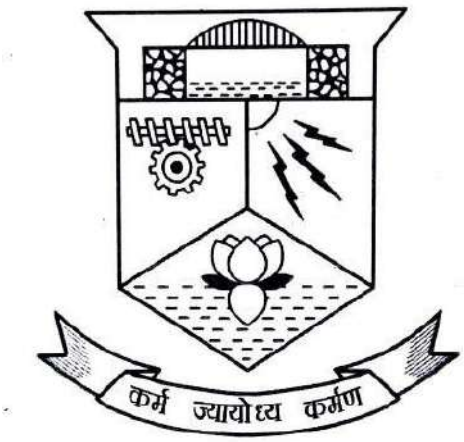
ANALOG INTEGRATED CIRCUITS AND SIMULATION LAB -ECL 331



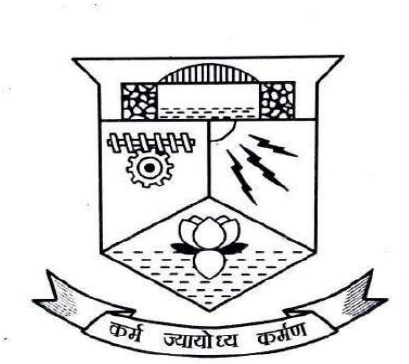
DEPARTMENT OF ELECTRONICS & COMMUNICATION ENGINEERING

COLLEGE OF ENGINEERING, TRIVANDRUM

2019

DEPARTMENT OF ELECTRONICS & COMMUNICATION ENGINEERING

COLLEGE OF ENGINEERING, TRIVANDRUM



CERTIFICATE

This is a controlled document of ***Department of Electronics & Communication*** *of* ***College of Engineering, Trivandrum****. No part of this can be reproduced in any form by any means without the prior written permission of the Head of the Department, Electronics & Communication, College of Engineering, Trivandrum. This is prepared as per* ***2019 KTU B.Tech*** *Electronics & Communication scheme.*

Head of Dept. Dept. of ECE
College of Engineering Trivandrum

ELECTRONICS & COMMUNICATION

SYLLABUS

ECL331	ANALOG INTEGRATED CIRCUITS AND SIMULATION LAB	CATEGORY	L	T	P	CREDIT
		PCC	0	0	3	2

Preamble: This course aims to (i) familiarize students with the Analog Integrated Circuits and Design and implementation of application circuits using basic Analog Integrated Circuits (ii) familiarize students with simulation of basic Analog Integrated Circuits.

Prerequisite: ECL202 Analog Circuits and Simulation Lab

Course Outcomes: After the completion of the course the student will be able to

CO 1	Use data sheets of basic Analog Integrated Circuits and design and implement application circuits using Analog ICs.
CO 2	Design and simulate the application circuits with Analog Integrated Circuits using simulation tools.
CO 3	Function effectively as an individual and in a team to accomplish the given task.

Mapping of course outcomes with program outcomes

	PO1	PO 2	PO3	PO 4	PO5	PO 6	PO7	PO8	PO9	PO 10	PO 11	PO 12
CO1	3	3	3						2			2
CO2	3	3	3	2	3				2			2
CO3	2	2	2		2				3	2		3

Assessment

Mark distribution

Total Marks	CIE	ESE	ESE Duration
150	75	75	3 hours

Continuous Evaluation Pattern

Attendance : 15 marks
Continuous Assessment : 30 marks
Internal Test (Immediately before the second series test) : 30 marks

End Semester Examination Pattern: The following guidelines should be followed regarding awardof marks

(a) Preliminary work : 15 Marks

(b) Implementing the work/Conducting the experiment : 10 Marks

(c) Performance, result and inference (usage of equipments and trouble shooting): 25 Marks

(d) Viva voce : 20 marks

(e) Record : 5 Marks

General instructions: End-semester practical examination is to be conducted immediately after the second series test covering entire syllabus given below. Evaluation is to be conducted under the equal responsibility of both the internal and external examiners. The number of candidates evaluated per day should not exceed 20. Students shall be allowed for the examination only on submitting the duly certified record. The external examiner shall endorse the record.

Course Level Assessment Questions (Examples only)

Course Outcome 1 (CO1): Use data sheets of basic Analog Integrated Circuits and design and implement application circuits using Analog ICs.

1. Measure important opamp parameters of µA 741 and compare them with the data provided in the data sheet
2. Design and implement a variable timer circuit using opamp
3. Design and implement a filter circuit to eliminate 50 Hz power line noise.

Course Outcome 2 and 3 (CO2 and CO3): Design and simulate the application circuits with Analog Integrated Circuits using simulation tools.

1. Design a precission rectifier circuit using opamps and simulste it using SPICE
2. Design and simulate a counter ramp ADC

List of Experiments

I. Fundamentals of operational amplifiers and basic circuits [Minimum seven experiments are to be done]

1. Familiarization of Operational amplifiers - Inverting and Non inverting amplifiers, frequency response, Adder, Integrator, Comparators.

2. Measurement of Op-Amp parameters.

3. Difference Amplifier and Instrumentation amplifier.

4. Schmitt trigger circuit using Op–Amps.

5. Astable and Monostable multivibrator using Op-Amps.

6. Waveform generators using Op-Amps - Triangular and saw tooth

7. Wien bridge oscillator using Op-Amp - without & with amplitude stabilization.

8. RC Phase shift Oscillator.
9. Active second order filters using Op-Amp (LPF, HPF, BPF and BSF).
10. Notch filters to eliminate the 50Hz power line frequency.
11. Precision rectifiers using Op-Amp.

II. Application circuits of **555 Timer/565 PLL/ Regulator(IC 723) ICs** [Minimum three experiments are to be done]

1. Astable and Monostable multivibrator using Timer IC NE555
2. DC power supply using IC 723: Low voltage and high voltage configurations, Short circuit and Fold-back protection.
3. A/D converters- counter ramp and flash type.
4. D/A Converters - R-2R ladder circuit
5. Study of PLL IC: free running frequency lock range capture range

III. Simulation experiments [The experiments shall be conducted using SPICE]

1. Simulation of any three circuits from Experiments 3, 5, 6, 7, 8, 9, 10 and 11 of section I
2. Simulation of Experiments 3 or 4 from section II

Textbooks

1. D. Roy Choudhary, Shail B Jain, “Linear Integrated Circuits,”
2. M. H. Rashid, “Introduction to Pspice Using Orcad for Circuits and Electronics”, Prentice Hall

ELECTRONICS & COMMUNICATION

INDEX

Exp. No	Date	Name of the Experiment	Page No.	Marks	Signature
1		Familiarization of Operational amplifiers - Inverting and Non inverting amplifiers, frequency response, Adder, Integrator, comparators.			
2		Measurement of Op-Amp parameters.			
3		Difference Amplifier and Instrumentation amplifier.			
4		Schmitt trigger circuit using Op –Amps.			
5		Astable and Monostable multivibrator using Op - Amps.			
6		Timer IC NE555			
7		Triangular and square wave generators using Op-Amps.			
8		Wien bridge oscillator using Op-Amp - without & with amplitude stabilization.			
9		RC Phase shift Oscillator			
10		Precision rectifiers using Op-Amp.			
11		Active second order filters using Op-Amp (LPF, HPF, BPF and BSF).			
12		Notch filters to eliminate the 50Hz power line frequency			
13		IC Voltage Regulators			
14		A/D converters- counter ramp and flash type.			
15		D/A Converters- ladder circuit.			
16		Simulation Experiments:			

EXP 1 **DATE**

Familiarization of operational amplifiers - inverting and non inverting amplifiers, frequency response, adder, integrator, comparators.

Aim: To familiarize with basic operational amplifier integrated circuits.

Components required: Op-amp, resistors, capacitors, breadboard, CRO, function generator and power supplies.

Theory: Operational amplifier, in short, op-amp is a versatile device used to amplify AC and DC signals. Though it was originally designed for computing mathematical operations such as addition, multiplication, differentiation, integration etc., it is widely used for variety of applications like oscillators, filters, regulators, clipping circuits, waveform generators etc.

The symbol of op-amp represents a circuit with two input terminals an output terminal and two bias supply points.

The **741 IC:** It is a frequency compensated and short circuit protected IC.741C is its commercial version with operating temperature ranges from 0^0C to $+70^0$C. 741 needs positive and negative dc sources for bias supply connections V^+ and V^-. This is provided by either a dual power supply or two power supplies. When dual power supply is used its positive terminal is connected to the V^+ pin of the IC and the negative terminal is connected to the V^- pin of the IC. The ground terminal of the dual power supply is connected to the ground point of the circuit. When two power supplies are used positive terminal of one supply and negative terminal of the other power supply are connected to the V^+ and V^- pins of the IC respectively.

Inverting Amplifier: This is one of the most popular op-amp circuits. The polarity of the input voltage gets inverted at the output. If a sine wave is fed to the input of this amplifier, the output will be an amplified sine wave with 180^0 phase shift. The gain of the inverting amplifier is given by $A= -R_f/R_i$. where R_f is the feedback resistance and R_i is the input resistance.

Noninverting Amplifier: This circuit provides a gain to the input signal without any change in polarity. The gain of noninverting amplifier is given by $A=1+R_f/R_i$. Input impedance is extremely large.

Adder: This circuit gives the sum of two input voltages. Here an input voltage V_i and a dc voltage V_{ref} are given as inputs to the adder. This is an inverting summing amplifier because the out is the sum of inputs with a sign change. The minus sign in the expression for the output can be avoided if necessary, by inverting the output once again using a unity gain inverting amplifier. The output can be scaled by selecting the ratio R_f/R_i. If this ratio is greater than 1 the circuit will function as a summing amplifier.

Integrator: In an integrator circuit, the output voltage is integral of the input signal. The output voltage of an integrator is given by

$$V_o = -1/R_1C_f \int_o^t V_i\, dt$$

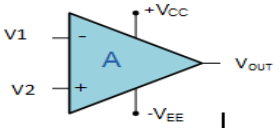
At low frequencies the gain becomes infinite, so the capacitor is fully charged and behaves like an open circuit. The gain of an integrator at low frequency can be limited by connecting a resistor in shunt with capacitor.

Comparator: This circuit compares one analogue voltage level with another analogue voltage level, or some preset reference voltage, V_{REF} and produces an output signal based on this voltage comparison. The op-amp voltage comparator compares the magnitudes of two voltage inputs and determines which is the largest of the two.

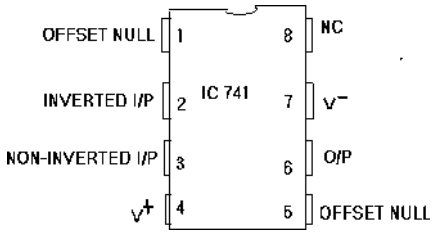
The output saturation voltages are about 2V below the magnitudes of the dc power supply levels. For supply voltages of ($^{+}$-15V),V_{sat} will be approximately ($^{+}$-13V).

Sr. No.	Characteristics	Value for IC 741	Ideal value
1	Input resistance Ri	2 MΩ	∞
2	Output resistance Ro	75 Ω	0
3	Voltage gain Av	2 X 10^5	∞
4	Bandwidth BW	1 MHz	∞
5	CMRR	90 dB	∞
6	Slew rate S	0.5 V/μS	∞
7	Input offset voltage	2 mV	0
8	PSRR	150 μV/V	0
9	Input bias current	50 nA	0
10	Input offset current	6 nA	0

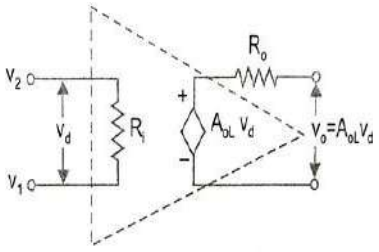
Circuit diagram and waveforms:



op-amp symbol

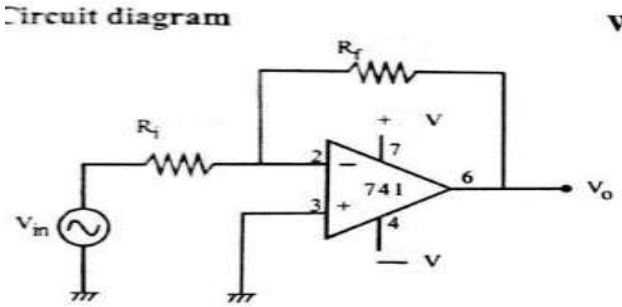


op-amp pin out



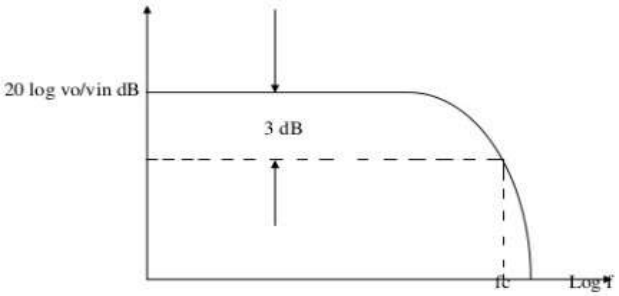
equivalent circuit

Inverting amplifier



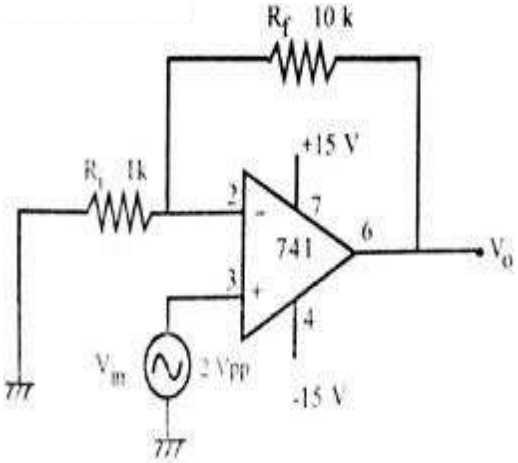
Tabular Column & Frequency Response

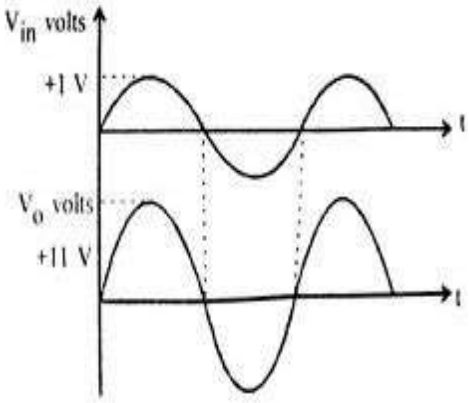
F(Hz)	V_0 (volts)	log f	$20\log(V_0/V_{in})$db



Non inverting amplifier

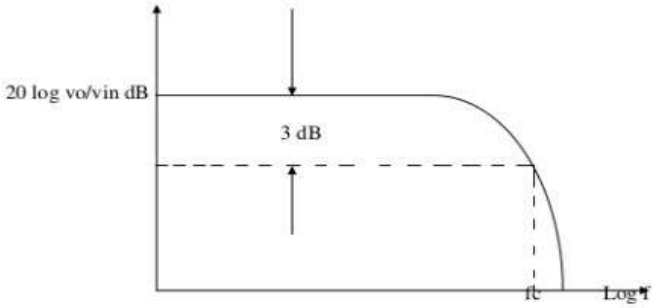
Circuit Diagram & Waveform





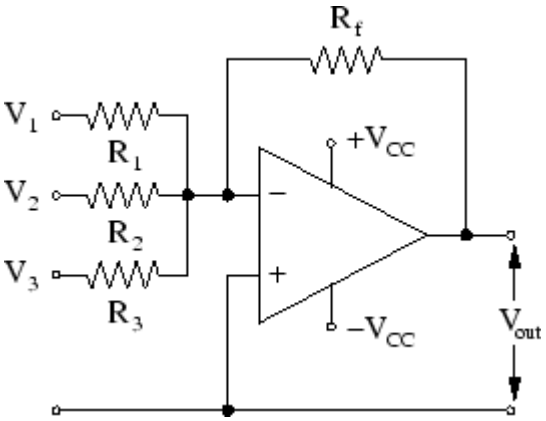
Tabular Column & Frequency Response

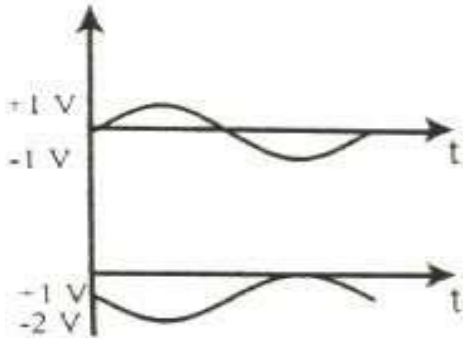
F(Hz)	V_0 (volts)	log f	$20\log(V_0/V_{in})$db



Adder

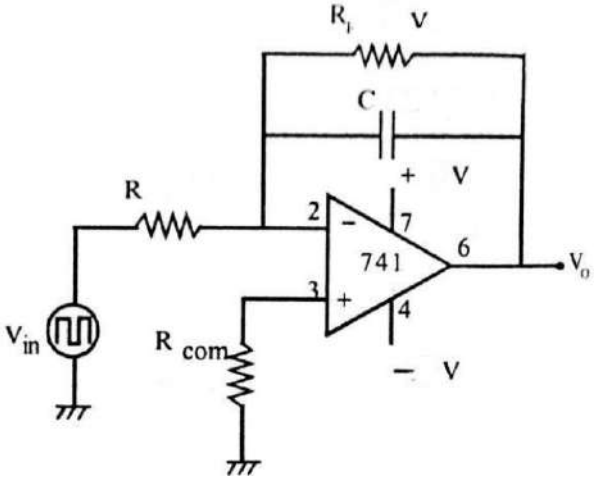
Circuit Diagram & Waveforms

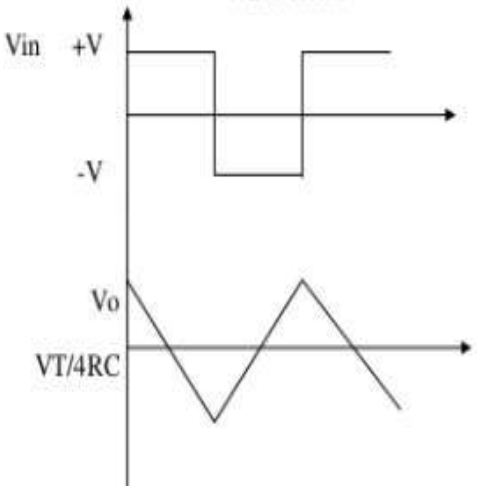




Integrator

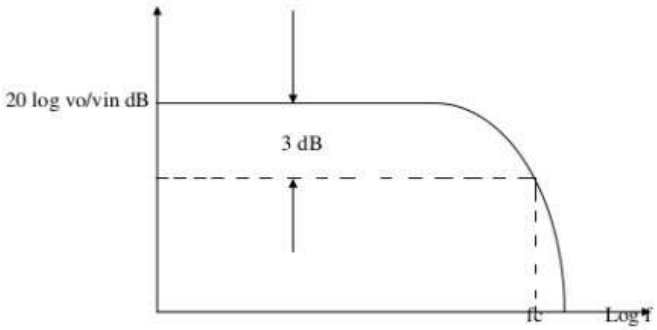
Circuit Diagram & Waveforms





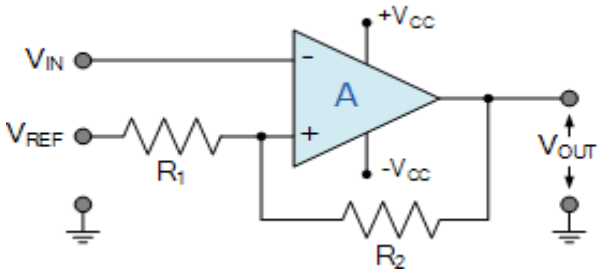
Tabular Column & Frequency Response

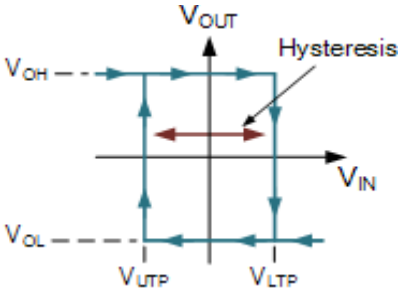
F(Hz)	V_0 (volts)	log f	20log(V_0/V_{in})db



Comparator

Circuit Diagram & Waveforms





Design:

Inverting amplifier:

A = -Rf/R1

Take A = 1

Rf = R1

Choose Rf = , R1=

Non inverting amplifier:

A = 1+ Rf/R1

Take A =

Rf = R1

Choose Rf = , R1=

Adder:

$$V_{out} = -R_f \left(\frac{V_1}{R_1} + \frac{V_2}{R_2} + \frac{V_3}{R_3} \right)$$

Integrator:

Let input frequency be ,f= 1 kHz

$f = 1/ (2\pi RC)$

Take C= . Then R= . Use …. std.

Select $R_f = 10$ R =…. so that break frequency is ….. Hz. Select R_{com} =

Procedure:

I. Inverting amplifier and Non inverting amplifier

1. Set up the inverting amplifier on the bread board.

2. Feed a sine wave and observe the input and output simultaneously on CRO. Verify whether the output is …. sine wave in phase with input.

3. Set up the non inverting amplifier on the bread board.

4. Feed a sine wave and observe the input and output simultaneously on CRO. Verify whether the output is ….. sine wave with 180^0 out of phase with input.

II. Adder

1. Set up the adder circuit.
2. Feed the inputs and verify the output.

III. Integrator

1. Set up the integrator circuit.

2. Feed …., …. square wave at the input and observe the input and output simultaneously on CRO.

3. Feed a sine wave to the input and note down the output amplitude by varying the frequency of the sine wave. Enter it in tabular column and plot the frequency response

IV. Comparator.

1. Set up the comparator circuit.
2. Feed the inputs and verify the output.

Results: Familiarized with basic operational amplifier integrated

circuits. Gain of inverting amplifier :…….

Gain of non inverting amplifier: :…….

EXP 2 **DATE**

Measurement of Op-Amp parameters.

Aim: To measure the following parameters of an Op-amp i.e, input bias current, input offset voltage, input offset current, CMRR and slew rate.

Components required: Op-amp, resistors, capacitors, breadboard, CRO, function generator and power supplies.

Theory:

Input bias current I_B: It is defined as the average of the currents entering into the inverting and non-inverting terminals of an op-amp. $I_B=(I_{b1}+I_{b2})/2$. Typical value of input bias current is 80nA.

Input offset current I_{Os}: It is defined as the algebraic difference between the currents entering into the inverting and non-inverting terminals of an op-amp. $I_O=|I_{b1}-I_{b2}|$. Typical value of input offset current is 20nA.

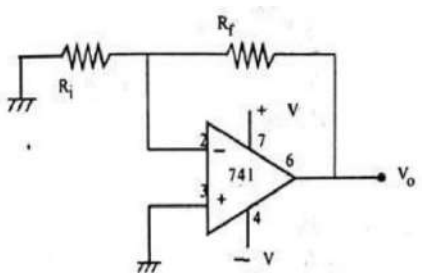
Input offset voltage: It is defined as the small voltage which is applied to overcome circuit imbalances due to which the output voltage is not zero for zero input voltage, ie voltage applied between the input terminals of an op-amp to nullify the output voltage. Typical value of input offset voltage is 2mV.

CMRR: It is the ratio of differential mode gain to common mode gain and is expressed in dB. CMRR=20 log (A_d/A_c) in dB.

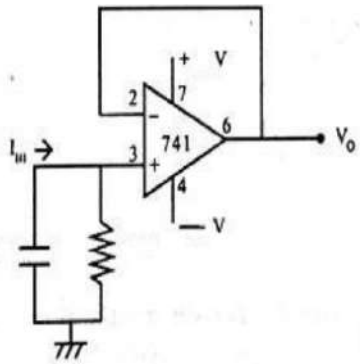
Slew rate: It is the rate of rise of output voltage. It is a measure of fastness of op-amp. It is expressed in v/μs.

Circuit diagrams

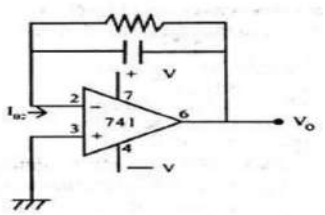
Circuit to measure input offset voltage



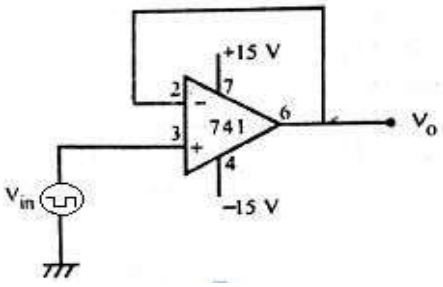
Circuit to measure I_{B1}



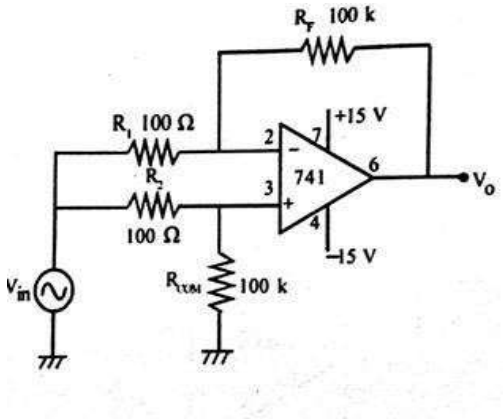
Circuit to measure I_{B1}



Circuit to measure slew rate



Circuit to measure CMRR.



Procedure

1. Set up the circuit to find the input offset voltage.
2. Measure the output voltage using the expression, $V_{iO} = V_O R_i/(R_f + R_i)$; where V_O is the output voltage and V_{iO} is the input offset voltage..
3. Set up the circuits for measuring input bias current and input bias voltage

4. Measure the output voltage using the expressions $V_O = I_{b1}R$ and $V_O = I_{b2}R$.
5. Calculate I_{B1} and I_{B2} and measure the bias and offset currents using the expression $I_B = (I_{b1}+I_{b2})/2$ and $I_{OS} = |I_{b1}-I_{b2}|$. Where I_B is bias current, I_O is offset current.
6. Setup the circuit to calculate the slew rate. Give a square input of …., …. Vary the input frequency and observe the output. Note down the frequency at which the output gets disturbed. Calculate the slew rate using the expression $SR = (2\pi f V_m)/10^6$

7. Set up the circuits for finding CMRR and apply a dc signal of …v to input and measure V_O. Calculate the CMRR using the expression $CMRR=V_i(R_f/R_i)/V_O$. Express the CMRR in dB using the expression 20 log(CMRR).

Results:

Measured the Op-amp parameters and obtain the following

results Input offset voltage.....= mV

Input bias current =.....A

Input offset current =.....A

Slew rate =.....V/μs.

CMRR = ……..

EXP 3 **DATE**

Difference Amplifier And Instrumentation Amplifier.

Aim: To design and setup a difference amplifier and instrumentation amplifier using opamp.

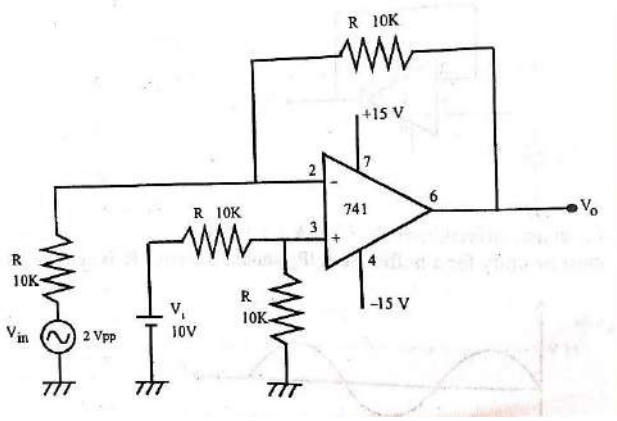
Components required: Op-amp, resistors, capacitors, breadboard, CRO, function generator power supplies.

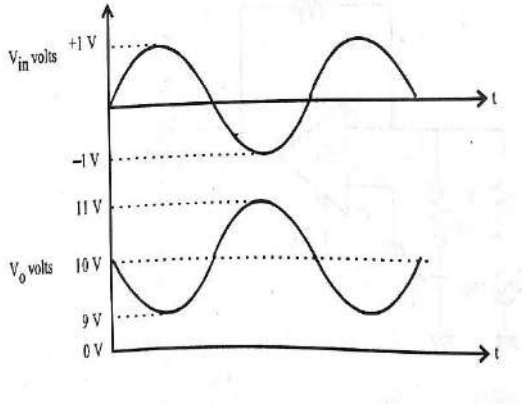
Theory: The difference amplifier circuit is very useful in detecting very small differences in the signal. since the gain is R_F/R_1 can be selected to be very large. The output $V_O = -R_F/R_1(V2-V1)$.If all the external resistors are of equal value, the gain of the amplifier becomes one. Thus the output is V_2-V_1. Hence the name subtractor.

Instrumentation amplifiers are widely used in data acquisition systems, remote sensing applications and instrumentation systems to measure temperature, humidity, light intensity and weight etc. Most of the instrumentation systems use a transducer in a bridge circuit. Instrumentation amplifier facilitates the amplification of potential difference take place due to the imbalance of the bridge circuit proportional to a change in physical quantity. The main feature of instrumentation amplifiers are high gain, high input resistance, high CMRR etc.

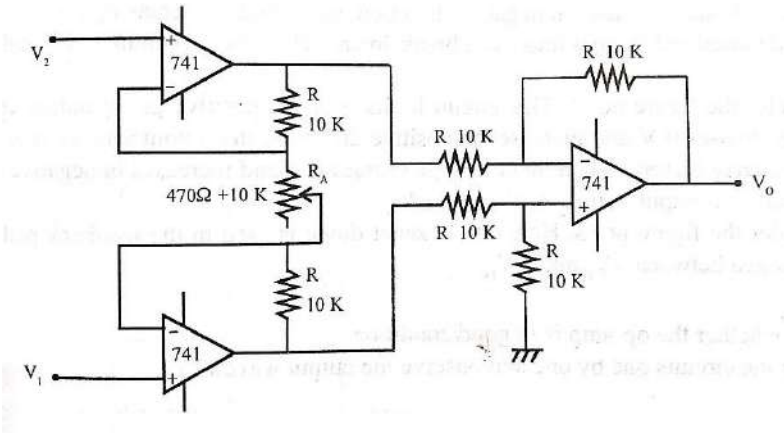
Circuit diagram and waveforms:

Difference amplifier





Instrumentation amplifier



Design:

Difference amplifier:

Let R=…K

$V1$ R

$$V0 = \frac{}{2}(1 + R/R) - V2\ \frac{}{R}$$
$$= V1 - V2$$

ELECTRONICS & COMMUNICATION

Instrumentation amplifier

We have, $V_0=(V_1-V_2)[1+2R/R_A max]$

Given, $1+2R/R_A max=….$

Take R and R_A max=…. Use … pot in series with ….

Procedure:

1.Verify the condition of op-amps.

2.Setup the circuit..

3.verify the output.

Result: Designed the difference and instrumentation amplifier.

EXP 4 **DATE**

Schmitt trigger circuit using op amps

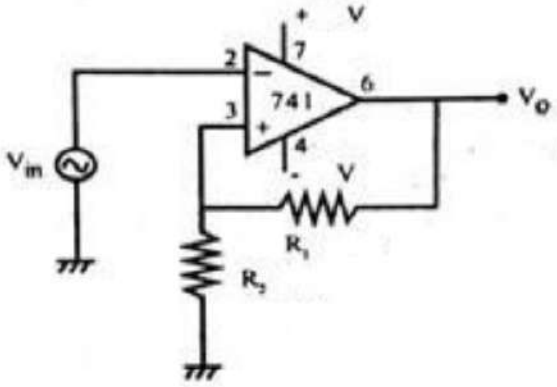
Aim: To design and set up a schmitt trigger circuit using op-amps for various LTP and UTP.

Components required: Op-amp, resistors, capacitors, breadboard, CRO, function generator and power supplies.

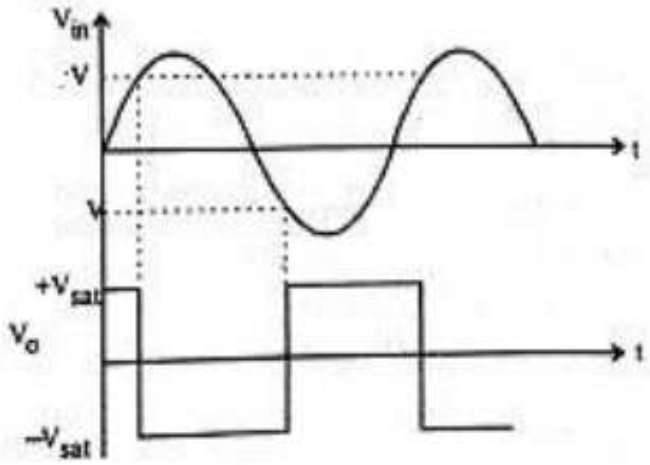
Theory: Schmitt Trigger converts an irregular shaped waveform to a square wave or pulse. Here, the input voltage triggers the output voltage every time it exceeds certain voltage levels called the upper threshold voltage VUTP and lower threshold voltage VLTP. The input voltage is applied to the inverting input. Because the feedback voltage is aiding the input voltage, the feedback is positive. A comparator using positive feedback is usually called a Schmitt Trigger. Schmitt Trigger is used as a squaring circuit, in digital circuitry, amplitude comparator, etc.

Circuit diagram

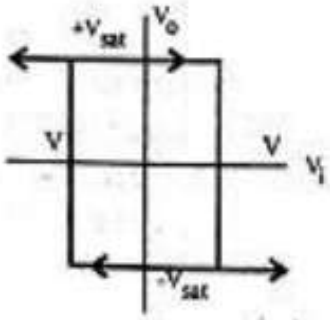
Schmitt trigger LTP=……V and UTP=…..V



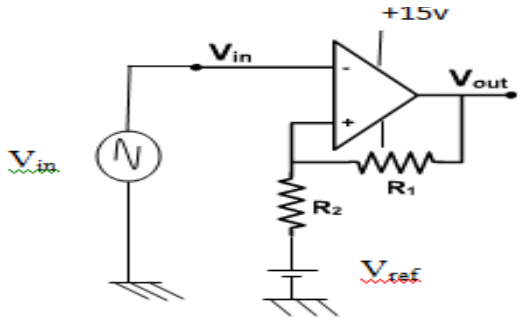
Waveforms



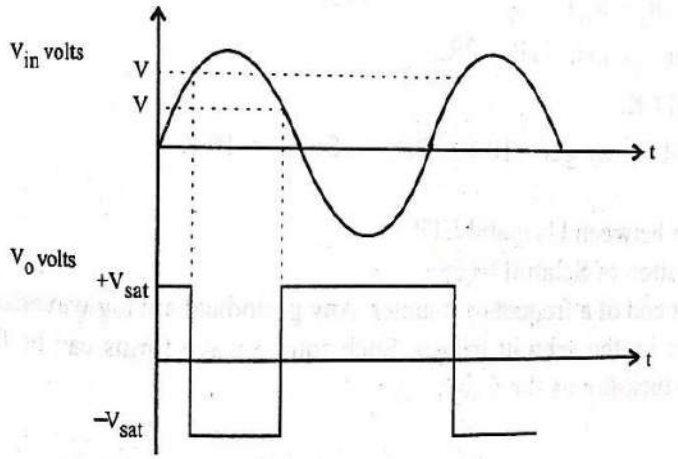
Transfer characteristics



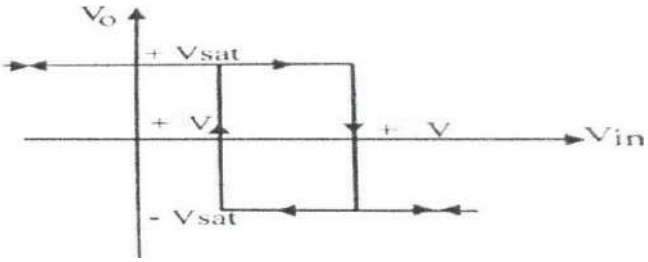
Schmitt trigger LTP = …..V and UTP =…….V



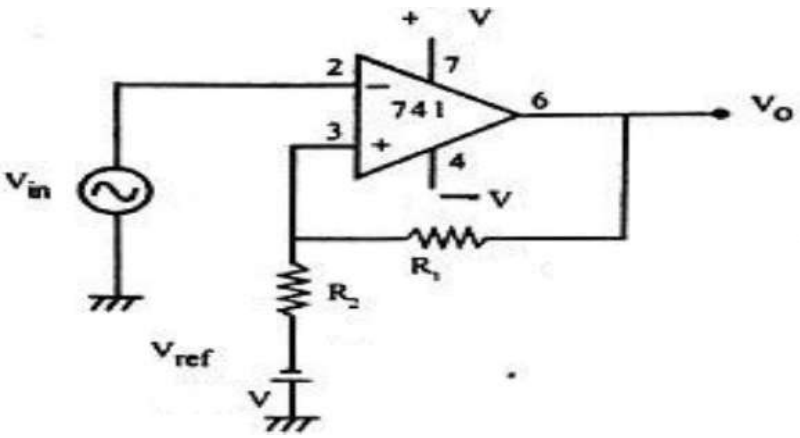
Waveform



Transfer Characteristics



Schmitt trigger LTP= …..V and UTP= …..V



Design:

Schmitt trigger LTP=…V and UTP=…V

Let the required LTP be ..V and UTP be…V

Normally, V_{sat}=..V when V+= ..V

LTP=….V= $-13R_2/(R_1+R_2)$

UTP=…..V=$13R_2/(R_{1+}R_2)$

Take R_2=….K and

R_1=….K

Schmitt trigger LTP=…..V and UTP=…..V

Let the required LTP be …..V and UTP be…..V

Normally, V_{sat}=…..V when V+=….V ,Use a reference voltage.

LTP=…..V=($-13R_2/(R_1+R_2)$)+(V_{ref} $R_1/(R_1+R_2)$)

UTP=…..V=($13R_2/(R_1+R_2)$)+(V_{ref} $R_1/(R_1+R_2)$)

Take R_2=…..K and R_1=…..K we get reference voltage V_{ref} =…. V

Schmitt trigger LTP= …..V and UTP= ….V

Let the required LTP be ….V and UTP be….V

Normally, V_{sat}=…..V when V+=….V ,Use a reference voltage.

LTP= ….V=($-13R_2/(R_1+R_2)$)+(V

$_{ref}$ $R_1/(R_1+R_2)$)

UTP= ….V=($13R_2/(R_1+R_2)$)+(V_{ref} $R_1/(R_1+R_2)$)

Take R_2=…K and R_1=….K we get reference voltage V_{ref} = ….V

Procedure:

1. Verify whether the op-amp was in good condition.
2. Set up the circuit for Schmitt trigger and switch on the supplies and observe the input and output on the CRO screen.
3. Observe the transfer characteristics.

Result : Designed the Schmitt trigger and obtained the output

EXP 5 **DATE**

Astable and Monostable multivibrator using Op –Amps

Aim: To design set up a astable and monostable multivibrators using op-amps for a frequency of 1kHz.

Components required: Op-amp, resistors, capacitors, breadboard, CRO, function generator and power supplies.

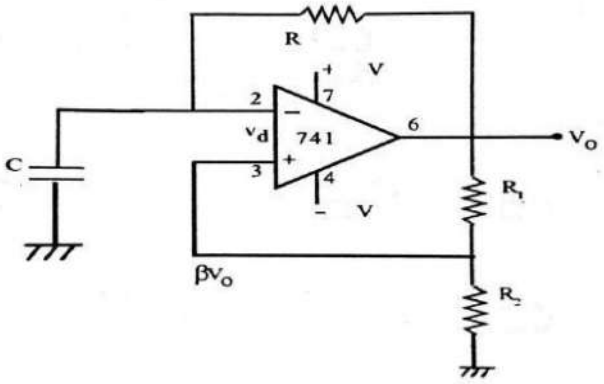
Theory:

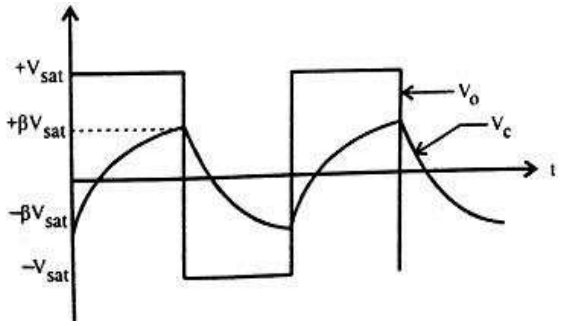
Astable multivibrator: Astable multivibrators are capable of producing square wave for given frequency, amplitude and duty cycle. The output of an op-amp is forced to swing repetitively between positive saturation +Vsat and negative saturation –Vsat resulting in asquare wave output. This circuit is also called free running multivibrator or square wave generator. The output of the op-amp will be in positive saturation if differential input voltage is negative and vice versa. The differential voltage Vd = Vc -βVsat where β is the feedback factor. βVsat is the potential at non-inverting terminal of op-amp. Consider the instant at which Vo=+Vsat. Now the capacitor charges exponentially towards +Vsat through R. Automatically Vd increases and crosses zero. This happens when Vc changes to –Vsat. Now capacitor starts to discharge to zero and recharge towards –Vsat. Now Vd decreases and crosses zero. This happens when Vc=-βVsat. The moment Vd becomes negative again, output changes to +Vsat. This completes one cycle. The time period T of the square wave is T= 2RCln(1+β)/(1-β).If β is made ½, T= 2.2RC. Astable multivibrator is particularly useful for the generation of frequency in the audio frequency range. Higher frequencies are limited by the delay time and slew rate of the op-amp.

Monostable multivibrator: A Monostable Multivibrator, often called a one-shot Multivibrator. It ha a stable state and and a quasi stable state. The circuit remains in stable state until triggering signal causes a transition to quasi stable state. After a time interval, it returns to the stable state. So a single pulse of predetermined duration can be generated using this circuit. Consider the instant at which the output Vo=+Vsat. Now the diode D1 clamps the capacitor voltage Vc at 0.7V. feedback voltage available at non inverting terminal is +βVsat. When the negative going trigger is applied such that potential at non inverting terminal becomes less than
0.7 V , the output switches to –Vsat. Now the capacitor charges through R towards –Vsat, because the diode becomes reverse biased. When the capacitor voltage become more negative than –Vsat, the comparator switches back to +Vsat, and the capacitor C starts charging to +Vsat through R until Vc reaches 0.7.

Circuit diagrams and waveforms :

Astable multivibrator





DESIGN

Required period of oscillation T=….ms with duty cycle

……% Time period T =$T_{1+}T_2$ = 2RC ln(1+β)/(1-β)

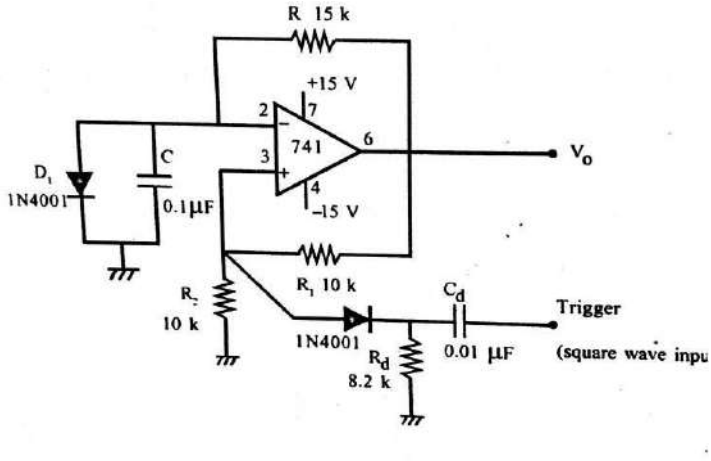
Where β, the feedback factor=$R_2/(R_1+R_2)$

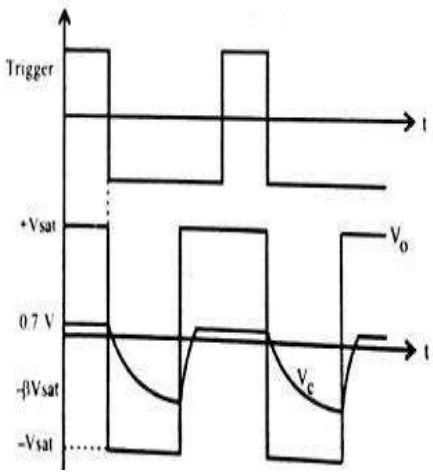
Take β=….. and R_2=….K. Then

R_1=…..K. When β=….., T=….RC

Let C be …µF. Then R= …..K.

Monostable multivibrator





DESIGN

Required period of oscillation T=…..ms with duty cycle

…..% Time period T = RC ln[1/(1-β)]T=…..RC

Where β, the feedback factor=$R_2/(R_1+R_2)$

Take β=….. and R_2=…..K. Then

R_1=…..K. Let C be …..µF. Then R=…..K

Use …..K

Design of differentiating circuit: $R_dC_d<0.016T_t$.

Take trigger time period T_t=….ms and C_d=…..µF Then R_d=….K

Procedure:

1. Verify the conditions of op-amp.

2. Set up the circuit astable multivibrator and observe the output waveform. Note down their frequencies and amplitudes.

3. Set up the circuit monostable multivibrator and feed …..Vpp,….Hz square wave at the trigger input and observe the output waveform. Note down their frequencies and amplitudes.

Results: Designed the astable and monostable multivibrator using opamp.

EXP 6 **DATE**

Timer IC NE555

Aim: To study NE555 Timer and to design and setup an astable multivibrator and monostable multivibrator using NE555 timer for a frequency of 1KHz.

Components required: NE555 timer, resistors, capacitors, breadboard, CRO, function generator and power supplies.

Theory: The 555 timer is a highly stable device for generating accurate time delay or oscillation. Signetics Corporation introduced this device as SE555/NE555 in 1971. The 555 timer is available with the supply voltage between +4.5 to +18v, supply current 3 to 6 mA. It is compatible with both TTL and CMOS logic circuits. The functional block diagram of 555 consist of two comparators, a flip-flop, an output stage, two BJT Q_1 and Q_2 and a voltage divider network. The comparators are devices whose outputs are HIGH when the positive(+) input voltage is greater than the negative (-) input voltage and LOW when the negative (-) input voltage is greater than positive (+) input voltage. The voltage divider consisting of three 5KΩ resistors provide a trigger level of $1/3V_{CC}$ and a threshold levels of $2/3V_{CC}$. The control voltage input can be used externally adjust the trigger and threshold levels to other values, if necessary.

When the normally HIGH trigger input momentarily goes below $1/3V_{CC}$, the output of comparator 2 switches from LOW to HIGH and set the S-R flip flop, causing the output to go HIGH and turning the discharge transistors Q_1 OFF. The output will remain HIGH until the normally LOW threshold input goes above $2/3V_{CC}$ and causes the output of comparator 1 to switch from LOW to HIGH. This resets the flip flop, causing the output to go back LOW and turning the discharge transistor ON. The external reset input can be used to reset the flip flop independent of threshold circuit. The trigger and threshold inputs are controlled by external components connected to produce either monostable or astable action.

So the output of timer becomes HIGH when the trigger input voltage is less than $1/3V_{CC}$ and the output becomes LOW when the threshold voltage is greater than $2/3V_{CC}$. Also in stable state, the output of timer is LOW.

Astable multivibrator: When the power supply V_{CC} is connected, the external timing capacitor 'C" charges towards V_{CC} with a time constant (R_A+R_B) C. During this time, pin 3 is high ($\approx V_{CC}$) as Reset R=0, Set S=1 and this combination makes $\overline{Q}$ =0 which has unclamped the timing capacitor 'C'.

When the capacitor voltage equals 2/3 V_{CC}, the upper comparator triggers the control flip flop on that $\overline{Q}$ =1. It makes Q1 ON and capacitor 'C' starts discharging towards ground through R_B and transistor Q1 with a time constant R_BC. Current also flows into Q1 through R_A. Resistors R_A and R_B must be large enough to limit this current and prevent damage to the discharge transistor Q1. The minimum value of R_A is approximately equal to $V_{CC}/0.2$.

During the discharge of the timing capacitor C, as it reaches $V_{CC}/3$, the lower comparator is triggered and at this stage S=1, R=0 which turns $\overline{Q}$ =0. Now $\overline{Q}$ =0 unclamps the external timing capacitor C. The capacitor C is thus periodically charged and discharged between 2/3 V_{CC} and 1/3 V_{CC} respectively. The length of time that the output remains HIGH is the time for the capacitor to charge from 1/3 V_{CC} to 2/3 V_{CC}.

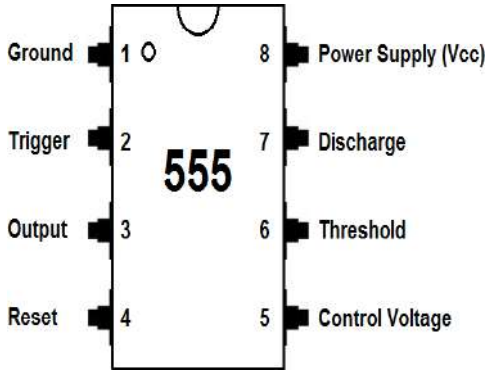
The charging period of capacitor = 0.69 (R_A + R_B) C.

The discharging period of capacitor = 0.69 R_B C.

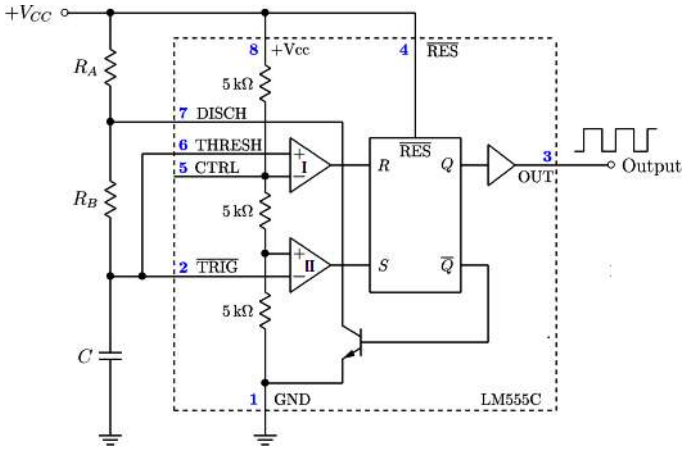
Monostable multivibrator: A Monostable Multivibrator, often called a one-shot Multivibrator, is a pulse-generating circuit in which the duration of the pulse is determined by the RC network connected externally to the 555 timer. In a stable or stand by mode Q is high and in turn, Q1 is turned ON and output is low. When the negative going trigger passes through $V_{CC}/3$, the FF is set i.e. Q =0. This makes transistor Q1 off. The capacitor starts charging towards V_{CC}, which was earlier clamped to zero. After a time period, the capacitor voltage becomes greater than 2/3 V_{CC} and upper comparator resets the FF, i.e. R=1, S=0. This makes Q =1. In turn the transistor Q1 turns ON and thereby discharging the capacitor C rapidly to ground potential. Mononostable circuit has only one stable state (output low), hence the name monostable. Normally the output of the Monostable Multivibrator is low.

Circuit diagram:

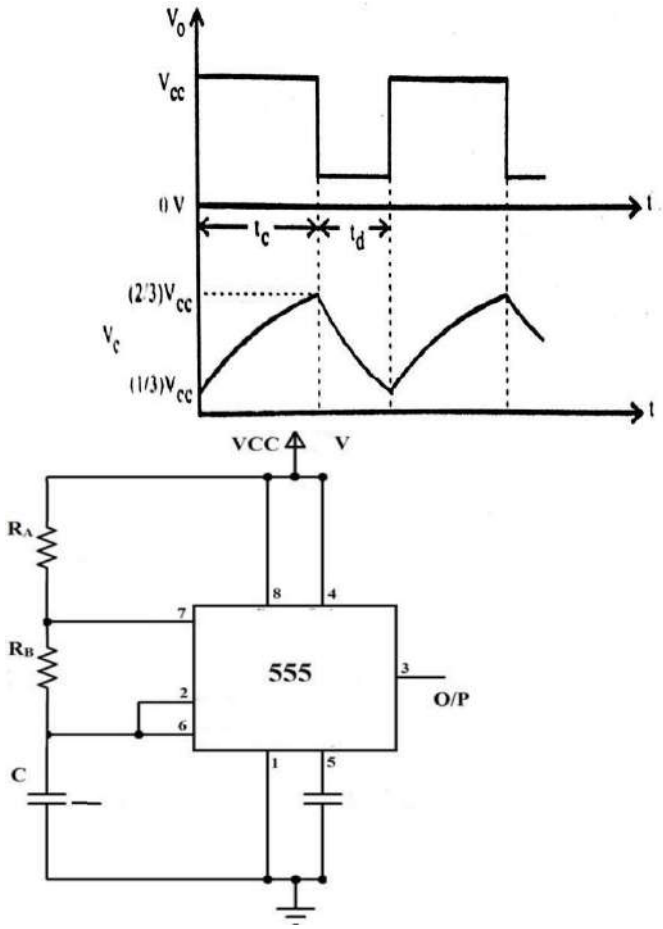
Pin out of 555 timer IC



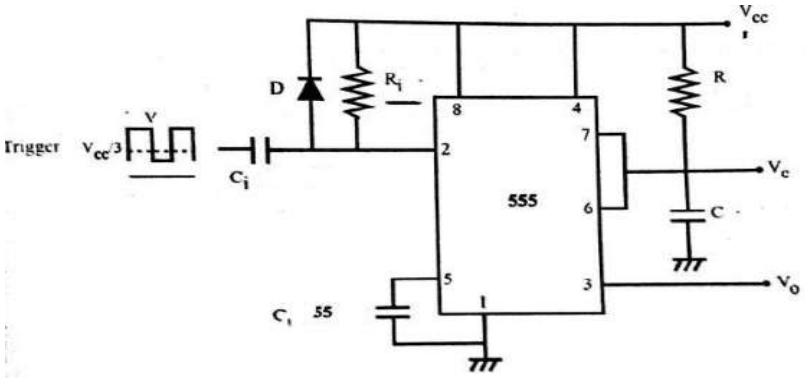
Functional block diagram of 555 timer

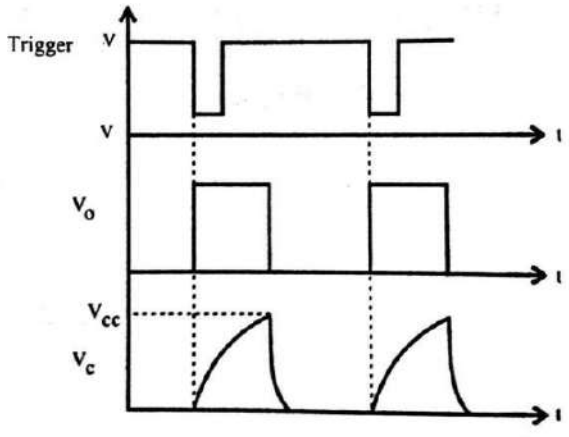


Astable multivibrator using 555 timer



Monostable multivibrator using 555 timer





Design:

1.Astable multivibrator

Take $V_{CC=}$…..V and t_c=….ms and

t_d=….ms We have, t_c=0.69(R_A+R_B)C and

t_d=….R_B

The R_A and R_B should be in the range of 1K to 10K to limit the collector current of the internal transistor. Take R_A=R_B=….KΩ.

Let C=…..µF. Choose C_1=….µF.

2.Monostable multivibrator

Take V_{CC}=……V and T=…ms.

We have, T=…..RC

Take R=…..K to limit current through the internal transistor to

….Ma. Then C=…..µF.

Design of triggering circuit we have $R_iC_i \leq 0.0016T_t$ where T_t is the time period of the trigger.

Take T_t=…..ms. Take R_i=…..KΩ to avoid loading. Then C_i=…..µF. Choose C_1=…..µF.

Procedure:

1.Set up the astable multivibrator circuit after verifying the condition of the IC

2.Observe the output waveform at pin no.3 and 6 of the IC.

3. Set up the monostable multivibrator circuit.

4. Use positive pulses of amplitude Vcc and frequency ….. Hz as the trigger.

5. Observe the output waveform at pin no.3 and 6 of the IC

Result: Familiarized the NE555 Timer and designed an astable multivibrator and monostable multivibrator using NE555 timer for a frequency of 1KHz

EXP 7 **DATE**

Triangular, square wave and sawtooth generators using Op- Amps

Aim: To set up and study a saw-tooth and triangular wave form generator using Op-Amp for 1KHz frequency.

Components required: Op-amp, resistors, capacitors, breadboard, CRO, function generator and power supplies.

Theory:

Triangular wave generator:

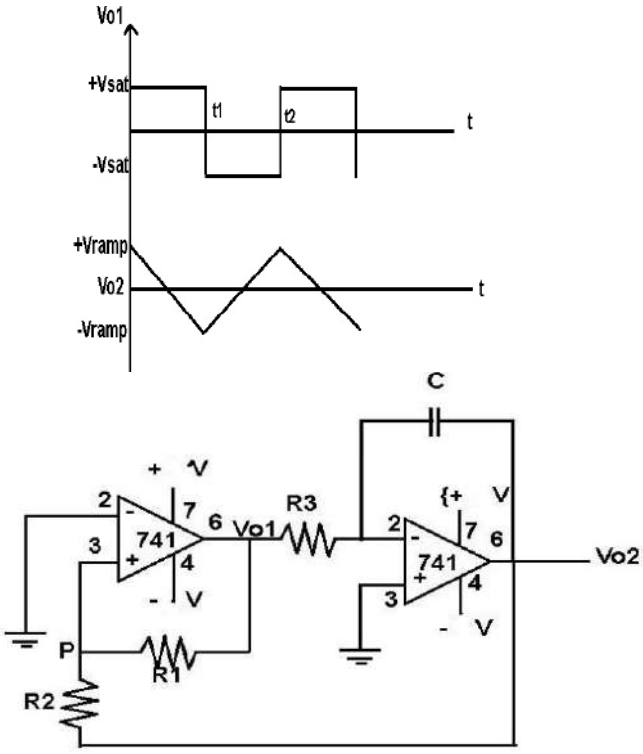
This circuit uses two op-amps. One functions as a comparator and other as an integrator. Comparator compares the voltage at point P continuously with respect to the point voltage at the inverting input which is at zero volt. When voltage at P goes slightly above zero, the output of A will switches to negative saturation. Suppose the output of A is at positive saturation +Vsat, since this voltage is at the input of integrator, the output of A2 will be a negative going ramp. Thus one end of voltage divider R1 and R2 is at +Vsat and other end is at negative going ramp. At the time t=t1, when the negative going ramp attains the value of -Vramp, the effective voltage at P becomes slightly less than zero volt. This switches output of A1from +Vsat to –Vsat level. The output of A2 increases in the positive direction. At the instant t=t2 , voltage at P becomes just above zero volt thereby switching the output of A from –Vsat to + Vsat . The cycle repeats and generates a triangular waveform. Frequency of triangular waveform f = (R1/4R2R3C) .Peak to peak amplitude of ramp voltage is 2(R2/R1) Vsat.

Saw tooth waveform generator:

In sawtooth waveform generator the rise time is much higher than its fall time or vice - versa. The triangular waveform generator can be converted into a Sawtooth waveform generator by including a variable dc voltage into non inverting terminal of the integrator.This can be done by using a port. When the wiper of the port is at the centre ,the output will be a triangular wave since the duty cycle is 50%.If the wiper moves towards negative ,the rise time of Sawtooth becomes larger than fall time. If the wiper moves towards positive ,the fall time becomes larger than rise time. The Sawtooth waveform generators have wide applications in time base generators and pulse width modulation circuits.

Circuit diagram and waveforms:

Triangular wave generator



Design:

Triangular wave generator:

Frequency,f =R1/(4*R2*R3*C)

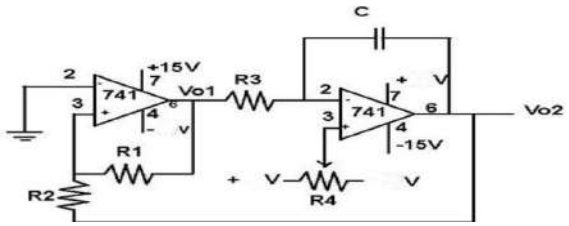
Peak-peak output of ramp V_{pp} = 2R2/R1

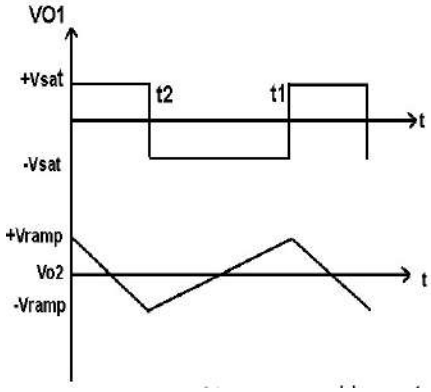
Let the required V_{pp} = …..V and V_{sat} =

….. V Assume R1 = …..KΩ then R2 =

…..Ω Take C = …. µF, so R3 = …... KΩ

Sawtooth wave generator


C
+15V
741
Vo1
R3
741
Vo2
-15V
R1
R2
R4


VO1
+Vsat
t2
t1
t
-Vsat
+Vramp
Vo2
t
-Vramp

Design:

Sawtooth wave generator

Frequency,f =R1/(4*R2*R3*C)

Peak-peak output of ramp Vpp = 2R2/R1

Let the required V_{pp} = ...V and V_{sat} = … V

Assume R1 = …KΩ then R2 = …. Ω Use …. Ω standard

Take C = …. µF, so R3 = …. KΩ Use …..KΩ pot

Select R4 = ….. KΩ

Procedure:

1. Set up the waveform generator circuit.

2. Obtain the output and note down the amplitude and frequency.

3. Set up the circuit of saw tooth wave generator.

4. Observe the output of both op-amps and note down the rise time and fall time.

5. Obtain the output by moving the wiper of port in both directions and observe the changes taking place in waveforms.

Results: Designed and studied the saw tooth and triangular wave generator.

EXP 8 **DATE**

WIENBRIDGE OSCILLATOR USING OP-AMP WITH AND WITHOUT AMPLITUDE STAILIZATION

Aim

:

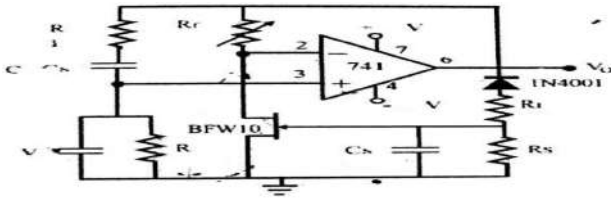
1. To design set up a wien bridge oscillator incorporating amplitude stabilization.
2. Design and set up a Wien bridge oscillator using an op-amp for a frequency of 1kHz.

Components required: Op-amp, resistors, capacitors, breadboard, CRO, function generator and power supplies

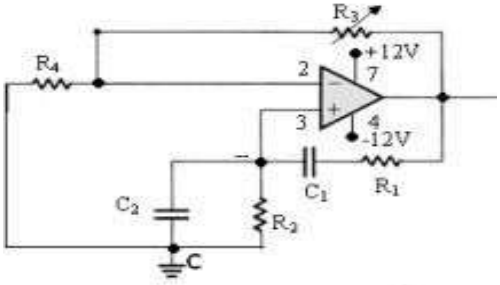
Theory: An FET circuit in association with the wien bridge oscillator, helps the stabilization of the amplitude of oscillation. The N-channel JFET acts as a voltage controlled resistor and the diode circuit function as a negative peak detector. The dc voltage at the gate of FET becomes more negative when amplitude of oscillation increases. Then gate of FET gets reverse biased and effective resistance from drain to source increases. This causes to decrease the gain according to the relation $A=1+(R_f /R_i)$ and amplitude is brought back to a stable level. When output peak starts to decrease, opposite patter occurs. This is an audio frequency oscillator of high stability and simplicity.The feedback signal in the circuit is connected to the non-inverting input terminal so that the op-amp is working as a non-inverting amplifier. Therefore, a feedback network need not provide any phase shift.The circuit can be viewed as a Wien bridge with a series RC network in one arm parallel RC network in the adjoining arm.Resisrors Ri and Rf are connected in the remaining two arms.The conditionof zero phase shift around the circuit is achieved by balancing the bridge.The frequency of oscillation is the resonant frequency of the balanced bridge and is given by the expression $fo=1/(2\pi RC)$.From the analysis of the circuit,it can be seen that the feedback factor $\beta=1/3$ at the frequency of oscillation. Therefore, for the sustained oscillation,the amplifier must have a gain of 3.

Circuit diagram:

With amplitude stabilization



Without amplitude stabilization



Design:

Without amplitude stabilization

Required frequency,fo= …

kHz. Given fo=1/(2πRC)

Let C=…..µF. R_1=R_2=R=….kΩ

Use …..kΩ std.

Gain 1+Rf/Ri=3, Then Ri=…k.

Then R_3= Rf=….k,use …k potentiometer.

Select R_4= ….K, andC_1 and C_2=….µF.

With amplitude stabilization

Required frequency, fo= ….

kHz. Given fo=1/(2πRC)

Let C=….µF. R_1=R_2=R=….k Use …..k std.

Gain 1+Rf/Ri=3, Then Ri=….k.

Then R_3= Rf=…..k,use ….k

potentiometer. Select R_s= ….M,

andC_S=….µF

Procedure:

1. Verify the conditions of op-amp and JFET.

2. Set up the circuit and observe the output waveform. Note down the frequency and amplitude of oscillation.

Result:

Designed the wienbridge oscillator using op-amp with and without amplitude stabilization

EXP 9 **DATE**

RC PHASE SHIFT OSCILLATOR

Aim: To design and set up an RC phase shift oscillator using op-amp for a frequency of

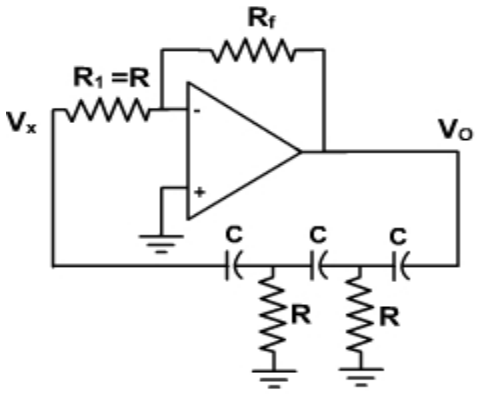
1 kHz.

Components required: Op-amp, resistors, capacitors, breadboard, CRO, function generator and power supplies

Theory: RC phase shift oscillator consists of an op-amp as the amplifying stage and three RC cascaded networks as the feedback network. The feedback network provides a fraction of the output voltage back to the input of the amplifier.The op-amp is functioning in the inverting mode.Therefore any signal which appears at the inverting terminal is shifted by 180^{o} at the output.An additional 180^{o} phase shift required for oscillationas per Barkhausen criteria,is provided by the cascaded RC network.Thus the total phase shift around the loop becomes 0^{o}.

The frequency of oscillation is given by,$f_o=1/(2\pi\sqrt{6}RC)$.The gain of the inverting op-amp should be atleast 29 at this frequency because the attenuation provided by the feedback network is 1/29.The gain is kept slightly greater than 29 to ensure that the variations in circuit parameters will not make the loop gain less than unity,and thus oscillations died out.For lower frequencies(<1 MHz),op-amp 741 may be used,however for higher frequencies,LM318 or LF351 should be used.

Circuit diagram:



Design:

Let the required frequency fo be,$1/(2\pi\sqrt{RC})$=…..kHz.

Take C=….Mf.Then R=….Ω. Use ……Ω.

Gain R_f/R_1=…..Take R_1=….k and R_f=…..k pot.

Procedure:

1.Verify whether the op-amp is in good condition and set up the circuit as shown in the circuit diagram.

2.Note down the amplitude and frequency of output waveform.

Result:

Designed the RC phase shift oscillator using op-amp.

EXP 10 **DATE**

Precision rectifiers using Op-Amp.

Aim: To design and setup a precision rectifier using op amp

Components required: Op-amp, resistors, capacitors, breadboard, CRO, function generator and power supplies.

Theory: The precision rectifier, also known as a super diode, is a configuration obtained with an operational amplifier in order to have a circuit behave like an ideal diode and rectifier. It is useful for high-precision signal processing.

In a precision half wave rectifier a diode is placed in the negative feedback path of the op amp. If V_{in} becomes positive, the output of the op amp will become positive. Hence the diode conducts and a closed feedback path is established between the op amp's output terminal and the negative input terminal. This negative feedback path will cause a virtual short circuit to appear between two input terminal. Thus $V_0=V_{in.}$ For the circuit to start working V_{in} has to exceed the voltage equal to diode drop/op amp open loop gain. Since the op amp open loop voltage is very high,starting voltage is negligiably small. Hence the circuit behaves like an ideal diode.

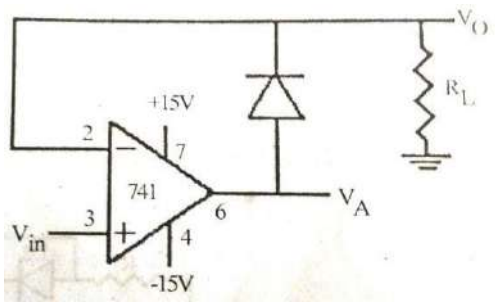
If V_{in} becomes negative, the output V_o of the op amp will be negative. This will reverse bias the diode and no current will flow through the load resistance $R_{L,}$ causing the output V_O to remain at 0V. Since the diode is off, the op amp will be operating as open loop circuit and its output will b at $-V_{sat}$.

In a practical half wave precision rectifier an inverting amplifier is converted to a rectifier by adding two diodes. The resistors are designed such that gain of the amplifier is one. Duing the positive half cycle of the input, the output becomes negative. At this moment D_2 becomes forward biased and D_1 becomes reverse biased. Hence V_{01} is zero and V_{02} is negative. During the negative half cycle, the output of the inverting amplifier becomes positive and D_1 becomes forward biased and D_2 becomes reverse biased. Hence the V_{01} is positive and V_{02} is zero.

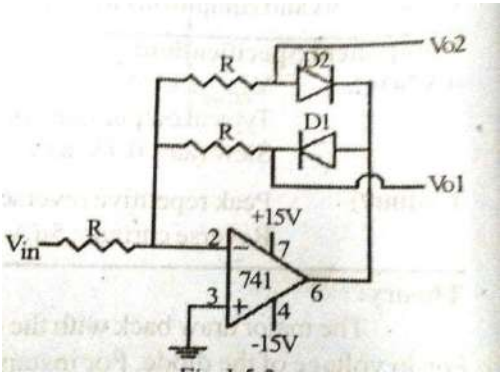
A full wave precision rectifier consists of a half wave rectifier and an inverting adder. We can vary the output gain by changing the values of resistors as required . Input signal is directly given to the adder at X. T he output V_{02} which gives the negative ripples only of the half wave rectifier is also given to the adder at Y. The mathematical addition of these two signals will gives the output of a full wave rectifier. The resistance value of the adder is adjusted such that the output is –(2Y+X).

Circuit diagram and expected waveforms:

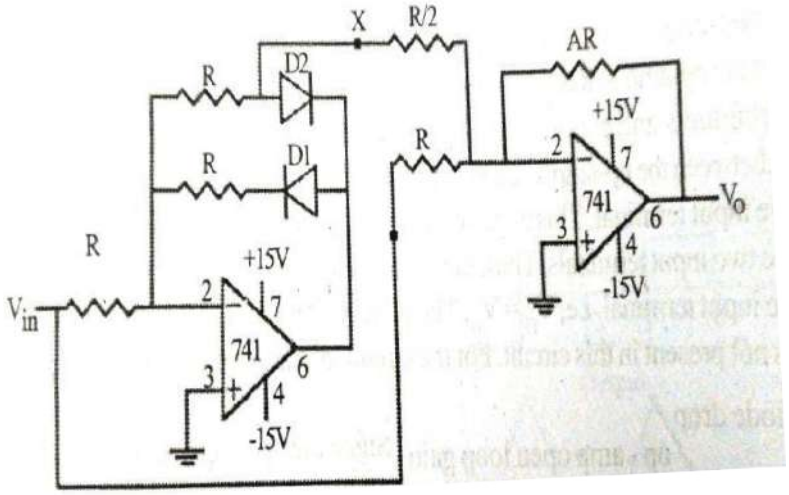
Half wave precision rectifier

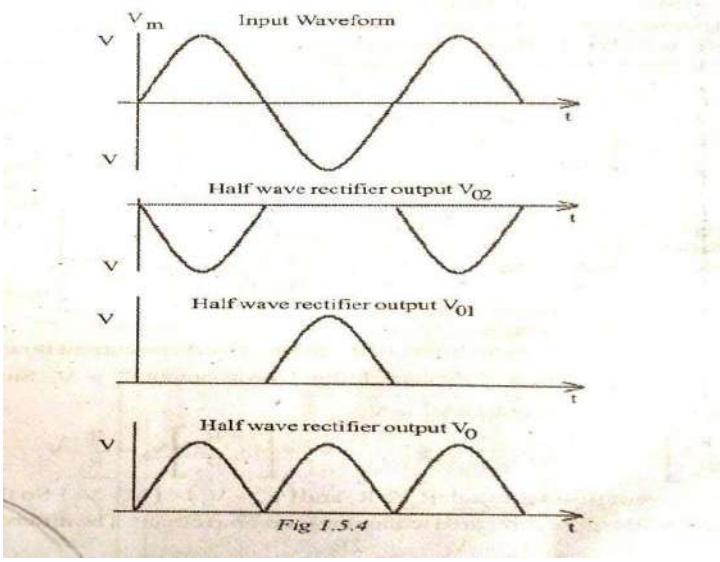


Practical half wave precision rectifier



Full wave precision rectifier





Design:

For unity gain select R=…..K

Procedure:

1. Set up the circuit.
2. Apply a sine wave of milli-volt range and frequency less than …..KHz.
3. Observe the output in the CRO.
4. Repeat the procedure for the full wave circuit.

Results: Designed the circuit for precision rectifier and obtained the result.

EXP 11 **DATE**

Active second order filters using Op-Amp (LPF, HPF, BPF and BSF).

Aim: To design and set up the active second order filters using op amp.

Components required: Op-amp, resistors, capacitors, breadboard, CRO, function generator and power supplies.

Theory : A stop band having a 40 db/decade roll off is obtained with the second order filters. They are important because higher order filters can be realized using them. The gain of the second order filters can be fixed by R_1 & R_F, while the cutoff frequency can be obtained from R_2, R_3, C_2, and C_3 as $f=1/(2\pi\sqrt{(R_2 R_3 C_2 C_3)})$ and voltage gain $A_V= 20 \log(V_0/Vin)$.

For a second order low pass filter, the voltage gain magnitude is $|V_0/V_i| = A_F/\sqrt{(1+(f/f_H)^4)}$

For a second order high pass filter, the voltage gain magnitude is $|V_0/V_i| = A_F/\sqrt{(1+(f_L/f)^4)}$

For a second order band pass filter, the voltage gain magnitude is

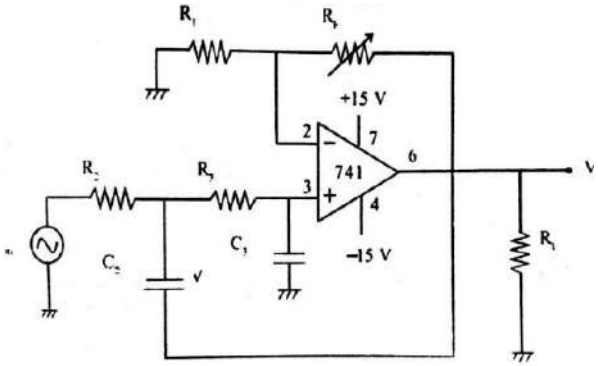
$$|V_0/V_i| = A_F (f/f_L)/\sqrt{\{[1+(f/f_L)^2][1+(f/f_H)^2]\}}$$

A band pass filter of -40 dB/decade fall off rate can be formed by cascading a second order HPF and LPF.

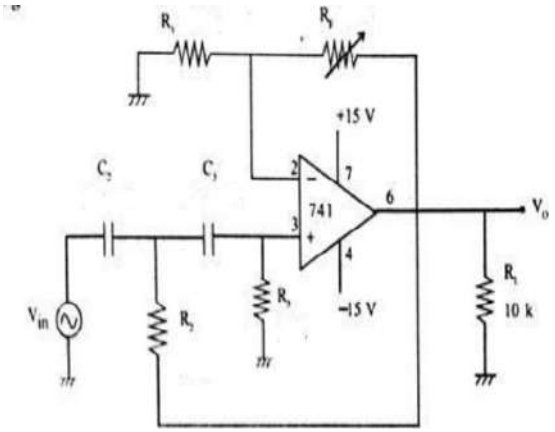
A band reject filter is obtained by cascading a LPF, HPF and a summer circuit.

Circuit diagram:

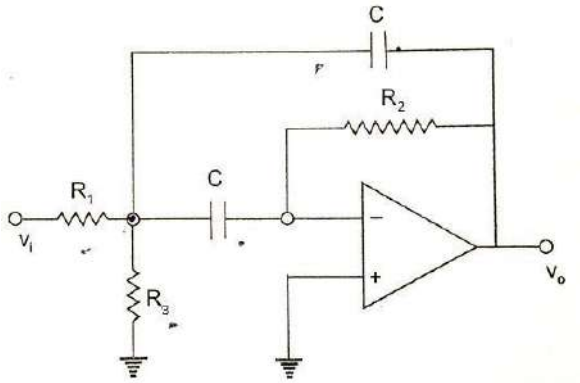
Low pass filter



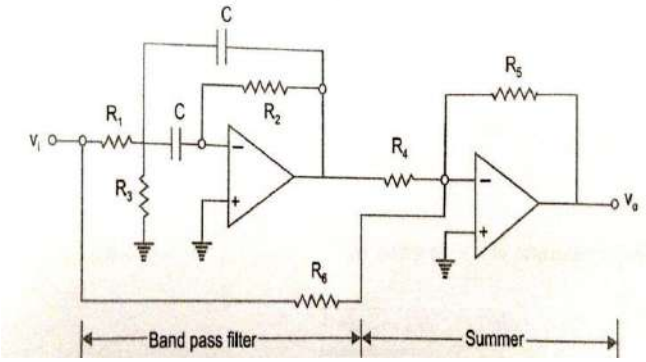
High pass filter



Band pass filter



Band reject filter



Design:

<u>Low pass filter design</u>

Required cutoff frequency f_H = …

KHz We have $f_H = 1/2\pi\sqrt{(R_1R_2C_2C_3)}$

Let $C_2=C_3$= ……µF. Then $R_2=R_3$=…..K

For $R_2=R_3$and $C_2=C_3$,

The pass band gain $A_F=(1+R_F/R_1)$ must be …..

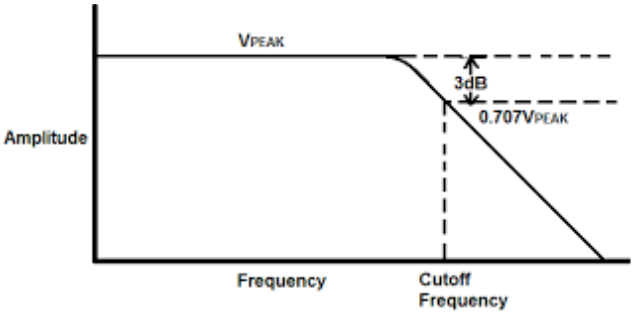
That is R_F= ……R_1

Let R_1=……K. Then R_F=…..K.(use ….k)

Tabular Column V_{in} = …. volt

f(Hz)	V_0(V)	Log(f)	A_v in dB

Graph



<u>High pass filter design</u>

Given cutoff frequency f_L=

….KHz We have,

$f_L=1/2\pi\sqrt{(R_2R_3C_2C_3)}$

Take $C_2=C_3=C$ and $R_2=R_3=R$

Then $f_L=1/2\Pi RC$

Assume C=….µF. Then R=…..K.Use ….k std.

The pass band gain $A_F=(1+R_{F/}R_1)$ must be ……. for butterworth filter.

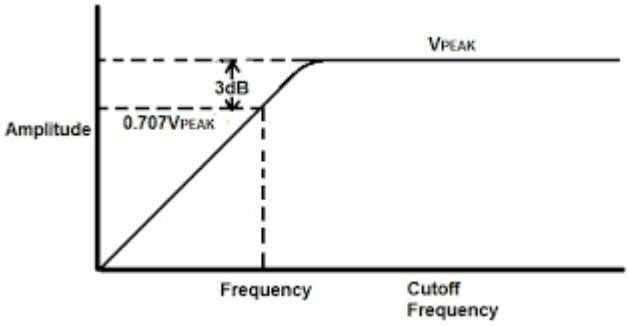
ie R=…..R_1

Take R_1=…..K .Then R_F=….. K . Use ….K pot

Tabular Column V_{in} = ……volt

f(Hz)	V_0(V)	Log(f)	A_v in dB

Graph

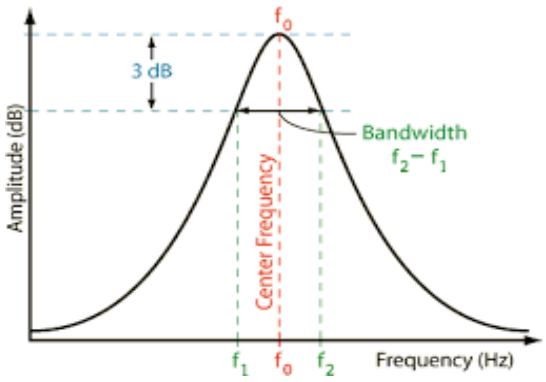


<u>Band pass filter</u>

Tabular Column V_{in} = …..volt

f(Hz)	V_0(V)	Log(f)	A_v in dB

Graph

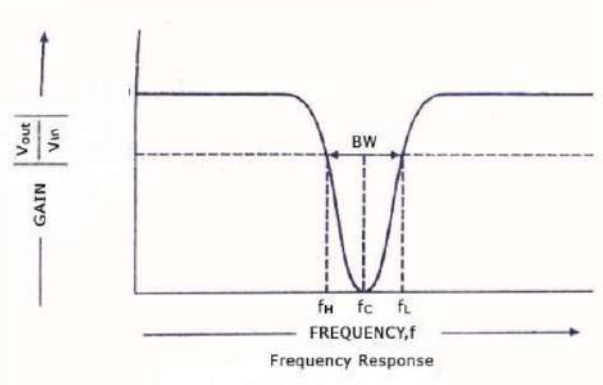


Band reject filter

Tabular Column V_{in} = …..volt

f(Hz)	V_0(V)	Log(f)	A_v in dB

Graph



Frequency Response

Procedure:

1. Set up the circuits and feed a ….Vpp sine wave from the signal generator.
2. Vary the frequency in steps and note the output voltage.
3. Plot the frequency response.
4. Mark the lower cut-off frequency and calculate the roll-off in dB/decade

Results:

Designed the second order filters using op amp and obtained the results as

Cutoff frequency of low pass filter =.....KHz

Roll off of low pass filter =……..

Cutoff frequency of high pass filter =.....KHz

Roll off of high pass filter =…….

Cutoff frequency of band pass filter =.....KHz

Roll off of band pass filter =……..

Cutoff frequency of band reject filter =.....KHz

Roll off of band reject filter =……..

EXP 12 **DATE**

Notch filters to eliminate the 50Hz power line frequency

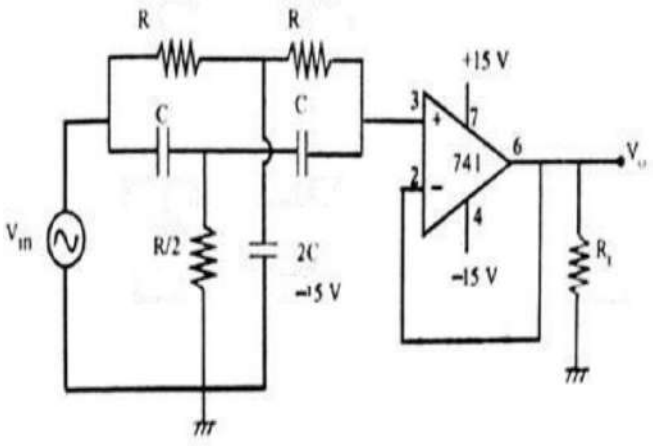
Aim: Design and setup a notch filters to eliminate the 50Hz power line frequency.

Components required: Op-amp, resistors, capacitors, breadboard, CRO, function generator and power supplies.

Theory: Band elimination filter is also known as band rejection filter or band stop filter. Wideband rejection can be setup by connecting a low pass filter and a high pass filter in parallel. If an LPF with f_L is connected in parallel with HPF with f_H such that $f_H > f_L$, it forms BEF with bandwidth $f_H - f_L$.

Narrow band rejection filter is also known as notch filter. It provides maximum attenuation at f_0. This is achieved by a twin-T RC network. Passive twin T network has relatively low figure of merit Q. Q can be increased by associating with a voltage follower using op amp. Notch filter has wide applications in communication field. It is used to eliminate undesired frequencies. The very common application is to remove power supply that access at 50 Hz.

Circuit diagram:



Design:

Required notch frequency $f_N = 1/2\Pi RC = \ldots.KHz$

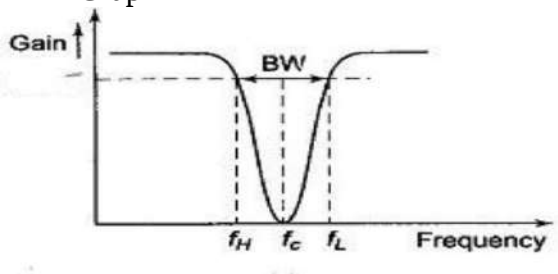
Take C=…..µF. Then R=…..K

Take 2C=……1µF and R/2=…..K

Tabular Column V_{in} = …… volt

f(Hz)	V_0(V)	Log(f)	A_v in dB

Graph



Procedure:

1. Set the signal generator output as …..V sine wave
2. Vary the frequency of sine wave and note down the output voltage.
3. Plot the frequency response on graph sheet.

Result:

Designed a notch filters to eliminate the 50Hz power line frequency.

EXP 13 **DATE**

IC voltage regulators.

Aim: To design and set up a low voltage and a high voltage regulator using IC RC723.

Components required: Op-amp, resistors, capacitors, breadboard, CRO, function generator and power supplies.

Theory: 723 is a general purpose regulator that can be adjusted over a wide range of both positive and negative regulated voltage. It has two sectios 1- a zener diode, a constant current source and a reference amplifier that produces a fixed voltage of 7.15 at the terminal $V_{ref.}$. The constant current source forces the zener to operate at a fixed point so that the zener outputs a fixed voltage. 2- it consist of an error amplifier, a series pass transistor Q_1 and a current limiting transistor Q_2 .T he error amplifier compares a sample of the output voltage applied at the INV input terminal. The error signal controls the conduction of Q_1. These two sections are not internally connected but various points are brought out on the IC package.723 regulator IC are available in a 14 pin dual in line package or 10 pin metal can. It is inherently a low current device, but it can be boosted to provide 5A or more current by connecting external components. But it has no built in thermal protection .It also has no short circuit current limits. It can operate with an input voltage from 9.5V to 40V and provide output voltage from 2V to 37V.

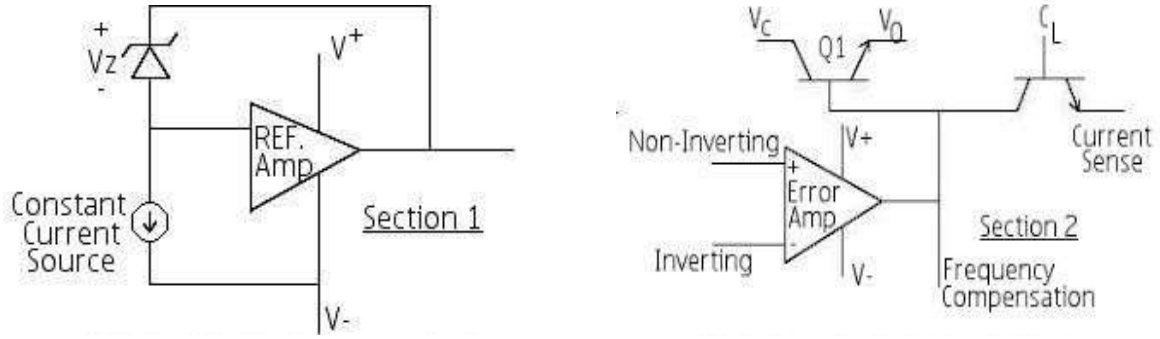
Low voltage regulator: A positive low voltage regulator using 723 is as shown. The voltage at NI terminal of the error amplifier due to R_1R_2 divider is $V_{IN}= V_{ref}(R_2/R_{1+}R)$. T he difference between V_{IN} and the output voltage V_0 is directly fed back to the INV terminal is amplified by the error amplifier. The output of the error amplifier drives the pass transistor Q_1 so as to minimize the difference the NI and INV input of error amplifier. Since Q_1 is operating as an emitter follower $V_0=V_{ref}(R_2/R_1+R_2)$. If the output voltage becomes low, the voltage at the INV terminal of the error amplifier also goes down. This makes the output of the error amplifier to become more positive, thereby driving transistor Q_1 more into conduction .This reduces the voltage across Q_1 and drive more current into the load causing the voltage across the load to increase. So the initial drop in the load voltage has been compensated. Similarly any increase in load voltage, or changes in the input voltage get regulated. The reference voltage typically 7.15volt, so the output voltage $V_0=7.15(R_2/R_1+R_2)$. This will be always being less than 7.15V.

High voltage regulator :

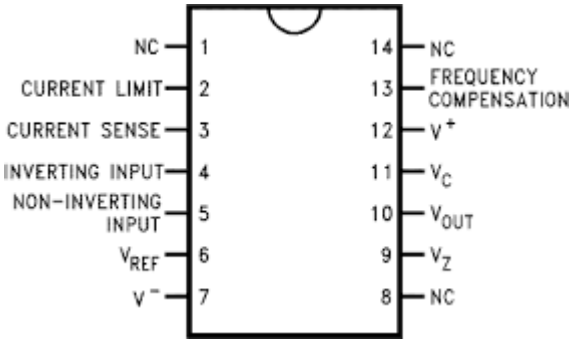
If it is desired to produce regulated output voltage greater than 7V, a small change should be made in the circuit for low voltage regulator. The non-inverting terminal is connected directly to V_{ref} through R3. So the voltage at the non-inverting terminal is V_{ref}. The error amplifier operates as a non-inverting amplifier with a voltage gain of $Av=1+R_1/R_2$. Notice that Av is always greater than 1. So the output voltage of the circuit is $Vo=7.15(1+ R_1/R_2)$.

Circuit diagram

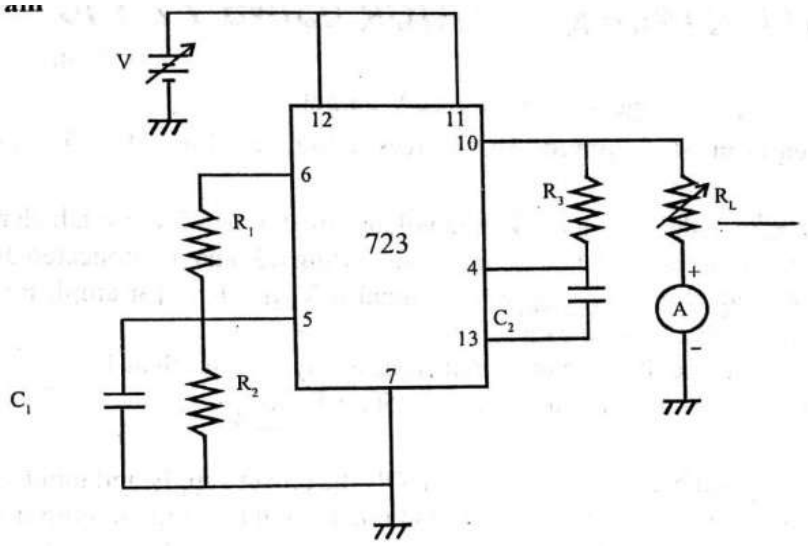
Functional block diagram of 723 regulator



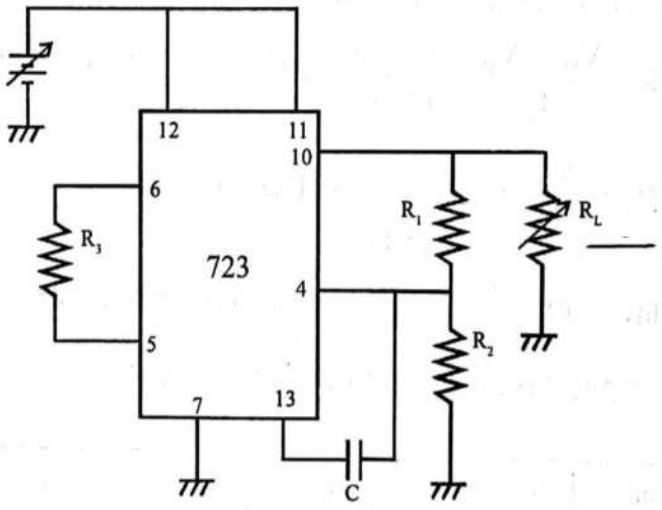
Pin diagram of IC 723



Low voltage regulator



High voltage regulator



Design

- Low voltage regulator

V_0 =
R2/(R1+R2) = …..V Vref = ….. V

R1=(Vref – V_0)/I_D = (7.15-6)/1mA =…..kΩ $V_{0=...}$V
R2=V_0/I_D =6V/1mA = ….kΩ ≈….kΩ I_D = ..mA
Take C = …..pF R3 = ……. kΩ

- High voltage Regulator

V_0 = 7.15(1 +(R1/ R2))
Take R1 = ….kΩ

1+(R1/R2)= 12/7.15 R2 =…. kΩ

Use R2= ….kΩ std
C= …..pF Take R_L= …..Ω rheostat

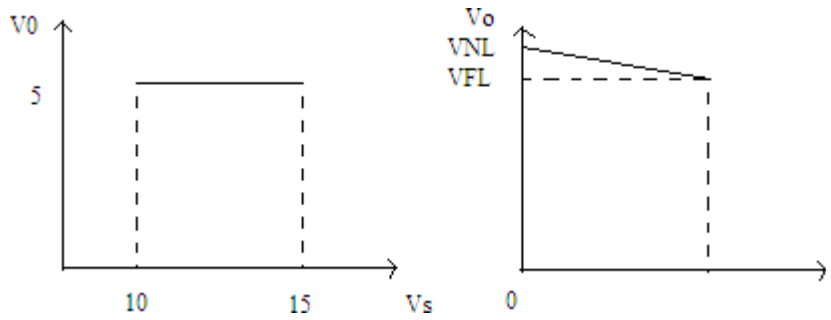
Low voltage regulator

Observations:

For line regulation	
Vs(V)	Vo(V)
10	
11	
12	
13	
14	
15	

For load regulation	
I_L(mA)	Vo(V)
0	
5	
10	
20	
40	
70	
80	
100	

Typical graph



Line regulation Load Regulation

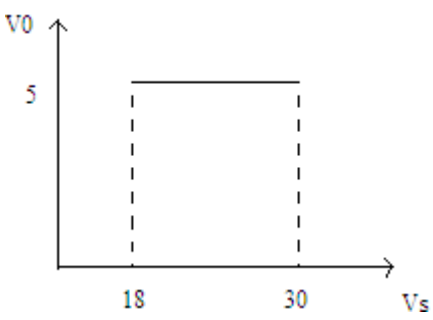
Low voltage regulator

Observations:

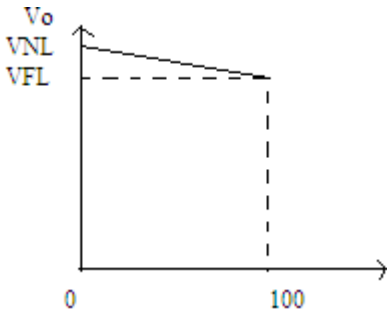
For line regulation	
Vs(V)	Vo(V)
18	
20	
22	
24	
28	
30	

For load regulation	
I_L(mA)	Vo(V)
0	
5	
10	
20	
40	
70	
80	
100	

Typical graph



Line regulation



Load Regulation

Procedure:

Low voltage regulator:

1. Set up the circuit. Switch on the power supply and input voltage sources.
2. Vary the input voltage from ….V to ….V and observe the output voltage. Note down it in tabular column.
3. Vary the rheostat and note the change in output current.
4. Draw the regulation characteristics with input on X-axis and output on Y-axis.
5. Calculate the % line regulation using the expression:
   S_V = change in output voltage / change in input voltage

6. Calculate the % load regulation using the
   expression: $S_L = (V_{NL} – V_F)/ V_{NL}$

High voltage regulator:

1. Set up the circuit. Switch on the power supply and input voltage sources.
2. Vary the input voltage from ….V to …..V and observe the output voltage. Note down it in tabular column.
3. Vary the rheostat and note the change in output current.
4. Draw the regulation characteristics with input on X-axis and output on Y-axis.

Result:

Designed the voltage regulator circuits using IC 723 Load regulation for

Low voltage regulator =……

High voltage regulator=……

EXP 14 **DATE**

A/D converters- counter ramp and flash type.

Aim: To design and setup a counter ramp and flash type ADC.

Components required: Op-amp, diode, resistors, capacitors, comparator LM311,IC's 7408,7493,741, breadboard, CRO, function generator and power supplies.

Theory:

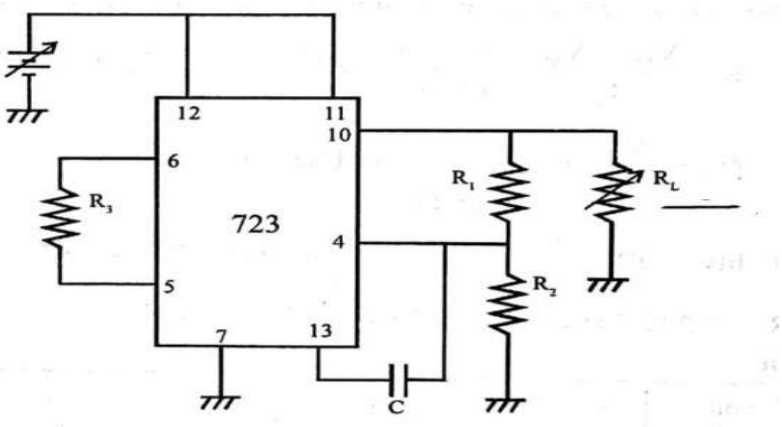
Counter ramp ADC: It displays the digital equivalent of input analog signal. Basically, a comparator opens a gate for a period of time and a counter counts the number of pulses flowing through the gate. Comparator keeps the gate open until the analog equivalent of the digital output of the counter equals the input voltage that to be digitized.

A four bit binary counter 7493 is used to count the pulses. An op amp with R-2R ladder network is used as a digital to analog converter. Comparator output provides high output as long as $V_{in} > V_{a.}$ V_{in} is the input to be digitized and $V_{a.}$ analog equivalent of the instantaneous digital output. When $V_{in} < V_{a}$, gate closes and pulses stop to flow to binary counter. Digital output remains standstill at its value.

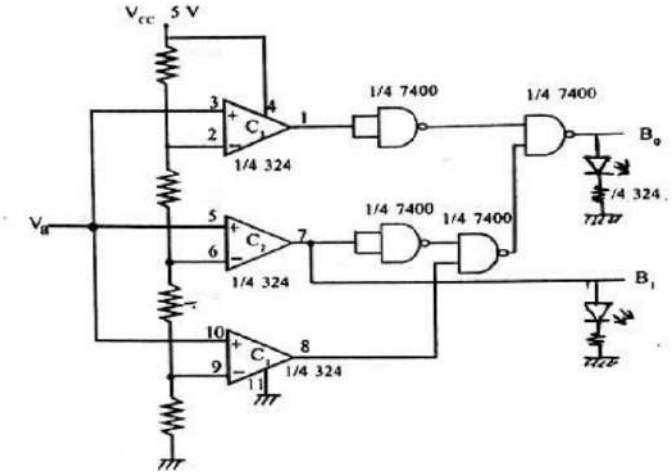
2-bit flash ADC: If the analog signal exceeds the reference signal to any comparator, that comparator turns on. If all comparators are off, analog input will be between 0 and +V/4.If C1 is high and C2 and C3 are low, input will be between +V/4 and +V/2.If C1 andC2 are high and C3 is low input will be between +V/2 and +3V/4. If all comparators are high , analog input will be between +3V/4 and +V. the outputs of three comparators are then fed to a coding network to provide 2 bits which are equivalent to the input analog voltage.

Circuit diagram:

Counter ramp ADC



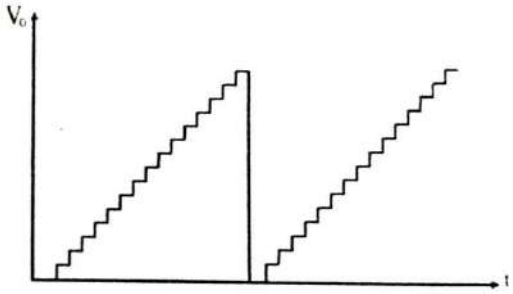
2 bit FLASH ADC



Tabular column

V (Volts)	S_3	S_2	S_1	S_0
	0	0	0	0
	0	0	0	1
	0	0	1	0
	0	0	1	1
	1	1	0	0
	1	1	0	1
	1	1	1	0
	1	1	1	1

Waveform



Procedure:

1. Set up the circuit for counter ramp ADC and …..bit flash ADC

2. Vary the analog input from …..V to …….V and observe the output bits.

Result:

Designed and setup the ADC circuits

.

EXP 15 **DATE**

D/A Converters- ladder circuit.

Aim:

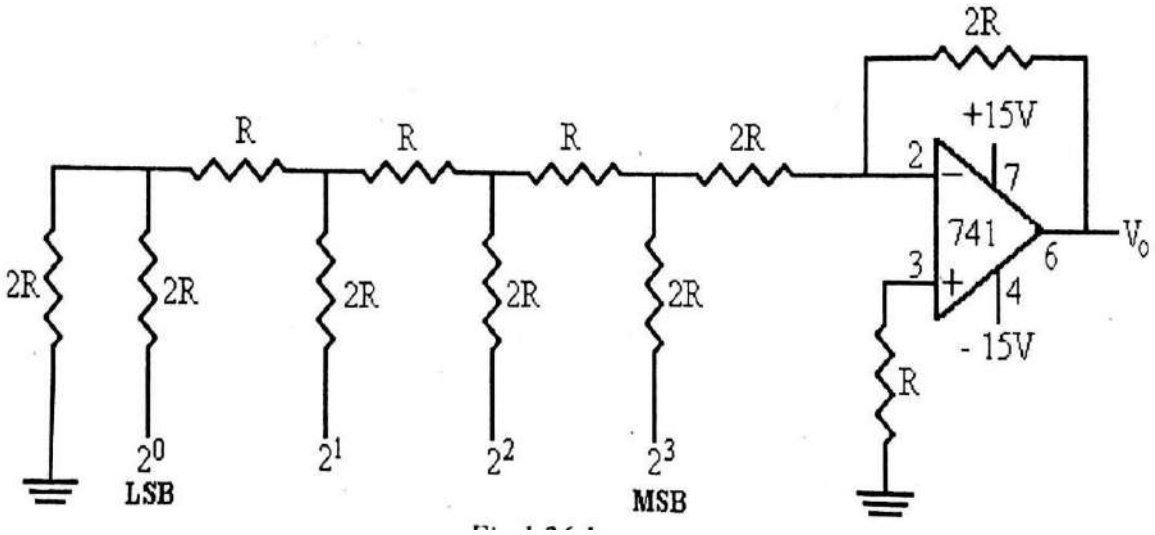
To design and set up a R-2R ladder type DAC.

Components required:

Op-amp, resistors, capacitors, breadboard, CRO, function generator and power supplies.

Theory: An R-2R ladder DAC uses fewer unique resistor values. Only two resistance values are used anywhere in the entire circuit. This means that only two values of resistance in the ratio 2:1.Current flowing through any input resistor (2R) encounters two possible paths at the far end. The effective resistances of both paths are the same, so the incoming current splits equally along both paths. The half current that flows back towards lower orders of magnitude does not reach the op amp, and therefore has no effect on the output voltage. The half that takes the path towards the op amp along the ladder can affect the output.

Circuit diagram:

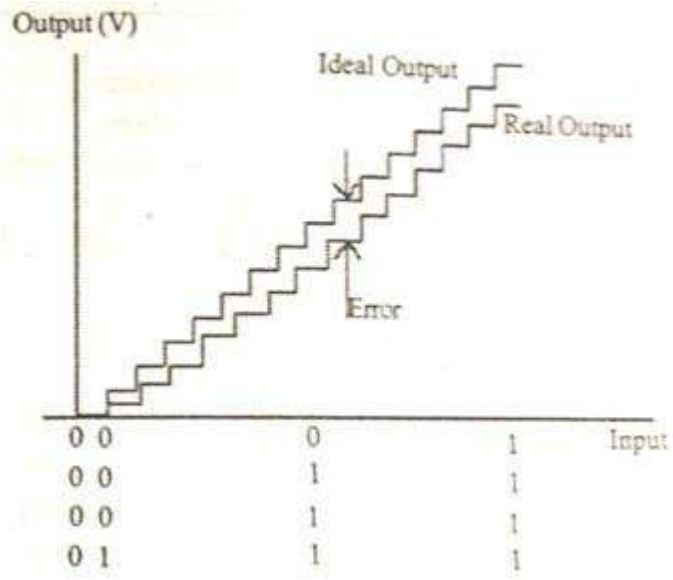


Design:

Let R=……KΩ & 2R=…..KΩ

Observations and typical response curve

$Q_3Q_2Q_1Q_0$	V0(Volts)
0000 0001 0010 …… …… 1101 1110 1111	



Procedure:

1. Verify the conditions of op-amp.

2. Set up the DAC circuit and manually enter binary inputs ….. to ….

3. Measure the output voltage using a multimeter and tabulate the readings.

4. Draw the response with analog output on Y-axis and binary output on X-axis.

Results:

Designed the ladder circuit DAC Error in the output.....%

EXP 16 **DATE**

Study of PLL IC: free running frequency lock range capture range

Aim: To design set up a PLL circuit and study its functional characteristics.

Components required: 565 PLL IC, Power Supply, Function generator, CRO, Resistors, Capacitors, Bread board

Theory: PLL is a control system that generates an output signal whose phase is related to the phase of input reference signal. It mainly consists of a phase detector, an LPF and a VCO. Phase comparator or phase detector compare the frequency of input signal f_s with frequency of VCO output f_o and it generates a signal which is function of difference between the phase of input signal and phase of feedback signal which is basically a d.c voltage mixed with high frequency noise. LPF remove high frequency noise voltage. Output is error voltage. If control voltage of VCO is 0, then frequency is center frequency (f_o) and mode is free running mode. Application of control voltage shifts the output frequency of VCO from f_o to f. On application of error voltage, difference between f_s & f tends to decrease and VCO is said to be locked. While in locked condition, the PLL tracks the changes of frequency of input signal.

Center frequency (free running frequency) $f_o = 1.2/4R_1C_1$ Hz

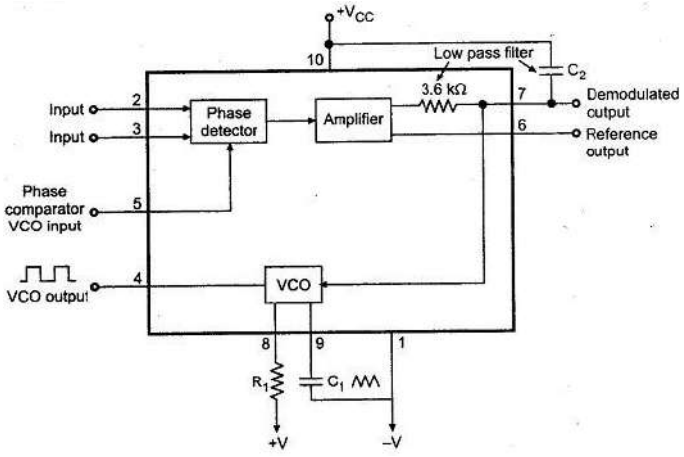
Lock range $f_L = \pm 8\, f_o/V$ Hz

$V = (+V) - (-V)$

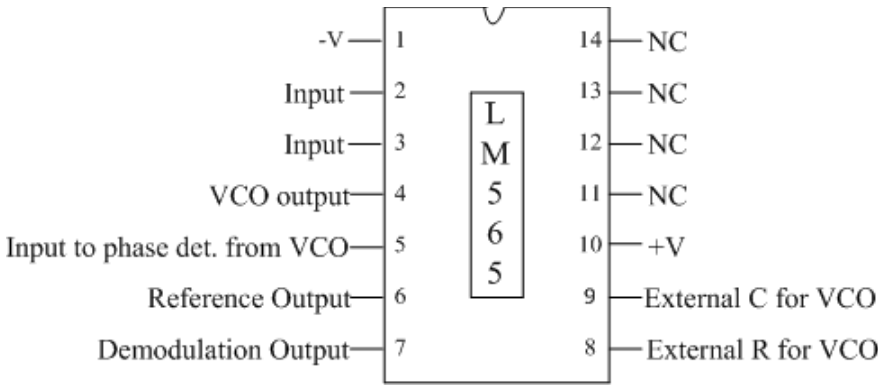
Capture range $f_c = \pm \left[\dfrac{f_L}{2\Pi(3.6)x10^3\, xC2} \right]^{1/2}$

Circuit diagram

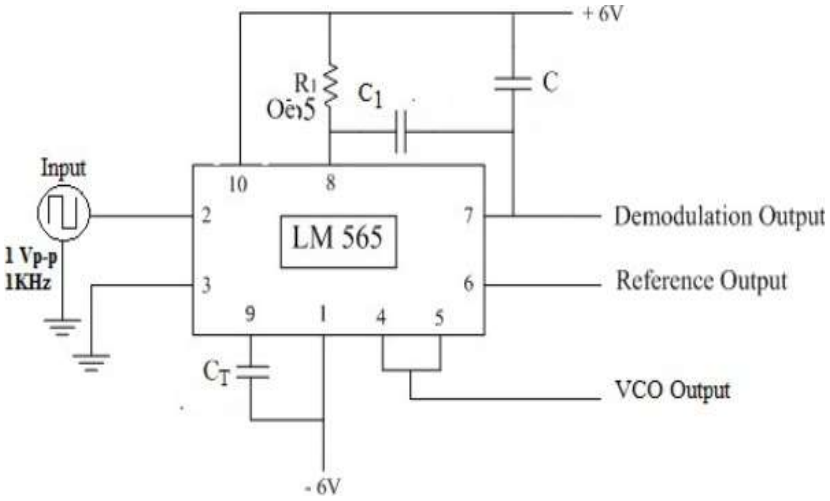
Block diagram of 565 PLL



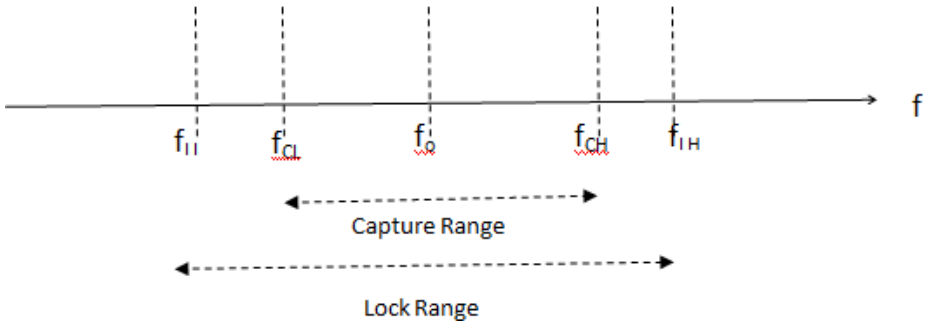
Pin out of 565



Circuit diagram of PLL



Graph



Design:

Take V+=…..V and V-=…..V

Let the free running frequency f_0 be 2.5KHz=$1.2/4R_1C_1$

Take C_1=…..µF. Then R_1=…..K.

The value of R_1 satisfies the required condition $2k < R_2$

Take C_3 =….µF C_2=…µF Then the theoretical values of f_L and f_C

$f_L=\pm 8*2.5*10^3/10-(-10)$ =….KHz

$f_c=\sqrt{(10^3)}/\sqrt{(2\pi*3.6*10^3*10*10^{-6})}$ = …… Hz

Procedure:

1. Verify the condition of components.
2. Set up the circuit and observe the output waveform and note down the VCO frequency.
3. Feed a square wave to the pin no.2 of 565 PLL IC and vary its frequency from …..Hz to …… MHz and note down f_{C1} and f_{L2}. Then decrease the frequency from …..MHz to …..Hz note down f_{C2} and f_{L1}
4. Calculate capture range and lock range.

Results: Designed the PLL IC and obtain the results.

Free running frequency:……

Capture range :……..

Lock range :…….