PEC – 451

ANALOG INTEGRATED CIRCUITS

Laboratory Manual

For

Electronics & Communication Engineering

2ND Year Students

FOREWORD

It is my great pleasure to present this laboratory manual for second year engineering students for the subject of Analog Integrated Circuits keeping in view the vast coverage required for visualization of concepts of Analog Integrated circuit.

As a student, many of you may be wondering with some of the questions in your mind regarding the subject and exactly that has been tried to answer through this manual.

Faculty members are also advised that covering these aspects in initial stage itself will greatly relieve them in future, as much of the load will be taken care by the enthusiastic energies of the students, once they are conceptually clear.

Ajay Kumar Gautam
(Asst. Prof.)

LABORATORY MANUAL CONTENTS

This manual is intended for the Second year students of Electronics & Communication branch in the subject of Analog Integrated Circuit. This manual typically contains Practical/Lab Sessions related to Analog Integrated Circuit covering various aspects related the subject to enhance understanding of the subject.

Students are advised to thoroughly go through this manual rather than only topics mentioned in the syllabus, as practical aspects are the key to understanding conceptual visualization of theoretical aspects covered in the books.

Good Luck for your Enjoyable Laboratory Sessions

Ajay Kumar Gautam

(Asst. Prof.)

Requirements to complete a laboratory

Each laboratory will consist of three parts:

1. Laboratory preparation and home exercises (expected to be completed before coming to the laboratory).
2. Execution of the laboratory tasks.
3. Submission of a complete laboratory report.

Notice: A laboratory is not approved until all three requirements are fulfilled. A laboratory report is to be submitted one week after the laboratory was performed.

SUBJECT INDEX

1. Do’s and Don’ts
2. Lab Exercise:
3. Quiz on the subject
4. Conduction of Viva Voce Examination
5. Evaluation and Marking Systems

Dos and Don't s in Laboratory

1. Do not handle any equipment before reading the instructions/Instruction manuals.
2. Do not apply voltage more than 15 V to IC 741.
3. Check CRO probe before connecting it.
4. Strictly observe the instructions given by the teacher/Lab Instructor.
5. Good English and good technical communications are necessary and will be taken into account in grading.

Experiment No. XX

OBJECTIVE

State what you understand to be the objective for the experiment. Usually, it is easier to appreciate the objective of the experiment after the whole experiment has been performed and the results analyzed.

An example objective for Experiment 1 might read as follows: *To study Square wave generator circuit using op-amp 741*.

STATEMENT

See design statement in Lab Manual

APPARATUS REQUIRED

See requirements in Lab Manual

DESIGN

- Show circuit used and all components and their values on.
- Give a brief description of the operation of the circuit.
- Show design equations and calculated values of circuit components (or other parameters).

EXPERIMENTAL RESULTS/ANALYSIS

- State what was done, for example, data or graphs obtained, analysis performed.
- Give equations and sample of calculations used.
- Give data, graphs, and experimental results, using tables whenever possible.
- Carry out any specific instructions and respond to any questions asked.

DISCUSSION

- Describe any unexpected results, which are different from theoretical expectations.
- Try your best to explain possible causes for any discrepancies.

CONCLUSSION

- Indicate how well the experiment conformed to expectations, which tasks could not be completed and why, and how well the experimental objectives were achieved.
- Give some indication of what you learnt from the experiment.

NOTES

- The Report must be written in good English.
- Some of the issues in the sections listed above may not be relevant in some experiments.

PEC 451 ANALOG INTEGRATED CIRCUIT LAB

Objective: - To design and implement the circuits to gain knowledge on performance of the circuit and its application. These circuits should also be simulated on Pspice.

1. Log and antilog amplifiers.
2. Voltage comparator and zero crossing detectors.
3. Second order filters using operational amplifier for–
   - Low pass filter of cutoff frequency 1 KHz.
   - High pass filter of frequency 12 KHz.
   - Band pass filter with unit gain of pass band from 1 KHz to 12 KHz.
4. Wien bridge oscillator using operational amplifier.
5. Determine capture range; lock in range and free running frequency of PLL.
6. Voltage regulator using operational amplifier to produce output of 12V with maximum load current of 50 mA.
7. Voltage to current and current to voltage convertors.
8. Function generator using operational amplifier (sine, triangular & square wave)
9. Astable and monostable multiviberator using IC 555

Experiment No. :1

OBJECTIVE: To Design a Low pass, first order Butterworth Filter with a cut-off frequency of f_H=1.0 kHz.

EQUIPMENTS AND COMPONENTS:

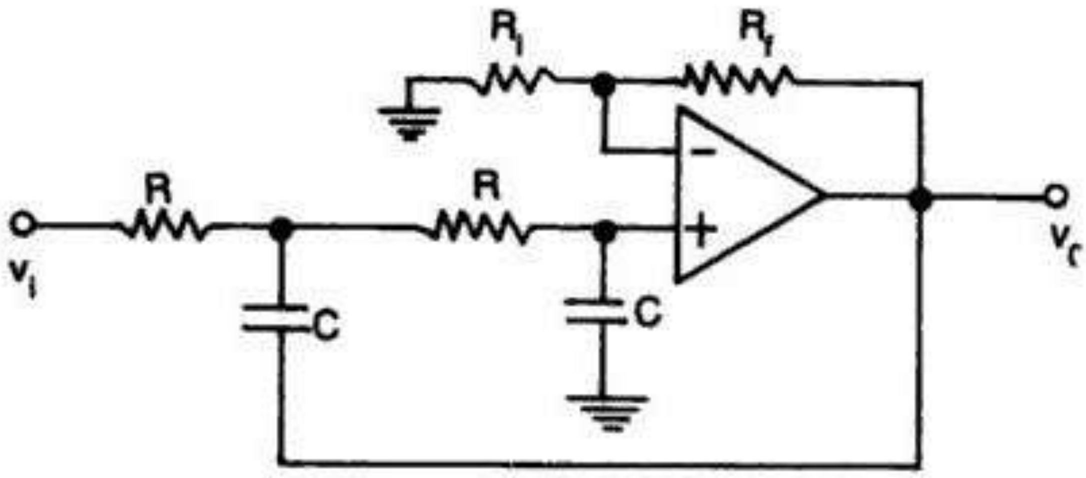
1. CRO (Dual trace) - 1 No
2. Signal Generator - 1 No
3. Bread Board - 1 No.
4. dual channel power supply- 1 No

0 to 20 volts peak to peak independently.

THEORY:

A frequency selective electric circuit that passes electric signals of specified band of frequencies and attenuates the signals of frequencies outside the brand is called an electric filter. The first order low pass filter consists of a single RC network connected to the non-inverting input terminal of the operational amplifier. Resisters R1 and R_F determine the gain of the filter in the pass band. The low pass filter as maximum gain at f = 0 Hz. The frequency range from 0 to F_H is called the pass band the frequency range f > fh is called the stop band.

CIRCUIT DIAGRAM:



PROCEDURE:

1. Construct the circuit as shown in circuit diagram.
2. Apply an input sine wave and measure the amplitude of output waveform for different values of input frequencies.
3. Calculate the gain in dB.
4. Plot the frequency response.

OBSERVATIONS:

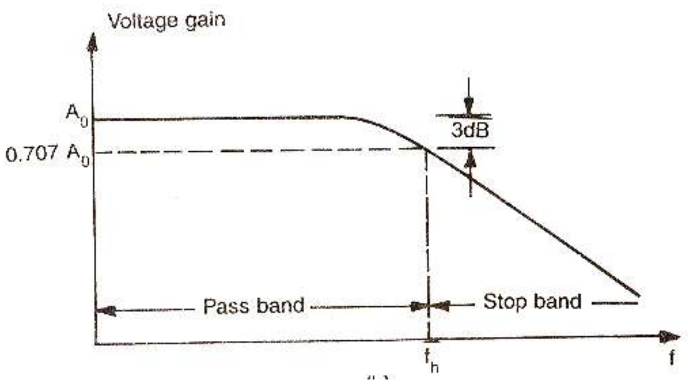
Sl. No	Input frequency	Output voltage	Gain	20 log Vo/Vi
1.	100			
2	200			
3	500			
4.	1kHz			
5.	1.5KHz			
6.	2.0kHz			
7.	5kHz			

CALCULATIONS:

i. $f_H = \frac{1}{2\pi RC}$

ii. $A_o = 1 + \frac{R_F}{R_1}$

GRAPH:



RESULT:

i. The cut-off frequency of the low pass filter = kHz

ii.The pass band gain of low pass filter =

CONCLUSION:

i. The working of active low pass filter is observed and the output is plotted.

ii. The frequency response of the low pass filter is plotted on a semi-log graph paper.

iii. It is observed that the gain rolls of at the rate of 20dB per decade at the cut of frequency.

REVIEW QUESTIONS

i.Define an electric filter.

ii.Classify filters

iii.Discuss the disadvantages of passive filters

iv.Why are active filters preferred?

v.List the commonly used filters.

vi.Define pass band and stop band of a filter.

vii.What is roll-off rate of a first order filter?

viii.Why do we use higher order filters?

ix.On what does the damping coefficient of a filter depend?

x.What is sallen key filter?

EXPERIMENT NO:2

OBJECTIVE: To Design a High pass, first order Butterworth Filter with a cut-off frequency of f_L=1.0 kHz..

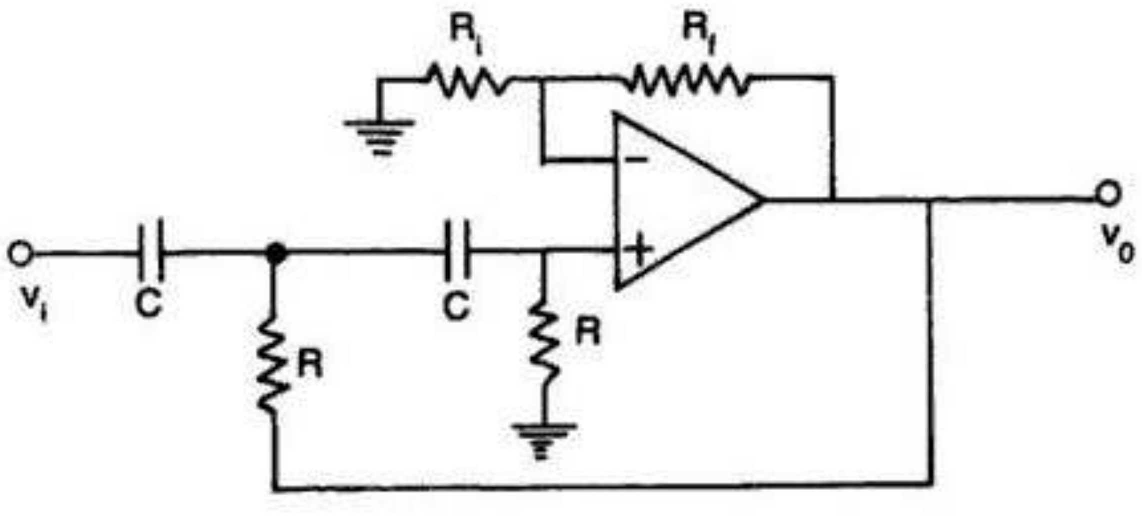
EQUIPMENTS AND COMPONENTS APPARATUS:

1. CRO (Dual channel) - 1 No
2. Signal Generator - 1 No
3. Bread Board - 1 No.
4. Dual channel power supply –1 No

III. THEORY:

A frequency selective electric circuit that passes electric signals of specified band of frequencies and attenuates the signals of frequencies outside the brand is called an electric filter. The first order high pass filter consists of a single RC network connected to the non-inverting input terminal of the operational amplifier. Resisters R1 and R_F determine the gain of the filter in the pass band. The high pass filter has maximum gain at f = f_l Hz. The frequency range from 0 to F_l is called the stop band the frequency range f > fl is called the pass band.

CIRCUIT DIAGRAM:



PROCEDURE:

i. Construct the circuit as shown in circuit diagram.

ii.Apply an input sine wave and measure the amplitude of output waveform for different values of input frequencies.

iii.Calculate the gain in dB.

iv.Plot the frequency response.

OBSERVATIONS:

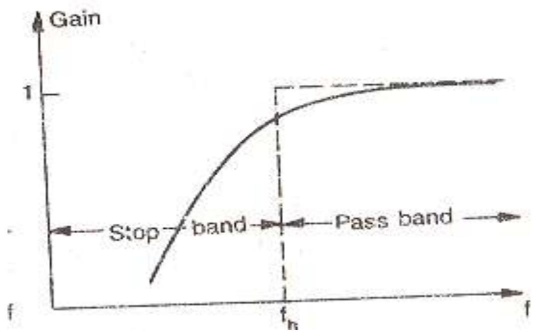
Sl.No	Input frequency	Output voltage	Gain	20 log Vo/Vi
1.	100			
2	200			
3	500			
4.	1kHz			
5.	1.5KHz			
6.	2.0kHz			
7.	5kHz			

CALCULATIONS:

i. $f_H = \frac{1}{2\pi RC}$

ii. $A_o=1+\frac{RF}{R_1}$

GRAPH:



RESULT:

The lower cutoff frequency of the high-pass filter = ------- KHz.
The pass band gain = -------

CONCLUSION:

i. The working of active high pass filter is observed and the output is plotted.
ii. The frequency response of the high pass filter is plotted on a semi-log graph paper.
iii. It is observed that the gain increases at the rate of 20dB per decade at the cut of frequency.

REVIEW QUESTIONS

i.Define Bessel , Butterworth and Chebyshev filters, and compare their response.
ii.What are the important parameters of a band pass filter?
iii.Define Notch filter.
iv.How do we get a notch filter from a band pass filter?
v.Define state variable filter.
vi.What is switched capacitor ?
vii.Discuss the importance of switched capacitors.
viii.Give the circuit of a switched capacitor low pass filter
ix.Discuss the advantages of active filters
x.What is the roll-off rate of second order filter?

WEIN BRIDGE OSCILLATOR

AIM: To Design Wein Bridge Oscillator so that the output frequency is 965 Hz.

II EQUIPMENTS AND COMPONENTS:

(i).APPARATUS

1. CRO (Dual channel) - 1 No
2. Bread Board – 1 No
3. Dual Channel Power Supply – 1 No

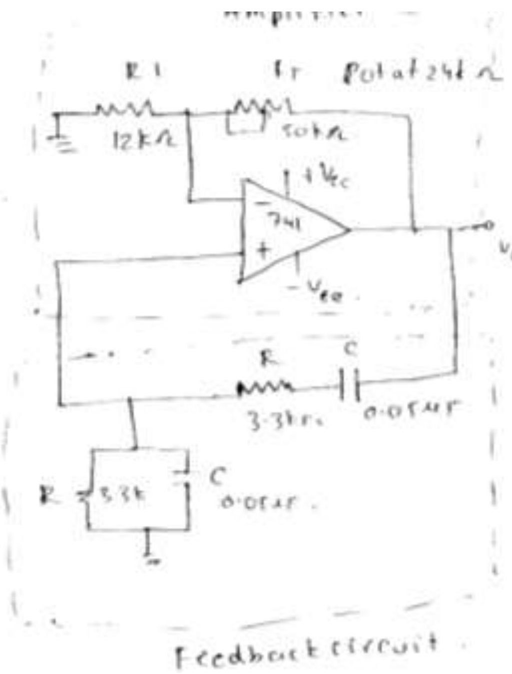
3. COMPONENTS:

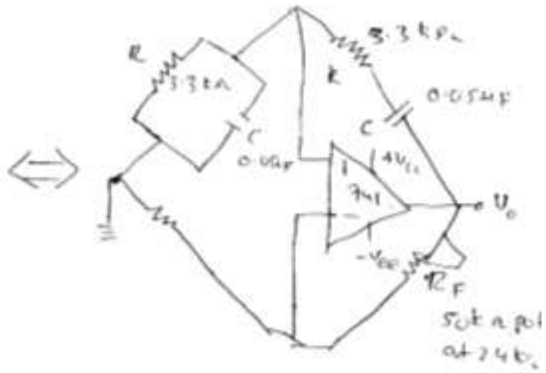
1. 12kΩ Resistor – 1 No.
2. 50kΩ Resistor – 1 No.
3. 3.3kΩ Resistor – 1 No.
4. 0.05μF Capacitor – 1 No.
5. Operational Amplifier – 1 No.

III. THEORY:

In this oscillator the Wein Bridge Circuit is connected between the amplifier input terminals and the output terminal. The bridge has a series RC network in one arm and parallel RC network in the adjoining arm. In the remaining two arms of the bridge resistors R1 and RF are connected. The total phase-shift around the circuit is 0^o when the bridge is balanced.

IV. CIRCUIT DIAGRAM:





V. PROCEDURE:

i. Construct the circuit as shown in the circuit diagram.

ii. Adjust the potentiometer R_f such that an output wave form is obtained.

iii. Calculate the output wave form frequency and peak to peak voltage.

iv. Compare the theoretical and practical values of the output waveform frequency.

VI. OBSERVATIONS:

The frequency of oscillation = ______

VII. CALCULATIONS:

The frequency of oscillation f_o is exactly the resonant frequency of the balanced Wein Bridge and is given by $f_o = 1/(2\pi RC)$

$= 0.159 / RC$

The gain required for sustained oscillations is given by $A_v = 3$. i.e., $R_f = 2R_1$

Let $C = 0.05\ \mu F$

Then $f_o = 1/(2\pi RC)$

→ $R = 1/(2\pi f_o C)$

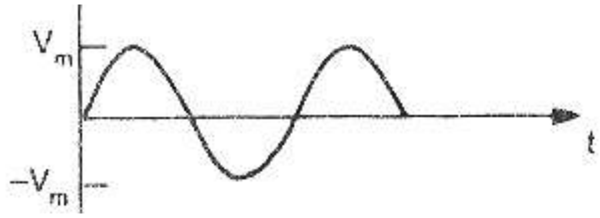
→ $= 3.3\ k\Omega$

Now let $R_1 = 12\ k\Omega$

Then $R_f = 2R_1 = 24\ k\Omega$

Use $R_f = 50\ k\Omega$ potential meter.

VIII. GRAPH:



IX: RESULT:

The frequency of oscillation of the Wein Bridge oscillator =----------

X: INFERENCE:

i. The working of Wein Bridge oscillator is observed and the output is plotted.
ii. The frequency response of the Wein Bridge oscillator is plotted
iii. It is observed that the gain doesn't sustain beyond 5 KHz

i. Design the circuit for different frequencies of oscillations.

XV. REVIEW QUESTIONS

i.State the two condition of oscillations
ii.Classify the oscillators
iii. What is the phase shift in case of the phase shift oscillator?
iv.Explain how to measure the phase difference of two signals
v.In WEIN BRIDGE oscillator what phase shift does the opamp provide?
vi. In what mode the opamp is used in the wein bridge oscillator?
vii.What phase shift is provided by the feedback network?
viii.What is the minimum gain that the inverting opamp should have?
ix.For high frequencies which kind of opamp should be used?

x.What is the condition for so that the oscillations will not die out?

IC 723 VOLTAGE REGULATOR

EXPERIMENT NO:5

I AIM:

To study the operation of IC723 voltage regulator

II EQUIPMENTS AND COMPONENTS:

(i).APPARATUS

1. DC power supply - 1 No.
2. Digital Multimeter - 1 No.
3. Ammeter - 1 No.
4. Bread Board - 1 No.

.

3. COMPONENTS:

1. 1kΩ Resistor – 1 No.
2. 33Ω Resistor – 1 No.
3. 10k Ω Resistor – 1 No
4. 680 Ω Resistor – 1 No.
5. 2.2 kΩ Resistor – 1 No.
6. 100 pF Capacitor – 1 No
7. IC723

III. THEORY:

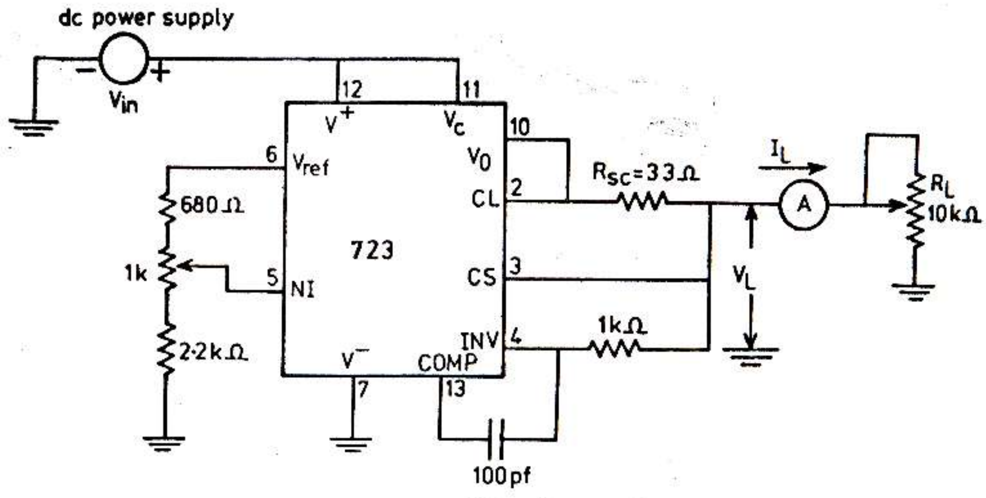
The three-terminal regulators have the following limitations

1. No short-circuit protection
2. Output voltage (+ve or –ve) is fixed

These limitations have been overcome in 723 general purpose regulator. This IC is inherently low current device but can be boosted to provide 5 amps or more current by connecting external components. The limitation of 723 is that it has no in-built thermal protection. It also has no short-circuit current limits. The IC723 has two sections. The first section consists of Zener Diode constant current source and a reference amplifier. The other section of the IC consists of an error amplifier series pass transistor and a current limit transistor. This is a 14-pin DIP package. The main

Features of 723 include an input voltage of 40v max, output voltage is adjustable from 2V to 37V, 150 mA output current without external pass resistor, can be used as either a linear or a switching regulator.

IV. CIRCUIT DIAGRAM:



723 voltage regulator

V. PROCEDURE:

1. Connect the 723 regulator as shown in the circuit diagram
2. Set Dc power supply voltage V_{in} to +10V measure and record V_{ref} with respect to ground. With load R_L(10kΩ pot) removed from the circuit (output open). Measure the minimum and maximum output voltage by rotating the 1kΩ pot through its full range.
3. Now adjust the 1kΩ pot so that V_o is +5V dc. Measure the voltage between the wiper arm of the 1 kΩ pot and ground.
4. Adjust the load R_L (10 kΩ) pot until the load current I_L = 1 mA. Record V_L. Repeat for different values of load currents 5mA, 10mA, 15mA, 18mA. Calculate the load regulation and compare with manufacturer's specifications
5. Gradually increase the load current above 18mA, you will see that the load voltage suddenly decreases when the load current is about 18 to 20 mA. Now the voltage across R_{SC} is enough to begin current limiting. Measure and record a few values of load current and load voltage below and above the current limiting point. Plot a graph of V_L vs I_L from the data obtained in steps 4 and 5

VI. OBSERVATIONS:

1. The load regulation = _____ %
2. The line regulation = ______%

VII. CALCULATIONS:

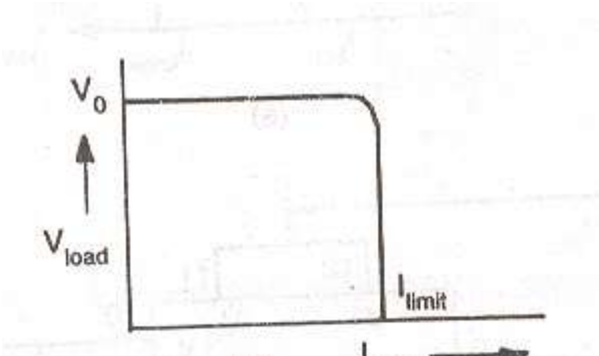
1. The load regulation can be calculated by using the below formulae

%load regulation = $((V_{fl} - V_{nl}) / (V_{fl})) * 100$

2. The line regulation can be calculated by using the below formulae

%line regulation = $(\Delta V_o / \Delta V_i)$

VIII. GRAPH:



(a)

IX: RESULT:

i.The % load regulation =
ii. The % line regulation =

X: INFERENCE:

i. The working of 723 regulator is observed and the output is plotted.
ii. The load regulation is calculated
iii. The line regulation is calculated

XV REVIEW QUESTIONS

i.What is the maximum input voltage that we can give to 723 regulator?
ii.What output voltage range we can obtain from 723 regulator?
iii.What is the output current in case of 723 regulator?
iv.What are the applications of 723 regulator?
v.Define line regulation
vi.Define load regulation
vii.Define ripple rejection
viii.Define long term stability
ix.What is the current limit protection
x.What are the ideal values of load and line regulations

6. OP-AMP 741 AS DIFFRENTIATOR AND INTEGRATOR

EXPERIMENT NO:6

I AIM:

To design and test an op-amp differentiator and integrator

II EQUIPMENTS AND COMPONENTS:

(i).APPARATUS

1. DC power supply	-	1 No.
2. CRO	-	1 No.
3. Bread Board	-	1 No.
4. Function Generator	-	1 No.

3. COMPONENTS:

1. 15 kΩ Resistor – 2 No.
2. 820 Ω Resistor – 1 No.
3. 1.5 kΩ Resistor – 1 No.
4 0.01 μF Capacitor – 2 No
5 0.5 nF Capacitor – 1 No
5 IC741 - 1 No.

III. THEORY

The operational amplifier can e used in many applications. It can be used as differentiator and integrator. In differentiator the circuit performs the mathematical operation of differentiation that is the output waveform is the derivative of the input wave form for good differentiation, one must ensure that he time period of the input signal is larger than or equal to RfC1.the practical differentiator eliminates the problem of instability and high frequency noise.

IV. CIRCUIT DIAGRAM:

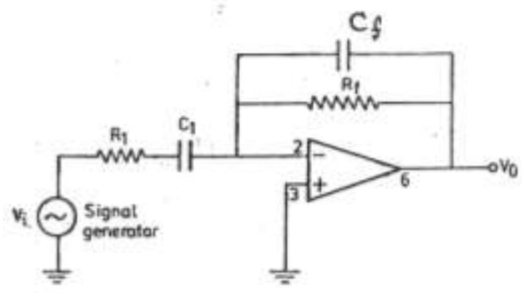


Fig 1 differentiator

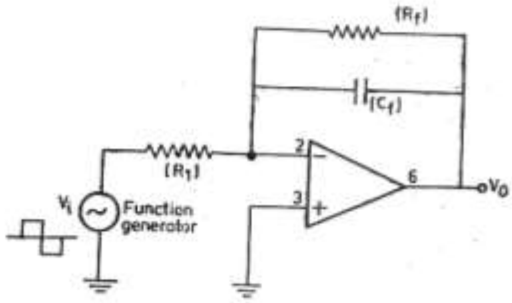


Fig. 2 integrator

V. PROCEDURE:

1 connect the differentiator circuit as shown in fig 1.adjust the signal generator to produce a 5 volt peak sine wave at 100 Hz.

2 observe input Vi and Vo simultaneously on the oscilloscope measure and record the peak value of Vo and the phase angle of Vo with respect to Vi.

3.Repeat step 2 while increasing the frequency of the input signal. Find the maximum frequency at which circuit offers differentiation. Compare it with the calculated value of f_a Observe & sketch the input and output for square wave.

4. Connect the integrator circuit shown in Fig2. Set the function generator to produce a square wave of 1V peak-to-peak amplitude at 500 Hz. View simultaneously output V_o and $V_{i.}$

5. Slowly adjust the input frequency until the output is good triangular waveform. Measure the amplitude and frequency of the input and output waveforms.

6. Verify the following relationship between R_1C_f and input frequency for good integration $f>f_a$ & $T< R_1C_1$

Where R_1C_f is the time constant

7. Now set the function generator to a sine wave of 1 V peak-to-peak and frequency 500 Hz. Adjust the frequency of the input until the output is a negative going cosine wave. Measure the frequency and amplitude of the input and output waveforms.

VI. OBSERVATIONS:

1. The time period and amplitude of the output waveform of differentiator circuit
2. The time period and amplitude of the integrator waveform

VII. CALCULATIONS:

Design a differentiator to differentiate an input signal that varies in frequency from 10 Hz to 1 kHz.

$$f_a = \frac{1}{2\pi R_f C_1}$$

f_a = 1 kHz, the highest frequency of the input signal

Let C_1 = 0.01 μF,
Then R_f = 15.9 kΩ
Therefore choose R_f = 15.0 kΩ

$$f_a = \frac{1}{2\pi R_1 C_1}$$

Choose: f_b= 20x f_a = 20 KHz
Hence R_1 = 795 Ω
Therefore choose R_1 = 820 Ω
Since $R_1C_1 = R_f C_f$ (compensated attenuator)
C_f = 0.54 nF
Therefore choose C_f = 0.5nF

Integrator: Design an integrator that integrates a signal whose frequencies are between 1 KHz and 10 KHz

$$f_b = \frac{1}{2\pi R_1 C_f}$$

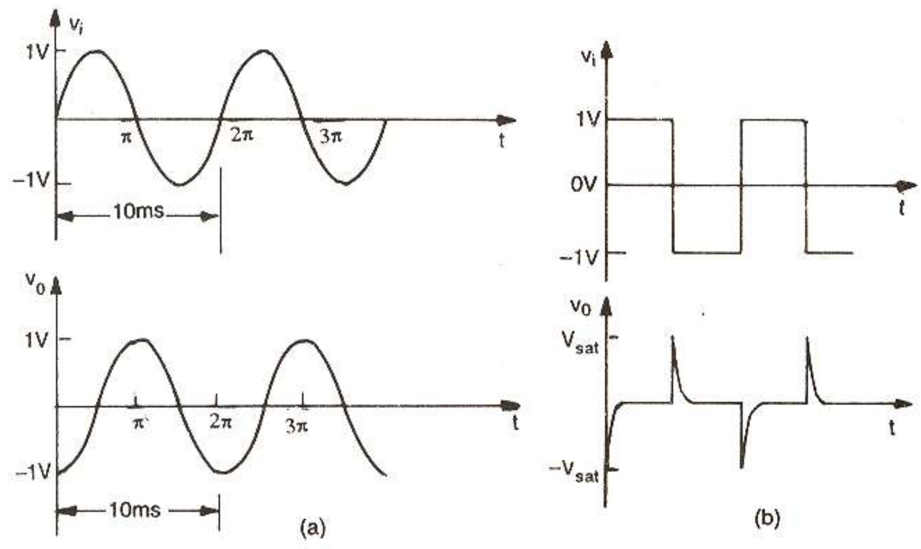
the frequency at which the gain is 0 dB.

$$f_a = \frac{1}{2\pi R_f C_f}$$

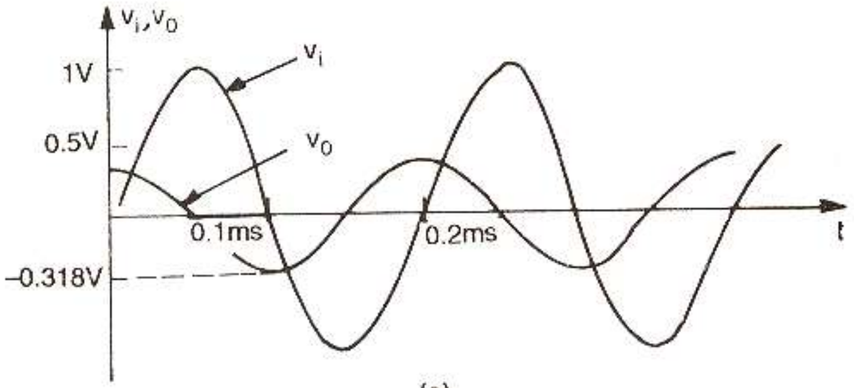
f_a: Gain limiting frequency,
The circuit acts as integrator for frequencies between f_a and f_b
Generally $f_a < f_b$ [Ref. Frequency response of the integrator]
Therefore choose f_a = 1KHz
F_b = 10 KHz
Let C_f = 0.01 μF
Therefore R_1= 1.59kΩ
Choose R_1= 1.5 KΩ
R_f = 15 KΩ

VIII. GRAPH:

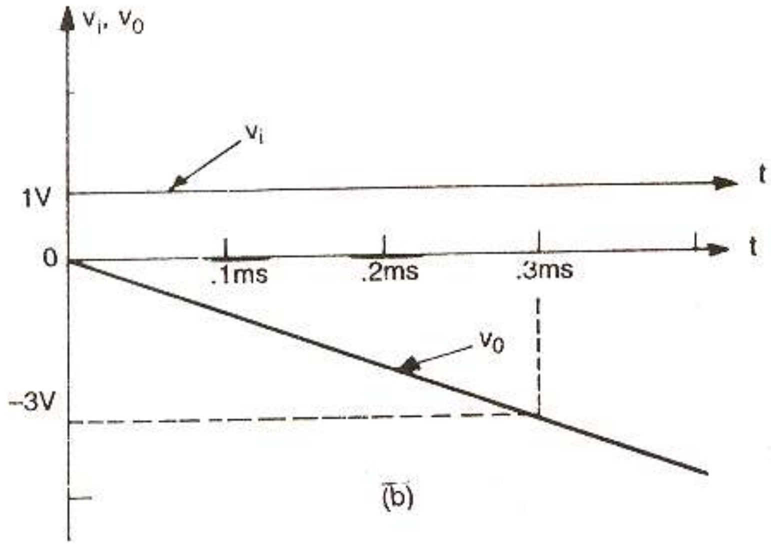
Differentiator



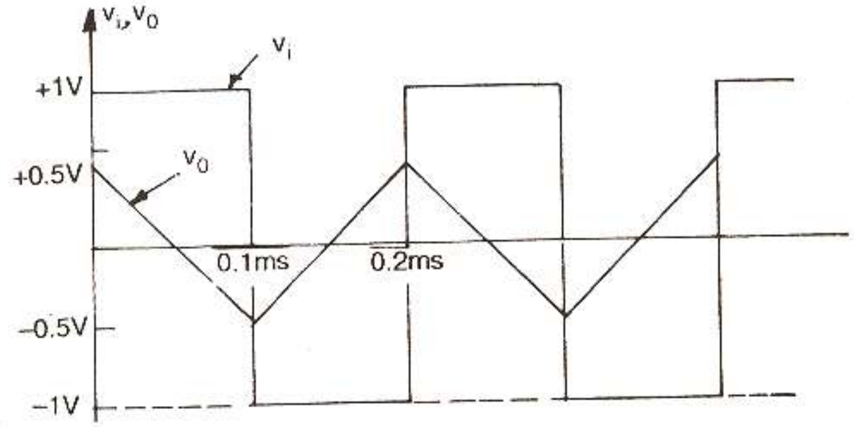
Integrator



(a)



(b)



(c)

IX: RESULT:

Differentiator

$$f_b = \frac{1}{2\pi R_1 C_f} = \_\_\_\_\_\_$$

T> $R_f C_1$ = ________

Integrator

$$f_a = \frac{1}{2\pi R_f C_f} = \_\_\_\_\_\_$$

T = _________

X: INFERENCE:

i. The working of differentiator and integrator is observed and the output is plotted.
ii. The time period of the output waveform is calculated
iii. The maximum frequency of differentiation and integration is observed

XV REVIEW QUESTIONS

i.Define differentiator
ii.Define integrator
iii.What are the limitations of an ordinary differentiator?
iv. Explain how the practical differentiator will overcome the limitations
v.What are the limitations of an ideal integrator?
vi.What are the initial conditions of a loss integrator?
vii.What are the differences between integrator and differentiator
viii.State the applications of integrator
ix.State the applications of differentiator
x.Explain why integrators are preferred over differentiators in analog computer.

7. 555 TIMER-ASTABLE MULTI-VIBRATOR

EXPERIMENT NO:7

I AIM:

To design and test astable multi-vibrator with fixed and adjustable duty cycle.

II EQUIPMENTS AND COMPONENTS:

(i).APPARATUS

1. DC power supply - 1 No.
2. CRO - 1 No.
3. Bread Board - 1 No.

3. COMPONENTS:

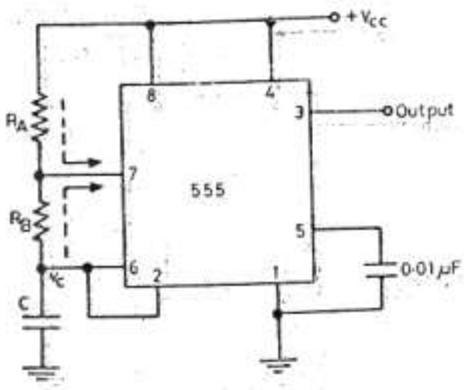
1. 3.62 kΩ Resistor – 1 No.
2. 7kΩ Resistor – 1 No.
3. 0.1 μF Capacitor – 1 No
4. IC555 - 1 No.
5. Diode – 1 No.

III. THEORY:

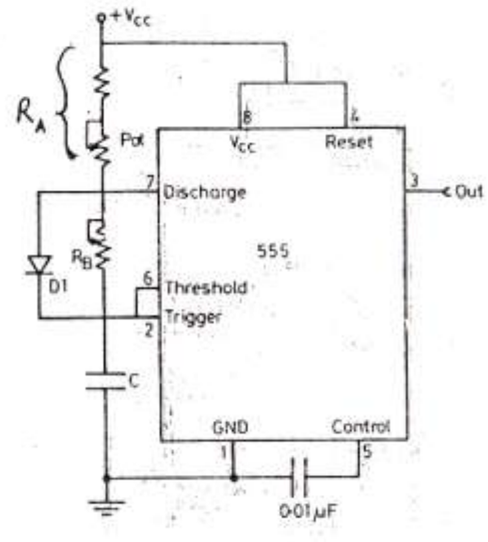
The 555 timer can be used with supply voltage in the range of + 5 v to + 18 v and can drive upto 200 mAmps. It is compatible with both TTL and CMOS logic circuits because of the wide range of supply voltage the 555 timer is versatile and easy to use in the astable multivibrator. The timer is oscillated between two threshold levels 1/3 V_{cc} and 2/3 V_{cc} in order to generate a square wave form. No external signal source is required for such generation and hence this is called as a free running multivibrator.

IV. CIRCUIT DIAGRAM:

CIRCUIT DIAGRAMS



Astable multivibrator using 555 timer



Adjustable duty cycle rectangular wave generator

V. PROCEDURE:

i.Connect the circuit using the component values as per the design of part – i

ii.Observe and sketch the capacitor voltage wave form (pin-6) and output waveform (pin-3) measure the frequency and duty cycle of the output wave form

iii.Connect the circuit of fig2 using component values as per the design of part – ii and repeat the step 2 by adjusting both the potential meters for duty cycle of 10%, 50% and 90% with a frequency of 1kHz.

VI. OBSERVATIONS:

1.t_{ON} = 69 ($R_A + R_B$) C

2.t_{OFF} = 0.69 R_BC

3. $$\%Dutycycle = \frac{t_{ON}}{t_{ON} + t_{OFF}} x100$$

VII. CALCULATIONS:

i. Design of astable multi-vibrator of 1 kHz with a 75% duty cycle

t_{ON} = 69 ($R_A + R_B$) C

t_{OFF} = 0.69 R_BC

$$\%Dutycycle = \frac{t_{ON}}{t_{ON} + t_{OFF}} x100 = \frac{t_{ON}}{T} x100$$

Where T = 1/f, the time period of the output waveform

T= 1msec, t_{ON} = 0.75 m/sec, t_{OFF} 0.25msec

Lec C = 0.1 µF

→ R_B = 3.62 k Ω

Choose R_B = 1.8 k Ω + 1.8 kΩ

$$f = \frac{1}{T} = \frac{1.45}{(R_A + 2R_B)c}$$ → Eq. (1)

ii. Design of astable multivibrator to produce 1 KHz output waveform with adjustable duty cycle of 10% to 90%

The circuit is as shown in Fig. 2

During the charging period, the diode 'D' is forward biased, R_B is bypassed

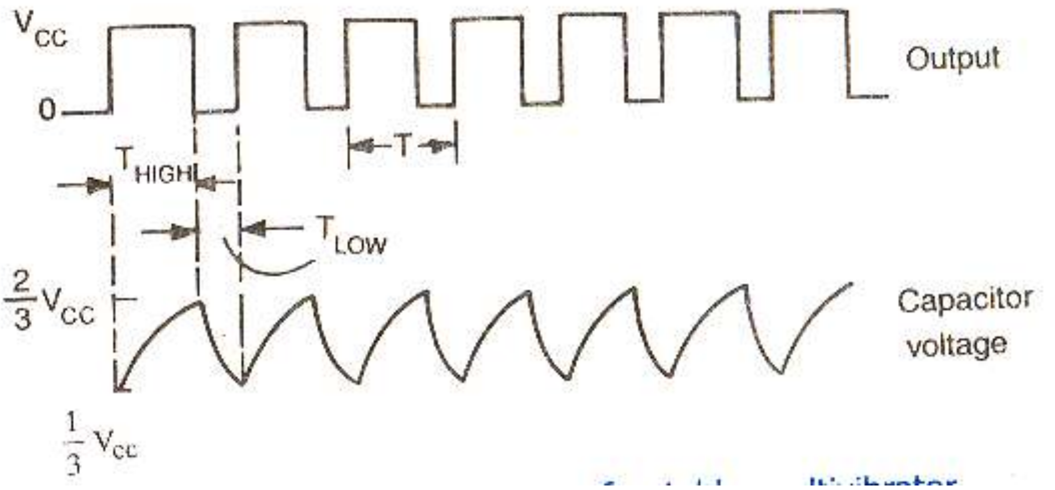
Hence t_{on} = 0.69 R_AC.

During the discharge period, the discharging transistor is shorted (ON) and the diode 'D' is reverse biased.
Hence $t_{OFF} = 0.69R_B\ C$
Output frequency f= 1 KHz is assumed

$$f = \frac{1}{T} = \frac{1.45}{(R_A + R_B)c}$$ → Eq. (2)

Let C = 0.1 μF
→ $R_A + R_B = 14.5\ K\Omega$
Where t_{ON}= 0.1 m/sec (for 10% duty cycle)
→ R_A= 1.44 K Ω
When t_{ON} = 0.9 m/sec (for 90% duty cycle)
R_A = 12.96 K Ω
R_A is to be varied form 1.414 KΩ to 12.96KΩ for 10% to 90% duty cycle.
Therefore choose R_A as 1.0 KΩ fixed resistor + 15.0kΩ pot, similarly R_B

VIII. GRAPH:



IX: RESULT:

t_{ON} = 69 ($R_A + R_B$) C =
t_{OFF} = 0.69 R_BC =

$$\%Dutycycle = \frac{t_{ON}}{t_{ON} + t_{OFF}} x100 = \frac{t_{ON}}{T} x100 =$$

X: INFERENCE:

i. The working of 555 timer astable multivibrator is observed and the output is plotted.
ii. The duty cycle is calculated
iii. Frequency of the output wave form is calculated

XV REVIEW QUESTIONS

i.Explain the functional block diagram of a 555 timer
ii.Explain the function of reset
iii.What are the modes of operation of timer?
iv.What is the expression of time delay of a astable multivibrator?
v.Discuss some applications of timer in astable mode.
vi.Define duty cycle
vii.Give methods of obtaining symmetrical waveform.
viii.How is an astable multivibrator connected into a pulse position modulator
ix.How Schmitt trigger circuit is constructed using 555 timer
x.Draw the pin diagram of 555 timer.

8.555 TIMER-MONOSTABLE MULTI-VIBRATOR

EXPERIMENT NO:8

I AIM:

To design and test monostable multi-vibrator using IC555 Timer.

II EQUIPMENTS AND COMPONENTS:

(i).APPARATUS

1. DC power supply - 1 No.
2. CRO - 1 No.
3. Bread Board - 1 No.
4. Function Generator - 1 No.

3. COMPONENTS:

1. 11.8 kΩ Resistor – 1 No.
2. 1kΩ Resistor – 1 No.
3. 0.1 μF Capacitor – 1 No
4. 0.01 μF Capacitor – 1 No
5. IC555 - 1 No.
6. 1N4148 Diode – 1 No.

III. THEORY:

The 555 timer can be used with supply voltage in the range of +5 v to +18 v and can drive upto 200 mAmps. It is compatible with both TTL and CMOS logic circuits because of the wide range of supply voltage the 555 timer is versatile and easy to use in monostable multivibrator we will provide external triggering in order to make the timer to switch over to high state (unstable). This is also called as one-short multivibrator.

IV. CIRCUIT DIAGRAM:

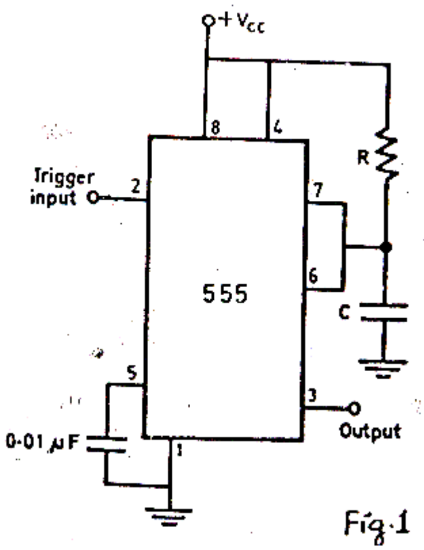


Fig.1

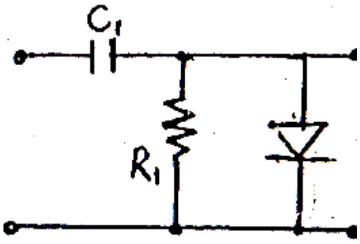


Fig.2

V. PROCEDURE:

1. Connect the circuit using the component values as per the design
2. Set the square wave 2.5V peak and 1KHz trigger input on function generator
3. Apply the trigger input at pin-2 through capacitor C1. Observe both trigger input and the output of the multivibrator on CRO simultaneously and sketch the waveforms
4. Repeat the step 3 for trigger input of 2KHz frequency

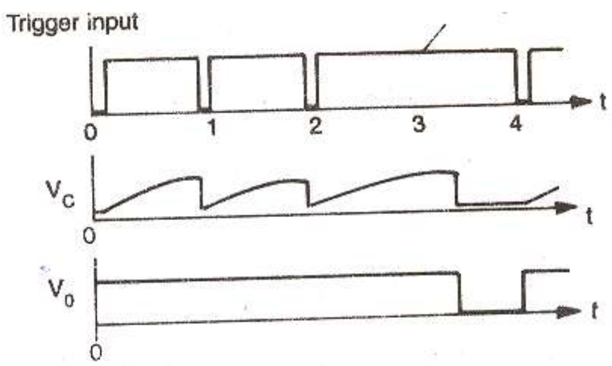
VI. OBSERVATIONS:

1 T_P = 1.1 R.C. = 1.3 m/sec
2. T = 1/ f = 1m/sec

VII. CALCULATIONS:

i. To produce a pulse of 1.3 m/sec duration:
T_P = 1.3mSec
T_P = 1.1 R.C.
Let C = 0.1μ F
R = 11.8KΩ
To provide negative edge triggering a circuit of fig.2 is to be connected between pin 2 and 8.
Design of Differentiator:
Let the trigger input frequency is 1KHz
i.e., T = 1/f = 1mSec
Assume $\tau_1 = R_1 C_1$ = 0.01mSec
Let C = 0.01 μF
Therefore R_1 = 1.0 KΩ

VIII. GRAPH:



IX: RESULT:

T_p = 1.1R.C. =

X: INFERENCE:

i. The working of 555 timer monostable multivibrator is observed and the output is plotted.

ii. The time period of the output waveform is calculated
iii. Frequency of the output wave form is calculated

XV REVIEW QUESTIONS

i. Explain the functional block diagram of a 555 timer
ii. Explain the function of reset
iii. What are the modes of operation of timer?
iv. What is the expression of time delay of a monostable multivibrator?
v. Discuss some applications of timer in monostable mode.
vi. Define duty cycle
vii. Give methods of obtaining symmetrical waveform.
viii. How is an monostable multivibrator connected into a pulse position modulator
ix. How Schmitt trigger circuit is constructed using 555 timer
x. Draw the pin diagram of 555 timer.

9. IC565 PLL

EXPERIMENT NO:9

I AIM:

1. To study the operation of NE565 PLL
2. To use NE565 as a multiplier

II EQUIPMENTS AND COMPONENTS:

(i).APPARATUS

1. DC power supply	-	1 No.
2. CRO	-	1 No.
3. Bread Board	-	1 No.
4. Function Generator	-	1 No.

3. COMPONENTS:

1. 6.8 kΩ Resistor – 1 No.
2. 0.1 μF Capacitor – 1 No
5. 0.001 μF Capacitor – 2 Nos
6. IC565 - 1 No.

III. THEORY:

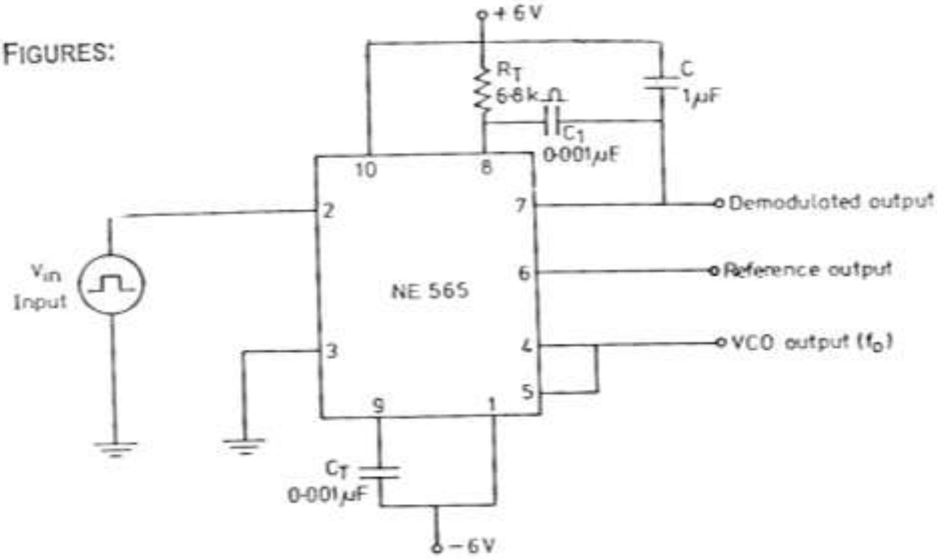
The 565 is available as a 14-pin DIP package. It is produced by signatic corporation. The output frequency of the VCO can be rewritten as

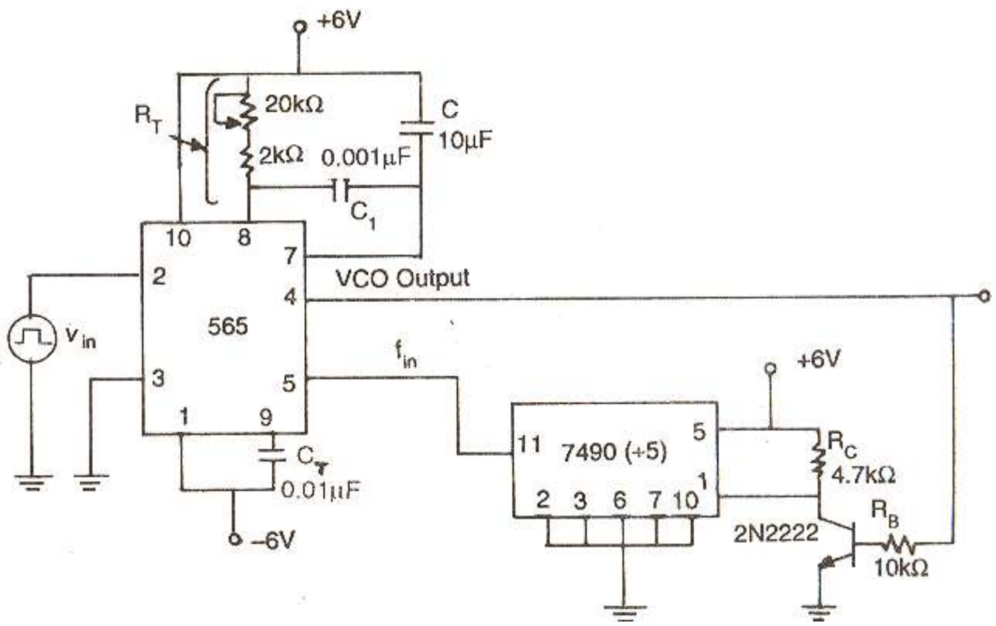
$$f_o = \frac{0.25}{R_T C_T} Hz$$

where R_T and $C_T$are the external resistor and capacitor connected to pin 8 and pin 9. A value between 2 kΩ and 20 kΩ is recommended for R_T. The VCO free running frequency is adjusted with R_T and C_T to be at the centre fo the input frequency range.

IV. CIRCUIT DIAGRAM:

FIGURES:





V. PROCEDURE:

i. Connect the circuit using the component values as shown in the figure

ii .Measure the free running frequency of VCO at pin 4 with the input signal V_{in} set = zero. Compare it with the calculated value = $0.25/ R_T C_T$

iii. Now apply the input signal of 1Vpp square wave at a 1kHz to pin 2

iv. Connect 1 channel of the scope to pin 2 and display this signal on the scope

v .Gradually increase the input frequency till the PLL is locked to the input frequency. This frequency $f_1$gives the lower ends of the capture range. Go on increase the input frequency, till PLL tracks the input signal, say to a frequency f_2. This frequency f_2 gives the upper end of the lock range. If the input frequency is increased further the loop will get unlocked.

vi. Now gradually decrease the input frequency till the PLL is again locked. This is the frequency f_3, the upper end of the capture range. Keep on decreasing the input frequency until the loop is unlocked. This frequency f_4 gives the lower end of the lock range

vii .The lock range $\Delta f_L = (f_2 - f_4)$ compare it with the calculated value of $\frac{\pm 7.8 fo}{12}$

Also the capture range is $\Delta f_c = (f_3 - f_1)$. Compare it with the calculated value of capture range.

$$\Delta f_c = \pm \left[\frac{\Delta f_L}{(2\pi)(3.6)(10^3) x C} \right]^{1/2}$$

viii. To use PLL as a multiplie5r, make connections as showin in fig. The circuit uses a 4-bit binary counter 7490 used as a divide-by-5 circuit.

ix. Set the input signal at 1Vpp square wave at 500Hz

x.. Vary the VCO frequency by adjusting the 20KΩ potentiometer till the PLL is locked. Measure the output frequency

xi. Repeat step 9 and 10 for input frequency of 1kHz and 1.5kHz.

VI. OBSERVATIONS:

fo = ___________

f_L = ___________

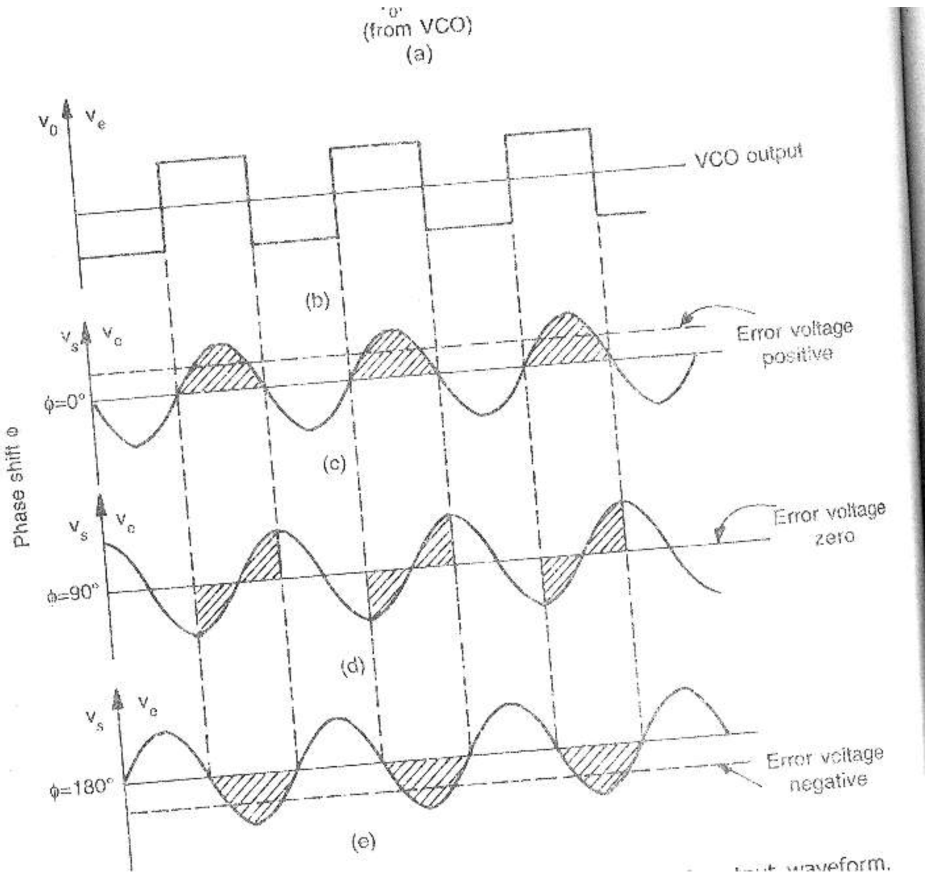
f_C = ___________

VII. CALCULATIONS:

$\Delta f_L = (f_2 - f_4) = \frac{\pm 7.8 fo}{12}$

$\Delta f_c = (f_3 - f_1) = \pm \left[\frac{\Delta f_L}{(2\pi)(3.6)(10^3) x C} \right]^{1/2}$

VIII. GRAPH:



IX: RESULT:

fo = __________

f_L = __________

f_C = __________

X: INFERENCE:

i. The working of 565 PLL is observed and the output is plotted.
ii. The time period of the output waveform is calculated
iii. Frequency of the output wave form is calculated
iv. The Lock range and Capture range of the PLL are calculated.

XV REVIEW QUESTIONS

i. List the basic building blocks of a PLL.
ii. Define capture range
iii. Define lock range
iv. Define pull in time

v. Which is greater capture range or lock range?
vi. What is the major difference between digital and analog PLLs
vii. List the applications of PLL,
viii. Explain about phase detector
ix. Explain about PLL based AM detector
x. Explain the operation of multiplier circuit.

10. IC566 VCO

EXPERIMENT NO:10

I AIM:

To study the operation of NE566 VCO

II EQUIPMENTS AND COMPONENTS:

(i).APPARATUS

1. DC power supply - 1 No.
2. CRO - 1 No.
3. Bread Board - 1 No.
4. Function Generator - 1 No.

3. COMPONENTS:

1. 6.8 kΩ Resistor – 1 No.
2. 15 kΩ Resistor – 1 No.
3. 100 kΩ Resistor – 1 No.
4. 75 pF Capacitor – 1 No
5. IC566 - 1 No.

.

III. THEORY:

A common type of VCO available in IC form in Stigmatic NE/SE566. Referring to the circuit diagram a timing capacitor C_T is linearly charged or discharged by a constant current source/sink. The amount of current can be controlled by changing the voltage v_c applied at the modulating input (pin5) or by changing the timing resistor R_T external to IC chip. The voltage at pin 6 is held at the same voltage as pin 5. Thus, if the modulating voltage at pin5 is increased, the voltage pin 6 also increases, resulting in less voltage across R_T and thereby decreasing the charging current.

The voltage across the capacitor CT is applied to the inverting input terminal of Schmitt Trigger the output voltage swing of the Schmitt Trigger is designed to Vcc and 0.5 Vcc. The output frequency of the VCO can be calculated as follows:

The total voltage on the capacitor changes from 0.25 Vcc to 0.5 Vcc. Thus Δv = 0.25 Vcc. The capacitor charges with a constant current source.

So, $\dfrac{\Delta v}{\Delta t} = \dfrac{i}{C_T}$

$$\frac{0.25Vcc}{\Delta t} = \frac{i}{C_T}$$

$$\Delta t = \frac{0.25 Vcc C_T}{i}$$

The time period T of the triangular wave form equal to 2Δt. The frequency of oscillator fo is,

$$fo = \frac{1}{T} = \frac{1}{2\Delta t}$$

$$= \frac{i}{0.5 VccC \ C_T}$$

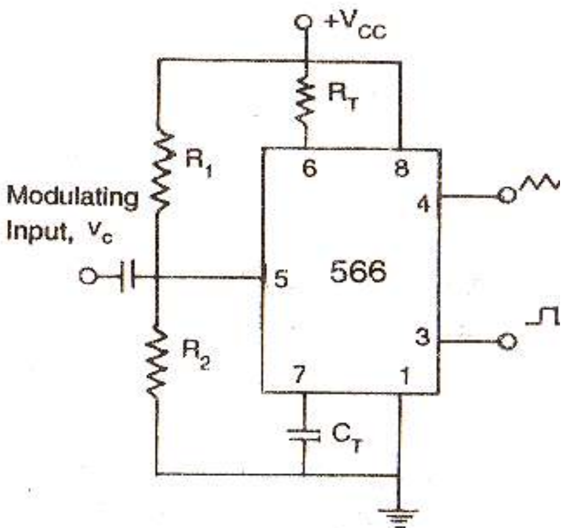
$$\text{But } i = \frac{V_{CC} - Vc}{R_T}$$

Where Vc is the voltage at pin 5. Therefore,

$$fo = \frac{2(V_{CC} - Vc)}{C_T R_T V_{CC}}$$

The output frequency of the VCO can be changed either by (i) R_T, (ii) C_T are (iii) The voltage V_C at the modulating input terminal pin 5. The voltage V_C can be varied by connecting a R_1R_2 circuit.

IV. CIRCUIT DIAGRAM:



V. PROCEDURE:

1) Make connections of the VCO as show in the Circuit diagram.
2) Measure the free running frequency of VCO at Pin 4, with the input signal V in set = 0. Compare it with the theoretical calculated value.
3) Draw the output wave form obtained at Pin No 3.
4) Draw the output wave form obtained at Pin No 4
5) Compare the theoretical and practical values of free running frequency.

VI. OBSERVATIONS:

fo = $\frac{2(V_{CC} - V_C)}{C_T R_T V_{CC}}$ = ___________

Output amplitude at Pin 4 = ______________

Output amplitude at Pin 3 = ______________

VII. CALCULATIONS:

$$\frac{\Delta v}{\Delta t} = \frac{i}{C_T}$$

$$\frac{0.25Vcc}{\Delta t} = \frac{i}{C_T}$$

$$\Delta t = \frac{0.25VccC_T}{i}$$

The time period T of the triangular wave form equal to 2Δt. The frequency of oscillator fo is,

$$fo = \frac{1}{T} = \frac{1}{2\Delta t}$$

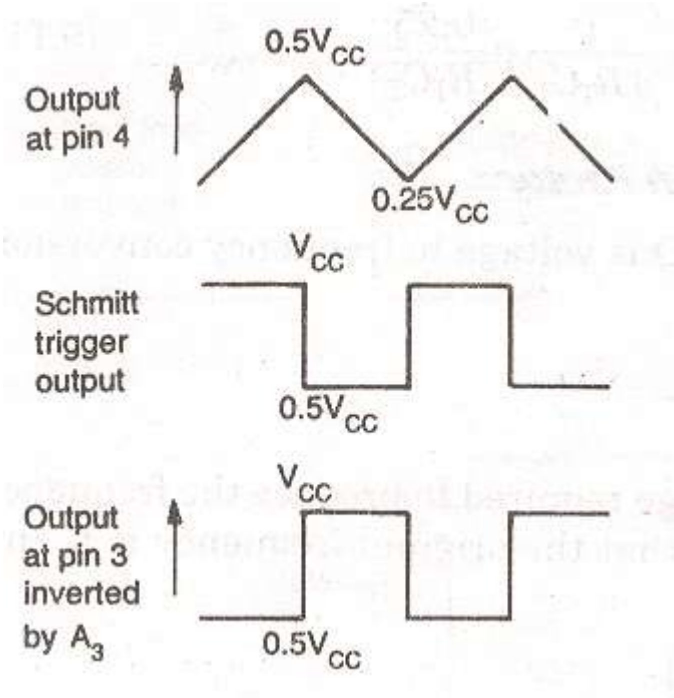
$$= \frac{i}{0.5VccC\ \ C_T}$$

But i = $\frac{V_{CC} - V_C}{R_T}$

Where Vc is the voltage at pin 5. Therefore,

fo = $\frac{2(V_{CC} - V_C)}{C_T R_T V_{CC}}$

VIII. GRAPH:



(c)

IX: RESULT:

fo = _________

X: INFERENCE:

i. The working of 566PLL is observed and the output is plotted.
ii. The time period of the output waveform is calculated
iii. Frequency of the output wave form is calculated

XV REVIEW QUESTIONS

i. List the basic building blocks of a VCO.
ii. Define free running frequency.
Iii .Define lock range
Iv .Explain the block diagram of IC 566 VCO.
v. What is the range of modulating input voltage applied to a VCO?
vi. What is the frequency transfer coefficient of VCO.
vii. List the applications of VCO,
viii. Explain about Schmitt trigger circuit.
Ix .Explain how VCO is used in PLL.
X .Explain the operation of constraint current source.

11. COMPARATOR

EXPERIMENT NO: 11

I AIM:

To Study the operation of comparator

II EQUIPMENTS AND COMPONENTS:

(i).APPARATUS

1. DC power supply - 1 No.
2. CRO - 1 No.
3. Bread Board - 1 No.
4. Function Generator - 1 No.

3. COMPONENTS:

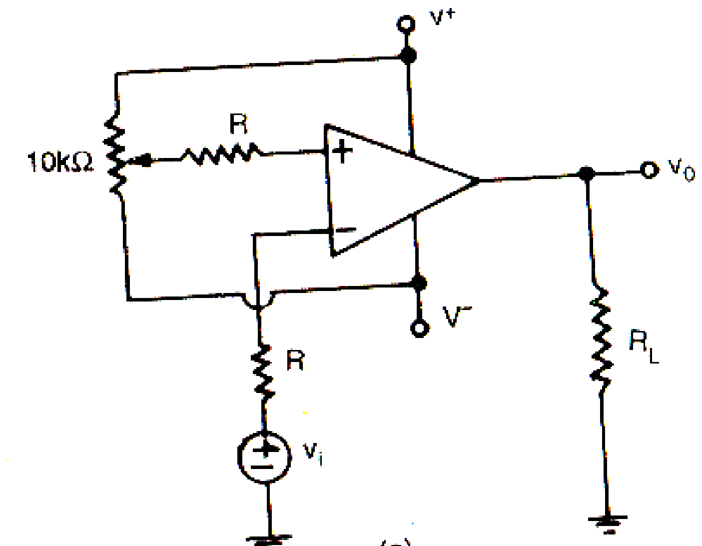
i.10 kΩ Resistor – 1 No.
ii.1kΩ Resistor – 1 No.
iii.IC741 - 1 No.

.

III. THEORY:

A comparator is a circuit which compares a signal voltage applied at one input of an OPAMP with a known inference voltage at the other input. It is basically an open loop OPAMP with output $\pm V_{sat}$. It may be seen that change in output state takes place with an increment in input V_i of only 2mV. There are basically two types of comparators, non-inverting comparator and inverting comparator.

IV. CIRCUIT DIAGRAM:



(a)

V. PROCEDURE:

1. Connect the circuit shown in Fig. And adjust the 10 kΩ potentiometer so that V_{ref} = +0.5V
2. Adjust the signal generator so that v_i = 2V pp sine wave at 1 kHz.
3. Using a CRO observe the input and output waveform simultaneously. Plot the output waveform.
4. Adjust the 10 kΩ potentiometer so that V_{ref} = -0.5V. Repeat step 3
5. To make a zero crossing detector, set V_{ref} = 0V and observe the output waveforms.

VI. OBSERVATIONS:

1. V_{out} = __________
2. V_{in} = __________

VII. CALCULATIONS:

V_{ref} = 0.5 V

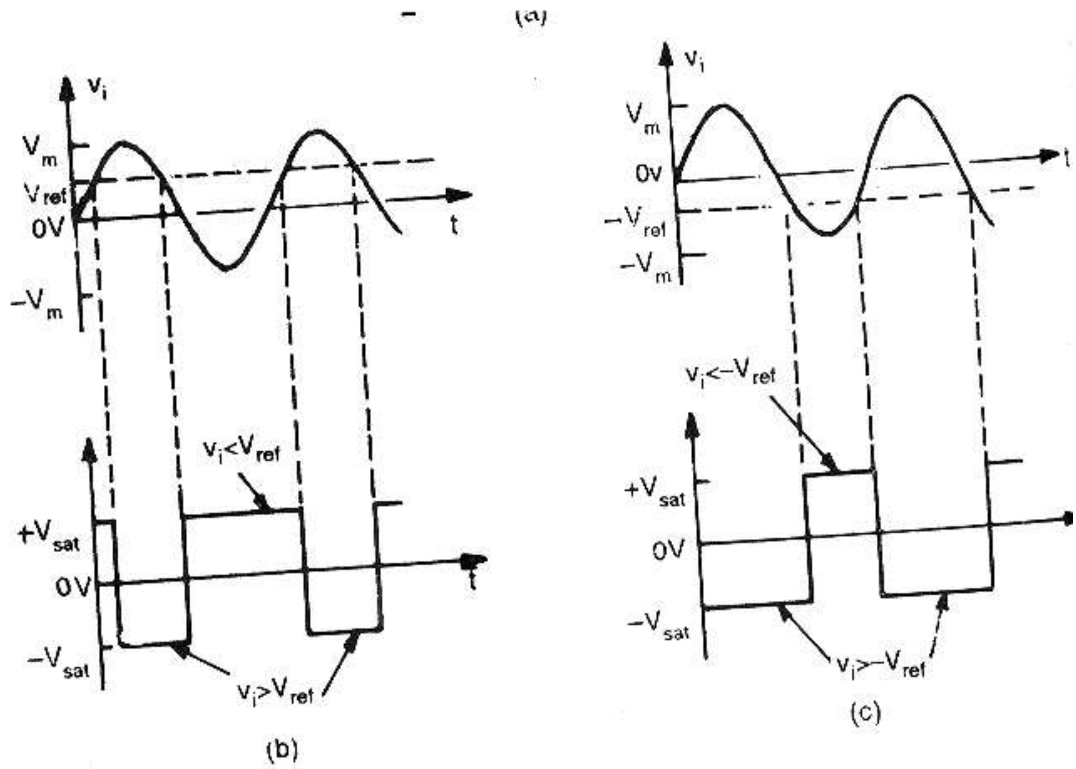
V_O = ________

V_{ref} = -0.5V

V_O = _______

Time period of output waveform = ___________

VIII. GRAPH:



IX: RESULT:

V_{in} = _______

V_O = _______

V_{ref} = ________

X: INFERENCE:

i. The working of comparator is observed and the output is plotted.
ii. The time period of the output waveform is calculated
iii. Frequency of the output wave form is calculated

XV. REVIEW QUESTIONS

i.Discuss the characteristics of an ideal comparator.
ii.List the different types of comparators.
iii.What is the meaning of Voltage limiting?
iv.What is window detector?
v.What is zero crossing detector?
vi.What is Schmitt trigger circuit?
vii.What is time maker generator?
viii.What are the differences between ideal and practical comparator?
ix.What are the applications of comparator?
x. In which mode the operational amplifier is connected in the comparator circuit.

12. D/A CONVERTER

EXPERIMENT NO:12

I AIM:

To construct a 4-bit R – 2 R ladde type D/A converter. Plot the transfer characteristics, that is, binary input vs output voltage. Calculate the resolution and linearity of the converter from the graph.

II EQUIPMENTS AND COMPONENTS:

(i).APPARATUS

1. DC power supply - 1 No.
2. CRO - 1 No.
3. Bread Board - 1 No.
4. Function Generator - 1 No.

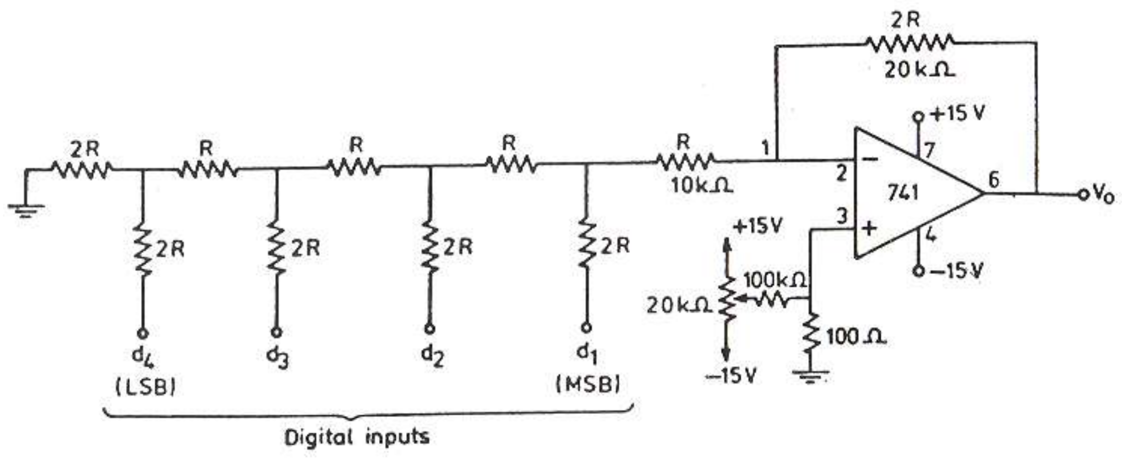
3. COMPONENTS:

1.10 kΩ Resistor – 1 No.
2.20 kΩ Resistor – 1 No.
3.IC741 - 1 No.
.Bandwidth adjustment range = < ± 1 to ±60%

III. THEORY:

Most of the real world physical quantities such as voltage current temperature pressure are available in analog form. It is very difficult to process the signal in analog form, hence ADC and DAC are used. The DAC is to convert digital signal into analog and hence the functioning of DAC is exactly opposite to that of ADC. The DAC is usually operated at the same frequency as the ADC. The output of the DAC is commonly staircase. This staircase like digital output is passed through a smoothing filter to reduce the effect of quantization noise. There are three types of DAC techniques (i) Weighted resistor DAC (ii) R-2R ladder. (iii) Inverted R-2R ladder. Wide range of resistors is required in binary weighted resistor type DAC. This can be avoided by using R-2R ladder type DAC where only two values of resistors are required it is well suited for integrated circuit realization.

IV. CIRCUIT DIAGRAM:



A 4-bit R-2R ladder D/A converter

V. PROCEDURE:

i. Set up the circuit shown in Fig.

ii. With all inputs (d_0 to d_3) shorted to ground, adjust the 20 kΩ pot until the output is 0V. This will nullify any offset voltage at the input of the op-amp.

Iii ..Measure the output voltage for all binary input states (0000 to 1111) and plot a graph of binary inputs vs output voltage.

iv. .Measure the size of each step and hence calculate resolution

v. .Calculate linearity

VI. OBSERVATIONS:

Output Voltage = ____________

Size of each step = ____________

Resolution = ____________

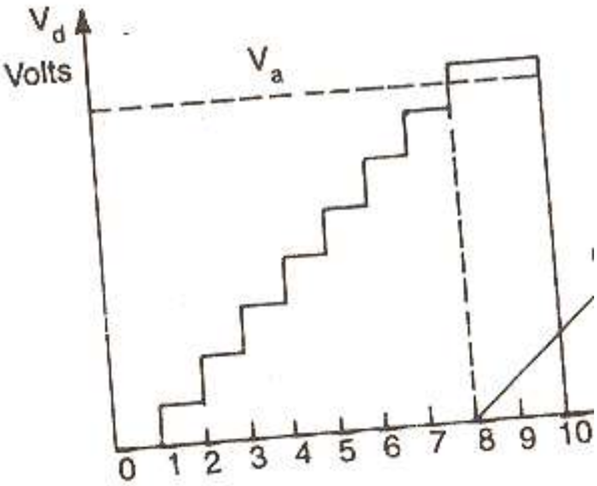
Linearity = ____________

VII. CALCULATIONS:

$V_O = V_R / 2 = V_{FS} / 2$

Resolution (in volts) = $V_{FS} / (2^n - 1)$ = 1 LSB increment.

VIII. GRAPH:



.

IX: RESULT:

Output Voltage = ____________
Size of each step = ____________
Resolution = ____________
Linearity = ____________

X: INFERENCE:

i. The working of D / A converter is observed and the output is plotted.
ii. The staircase wave form is plotted.
iii. Resolution is calculated.

XV REVIEW QUESTIONS

i. Classify DAC on the basis of their output.
ii. Name the essential parts of a DAC.
iii. Describe the various types of electronic switches used in DAC.
vi. How many resistors are required in 12 bit weighted resistor DAC?
v. Why is an inverted R-2R ladder network DAC is better than R-2R ladder DAC.
vi. Define resolution.
vii. Define linearity.
Viii .Define monotonicity.
Ix .Define step size.
x. Define settling time.