DEPARTMENT
OF
ELECTRONICS & COMMUNICATION ENGG

ANALOG COMMUNICATIONS +
LINEAR INTEGRATED CIRCUITS LAB MANUAL
AC + LIC LAB

IV Semester (17EC4DLACL)
Autonomous Course
2018-19

Name of the Student	:	
Semester /Section	:	
USN	:	
Batch	:	

Dayananda Sagar College of Engineering
Shavige Malleshwara Hills, Kumaraswamy Layout,
Banashankari, Bangalore-560078, Karnataka
Tel : +91 80 26662226 26661104 Extn : 2731 Fax : +90 80 2666 0789
Web - http://www.dayanandasagar.edu Email : hod-ece@dayanandasagar.edu
(An Autonomous Institute Affiliated to VTU, Approved by AICTE & ISO 9001:2008 Certified)
(Accredited by NBA, National Assessment & Accreditation Council (NAAC) with 'A' grade)

DEPARTMENT
OF
ELECTRONICS & COMMUNICATION ENGINEERING

AC + LIC LABORATORY MANUAL

IV Semester (17EC4DLACL)

Autonomous Course

2018-19

Staff incharges : Sapna P.J.
Navya Holla K., Trupti Tagare, Priyanka N.
Lab Instructor : M.S. Puttaraju
HOD : Dr. T.C.Manjunath

Name of the Student	:	
Semester /Section	:	
USN	:	
Batch	:	

Dayananda Sagar College of Engineering
Shavige Malleshwara Hills, Kumaraswamy Layout,
Banashankari, Bangalore-560078, Karnataka
Tel : +91 80 26662226 26661104 Extn : 2731 Fax : +90 80 2666 0789
Web - http://www.dayanandasagar.edu Email : hod-ece@dayanandasagar.edu
(An Autonomous Institute Affiliated to VTU, Approved by AICTE & ISO 9001:2008 Certified)
(Accredited by NBA, National Assessment & Accreditation Council (NAAC) with 'A' grade)

Dayananda Sagar College of Engineering

Dept. of E & C Engg

Name of the Laboratory	:	Analog Communication and Linear Integrated Circuits Lab AC + LIC Lab
17EC4DLACL **Semester/Year**	:	IV/2017 & 2018 (Autonomous)
No. of Students/Batch	:	20
No. of equipment's	:	20
Major Equipment's	:	Digital Storage Oscilloscope Function generator Power Supply Fixed Power Supply Multi Meter, Bread Boards Resistance Box Capacitance Box
Area in square meters	:	72.9 Sq mts
Location	:	Level – 3 (Room No. 17215)
Total Cost of Lab	:	Rs. 5,10,000/-

Lab Incharge/s: Mrs. Sapna P.J. / Trupti Tagare

Mrs. Navya Holla K / Priyanka N.

Instructor : Mr. M.S. Puttaraju

HOD : Dr. T.C. Manjunath, Ph.D. (IIT Bombay)

About the college & the department

The Dayananda Sagar College of Engineering was established in 1979, was founded by Sri R. Dayananda Sagar and is run by the Mahatma Gandhi Vidya Peetha Trust (MGVP). The college offers undergraduate, post-graduates and doctoral programmes under Visvesvaraya Technological University & is currently autonomous institution. MGVP Trust is an educational trust and was promoted by Late. Shri. R. Dayananda Sagar in 1960. The Trust manages 28 educational institutions in the name of "Dayananda Sagar Institutions" (DSI) and multi – Specialty hospitals in the name of Sagar Hospitals - Bangalore, India. Dayananda Sagar College of Engineering is approved by All India Council for Technical Education (AICTE), Govt. of India and affiliated to Visvesvaraya Technological University. It has widest choice of engineering branches having 16 Under Graduate courses & 17 Post Graduate courses. In addition, it has 21 Research Centres in different branches of Engineering catering to research scholars for obtaining Ph.D under VTU. Various courses are accredited by NBA & the college has a NAAC with ISO certification. One of the vibrant & oldest dept is the ECE dept. & is the biggest in the DSI group with 70 staffs & 1200+ students with 10 Ph.D.'s & 30^{+} staffs pursuing their research in various universities. At present, the department runs a UG course (BE) with an intake of 240 & 2 PG courses (M.Tech.), viz., VLSI Design Embedded Systems & Digital Electronics & Communications with an intake of 18 students each. The department has got an excellent infrastructure of 10 sophisticated labs & dozen class room, R & D centre, etc…

Vision & Mission of the Institute

Vision:

- To impart quality technical education with a focus on Research and Innovation emphasizing on Development of Sustainable and Inclusive Technology for the benefit of society.

Mission:

- To provide an environment that enhances creativity and Innovation in pursuit of Excellence.
- To nurture teamwork in order to transform individuals as responsible leaders and entrepreneurs.
- To train the students to the changing technical scenario and make them to understand the importance of sustainable and inclusive technologies.

Vision & Mission of the Department

Vision :

- To achieve continuous improvement in quality technical education for global competence with focus on industry, societal needs, research and professional success.

Mission:

- Offering quality education in Electronics and Communication Engineering with effective teaching learning process in multidisciplinary environment.
- Training the students to take-up projects in emerging technologies and work with team spirit.
- To imbibe professional ethics, development of skills and research culture for better placement opportunities.

Program Education Objectives

After four years, the students will be

PEO1 : ready to apply the state-of-art technology in industry and meeting the societal needs with knowledge of Electronics and Communication Engineering due to strong academic culture.

PEO2 : competent in technical and soft skills to be employed with capability of working in multidisciplinary domains.

PEO3 : professionals, capable of pursuing higher studies in technical, research or management programs.

Program Specific Outcomes

Students will be able to

PSO1 : Design, develop and integrate electronic circuits and systems using current practices and standards.

PSO2 : Apply knowledge of hardware and software in designing Embedded and Communication systems.

Course Outcomes

After the completion of this laboratory the students will have the ability to ….

1. Design low pass, high pass, band pass and band reject filters for different cut-off frequencies.
2. Design Schmitt trigger circuit for the given values of UTP and LTP
3. Understand and analyze the of different modulation techniques such as Amplitude Modulation, Pulse Amplitude Modulation, Pulse Width Modulation, Pulse position Modulation and Frequency Modulation.
4. Design and verify the R-2R DAC circuit.
5. Design and verify the monostable/astable multivibrators for specified pulse width/duty cycle.
6. Understand and analyze the Precision Rectifier circuit using Op-Amp.

CO1	Gain hands-on experience in FM generation
CO2	Make the right choice of an IC and design the circuit for a given application
CO3	Design filters for different cut-off frequencies.
CO4	Design and verify the performance of Multivibrators for specified pulse width/duty cycle.
CO5	Understand the applications of Linear IC for Rectification, Conversion of different signals
CO6	Understand and analyze the of different analog modulation and pulse modulation techniques

Mapping of Course outcomes to Program outcomes

	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12
CO1	2	-	2	-	1	-	-	-	2	-	1	1
CO2	2	1	3	-	1	-	-	-	2	-	1	1
CO3	2	1	3	-	1	-	-	-	2	-	1	1
CO4	2	1	3	-	1	-	-	-	2	-	1	1
CO5	2	1	2	-	1	-	-	-	2	-	1	1
CO6	2	1	-	-	1	-	-	-	2	-	1	1

ANALOG COMMUNICATION AND LINEAR INTERGRATED CIRCUITS LAB

Course code : 17EC4DLACL Credits: 2 & 2 hrs per lab

L : P : T : S : 1 : 2 : 0 : 0 CIE Marks: 50

Exam Hours : 3 SEE Marks: 50

EXPT	Course Content	Hours	COs
1	Second order active LPF and HPF	02	CO2 CO3
2	Second order active BPF and BEF	02	CO2 CO3
3	Schmitt Trigger Design and test a Schmitt trigger circuit for the given values of UTP and LTP	02	CO2 CO5
4	Design and test R-2R DAC using op-amp	02	CO2 CO5
5	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.	02	CO2 CO4
6	Amplitude modulation using transistor (Generation and detection)	02	CO6
7	Pulse Amplitude Modulation and Detection	02	CO2 CO6
8	Pulse Width Modulation and Pulse Position Modulation	02	CO2 CO6
9	Frequency modulation using 8038/2206	02	CO1
10	Precision rectifiers – both Full Wave and Half Wave	02	CO2 CO5

Cycle of experiments

DO’s

- Students should follow the dress code of the laboratory compulsorily.
- Keep your belongings in the corner of the laboratory.
- Students have to enter their name, USN, time in/out and signature in the log register maintained in the laboratory.
- Students are required to enter components in the components register related to the experiment and handle the equipment’s smoothly.
- Check the components, range and polarities of the meters before connecting to the circuit.
- Come prepare for the experiment and background theory.
- Before connecting to the circuit refer the designed circuit diagram properly. Debug the circuit for proper output.
- Students should maintain discipline in the laboratory and keep the laboratory clean and tidy.
- Observation book and Record book should be complete in all respects and get it corrected by the staff members.
- Clarify the doubts with staff members and instructors.
- Experiment once conducted, in the next lab, the entire record should be complete in all respects, else the student will lose the marks.
- For programming lab, show the output to the concerned faculty.
- All the students should come to LAB on time with proper dress code and identity cards
- Keep your belongings in the corner of laboratory.
- Students have to enter their name, USN, time-in/out and signature in the log register maintained in the laboratory.
- All the students should submit their records before the commencement of Laboratory experiments.
- Students should come to the lab well prepared for the experiments which are to be performed in that particular session.
- Students are asked to do the experiments on their own and should not waste their precious time by talking, roaming and sitting idle in the labs.
- Observation book and record book should be complete in all respects and it should be corrected by the staff member.
- Before leaving the laboratory students should arrange their chairs and leave in orderly manner after completion of their scheduled time.

- Prior permission to be taken, if for some reasons, they cannot attend lab.
- Immediately report any sparks/ accidents/ injuries/ any other untoward incident to the faculty /instructor.
- In case of an emergency or accident, follow the safety procedure.
- Switch OFF the power supply after completion of experiment.

DONT's

- Do not switch on the power supply before verification of the connected circuits by concerned staff.
- Do not feed higher voltages than rated to the device.
- Do not upload, delete or alter any software on the laboratory PC's.
- Do not write or mark on the equipment's.
- Usage of mobile phone is strictly prohibited.
- Ragging is punishable.
- If student damages the equipment or any of the component in the lab, then he / she is solely responsible for replacing that entire amount of the equipment or else, replace the equipment.
- The use of mobile/ any other personal electronic gadgets is prohibited in the laboratory.
- Do not make noise in the Laboratory & do not sit on experiment table.
- Do not make loose connections and avoid overlapping of wires.
- Don't switch on power supply without prior permission from the concerned staff.
- Never point/touch the CRO/Monitor screen with the tip of the open.

Experiment No. : 1	Date : / / .

ACTIVE LOW PASS AND HIGH PASS FILTER (2nd ORDER)

Aim : To design a 2nd order low pass and high pass filter and to draw the frequency response.

Apparatus/Components required:

Op-amp IC741, Resistors, Capacitors, Power supply, Signal generator, CRO

Theory: A filter is frequency selective circuit that passes a specified band of frequencies and blocks signals of frequencies outside this band. Filters may be classified as follows. Analog & Digital, Passive & Active, Audio frequency & Radio frequency filters. Depending on the elements used in construction, filters may be classified as passive and active. Elements used in passive filters are: Resistors, Capacitors and Inductors, and in active filters Op-amps or Transistors are used along with resistors and capacitors. There are many advantages of using active filters:

a) Since opamp is used in the construction of active filter, there is flexibility in gain and frequency adjustment.
b) Since opamp input impedances is infinite, no loading problem on filter characteristics.
c) This is applicable over a wide range of frequency.
d) This is cheaper in cost wise since big inductors are not used.

Working of the circuit: At low frequencies the reactance of the capacitor is high, hence capacitor acts as open circuit and all low frequency signals are passed and hence filter has a constant gain from 0 Hz to high cut off frequency F_h, the gain decreases with increase in frequency and at F_h the gain is down by 3dB. A high pass filter does not pass any signal up to lower cutoff F_L, after F_L gain increases with increase in frequency. 2nd order filter have roll of 40 dB/decade, i.e., the gain increases or decreases at the rate of 40 dB/decade in the stop band. Active filters are mainly used in the field of DSP, Radio, T.V. and Radar.

Design :

$$F_h = \frac{1}{2\pi\sqrt{R_3 R_2 C_3 C_2}}$$

Let $R_3 = R_2 = R$ & $C_3 = C_2 = C$

$$F_h = \frac{1}{2\pi\sqrt{R^2 C^2}} = \frac{1}{2\pi RC}$$

Let F_h = 1 KHz & C = 0.01 μF

$$F_h = \frac{1}{2\,\pi\,F_h\,C}$$

R = 15.9 KΩ

$$A = 1 + \frac{R_f}{R_1}$$

For 2nd order filter, A = 1.586

R_1 = 10 KΩ

$\therefore$ R_f = 5.86 KΩ

Procedure:

1) Connections are made as per the circuit diagram.
2) Input voltage (V_i) of amplitude 1 V/2 V is applied from the signal generator.
3) Input frequency is varied in steps from 10 Hz to 100 KHz and corresponding output voltage (V_0) is noted.
4) Gain in dB is calculated by using equation 20 $\log_{10}$ (V_0/V_i).
5) Graph of Frequency versus Gain in dB is plotted in semi log sheet and roll off is calculated which is approximately equal to 40 dB/decade.

Circuit Diagram : Active Low Pass Filter

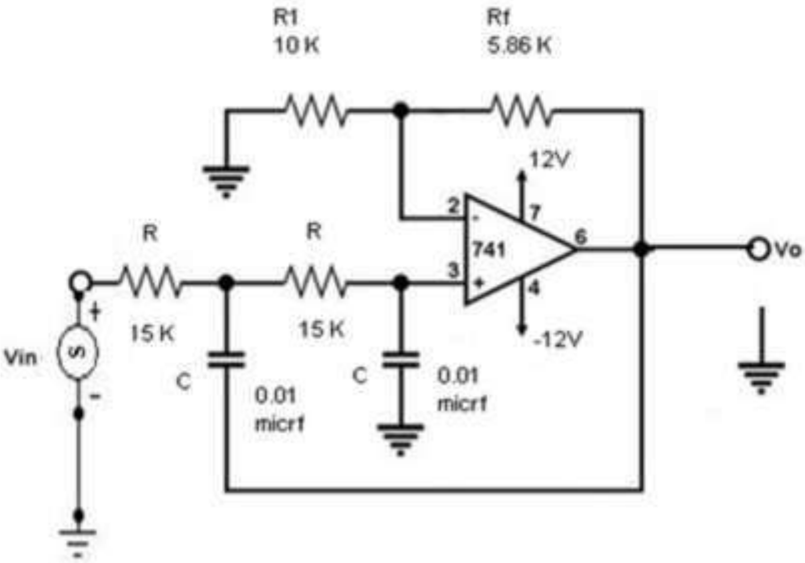


Fig. : Active Low Pass Filter

Frequency Response

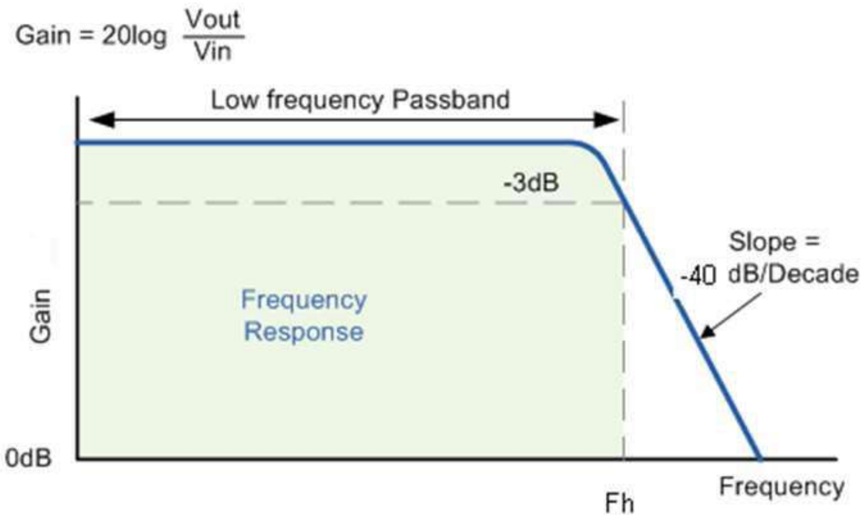


Fig. : Frequency Response

Tabular Column:

Input voltage : V_i = 1 V or 2 V

Frequency in Hz	Output Voltage V_0 (V)	Gain dB = 20 $\log_{10}$ (V_0/V_i)
100 Hz		
15 KHz		

Results :

The experiment was conducted successfully, the frequency response is drawn and the roll off factor is calculated.

Roll off factor for LPF = ___________________

Applications:

1.

2.

3.

4.

ACTIVE HIGH PASS FILTER

Design:

$$F_1 = \frac{1}{2\pi RC}$$

Let F_1 = 1 KHz, C = 0.01 μF

$$R = \frac{1}{2\pi C F_1} = 15.9\ \text{K}\Omega$$

$$A = 1 + \frac{R_f}{R_1}$$

For 2nd order filter, A = 1.586

Let R_1 = 10 KΩ, therefore R_f = 5.86 KΩ

Circuit Diagram : ACTIVE HIGH PASS FILTER

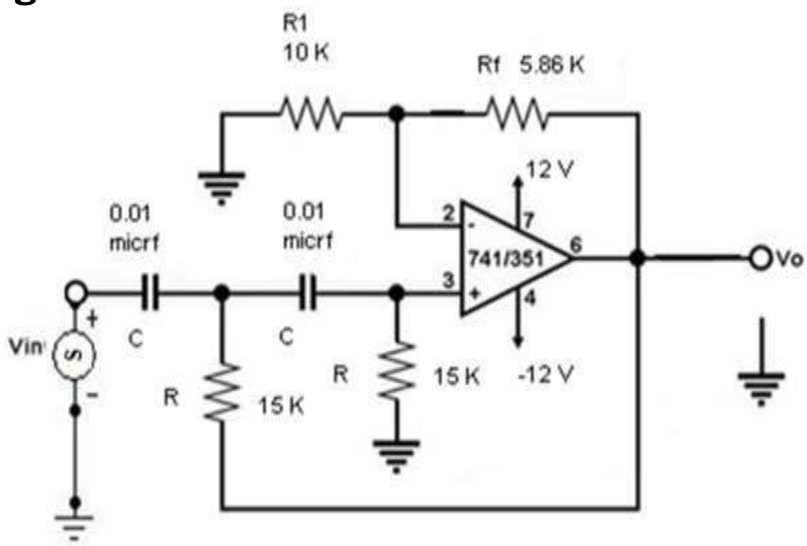


Fig.: Active high pass filter

Frequency Response

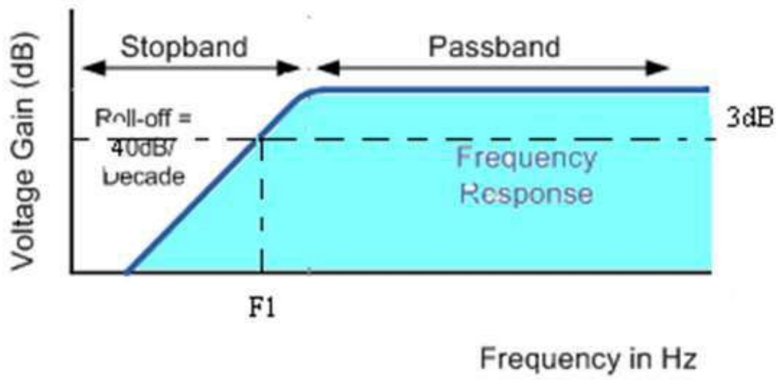


Fig. : Frequency Response

Tabular Column:

Input voltage: V_i = 1 V or 2 V

Frequency in Hz	Output Voltage V_0 (V)	Gain dB = 20 $\log_{10}$ (V_0/V_i)
50 Hz		
5 KHz		

Results:

The experiment was conducted successfully, the frequency response is drawn and the roll off factor is calculated.

Roll off factor for HPF = ___________________

Applications:

1.
2.
3.
4.

Remarks:

Signature of staff incharge with date:

Probable viva questions:

1. Define a filter. How are filters classified?
2. Define gain, stop band and pass band of a filter.
3. Explain the operation of a second order High pass filter.
4. Explain the operation of a second order Low pass filter.
5. How can a first order low pass filter can be converted into second order low pass filter.
6. What is Roll off factor of filter ?
7. What is the pass band voltage gain of a second order low pass butterworth filter.

References:

1. http://www.nptel.ac.in/courses/117107094/lecturers/lecture_16/lecture16_page1.htm
2. http://textofvideo.nptel.iitm.ac.in/122106025/lec37.pdf

Experiment No. : 2	Date : / / .

ACTIVE BAND PASS AND BAND REJECT FILTER (2nd ORDER)

Aim: To design a 2nd order band pass and band reject filter and to draw the frequency response.

Apparatus/Components required:

Op-amp IC741, Resistors, Capacitors, Power supply, Signal generator, CRO.

Theory:

The band pass filter has pass band between two cut-off frequencies F_2 and F_l such that $F_2 > F_l$. There are two types of band pass filters, namely wide band and narrow band pass filter. A filter is defined as wide band pass filter if its figure of merit or quality factor Q < 10 and also $A_f < 2Q^2$.The band reject filter is also called Band elimination filter and in this filter frequencies are attenuated in the stop band while they are passed outside this band. The narrow band reject filter is also called notch filter and it is mainly used in rejection of single frequency such as 60 Hz power line frequency used in power line.

Design:

Let

$$F_C = 5\text{ KHz} \ \& \ C_1 = C_2 = C = 0.001\ \mu F$$

$$B.W. = 500\text{ Hz} \ \& \ A_f = 5$$

$$Q = \frac{F_C}{BW} = 10$$

$$R_1 = \frac{Q}{2\pi F_C C A_f} = 63.66\ K\Omega$$

$$R_2 = \frac{Q}{2\pi F_C C\left(2Q^2 - A_f\right)} = 1.6\ K\Omega$$

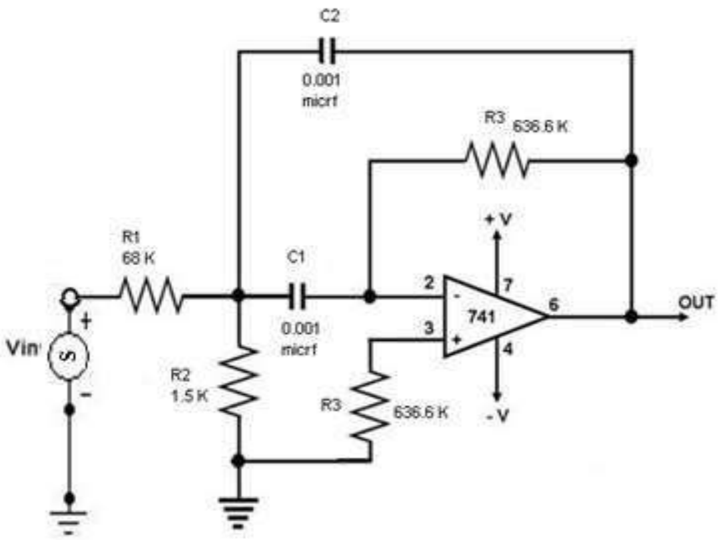
$$R_3 = \frac{Q}{\pi F_C C} = 636.6\ K\Omega$$

Procedure:

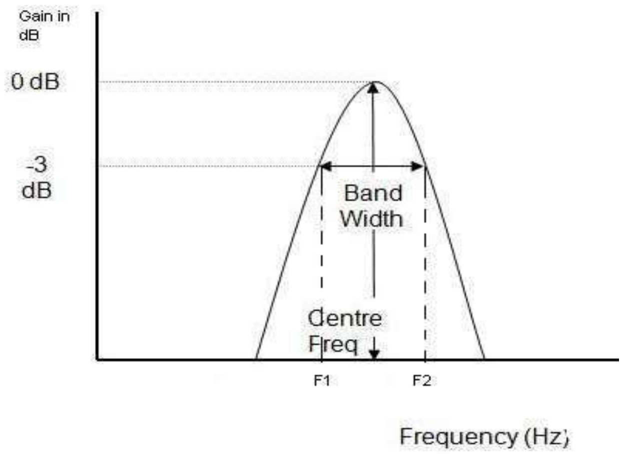
1. Connections are made as per the circuit diagram.
2. Input voltage (Vi) of amplitude 1 V-2 V is applied from the signal generator.
3. Input frequency is varied in steps from 10 Hz to 100 KHz and corresponding output voltage (V_0) is noted.
4. Gain in dB is calculated by using equation 20 $\log_{10}(V_0/V_i)$.
5. Graph of frequency versus gain in dB is plotted in semi log sheet.

Circuit Diagram:

<u>ACTIVE BAND PASS FILTER</u>



Frequency Response:



Tabular Column:

Input voltage : V_i = 1V or 2V

Frequency in Hz	Output Voltage V_0 (V)	Gain dB = 20 $\log_{10}$ (V_0/V_i)
1K Hz		
10K Hz		

ACTIVE BAND REJECT FILTER

Design:

Let

$$F_C = 100 \text{ Hz}$$

$$C = 0.1\ \mu F$$

$$B.W. = 10 \text{ Hz}$$

$$F_C = \frac{1}{2\pi RC}$$

$$R = \frac{1}{2\pi F_C C} = 15.9 \text{KHz}$$

$$\text{Choose } R_1 = R_2 = R = 15\ K\Omega$$

$$C_1 = C_2 = C = 0.1\ \mu F$$

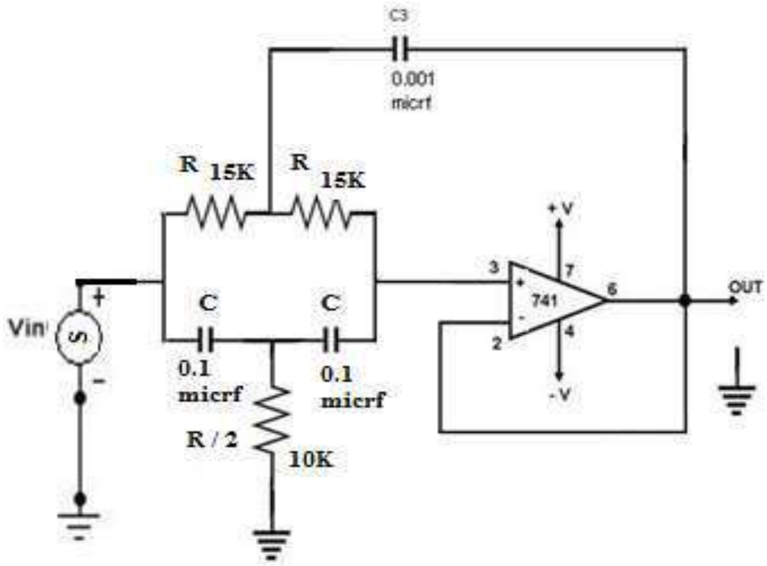
$$\text{and } C_3 = 2C = 0.001\ \mu F$$

$$R_3 = \frac{R}{2} = 8\ K\Omega \approx 10\ K\Omega$$

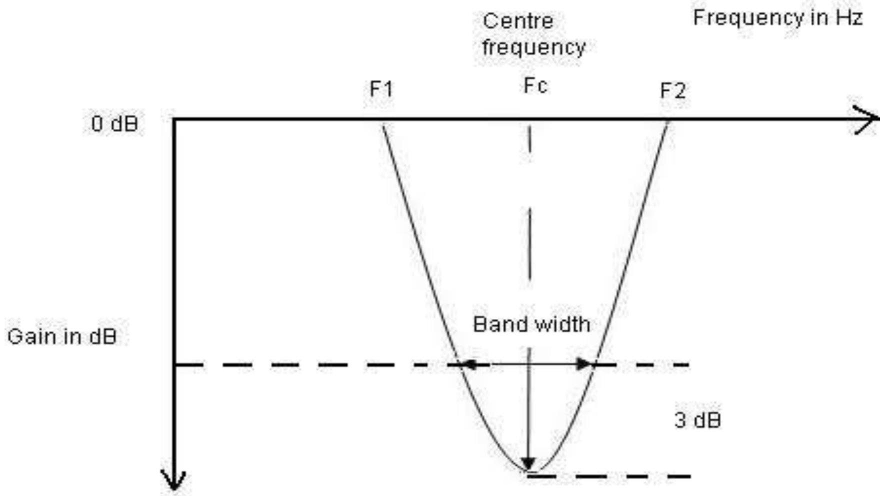
Procedure:

1. Connections are made as per the circuit diagram.
2. Input voltage (V_i) of amplitude 1 V - 2 V is applied from the signal generator.
3. Input frequency is varied in steps from 10 Hz to 100 KHz and corresponding output voltage (V_0) is noted.
4. Gain in dB is calculated by using equation 20 log10 (Vo/Vi).
5. Graph of frequency versus gain in dB is plotted in semi log sheet.

Circuit Diagram:



Frequency Response:



Tabular Column:

Input voltage : V_i = 1 V or 2 V

Frequency in Hz	Output Voltage V_0 (V)	Gain dB = 20 $\log_{10}$ (V_0/V_i)
10 Hz		
300 Hz		

Results: The experiment was conducted & the frequency response is drawn

Bandwidth of BPF was ________

Applications:

1.

2.

3.

4.

Remarks :

Signature of staff incharge with date:

Probable viva questions:

1. What is band pass filter ? Which are the 2 types of BPF?
2. What is a notch filter?
3. Give some applications of filters as applied to communication.
4. What is the difference between Butterworth and Chebyshev filter?
5. Draw the frequency response of a band elimination filter?
6. What is all pass filters? What is its application?

References:

1. http://nptel.ac.in/courses/117107094/15

Experiment No. : 3	Date : / / .

DESIGN AND TEST R-2R DAC USING OP-AMP

Aim: To study the operation of 4 bit DAC using R-2R ladder network and to generate staircase wave using DAC circuit.

Apparatus/Components required: Resistors, Breadboard, Multimeter, Op-amp [µA 741], Connecting wires, Power supply.

Theory: A ladder is a series/parallel resistor network. An R-2R ladder type DAC is Shown in the figure. It needs only 2 values R and 2R. Here D_0, D_1, D_2 and D_3 are electronic switches, which are digitally controlled. When 1 is present on the MSB line, switch D_3 connects the resistor 2R to V_{ref}, conversely when 0 is present on the MSB line, the resistor 2R is connected to ground. Since the ladder is composed of linear resistors, it is a linear n/w and the principal of superposition can be used. This means that the total o/p voltage due to a combination of i/p digital levels can be found by simply taking the sum of the o/p that are caused by each of the many digital i/p's, all acting individually.

Design:

- Op-amp voltage follower acts as buffer stage.
- D_0, D_1, D_2 and D_3 are digital I/P may be low (0) or high (1).
- V_R (0) = 0 Volts
- V_R (1) = V_R = reference voltage can be selected depending on maximum analog o/p voltage required.
- If the digital i/p are obtained from a digital IC trainer then V_R = +5 V fixed.
- The analog voltage V_0 for a 4 bit DAC is given as

$$V_0 = \left[2^3 D_3 + 2^2 D_2 + 2^1 D_1 + 2^0 D_0\right]\left[\frac{V_R}{2^4}\right] \times \left[\frac{2R}{3R}\right]$$

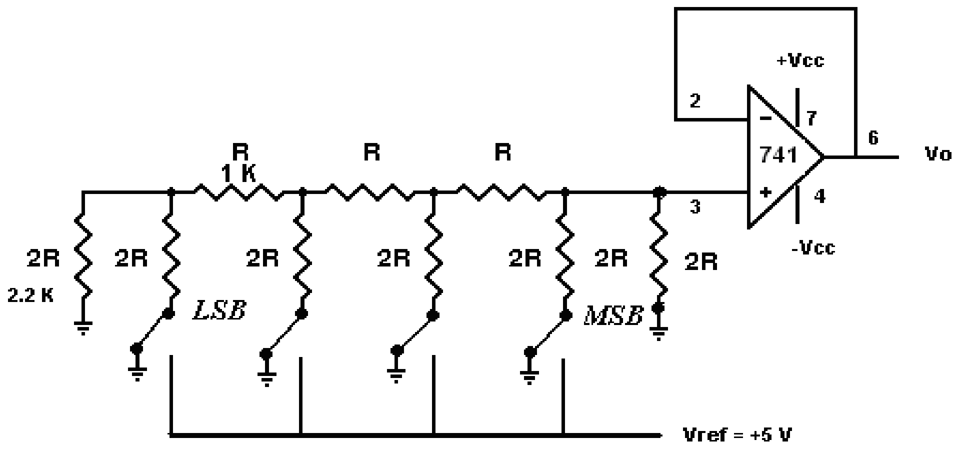
- If V_R = +5 volts, then

$$V_0 = \left[\frac{5}{2^4}\right].\left[2^3 D_3 + 2^2 D_2 + 2^1 D_1 + 2^0 D_0\right]\left[\frac{2}{3}\right]$$

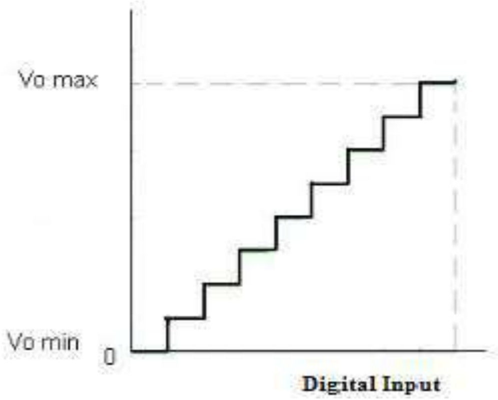
Procedure:

1. Rig up the circuit as per the circuit diagram.
2. Switch on the supply and apply the DC reference voltage of +5V for logic 1 and 0 for logic 0.
3. Vary them in stage from 0000 to 1111 as shown in the tabular column and measure the corresponding voltage using multi meter.
4. Readings are tabulated and verified

Circuit Diagram :



Waveform :



Tabular Column :

Decimal no	Digital i/p	Theoretical value	Expt value
0	0000	0	
1	0001	0.20883	
2	0010	0.4166	
3	0011	0.625	
4	0100	0.833	
5	0101		
6	0111		
7			
8			
9			
10			
11			
12			
13			
14			
15	1111		

Results:

R-2R DAC is designed and tested.

The theoretical and experimental values of O/P voltage are compared.

Applications:

1.

2.

3.

4.

Remarks :

Signature of Staff Incharge with date:

Probable viva questions:

1. What is the difference between A/D and D/A converters ?
2. Explain R/2R ladder technique of D/A conversion.
3. What are the different types of D/A conversion ?
4. Why the switches used in weighted resistor DAC are of single pole double throw (SPDT) type ?
5. What is the disadvantage of binary weighted type DAC ?

References:

1. "Operational Amplifiers and Linear IC's", David A. Bell, 2^{nd} edition, PHI/Pearson, 2004.

Experiment No. : 4	Date : / / .

AMPLITUDE MODULATION USING TRANSISTOR (GENERATION AND DETECTION)

Aim:

1. Design and construct collector Amplitude modulator for given carrier frequency 500 kHz and modulating frequency 1 kHz.
2. Study the variation in the modulation index as a function of modulating voltage.
3. Display the amplitude modulator output and trapezoidal pattern & to determine depth of modulation.

Apparatus/Components required: IFT, Transistor (SL100), resistor (470k, 120, 10k), capacitor 0.01 F (2 no's).

Theory: Amplitude modulation is a type of modulation where the amplitude of the carrier signal is varied in accordance with the information bearing signal. The envelope, or boundary, of the amplitude modulated signal embeds the information bearing signal. The total power of the transmitted signal varies with the modulating signal, whereas the carrier power remains constant. Nonlinear device such as a transistor is used to combine the carrier and the modulating signal to generate an amplitude modulated signal. The output of the nonlinear device consists of discrete upper and lower sidebands. The circuit used gives out 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. Modulation signal m(t) is applied in series with VCC through low frequency transformer. Modulated output is obtained through IFT.

Procedure:

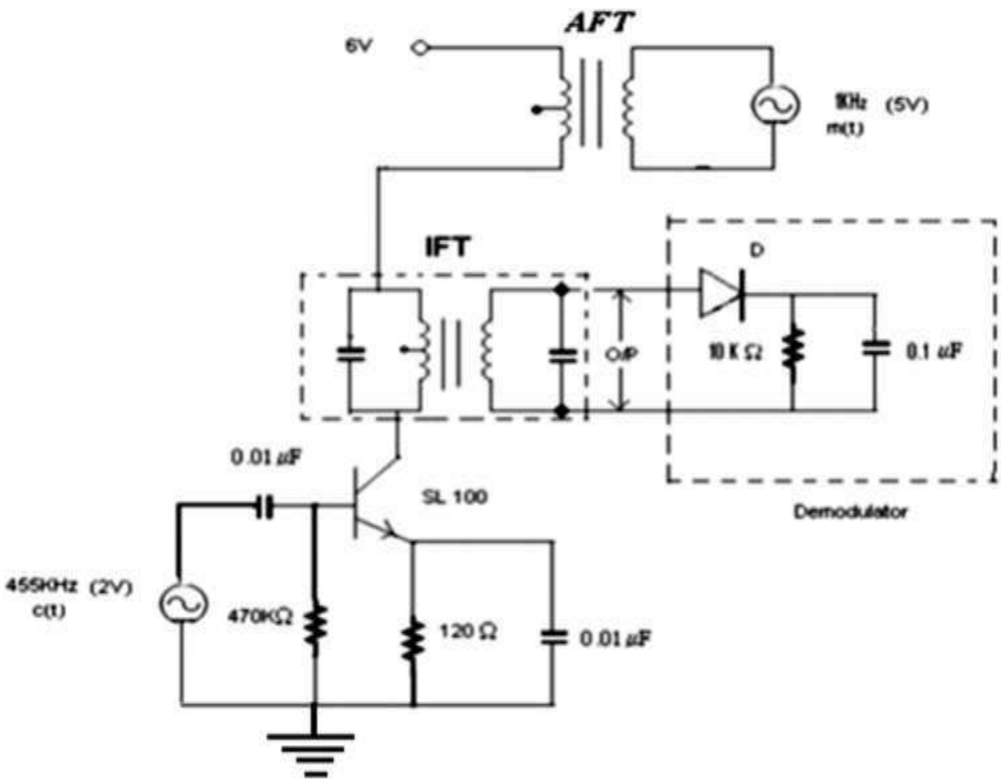
1. Connect the circuit as shown in the figure/circuit diagram.
2. Adjust the frequency of carrier wave around 500 kHz and amplitude 1v (p-p) to get a undistorted wave.
3. Adjust the frequency of m(t) around 500 Hz to 1 kHz and 10 v(p-p).
4. Feeding A_m output to Y plates and modulation signal to X plates of CRO obtain trapezoidal pattern.

$$m = \frac{L_1 - L_2}{L_1 + L_2}$$

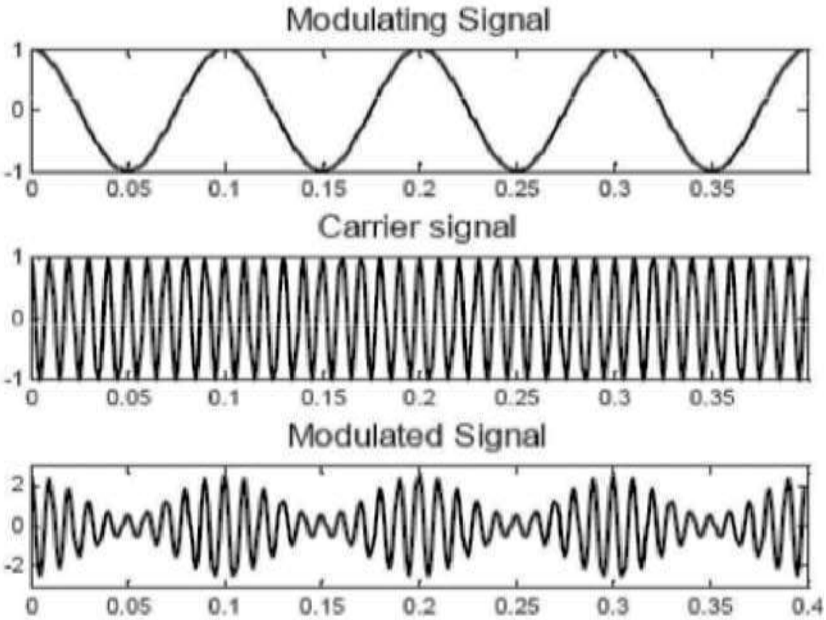
where, $L_1 = E_{max}$, $L_2 = E_{min}$

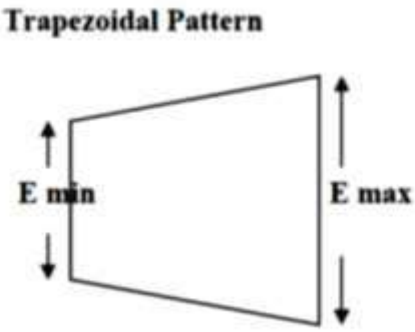
5. Keeping carrier amplitude constant vary modulation signal voltage in appropriate steps and measure modulation index, $m = \frac{E_{max} - E_{min}}{E_{max} + E_{min}}$
6. Tabulate results and draw graph of m v/s modulation voltage amplitude.

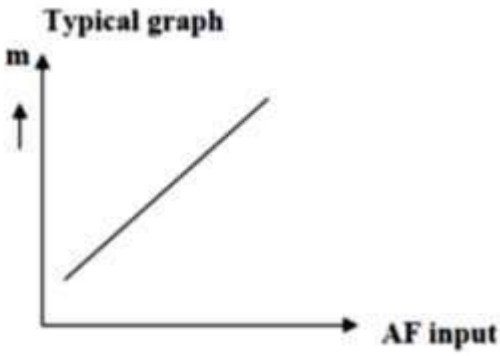
Circuit Diagram



Waveform:







Tabular Column:

AF input	E_{max}	E_{min}	m

$$m = \frac{E_{\max} - E_{\min}}{E_{\max} + E_{\min}}$$

Results: The experiment was conducted & the results are tabulated, the graph is plotted for m v/s modulating voltage (AF input)

Applications:

1.

2.

3.

4.

Remarks :

Signature of Staff Incharge with date:

Probable viva questions:

1. What are the basic components of electronic communication ?
2. What is a communication channel? State two types of communication channel.
3. Define analog and digital signals.
4. What is a carrier signal ?
5. Name the basic types of continuous modulation.
6. What is over modulation ? Why over modulation is not preferred ?
7. How much power will be saved if the carrier of a 100 percent modulated AM wave is suppressed ?
8. What is the position of the operating point of class-C amplifier used in collector modulation ?

References:

1. An Introduction to Analog and Digital Communication, Simon Haykins, John Wiley India Pvt. Ltd., 2008.

Experiment No. : 5	Date : / / .

PULSE AMPLITUDE MODULATION AND DEMODULATION

Aim: To conduct an experiment to generate PAM signal and also design a circuit to demodulate the obtained PAM signal and verify sampling theorem and Plot relevant Waveforms.

Apparatus/Components required: Transistor, Resistors, Capacitor, Power Supply, Signal generators, CRO.

Theory: In Pulse Amplitude Modulate (PAM), the amplitude of the periodic pulses varies with instantaneous sample values of a continuous message signal. PAM modulator is a simple emitter follower circuit. In the absence of clock pulse output follows the input base of transistor coupled with modulating signal and carrier. Amplitude of carrier is kept high to bring transistor in the cut off region. When pulse signal is high transistor circuit behaves as an emitter follower and output follows the input-modulating signal. When the clock signal is low the transition is cut off and output is zero. PAM demodulator is simple RC low pass circuit where RC time constant has a cut off frequency of F_m (i.e.,) higher frequency component of modulating signal. The sampled signal can be recovered exactly when sampling frequency is equal or greater than twice the highest frequency component of modulating signal. PAM signal is often used as input to a PCM that will change analog signals to binary code.

Design :

MODULATOR circuit is designed as follows ….

Let

$I_c = I_E = 0.5$ mA, when Q is on,

$$I_E = \frac{V_{CC} - V_{CE}}{R_E} = \frac{5 - 2.5}{R_E}$$

$$R_E = \frac{2.5\text{V}}{0.5m\text{A}} = 5K\Omega \approx 4.7\text{K}\Omega$$

$I_B = I\ R_1$ [with m(t) = 0]

$$I_\text{B} = \frac{I_\text{C}}{\beta}$$

Let $\beta = 2$

$$I_\text{B} = \frac{0.5m\text{A}}{2} = 250\text{ A}$$

$V_E = I_E R_E = (0.5 \times 10^{-3})(5 \times 10^3) = 2.5\text{V}$

$V_A = V_E + V_{BE} = 3.2\text{V}$

$$R_1 = \frac{V_{\text{g1}} - V_\text{A}}{I_\text{B}} = \frac{5 - 3.2}{250} = 0.0072\,\Omega$$

$R_1 = 7.2$ K = 10 KΩ

$$I_{\text{R}_2} = \frac{I_\text{B}}{2} = \frac{250}{2} = 125\text{ A}$$

$$R_2 = \frac{V_\text{A} - V_{\text{g2}}}{I_{\text{R}_2}} = \frac{3.2 - 1}{125} = 18\,\text{K}\Omega$$

Choose R_2 = 22 KΩ

Procedure:

1. Before wiring the circuit, check all the components using the multimeter.
2. Rig up the connection as shown in the circuit diagram.
3. Set the C(t) frequency to 2 KHz and amplitude 10V P-P. (V_{g1}= 5Vpeak)
4. Set the m(t) frequency to 200 Hz and amplitude 2V P-P. (V_{g2} = 1V peak)
5. Check the PAM O/P at the emitter by varying the C(t) and m(t) amplitude if necessary.
6. Rig up the demodulator circuit.
7. Give PAM output as input.
8. To get undistorted m(t) change the C(t) frequency to higher value; to 10 KHz

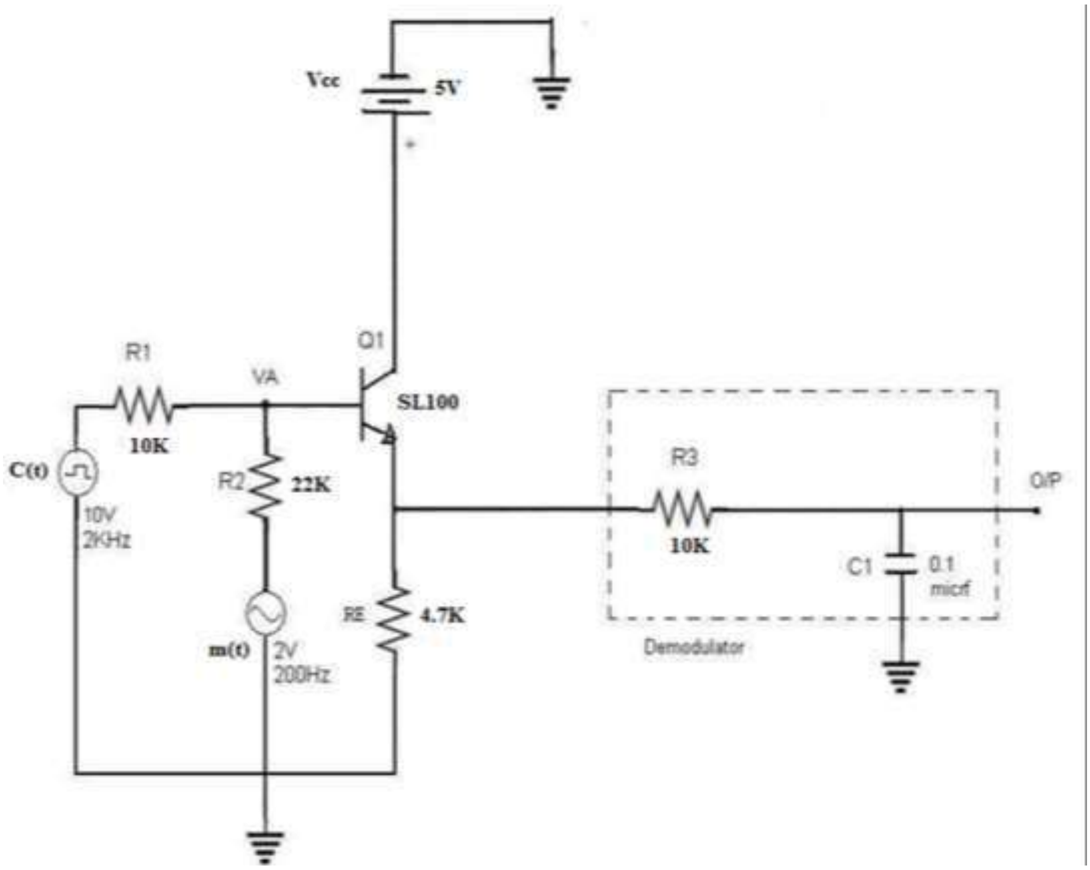
9. Verification of sampling theorem

a) Over sampling $f_s > 2f_m$ (f_m = 200 Hz, f_s = 2 KHz)

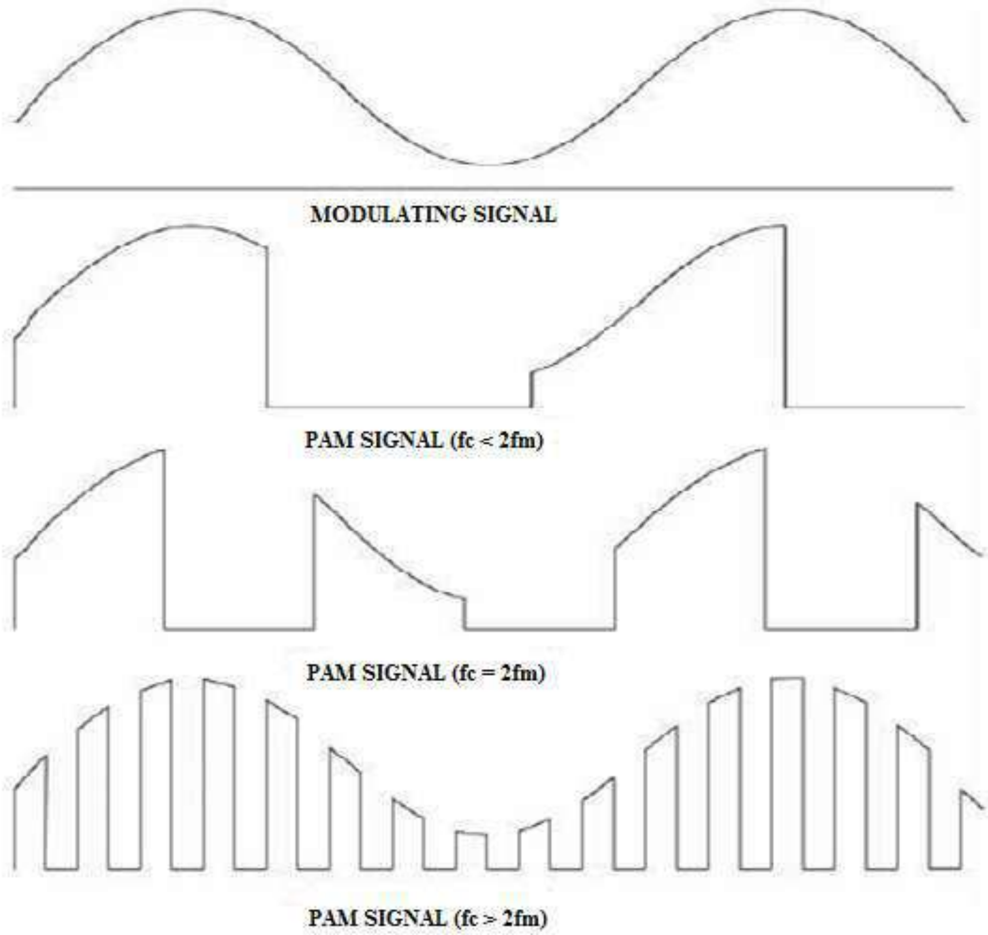
b) Under sampling $f_s < 2f_m$ (f_m = 200 Hz, f_s = 40 Hz)

c) Correct sampling $f_s = 2f_m$ (f_m = 200 Hz, f_s = 400 Hz)

Circuit Diagram :



Waveform :



Results: The experiment was conducted & the PAM output is verified.

Applications:

1.

2.

3.

4.

Remarks :

Signature of Staff Incharge with date:

Probable viva questions:

1. What is PAM ?
2. What is the difference between AM and PAM ?
3. What is the basic principle of PAM ?
4. Mention the applications of PAM.
5. What is the process of sampling an analog signal.
6. Which multiplexing technique is used in PAM ?
7. What do you mean by Nyquist rate ?

References:

1. Communication Systems, Simon Haykins, 5th Edition, John Willey, India Pvt. Ltd, 2009.

Experiment No. : 6A	Date : / / .

DESIGN AND TESTING OF ASTABLE MULTIVIBRATOR FOR THE GIVEN FREQUENCY AND DUTY CYCLE USING IC 555

Aim: To Design a symmetrical and unsymmetrical astable multivibrator using IC 555 timer.

Apparatus/Components required: IC555, Resistance 6.8 KΩ & 3.3 KΩ, capacitance = 0.1 & 0.01 µF.

Theory: IC555, an astable multivibrator and in this mode of operation it generates rectangular pulses, by suitably modifying the circuit, square wave can be generated. This circuit can be made to work as square wave generator. This has 2 o/p states both of this states are quasi-stable states, hence the name free running oscillator. The multivibrator keeps on switching b/w 2 states by itself and it does not need any external triggering. The circuit diagram is as shown in the figure. If the time taken by capacitor is equal to the time for discharge then the $T_{ON}= T_{OFF}$, it is called symmetrical astable multivibrator and if T_{ON} not equal to T_{OFF}, i.e., time taken for charging is not equal to the time taken for discharging then it's an unsymmetrical astable multivibrator.

Working: Initially when the output is high capacitor C starts charging towards V_{CC} through R_A and R_B. However as soon as the voltage across the capacitor equals 2/3 V_{CC}, comparator 1 triggers the flip-flop and the output switches to low state. Now capacitor C discharges through R_B and the transistor Q1. When voltage across C equals 1/3 V_{cc}, comparator 2's output triggers the flip-flop and the output goes high. Then the cycle repeats. The capacitor is periodically charged and discharged between 2/3 V_{CC} and 1/3 V_{CC} respectively. The time during which the capacitor charges from 1/3 V_{cc} to 2/3 V_{cc} is equal to the time the output remains high and is given by

$$t_c = 0.693\ (R_A + R_B)\ C$$

where, R_A and R_B are in ohms and C is in Farads.

Similarly, the time during which the capacitor discharges from 2/3 V_{cc} to 1/3 V_{cc} is equal to the time the output is low and is given by

$$t_d = 0.693\ R_B\ C$$

Thus, the total time period of the output waveform is

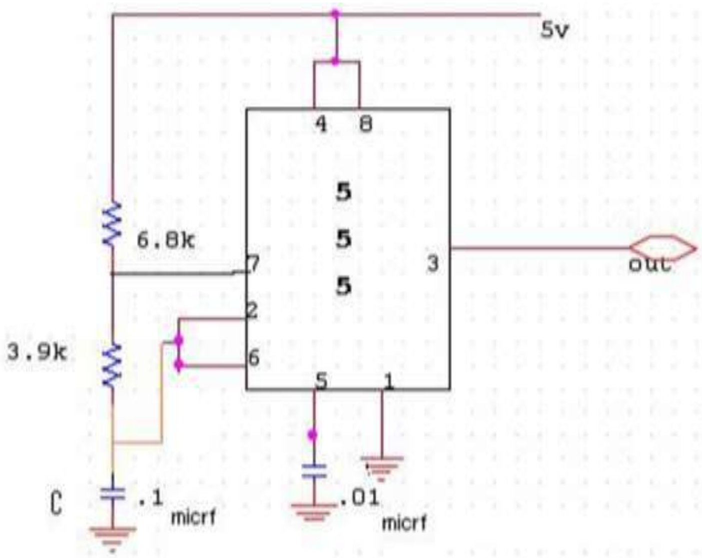
$$T = t_c + t_d = 0.693\ (R_A + 2R_B)\ C$$

Procedure:

<u>FOR UNSYMMETRICAL SQUARE WAVES</u>

1) Generate a signal of frequency 1 KHz.
2) Set up the circuit as shown in the figure with R_A = 6.8 KΩ R_B = 3.3 KΩ
3) Observe the out wave forms at PIN 3
4) Verify V_{TL} =1/3 V_{CC} and V_{TH} = 2/3 V_{CC} at PIN 6.

Circuit Diagram



Observations and calculations :

$$F = 1\ KHz$$

$$D = 0.75$$

$$C = 0.1\ F$$

$$T_{ON} = 0.693\ (R_A + R_B)\ C$$

$$T_{OFF} = 0.693\ (R_B)\ C$$

$$T = T_{ON} + T_{OFF}$$

$$T = 0.693\ (R_A + 2R_B)\ C \ \ldots\ldots\ldots\ldots\ldots\ (1)$$

$$D = \frac{T_{ON}}{T_{ON} + T_{OFF}}$$

$$D = \frac{R_A + R_B}{R_A + 2R_B} \ \ldots\ldots\ldots\ldots\ldots\ (2)$$

$$F = \frac{1}{T} \ \ldots\ldots \text{ using eqn. (1)}$$

$$F = \frac{1.44}{[(R_A + 2R_B)C]}$$

$$R_A + 2R_B = 14.4\ K\Omega \ \ldots\ldots\ldots\ldots\ldots\ (3)$$

Substitute this in eqn (2) with D = 0.75

$$R_A + R_B = 10.8\ K\Omega \ \ldots\ldots\ldots\ldots\ldots\ (4)$$

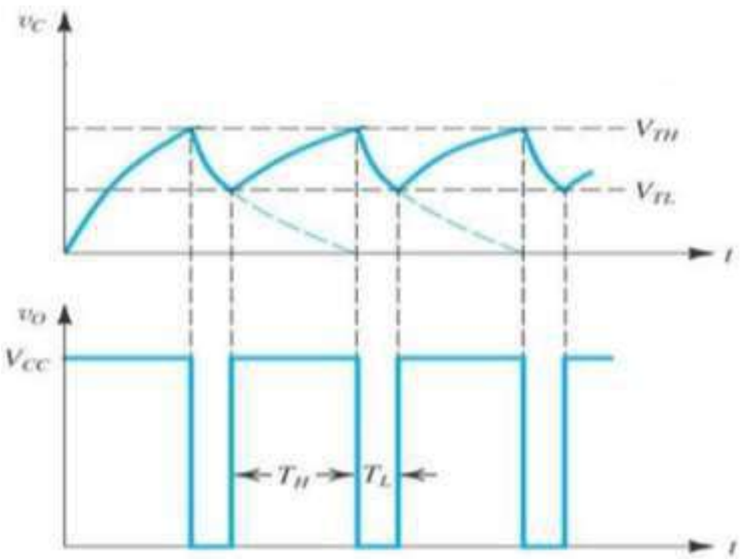
Solving eqns. (3) and (4), we get

$$R_A = 7.2\ K\Omega$$

$$R_B = 3.6\ K\Omega$$

Therefore, $R_A = 6.8\ K\Omega$ and $R_B = 3.3\ K\Omega$ is used.

Waveforms :



OBSERVATIONS AND CALCULATIONS:

V_{TH} =

V_{TL} =

T_H =

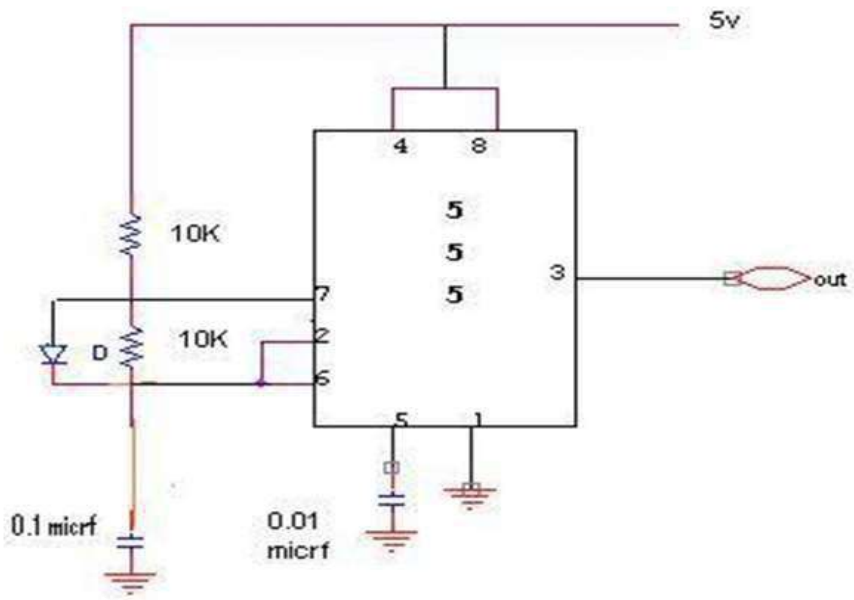
T_L =

D =

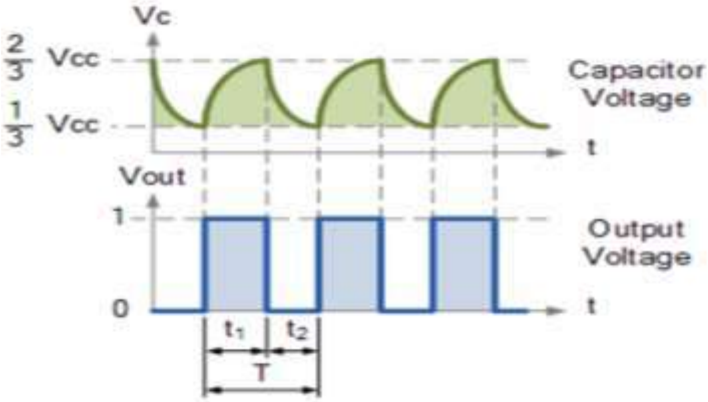
Procedure: FOR GENERATING SYMMETRICAL SQUARE WAVE

1. Connect the point contact diode (0A79) in parallel with RB (Anode of the diode at pin 7)
2. The duty cycle of the o/p becomes 50 %.
3. Now o/p wave form timings are T_{ON} = 0.695 R_AC & T_{OFF} = 0.695 R_BC
4. Choose R_A = R_B = 10 KΩ & C = 0.1 F and setup the circuit and observe the o/p wave forms and verify the observation of the circuit.

Circuit Diagram :



Waveform:



CALCULATIONS :

T_{ON} =

T_{OFF} =

D =

Results: The circuit is constructed successfully and the working is verified.

Applications:

1.
2.
3.
4.

Remarks :

Signature of Staff Incharge with date:

Probable viva questions:

1. Explain the functional block diagram of IC 555.
2. State the working of IC 555 as an astable multivibrator ?
3. Define duty cycle.
4. How astable mode of 555 can be modified to get a symmetrical square wav ?
5. What is a multivibrator ?
6. Why is an astable multivibrator called so ?
7. Mention the applications of astable multivibrator ?

References:

1. "Operational Amplifiers and Linear IC's", David A. Bell, 2nd edition, PHI/Pearson, 2004.

Experiment No. : 6B	Date : / / .

DESIGN AND TEST MONOSTABLE MULTIVIBRATOR FOR THE GIVEN PULSE WIDTH W USING IC 555

Aim: To connect IC555 for monostable operation & to design and test a monostable multivibrator for the given pulse width w using IC 555.

Apparatus/Components required: IC555 Timer, Resistor (as per calculation) & CRO with connecting wires, etc.

Theory: Monostable multivibrator is a circuit, in which one of the states is stable, but the other is not—the circuit will flip into the unstable state for a determined period, but will eventually return to the stable state. Such a circuit is useful for creating a timing period of fixed duration in response to some external event. This circuit is also known as a one shot. A common application is in eliminating switch bounce.

Working operation: Initially when the circuit is in the stable state i.e., when the output is low, transistor Q1 is ON and the capacitor C is shorted out to ground. Upon the application of a negative trigger pulse to pin 2, transistor Q1 is turned OFF, which releases the short circuit across the external capacitor C and drives the output high. The capacitor C now starts charging up towards V_{CC} through R. When the voltage across the capacitor equals 2/3 $V_{C\,C}$, comparator 1's output switches from low to high, which in turn drives the output to its low state via the output of the flip-flop. At the same time the output of the flip-flop turns transistor Q1 ON and hence the capacitor C rapidly discharges through the transistor. The output of the monostable remain slow until a trigger pulse is again applied. Then the cycle repeats. The pulse width of the trigger input must be smaller than the expected pulse width of the output waveform. Also the trigger pulse must be a negative going input signal with amplitude larger than 1/3 V_{CC}.

The time during which the output remains high is given by t = 1.1 RC, where R is in Ohms and C is in Farads. Once triggered, the circuit's output will remain in the high state until the set time, *t* elapses. The output will not change its state even if an input trigger is applied again during this time interval *t*. The circuit can be reset during the timing cycle by applying negative pulse to the reset terminal. The output will remain in the low state until a trigger is again applied.

Design Procedure :

R = 10 KΩ, C = 0.1μF

Voltage across capacitor $V_C = V_{CC}\left(1 - e^{-t/RC}\right)$

At t = T, $V_C = \frac{2}{3}V_{CC}$

Equating the above 2 equations, we get

$$\frac{2}{3}V_{CC} = V_{CC}\left(1 - e^{-t/RC}\right) = 1.1RC$$

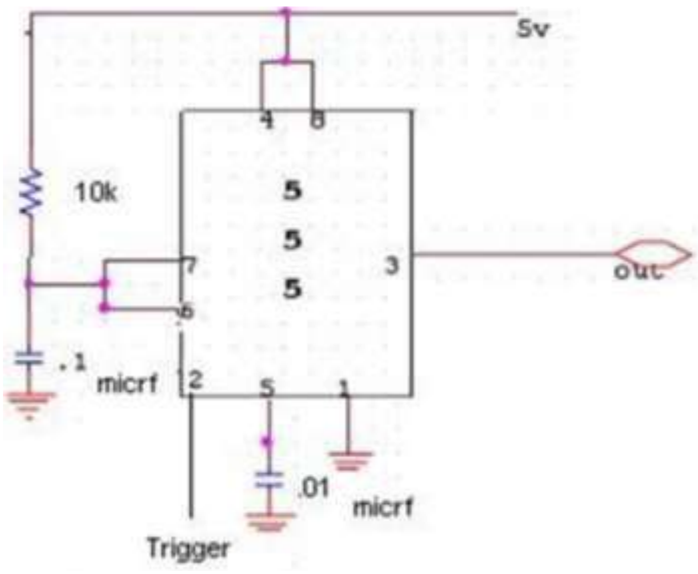
Substitute the values of R & C, we get

T = 1.1 ms

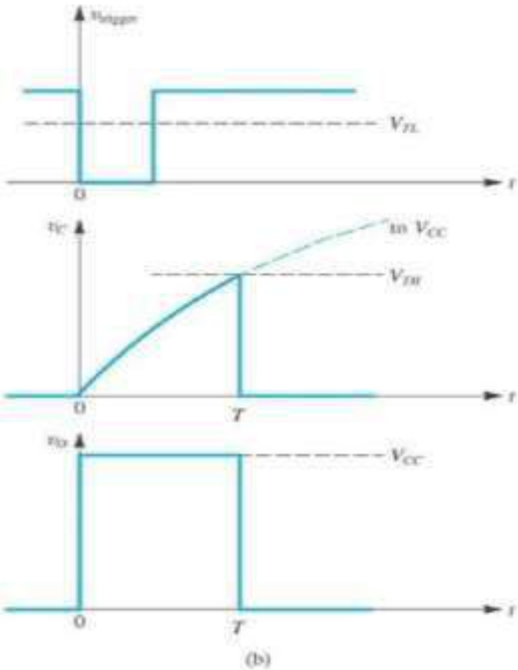
Procedure: <u>As a non–retriggerable one shot</u>

1) Set up the circuit as shown in figure below in the circuit diagram.
2) Apply Negative pluses of 2 kHz frequency at the trigger input (pin 2) [Note : usually the width of triggering pulse must be less than T_{ON}]
3) Observe the output (pin3) and measure pulse duration
4) Calculate pulse duration as T_{ON} =1.1 RC and compare it with the Observed experiment value.
5) Choose R_A = R_B = 10 KΩ and C = 0.1 F and set up the circuit.
6) Observe the output waveforms and verify the operation of the circuit.

Circuit Diagram :



Waveform:



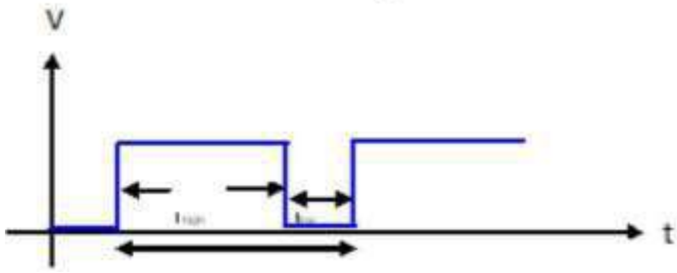


Fig. : Waveform for continuous trigger signal

Results: The experiment was conducted for the design and testing of a monostable multivibrator for the given pulse width W using IC555 & the results are observed.

Applications:

1.
2.
3.
4.

Remarks :

Signature of Staff Incharge with date:

Probable viva questions:

1. Explain the use of IC555 as a monostable multivibrator.
2. Why is an monostable multivibrator called so ?
3. What do you mean by timer ?
4. Mention applications of monostable multivibrator.
5. What type of input is given to a monostable multivibrator ?

References:

1. "Operational Amplifiers and Linear IC's", David A. Bell, 2nd edition, PHI/Pearson, 2004.

Experiment No. : 7	Date : / / .

HALF WAVE AND FULL WAVE PRECISION RECTIFIERS

Aim: To design a half wave and full wave precisions rectifier using operational amplifiers

Apparatus/Components required: Op amp µA 741, diode (in 4001 x 2), resistors (1 KΩ x 4),10 KΩ, Signal Generator, CRO, dual power supply, Wires.

Theory: Rectifier circuits are used in the design of power supply circuits. In such applications, the voltages being rectified are usually much greater than the diode voltage drop, rendering the exact value of the diode drop unimportant to the proper operation of the rectifier. Other applications exist, however, where this is not the case. For example, in instrumentation applications, the signal to be rectified can be of very small amplitude, say 0.1 V, making it impossible to employ the conventional rectifier circuits. Also the need arises for very precise transfer characteristics. The precision rectifier, which is also known as a super diode, is a configuration obtained with an OPAMP in order to have a circuit behaving like an ideal diode and rectifier. It can be useful for high-precision signal processing. The HWR circuit in the figure accepts an incoming waveform and as usual with op amps, inverts it. However, only the positive-going portions of the output waveform, which correspond to the negative-going portions of the input signal, actually reach the output. The direct feedback diode shunts any negative-going output back to the "" input directly, preventing it from being reproduced. The slight voltage drop across the diode itself is blocked from the output by the second diode. The second diode allows positive-going output voltage to reach the output. Furthermore, since the output voltage is taken from beyond the output diode itself, the op amp will necessarily compensate for any non-linear characteristics of the diode itself. As a result, the output voltage is a true and accurate (but inverted) reproduction of the negative portions of the input signal. Thus, this circuit operates as a precision half-wave rectifier. If R_f is equal to R_{in} as is the usual case, the output voltage will have the same amplitude as the input voltage.

Design:

In inverting mode

$$A = \frac{R_f}{R_1} \text{ or } V_0 = -\frac{R_f}{R_1}(V_i)$$

Therefore,

$$\frac{R_f}{R_1} = \text{Slope} = 10$$

$$R_f = 10\ R_1;$$

Let

$$R_1 = 1\ K\Omega\ \ \&\ R_f = 10\ K\Omega$$

For an Half Wave rectifier,

$$R_f = R_1, \text{ i.e., slope} = 1$$

$$V_i = 5\ V\ p\text{-}p$$

$$V_0 = -\frac{R_f}{R_1}(V_i) = -5\ V$$

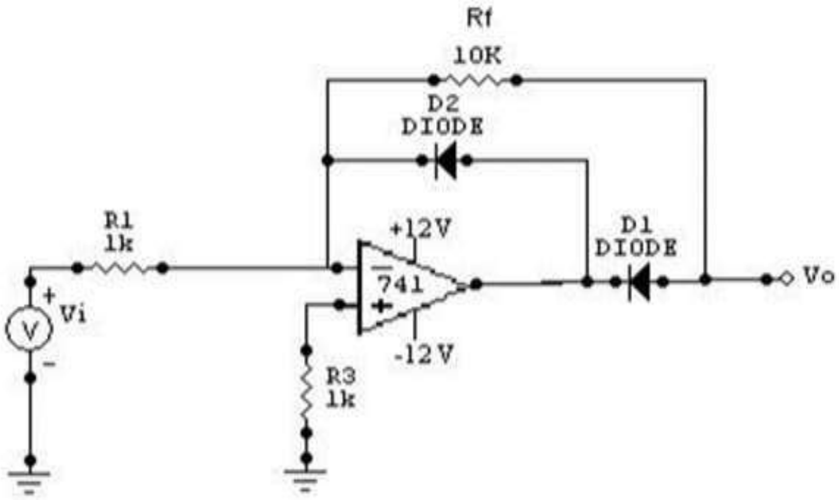
Let,

$$R_f = R_1 = 1\ K\Omega$$

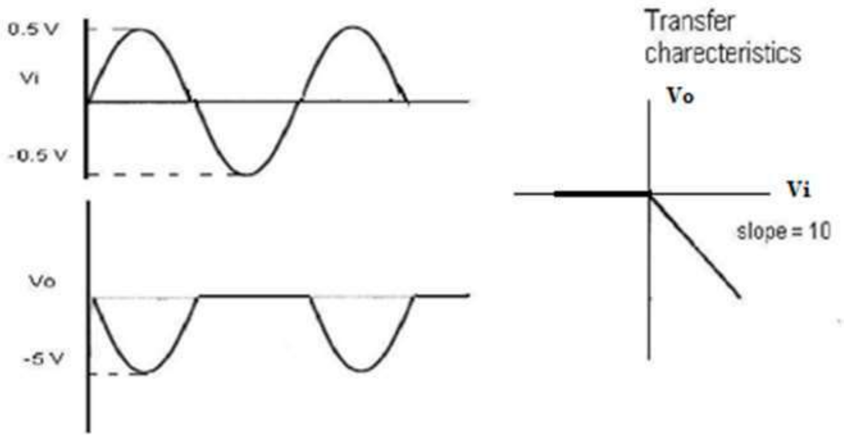
Procedure:

1) Connect the circuit as shown above.
2) Give i/p of 0.5 V peak to peak sinusoidal signal and frequency 1 Khz.
3) Display the o/p waveform on CRO.
4) Press x-y mode on the CRO and observe the transfer characteristics.
5) Connect the circuit for a Full wave rectifier.
6) And repeat the same procedure as above.
7) Observe the o/p waveform and transfer characteristics for FWPR and verify it with the given waveforms above.

Circuit Diagram: Half wave rectifier



Waveforms:



Full wave rectifier :

$$R_1 = R_2 = R_3 = R_4 = R = 1\ K\Omega$$

$$R_5 = \frac{R}{2} = 500\ \Omega$$

Choose

$$R_5 = 470\ \Omega$$

Design

During the positive half cycle,

$$V_0 = -\frac{V_i}{R_1}(R_f)$$

$$V_i = 0.5\ v\ (\text{peak to peak}) = \ = +0.25\ v \text{ to } -0.25\ v$$

Therefore,

$$\frac{V_0}{V_i} = 1 \quad \Rightarrow \quad R_f = R_i$$

Choose

$$R_i = 1\ K\Omega \Rightarrow R_f = 1\ K\Omega$$

o/p of first op-amp is

$$V_S = -V_i\left(\frac{R_2}{R_1}\right)$$

Since

$$R_2 = R_1\ V_s = -V_i$$

$$V_0 = -V_S \times \left(\frac{R_4}{R_2}\right) \times V_0 = V_i$$

During negative half cycle,

KCL equation at point 2 of first op-amp is

$$\frac{V_i}{R_1} = \frac{V}{2R} + \frac{V}{R_5}$$

$$R_1 = R_5 = R$$

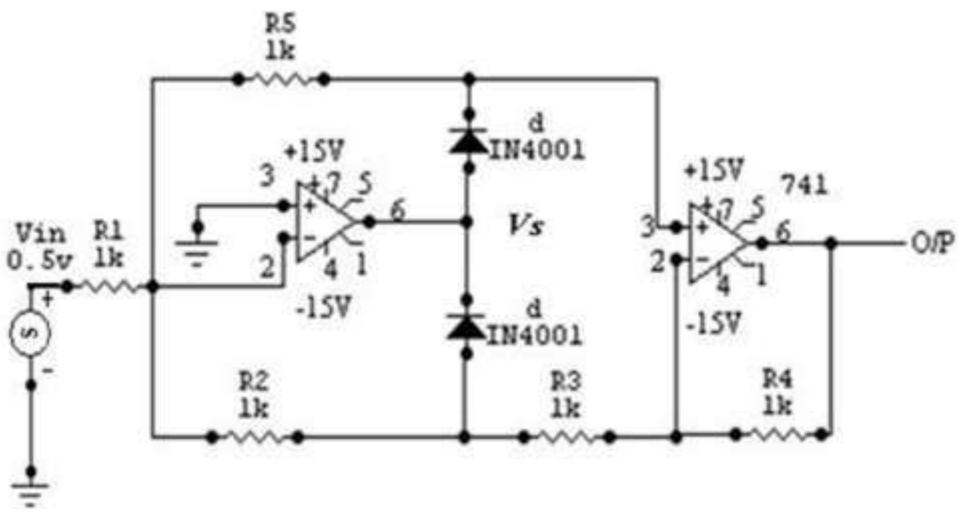
$$V = -\left(\frac{2}{3}\right)V_i$$

$$V_0 = \left[1 + \frac{R_4}{2R}\right] \times \left[-\left(\frac{2}{3}\right) \times V_i\right]$$

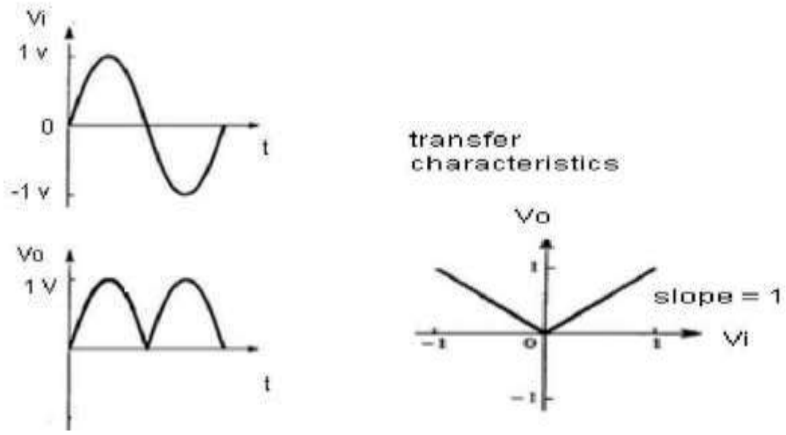
$$R_4 = R$$

$$V_o = V_i$$

Circuit Diagram: Full wave rectifier



Waveforms:



Results: Half wave and full wave rectifiers are constructed using op amp, the experiment was conducted and the input waveform are observed on the CRO along with the transfer characteristics.

Applications:

1.
2.
3.
4.

Remarks:

Signature of Staff Incharge with date:

Probable viva questions:

1. What do you understand by precision rectifiers ?
2. How do they differ from conventional rectifier ?
3. Why use an op-amp as a rectifier ?
4. What are the advantages of Precision rectifier ?
5. How do precision rectifiers achieve rectification below cut-in voltage of diode ?
6. Why diode is used in the feedback loop in the precision rectifier circuit ?

References:

1. "Operational Amplifiers and Linear IC's", David A. Bell, 2nd edition, PHI/Pearson, 2004.

Experiment No. : 8	Date : / / .

PULSE WIDTH MODULATION

Aim: Conduct an experiment to generate a PWM signal for a given analog signal of frequency less than 1 KHz also determine the dynamic range.

Apparatus/Components required: 555 Timer, Resistor, Capacitor, IN4001 diode, Signal generator, Op-amp 741, connecting wires, bread board, etc..

Theory: Pulse width changes with instantaneous value of modulating signal. Pulse width modulator uses a monostable multivibrator with a modulating input signal applied to pin no 5. By the application of continuous trigger signal at pin no 2, a series of output pulses are obtained and the duration of these pulses depends on modulating input at pin no 5. Modulating signal applied at pin no 5 gets superimposed upon the already existing voltage 2/3 V_{cc} at the inverting input terminal of upper comparator. This in turn changes the threshold level of upper comparator and output pulse width modulation takes place. It is observed that pulse duration only varies keeping frequency same as that of continuous input pulse train. PWM signal is used to amplitude modulate the station carrier signal in high-level modulator.

Design

$$\text{Let } T_p = 1.1\ RC$$

$$V_{CC} = 5\ V$$

$$\text{Let } C = 0.1\ \mu F$$

$$T_p = 1.1\ ms$$

$$R = 10\ K\Omega$$

Design of the triggering circuit is shown as below …

$$R_t\ C_t << T$$

$$\text{Let } T = 1\ ms$$

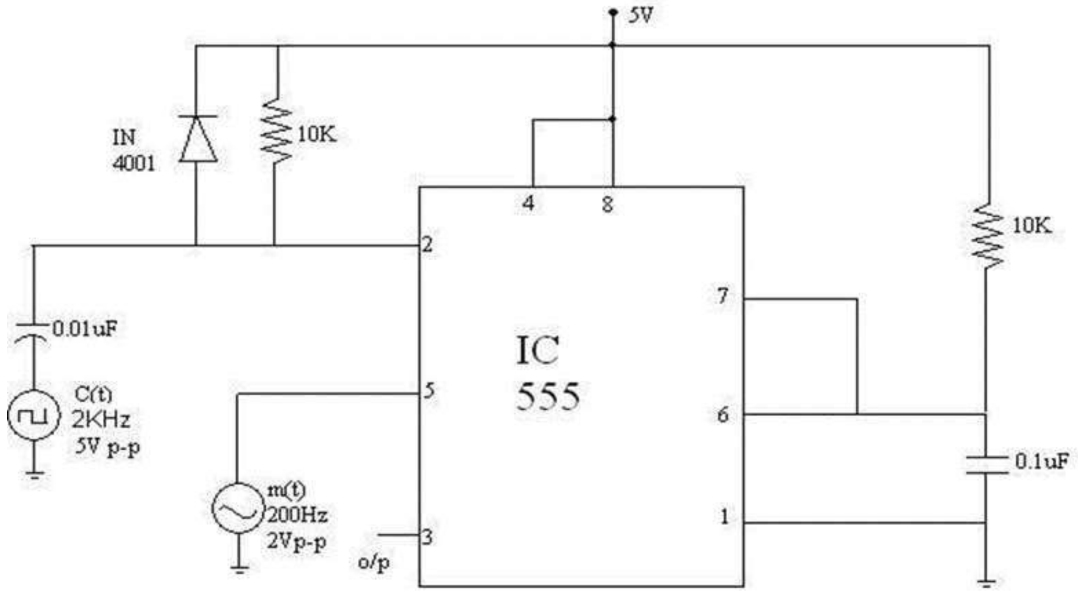
$$R_t\ C_t = 0.1\ T$$

$$\text{If } C_t = 0.01\ \mu F, \text{ then } R_t = 10\ K\Omega$$

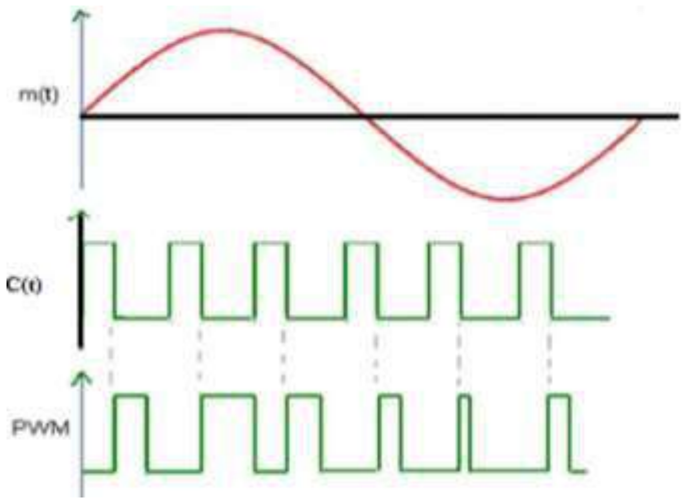
Procedure:

1. Connect the circuit as shown in figure below in the circuit diagram.
2. Adjust the C(t) frequency to around (800 Hz – 2 KHz) and amplitude to 5V (p-p)
3. Adjust the m(t) frequency to around 200 Hz and amplitude to 2 V(p-p)
4. Slightly vary the amplitude of m(t) & frequency to 139 Hz and frequency of C(t) to 850 Hz
5. See the o/p of PWM at pin 3.

Circuit Diagram :



Waveforms :



Results: The experiment was conducted & the PWM signal is generated and output is verified & conclusions are drawn.

Applications:

1.
2.
3.
4.

Remarks:

Signature of Staff Incharge with date:

Probable viva questions:

1. State various Pulse modulation methods.
2. What are the advantages of PWM over PAM ?
3. What are the disadvantages of PWM ?
4. Explain the principle of PWM.
5. Mention the applications of PWM.

References:

1. Communication Systems, Simon Haykins, 5th Edition, John Willey, India Pvt. Ltd, 2009.

Experiment No. : 9	Date : / / .

PULSE POSITION MODULATION

Aim: To conduct an experiment to generate a PPM signal of pulse width for a given modulating signal.

Apparatus/Components required: 555Timer, Resistor, Capacitor, IN4001 diode, signal generator, Op-amp 741, bread board, connecting wires, etc…

Theory: The position of the pulse varies in accordance with amplitude of sampled waveform. PPM can be derived from PWM through process of differentiating. PWM signal is applied to a pin no 2 of IC 555 timer through a diode RC combination. Thus input to pin no 2 is negative trigger pulses, which corresponds to trailing edges of PWM waveform. IC555 timer is working in a monostable mode and pulse width is constant. The negative trigger pulses decide the starting time of output pulses and thus output at pin no 3 is desired pulse position modulation. PPM is used in speed control of DC motors.

Design :

Let

PPM pulse width T_P = 100 sec

T_m = minimum width of PWM signal

$T_P = 1.1\ RC$

Let C = 0.01 µF

$$R = \frac{200 \times 10^{-6}}{1.1 \times 0.01 \times 10^{-6}} = 10\ K\Omega$$

Triggering circuit design is done as ....

To get spikes,

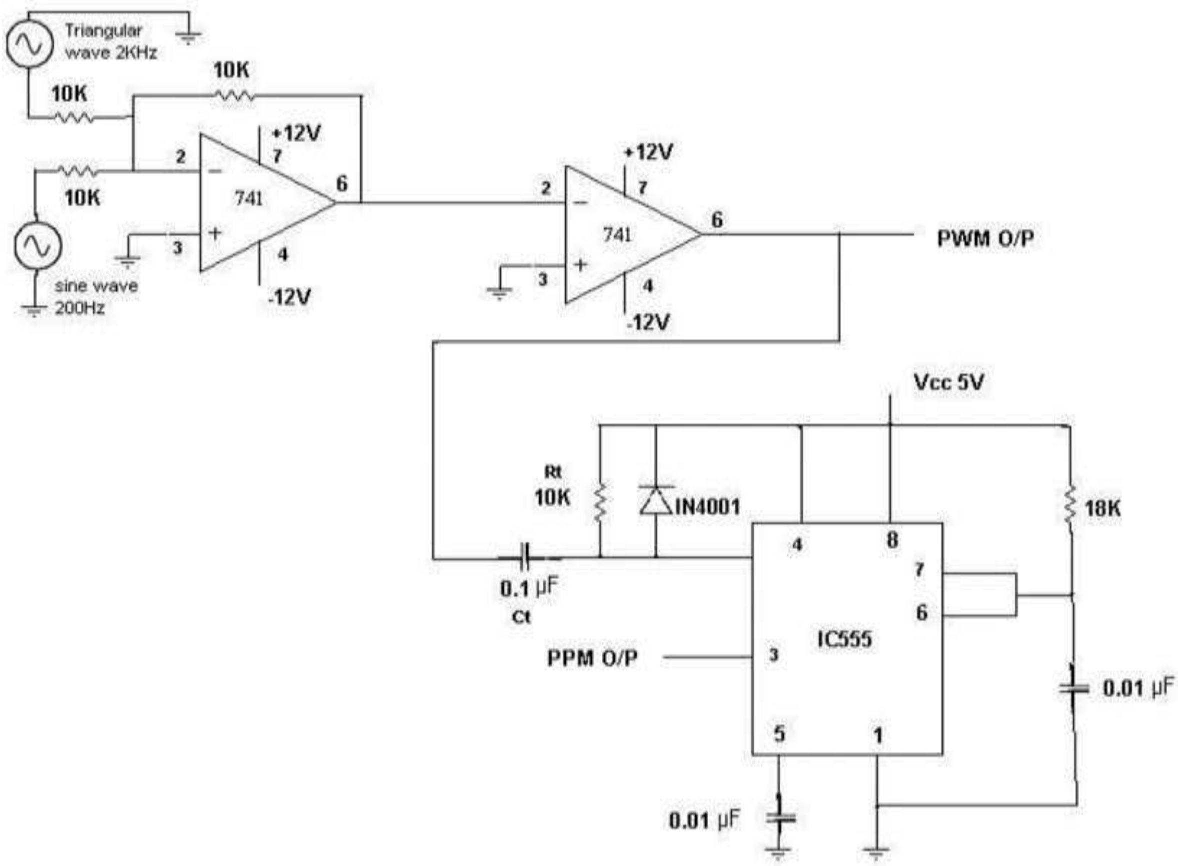
$$R_t C_t << T_m$$

$$\text{Let } R_t\, C_t = 0.1\ T_m$$

$$\text{Let } C_t = 0.1\ \mu F$$

$$R_t = 10\ K\Omega$$

Circuit Diagram :



Procedure:

1. Rig up the circuit as shown in figure.

2. Set the sin wave m(t) around 200 Hz and amplitude of 2 V.

3. Set the triangular wave frequency C(t) around 2 KHz & amplitude of

around 2 V.

4. Now check the PWM o/p at pin 6 of op-amp.
5. Give this o/p as i/p at pin 2 of IC 555.
6. We will get PPM o/p at pin 3 hence the position of pulse width keeps change.

Waveforms :

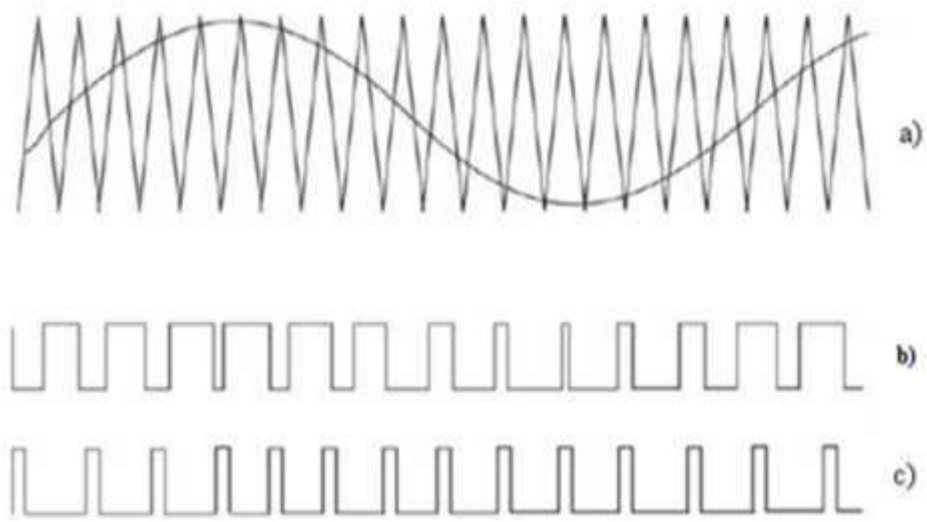


Fig. a) Modulating signal and triangular signal b) PWM c) PPM

Results: PPM signal is generated and o/p is observed.

Applications:

1.
2.
3.
4.

Remarks:

Signature of Staff Incharge with date:

Probable viva questions:

1. What is pulse position modulation ?
2. Explain the principle of PPM ?
3. What is the advantage of PPM over PWM and PAM ?
4. Explain the principle of PPM.
5. What are the applications of PPM ?
6. What are the different circuits that are used in the block diagram to generate a PPM ?

References:

1. Communication Systems, Simon Haykins, 5^{th} Edition, John Willey, India Pvt. Ltd, 2009.

Experiment No. : 10	Date : / / .

FREQUENCY MODULATION USING 8038

Aim: Design and conduct a suitable experiment to generate a FM wave using IC 8038. Find the modulation index and the bandwidth of operation (T) and to display various wave forms.

Apparatus/Components required: IC 8038, resistors –10 KΩ (3 no's), 6.8 KΩ, 8.2 KΩ, Capacitors -1 F, Signal generator, Connecting wires, etc…

Theory: Frequency modulation conveys information over a carrier wave by varying its instantaneous frequency. An important concept in the understanding of FM is that of frequency deviation. The amount of frequency deviation a signal experiences is a measure of the change in transmitter output frequency from the rest frequency of the transmitter. The rest frequency of a transmitter is defined as the output frequency with no modulating signal applied. For a transmitter with linear modulation characteristics, the frequency deviation of the carrier is directly proportional to the amplitude of the applied modulating signal. Thus, an FM transmitter is said to have a modulation sensitivity, represented by a constant, k_f, where

$$k_f = \text{frequency deviation/V} = k_f \ \text{kHz/V.}$$

IC 8038 a waveform generator is used in F.M generation frequently. Modulated signal is given by

$$s(t) = A_c \cos [\omega_c t + \sin \omega_m t]$$

$$m(t) = A_m \cos \omega_m t$$

$$\text{Modulation index} = K_f \frac{A_m}{F_m}$$

where < 0.3 it is narrow band F.M. and > 0.3 it is wide band F.M.

- 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 0.22 V_{cc} and V_{cc} varies between ±5 v to ±15 v.
- This IC also generates square wave and triangular wave.
- Pin 4 and 5 are used for duty cycle.
- A external capacitor connected to pin 10 along with resistors connected to pin 5 and determines output frequency.
- Pin 8 is used for F.M sweep input.
- Modulating signal is applied to pin8 sweep voltage should be within the range 2/3 V_{cc} < V_{sweep} < V_{cc}, where V_{cc} is total voltage.
- Sweep frequency of F.M. is 10 khz.
- An external capacitor connected to timing capacitor with timing resistors connected to 4 & 5 determines the frequency of output waveforms.

Design :

Given frequency f = 25 KHz

$$f = \frac{1}{\frac{5}{3} R_A C \left[1 + \frac{R_B}{2R_A - R_B} \right]}$$

$R_A = R_B = R$

$$f = \frac{0.3}{RC}$$

Let R = 10 KΩ & C = 0.001 μF

Procedure:

1) Give ± 10 V as V_{cc} to IC 8038 and shorting pin no's 7 and 8 observe output at 9, 3, 2 on CRO.
2) Measure sine wave amplitude and frequency.
3) Connect modulating signal of V_m = 5 v(p-p) and frequency of 800 Hz to 1.5 kHZ. Between pin 7 & 8, through R-C as shown in the figure.
4) Observe FM output at pin 2.

5) Draw output waveform.

6) Determine maximum phase deviations following steps are carried out.

   a) Shot 7 and 8, without RC connects a variable DC supply.

   b) Vary DC voltage from (0-6 v) and observe frequency variation at pin 2.

   c) Note down DC input and output frequency on CRO.

   d) Draw graph of frequency v/s input voltage.

Circuit Diagram :

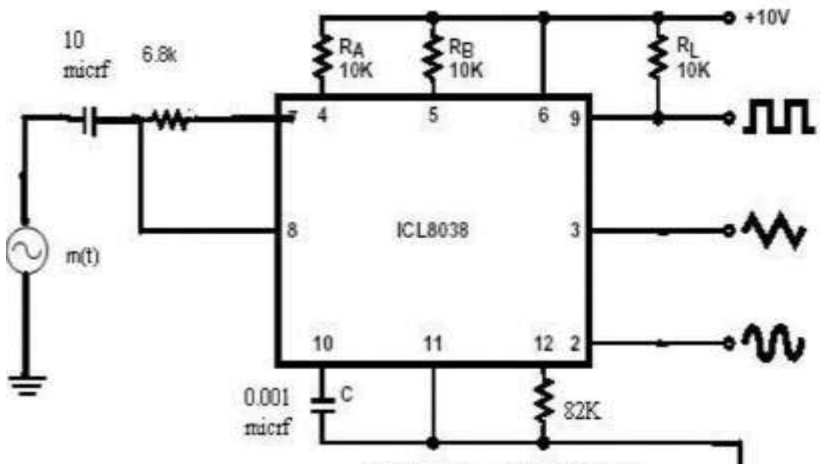
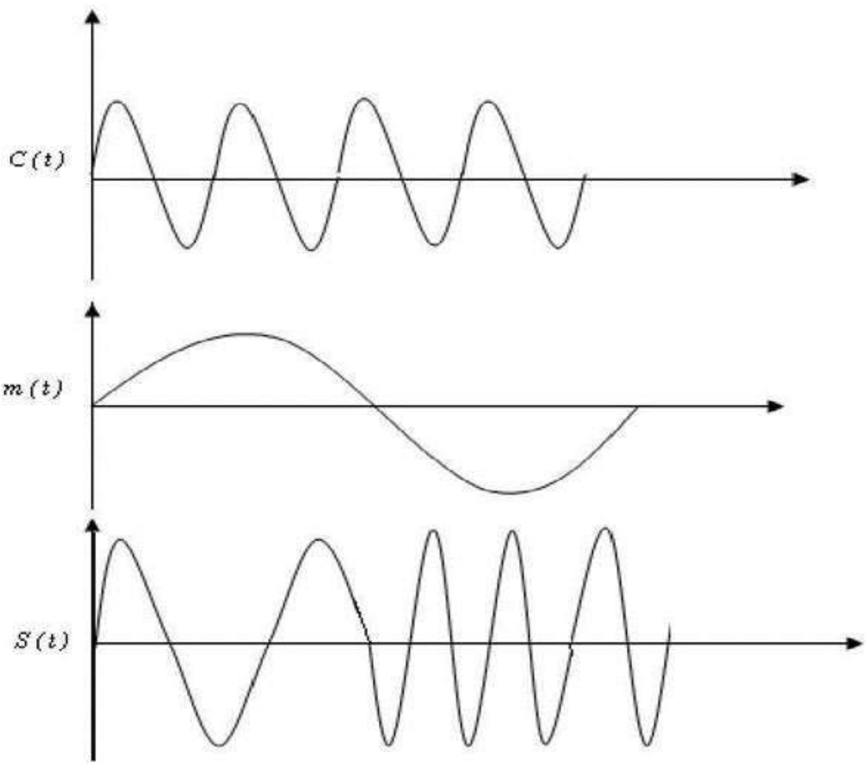
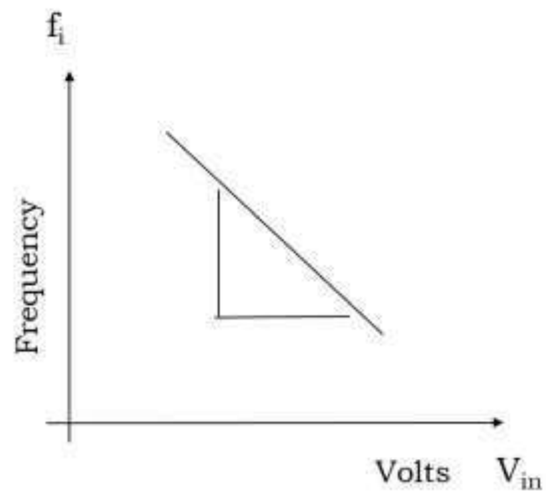


FIGURE 1. TEST CIRCUIT

Waveforms:





Sensitivity-(from the graph) Slope S = $\frac{\Delta f_i}{\Delta V_i}$

Maximum deviation: δ = S × V_m = $\frac{\delta}{f_m}$

Tabular Column:

DC input volts (V_i)	FM output (f)

For carrier sine wave: Amplitude = 2.2 v

Results: The modulation index is =

Applications:

1.
2.
3.
4.

Remarks :

Signature of Staff Incharge with date:

Probable viva questions:

1. Why frequency modulation is better than amplitude modulation ?
2. What is frequency deviation in FM ?
3. What is transmission BW for FM ?
4. When maximum frequency deviation occurs in FM ?
5. List some advantages of FM over AM.
6. What is the carson's rule for bandwidth of FM wave ?
7. What are the standard intermediate frequency values of AM radio and FM radio ?

References:

1. An Introduction to Analog and Digital Communication, Simon Haykins, John Wiley India Pvt. Ltd., 2008.

Experiment No. : 11	Date : / / .

SCHMITT TRIGGER

Aim: To design op-amp Schmitt trigger for given specification

Apparatus/Components required: Op-amp IC 741, resistor [1K, 15K], power supply [DC], signal generator, CRO, connecting wires.

Theory: A Schmitt trigger is an inverting comparator with a positive feedback. The applied voltage V_{in} triggers (changes the state of the o/p) the o/p V_{out} every time the i/p voltage exceeds a certain voltage level called the upper threshold voltage [UTP]. The applied voltage V_{in} also changes the state of the o/p, i.e., triggering the o/p when the value of V_{in} falls below a certain reference voltage called lower threshold voltage [LTP]. The values of the upper threshold voltage [UTP] and the lower threshold voltage [LTP] are obtained by using the voltage divider circuit R_1 & R_2. When the i/p voltage is above the upper threshold, the o/p voltage is constant at the lower saturation point ($-V_{sat}$) this stage the voltage across the resistor R_2 is V_{Lt} = lower threshold voltage. When V_{in} is less than the lower threshold, the o/p is constant at the upper saturation point V_{sat}. At this stage, the voltage across the resistor R_2 is V_{ut} = upper threshold voltage. The width of hysterics curve = $V_{utp} - V_{Lltp}$

Design :

Design a Schmitt trigger for UTP = 2.5 V and LTP = 1.0 V, V_{sat} = 12 V

$$UTP + LTP = 2\ V_{ref}\left[\frac{R_1}{(R_1+R_2)}\right] \quad \ldots\ldots\ldots\ldots\ldots\ldots (1)$$

$$UTP - LTP = 2\ V_{sat}\left[\frac{R_2}{(R_1+R_2)}\right] \quad \ldots\ldots\ldots\ldots\ldots\ldots (2)$$

$$\text{From eqn. (2) : } 2\left[\frac{R_2}{(R_1+R_2)}\right] = \frac{1.5}{12}$$

$$\left[\frac{R_2}{(R_1 + R_2)}\right] = 0.0625$$

$$R_1 + R_2 = 16\ R_2$$

$$R_1 = 15\ R_2$$

Choose $R_2 = 1\ K\Omega$, then $R_1 = 15\ K\Omega$

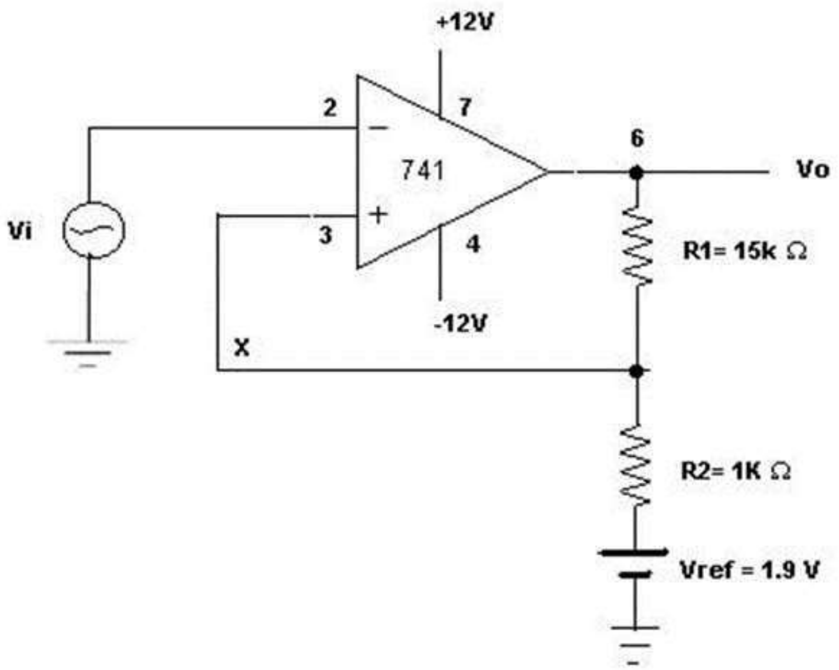
Form eqn. (1) : $3.5 = 2\ V_{ref}\frac{15R_2}{16R_2}$

Therefore, $V_{ref} = 1.87\ V \approx 1.9\ V$

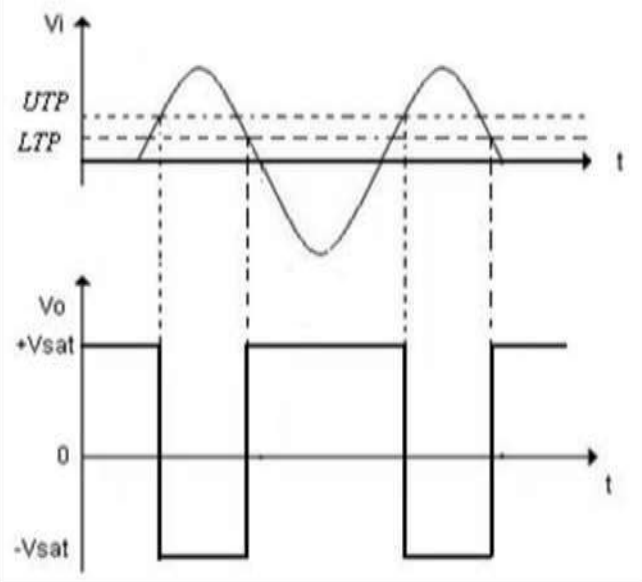
Procedure:

1) Rig up the circuit as per the circuit diagram**.**
2) Use a sinusoidal signal of 500Hz as i/p and amplitude of 6V peak to peak.
3) Display o/p rectangular wave on CRO and measure UTP and LTP.
4) Use X and Y mode and display the hysteresis curve on CRO, measure UTP and LTP and compare it with the designed values.

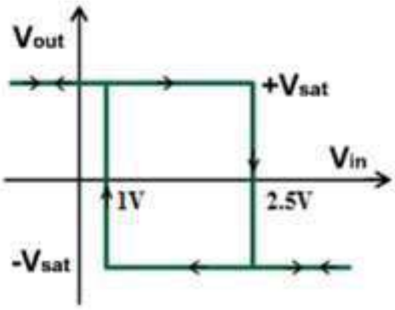
Circuit Diagram :



Waveforms :



Transfer characteristics :



Here, V_i is linear varying signal ; say a sinusoidal or triangular waveform,

Working : say initially $V_i < 0$ then $V_0 = + V_{sat}$.

Let us find voltage at node X.

$$V_X = V_{sat} \frac{R_2}{R_1 + R_2} + V_{ref} \frac{R_1}{R_1 + R_2}$$

Until V_i reaches V_x , the o/p cannot change its state.

This value of i/p which changes the o/p form + V_{sat} to $-V_{sat}$ is called upper trip point (UTP)

Therefore,

$$UTP = \frac{R_2}{R_1 + R_2}(V_{sat}) + \frac{R_1}{R_1 + R_2}(V_{ref})$$

To regain original state the V_i must decrease from value greater than UTP.

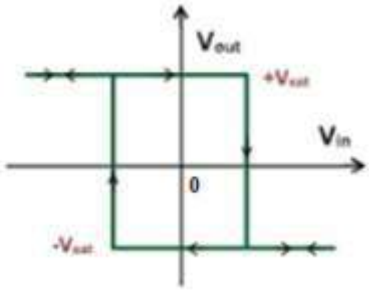
i.e., $V_o = V_{sat}$

Then,

$$V_X = \frac{R_2}{R_1 + R_2}(-V_{sat}) + \frac{R_1}{R_1 + R_2}(V_{ref})$$

Therefore,

$$LTP = \frac{R_2}{R_1 + R_2}(-V_{sat}) + \frac{R_1}{R_1 + R_2}(V_{ref})$$



when V_{ref} = 0 V

UTP = LTP

Results: The experiment was conducted, the results was observed.

Applications:

1.
2.
3.
4.

Remarks:

Signature of Staff Incharge with date:

Probable viva questions:

1. Explain op-amp as a Schmitt trigger.
2. What are the applications of Schmitt trigger.
3. Why is a Schmitt trigger called a regenerative comparator.
4. What type of feedback is used in Schmitt trigger.
5. Which circuit converts irregularly shaped waveform to regular shaped waveforms ?
6. In which configuration a dead band condition occurs in Schmitt trigger ?

References:

1. "Operational Amplifiers and Linear IC's", David A. Bell, 2nd edition, PHI/Pearson, 2004.

AC+LIC LABORATORY (ECL47)

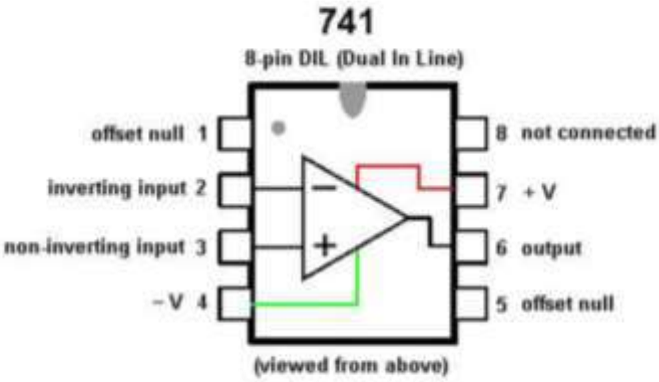
PROBABLE/SUGGESTED QUESTION BANK

1. Design a second order active Low Pass Filter for a given cutoff frequency f_c = ____ Hz (< 10 KHz). Conduct an experiment to draw the frequency response.
2. Design a second order active High Pass Filter for a given cut-off frequency f_c = ____ Hz (< 10 KHz). Conduct an experiment to draw the frequency response.
3. Design a Band Pass Filter for f_c = ____ Hz & draw the frequency response.
4. Design a Band Elimination Filter for f_c = ____ Hz & draw the frequency response.
5. Design an Op-Amp Schmitt trigger for UTP = ____ and LTP = ____
6. Design a symmetrical astable multivibrator using IC555.
7. Design a monostable multivibrator for the ____ pulse width.
8. Conduct an experiment to generate an AM signal using collector modulation for an F_c = _____ KHz and F_m = ____ Hz. Plot the variations of modulating signal amplitude v/s modulation index. Also demodulate it.
9. Design and conduct a suitable experiment to generate an FM wave using IC 8038. Find the modulation index and bandwidth. Display the various waveforms.
10. Conduct an experiment to generate PAM signal and also design a circuit to demodulate the obtained PAM signal. Plot the relevant waveforms.
11. Conduct an expt to generate a PPM signal for a given modulating signal.
12. Conduct an experiment to generate a PWM signal, also determine the dynamic range.
13. Design and test R-2R DAC using Op-Amp.
14. Design and test Half wave and Full wave precision rectifier circuits using Op-Amp.
15. Design a PWM circuit using summing amplifier and ZCD.

Appendix

LM 741 OPAMP

One of the cheapest and most-popular OP-AMP is the 741. Operational amplifiers can be used to perform mathematical operations on voltage signals such as inversion, addition, subtraction, integration, differentiation, and multiplication by a constant. The pinout for an LM741 is shown below:



The basic parameters for a 741 are :

Rail voltages : +/– 15v DC (+/– 5v min, +/– 18v max)

Input impedance: approx 2M

Low Frequency voltage gain: approx 2,00,000

Input bias current: 80 nA

Slew rate: 0.5 v per microsecond

Maximum output current: 20 mA

Recommended output load: not less than 2 kΩ

555 Timer

The 555 timer is an integrated circuit (chip) implementing a variety of timer and multivibrator applications. It was produced by Signetics Corporation in early 1970. The original name was the SE555/NE555 and was called "*The IC Time Machine*". The 555 gets its name from the three 5-KΩ resistors used in typical early implementations. It is widely used because of its ease to use, low price and reliability. It is one of the most popular and versatile integrated circuits which can be used to build lots of different circuits.

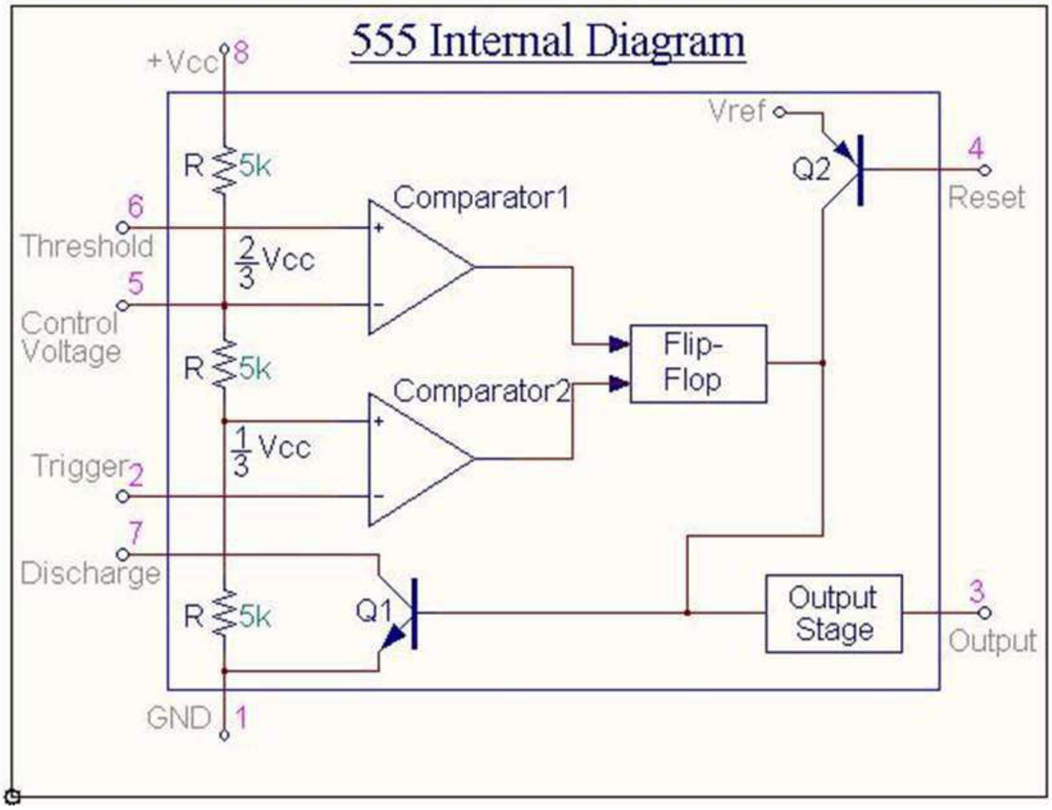
It includes 23 transistors, 2 diodes and 16 resistors on a silicon chip installed in an 8-pin mini dual-in-line package (DIP-8). The 555 Timer is a monolithic timing circuit that can produce accurate and highly stable time delays or oscillations. The timer basically operates in one of the two modes—monostable (one-shot) multivibrator or as an astable (free-running) multivibrator. In the monostable mode, it can produce accurate time delays from microseconds to hours. In the astable mode, it can produce rectangular waves with a variable duty cycle. Frequently, the 555 is used in astable mode to generate a continuous series of pulses, but you can also use the 555 to make a one-shot or monostable circuit.

The 555 can source or sink 200 mA of output current, and is capable of driving wide range of output devices. The output can drive TTL (Transistor-Transistor Logic) and has a temperature stability of 50 parts per million (ppm) per degree Celsius change in temperature, or equivalently 0.005 %/°C.

Applications of 555 timer in monostable mode include timers, missing pulse detection, bounce free switches, touch switches, frequency divider, capacitance measurement, pulse width modulation (PWM) etc.

In astable or free running mode, the 555 can operate as an oscillator. The uses include LED and lamp flashers, logic clocks, security alarms, pulse generation, tone generation, pulse position modulation, etc. In the bistable mode, the 555 can operate as a flip-flop and is used to make bounce-free latched switches, etc.

Refer to the figure below for the brief description of the pin connections. The pin numbers used refer to the 8-pin mini DIP and 8-pin metal can packages. The 555 can be used with a supply voltage (V_{CC}) in the range 4.5 to 15 V (18 V absolute maximum).



Pin 1: Ground. All voltages are measured with respect to this terminal.

Pin 2: Trigger. The output of the timer depends on the amplitude of the external trigger pulse applied to this pin. The output is low if the voltage at this pin is greater than 2/3 V_{cc}. When a negative going pulse of amplitude greater than 1/3 V_{CC} is applied to this pin, comparator 2 output goes low, which in turn switches the output of the timer high. The output remains high as long as the trigger terminal is held at a low voltage.

Pin 3: Output. There are two ways by which a load can be connected to the output terminal: either between pin 3 and ground or between pin3 and supply voltage $+V_{CC}$. When the output is low the load current flows through the load connected between pin 3 and $+V_{CC}$ into the output terminal and is called sink current. The current through the grounded load is zero when the output is low. For this reason the load connected between pin 3 and $+V_{CC}$ is called the *normally on load* and that connected between pin 3 and ground is called *normally off-load*. On the other hand, when the output is high the current through the load connected between pin 3 and $+V_{CC}$ is zero. The output terminal supplies current to the normally off load. This current is called source current. The maximum value of sink or source current is 200mA.

Pin 4: *Reset*. The 555 timer can be reset (*disabled*) by applying a negative pulse to this pin. When the reset function is not in use, the reset terminal should be connected to $+V_{CC}$ to avoid any possibility of false triggering.

Pin 5: *Control Voltage*. An external voltage applied to this terminal changes the threshold as well as trigger voltage. Thus by imposing a voltage on this pin or by connecting a *pot* between this pin and ground, the pulse width of the output waveform can be varied. When not used, the control pin should be bypassed to ground with a 0.01 μF Capacitor to prevent any noise problems.

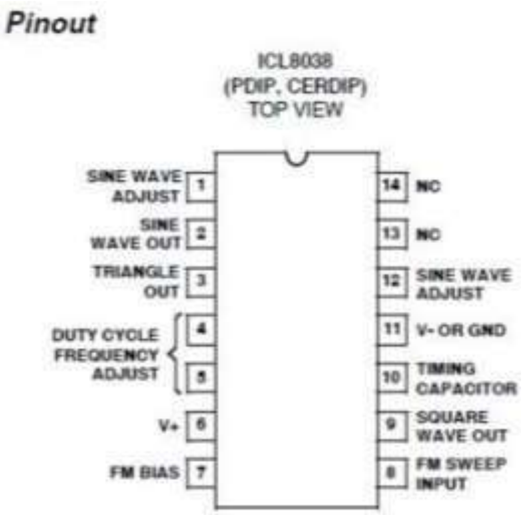
Pin 6: *Threshold*. This is the non-inverting input of comparator 1, which monitors the voltage across the external capacitor. When the voltage at this pin is greater than or equal to the threshold voltage 2/3 V_{CC}, the output of comparator 1 goes high, which in turn switches the output of the timer low.

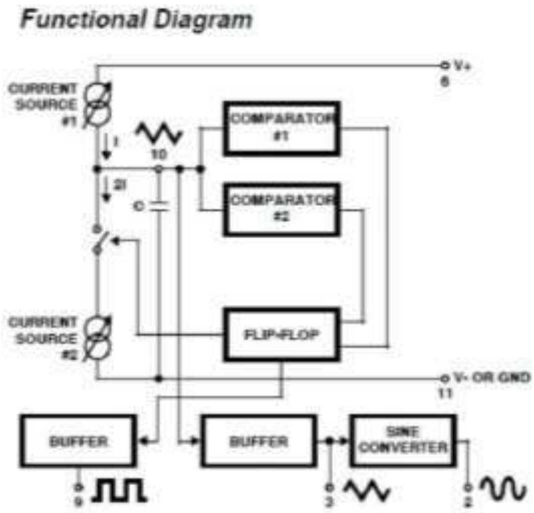
Pin 7: *Discharge*. This pin is connected internally to the collector of transistor Q1. When the output is high Q1 is OFF and acts as an open circuit to external capacitor C connected across it. On the other hand, when the output is low, Q1 is saturated and acts as a short circuit, shorting out the external capacitor C to ground.

Pin 8: $+V_{CC}$. The supply voltage of +5V to + 18V is applied to this pin with respect to ground.

ICL 8038

The ICL8038 waveform generator is a monolithic integrated circuit capable of producing high accuracy sine, square, triangular, saw tooth and pulse waveforms with a minimum of external components. The frequency (or repetition rate) can be selected externally from 0.001Hz to more than 300 kHz using either resistors or capacitors, and frequency modulation and sweeping can be accomplished with an external voltage. The ICL8038 is fabricated with advanced monolithic technology, using Schottky barrier diodes and thin film resistors, and the output is stable over a wide range of temperature and supply variations. These devices may be interfaced with phase locked loop circuitry to reduce temperature drift to less than 250 ppm/^{O}C.





Vision of the Institute

- To impart quality technical education with a focus on Research and Innovation emphasizing on Development of Sustainable and Inclusive Technology for the benefit of society.

Mission of the Institute

- To provide an environment that enhances creativity and Innovation in pursuit of Excellence.
- To nurture teamwork in order to transform individuals as responsible leaders and entrepreneurs.
- To train the students to the changing technical scenario and make them to understand the importance of sustainable and inclusive technologies.

Vision of the ECE Department

- To achieve continuous improvement in quality technical education for global competence with focus on industry, societal needs, research and professional success.

Mission of the ECE Department

- Offering quality education in Electronics and Communication Engineering with effective teaching learning process in multidisciplinary environment.
- Training the students to take-up projects in emerging technologies and work with team spirit.
- To imbibe professional ethics, development of skills and research culture for better placement opportunities.

DEPARTMENT
OF
ELECTRONICS & COMMUNICATION ENGINEERING