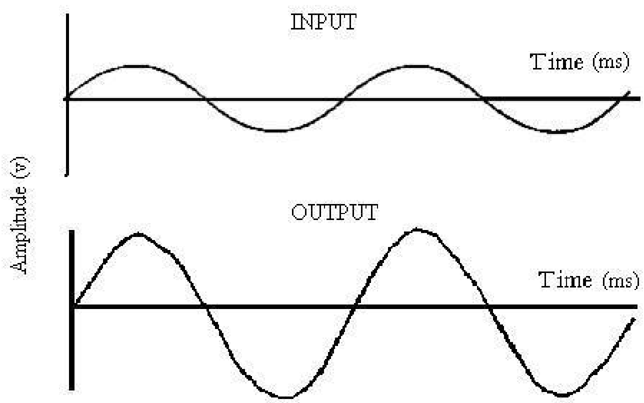


Fig 8.1b Block diagram for Frequency Division Multiplexing

Tabular Column:

SIGNAL	AMPLITUDE (V)	TIME PERIOD (s)	Frequency(Hz)
Input 1			
Input 1			
Modulated Input			
Demodulated Output 1			
Demodulated Output 2			

Model graph



RESULT

FDM Modulation and Demodulation are verified in the hardware kit and its waveforms are studied.

Ex No:8

Date:

PRE EMPHASIS AND DE-EMPHASIS IN FM

Aim:

I) To observe the effects of pre-emphasis on given input signal.

ii) To observe the effects of De-emphasis on given input signal.

Apparatus Required:

Name of the Component/Equipment	Specifications/Range	Quantity
Transistor (BC 107)	f_T = 300 MHz P_d = 1W I_c(max) = 100 mA	1
Resistors	10 K, 7.5 K, 6.8 K	1 each
Capacitors	10 nF 0.1 µF	1 2
CRO	20MHZ	1
Function Generator	1MHZ	1
Regulated Power Supply	0-30V, 1A	1

Theory:

The noise has a effect on the higher modulating frequencies than on the lower ones. Thus, if the higher frequencies were artificially boosted at the transmitter and correspondingly cut at the receiver, an improvement in noise immunity could be expected, there by increasing the SNR ratio. This boosting of the higher modulating frequencies at the transmitter is known as pre-emphasis and the compensation at the receiver is called de-emphasis.

Pre-emphasis

The circuits are the transmitting side of the frequency modulator. It is used to increase the gain of the higher frequency component as the input signal frequency increased, the impendence of the collector voltage increase. If the signal frequency is lesser then the impendence decrease which increase the collector current and hence decrease the voltage.

De-emphasis

The circuit is placed at the receiving side. It acts as allow pass filter. The boosting gain for higher frequency signal in the transmitting side is done by the pre-emphasis circuit is filtered to the

same value by the low pass filter. The cut off frequency is given by the formula

$$f_c = 1/(2p\ RC) \qquad (4\text{-}1)$$

Where R = 2 p fc L

Circuit Diagrams:

For Pre-emphasis:

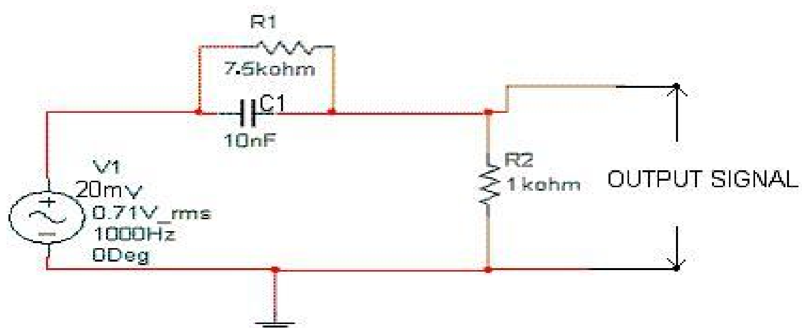


Fig.1. Pre-emphasis circuit

For De-emphasis:

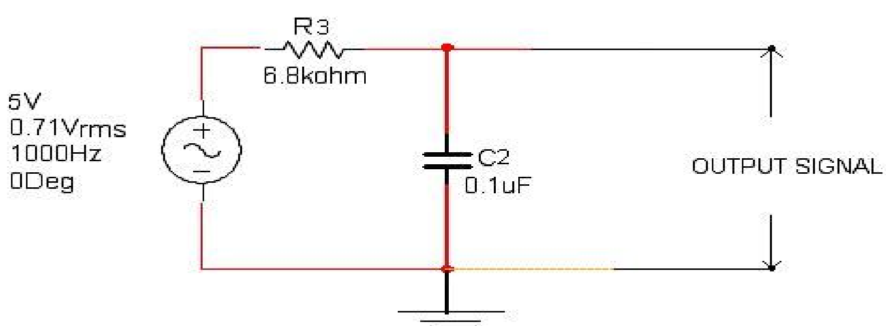


Fig.2. De-emphasis circuit

Procedure:

1. Connect the circuit as per circuit diagram as shown in Fig.1.

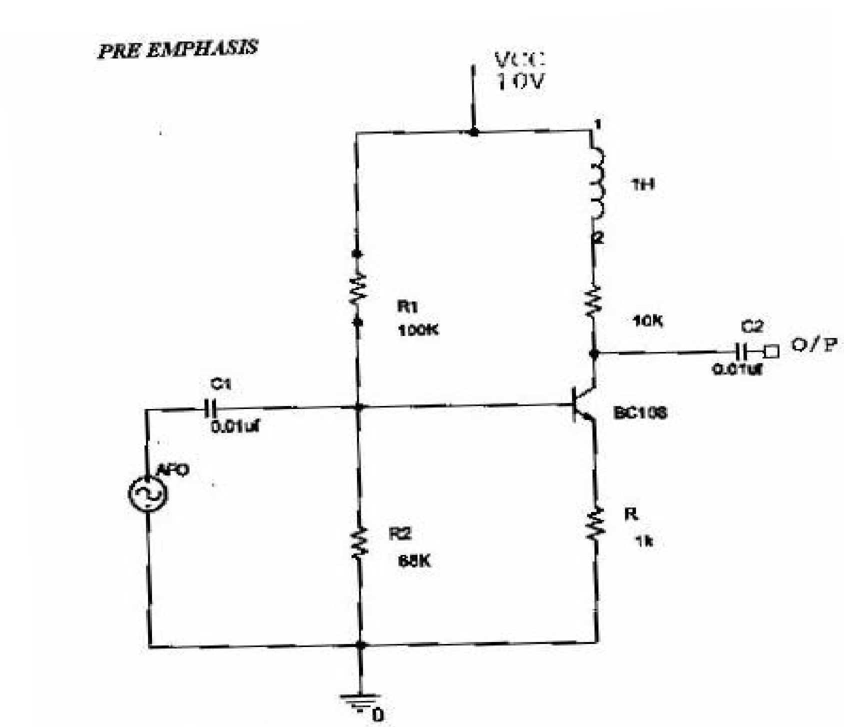
2. Apply the sinusoidal signal of amplitude 20mV as input signal to pre emphasis circuit.

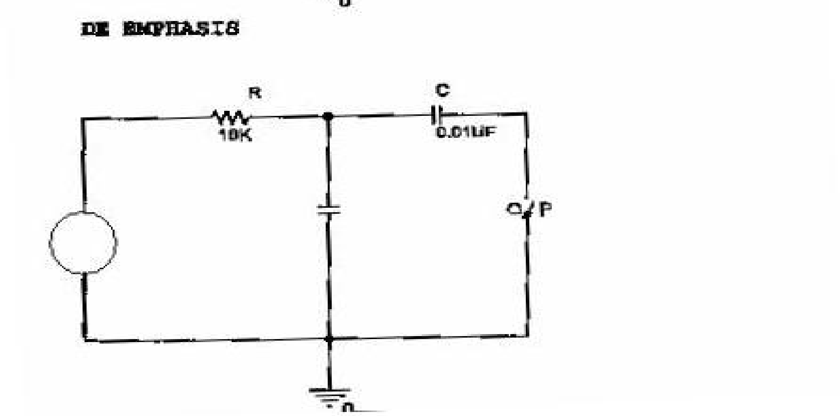
3. Then by increasing the input signal frequency from 500Hz to 20KHz, observe the output voltage (v_o) and calculate gain (20 log (v_o/v_i)).

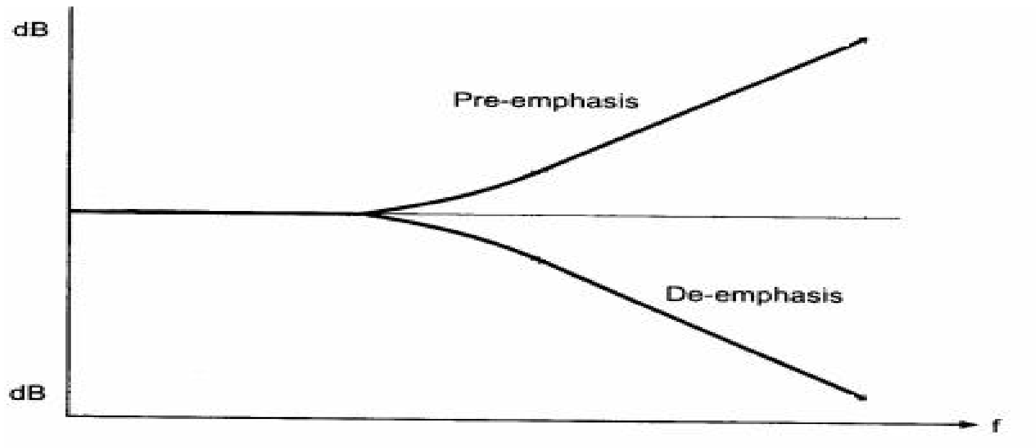
4. Plot the graph between gain Vs frequency.

5. Repeat above steps 2 to 4 for de-emphasis circuit (shown in Fig.2). by applying the sinusoidal signal of 5V as input signal.

(OR)







Pre-Emphasis & De-Emphasis Model Graph

DESIGN FORMULA

f_C **=** 1/(2 p RC)   (assume =R = 10 KO, C = 0.01µf)

R = 2 pf_CL;   L=1/(2pf_C)

PROCEDURE:

1. The circuit connection are made as shown in the circuit diagram for the pre-emphasis and de-emphasis circuits
2. A power supply of 10V is given to the circuit
3. For a constant value of input voltage the values of the frequency is varied and the output is noted on the CRO
4. A graph is plotted between gain and frequency

The cut frequencies are practical values of the values of cut off frequency \are found, compared and verified

RESULTS:

The characteristics of pre –emphasis and de emphasis circuits were studied and a graph was

drawn between gain (in db) and frequency.

Ex No:9

Date:

SIMULATION EXPERIMENTS USING P-SPICE AND MATLAB.

Aim:

i) AM modulator with AWGN noise in Matlab.
ii) Pre-emphasis and De-emphasis in FM using P-SPICE

AM MODULATION WITH AWGN NOISE IN MATLAB

Aim:

To write and simulate a MATLAB program for amplitude modulation with $v_c > v_m$, $f_c > f_m$ and add an additive white Gaussian noise with SNR=10dB and analyze by varying the SNR value.

SOFTWARE REQUIRED:

MATLAB, Computer installed with Windows XP or higher Version.

MATLAB® INTRODUCTION

MATLAB® is a programming language and numerical computing environment. The name MATLAB® is an acronym for "Matrix Laboratory". As it name suggests it allows easy manipulation of matrix and vectors. Plotting functions and data is made easy with MATLAB®. It has a good Graphic User Interface and conversion of matlab files to C/C++ is possible. It has several toolboxes that possess specific functions for specific applications. For example Image Processing, Neural Networks, CDMA toolboxes are name a few. An additional package, Simulink, adds graphical multidomain simulation and Model-Based Design for dynamic and embedded systems. Simulink contains Blocksets that is analogous to Toolboxes. It was created by Mathworks Incorporation, USA. Writing MATLAB programs for modulation applications require knowledge on very few functions and operators. The operators mostly used are arithmetic operators and matrix operators. To know more type in the command prompt 'help ops'. MATLAB will give a list in that to know on specific operator say addition type in the command prompt 'help plus'. MATLAB will give how to use and other relevant information.

Commonly used graphical functions are plot, figure, subplot, title, and mathematical functions are sin and cos only. The mathematical functions sin and cos are self-explanatory. The graphical function figure will create a new window and then subsequent graphical commands can be applied. The plot function usually takes two vectors and plot data points according to given vector data. Subplot function is used when two or more plots are drawn on the same figure. As title function suggests it helps to write title of the graph in the figure. For further details type ‘help plot’ or ‘help subplot’ in the command prompt and learn the syntax.

ALGORITHM:

1. Create a vector ‘t’ (time) that varies from zero to two or three cycles.

2. Create a message signal with single sine frequency or combination of few sine frequencies. All the frequency used in message signal should be less than the carrier frequency likewise amplitude of message signals should be less than carrier amplitude.[$A_m Sin(2\pi f_m t)$]

3. Create a carrier signal. [$A_c Sin(2\pi f_c t)$]

4. Create the modulated signal using the AM equation A_m [$(1+m_a Sin(2\pi f_m t)]Sin(2\pi f_c t)$

5. Introduce AWGN noise and observe the changes in the amplitude modulated signal.

6. Plot the message signal, carrier signal, and amplitude modulated signal with AWGN noise.

PROGRAM FOR AMPLITUDE MODULATION

```
clc;

clear all;

t=0:0.001:1;

vm=5;

vc=10;

fm=2;
```

```
fc=25;
m=vm*sin(2*pi*fm*t);
c=vc*sin(2*pi*fc*t);
amp=vc+vm*sin(2*pi*fm*t);
am=amp.*sin(2*pi*fc*t);
y=awgn(am,10,'measured');
subplot(4,1,1);
plot(t,m);
xlabel('time');
ylabel('amplitude');
title('message signal');
subplot(4,1,2); plot(t,C);
xlabel('time');
ylabel('amplitude');
title('carrier signal');
subplot(4,1,3); plot(t,AM);
xlabel('time');
ylabel('amplitude');
title('amplitude modulated signal');
subplot(4,1,4);
plot(t,y);
xlabel('time');
ylabel('amplitude');
title('amplitude modulated signal with AWGN');
```

WITH VARYING SNR

```
clc;
clear all; t=0:0.001:1;
vm=5;
```

```
vc=10;
fm=2;
fc=25;
m=vm*sin(2*pi*fm*t);
c=vc*sin(2*pi*fc*t);
amp=vc+vm*sin(2*pi*fm*t);
am=amp.*sin(2*pi*fc*t);
y1=awgn(am,10,'measured');
y2=awgn(am,100,'measured');
y3=awgn(am,1000,'measured');
subplot(4,1,1);
plot(t,am);
xlabel('time');
ylabel('amplitude'); title('amplitude
modulated signal'); subplot(4,1,2);
plot(t,y1);
xlabel('time');
ylabel('amplitude');
title('amplitude modulated signal with AWGN [snr10]');
subplot(4,1,3);
plot(t,y2);
xlabel('time');
ylabel('amplitude');
title('amplitude modulated signal with AWGN[snr100]');
subplot(4,1,4);
```

```
plot(t,y3);

xlabel('time');

ylabel('amplitude');

title('amplitude modulated signal with AWGN[snr1000]');
```

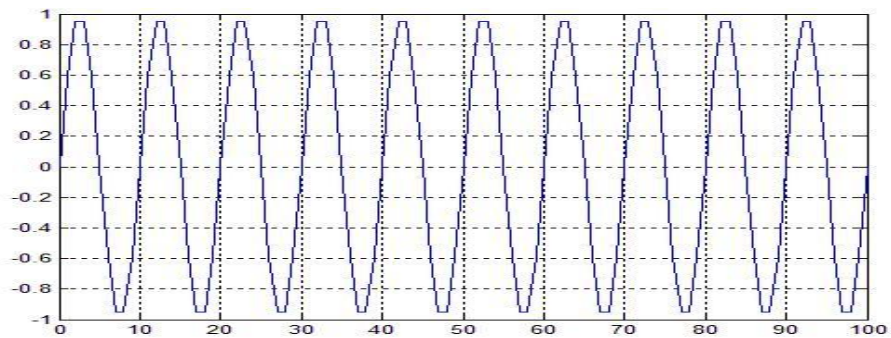
PRE LAB QUESTIONS:

1. What is Matlab?
2. What is Matlab working environment?
3. Can we run Matlab without graphics?
4. What is Simulink?
5. What is AWGN noise?

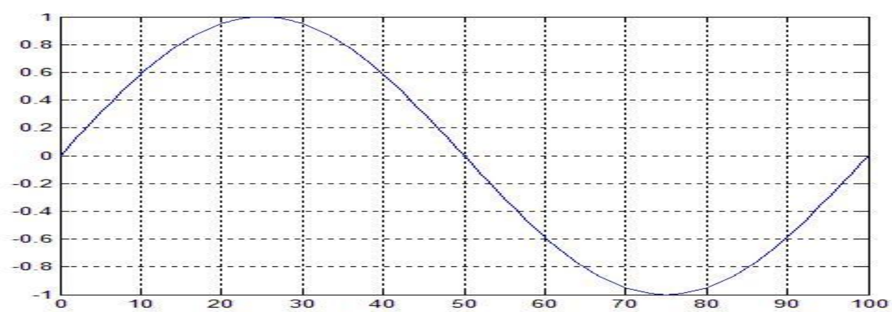
LAB PROCEDURE:

1. Open the MATLAB® software by double clicking its icon.
2. MATLAB® logo will appear and after few moments Command Prompt will appear.
3. Go to the File Menu and select a New M-file. (File New M-file) or in the left hand corner a blank white paper icon will be there. Click it once.
4. A blank M-file will appear with a title 'untitled'
5. Now start typing your program. After completing, save the M-file with appropriate name. To execute the program Press F5 or go to Debug Menu and select Run.
6. After execution output will appear in the Command window .If there is an error then with an alarm, type of error will appear in red color.
7. Rectify the error if any and go to Debug Menu and select Run.

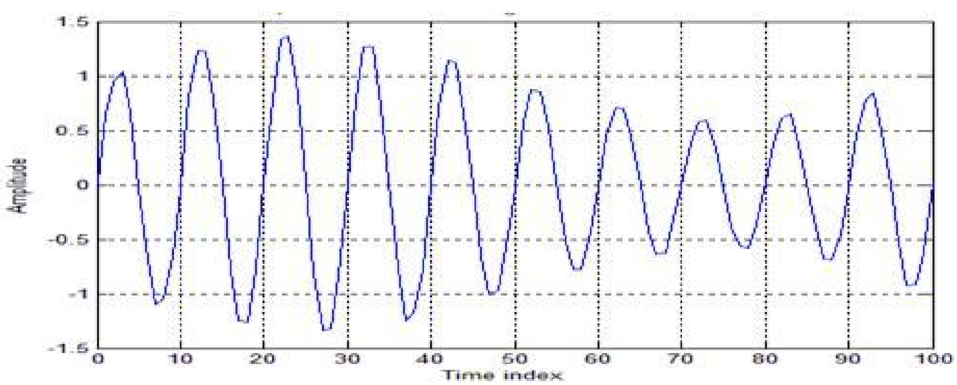
MODEL GRAPH



High Freq Signal



Message Signal



Ampitude Modulated Signal with AWGN Channel

RESULT:

Thus the amplitude modulation was simulated with AWGN noise.

PRE –EMPHASIS AND DE-EMPHASIS IN FM USING PSPICE

Aim:

To simulate the Pre emphasis and De emphasis circuit using PSPICE Program and verify the output response characteristics.

SOFTWARE REQUIRED

PSPICE – OrCAD 9.2 lite, Computer installed with Windows XP or higher Version.

ABOUT PSPICE:

SPICE is software that stimulates electronic circuits. SPICE can perform various analyses such as DC analysis, Transient analysis, AC analysis and operating point measurements. SPICE contains model for common circuit elements active as well as passive and it is possible of stimulating most electronic circuits. The abbreviation of SPICE is **S**timulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis.

A circuit is described to a computer by using a file called circuit file. The circuit file contains the circuit details of component and elements, ht information about the sources and the commands for what to calculate and what to provide as output. The circuit file is the input file to the SPICE program, which after executing the commands, produces the result in another file called output file.

The description of analysis of a circuit require specifying the following:

Element values	Types of analysis
Nodes	Output variables
Circuit elements	PSPICE output commands
Element modes	Format of output files
Sources	Format of circuit files

ELEMENT VALUES:

The elements values are written in standard floating point notation with optimal scale and unit suffixs

Ex: V=Volt, A=Ampere, Hz= Hertz, OHM= ohm

Node voltage:

The voltage of a point with respect to ground references point.

Vname	+node	-node	Value
Ex:	Vcc	10	30v

The passive component is also indicated as

Rname	+node	-node	Values
R1	2	3	10k

Format of circuit files:

The circuit file that can be read by PSPICE may be divided into 5 parts Title

Circuit description Analysis

description Output

description

.END (end of statement)

Notes:

1. The first line is the title line and it may contain any type of text.
2. The last line must be the .END command.
3. The order of the remaining lines is not important and does not affect the results of simulations.
4. If the statement is more than one line, the statements can continue on the next line. A continuation line is identified by a plus sign(+) in the first column of the next line
5. The comment line may be included anywhere, precede by an asterisk (*). Within a statement. A comment is preceded by a semicolon (;).

6. The number of blanks between items is not significant. The tabs and commas are equivalent to blanks. The tabs and commas are equivalent.

7. The statement or comments can be in either lower case or upper case.

8. If you are not sure of any command or statement, the best thing is to run the circuit file by using that command or statement and see what happened. PSPICE is user friendly software; it gives an error message in the output file that identifies a problem.

PRINT statement

.Print dc [output variables]

The print statement used to get the DC outputs. The maximum number of output variables is 8. .Print statement can be used to print all the desired output variables. The values of the output variables are printed as a table with each column corresponding to one output variable.

PLOT statement:

The result from dc analysis van also be obtained in the form of line printer plots. The plots are drawn by suing characters.

plot dc<output variables>

+[<(lower limit),value><(upper limit), value>]

The plot statement can be used to plot all the desired output variables.

PROBE Statement:

Probe is a graphics post processor/ waveform analyser for PSPICE.

probe

.probe < one or more output variables>

In the first form no output variables is specified, the .probe command writes all the node voltages and all the element currents in to the probe.dat file. In the second form where the output variables are specified, only the specified output variables to the probe.dat file.

Run the program by typing filename.cir which should place you in probe with an empty screen and a message saying that " all voltages and currents are available". That is, data for all node voltages and element currents in the circuit have been passed from pspice to probe

Select Add_Trace. Functional graphs are called "traces" by probe. At this point type the names of variables to be traced. Push ENTER to get the trace. Notice that both axes are cutomatically scaled to suit the curve.

The Zoom feature of probe displays is very useful if you are interested in an small region of the highlighted. The Zoom in on the peak of the power curve, select Zoom, and select Specify region which is highlighted.

The cursor feature of probe is very useful to find maxima or minima values. Select Cursor to go to the cursor submenu.

Once the results of the simulations are processed by the .probe command the result are available for graphical displays.

Tran statement:

Transient analysis can be performed by the .Tran command

.tran[/op]   print_step   end_time   (no_print_time [step_ceiling])   [UIC]

/op = to print detailed information about the transient analysis operating point

Print_step = it specifies the time increment between the results which are printed in tabular form into the output file.

End_time = it specifies the final time of the transient calculation.

No_print_time = it specifies a value of time, before which results will not be printed to probe for graphing.

Step_ceiling = it is the maximum time interval used by the pspice in its internal calculations which is default by end time/50.

UIC = directs pspice to use initial conditions given to inductors and capacitors in their defininf L or C statement.

Ac source:

The general transient analysis for SIN is

Sin(offset amplitude freq delay damping factor phase)

Frequency response:

.ac sweeptype points fstart fstop

Sweeptype = LIN,DEC,OCT

Fstart = starting frequency must specified in Hz. They also must be positive.

Fstop = ending frequency must specified in Hz. They also must be positive.

Points = it specifies the number of calculations for every power of ten in the frequency range.

Transfer function:

It can be used to compute the small signal dc gain, the input resistance and the output resistance of a circuit.

.Tf Vout Vin

.Tf Iout Iin

The .tf command calculates the parameters of Thevenin's and Norton equivalent circuit for the circuit file.It automatically prints the output and does not require .print,.plot or .probe statement.

Dc sweep:

The Dc sweep is also known as dc transfer characteristics. The input variable is varied over a range of values. For each value of the input variable. The dc operating and the small signal dc gain are computed by calling the small signal transfer function.

.dc LIN swname sstart send sinc

.dc oct swname sstart send np

.dc DEC swname sstart send np

.dc swname list <value>

Swname = sweep variable name may be V orI

Sstart = start value

Send = end value

Sinc = increment value of the sweep variable. It should be positive.

Np = number of steps.

.Lib statement:

A library file may be referenced in the circuit file by using the following statement.

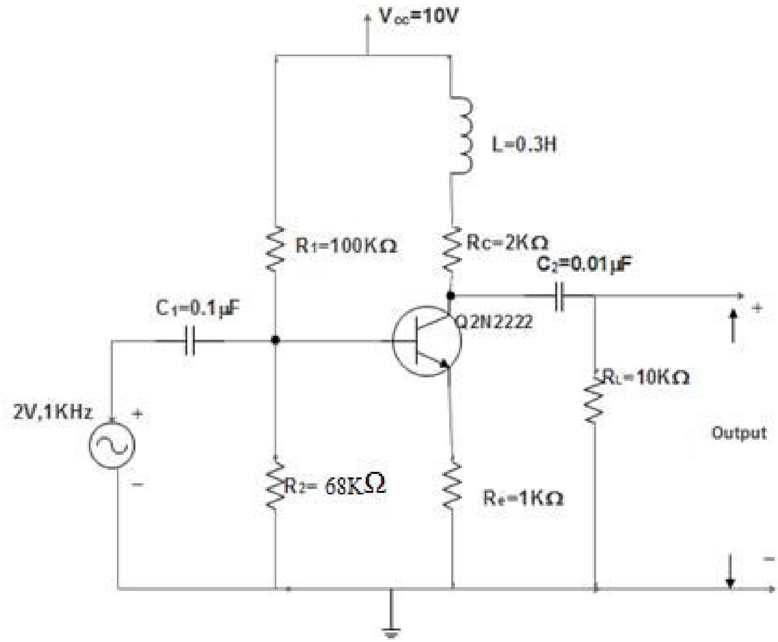
.lib fname

Fname = name of the library file.

A library file may contain comments, .model statements, sub circuit definition, .lib statements and .end statements. No other statements are permitted. If fname is omitted, pspice looks for the default file EVAL.LIB.

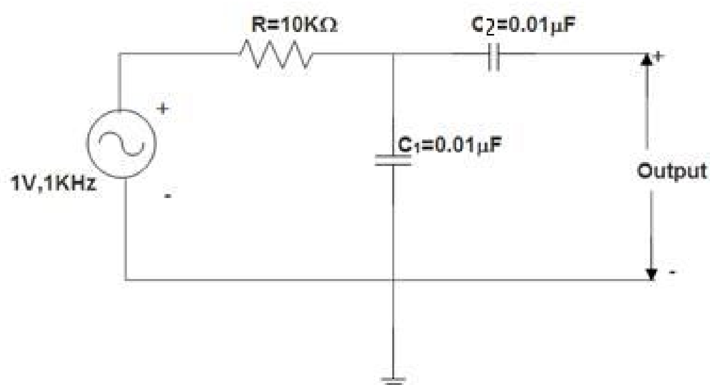
When a .lib command calls for a file, it does not bring the whole text of the library file into the circuit file. It simply reads those models or sub circuits that are called by the main circuit file.

PRE-EMPHASIS CIRCUIT DIAGRAM



Pre-emphasis Circuit

DE-EMPHASIS CIRCUIT DIAGRAM:



De-Emphasis Circuit

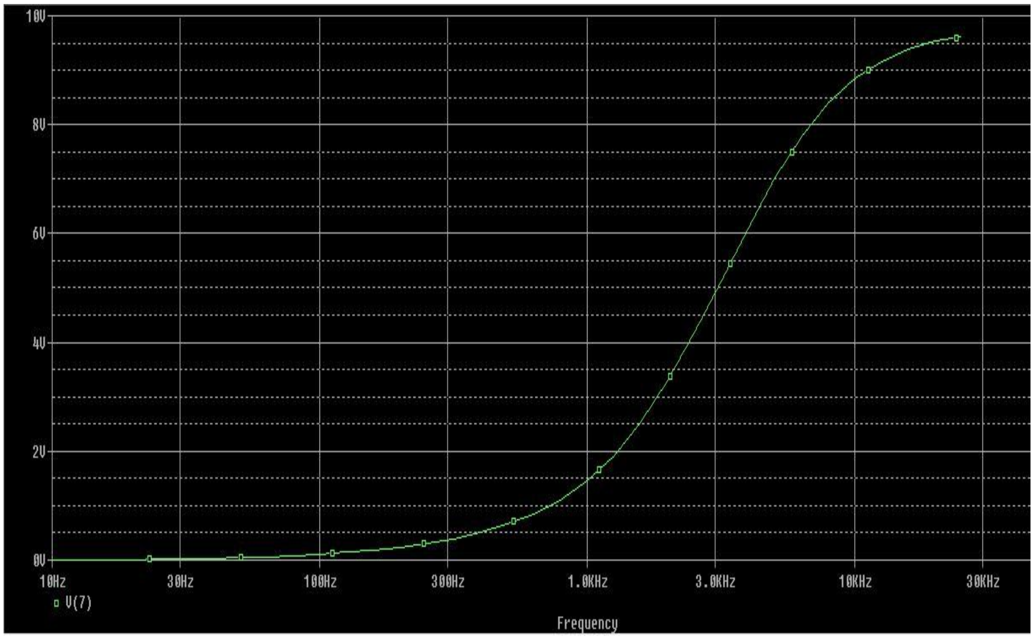
PRE-EMPHASIS PROGRAM:

```
Vm 6 0 AC 1V sin(0 1V 1KHz)

Vcc 1 0 DC 10V

R1 1 3 100K

R2 3 0 68K

L1 1 5 0.3H

R3 5 2 2K

R4 4 0 1K

C1 6 3 0.1uF

C2 2 7 0.01uF

R5 7 0 10k

Q1 2 3 4 Q2N2222

.LIB

.AC DEC 10 10Hz 20KHz

.PROBE

.END
```

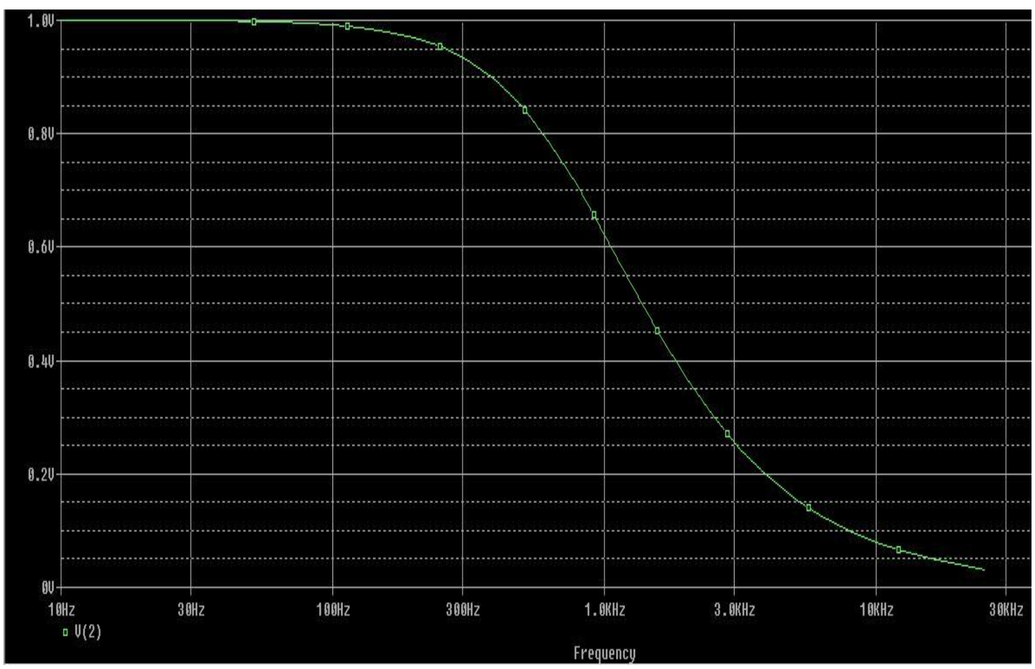
DE-EMPHASIS PROGRAM:

```
Vm 1 0 AC 1V sin(0 1V 1KHz)

R1 1 2 10k

C1 2 0 0.01uF

C2 2 0 0.01uF

.LIB

.AC dec 10 10Hz 20KHz

.PROBE

.END
```

MODEL GRAPH



Pre-emphasis Output waveform



De-emphasis output wave form

PRELAB QUESTIONS:

1. How to include the file in the circuit file?
2. What is the use of Probe comment?
3. What is threshold effect?
4. Which range of frequency is affected by noise interference?
5. How to include a device which is not already existed in pspice library?

LAB PROCEDURE:

1. Open the PSPICE AD Lit software by double clicking its icon.
2. After few moments Command window will appear.
3. Go to the File Menu and select a New text file. (File New text file)
4. A blank text file will appear with a title ‘untitled’
5. Now start typing your program. After completing, save the text file as .cir with appropriate name. To execute the program go to Debug Menu and select Run.
6. After execution output will appear in the Command window .If there is an error then with an alarm, type of error will appear.
7. Rectify the error if any and go to Debug Menu and select Run.
8. If there is no errors, go to Trace menu and click add trace. Enter the output node voltage and click ok then the output will display.

RESULT:

Thus the net list for the given pre-emphasis and de-emphasis circuit was Written and the output waveforms were plotted.