DEPARTMENT OF ELECTRONICS & COMMUNICATION ENGINEERING

ANALOG COMMUNICATION LAB + LIC LAB (10ECL58)

V SEMESTER

2016-2017

Prepared by: Soumya M J
Dept. of ECE
GCEM

Reviewed by: Kavitha M V
Head of the Department
Dept. of ECE
GCEM

Approved by: Dr. A.A. Powly Thomas
Principal
GCEM

81/1, 182/1, Hoodi Village, Sonnenahalli, K.R. Puram, Bengaluru, Karnataka-560048.
## CONTENTS

<table>
<thead>
<tr>
<th>S.No</th>
<th>Title</th>
<th>Page No</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Syllabus</td>
<td>ii</td>
</tr>
<tr>
<td>2.</td>
<td>Course objective</td>
<td>iii</td>
</tr>
<tr>
<td>3.</td>
<td>Course outcome</td>
<td>iv</td>
</tr>
<tr>
<td>4.</td>
<td>Do’s &amp; Don’ts</td>
<td>v</td>
</tr>
<tr>
<td>5.</td>
<td>List of experiments</td>
<td>vi</td>
</tr>
<tr>
<td>6.</td>
<td>Experiments</td>
<td>1-41</td>
</tr>
<tr>
<td>7.</td>
<td>Viva questions</td>
<td>42-44</td>
</tr>
<tr>
<td>8.</td>
<td>Appendix</td>
<td>45-46</td>
</tr>
</tbody>
</table>
SYLLABUS

ANALOG COMMUNICATION LAB + LIC LAB

Subject Code: 10ECL58

ia Marks: 25

No. of Practical Hrs/Week: 03

Exam Hours: 03

Total no. of Practical Hrs.: 42

Exam Marks: 50

EXPERIMENTS USING DESCERTE COMPONENTS and LABVIEW- 2009 CAN BE USED FOR VERIFICATION AND TESTING.

1. Second order active LPF and HPF
2. Second order active BPF and BE
3. Schmitt Trigger Design and test a Schmitt trigger circuit for the given values of UTP and LTP
4. Frequency synthesis using PLL.
5. Design and test R-2R DAC using op-amp
6. Design and test the following circuits using IC 555
   a. Astable multivibrator for given frequency and duty cycle
   b. Monostable multivibrator for given pulse width W
7. IF amplifier design
8. Amplitude modulation using transistor/FET (Generation and detection)
9. Pulse amplitude modulation and detection
10. PWM and PPM
11. Frequency modulation using 8038/2206
12. Precision rectifiers – both Full Wave and Half Wave.
COURSE OBJECTIVES:

- Study the second order low pass filter and its characteristics.
- Study the second order High pass filter and its characteristics.
- Study about the Second order active Band Pass Filter.
- Study about the Second order active Band Stop Filter.
- Design the Schmitt trigger circuit for given values of UTP and LTP. Test the Schmitt trigger circuit for the given values of UTP and LTP.
- Study the technique of generating higher frequencies using PLL. Understand circuit operations for Frequency synthesis using PLL.
- Design of R-2R DAC using op-amp. Testing of the designed R-2R DAC.
- Design and Test an Astable multivibrator for a given frequency and duty cycle.
- Design and test a Monostable multivibrator for a given Gate Width.
- Study about IF amplifiers and Mixer circuits.
- Study the technique of generation of Amplitude modulation using transistor/FET.
- Understand the method of demodulation using Envelope detectors.
- Study simple circuit for the generation of Pulse amplitude modulation waveforms.
- Study de-modulation technique.
- Study generation of Pulse Width Modulation and Pulse Position Modulation using 555 timer circuit.
- Study how the Frequency modulation output is generated using 8038/2206.
- Study Precision rectifier circuits – both Full Wave and Half Wave.
COURSE OUTCOMES:

- It helps students to learn the design of filters which are the basic building blocks in any communication system and will understand various terminologies used therein.
- LPF are used in the transmitter circuits for band limiting the base band signals and at the receiver end to demodulate and get back the original base band data.
- It helps the students to learn the basic working of band pass filters and band elimination filters.
- Band pass filters are used in several analog modulation techniques like SSB, VSB etc.
- It helps the students design a Schmitt Trigger for the given threshold points.
- They analyze the sine to square wave conversion of the signals using op-amp. It’s a type of comparator with 2 different threshold voltage levels. Student will learn about comparators and zero cross detectors.
- Students learn to manipulate the VCO pins for up-conversion and down conversion.
- GSM local oscillator modules are typically built with a frequency synthesizer IC and discrete resonator VCOs.
- Students learn the basics of Digital to Analog conversion. Students can understand other techniques of DAC
- Student learns the technique of generating square wave signals of different period and duty cycles.
- Student learns how to generate gate pulses using 555 monostable multivibrators for counter applications/TDM applications and provides ideas for VLSI design.
- Students understand the concept high level modulation and low level modulation. (AM).
- Students understand the principles of Pulse amplitude modulation.
- Students understand, step –by – step, usage of ladder filters for demodulation of PAM
- Student learns the basic principle of PWM and PPM.
- Student learns the art of rectifying signals whose voltages are below the cut-in voltage of the rectifying diode. They also understand how to analyze such a circuit.
- The student learns basics behind the generation of FM using IC chips.
**DOS & DON’TS**

**DOS**

It is COMPULSORY to wear covered shoes when entering the lab.

You must keep your bags at the designated area. Bags should NOT be placed on or under the workbench.

Chairs and stools should be kept under the workbenches when not in use. Sit upright on chairs or stools, keeping feet on the floor.

Follow the instructions of your lab demonstrator while conducting the experiments.

Power must be switched off whenever an experiment or project is being assemble, disassembled or modified.

In an emergency, all power in the lab can be switched off by pressing the button on the main breaker panel. It is to be used for emergencies ONLY.

**DON’TS**

NO food and drinks are allowed in this lab.

Do NOT transfer equipment to other workbench or other labs without permission.

Do NOT touch any equipment until you are told to do so.

Wearing a ring or watch can be hazardous in an electrical lab since such items make good electrodes for the human body.

NEVER touch any equipments or components with wet or damp hands.
## LIST OF EXPERIMENTS

<table>
<thead>
<tr>
<th>Sl. No.</th>
<th>EXPERIMENT</th>
<th>PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>SECOND ORDER ACTIVE LOW PASS FILTER</td>
<td>01</td>
</tr>
<tr>
<td>2.</td>
<td>ACTIVE BAND PASS AND BAND ELIMINATION FILTER</td>
<td>06</td>
</tr>
<tr>
<td>3.</td>
<td>SCHMITT-TRIGGER CIRCUIT</td>
<td>10</td>
</tr>
<tr>
<td>4.</td>
<td>R- 2R DAC</td>
<td>13</td>
</tr>
<tr>
<td>5.</td>
<td>ASTABLE AND MONOSTABLE MULTIVIBRATORS</td>
<td>16</td>
</tr>
<tr>
<td>6.</td>
<td>AMPLITITUDE MODULATION USING TRANSISTOR</td>
<td>21</td>
</tr>
<tr>
<td>7.</td>
<td>PULSE AMPLITUDE MODULATION &amp; DEMODULATION</td>
<td>25</td>
</tr>
<tr>
<td>8.</td>
<td>(A) PULSE WIDTH MODULATION &amp; DEMODULATION</td>
<td>27</td>
</tr>
<tr>
<td></td>
<td>(B) PULSE POSITION MODULATION</td>
<td>29</td>
</tr>
<tr>
<td>9.</td>
<td>CLASS C SINGLE TUNED AMPLIFIER</td>
<td>31</td>
</tr>
<tr>
<td>10.</td>
<td>I. F. AMPLIFIER / MIXER CIRCUIT</td>
<td>34</td>
</tr>
<tr>
<td>11.</td>
<td>FM MODULATION USING IC 8038</td>
<td>36</td>
</tr>
<tr>
<td>12.</td>
<td>PRECISION RECTIFIERS</td>
<td>39</td>
</tr>
<tr>
<td>13.</td>
<td>VIVA QUESTIONS</td>
<td>42</td>
</tr>
</tbody>
</table>
Experiment No.: 01

**SECOND ORDER ACTIVE LOW PASS FILTER & HIGH PASS FILTER**

**AIM:** To design a second order Butterworth Active Low Pass filter (LPF) and High Pass Filter (HPF) for a given cutoff frequency of 1 KHz and to draw its frequency response. Calculate there from the cut-off frequency and Roll-off factor.

**APPARATUS REQUIRED:** Op-amp (µA741), Resistors 5.6KΩ, 10KΩ 2 nos, 15KΩ 2 nos, Ceramic (Disk) Capacitors 0.01µF 2nos, Power supply, Signal generator, CRO Connecting wires etc.

**THEORY:**

Almost all communication systems use filters. A filter passes one band of frequencies while rejecting another. A filter can be either passive or active. Passive filter are built with resistors, capacitors and industries. They are generally used above 1MHz, have no power gain, and relatively difficult to tune. Active filters are useful below 1MHz, have power gain, and are relatively easy to tune.

Filters can separate desired signals from undesired signals, block interfering signal, enhance speech and video, and alter signals in other ways.

A low pass filter passes the frequencies between zero and the cutoff frequency called the pass band. The frequencies above the above the cut-off frequencies are called the stop band. The roll off region between the pass band and the stop band is called the transition. An ideal low pass filter has zero attenuation in the pass band, infinite attenuation in the stop band, and a vertical transition.

The ideal low pass filter has zero phase-shift for all frequencies in the pass band. Zero phase shift is important when the input signal is non sinusoidal. When a filter has zero phase shift, the shape of the non sinusoidal signal is preserved as it passes through the ideal filter.

A high pass filter has a stop band 0<f<fc and a pass band f>fc. fc is the low cutoff frequency and f is the operating frequency. The gain increases with an increase in frequency till fc. At f=fc the gain is down by 3dB. After fc the gain remains constant.
CIRCUIT DIAGRAM OF LPF:

![Circuit Diagram]

DESIGN OF LPF:

Gain: $K = (1 + \frac{R_f}{R_1}) = 1.5858$; Cut-off Frequency: $f_c = \frac{1}{2\pi CR}$

Given $f_c = 1 \, \text{kHz}$; Let $C_2 = C_3 = C = 0.01 \mu\text{f}$

Then $R_2 = R_3 = R = \frac{1}{2\pi f_c C} = \frac{1}{(2\pi \times 1\,\text{kHz} \times 0.01 \mu\text{f})} = 15.9 \, \Omega$ (Remember $1/2\pi \approx 0.159$)

Use $R_1 = R_2 = R = (12 + 3.9 \, \Omega)$ Series combination

The pass band voltage gain, $K = 1.5858 \, \frac{R_f}{R_1} = 0.5858$ or $R_f = 0.5858 \, R_1$

Let $R_1 = 10 \, \Omega$ then $R_f = 5.8 \, \Omega$

Use nearest available single values: $R_1 = 10 \, \Omega$, $R_f = 5.8 \, \Omega$

PROCEDURE:

- Make connections as shown in the circuit diagram.
  Apply the sinusoidal I/P signal of amplitude $10\,\text{V}$, (P-P) with a frequency less than so that the O/P is undistorted and measure the O/P voltage (P – P). Record this as $V_{\text{Max}}$.
- Keeping the input signal amplitude constant, increase the frequency until the O/P voltage reduces to $0.707V_{\text{Max}}$. The frequency at which this happens is your $f_c$, the filter cutoff frequency.
- Keeping the I/P signal amplitude constant, measure and record at least 6 readings below and 6 readings above the cutoff frequency and one reading at the cut off frequency. (Note: Whenever you change the frequency button /setting, you should check and adjust the I/P to its previous constant value)
- Plot the graph of Gain in dB V/S frequency on a semi-log graph paper.
- Identify the 3dB point on your graph and identify the cut off frequency.
- **Increase the frequency to** $f_1 = 10f_c$. Increase I/P signal amplitude also so as to be able to observe some O/P on the CRO. Measure the O/P when there is no distortion ($V_{o1}$)
- **Increase the frequency to** $f_2 = 20f_c$. Maintaining same I/P amplitude, measure the O/P voltage ($V_{o2}$)
- Compute **Roll–Off factor** of the filter using: dB/Octave
  
  \[ M_1 = 20 \log \left( \frac{V_{o1}}{V_i} \right) \text{ dB} \]
  
  \[ M_2 = 20 \log \left( \frac{V_{o2}}{V_i} \right) \text{ dB} \]
  
  Change in magnitude as the frequency is increased is
  \[ M_2 - M_1 = 20 \log \left( \frac{V_{o2}}{V_{o1}} \right) \text{ dB} \]
  
  This is the Roll – off factor. The answer is in dB/Octave because the two frequencies selected are in the ratio of 1:2. To convert the answer to dB/Decade, multiply the answer obtained in dB / Octave by (20/6) because 20n dB/Decade = 6n dB/Octave

- Alternatively, measure $v_{o1}$ at $f_c/10$ and $v_{o2}$ at $10f_c$. Roll-off factor is given by

  \[ \text{Roll-off factor} = 20 \log \left( \frac{v_{o2}}{v_{o1}} \right) \text{ dB/decade} \]

**EXPECTED FREQUENCY RESPONSE GRAPH:**

**TABULAR COLUMN:**

<table>
<thead>
<tr>
<th>SL. NO.</th>
<th>Frequency(Hz)</th>
<th>Output voltage</th>
<th>Gain(Vo/Vin)</th>
<th>Gain in db $20\log_{10}(Vo/Vin)$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**V_{in} = ....... V** (p-p), Constant

- 20 dB/Decade
CIRCUIT DIAGRAM OF HPF:

![Circuit Diagram of HPF](image)

**DESIGN OF HPF:**

Gain, \( K = (1+R_f/R_1) = 1.5858 \) and Cut-off Frequency, \( f_c = 1/(2\pi CR) \)

Given \( f_c = 1 \, \text{KHz}; \) Let \( C_2=C_3=0.01\mu f \)

Then \( R_2=R_3= R = 1/2\pi f_c C = 1/ (2\pi \times 1\,\text{K} \times 0.01\mu f) = 15.9 \, \text{K}\Omega \)

Use \( R_1=R_2= (12\,\text{K}\Omega + 3.9\,\text{K}\Omega) \) Series combination or use \( R = 15\,\text{K}\Omega \) (Nearest single component)

The pass band voltage gain, \( K = 1.5858 \, R_f/R_1 = 0.5858 \) or \( R_f = 0.5858 \, R_1 \)

Let \( R_1=10\,\text{K}\Omega \) then \( R_f=5.8 \, \text{K}\Omega \)

Use nearest available single values: \( R_1=10\,\text{K}\Omega, R_f=5.8 \, \text{K}\Omega \)

**PROCEDURE:**

- Make connections as shown in the circuit diagram.
- Apply sinusoidal I/P signal of amplitude 1V, (P-P) or any other convenient value with frequency > 10 (Around 10V) such that the O/P is undistorted and measure the O/P voltage (P–P). Record this as \( V_{\text{Max}} \).
- Keeping the input signal amplitude constant, decrease the frequency until the O/P voltage reduces to 0.707\( V_{\text{Max}} \). The frequency at which this happens is your \( f_c \), the filter cutoff frequency.
- Keeping the I/P signal amplitude constant at a convenient value, measure and record at least 6 readings below and 6 readings above the cutoff frequency and one reading at the cut off frequency. (Note: Whenever you change the frequency button /setting, you should check and adjust the I/P to its previous constant value)
• Plot the graph of Gain in dB V/S frequency on a semi log graph paper.
• Identify the 3dB point on your graph and identify the cut off frequency.
• Decrease the frequency to . Increase I/P signal amplitude also, so as to be able to observe some O/P on the CRO. Measure the O/P when there is no distortion. ($V_{o1}$)
• Decrease the frequency to. Maintaining same I/P amplitude, measure the O/P voltage ($V_{o2}$)
• Compute the Roll – Off factor of your filter using the formula:

$$\text{Roll-off factor} = 20 \log \left( \frac{V_{o2}}{V_{o1}} \right) \text{ dB/decade}$$

Alternatively, measure $v_{o1}$ at $f_c/10$ and $v_{o2}$ at $10f_c$. Roll-off factor is given by

$$\text{Roll-off factor} = 20 \log \left( \frac{v_{o2}}{v_{o1}} \right) \text{ dB/decade}$$

**EXPECTED FREQUENCY RESPONSE GRAPH:**

**TABULAR COLUMN:**

$V_{in} = \ldots\ldots \text{V (p-p), Constant}$

<table>
<thead>
<tr>
<th>SL. NO.</th>
<th>Frequency(Hz)</th>
<th>Output voltage</th>
<th>Gain($V_o/V_i$)</th>
<th>Gain in db $20\log_{10}(V_o/V_i)$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

RESULT: -
Experiment No.:02

ACTIVE BAND PASS AND BAND ELIMINATION FILTER

AIM: To design an active second order Band pass filter for the given specifications. Conduct an experiment to draw the frequency response graph and verify the design specifications

APPARATUS REQUIRED: Op-amp (µA741) 2 nos, Resistors15KΩ - 1 no, 30KΩ - 1 no 300Ω- 1 no, 10KΩ - 3 nos, Capacitor 0.01µf 2 nos, Power supply, Signal generator, CRO Connecting wires etc.

CIRCUIT DIAGRAM BAND PASS FILTER:

EXPECTED FREQUENCY RESPONSE GRAPH:
PROCEDURE:

- Make connections as shown in the circuit diagram.
- Apply the sinusoidal I/P signal of frequency, the filter mid – frequency and amplitude of 1V, (P-P) or any other convenient value (10 V) so that the O/P is undistorted and measure the O/P voltage (P – P). Record this as \( V_{\text{Max}} \).
- Keeping the input signal amplitude constant, decrease the frequency until the O/P voltage reduces to 0.707\( V_{\text{Max}} \). The frequency at which this happens is your \( f_{c1} \), the lower cutoff frequency of the filter.
- Repeat step 3 but now increase the frequency until the O/P voltage reduces to 0.707\( V_{\text{Max}} \). The frequency at which this happens is your \( f_{c2} \), the upper cutoff frequency of the filter.
- Keeping the I/P signal amplitude constant at a convenient value, measure and record at least 6 readings below and 6 readings above the filter mid – frequency, \( f_m \). (Note: Whenever you change the frequency button /setting, you should check and adjust the I/P to its previous constant value)
- Plot the graph of Gain in dB V/S frequency.
- Identify the 3dB points on your graph and identify the mid - frequency. Mark all the salient points on your graph.
TABULAR COLUMN:
$V_{in} = \ldots\ldots\ V_{(p-p)}$, Constant

<table>
<thead>
<tr>
<th>SL. NO.</th>
<th>Frequency(Hz)</th>
<th>Output voltage</th>
<th>Gain($V_o/V_i$)</th>
<th>Gain in db $20\log_{10}(V_o/V_i)$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

EXPECTED FREQUENCY RESPONSE GRAPH

CIRCUIT DIAGRAM OF BAND ELIMINATION FILTER:
PROCEDURE:

- Make connections as shown in the circuit diagram.
- Apply the sinusoidal I/P signal of frequency $f_m/10$, one tenth of the filter mid-frequency and amplitude 1V, (P-P) or any other convenient value (10V or less) so that the O/P is undistorted and measure the O/P voltage (P – P).
- Keeping the input signal amplitude constant, increase the frequency until the O/P voltage reduces to a minimum value, $V_{Min}$. The frequency at which this happens is your mid-frequency, $f_m$.
- Now increase the frequency until the O/P voltage approaches a constant value. Now you are in the pass band.
- You can identify the frequencies $f_L$ and $f_H$ as you did in LPF/HPF
- Keeping the I/P signal amplitude constant at a convenient value, measure and record at least 6 readings below and 6 readings above the filter mid-frequency, $f_m$. (Note: Whenever you change the frequency button/setting, you should check and adjust the I/P to its previous constant value)
- Plot the graph of Gain in dB V/S frequency.
- Identify the 3dB points on your graph and identify the mid-frequency. Mark all the salient points on your graph.

TABULAR COLUMN:

$V_{in} = \ldots V(p-p)$, Constant

<table>
<thead>
<tr>
<th>SL. NO.</th>
<th>Frequency(Hz)</th>
<th>Output voltage</th>
<th>Gain(Vo/Vin)</th>
<th>Gain in db $20\log_{10}(Vo/Vin)$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

RESULT:
Experiment No.: 03

**SCHMITT-TRIGGER CIRCUIT**

**Aim:** To design and test a Schmitt trigger for the given UTP and LTP Values

**COMPONENTS/APPARATUS REQUIRED:** - Op-amp ($\mu$A741), Resistors $15k\Omega$, $1k\Omega$, $1.2K$, Signal generator, Power supply, CRO, Connecting wires etc.

**THEORY:**

In electronics a **Schmitt trigger** is a comparator circuit with hysteresis implemented by applying positive feedback to the noninverting input of a comparator or differential amplifier. It is an active circuit which converts an analog input signal to a digital output signal. The circuit is named a "trigger" because the output retains its value until the input changes sufficiently to trigger a change. In the non-inverting configuration, when the input is higher than a chosen threshold, the output is high. When the input is below a different (lower) chosen threshold the output is low, and when the input is between the two levels the output retains its value. This dual threshold action is called hysteresis and implies that the Schmitt trigger possesses memory and can act as a bistable circuit (latch or flip-flop). There is a close relation between the two kinds of circuits: a Schmitt trigger can be converted into a latch and a latch can be converted into a Schmitt trigger. Schmitt trigger devices are typically used in signal conditioning applications to remove noise from signals used in digital circuits, particularly mechanical switch bounce. They are also used in closed loop negative feedback configurations to implement relaxation oscillators, used in function generators and switching power supplies.

**CIRCUIT DIAGRAM**

![Inverting Schmitt Trigger Diagram](image-url)
PROCEDURE:

1. Make the circuit connections of the Inverting Schmitt Trigger as shown in the diagram.

2. Apply a sinusoidal input signal and set the input signal amplitude to a convenient value, at least greater than the UTP value.

3. Adjust the I/P frequency to a convenient value (say from 100 Hz to 2 KHz).

4. Display the input signal and output signal wave forms on the CRO using **dual mode**.

5. Keeping the I/P in a calibrated (Fixed) position, move the O/P square wave form so as to **merge the falling edge with the I/P wave** and measure the corresponding amplitude of the input wave. This gives your UTP.

6. Now **merge the raising edge of the O/P wave with the I/P wave** and measure the amplitude. This gives your LTP.

7. Put the CRO into XY mode and see the RHOMBUS shape (If frequency is >1 KHz). Measure the UTP, LTP and the Hysteresis = (UTP – LTP) as indicated on the diagram.

**Specifications:** $V_{cc} = \pm 12V$  
$V_{sat} = \pm 12V$  
$UTP = 4V$  
$LTP = 2V$

**Design:**

$$UTP = \frac{V_{sat}R_2}{(R_1+R_2)} + \frac{V_rR_1}{(R_1+R_2)} \ldots (1)$$

$$LTP = -\frac{V_{sat}R_2}{(R_1+R_2)} + \frac{V_rR_1}{(R_1+R_2)} \ldots (2)$$

Adding equations $(1)$ and $(2)$, we get: 

$$UTP + LTP = 2V_r \frac{R_1}{(R_1+R_2)} = \frac{4+2}{6} = 6V \ldots (3)$$

Subtracting equations $(1)$ and $(2)$, we get: 

$$UTP - LTP = 2V_{sat} \frac{R_2}{(R_1+R_2)} = \frac{4-2}{2} = 2V \ldots (4)$$

Substituting the values of $V_{sat}$ in equation $(4)$ we have, $R_1 = 11R_2$

Select $R_2 = 3.3K$

Hence $R_1 = 9R_2 = 9*3.3K = 29.7K$, so choose $R_1 = 30K$

Substituting the values of $R_1$, $R_2$ and $V_{sat}$ we have,

$V_r = 3.36V$

$R_3 = R_1 \mid \mid R_2$

Select $R_3 = 2.9K$. 
WAVEFORMS:

RESULT:
Experiment No.:04

R- 2R DAC

AIM: To design and test R-2R DAC using operational amplifier

APPARATUS REQUIRED: Resistors: 10kΩ, 4.7kΩ, IC 741, CRO, probes, connecting wires

THEORY:

A 4-bit DAC using R-2R ladder network and an Op-amp voltage follower acting as a buffer stage is shown in Fig 1. D₀, D₁, D₂ and D₃ are the digital inputs. Each digital input may be low (0) or high (1).

\[ V_R(0) = 0 \text{ and } V_R(1) = V_R = 5V. \] (Reference voltage can be selected depending on maximum Analog o/p voltage required. If the digital inputs are obtained from a Digital IC trainer, then \( V_R = +5 \text{ V = constant / DC reference voltage}. \)

The analog output voltage \( V_O \) for a 4-bit DAC shown in Fig can be written as below:

\[
Output \ V_O = (2^3 D_3 + 2^2 D_2 + 2^1 D_1 + 2^0 D_0) \frac{V_R}{24}
\]

Where \( V = \frac{V_R}{2^4 (\frac{2R}{3R})}; \) This gives \( V_O = (8D_3 + 4D_2 + 2D_1 + D_0) \frac{V_R}{24} \)

CIRCUIT DIAGRAM:
DESIGN:

Resolution = \( \frac{V_{\text{ref}}}{2^n - 1} \) where \( n \) is number of digital inputs to the DAC.

In this circuit number of digital inputs \( (n=4) \)

If \( V_{\text{ref}} \) is 5V then resolution = 0.33V

Choose \( R = 4.6k\Omega \) Hence \( 2R = 10k\Omega \)

Accuracy is a comparison of actual output voltage with expected output

\[ \text{Accuracy} = \frac{V_{\text{ofs}}}{(2^n - 1) \times 2} \]

Where \( V_{\text{ofs}} = D \) (decimal value of digital input) \( \times \) resolution (The output full scale voltage)

PROCEDURE:

- Rig up the circuit as shown in the circuit diagram and give the reference voltage
- Provide the digital input through \( V_{\text{ref}} \) to D0 to D3.
- Observe the analog output at \( V_{\text{out}} \)
- Verify the observed output with the theoretical values calculated.

EXPECTED WAVEFORM:
TABULAR COLUMN:

<table>
<thead>
<tr>
<th>DIGITAL INPUTS</th>
<th>THEORITICAL VALUES</th>
<th>PRACTICAL VALUES</th>
</tr>
</thead>
<tbody>
<tr>
<td>D3 D2 D1 D0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>0 0 0 0</td>
<td>0</td>
<td>-0.625</td>
</tr>
<tr>
<td>0 0 0 1</td>
<td>-0.625</td>
<td>-1.25</td>
</tr>
<tr>
<td>0 0 1 1</td>
<td>-1.25</td>
<td>-1.875</td>
</tr>
<tr>
<td>0 1 0 0</td>
<td>-2.5</td>
<td>-3.125</td>
</tr>
<tr>
<td>0 1 0 1</td>
<td>-3.125</td>
<td>-3.75</td>
</tr>
<tr>
<td>0 1 1 0</td>
<td>-3.75</td>
<td>-4.375</td>
</tr>
<tr>
<td>0 1 1 1</td>
<td>-4.375</td>
<td>-5</td>
</tr>
<tr>
<td>1 0 0 0</td>
<td>-5</td>
<td>-5.625</td>
</tr>
<tr>
<td>1 0 0 1</td>
<td>-5.625</td>
<td>-6.25</td>
</tr>
<tr>
<td>1 0 1 1</td>
<td>-6.25</td>
<td>-6.875</td>
</tr>
<tr>
<td>1 1 0 0</td>
<td>-7.5</td>
<td>-8.125</td>
</tr>
<tr>
<td>1 1 0 1</td>
<td>-8.125</td>
<td>-8.75</td>
</tr>
<tr>
<td>1 1 1 0</td>
<td>-8.75</td>
<td>-9.375</td>
</tr>
</tbody>
</table>

RESULT:
Experiment No.: 05

ASTABLE AND MONOSTABLE MULTIVIBRATORS

AIM: To design and test unsymmetrical and symmetrical astable multivibrator and a monostable multivibrator for a given duty cycle and frequency using IC555.

APPARATUS REQUIRED: Resistors: 2.886kΩ, 1.5kΩ, 10kΩ, 1kΩ
Capacitors: 0.001µf, 0.01µf, 0.1µf
IC555, BY127, connecting wires, probes, CRO

THEORY:
An Astable Multivibrator is an oscillator circuit that continuously produces rectangular wave without the aid of external triggering. So Astable Multivibrator is also known as Free Running Multivibrator. I have already posted about Astable Multivibrator using Transistors. Astable Multivibrator using 555 Timer is very simple, easy to design, very stable and low cost. It can be used for timing from microseconds to hours. Due to these reasons 555 has a large number of applications and it is a popular IC among electronics hobbyists.

CIRCUIT DIAGRAM OFASYMMETRICAL ASTABLE MULTIVIBRATOR:
EXPECTED OUTPUT WAVEFORM:

DESIGN:
- A square wave of given Duty cycle and frequency
  
  For unsymmetrical square wave let Duty cycle > 50%
  
  \[ T_{on} = 0.693(R_A + R_B)C \] for charging time
  
  \[ T_{off} = 0.693R_B C \] for discharging time
  
  Duty cycle = \( \frac{T_{on}}{T_{on} + T_{off}} = \frac{T_{on}}{T} \)
  
  \[ T_{off} = T - T_{on} \]
  
  Assume \( C = 0.1 \mu F \) \( T_{off} = 0.693R_B C \) \( T_{on} = 0.693(R_A + R_B)C \)
  
  Similarly for \( D < 50\% \) \( T_{on} < T_{off} \)
  
  \( T_{on} = 0.693R_A \) \( T_{off} = 0.693R_B \)
  
  Assume \( C \) for a given \( f \) and determine \( R_A \) and \( R_B \)

Observe the waveform for the same.
CIRCUIT DIAGRAM OF SYMMETRICAL ASTABLE MULTIVIBRATOR:

Design:

(i). A square wave of given Duty cycle and frequency

For unsymmetrical square wave  Let Duty cycle >50%

\[ T_{\text{on}} = 0.693(R_{\text{c}} + R_{\text{g}})C \]  
for charging time

\[ T_{\text{off}} = 0.693 R_{\text{g}} C \]  
for discharging time

(60%) Duty cycle  \[ \frac{T_{\text{on}}}{T_{\text{on}} + T_{\text{off}}} = \frac{T_{\text{off}}}{T} \]

\[ T_{\text{off}} = T - T_{\text{on}} \]

Note: To generate output waveform with duty cycle \( D = 50\% \), connect a diode (BY127) across \( R_{\text{g}} \) as shown. In that case, \( T_{\text{on}} = T_{\text{off}} \).

EXPECTED OUTPUT WAVEFORM:
MONOSTABLE MULTIVIBRATOR

THOERY:

A monostable multivibrator (MMV) often called a one-shot multivibrator, is a pulse generator circuit in which the duration of the pulse is determined by the R-C network, connected externally to the 555 timer. In such a vibrator, one state of output is stable while the other is quasi-stable (unstable). For auto-triggering of output from quasi-stable state to stable state energy is stored by an externally connected capacitor C to a reference level. The time taken in storage determines the pulse width. The transition of output from stable state to quasi-stable state is accomplished by external triggering.

CIRCUIT DIAGRAM OF MONOSTABLE MULTIVIBRATOR:

[Diagram of the 555 timer as a monostable multivibrator]

DESIGN:

Given Pulse Width \( W = 0.5 \text{ms} \)

We know that \( W = 1.1 \times R_A \times C \)

Assume \( R_A = 10K \) and calculate \( C \)

For differentiator circuit,

Choose \( R_t C_t << W \)

i.e., \( R_t C_t << W/10 \) (assume \( R_t = 1K \) & find \( C_t \))

Duty cycle \( D = W/T \) where \( T = 1/f \) (f = adjust trigger pulse input frequency to 1KHz)
EXPECTED OUTPUT WAVEFORM:

RESULT:
Experiment No.:06
AMPLITUDE MODULATION USING TRANSISTOR
(COLLECTOR MODULATION & DEMODULATION)

AIM: To design & conduct an experiment on collector modulation to generate a AM wave & to plot the variation of modulating signal amplitude Vs modulation index

THEORY:

AM modulation is defined as the varying of amplitude of the carrier wave in accordance with the instantaneous value of the modulation wave. The collector amplitude modulation is the most efficient high level modulation. Transistor is operated in class ‘C’ mode in which it is biased well below cut off. The carrier input to base must be sufficient to drive transistor into conduction over part of RF cycle during which collector current flows in the form of pulses.

Modulating signal Vm (t) is applied in series with Vcc through low frequency transformer. Modulating o/p is obtained through an IFT (IFT is tuned for carrier frequency). The RF voltage on the collector must swing the transistor into its saturation region over part of each RF cycle so that the modulation voltage have controlling effect on peak value of current pulse and hence amplitude modulated signal is obtained.

An attractive alternative to standard AM wave oscillograph is the trapezoidal wave pattern. The trapezoidal wave pattern allows observation of any complex modulation signal & provides a gauge for measuring % modulation. The taper of trapezoid is a measure of linearity of modulated wave

ENVELOPE DETECTOR:

The circuit produces output voltage which is proportional to envelope of the modulated signal & it is approximately same as that of modulating signal. Detector circuit is a half wave rectifier with on low pass RC filter. For distortion less operation, value of resistor of capacitor of a filter is designed such that 1/fm ≥ RdCd ≥ 1/fc.

When the diode is forward biased, the capacitor charges to peak value of input signal. When diode is reverse biased, the capacitor holds the positive charge previously received so that it discharge through R. Discharge constant RC must be large so that capacitor discharges slowly through R between +ve peaks of carrier. Detector circuits also provides an automatic voltage or automatic gain control & is used is IF & RF stages if receiver circuit positive or negative Depending on the polarity of the diode.
CIRCUIT DIAGRAM:

Modulation:

Demodulation:

DESIGN:

Modulation
Given: $F_{IF} = 455 \text{ KHz}$,
$T = 2.19\mu\text{s}$
$RC>>T$, $RC = 100T$
Choose $C=0.1\mu\text{F}$
$R=2.19\text{K}\Omega$, choose $R=2.2\text{ K}\Omega$

Demodulation:
Design:
Given: $F_c = 455 \text{ KHz}$, $F_m = 1 \text{ KHz}$
$1/F_m>RC>1/F_c$
$1\text{ms}>RC>2.2\mu\text{s}$
Let $RC = 100/F_c$
Choose $C= 0.1\mu\text{F}$
$R = 2.19\text{K}\Omega$, Choose $R = 2.2\text{ K}\Omega$

TABULAR COLUMN:

<table>
<thead>
<tr>
<th>Message Amp (V)</th>
<th>Emax (V)</th>
<th>Emin (V)</th>
<th>$m=\frac{\text{Emax}-\text{Emin}}{\text{Emax}+\text{Emin}}$</th>
<th>L1 (cm)</th>
<th>L2 (cm)</th>
<th>$m=L1-L2/L1+L2$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
TRAPEZOIDAL PATTERN:

COLLECTOR MODULATION & DEMODULATION WAVEFORMS:
PROCEDURE:

- Rig up the circuit as per the circuit diagram.
- Adjust the carrier frequency to about $f_c=455\text{Khz}$ & fine tune the signal to get maximum output.
- Keeping the carrier amplitude constant vary the modulating signal voltage in appropriate steps & measure the modulation index using the formula $m = \frac{E_{\text{max}} - E_{\text{min}}}{E_{\text{max}} + E_{\text{min}}}$
- Obtain the trapezoidal pattern & calculate the modulation index using formula $m = \frac{L_1 - L_2}{L_1 + L_2}$.
- Tabulate the results & draw the graph of modulation index Vs modulating voltage amplitude.
- Rig up the demodulation circuit & observe the demodulated O/P.

RESULT:
Experiment No.:07

**PULSE AMPLITUDE MODULATION & DEMODULATION**

**AIM:** To design & conduct an experiment to generate PAM signal to verify the sampling theorem & also to demodulate the PAM signal & also to plot the relevant waveforms.

**THEORY:**

Pulse-amplitude modulation (PAM) is a form of signal modulation where the message information is encoded in the amplitude of a series of signal pulses. It is an analog pulse modulation scheme in which the amplitudes of a train of carrier pulses are varied according to the sample value of the message signal. In PAM systems, the amplitude of each pulse is directly proportional to the instantaneous modulating-signal amplitude at the time the pulse occurs. Demodulation is performed by detecting the amplitude level of the carrier at every symbol period.

**CIRCUIT DIAGRAM:**

**DESIGN:**

**Modulation:**

Given: Ic = 1mA, hfe = 100, Vce (sat) = 0.3V, Vbe (sat) = 0.7V, Fm = 100 Hz

Vm (t) = IcRc + Vce(sat)

Let Vm (t) = 2.5V+3V dc shift = 5.5V

Then Rc = 5.2KΩ

Vc (t) = IbRb + Vbe (sat)

Vc(t)=2Vp-p

Let Ib = Ic/hfe = 10µA

Then Rb = 30KΩ
Demodulation:
Filter:
Cut off frequency $F_o = 500\text{Hz}$
$F_o = \frac{1}{2\pi RC}$
Let $C = 0.1\mu\text{F}$
Then $R = 3.3\Omega$

EXPECTED WAVEFORM:

PAM Modulation & Demodulation Waveforms:

PROCEDURE:

- Rig up the circuit as per the circuit diagram.
- Initially Apply square wave carrier of 2Vp-p with frequency $f_c=5\text{ KHz}$.
- Apply sine wave modulating signal of 5Vp-p amplitude & 3V dc shift with frequency $F_m=100\text{ Hz}$.
- Observe the PAM output.
- Observe the demodulated signal at the output of low pass filter.
- Plot the various waveforms.
- Repeat the steps from 2 to 6 for $f_c<2f_m$, $f_c=2f_m$ & $f_c>2f_m$.

RESULT:
Experiment No: 08(A)

**PULSE WIDTH MODULATION & DEMODULATION**

**AIM:** To design a circuit for pulse width modulation & also to demodulate the same.

**DESIGN:**

**Modulation:**
- F=10 KHz, T=0.1ms, duty cycle=0.7
- \( \delta = \frac{Ton}{T} \)
- Ton=0.07ms
- Toff=0.03ms
- Let Ct=0.1\( \mu \)F
- Ton= 0.693RaCt
- Ra=1.0K\( \Omega \)
- Toff= 0.693RbCt
- Rb=432 \( \Omega \)

Clamper circuit design:
- RC>>T
- RC=10T
- F=1 KHz, T=1ms
- Choose C= 1\( \mu \)F
- R= \( \frac{10T}{C} \), R=10K\( \Omega \)

**Demodulation design:** Cut off frequency Fo =1Khz
- Fo = \( \frac{1}{R_1C_1} \)
- Let C_1= 0.1\( \mu \)F
- R_1=1.591K\( \Omega \)

**CIRCUIT DIAGRAM:**
**MODULATION:**

![Circuit Diagram](image-url)
DEMODULATION:

PROCEDURE:

• Rig up the circuit as per the circuit diagram.
• Modulating signal is applied through pin no: 5 from the clamper.
• Observe the PWM output at pin no: 3.
• To find the Critical amplitude: It is that amplitude for which the pulses disappears for the first time either because its width has become zero (-ve half) or because the width of pulse is so large that it combines with the neighboring pulses (+ve half).
• Dynamic Range: Is the difference between the input voltage at which the PWM starts & the voltage where it vanishes.
• Rig up the demodulation circuit.
• By applying the PWM waveform as input & observe the original message signal at the output.

PWM MODULATION & DEMODULATION WAVEFORM:

RESULT: Dynamic range:

Critical amplitude:
Experiment No: 08(B)

PULSE POSITION MODULATION

AIM: To design & conduct an experiment pulse position modulation & Also to demodulate the same.

THEORY:

In pulse position modulation the position of pulses varies in accordance with amplitude of sampled waveform.

The spectrum of a PPM signal contains a fixed dc component and a set of carrier and phase modulated sidebands at each harmonic of the carrier frequency. The spectrum does not contain any baseband components, so the modulating signal cannot be recovered using a low pass filter. A phase detector is needed for demodulation, but again generation of this type is relatively easy.

PPM can be derived from PWM through the process of differentiation. PWM signal is applied to pin 2 of IC 555 timer through a diode RC combination. Thus input to pin 2 is negative trigger pulses which correspond to trailing edges of PWM waveform 555 timer is working in a monostable mode and pulse width is constant the negative trigger pulses decides the starting time of output pulses and thus output at pin 3 is desired pulses position modulation.

PPM totally rejects noise and is replacing the use of PAM and PWM. PPM is used in speed control of dc motors.

CIRCUIT DIAGRAM

MODULATION:

DESIGN:
Specifications: Pulse width=200µs  fc=1 KHz
Monostable Multivibrator
Pulse width = 1.1RCt
Choose $C_t=0.01\mu F$
$R=18K\Omega$

**Differentiator:**
$Rs*Cs = \frac{0.01}{fc} = 0.01ms$
Choose $Cs=0.001\mu F$
$Rs=10K\Omega$

**DEMODULATION:**

**PPM MODULATION WAVEFORMS:**

1. Audio signal
2. PWM signal
3. PPM signal
PROCEDURE:

- Connections are made as per the circuit diagram.
- Check the working of 555 timer as a monostable multivibrator by giving an unmodulated PWM signal. Verify the pulse width of output signal for the designed value.
- Critical amplitude of the modulating signal is that value of m(t) at which the pulse in PPM just disappears.
- Rig up the demodulation circuit (kit).
- By applying PPM as input observe the demodulated signal at the output.

RESULT: The critical amplitude of PPM is ______ volts.
Experiment No: 09

CLASS C SINGLE TUNED AMPLIFIER

AIM: To design and test a Class C single Tuned amplifier and also plot frequency response curve.

APPARATUS REQUIRED: Transistor SL 100, Resistors, Capacitors, DCB, DRB, Signal Generator, CRO, etc.

CIRCUIT DIAGRAM:

EXPECTED WAVEFORM:
PROCEDURE:

- Check out the components given and rig up the circuit.
- Adjust Input signal frequency using formula $f_0 = \frac{1}{2\pi R C}$.
- Vary the input Voltage for undistorted sine wave.
- For different load resistance note down output voltage.
- Calculate the efficiency. Plot the graph.

TABULAR COLUMN:

<table>
<thead>
<tr>
<th>$R_L$ in $\text{K}\Omega$</th>
<th>$V_o$ Volts</th>
<th>$I_d$ mA</th>
<th>$P_{ac} = \frac{V_o^2}{8R_L}$</th>
<th>$P_{dc} = \text{VCC*Idc}$</th>
<th>$\eta %$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

RESULT:
Experiment No: 10

I. F. AMPLIFIER / MIXER CIRCUIT

AIM: To design an IF amplifier/ Mixer Circuit and study its operation

APPARATUS REQUIRED: Transistor BF194/195, Resistor 390 KΩ, 150 K, 680 Ω, 10 K, 0.01µf 2nos, IFT, RF Signal generators 2nos, CRO.

CIRCUIT DIAGRAM:
**DESIGN:** Given: RF Transistor BF 194/195 for which the working $h_{FE} = 150$. For designing of a mixer we need to choose non – linear region of operation (Why?). (See the transfer characteristics shown). Choose near cutoff operation by selecting the base current to be $< 10$ A.

Thus Let $V_{cc} = 6V$; $I_B = 5A$. $I_C = 0.75mA$ $I_E$.

Let $R_E=680$ $V_E=I_ER_E= 0.51V$ $V_B= V_{BE} + V_E = 1.11V$ ($V_{BE} = 0.6V$ Si Transistor)

Let $R_2=150 K$ $I_2 = $ Current through $R_2=1.11V/150K = 7.4 A$

$I_1= $ Current through $R_1 = I_B + I_2 = 12.4 A$; $V_1 = $ Voltage across $R_1 = V_{cc} – V_B = 4.89 V$;

$R_1 = V_1/I_1 = 4.89V/12.4A = 394.35 K$. Use $R_1 = 390 K$

Use $C_c = 0.01 F$ coupling capacitors for both RF I/P and Local OSC I/P.

**PROCEDURE:**

- Make the connections as shown in circuit diagram.
- Set $V_{cc} = 6V$.
- Before switching on the Local oscillator signal generator, Vary the frequency of the RF I/P generator in the vicinity of 455 KHz and check for the resonant frequency of the IFT.

Switch on the Local oscillator and tabulate the readings as indicated in the TABLE below.

Find the Output Impedance of the IF amplifier using your 3rd semester method.

For each I/P frequency setting compute $I_o = V_o/R_0$

Compute the conversion trans-conductance of the IF amplifier, $G_c = V_{in}/I_o$.

<table>
<thead>
<tr>
<th>FRF</th>
<th>700 KHz</th>
<th>1.2 MHz</th>
<th>1.5 MHz</th>
<th>2.0 MHz</th>
<th>2.5 MHz</th>
</tr>
</thead>
<tbody>
<tr>
<td>FLO=FRF – 455 (UP Conversion)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FLO=FRF + 455 (Down Conversion)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Vin</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>V0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Gc</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**RESULT:**
Experiment No:11

**FM MODULATION USING IC 8038**

**AIM:** To design & conduct an experiment to generate FM wave using IC 8038 & to find the parameters the modulation index $\beta$, the bandwidth of operation $B_T$ & maximum frequency deviation $\delta$.

**THEORY:**

In angle modulation, the information signal may be used to vary the carrier frequency, giving rise to frequency modulation, or it may be used to vary the angle of phase lead or lag giving rise to phase modulation.

Compared to amplitude modulation, frequency modulation has certain advantages. Mainly, the signal to noise ratio can be increased without increasing transmitted power (but at the expense of an increase in frequency bandwidth required); certain forms of interference at the receiver are more easily suppressed; and the modulation process can take place at a low power stage in the transmitter, thus avoiding the need for large amounts of modulating power.

IC 8038, a waveform generator, is used in PM generation. IC 8038 is a 14 pin IC where pins 10, 11, 12 are used for sine wave adjust, Pin 2 gives sine wave output, the amplitude of this wave is 0.22Vcc where Vcc varies between ±5V to ±15V. This IC also generates square wave and triangular wave. Pin 4 & 5 are used for duty cycle/frequency adjustment. An external capacitor connected to pin 10 along with resistors connected to pin 4 & 5 determines output frequency. Pin 8 is used for FM sweep input.

**CIRCUIT DIAGRAM:**

![Circuit Diagram](image-url)
DESIGN:

Given: \( F_c = 170 \text{ KHz} \)
We have \( F_c = 0.159/R_t \cdot C_t \)
Choose \( C_t = 0.001\mu F \)
Therefore \( R_t = 0.159/F_c \cdot C_t \)
\( R_t = 935\Omega \)
Choose \( R_t = 820\Omega \)

TABULAR COLUMN

<table>
<thead>
<tr>
<th>Sl.no.</th>
<th>Fc (Hz)</th>
<th>Fm (Hz)</th>
<th>Vm (V)</th>
<th>Fc_{\text{max}} (Hz)</th>
<th>Fc_{\text{min}} (Hz)</th>
<th>\delta_1 (Hz)</th>
<th>\delta_2 (Hz)</th>
<th>\beta = \delta/Fm</th>
<th>BT = 2(\delta + F_m) (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

\( \delta_1 = F_{c_{\text{max}}} - F_c, \quad \delta_2 = F_c - F_{c_{\text{min}}} \)

\( \delta = \text{max of } \delta_1 \text{ or } \delta_2 \)

PROCEDURE:

- Rig up the circuit as per the circuit diagram.
- Switch OFF the message signal & note down the carrier frequency \( F_c \).
- Then Switch ON the message signal & note down the message frequency \( F_m \).
- Adjust the amplitude of the message signal & observe the FM waveform.
- Find \( F_{c_{\text{max}}} \) & \( F_{c_{\text{min}}} \) from the FM waveform.
- Calculate the maximum frequency deviation \( \delta \), modulation index \( \beta \) & bandwidth of operation \( B_T \).
RESULT:

Fc (prac) = 

Fc(theo) = 170Khz.

F_max =

F_min =

β =

δ =

B_T =
Experiment no: 12

**PRECISION RECTIFIERS**

**AIM:** To design and test circuit using op-amp to obtain the waveform given in the figure for a sinusoidal input.

**APPARATUS REQUIRED:**
IC 741
Resistors
Signal generator, CRO

**THEORY:**

The purpose of the rectifier section is to convert the incoming ac from a transformer or other ac power source to some form of pulsating dc. That is, it takes current that flows alternately in both directions as shown in the first figure to the right, and modifies it so that the output current flows only in one direction. The circuit required to do this may be nothing more than a single diode, or it may be considerably more complex. However, all rectifier circuits may be classified into one of two categories, as follows:

Half Wave Rectifier:

An easy way to convert ac to pulsating dc is to simply allow half of the ac cycle to pass, while blocking current to prevent it from flowing during the other half cycle. Such circuits are known as half-wave rectifiers because they only work on half of the incoming ac wave.

Full Wave Rectifier:

The more common approach is to manipulate the incoming ac wave so that both halves are used to cause output current to flow in the same direction. Because these circuits operate on the entire incoming ac wave, they are known as full-wave rectifiers. A half wave rectifier using OP amp is also known as a Precision rectifier or super diode, is a configuration obtained with an operational amplifier in order to have a circuit behaving like an ideal diode and rectifier.
CIRCUIT DIAGRAM:

WAVEFORMS & TRANSFER CHARACTERISTIC:
Design:

When $V_{in} = -ve$, $V_o' = -ve$, $D_1$ is reverse biased, $V_o'' = 0$,
$V_o = \left( -\frac{R_f V_o''}{R'} \right) + \left( -\frac{R_f}{V_{in}} \right)$

$V_o = -R_f V_{in} / R''$

As gain $(R_f / R'') = 10$, if $R_f = 22k$, $R'' = 2.2k$

When, $V_{in} = +ve$, $V_o' = -ve$, $V_o'' = -Vin$

$V_o = -(R_f V_o'' / R') + (R_f V_{in} / R'')$

$V_o = -(R_f / V_{in} / R') + (R_f V_{in} / R'')$

$V_o = (R_f V_{in} / R') - (R_f V_{in} / R'')$

$V_o = V_{in}[(R_f / R') (R_f / R'')]$

Gain $[(R_f / R'), (R_f / R'')] = 10$

$(R_f / R') = 10 = 10$

$R_f / R' = 10$

As $R_f = 22k$, $R' = 1.1k$

For figure c, choose $R_f / R'' =$ gain $= 6$. Therefore $R_f / R'' = 10$

$R_f / R'' = 16$ As $R_f = 22k$ Then $R'' = 1.375K$

**PROCEDURE:**

- Check all the components
- Make the connections as shown in the circuit diagram
- Set the input voltage amplitude and observe the output signal waveform on the CRO
- Plot the waveform on a graph sheet

**RESULT:**
**VIVA QUESTIONS**

Q1. What is amplitude modulation?
Ans: The process of amplitude modulation consists of varying the peak amplitude of a sinusoidal carrier wave in proportion to the instantaneous amplitude of the modulation signal.

Q2. What is modulation?
Ans: Modulation may be defined as the process by which some parameters of a high frequency signal termed as carrier, is varied in accordance with the signal to be transmitted.

Q3. What are the different types of analog modulation?
Ans: 1) Amplitude modulation 2) angle modulation.

Q4. What is the need for modulation?
Ans: Consider, for example, picture signal of a T.V camera. It has frequency spectra of DC to 5.5MHz. Such a wide band of frequency can’t be propagated through ionosphere. However, if this signal is modulated with a carrier in VHF and UHF range, the percentage bandwidth becomes very small and the signal becomes suitable for transmission through atmosphere.

Q5. What are the objectives met by modulation?
Ans: Length of antenna is shortened, signal loss is reduced, ease of radiation, adjustment of bandwidth, shifting signal frequency of the assigned value.

Q6. What are the advantage of PAM and PWM?
Ans: PWM system gives a greater signal to noise ratio as compared to PAM but requires a larger bandwidth to achieve this.

Q7. What is Pulse position modulation?
Ans: Pulse position modulation (PPM) is the process in which the position of a standard pulse is varied as a function of the amplitude of the sampled signal.

Q8. What is the advantage of PPM over PWM and PAM?
Ans: The phase deviations are usually small. The noise produces a smaller disturbing effect on the time position of the modulating pulse train and as a result, PPM waves have a better performance with respect to signal to noise ratio in comparison to PAM and PWM systems.

Q9. What are the applications of pulse position modulation?
Ans: It is primarily useful for optical communication systems, where there tends to be little or no multipath interference. Narrowband RF (Radio frequency) channels with low power and long wavelength (i.e., low frequency) are affected primarily by flat fading, and PPM is better suited.

Q10. What is the purpose of using differential pulse position modulation?
Ans: It is possible to limit the propagation of errors to adjacent symbols, so that an error in measuring the differential delay of one pulse will affect only two symbols, instead of effecting all successive measurements.

Q11. What is the advantage of PPM?
Ans: One of the principle advantages of pulse position modulation is that it is an M-ary modulation technique that can be implemented non-coherently, such that the receiver does not need to use a phase-locked loop (PLL) to track the phase of the carrier. This makes it a suitable candidate for optical communications systems, where coherent phase modulation and detecting are difficult and extremely expensive. The only other common M-ary non-coherent modulation technique is M-ary frequency shift keying, which is the frequency domain dual to PPM. The other advantages of pulse position modulation are:
• The amplitude is held constant thus less noise interference.
• Signal and noise separation is very easy.
• Due to constant pulse widths and amplitudes, transmission power for each pulse is same.

Q13. Explain the principle of PPM?
Ans: The amplitude and the width of the pulse is kept constant in this system, while the position of each pulse, in relation to the position of a recurrent reference pulse is varied by each instantaneous sampled value of the modulating wave. This means that the transmitter must send synchronizing pulses to operate timing circuits in the receiver. The PPM has the advantage of requiring constant transmitter power output, but the disadvantage of depending on transmitter-receiver synchronization.
Q14. What is the purpose of PPM?
Ans: PPM may be used to transmit analog information, such as continuous speech or data.

Q15. What are the analog analogies of PAM, PPM & PWM?
Ans: PAM is similar to AM; PPM and PWM is similar to angle modulation.

Q16. What is Frequency modulation (FM)?
Ans: Frequency modulation is the process of varying the frequency of a carrier wave in proportion to the instantaneous amplitude of the modulating signal without any variation in the amplitude of the carrier wave.

Q17. What is PWM or Pulse length modulation or pulse duration modulation?
Ans: In PWM, the pulse amplitude is kept constant but the leading edge, trailing edge or both may be varied as a function of the amplitude of the sampled signal and care must be taken to ensure that the pulse don’t overlap in a TDM system.

Q18. What are the disadvantages of PWM?
Ans: PWM, in general, requires a greater average power than PAM systems. Also, the PWM system requires a greater bandwidth than PAM.
APPENDIX

Pin diagram of the Operational Amplifier - IC 741

555 Internal Diagram

555 Timer Block Diagram
• **Pin 1. – Ground**, The ground pin connects the 555 timer to the negative (0v) supply rail.

• **Pin 2. – Trigger**, The negative input to comparator No 1. A negative pulse on this pin “sets” the internal Flip-flop when the voltage drops below 1/3Vcc causing the output to switch from a “LOW” to a “HIGH” state.

• **Pin 3. – Output**, The output pin can drive any TTL circuit and is capable of sourcing or sinking up to 200mA of current at an output voltage equal to approximately Vcc – 1.5V so small speakers, LEDs or motors can be connected directly to the output.

• **Pin 4. – Reset**, This pin is used to “reset” the internal Flip-flop controlling the state of the output, pin 3. This is an active-low input and is generally connected to a logic “1” level when not used to prevent any unwanted resetting of the output.

• **Pin 5. – Control Voltage**, This pin controls the timing of the 555 by overriding the 2/3Vcc level of the voltage divider network. By applying a voltage to this pin the width of the output signal can be varied independently of the RC timing network. When not used it is connected to ground via a 10nF capacitor to eliminate any noise.

• **Pin 6. – Threshold**, The positive input to comparator No 2. This pin is used to reset the Flip-flop when the voltage applied to it exceeds 2/3Vcc causing the output to switch from “HIGH” to “LOW” state. This pin connects directly to the RC timing circuit.

• **Pin 7. – Discharge**, The discharge pin is connected directly to the Collector of an internal NPN transistor which is used to “discharge” the timing capacitor to ground when the output at pin 3 switches “LOW”.

• **Pin 8. – Supply +Vcc**, This is the power supply pin and for general purpose TTL 555 timers is between 4.5V and 15V.