Varuvan Vadivelan Institute of Technology

Dharmapuri – 636 703

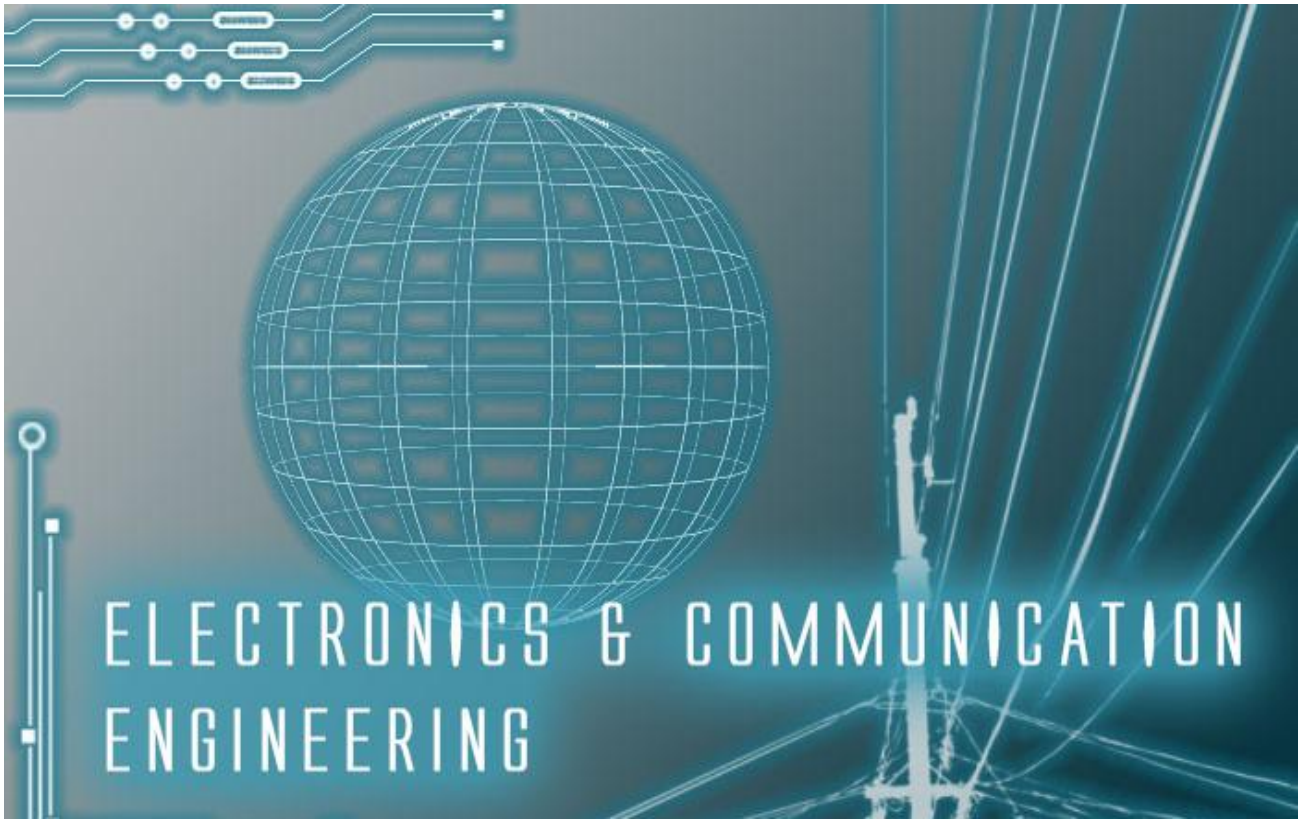
LAB MANUAL

Regulation : **2013**

Branch : ***B.E.*** **– ECE**

Year & Semester : **II Year / IV Semester**

EC6412- LINEAR INTEGRATED CIRCUIT LABORATORY



ANNA UNIVERSITY: CHENNAI
REGULATION 2013
EC6412 LINEAR INTEGRATED CIRCUITS LABORATORY

LIST OF EXPERIMENTS:

DESIGN AND TESTING OF

1. Inverting, Non inverting and Differential amplifiers.
2. Integrator and Differentiator.
3. Instrumentation amplifier
4. Active low-pass, High-pass and band-pass filters.
5. Astable &Monostable multivibrators and Schmitt Trigger using op-amp.
6. Phase shift and Wein bridge oscillators using op-amp.
7. Astable and monostable multivibrators using NE555 Timer.
8. PLL characteristics and its use as Frequency Multiplier.
9. DC power supply using LM317 and LM723.
10. Study of SMPS.

SIMULATION USING SPICE

1. Simulation of Experiments 3, 4, 5, 6 and 7.
2. D/A and A/D converters (Successive approximation)
3. Analog multiplier
4. CMOS Inverter, NAND and NOR

TOTAL-45 PERIODS

INDEX

Ex.No.	DATE	LIST OF THE EXPERIMENT	SIGNATURE OF THE STAFF	REMARKS
1		INVERTING & NON-INVERTING AMPLIFIERS USING OP-AMP		
2		DIFFERENTIAL AMPLIFIERS USING OP-AMP		
3		INTEGRATOR AND DIFFERENTIATOR USING OP-AMP.		
4		INSTRUMENTATION AMPLIFIER		
5		ASTABLE, MONOSTABLE MULTIVIBRATOR USING OP-AMP.		
6		SCHMITT TRIGGER USING OP-AMP		
7		RCPHASE SHIFT AND WEIN BRIDGE OSCILLATOR USINGOPAMP		
8		ACTIVE LOWPASS, HIGH PASS AND BAND PASS FILTER USINGOP-AMP.		
9		ASTABLE AND MONOSTABLE MULTIVIBRATOR USING IC555 TIMER		
10		PLL CHARACTERISTICS AND FREQUENCY MULTIPLIER USING PLL		

Ex.No.	DATE	LIST OF THE EXPERIMENT	SIGNATURE OF THE STAFF	REMARKS
11		DCPOWER SUPPLY USING LM317		
12		DCPOWER SUPPLY USING LM 723		
13		STUDY OF SMPS		
14		SIMULATION OF INSTRUMENTATION AMPLIFIER USING PSPICE		
15		SIMULATION OF ACTIVE LOWPASS, HIGH PASS AND BAND PASS FILTER USING PSPICE		
16		SIMULATION OF ASTABLE, MONOSTABLE MULTIVIBRATOR & SCHMITT TRIGGER USING PSPICE		
17		SIMULATION OF RC PHASE SHIFT AND WEIN BRIDGE OSCILLATOR USING PSPICE		
18		SIMULATION OF ASTABLE AND MONOSTABLE MULTI VIBRATOR USING IC555 TIMER		
19		SIMULATION OF ADC,DAC AND ANALOG MULTIPLIER USING PSPICE		
20		SIMULATION OF CMOS INVERTER,NAND AND NOR USING PSPICE		

INTRODUCTION OF LINEAR CIRCUIT

A linear circuit is an electronic circuit in which, for a sinusoidal input voltage of frequency f, any steady-state output of the circuit (the current through any component, or the voltage between any two points) is also sinusoidal with frequency f. Note that the output need not be in phase with the input.

An equivalent definition of a linear circuit is that it obeys the superposition principle. This means that the output of the circuit F(x) when a linear combination of signals ax1(t) + bx2(t) is applied to it is equal to the linear combination of the outputs due to the signals x1(t) and x2(t) applied separately:

It is called a linear circuit because the output of such a circuit is a linear function of its inputs. Informally, a linear circuit is one in which the electronic components' values (such as resistance, capacitance, inductance, gain, etc.) do not change with the level of voltage or current in the circuit. Linear circuits are important because they can amplify and process electronic signals without distortion. An example of an electronic device that uses linear circuits is a sound system.

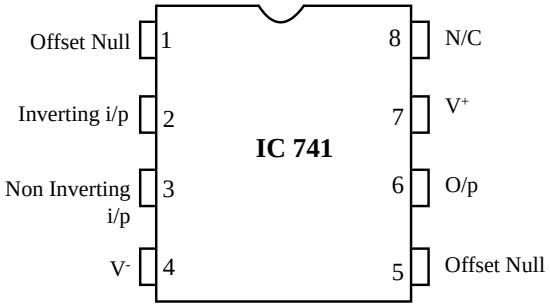
Linear circuits are important because they can process analog signals without introducing inter modulation distortion. This means that separate frequencies in the signal stay separate and do not mix, creating new frequencies (heterodynes).

STUDY OF OP-AMP

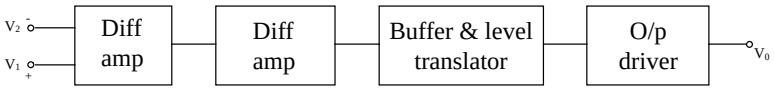
An operational amplifier or op-amp is a linear integrated circuit that has a very high voltage gain, high input impedance and low output impedance. Op-amp is basically a differential amplifier whose basic function is to amplify the difference between two input signals.

Op-amp has five basic terminals, that is, two input terminals, one o/p terminal and two power supply terminals. Pin2 is called the inverting input terminal and it gives opposite polarity at the output if a signal is applied to it. It produces a phase shift of 180^{o} between input and output. Pin3 is called the non-inverting terminal that amplifies the input signal without inversion, i.e., there is no phase shift or i/p is in phase with o/p. The op-amp usually amplifies the difference between the voltages applied to its two input terminals. Two further terminals pins 7 and 4 are provided for the connection of positive and negative power supply voltages respectively. Terminals 1 and 5 are used for dc offset. The pin 8 marked NC indicates 'No Connection'.

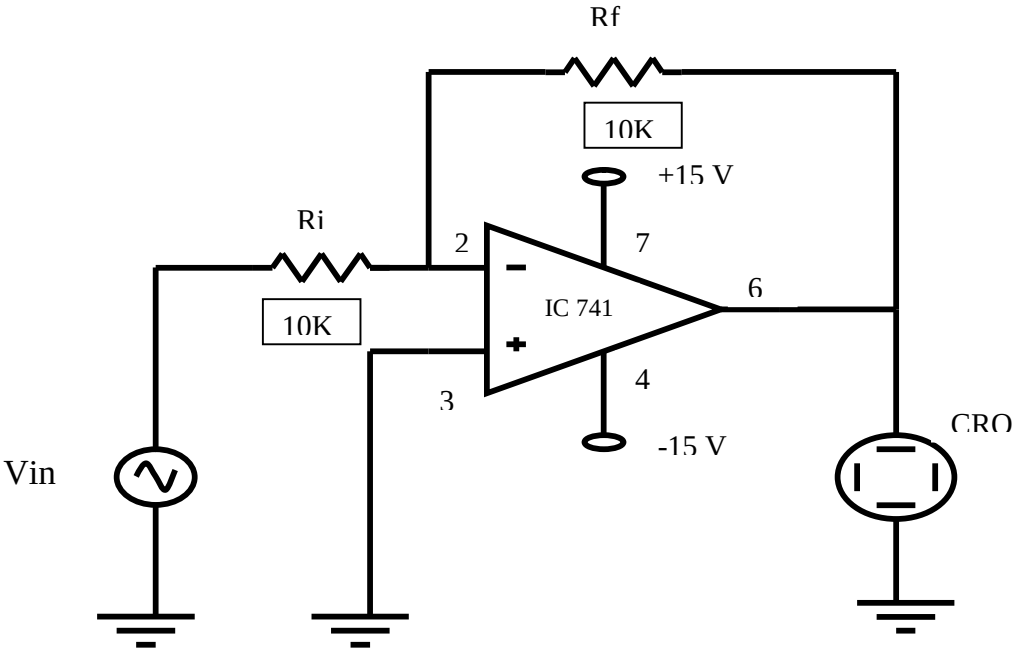
Study of op-amp



Block schematic of op-amp



CIRCUIT DIAGRAM- (INVERTING AMPLIFIER):



DESIGN PROCEDURE:

- $V_O = -I R_f$ ⟶ _____
- $I = V_{in} / R_i$ ⟶ _____
- $V_O = -(V_i / R_i) R_f$ ⟶ _____

- Gain $A_V = V_O / V_i = -(R_f / Ri)$ _____

Ex. No : 01	**DESIGN AND TESTING OF INVERTING AND NON INVERTING AMPLIFIERS**
DATE :	

AIM:

To design and test inverting and non inverting amplifiers using **IC µA 741**

APPARATUS REQUIRED:

S.NO.	APPARATUS	RANGE	QUANTITY
1	Dual power supply	(0 - +15)V	1
2	Signal generator	(0-3)MHz	1
3	CRO	(0-30)MHz	1
4	IC	µA 741	1
5	Resistor	10KΩ,5KΩ	2,1

THEORY:

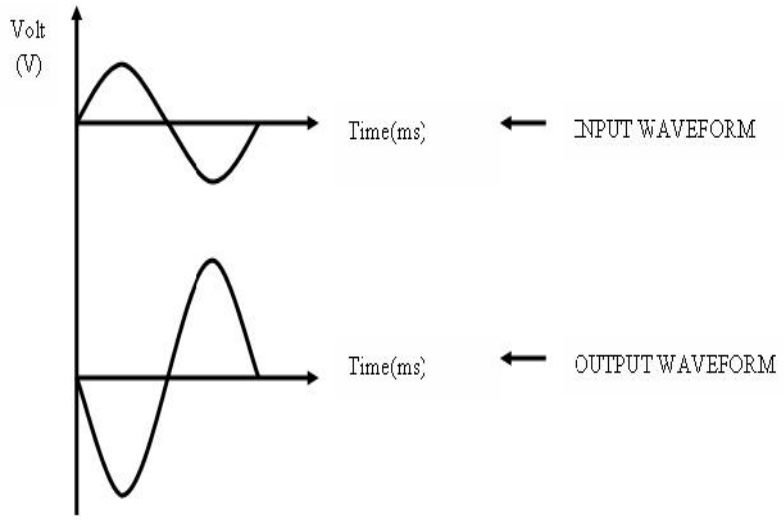
INVERTING AMPLIFIER:

The fundamental component of any analog computer is the operational amplifier or op-amp and the frequency configuration in which it is used as an inverting amplifier. An input voltage V_{in} is applied to the input voltage. It receives and inverts its polarity producing an output voltage (V_O). This same output voltage is also applied to a feedback resistor R_f, which is connected to the amplifier input analog with R_i. The amplifier itself has a very high voltage gain. If $R_f=R_i$ then $V_o=V_i$

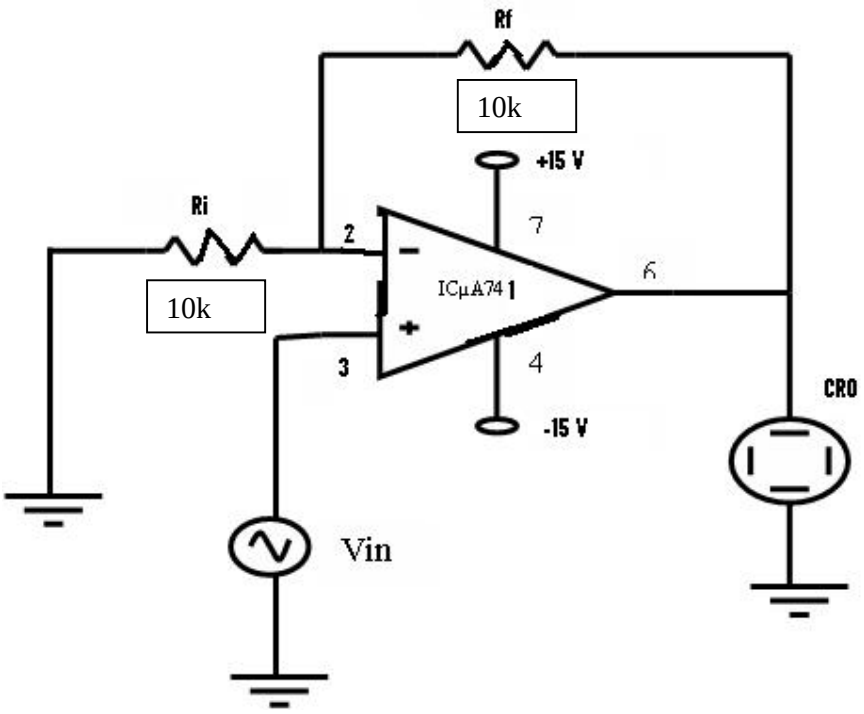
TABULATION –(INVERTING AMPLIFIER):

S.NO.	INPUT SIGNAL		OUTPUT SIGNAL	
	AMPLITUDE (Vi) *volts*	TIME (T) *ms*	AMPLITUDE (Vo) *volts*	TIME (T) *ms*

MODEL GRAPH:



CIRCUIT DIAGRAM –(NON INVERTING AMPLIFIER):



DESIGN PROCEDURE:

- Gain $A_v = V_o/V_i$ ⟶ _____

  $= (R_f+R_i) / R_i$ ⟶ _____

  $=1+ (R_f / R_i)$ ⟶ _____

- $V_{in} = V_o.R_i / (R_i+R_f)$ ⟶ _____

NON- INVERTING AMPLIFIER:

Although the standard op-amp configuration is as an inverting amplifier, there are some applications where such inversion is not wanted. However, we cannot just switch the inverting and non inverting inputs to the amplifier itself. We will still need negative feedback to control the working gain of the circuit .Therefore, we will need to leave the resistor structure around the op-amp intact and swap the input and ground connections to the overall circuit.

$V_O/V_I = (R_f/ R_i)+1$

From the calculations, we can see that the effective voltage gain of the non-inverting amplifier is set by the resistance ratio. Thus, if the two resistors are equal value, then the gain will be 2 rather than 1.

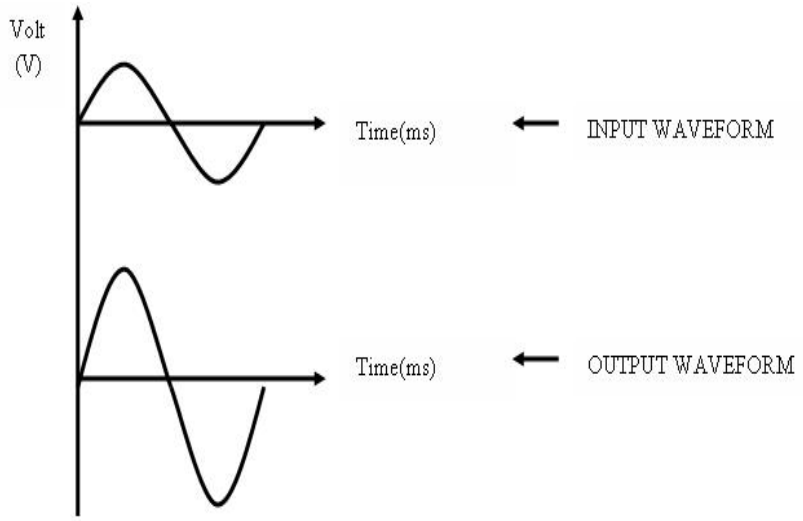
EXPERIMENTAL PROCEDURE:

1. Connect the circuit as shown in the diagram.
2. Give the input signal as specified.
3. Switch on the dual power supply
4. Note the outputs from the CRO.
5. Draw the necessary waveforms on the graph sheet.
6. Repeat the above procedure for non-inverting amplifier circuit.
7. Compare the practical gain with theoretically designed gain.

TABULATION –(NON- INVERTING AMPLIFIER):

S.NO.	INPUT SIGNAL		OUTPUT SIGNAL	
	AMPLITUDE (Vi) $volts$	TIME (T) ms	AMPLITUDE (Vo) $volts$	TIME (T) ms

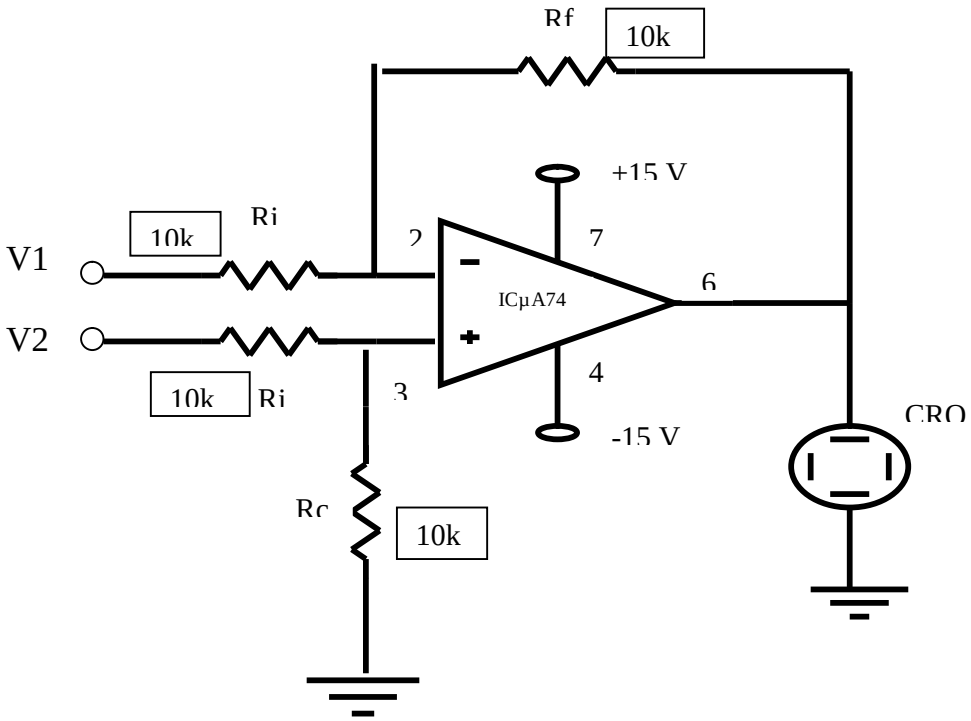
MODEL GRAPH



RESULT :

Thus the inverting and non inverting amplifier circuits using operational amplifier **Ic µA 741** are designed, constructed and tested.

CIRCUIT DIAGRAM - (DIFFERENTIAL AMPLIFIER)



DESIGN PROCEDURE:

- $V_1 = V_2 = V$ _____
- $V_C = (V_1 + V_2)/2 = V$ _____
- $A_C = V_0/V_C$ _____

Ex. No : 02	**DESIGN OF DIFFERENTIAL AMPLIFIER**
DATE :	

AIM:

To design and test a differential amplifier using operational amplifier **IC µA 741**

APPARATUS REQUIRED:

S.NO.	APPARATUS	RANGE	QUANTITY
1	Dual power supply	(0 - +15)V	1
2	Signal generator	(0-3)MHz	2
3	CRO	(0-30)MHz	1
4	IC	µA 741	1
5	Resistor	10KΩ	4

THEORY:

A circuit that amplifies the difference between two signals is called as a differential amplifier. This type of amplifiers is very useful in instrumentation circuits.

From the experimental setup of a differential amplifier, the voltage at the output of the operational amplifier is zero. The inverting and non-inverting terminals are at the same potential. Such a circuit is very useful in detecting very small differences in signals. Since the gain can be chosen to be very large. For example, if $R_2=100R_i$, then a small difference V_1-V_2 is amplified 100 times.

TABULATION –(DIFFERENTIAL AMPLIFIER):

S. NO.	V_1 $volts$	V_2 $volts$	OUTPUT $V_i = V_d = V_1 - V_2$ $volts$	V_o $volts$	GAIN	
					THEORETICAL	PRACTICAL

EXPERIMENTAL PROCEDURE:

1 Connections are made as per the EXPERIMENTAL SETUP.

2 The supply is switched ON.

3 Output is connected to anyone channel of CRO.

4 The V_1 and V_2 voltages are fixed and measured from the other channel of CRO and the corresponding output voltages are also noted from the CRO.

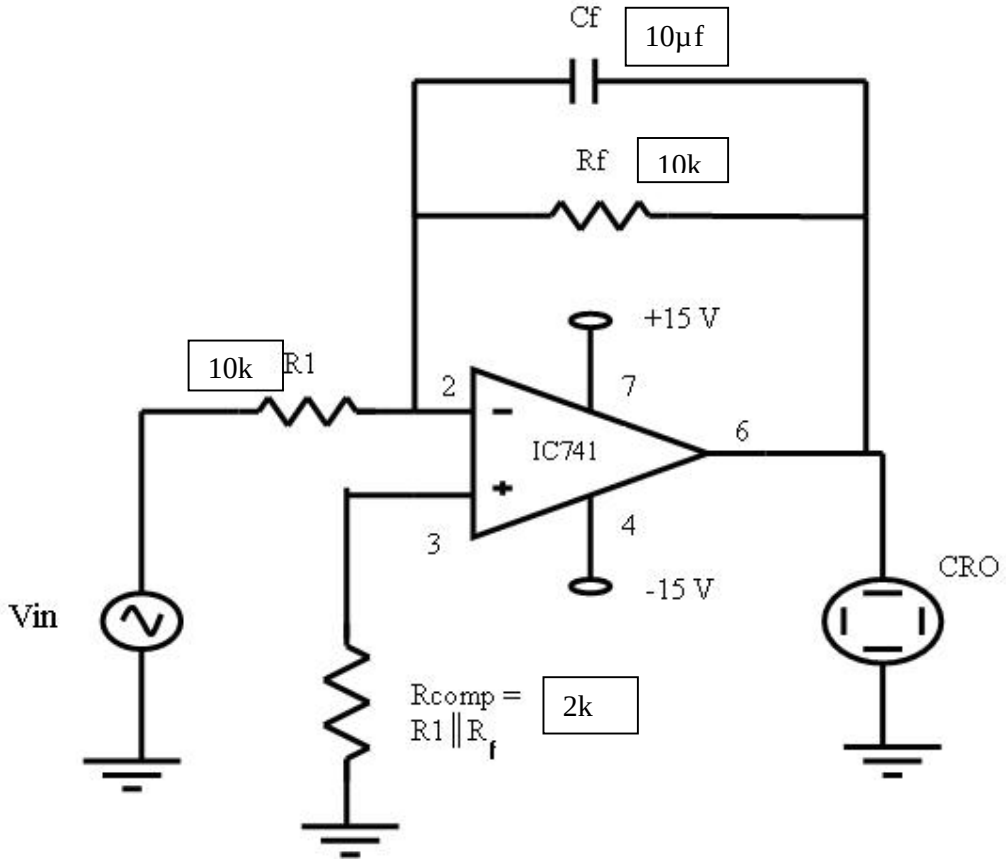
5 The above step is repeated for various values of V_1 and V_2.V_1 and V_2 may be AC or DC voltages from function generator or DC power supply.

6 Readings are tabulated and gain was calculated and composed with designed values.

RESULT:

Thus the differential amplifier is tested using operational amplifier **IC µA 741.**

CIRCUIT DIAGRAM - (INTEGRATOR);



DESIGN PROCEDURE:

- $V = -1/(R_f.C_f)\,[V_{in}(t).V_o(t)]$ _____
- $T = -1/(R_f.C_f)$ _____

Ex. No : 03	**INTEGRATOR AND DIFFERENTIATOR**
DATE :	

AIM:

To design and test the integrator and differentiator using operational amplifier IC µA 741

APPARATUS REQUIRED:

S.NO.	APPARATUS	RANGE	QUANTITY
1	Dual power supply	(0 - +15)V	1
2	Signal generator	(0-3)MHz	1
3	CRO	(0-30)MHz	1
4	IC	µA 741	1
5	Resistor	10KΩ,5KΩ	2,1
6	Capacitor	10µf	1

THEORY:

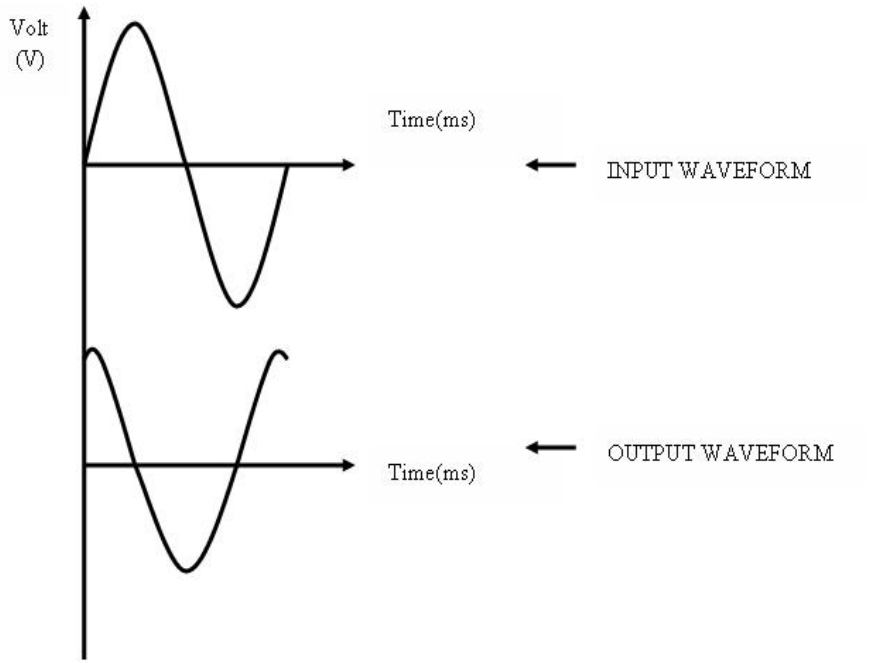
INTEGRATOR:

Op-amps allow us to make nearly perfect integrators such as the practical integrator The circuit incorporates a large resistor in parallel with the feedback capacitor. This is necessary because real op-amps have a small current flowing at their input terminals called the "bias current". This current is typically a few nano amps, and is neglected in many circuits where the currents of interest are in the micro amp to milliamp range. The feedback resistor gives a path for the bias current to flow. The effect of the resistor on the response is negligible at all but the lowest frequencies.

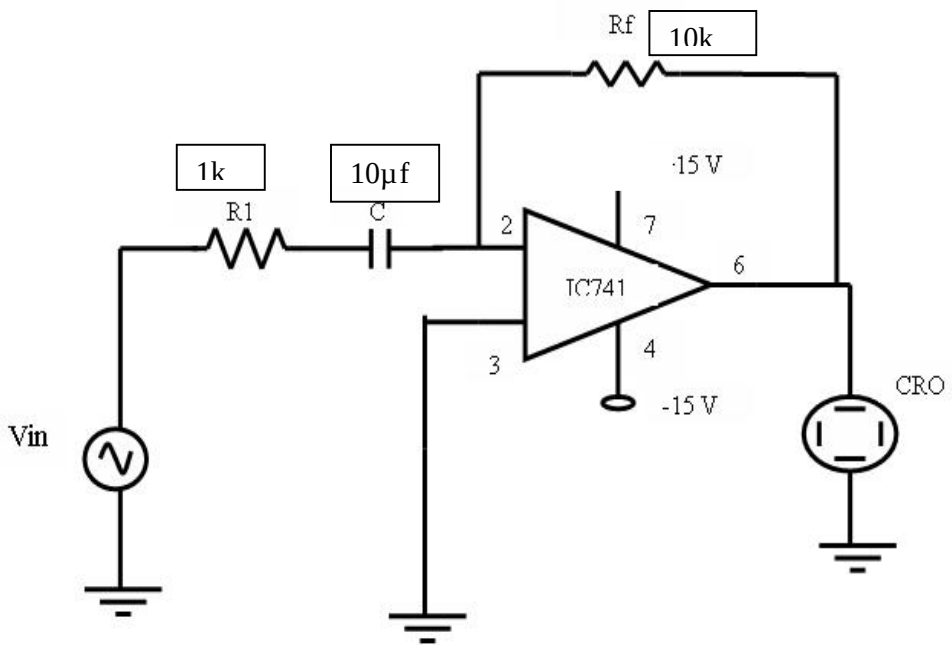
TABULATION-(INTEGRATOR):

INPUT WAVEFORM			OUTPUT WAVEFORM		
AMP. (V) $volts$	TIME (T) ms	F=1/T KHz	AMP. (V) $volts$	TIME (T) ms	F=1/T KHz

MODEL GRAPH:



CIRCUIT DIAGRAM- (DIFFERENTIATOR):



DESIGN PROCEDURE:

- $f_{max} = 1 / (2\pi C_1 R_f)$ _____
- $Vout = -1/R_f.C_f\ [D_{Vin}/ dt]$ _____
- $T = -\ R_f.C_f$ _____

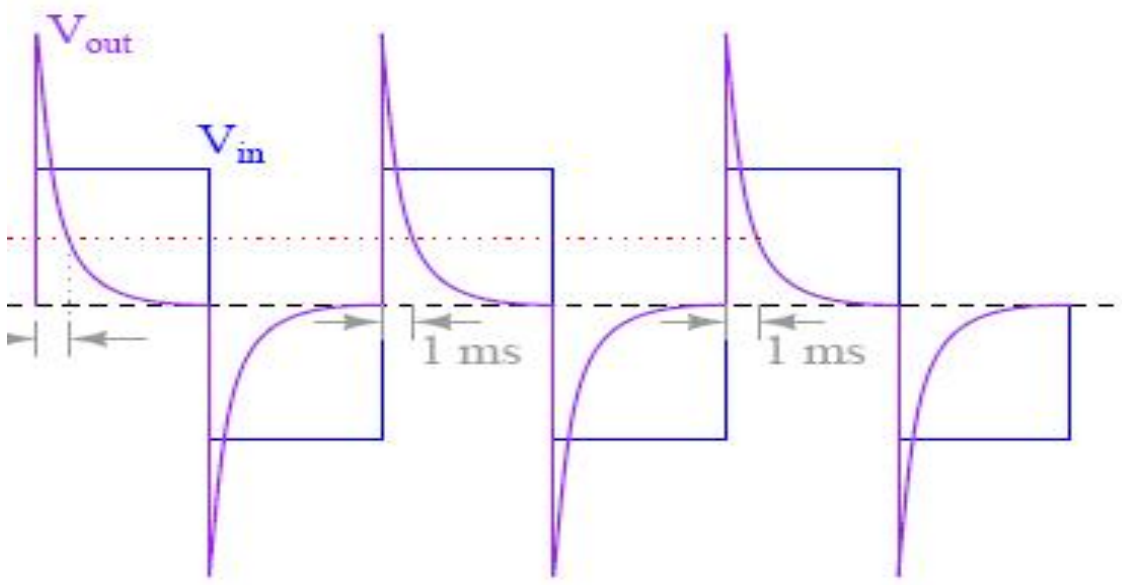
DIFFERENTIATOR:

One of the simplest of the operational amplifier that contains capacitor is differential amplifier. As the suggests, the circuit performs the mathematical operation of differentiation. The output is the derivative of the given input signal voltage. The minus sign indicates an 180^0 phase shift of the output waveform Vo with respect to the input signal.

TABULATION –(DIFFERENTIATOR) :

INPUT WAVEFORM			OUTPUT WAVEFORM		
AMP. (V) $volts$	TIME (T) ms	F=1/T KHz	AMP. (V) $volts$	TIME (T) ms	F=1/T KHz

MODEL GRAPH:



EXPERIMENTAL PROCEDURE:

1 The connections are given as per the EXPERIMENTAL SETUP.

2 The supply is switched ON after checking the circuit connections

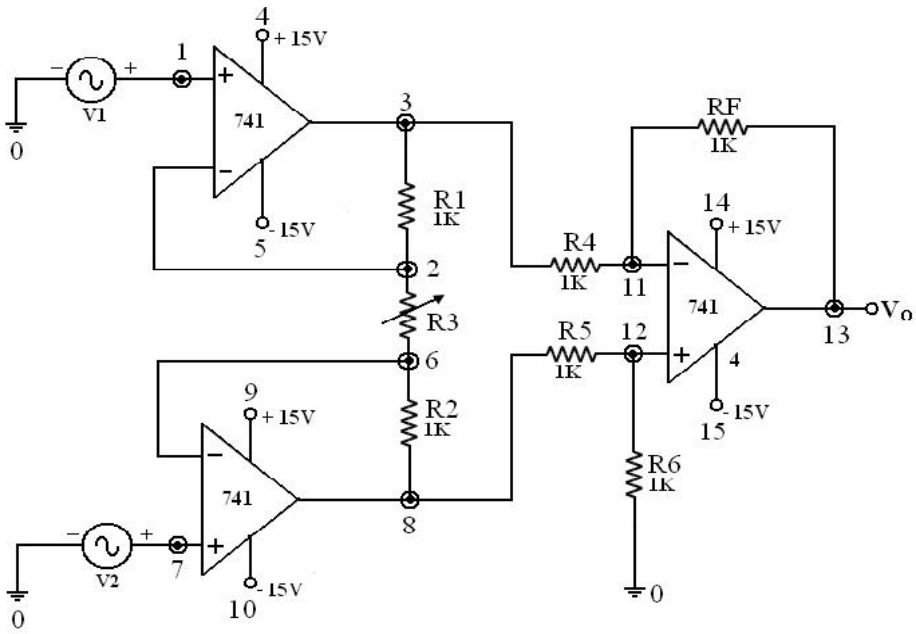
3 The input wave form is applied from the function generator and the corresponding output waveform is noted from the CRO.

4 The above mentioned procedure is repeated for differentiator also.

RESULT :

Thus the integrator and differentiator is designed and tested using operational amplifier IC µA 741

CIRCUIT DIAGRAM-(INSTRUMENTATION AMPLIFIER):



DESIGN PROCEDURE:

- $V_{01} = (1+R_2/R_1)V_1 - (R_2/R_1)V_2$,
- $V_{02} = (1+R_2/R_1)V_2 - (R_2/R_1)V_1$

- $V_0 = V_{02} - V_{01}$ _____

  $= (V_2-V_1)(1+2R_2/R_1)$ _____

Let $R_1=R_2=R_3=R_4=$ _____

- Gain = V_0/V_i
- $= V_o/(V_2-V_1)$
- Gain = _____

Ex. No :04	**INSTRUMENTATION AMPLIFIER**
DATE :	

AIM:

To design and test instrumentation amplifier using ICµA 741 for the given gain.

APPARATUS REQUIRED:

S.NO.	APPARATUS	RANGE	QUANTITY
1	Dual power supply	(0 - +15)V	1
2	Signal generator	(0-3)MHz	1
3	CRO	(0-30)MHz	1
4	IC	µA 741	1
5	Resistor	10kΩ	6

THEORY:

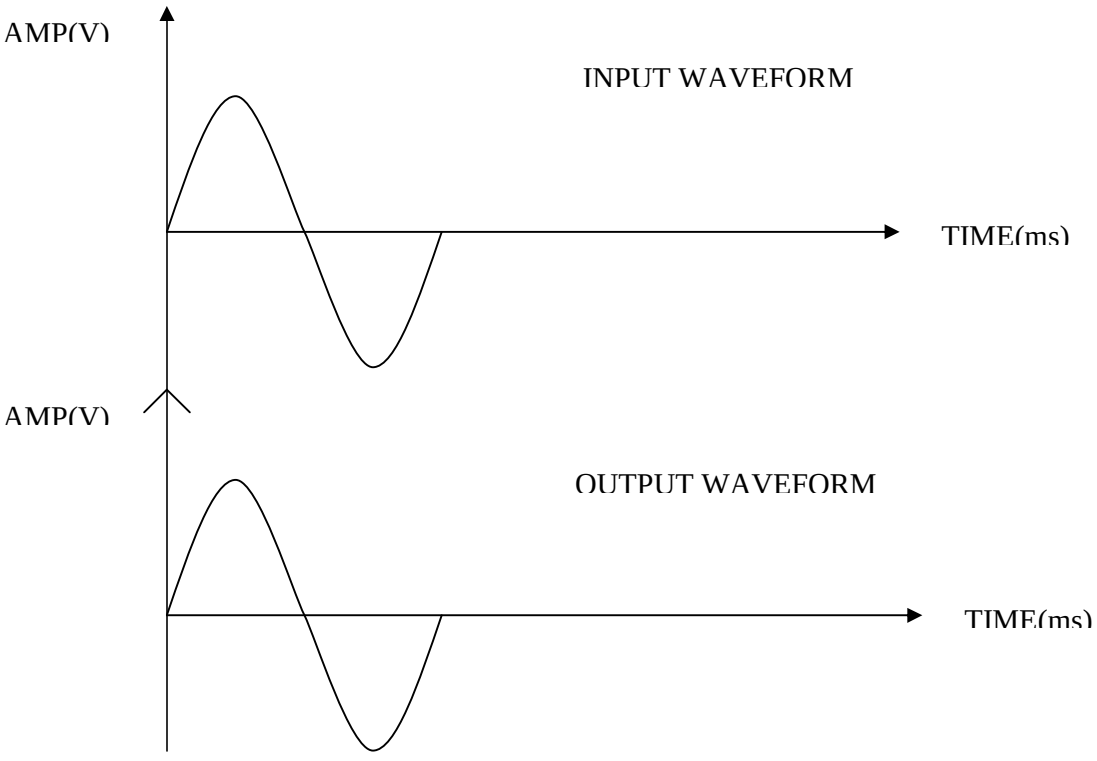
In a number of instrumentation and consumer applications one is required to measure and control the physical quantities. Some typical examples are measurement and control of temperature , humidity, light,
Intensity , water flow etc.

These physical quantities are usually measured with the help of transducer. The output of the transducer has to be amplified so that it can derive the indicator or display system. The function performed by an instrumentation amplifier are,

TABULATION –(INSTRUMENTATION AMPLIFIER):

S.NO	INPUT SIGNAL		OUTPUT SIGNAL	
	AMPLITUDE (Vi) *volts*	TIME (T) *ms*	AMPLITUDE (Vo) *volts*	TIME (T) *ms*

MODEL GRAPH:



- High CMRR
- High gain stability with low temperature coefficient
- Low dc offset
- Low input impedance

These are specially designed op-amp such as VA725 to meet the above started requirement of a good instrumentation amplifier. Monolithic instrumentation amplifiers are also available commercially such as AD521, AD524, AD624 by analog devices L40036, and L40037 by national semiconductors.

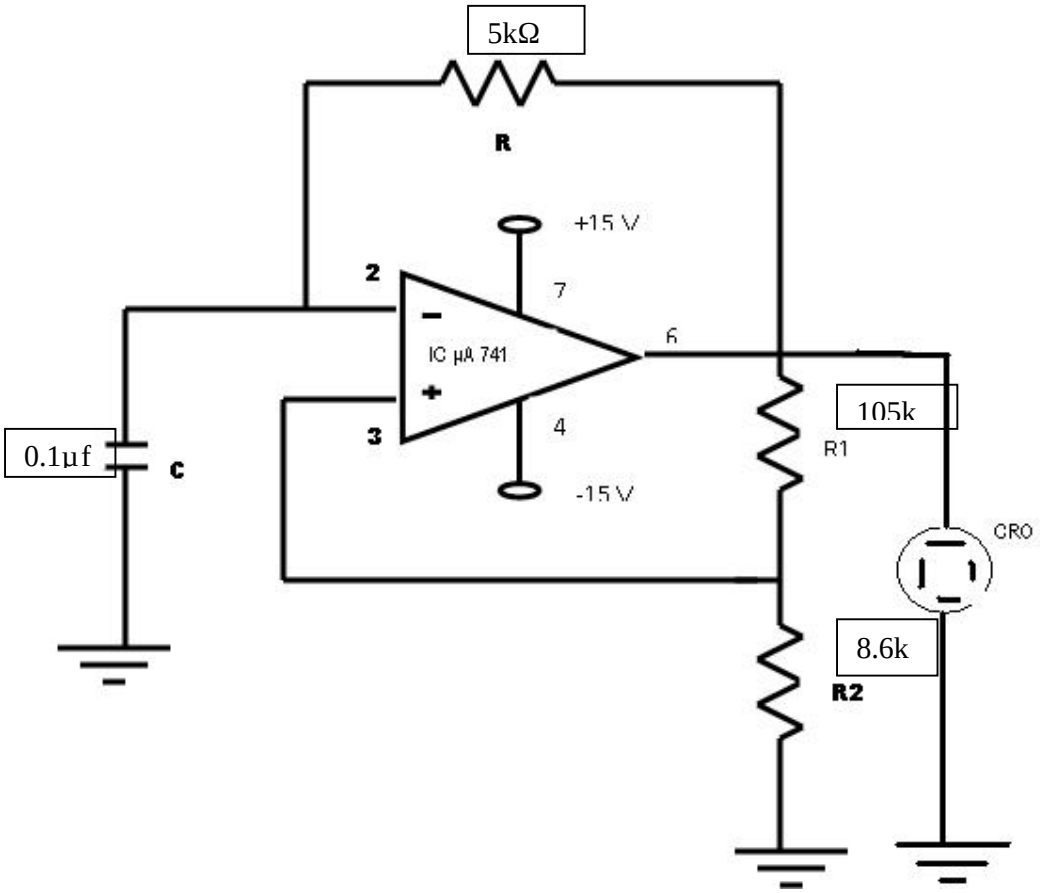
EXPERIMENTAL PROCEDURE:

1 Circuit connections are given as per the EXPERIMENTAL SETUP.
2 The input signal is given
3 The dual power supply is switched ON.
4 The input is varied in steps and the corresponding output readings are noted from CRO.
5 The practical gain is calculated from the readings and compared with the theoretically designed gain.

RESULT :

Thus the instrumentation amplifier is designed and tested using ICμA741.

CIRCUIT DIAGRAM-(ASTABLE MULTIVIBRATOR);



DESIGN PROCEDURE:

- $f_0 = 1/ (2RC)$ Let f_0= _____
- R1 = _____R2 Assume , C= _____

  R2 = _____ R=_____

  R1 = _____

Ex. No :05	**ASTABLE AND MONOSTABLE MULTIVIBRATOR**
DATE :	

AIM:

To design and test an astable and monostable multivibrator circuits using op-amp IC µA 741

APPARATUS REQUIRED:

S.NO.	APPARATUS	RANGE	QUANTITY
1	Dual power supply	(0 - +15)V	1
2	CRO	(0-30)MHz	1
3	IC	µA 741	1
4	Resistor	15kΩ,10kΩ	2,2
6	Capacitor	0.1µf	2

THEORY:

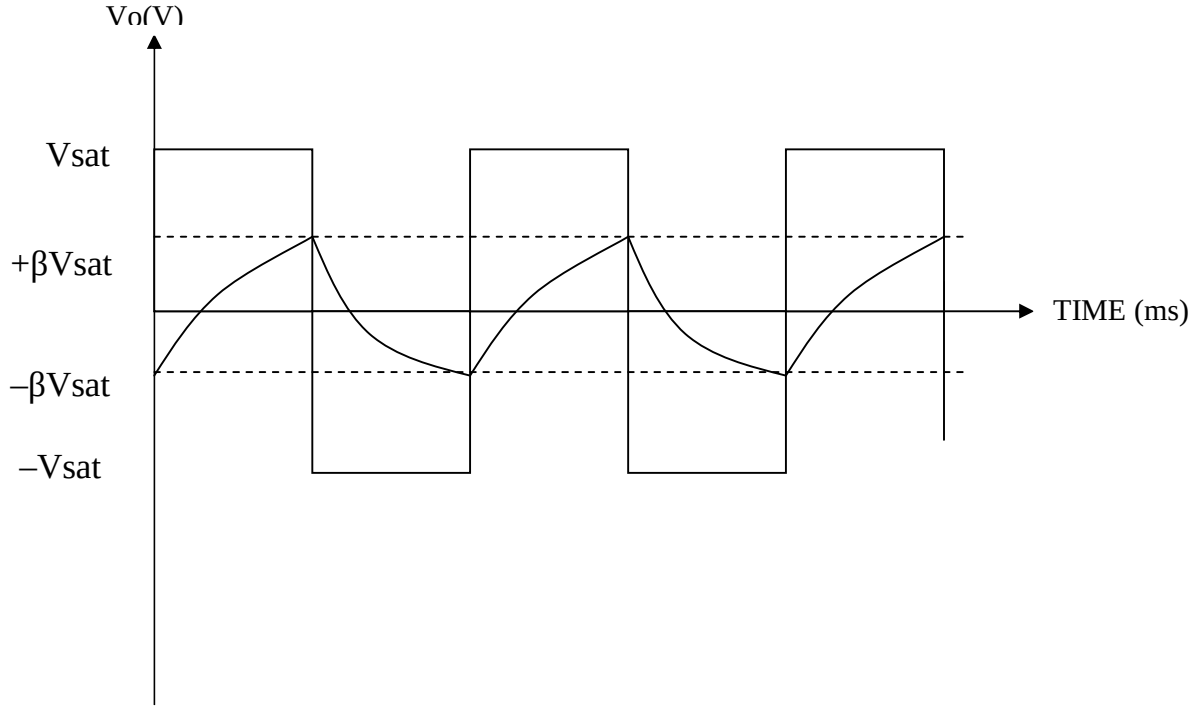
ASTABLE MULTIVIBRATOR:

The astable multivibrator is also known as free running oscillator. the principle of generation of square wave output is to force an op-amp to operate in saturation region. β = R2/(R1+R2) of the output is feedback to the positive input terminal. the reference voltage is Vo and may take the values as +βVsat and –βVsat. The output is also feedback to the negative input terminal after interchanging by a low pass RC combination. Whenever input terminal just exceeds Vref switching takes place resulting in square wave output. In this multivibrator both sates are quasi stable state

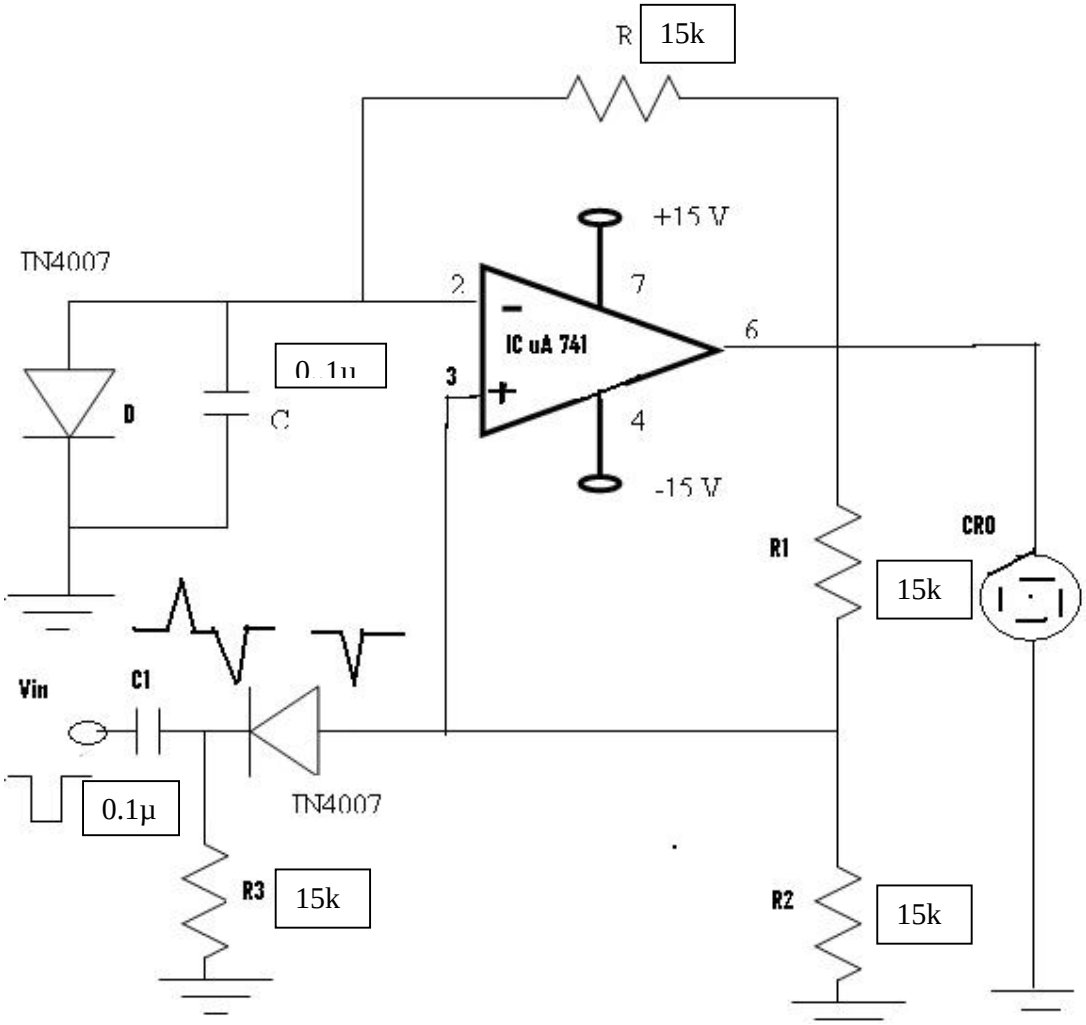
TABULATION-(ASTABLE MULTIVIBRATOR) :

S.NO.	WAVEFORM	AMPLITUDE (V) $volts$	TIME (T) ms	FREQUENCY F=1/T Hz
1	CAPACITOR			
2	OUTPUT			

MODEL GRAPH:



CIRCUIT DIAGRAM- (MONOSTABLE MULTIVIBRATOR)



DESIGN PROCEDURE:

- $T = RC \ln \frac{1+V_D/V_{SAT}}{1-\beta}$, where β = R2/(R!+R2)

If, $V_{SAT} > V_D$ & R1=R2 , β= 0.5 then ,

T= _____

MONOSTABLE MULTIVIBRATOR:

The monostable multivibrator is also called as one shot multivibrator. The circuit produces a single pulse of specified duration in response to each external trigger response. It is always have one stable state. When an external trigger is applied, the output changes the state. The new state is called quasi stable state. The circuit remain in this state for a fixed interval of time and then it returns to the original state after this interval. this time interval is determined by the charging and discharging of the capacitor.

EXPERIMENTAL PROCEDURE:

ASTABLE MULTIVIBRATOR:

1 Connections are given as per the EXPERIMENTAL SETUP.

2 Supply is switched ON after checking the circuit connections.

3 The output square wave form and the capacitor charging and discharging waveforms are noted from the CRO.

MONOSTABLE MULTIVIBRATOR:

1 Connections are given as per the EXPERIMENTAL SETUP.

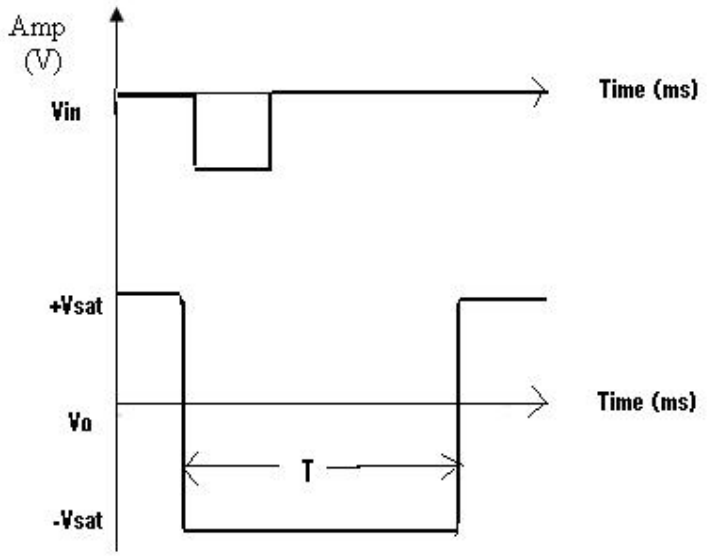
2 Supply is switched ON after checking the circuit connections.

3 The output square wave form and the capacitor charging and discharging waveforms are note down from the CRO.

TABULATION -(MONOSTABLE MULTIVIBRATOR):

S.NO.	WAVEFORM	AMPLITUDE (V) *volts*	TIME (T) *ms*	FREQUENCY F=1/T *Hz*
1	INPUT (TRIGGER)			
2	OUTPUT			

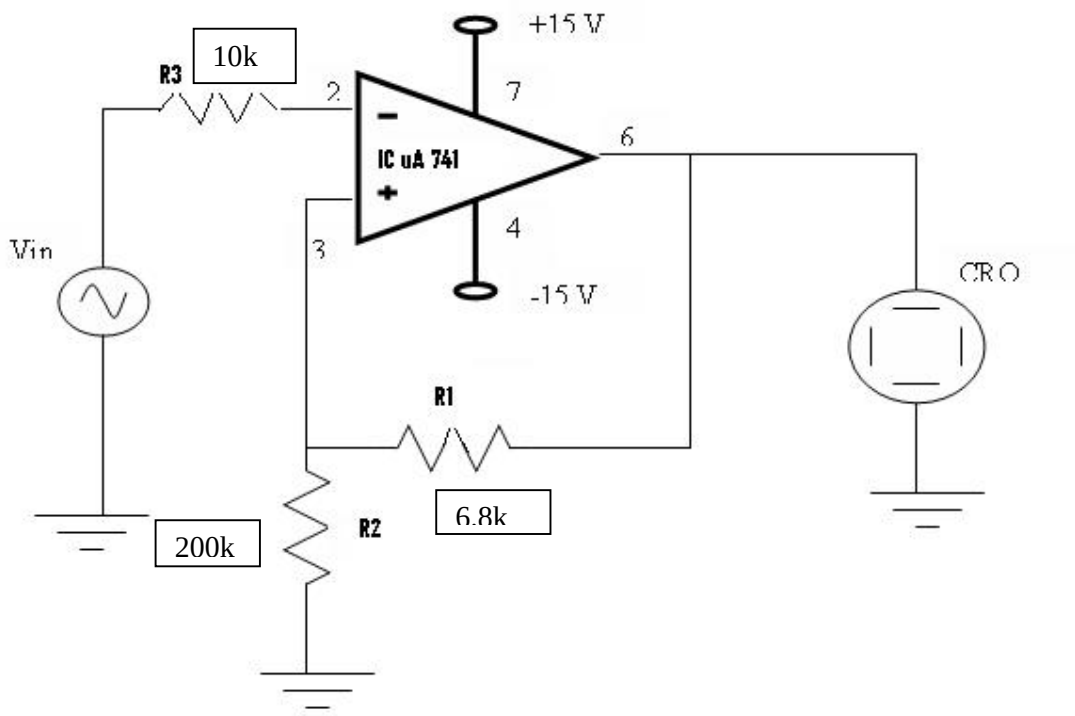
MODEL GRAPH:



RESULT :

Thus the astable and monostable multivibrator circuits were designed and tested using ICμA741.

CIRCUIT DIAGRAM-(SCHMITT TRIGGER):



DESIGN PROCEDURE:

$\pm V_{sat}$ = _____

- $V_{utp} = R_2(+V_{sat} / (R_1+R_2))$ _____
- $V_{ltp} = R_2(-V_{sat} / (R_1+R_2))$ _____

Ex. No :06	**SCHMITT TRIGGER**
DATE :	

AIM:

To design and test a Schmitt Trigger circuit using IC µA 741.

APPARATUS REQUIRED:

S.NO.	APPARATUS	RANGE	QUANTITY
1	Dual power supply	(0 - +15)V	1
2	CRO	(0-30)MHz	1
3	IC	µA 741	1
4	Resistor	6.8kΩ,200kΩ,10kΩ	2,2,1

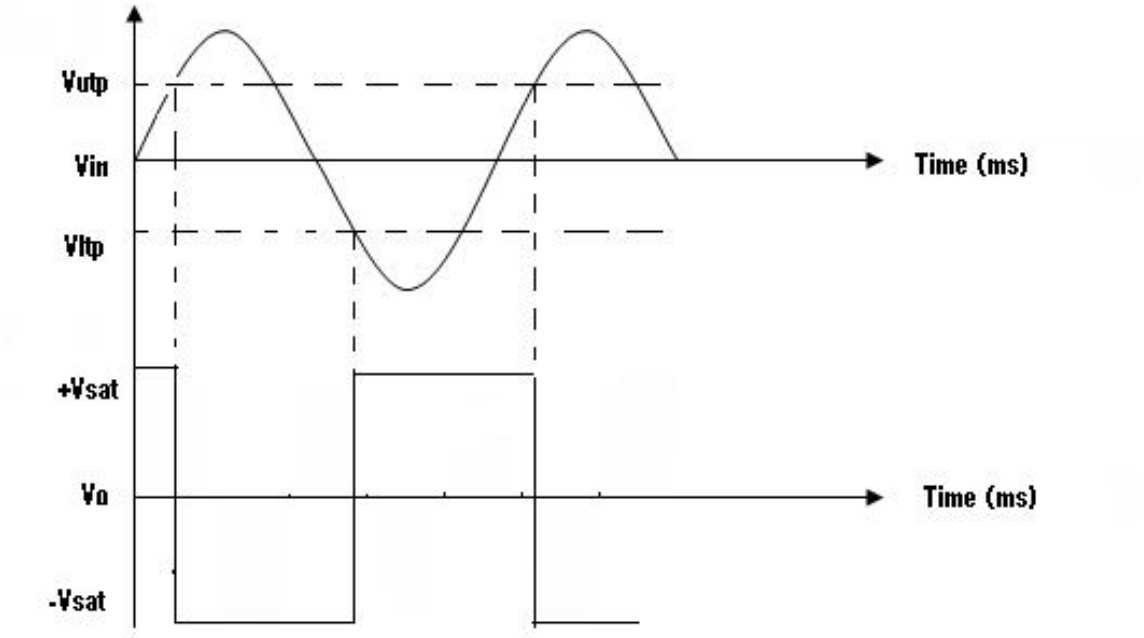
THEORY:

If the positive feedback is added to the comparator circuit means gain can be increased greatly. Consequently the transfer curve comparator becomes more close to the ideal curve theoretically. If the loop gain βfo is adjusted to unity then the gain with feedback average becomes extreme values of output voltage. in practical circuits, however it may not be possible to maintain loop gain exactly equal to unity for a long time because of supply voltage and temperature variations so a value greater than unity is chosen. This gives the output waveform virtually disconnected at the comparison voltage. This circuit however exhibits phenomenon called hystersis or backlash.

TABULATION –(SCHMITT TRIGGER) :

S.NO.	SIGNAL	VOLTAGE (V) $volts$	TIME (T) ms
	INPUT SIGNAL		
	OUTPUT SIGNAL		

MODEL GRAPH:



EXPERIMENTAL PROCEDURE:

1 Connections are given as per the EXPERIMENTAL SETUP.

2 The supply is switched ON.

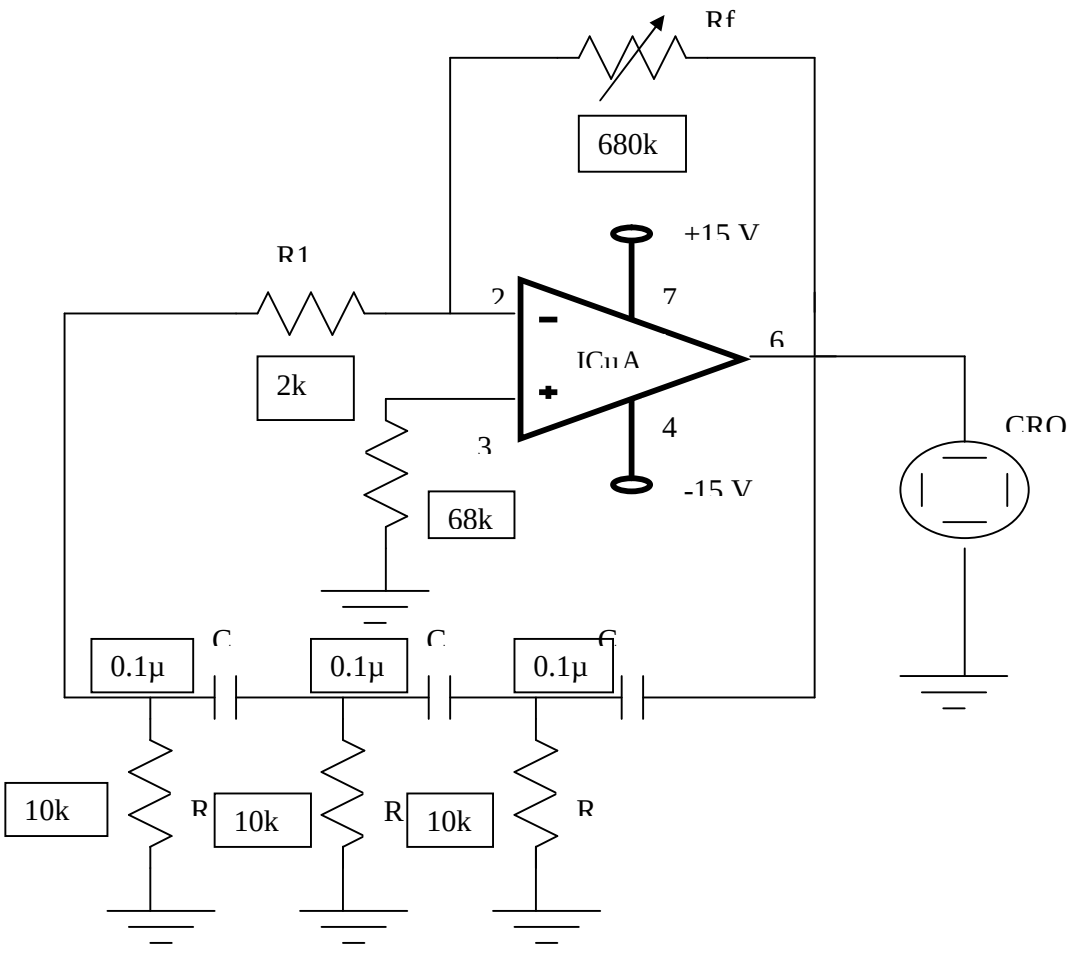
3 The output waveform was noted from CRO and UTP and LTP are noted. The graph is drawn.

4 The theoretical value of UTP and LTP are verified with the practical value.

RESULT :

Thus the Schmitt Trigger circuit using IC µA 741 was designed and tested.

CIRCUIT DIAGRAM – (RC PHASE SHIFT OSCILLATOR);



DESIGN PROCEDURE:

Design for RC

- f = 1/(2π√6 RC),

Assume C = _____

R= _____

Design for Gain

- Av= - R_f/R1, Let R1 = _____

R_f = _____

Ex. No :07	**RC PHASE SHIFT OSCILLATOR AND WEIN BRIDGE OSCILLATOR**
DATE :	

AIM:

To design and test RC phase shift and wein bridge oscillators using IC µA 741.

APPARATUS REQUIRED:

S.NO.	APPARATUS	RANGE	QUANTITY
1	Dual power supply	(0 - +15)V	1
2	Signal generator	(0-3)MHz	1
3	CRO	(0-30)MHz	1
4	IC	IC µA 741	1
5	Resistor	680kΩ,6.8kΩ,220kΩ, 4.7kΩ	3,2,2,1
6	Capacitor	0.1µf	3

THEORY:

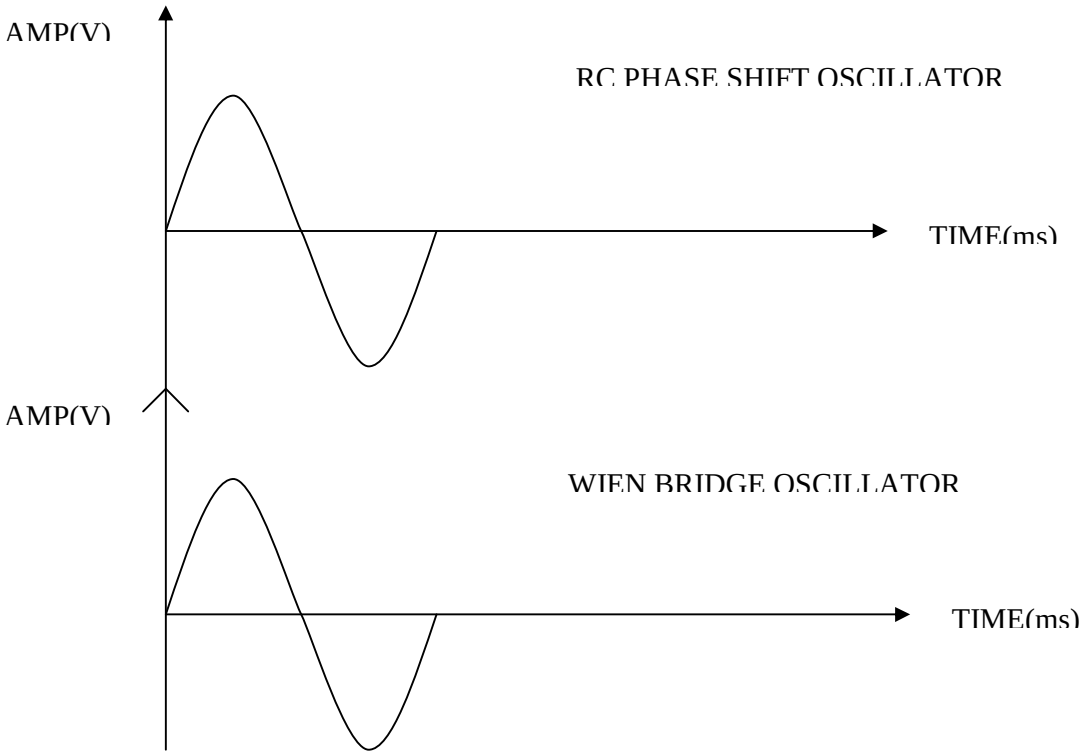
RC PHASE SHIFT OSCILLATOR:

RC phase shift oscillator using op-amp, in inverting amplifier mode. Thus it introduces a phase shift of 180^0 between the input and output. The feedback network consists of 3 RC sections producing each 60^0 phase shift. Such a circuit is known as RC phase shift network. The circuit is generating its own output signal and a stage of oscillator sustained. the phase shift produced by op-amp is 180^0.the op-amp with a gain of 29 and RC network is of equal resistor and capacitor connected feedback the op-amp output and input terminals..

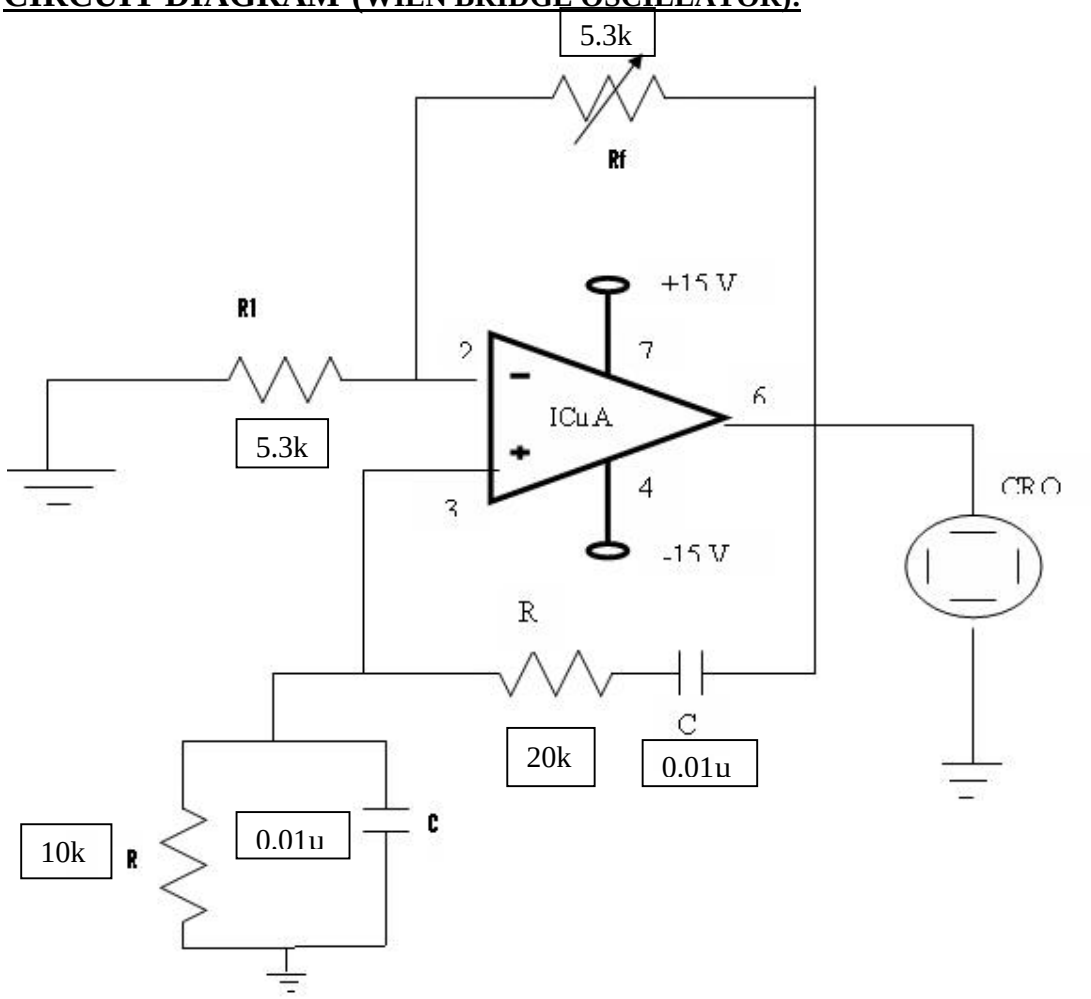
TABULATION-(RC PHASE SHIFT OSCILLATOR):

S.NO.	AMPLITUDE (V) $volts$	TIME (T) ms	F = 1/T Hz

MODEL GRAPH:



CIRCUIT DIAGRAM-(WIEN BRIDGE OSCILLATOR):



DESIGN PROCEDURE:

Design for RC,

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

Assume C = _____

R = _____

Design for Gain,

- R1 = _____ , Rf = _____

- Gain A= 1+Rf/R1 _____

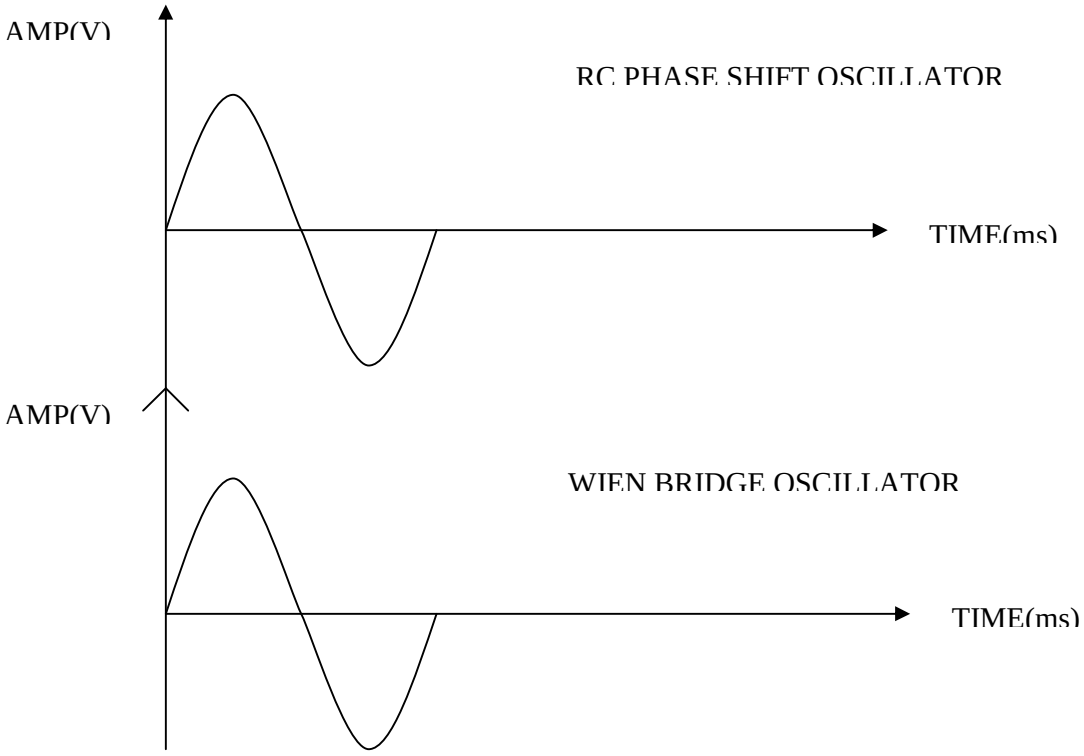
WEIN BRIDGE OSCILLATOR:

It is commonly used in audio frequency oscillator. The feedback signal is connected in the input terminal so that the output amplifier is working as an non-inverting amplifier. The wein bridge circuit is connected between amplifier input terminal and output terminal. The bridge has a series R network, in one arm and a parallel RC network in the adjoining arm. in the remaining two arms of the bridge, resistor R1 and R_f are connected. the phase angle criterion for oscillation is that the total phase shift around the circuit must be zero. This condition occurs when bridge is balanced. At resonance frequency of oscillation f_o is exactly the resonance frequency of balanced wein bridge and is given by f0 = 1/ (2πfC).assuming that the resistors are input impedance value and capacitance are equal to the value in the reactive stage of wein bridge. at this frequency, the gain required for sustained.

TABULATION –(WEIN BRIDGE OSCILLATOR) :

S.NO.	AMPLITUDE (V) $volts$	TIME (T) ms	F = 1/T Hz

MODEL GRAPH:



EXPERIMENTAL PROCEDURE:

RC PHASE SHIFT OSCILLATOR:

1 Circuit connections are given as per the EXPERIMENTAL SETUP.

2 Supply is switched ON.

3 360^0 phase shift output is obtained at the output.

4 The inverting op-amp produce 180^0 and RC network produce another 180^0

5 Frequency is calculated by the formula f =1/T

WIEN BRIDGE OSCILLATOR:

1 Connections are given as per the EXPERIMENTAL SETUP.

2 Resistor and capacitor values are verified simultaneously; the corresponding Rf value is noted.

3 The critical vale of frequency is noted correspondingly.

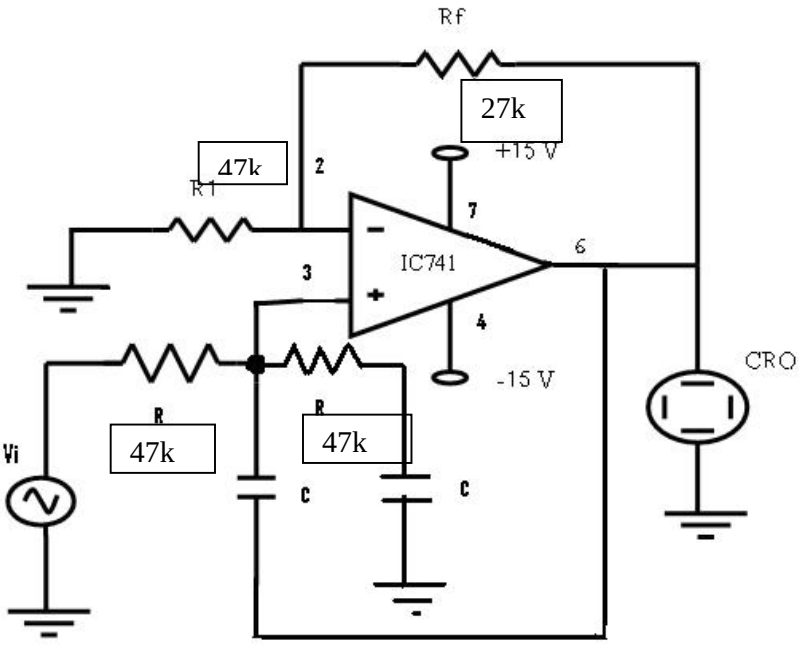
4 Check whether the calculated and observed frequency values are same.

5 Graph is drawn by taking amplitude along y-axis and time along x-axis.the graph will b sine waveform.

RESULT:

Thus the RC phase shift and wien bridge oscillator was designed and tested using IC µA 741.

CIRCUIT DIAGRAM-(ACTIVE LOW PASS FILTER):



DESIGN PROCEDURE:

- $f_c = 1/2\pi RC$,

Let C = _____

Therefore R = _____

Design for Gain:

- $A = 1+R_f/R_1$, Therefore,

R_f = _____

R_1= _____

Let A = _____

Ex. No :08	**ACTIVE LOW PASS, HIGH PASS AND BAND PASS FILTER**
DATE :	

AIM:

To design and test low pass, high pass and band pass filters using IC µA 741.

APPARATUS REQUIRED:

S.NO.	APPARATUS	RANGE	QUANTITY
1	Dual power supply	(0 - +15)V	1
2	CRO	(0-30)MHz	1
3	IC	µA 741	1
4	Resistor	10kΩ,5kΩ,	4,2
6	Capacitor	0.1µf,10µf	2,1

THEORY:

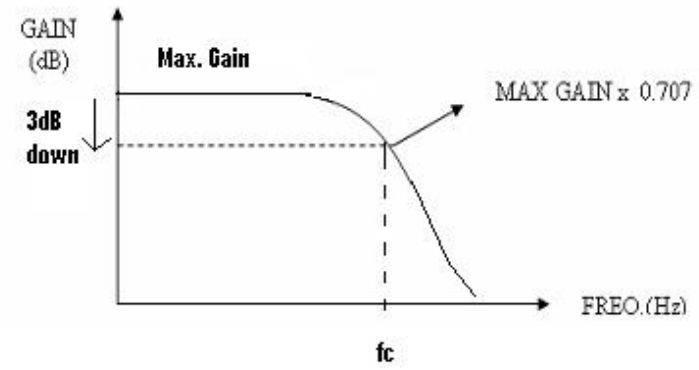
The first order low pass filter is realized RC circuit used along with an op-amp in non-inverting configuration. A low pass filter has constant gain from) Hz to $f_{H.}$. Bandwidth of this filter is $f_{H.}$ Bandwidh of electric filters are used in circuits which require the separation of signals according to their frequencies. a first order low pass filter consists of a single RC network connected to the positive input terminal of non-inverting op-amp amplifier. Resistors R_i and R_f determine the gain of the filter in the pass band.

TABULATION-(ACTIVE LOW PASS FILTER) :

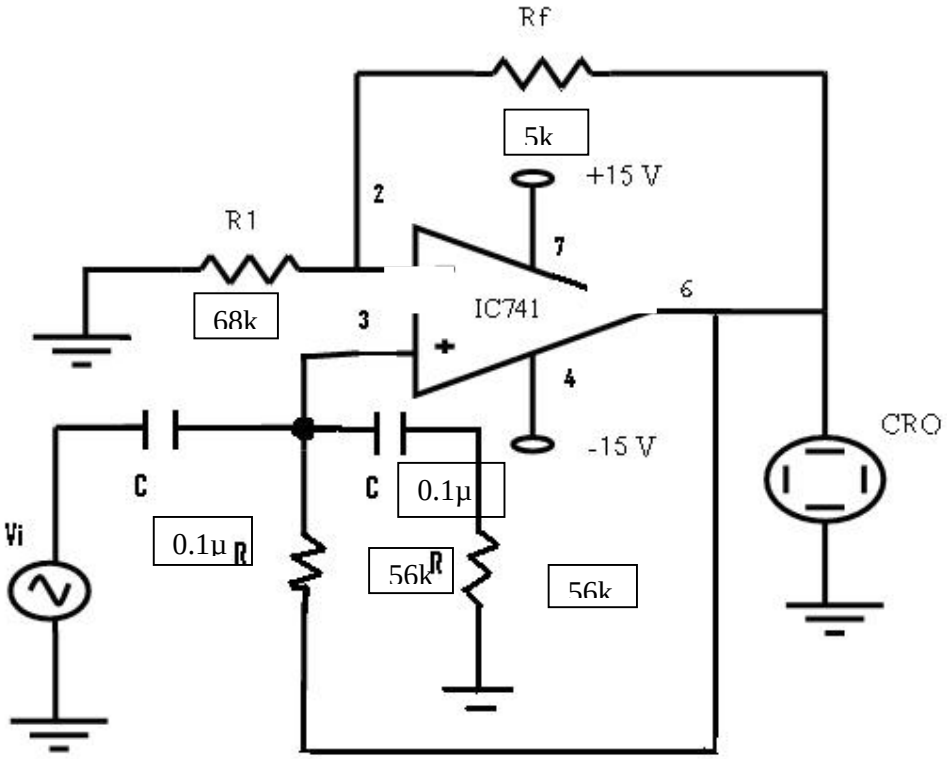
V_{in} =

S.NO	INPUT FREQUENCY (F_i) Hz	OUTPUT VOLTAGE (V_o) mV	GAIN = 20LOG(V_o/V_{in})

MODEL GRAPH:



CIRCUIT DIAGRAM-(ACTIVE HIGH PASS FILTER):



DESIGN PROCEDURE:

- $f_c = 1/2\pi RC$, Let C = _____

  Therefore R = _____

Design for Gain:

R1 = _____

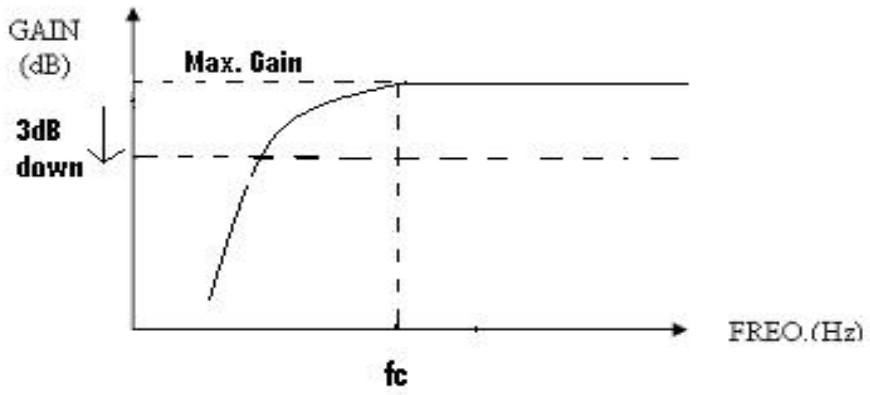
- A = 1+Rf/R1, R_f= _____

TABULATION –(ACTIVE HIGH PASS FILTER):

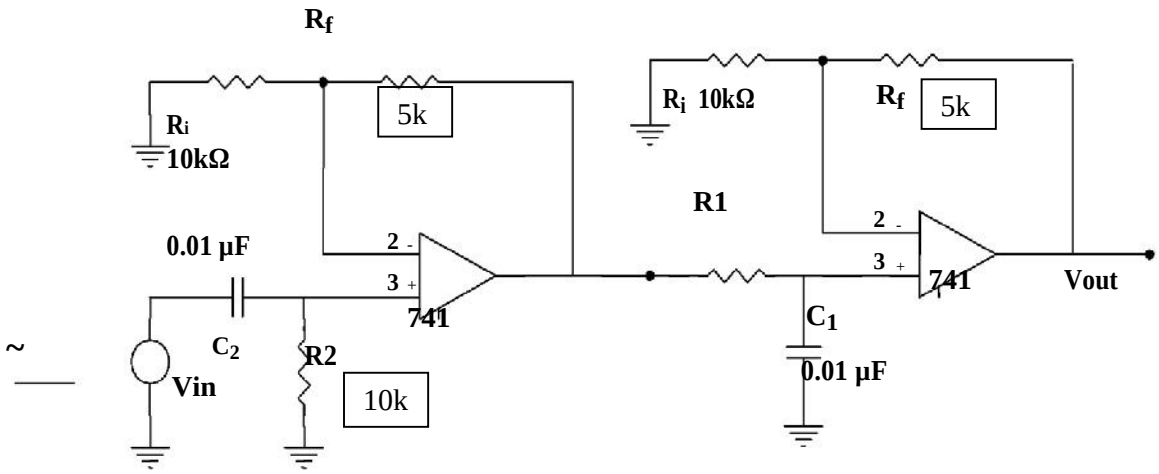
V_{in} =

S.NO.	INPUT FREQUENCY (F_i) Hz	OUTPUT VOLTAGE (Vo) mV	GAIN = 20LOG(V_o/V_{in})

MODEL GRAPH:



CIRCUIT DIAGRAM –(ACTIVE BAND PASS FILTER):



DESIGN PROCEDURE:

The parameters in the band pass filter are lower cutoff frequency, the upper cutoff frequency and the bandwidth, the central frequency gain A_o and selectivity Q. The higher the selectivity Q, the sharper the filter. Below 0.5fo all filters roll off at -20dB/decade independent of the value of Q. This is limited by the two RC pair of circuits.

EXPERIMENTAL PROCEDURE:

1 connections are given as per the EXPERIMENTAL SETUP

2 Supply is switched ON after checking the connections.

3 Input voltage is set to 1V and by changing the input frequency, output voltage is measured.

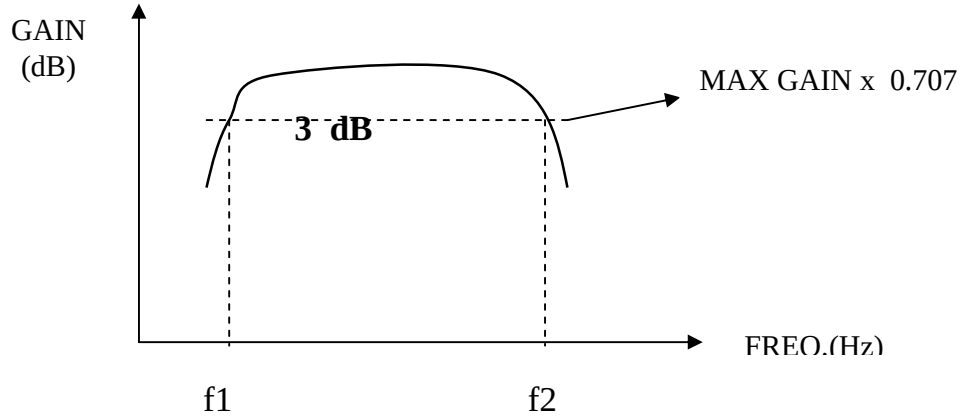
4 The procedure is applied to active low pass, high pass and band pass filters.

TABULATION –(ACTIVE BAND PASS FILTER):

Vin =

S.NO.	INPUT FREQUENCY (F_i) Hz	OUTPUT VOLTAGE (Vo) mV	GAIN = 20LOG(Vo/Vin)

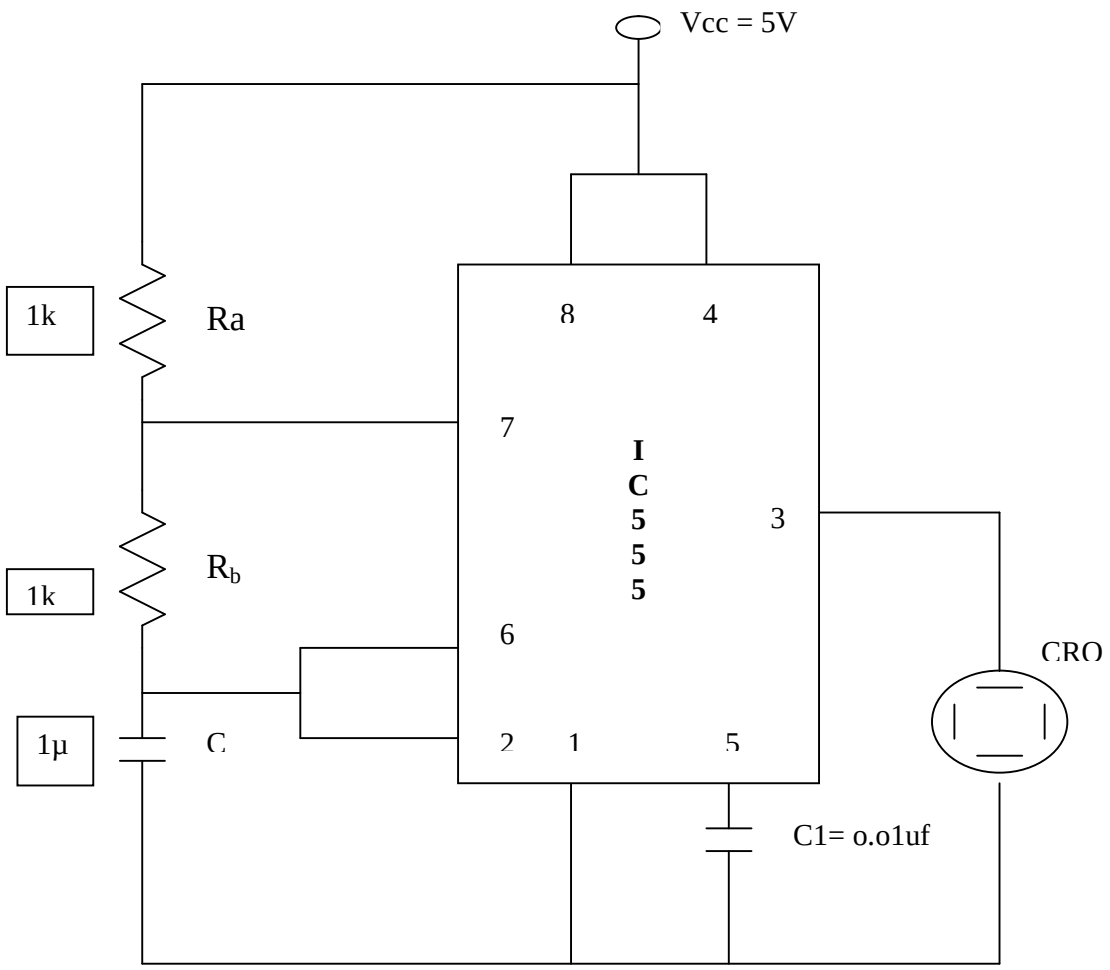
MODEL GRAPH:



RESULT :

Thus the active low pass, high pass and band pass filters were designed and tested using IC µA 741.

CIRCUIT DIAGRAM-(ASTABLE MULTIVIBRATOR):



DESIGN PROCEDURE:

- Duty cycle D = T_{ON} / (T_{ON} +T_{OFF})
  = [0.693(Ra+Rb)C] / [0.693(Ra+2Rb)C]
  = _____

- f = 1.45 / [(Ra+2Rb)C]
  f = _____

Ex. No :09	**ASTABLE MULTIVIBRATOR AND MONOSTABLE MULTIVIBRATOR USING 555 TIMERS**
DATE :	

AIM:

To design and test astable and monostable multivibrator circuits using IC 555.

APPARATUS REQUIRED:

S.NO.	APPARATUS	RANGE	QUANTITY
1	Dual power supply	(0 - +15)V	1
2	Signal generator	(0-1)MHz	1
3	CRO	(0-30)MHz	1
4	IC	IC 555	1
5	Resistor	1kΩ,2kΩ	1,1
6	Capacitor	0.1µf	1

THEORY:

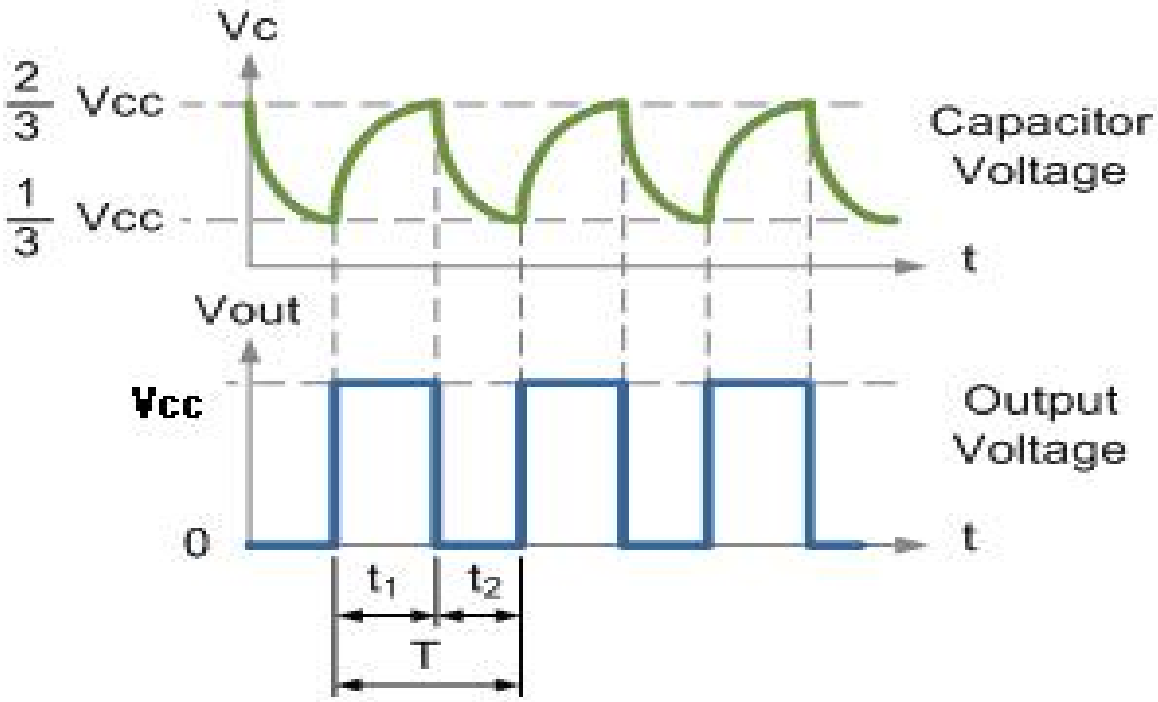
ASTABLE MULTIVIBRATOR:

The astable multivibrator is also called the free running multivibrator. It has two quasi states i.e. no stable states as such the circuit conditions oscillate between the components values used to decide the time for which circuit remains in each stable state. The principle of square wave output is to force the IC to operate in saturation region. Whenever input at the negative input terminal just exceeds Vref switching takes place resulting in a square wave output. In astable multivibrator both stable states and one quasi states are present.

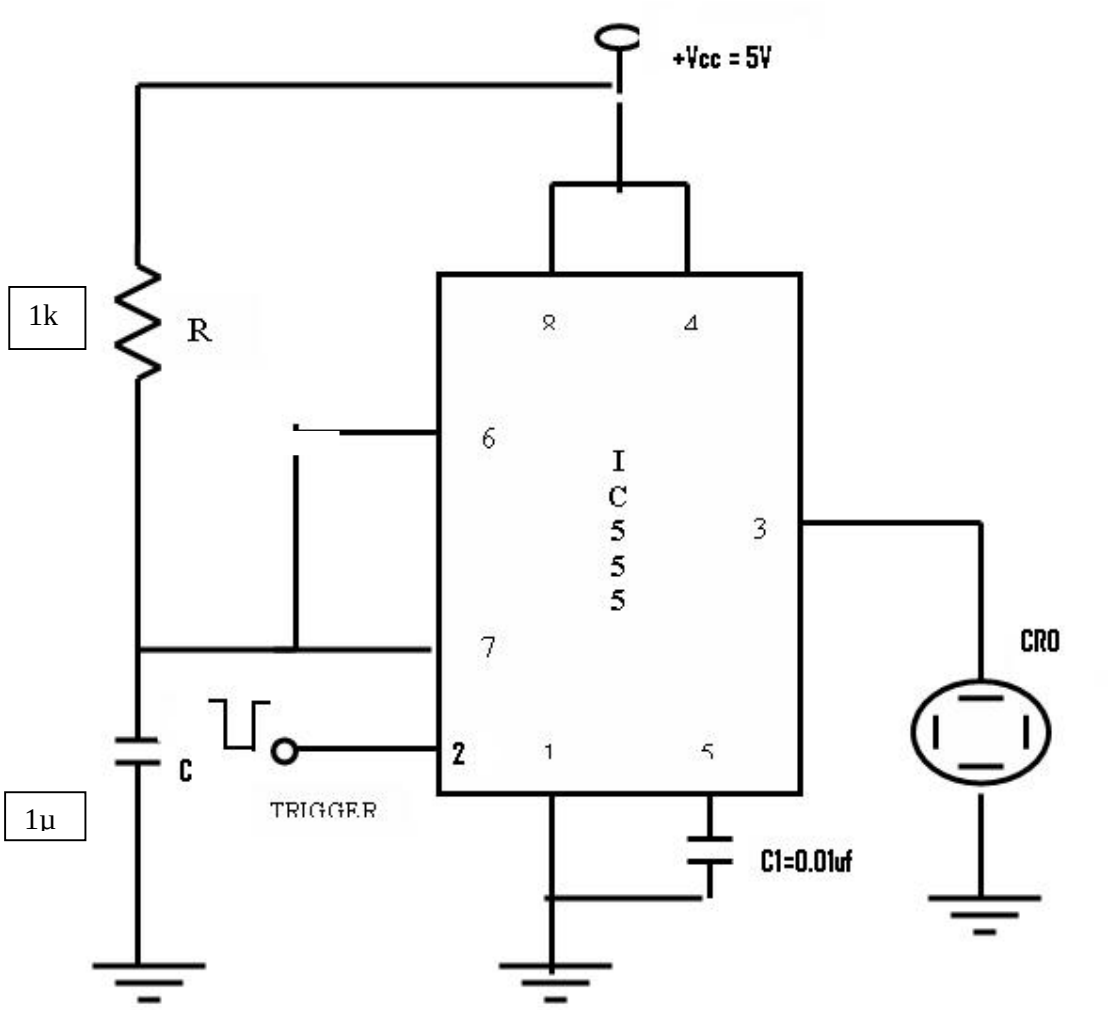
TABULATION-(ASTABLE MULTIVIBRATOR):

S.NO	WAVEFORM	AMPLITUDE (V) $VOLTS$	TIME (T) ms	F =1/T Hz
1	CAPACITOR			
2	OUTPUT			

MODEL GRAPH:(ASTABLE MULTIVIBRATOR)



CIRCUIT DIAGRAM- (MONOSTABLE MULTIVIBRATOR):



Design

T = 1.1*R*C, R = _____

= _____ **C** = _____

MONOSTABLE MULTIVIBRATOR:

These multivibrators are comprised of group of regenerative circuits that are commonly used in timing applications. The circuit produces a single pulse of applied duration in response to each external trigger pulse. For each circuit only one state exists. When an external trigger is applied the output changes its state. The new state is called quasi-stable state.

EXPERIMENTAL PROCEDURE:

1 Connections are as per the EXPERIMENTAL SETUP.

2 Supply is switched ON after checking the connections.

3 For monostable multivibrator trigger pulse is given and for stable it is not necessary.

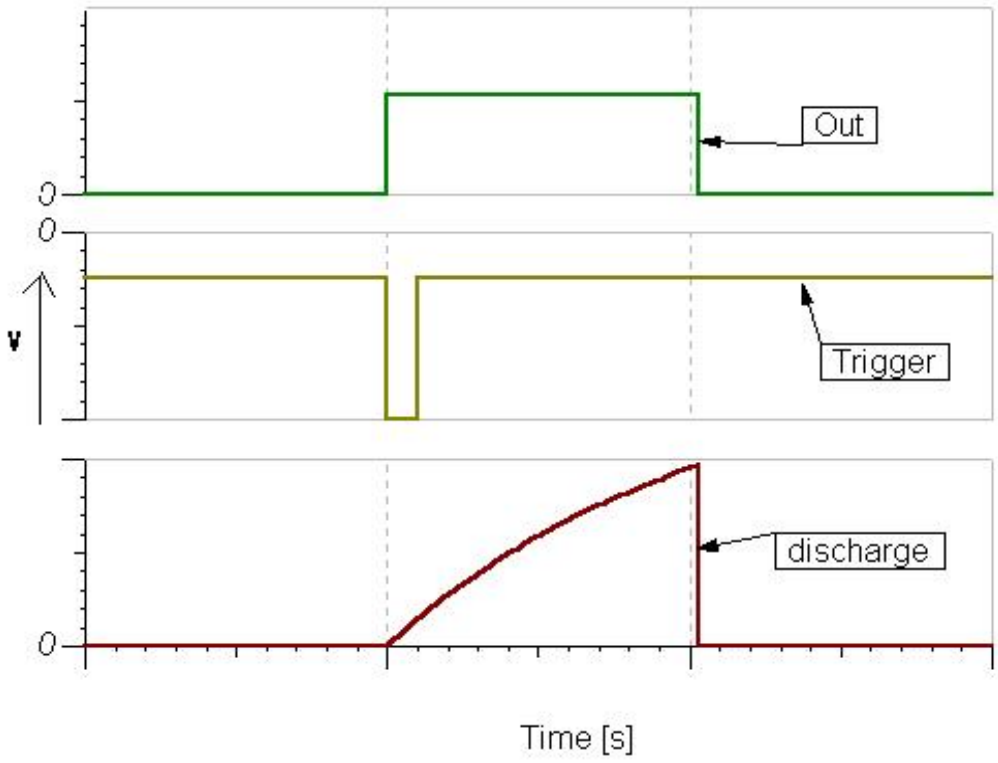
4 Output square wave is noted from CRO.

5 The frequency is calculated by input.

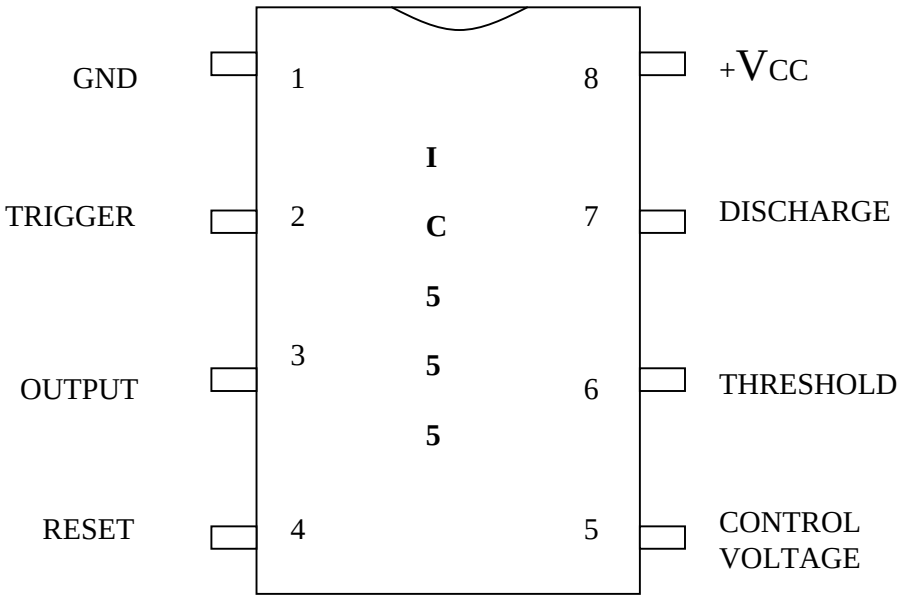
TABULATION –(MONOSTABLE MULTIVIBRATOR):

S.NO.	WAVEFORM	AMPLITUDE (V) $VOLTS$	TIME (T) ms	FREQ. f=1/T Hz
1	**CAPACITOR**			
2	**TRIGGER**			
3	**OUTPUT**			

MODEL GRAPH-(MONOSTABLE MULTIVIBRATOR):



PIN DIAGRAM –(IC 555 TIMER):



SPECIFICATIONS:

SUPPLY VOLTAGE : +5 V to +18 V

OPERATING TEMPERATURE : 0^0 TO 70^0 C

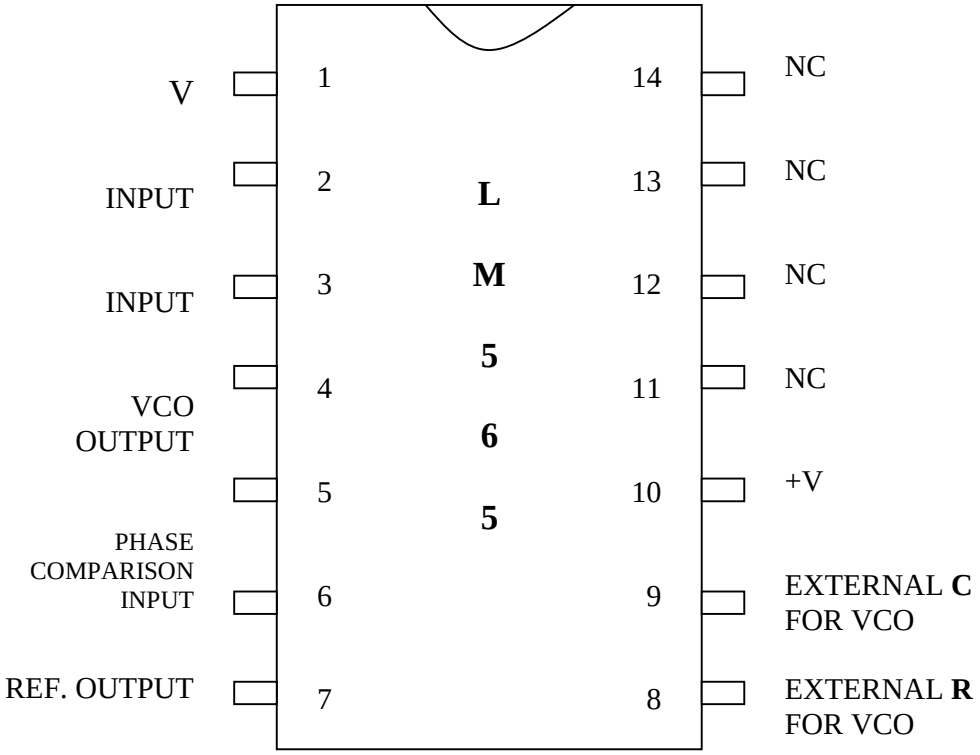
STORAGE TEMPERATURE RANGE : 65^0 TO 150^0 C

POWER DISSIPATION : 600mW

RESULT:

Thus the Astable and Monostable multivibrators were designed and tested using IC555.

PIN DIAGRAM-(LM565);



SPECIFICATIONS:

MAXIMUM SUPPLY VOLTAGE	:	2.6V
MAXIMUM POWER DISSIPATION	:	330mW
SUPPLY CURRENT	:	8mW
INPUT IMPEDENCE	:	5KΩ
OUTPUT IMPEDENCE	:	3.6KΩ

Ex. No :10	**PLL CHARACTERISTICS AND FREQUENCY MULTIPLIER USING PLL**
DATE :	

AIM:

To conduct an experiment on PLL using IC LM 565 and to draw the frequency response characteristics and also to design and to test the frequency multiplier using PLL.

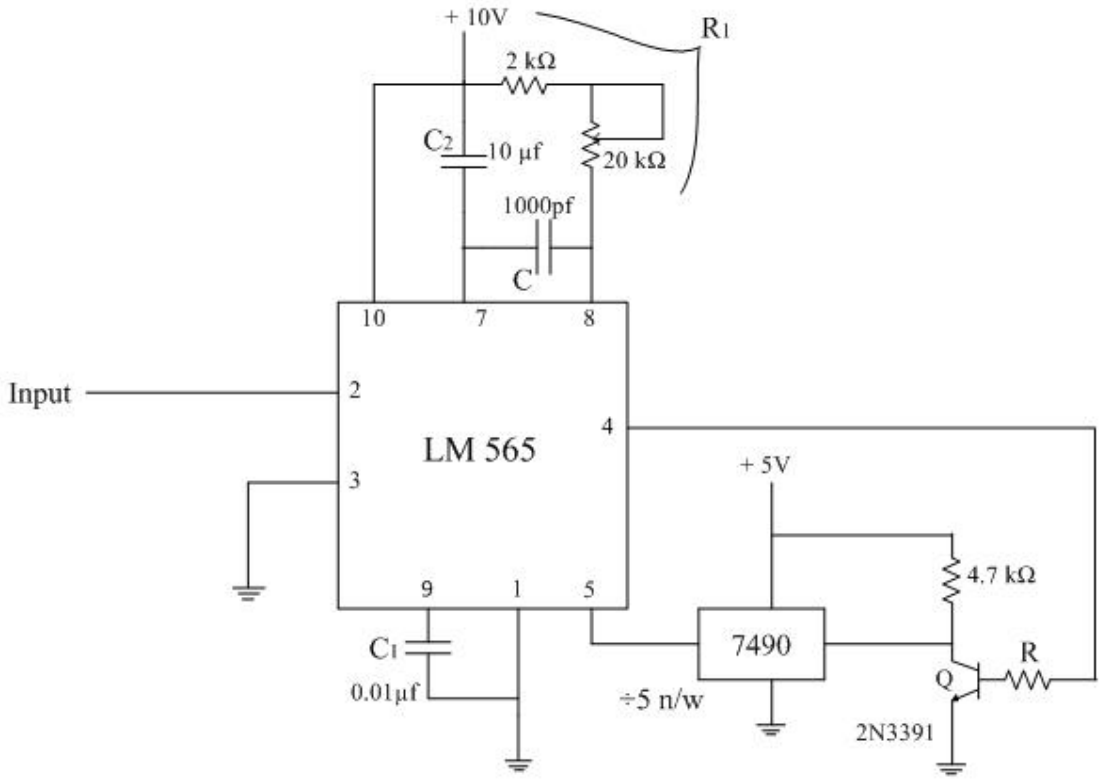
APPARATUS REQUIRED:

S.NO.	APPARATUS	RANGE	QUANTITY
1	Dual power supply	(0 - +15)V	1
2	CRO	(0-30)MHz	1
3	IC	LM 565	1
4	Resistor	20kΩ,2kΩ,4.7kΩ,10kΩ	Each 2
6	Capacitor	0.01µf,0.001µf,10µf	Each 1

THEORY:

The block diagram of LM565 PLL consists of base detector amplifier. Low pass filter and VCO as shown in the block diagram. The phase locked loop is not connected internally. It is necessary to connect output of VCO (pin 4) to phase comparator in pin 5 externally.

CIRCUIT DIAGRAM-(LM565):



In frequency multiplication applications a digital frequency driver is inserted into loop between pin 4 and pin 5.the centre frequency of PLL is determined by free running frequency multiplier of VCO given by free funning frequency of VCO which is given by $f_0 = 1.2/(4R1C1)$ Hz. the value of R_i is restricted from 2KΩ to 20KΩ but a capacitor can have any value. A capacitor C_2 is connected between pin 7 and to the positive supply from a first order low pass filter with an external resistance of 3.6 KΩ. The value of filter capacitor C_2 should be large enough to eliminate positive oscillator into VCO voltage.

$$F_L = I.8f_o/V \text{ Hz.}$$

Where, f_o = free running frequency in Hz

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

$$F_L = \pm (f_o /2\pi 3.6x10^3 C_2)^{1/2}$$

Where, C_2 is in farads

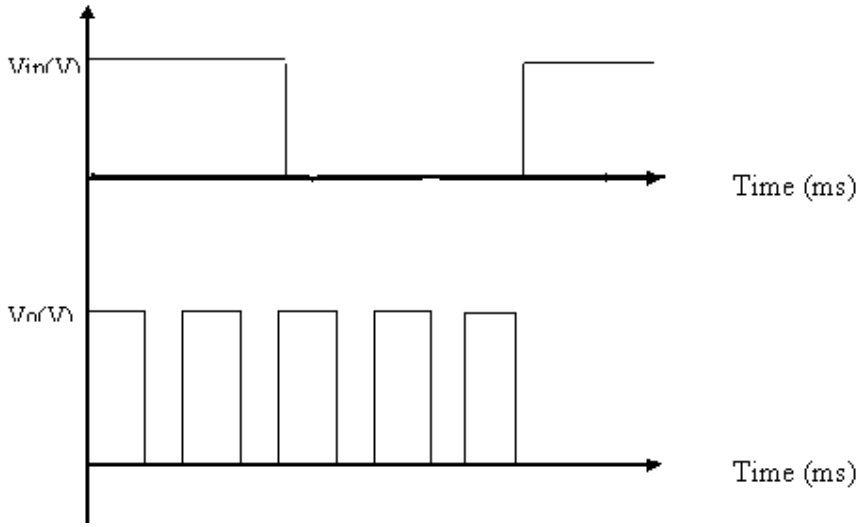
EXPERIMENTAL PROCEDURE:

1. Connections are given as per the EXPERIMENTAL SETUP.
2. Observe the waveform at pin 4 and pin 5 without any input signal. This is free running frequency of VCO (fo).
3. Switch ON the functional generator and give the square waveform of $1V_{pp}$ & 1KHZ .
4. Calculate the capture range and lock range.

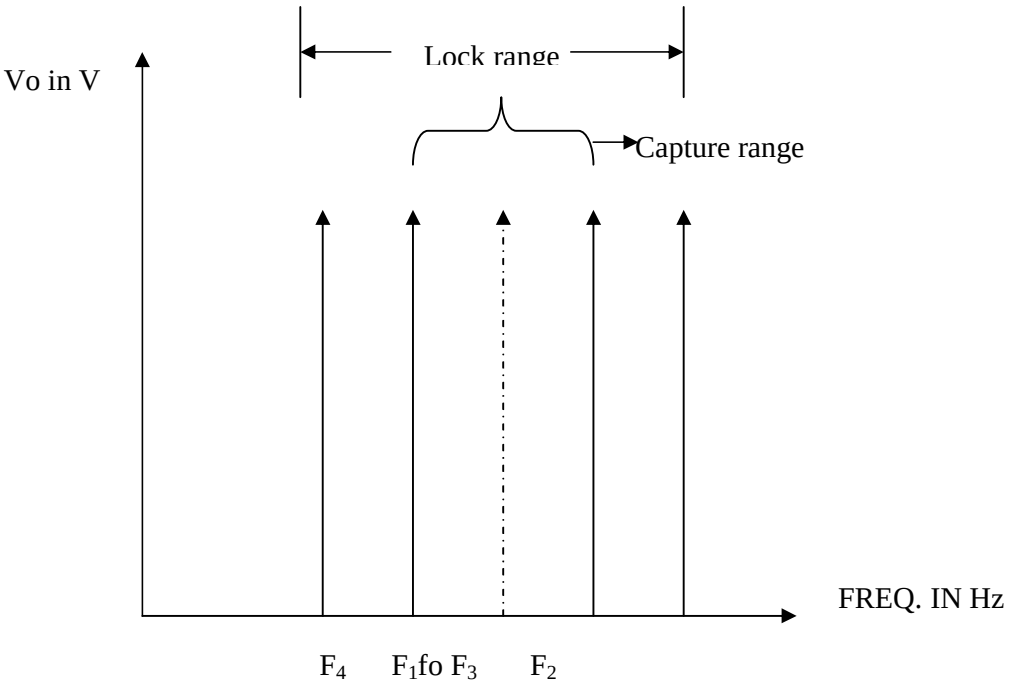
TABULATION –(FREQUENCY MULTIPLICATION):

S.NO.	WAVEFORM	FREQUENY F=1/T HZ	AMPLITUDE (V) $volts$
1	INPUT		
2	OUTPUT		

MODEL GRAPH:



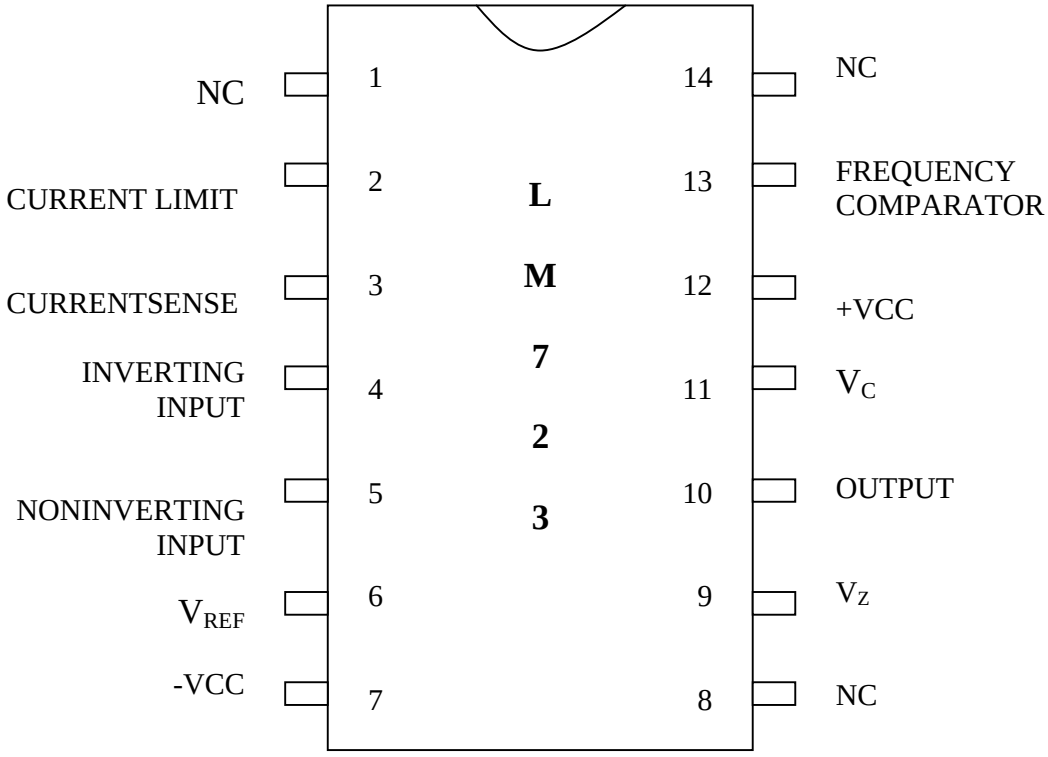
MODEL GRAPH:



RESULT:

Thus the experiment on LM 565 is conducted and the frequency response characteristic is drawn.

PIN DIAGRAM-(LM 723):



SPECIFICATIONS:

PEAK VOLTAGE FROM +VCC TO –VCC	:	50V
CONTINUOUS VOLTAGE FROM +VCC TO –VCC	:	40V
INPUT TO OUTPUT VOLTAGE DIFFERENTIAL	:	40V
DIFFERENTIAL INPUT VOLTAGE TO ERROR AMLIFIER	:	±5V
VOLTAGE BETWEEN NON-INVERTING INPUTS & -VCC	:	8V
CURRENT FROM V_Z	:	25mA
CURRENT FROM V_{REF}	:	15m

Ex. No :11	**DC POWER SUPPLY USING LM 723**
DATE :	

AIM:

To conduct an experiment in order to get regulated power supply output using LM 723.

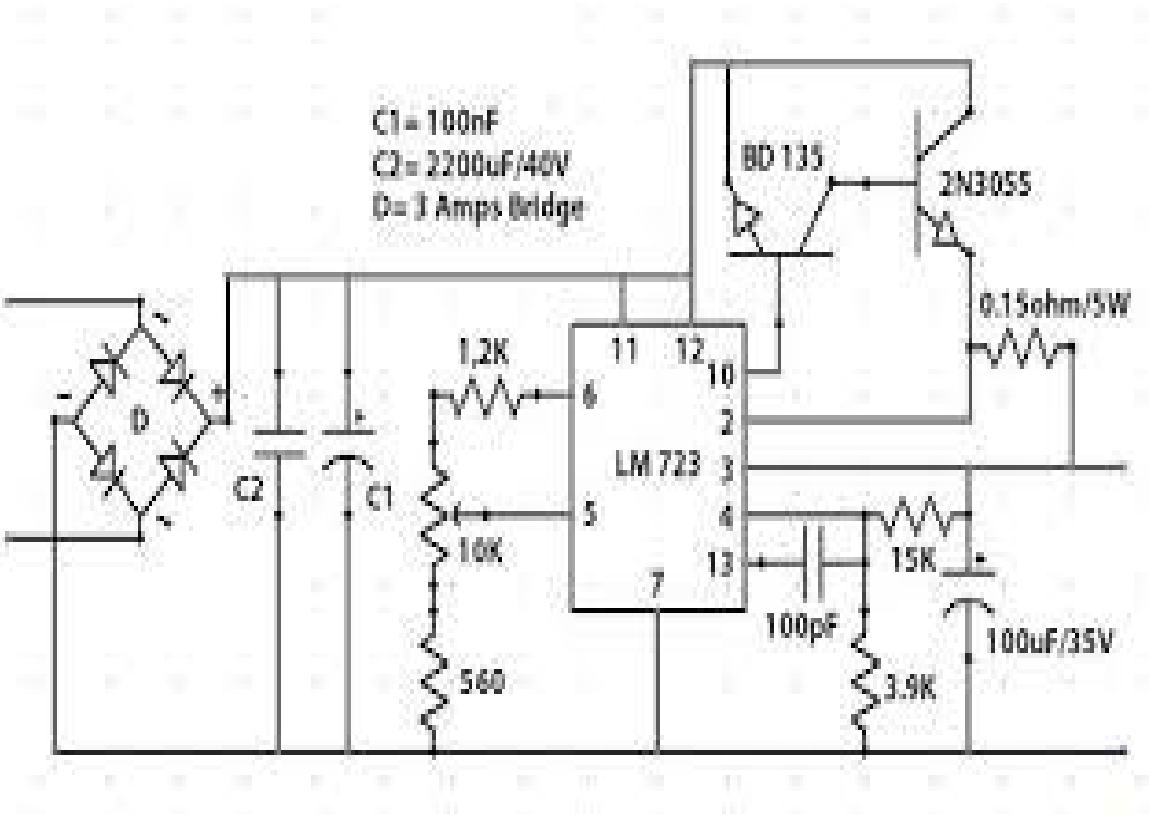
APPARATUS REQUIRED:

S.NO.	APPARATUS	RANGE	QUANTITY
1	Dual power supply	(0 - +15)V	1
2	Resistor	5.5KΩ , 1KΩ,846Ω	Each 1
3	Capacitor	100pF	1
4	Bread Board	-	1
5	Volt meter	(0-30)V	1

THEORY:

The basic voltage regulator in its simplest form consists of a) voltage reference Vr b) error amplifier c) feedback network d) active series or shunt control unit. the voltage reference generates a voltage level which is applied to the comparator circuit, which is generally error amplifier. The second input to the error amplifier obtained through feedback network. Generally using the potential divider, the feedback signal is derived by sampling the output voltage. The error amplifier converts the difference between the output sample and the reference voltage into an error signal. This error signal in turn controls the active element of the regulator circuit, in order to compensate the changes in the output voltage. Such an active element is generally a transistor.

CIRCUIT DIAGRAM –(DC POWER SUPPLY);



Let , Error amplifier controls the series pass transistor Q2 which acts as a variable resistor. The series pass transistor is small power transistor having about 800mW power dissipation. The unregulated power supply source of (< 36 V d.c) is connected to collector of series pass transistor.

Transistor Q2 acts as current limiter in case of short circuit condition. It senses drop across Rsc placed in series with regulated output voltage externally.

The frequency compensation terminal controls the frequency response of the error amplifier. The required roll-off is obtained by connecting a small capacitor of 100pF between frequency compensation and inverting input terminals.

EXPERIMENTAL PROCEDURE:

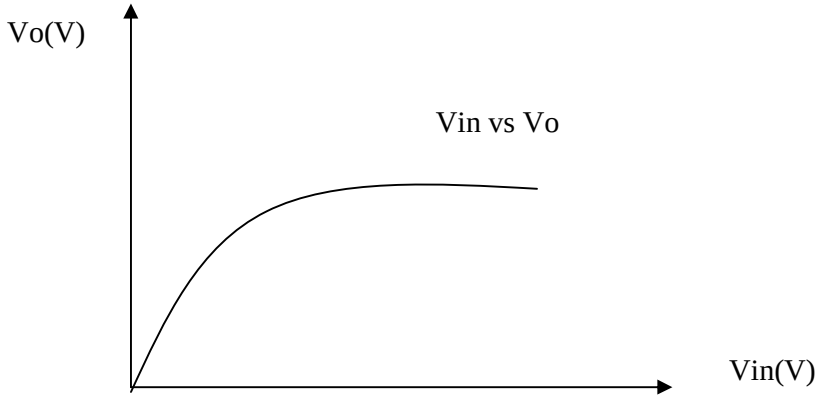
1 Connections are given as per the EXPERIMENTAL SETUP.

2 The input voltage is given to the circuit and the output voltage slowly varies from zero.

3 Then the output voltage attains the designed value and then it is irrespective of input voltage (the output becomes constant).

TABULATION–(DC POWER SUPPLY):

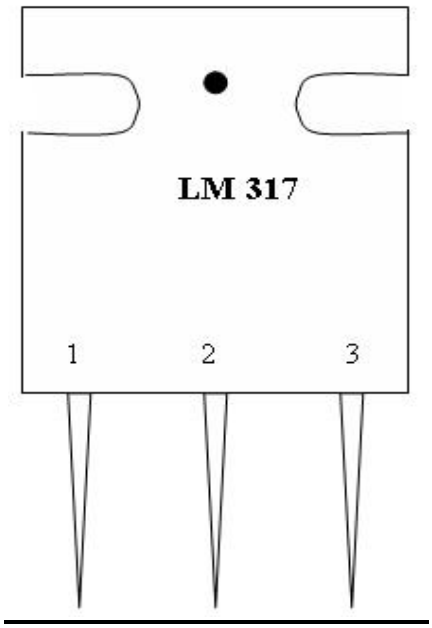
S.NO.	Vin *Volts*	Vo *Volts*



RESULT:

Thus the experiment is conducted using LM 723 and so the regulated output is obtained using the circuit.

PIN DIAGRAM:



SPECIFICATIONS:

MINIMUM OUTPUT VOLTAGE	:	1.2 V
MAXIMUM OUTPUT VOLTAGE	:	57 V
MAXIMUM OUTPUT CURRENT	:	1.5mA
OUTPUT RESISTANCE	:	17MΩ
RIPPLE REGECTION	:	62 dB
SHORT CIRCUIT CURRENT	:	750mA

Ex. No :12	**DC POWER SUPPLY USING LM 317**
DATE :	

AIM:

To conduct an experiment in order to get regulated output using LM 317.

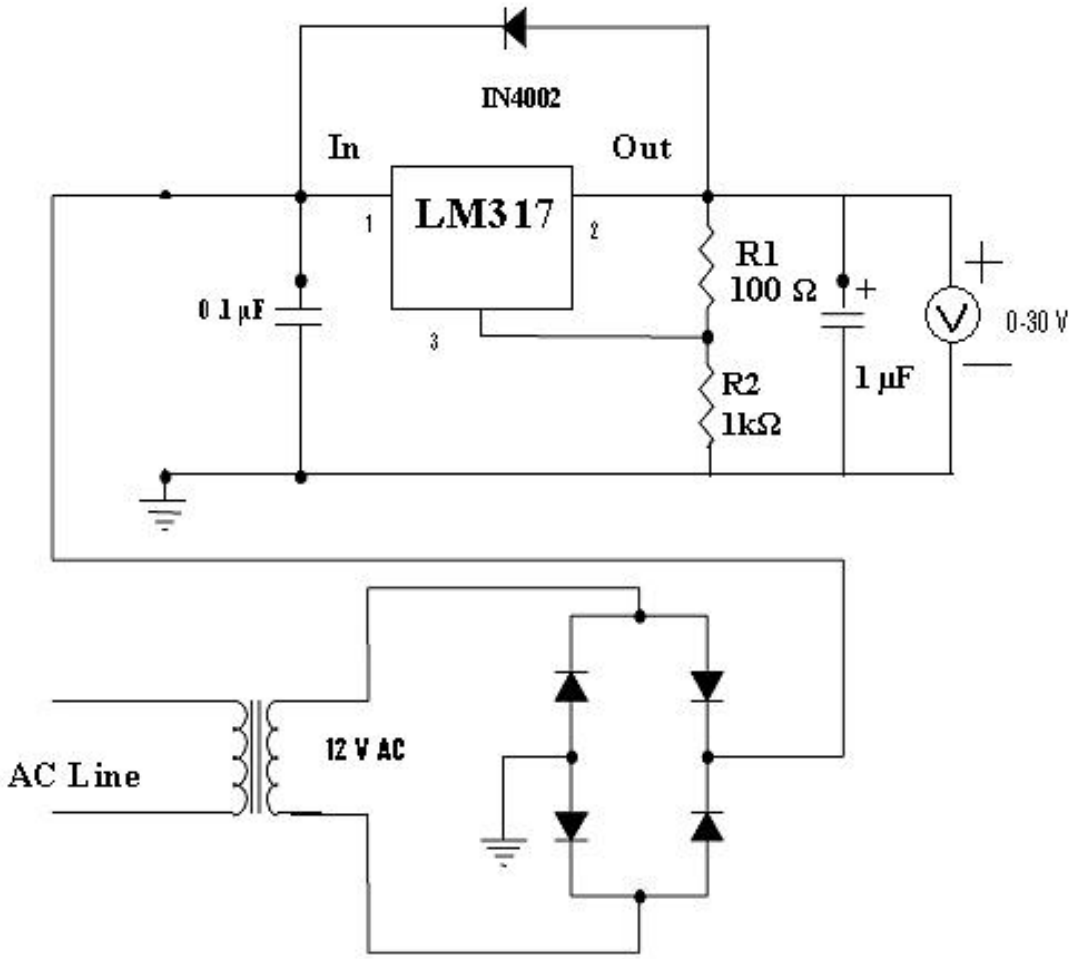
APPARATUS REQUIRED:

S.NO.	APPARATUS	RANGE	QUANTITY
1	Dual power supply	(0 - +15)V	1
2	Resistor	240Ω , 1.4KΩ,	Each 1
3	Capacitor	1µF	1
4	Bread Board	-	1

THEORY:

The basic voltage regulator in its simplest form consists of a) voltage reference Vr b) error amplifier c) feedback network d) active series or shunt control unit. the voltage reference generates a voltage level which is applied to the comparator circuit, which is generally error amplifier. The second input to the error amplifier obtained through feedback network. Generally using the potential divider, the feedback signal is derived by sampling the output voltage. The error amplifier converts the difference between the output sample and the reference voltage into an error signal. This error signal in turn controls the active element of the regulator circuit, in order to compensate the changes in the output voltage. Such an active element is generally a transistor.

CIRCUIT DIAGRAM-(LM317):



DESIGN PROCEDURE:

- $V_o = 1.25(1+R_2/R_1)$

Besides fixed voltage regulator, Ic voltage regulators are available which allow the adjustment of the output voltage. The output voltage can be adjusted from 1.2V to as high as 5.7V with the help of such regulators. in such regulator IC's common terminal plays the role of control input and hence called as adjustment terminal. The LM 317 series is the most commonly used three terminal adjustable regulators. These devices are available in a variety of packages which can be easily mounted and handled. The power rating of such regulators is 1.5Am.the maximum input voltage of LM 317 is 40V

EXPERIMENTAL PROCEDURE:

4 Connections are given as per the experimental setup.

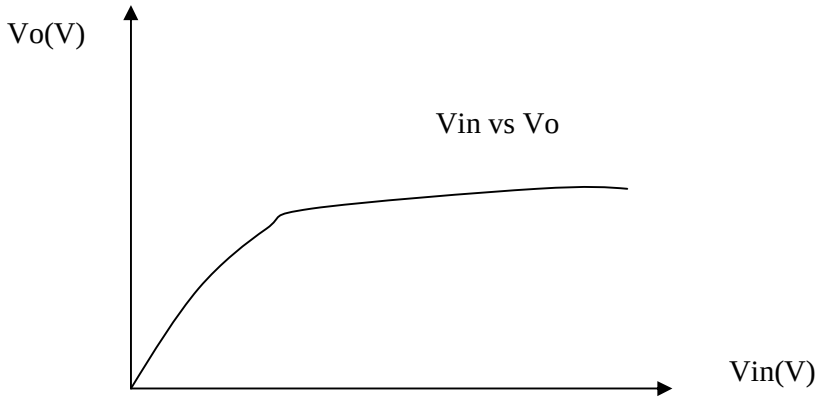
5 The input voltage is given to the circuit and the output voltage slowly varies from zero.

6 Then the output voltage attains the designed value and then it is irrespective of input voltage (the output becomes constant).

TABULATION-(DC POWER SUPPLY):

S.NO.	Vin $volts$	Vo $volts$

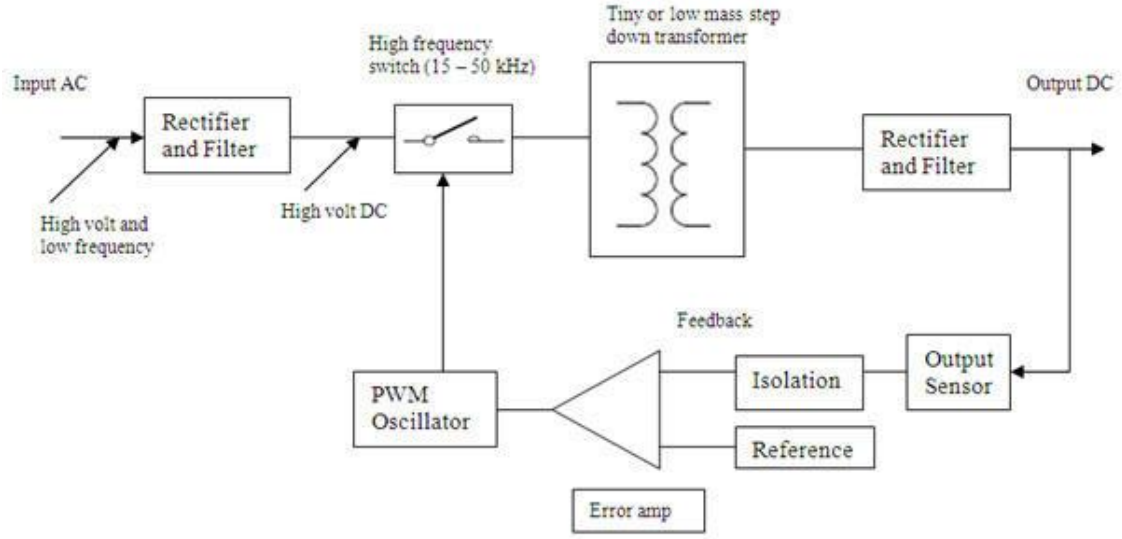
MODEL GRAPH:



RESULT :

Thus the experiment is conducted using LM 317 and so the regulated output is obtained using the circuit

BLOCK DIAGRAM-(SMPS):



BLOCK DIAGRAM ELEMENTS ARE:

1.Rectifier
2.Transformer
3.Filter
4.Pwm Oscillator
5.Amplifier
6.Isolation

Ex. No :13	**STUDY OF SMPS (SG 3524 AND SG 3525)**
DATE :	

AIM:

To study in detail about SMPS control unit of SG 3524 and SG 3525.

APPARATUS REQUIRED:

S.NO.	APPARATUS	QUANTITY
1	SMPS board	1
2	SG 3524 / SG 3525	1
3	RPS (0-30)V	1

DESCRIPTION:

The monolithic Ic contains all the control circuitry for a regulating power supply inverter or switching regulator. It also includes the error amplifier, oscillator, pulse width modulator, pulse steering flip flop, dual alternating output switches and current limiting and shut down circuitry.

The device can be used for switching regulators of either polarity, transformer coupled AC to DC converters, transformations, voltage doubling and polarity converters as well as other power control applications of 0^0C to 70^0C.

FEATURES:

- Complete PWM power control circuitry.
- Single ended or push pull outputs
- Line and load regulation of 0.2%
- 1 % maximum temperature variations.
- Total supply current is less than 10mA.
- Operation beyond 100 KHz.

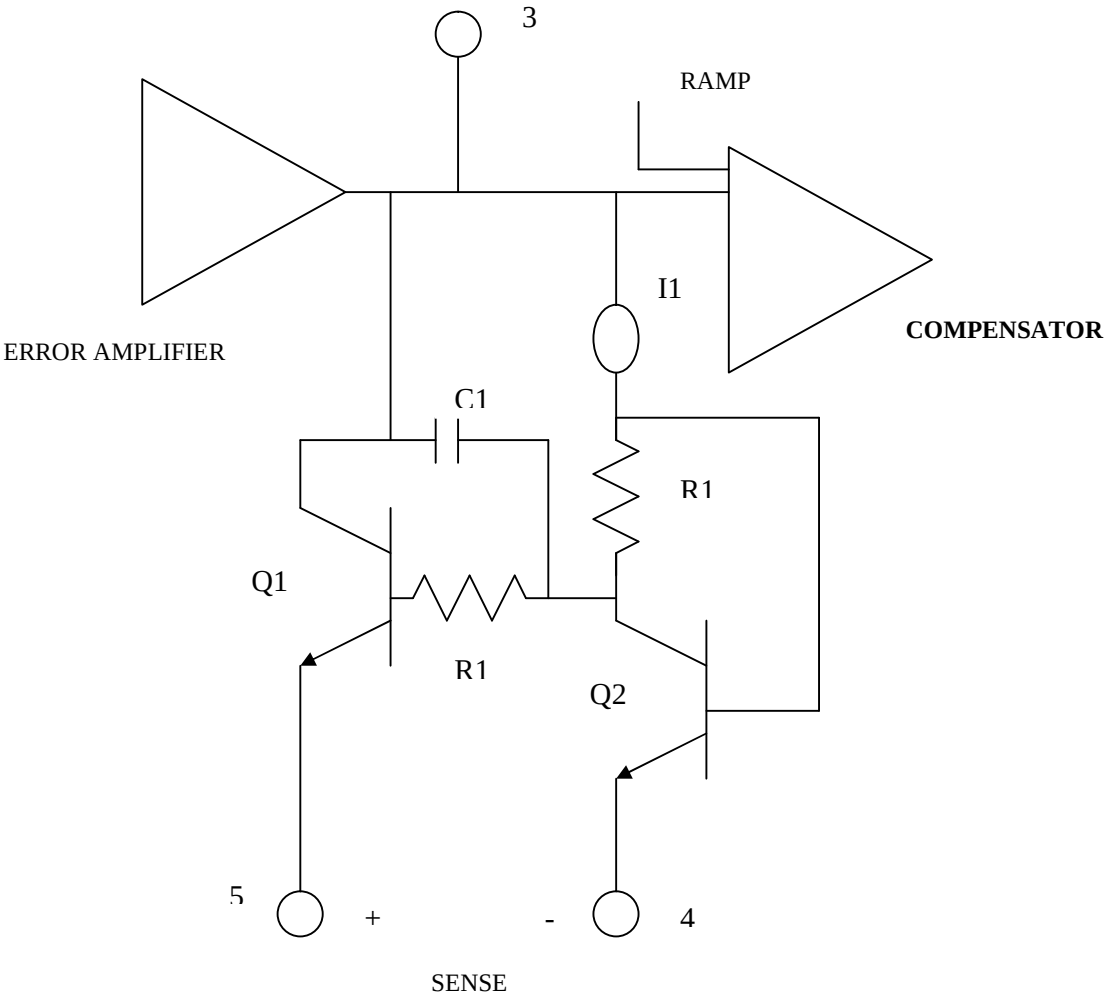
CURRENT LIMITING:

The current limiting circuitry of the SG3524 is shown in the figure. By matching the base-emitter voltages of Q1 and Q2 and also assuming a negligible voltage drop across the R1,

$$\text{Threshold} = V_{BE}(Q1) + I_1R_2 - V_{BE}(Q2)$$
$$= I_1R_2 - 200mV$$

Although this circuit provides a relatively small threshold with a negligible temperature coefficient, there are some limitations to its use. The most important of which is ±1V common mode range which require sensing in the ground state. Another factor to consider is that the frequency compensation provided by R_1C_1 and Q1 provides a roll off pole at approximately 300Hz.

CURRENT LIMITING CIRCUITRY OF SG 3524



Since the gain of this circuit is relatively low, there is a transition region as the current limit amplifier takes over pulse width control from the error amplifier. For testing purposes threshold is defined as the input voltage required getting 25% duty cycle with the error amplifier signaling maximum duty cycle

THEORY OF OPERATION:

VOLTAGE REFERENCE:

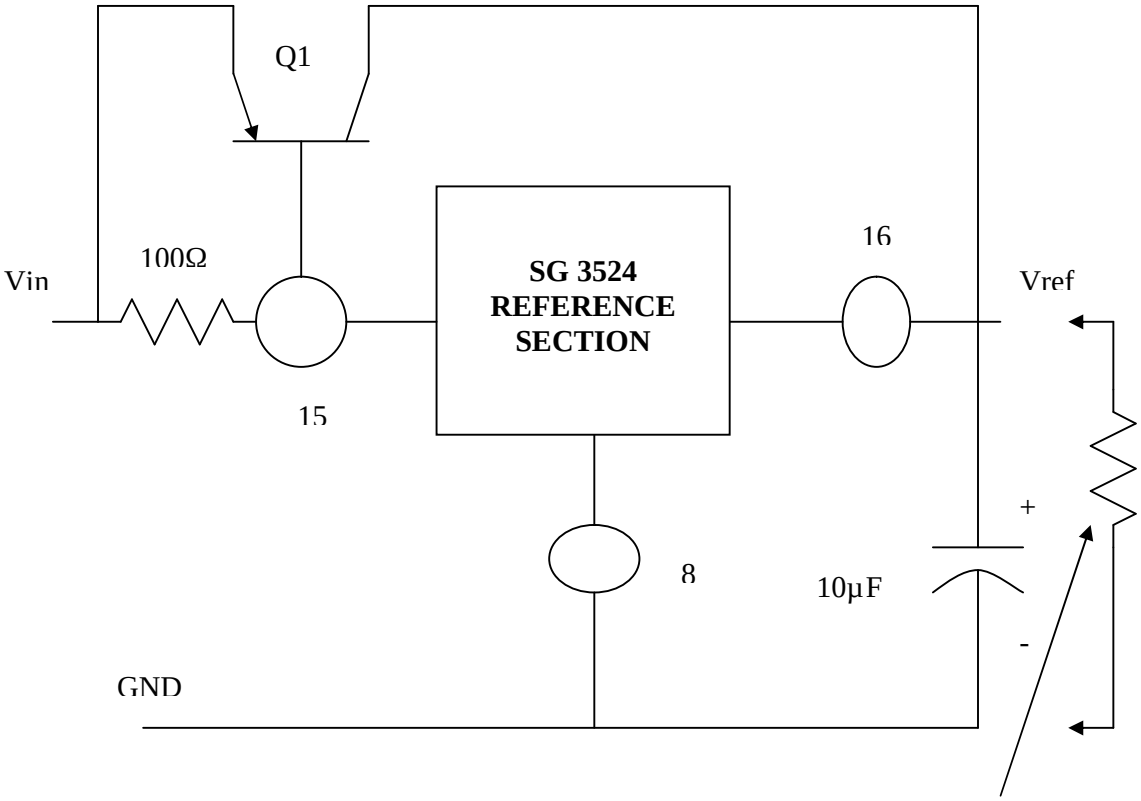
All the internal series regulator provides a nominal 5V output which is used both to generate source for all the internal timing and controlling circuitry. this regulator may be bypassed for operation from a fixed 5V supply by connecting pins 15 and 16 together to the input voltage of 5V

This reference regulator may be used as a 5V source for other circuitry. It will provide up to 50mA of current itself and can easily be expanded to higher currents with an external PNP.

EXTERNAL SYNCHRONIZATION:

If it is designed to synchronies the SG 3524 to an external clock, a pulse of approximately ±13V may be applied to the oscillator output terminal with $R_TC_T$at slightly greater than the clock pulse. the same consideration of pulse width is applied. The impedances to ground at this point is approximately 2KΩ.

EXPANDED REFERENCE CURRENT CAPABILITY



I_L OF 1Amp DEPENDING
ON CHOICE FOR Q1

If the two or more SG 3524 must be synchronized together, one must be designed at master with it.R_TC_T set for the correct period. the slaves should each have all R_TC_T set for approximately 10%. Layer period than the master with the added requirement that C_T(slave) = one half C_T(master).then connecting pins on he all units together will ensure that the master output pulse, which occurs first and has a wide pulse width will reset the slave units.

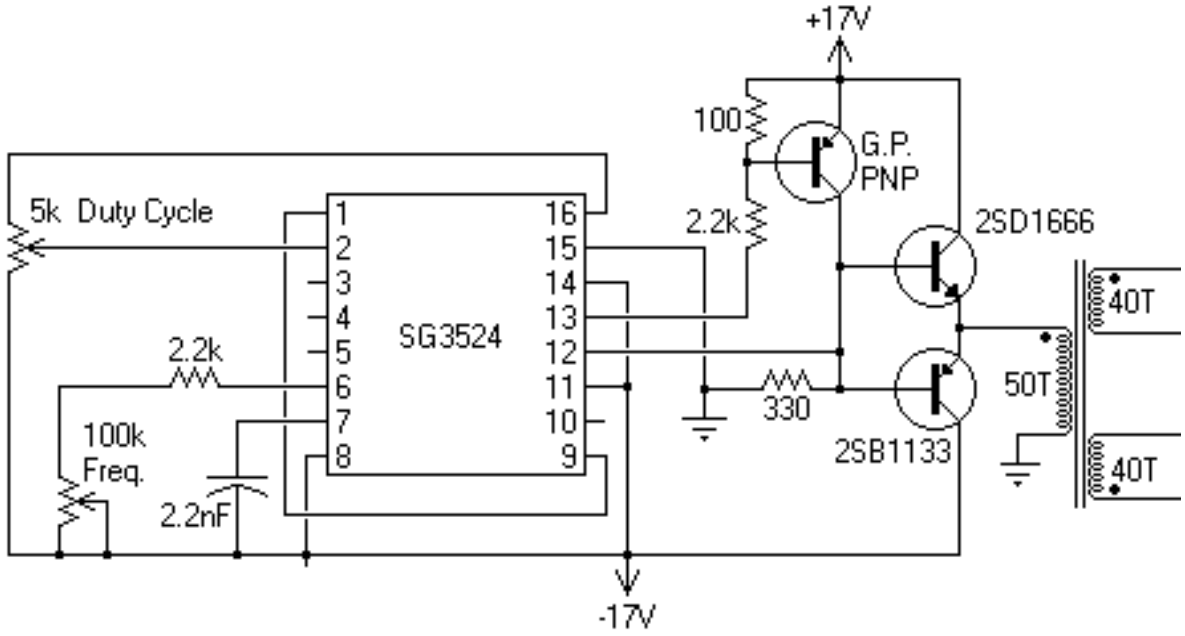
ERROR AMPLIFIER:

This circuitry is a simple differential input transconductance amplifier. the output is the compensation terminal pin 9 which is a higher impedance mode (R_LC_TMΩ).the gain is Av-g_mR_L-8IcR_L/2KT = 0.002 R_L and can be easily be reduced from a nominal of 10,000 by an external shunt impedance shunt resistance from pin 9 to ground.

In addition to DC gain control, the compensation terminal is also the place for AC phase compensation.

Typically most output filter designs will introduce one or more additional poles at a significantly lower frequency .therefore, the best stabilizing network is a series RC combination between pin 9 and ground which introduces a zero to cause one of the output filter poles. A good starting point is 50KΩ plus 0.001µF.

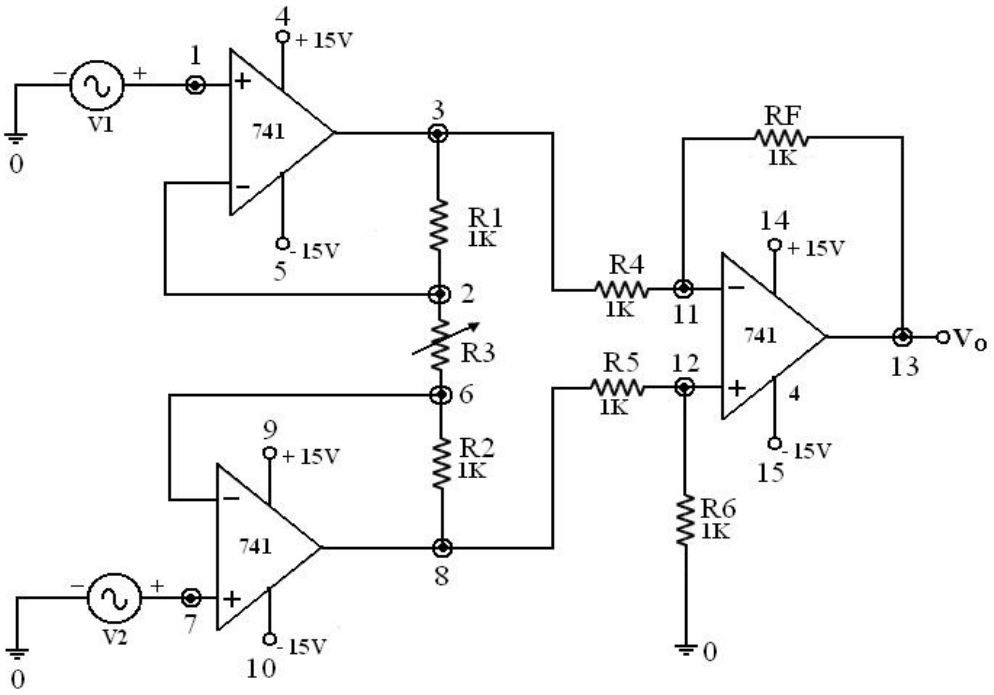
TEST CIRCUIT-(SMPS);



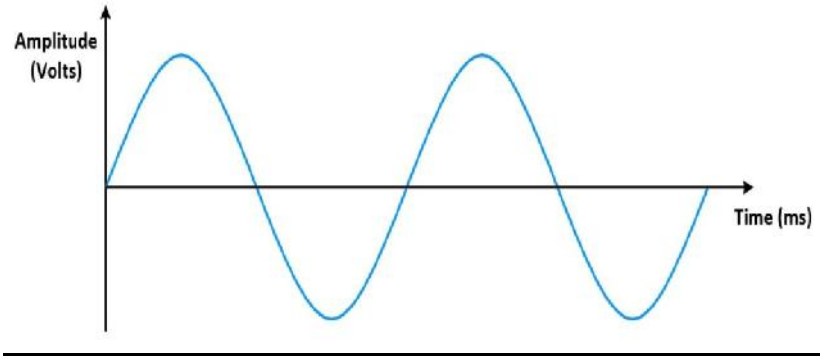
RESULT :

Thus the SMPS control unit of SG 3524 was studied.

CIRCUIT DIAGRAM –(INSTRUMENTATION AMPLIFIER):



MODEL GRAPH:



Ex. No :14	**SIMULATION OF INSTRUMENTATION AMPLIFIER USING PSPICE**
DATE :	

AIM:

To simulate and analyze the instrumentation amplifier using PSPICE.

SOFTWARE REQUIRED:

Or CAD SOFTWARE

PROCEDURE:

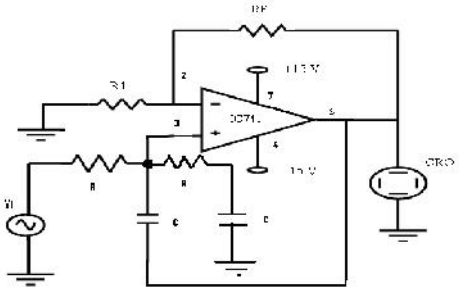
- Switch on the computer and select ORCAD PSPICE icon.
- Open a new project to design a circuit in the file menu.
- Select the required components from the library.
- Draw the circuit as shown in Fig .
- After completing save the project and go to simulation tool bar
- Verify the simulated output and take a print out.

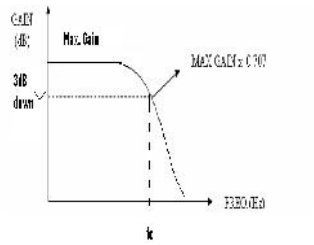
RESULT:

Thus the instrumentation amplifier using PSPICE was simulated and tested.

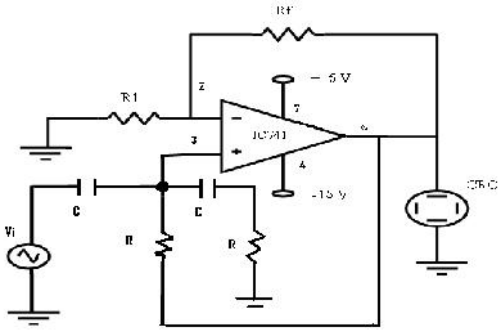
CIRCUIT DIAGRAM-(FILTERS):

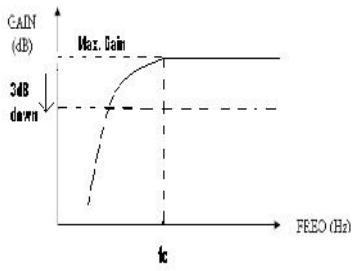
LOW PASS FILTER:



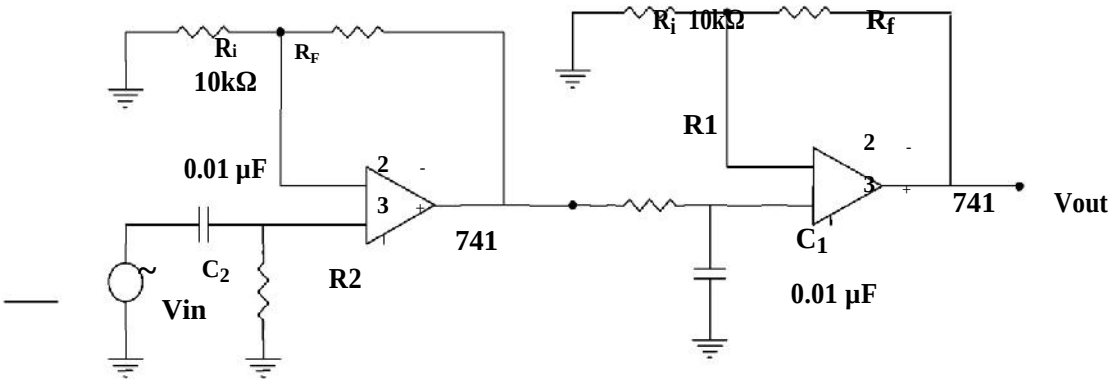


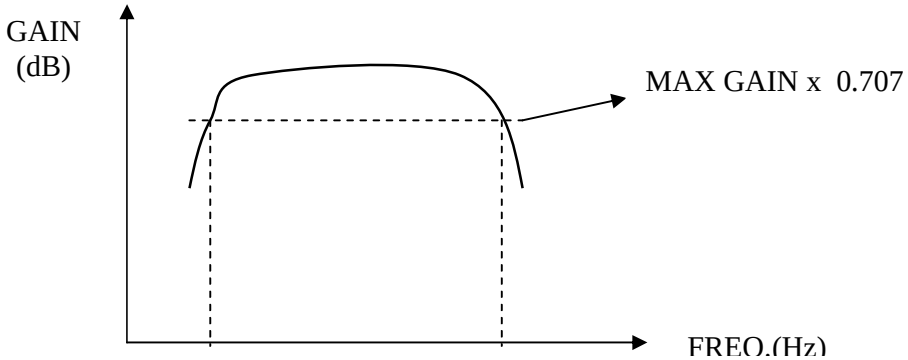
HIGH PASS FILTER:





BAND PASS FILTER





Ex. No :15	**SIMULATION OF ACTIVE LOWPASS, HIGHPASS AND BANDPASS FILTERS USING PSPICE**
DATE :	

AIM:

To simulate and analyze the Active Low pass, High pass and Band pass Filters using PSPICE.

SOFTWARE REQUIRED:

Or CAD software.

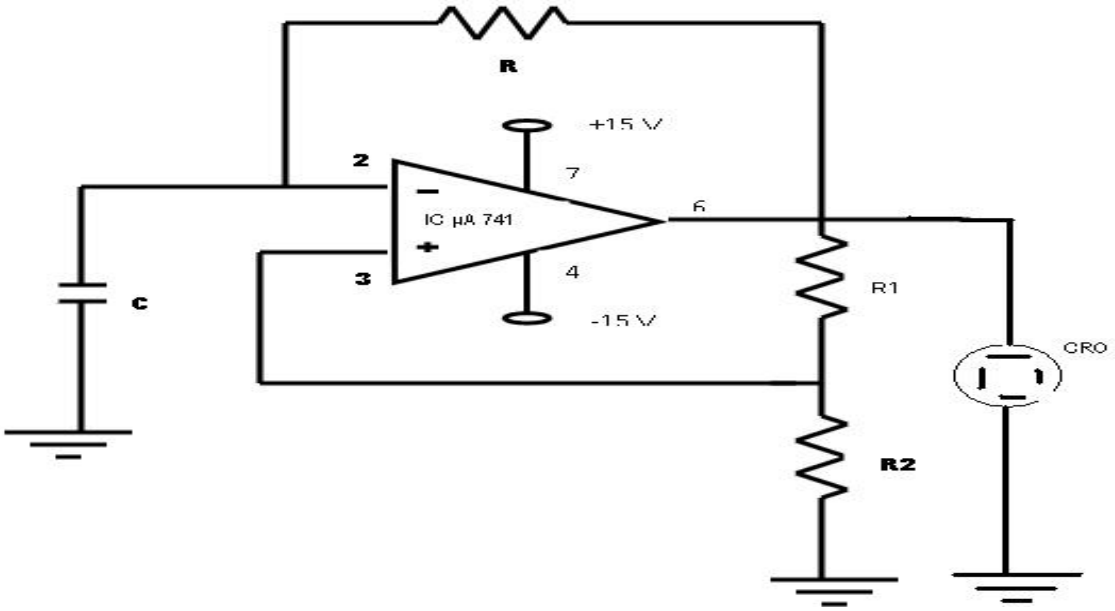
PROCEDURE:

- Switch on the computer and select ORCAD PSPICE icon.
- Open a new project to design a circuit in the file menu.
- Select the required components from the library.
- Draw the circuit as shown in Fig (1) &(2) tool bar.
- After completing save the project and go to simulation tool bar
- Verify the simulated output and take a print out.

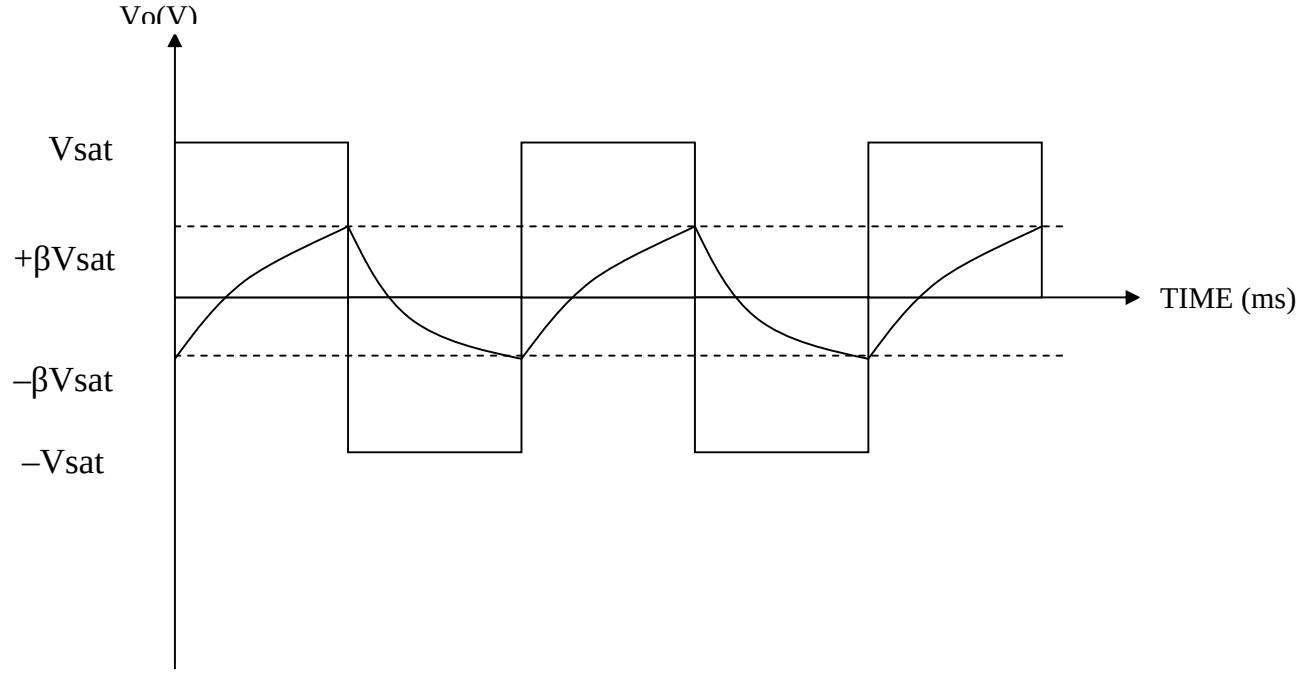
RESULT:

Thus the Active Low pass, High pass and Band pass Filters using PSPICE was simulated and tested.

CIRCUIT DIAGRM-(ASTABLE MULTIVIBRATOR):



MODEL GRAPH:



Ex. No :16	**SIMULATION OF ASTABLE & MONOSTABLE MULTIVIBRATORS AND SCHMITT TRIGGER USING PSPICE**
DATE :	

AIM:

To simulate and analyze the Astable &Monostable Multivibrators and Schmitt Trigger using PSPICE.

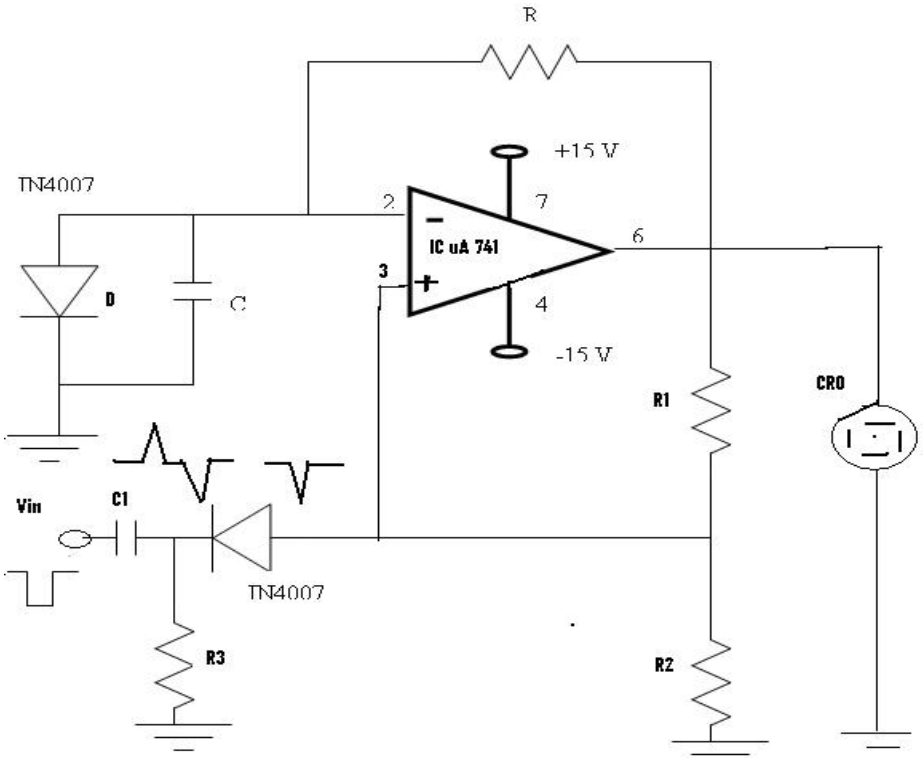
SOFTWARE REQUIRED:

Or CAD software.

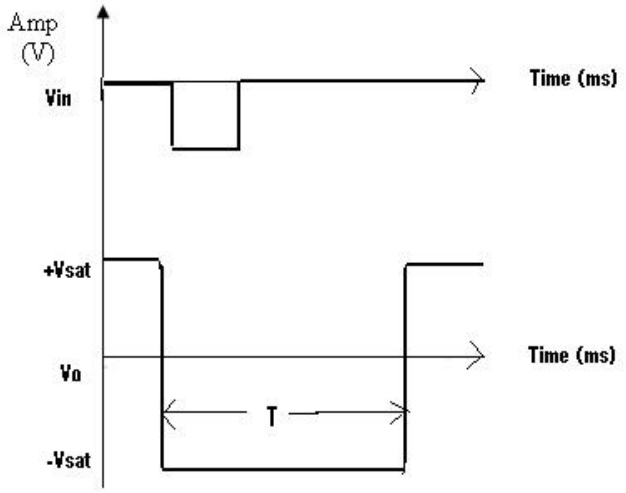
PROCEDURE:

- Switch on the computer and select ORCAD PSPICE ion.
- Open a new project to design a circuit in the file menu.
- Select the required components from the library.
- Draw the circuit as shown in Fig (1) &(2) tool bar.
- After completing save the project and go to simulation tool bar
- Verify the simulated output and take a print out.

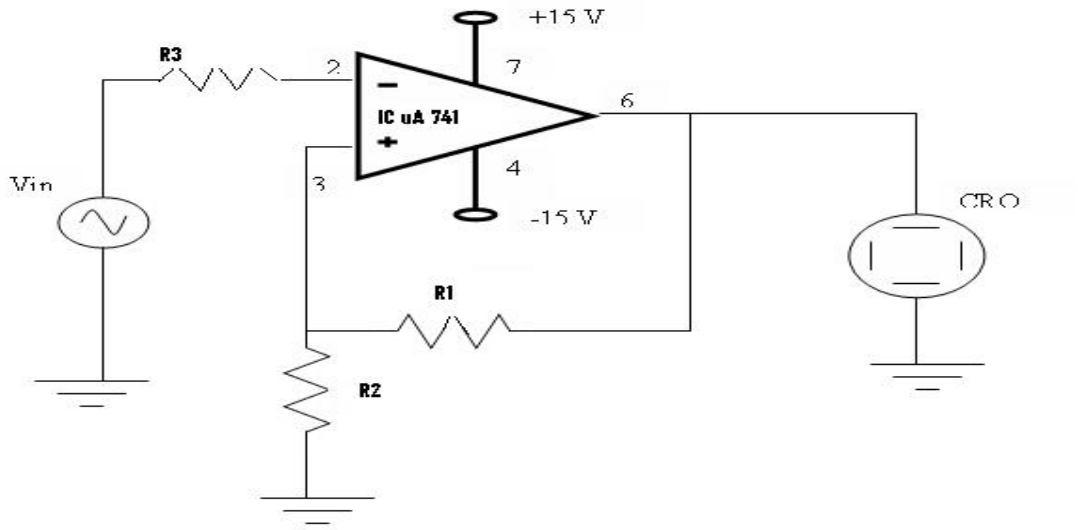
CIRCUIT DIAGRAM-(MONOSTABLE MULTIVIBRATOR):



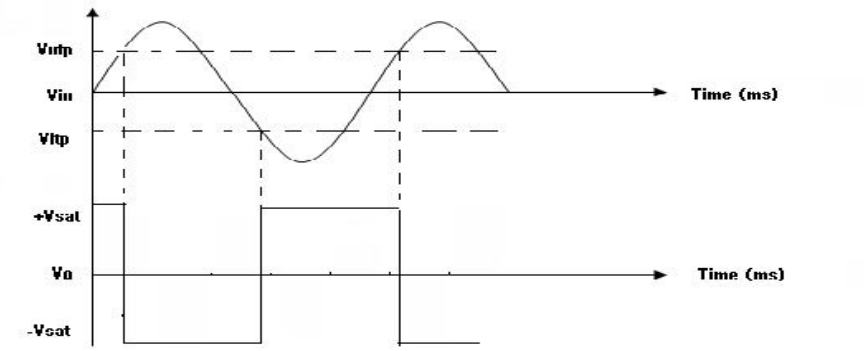
MODEL GRAPH:



CIRCUIT DIAGRAM-(SCHMITT TRIGER):



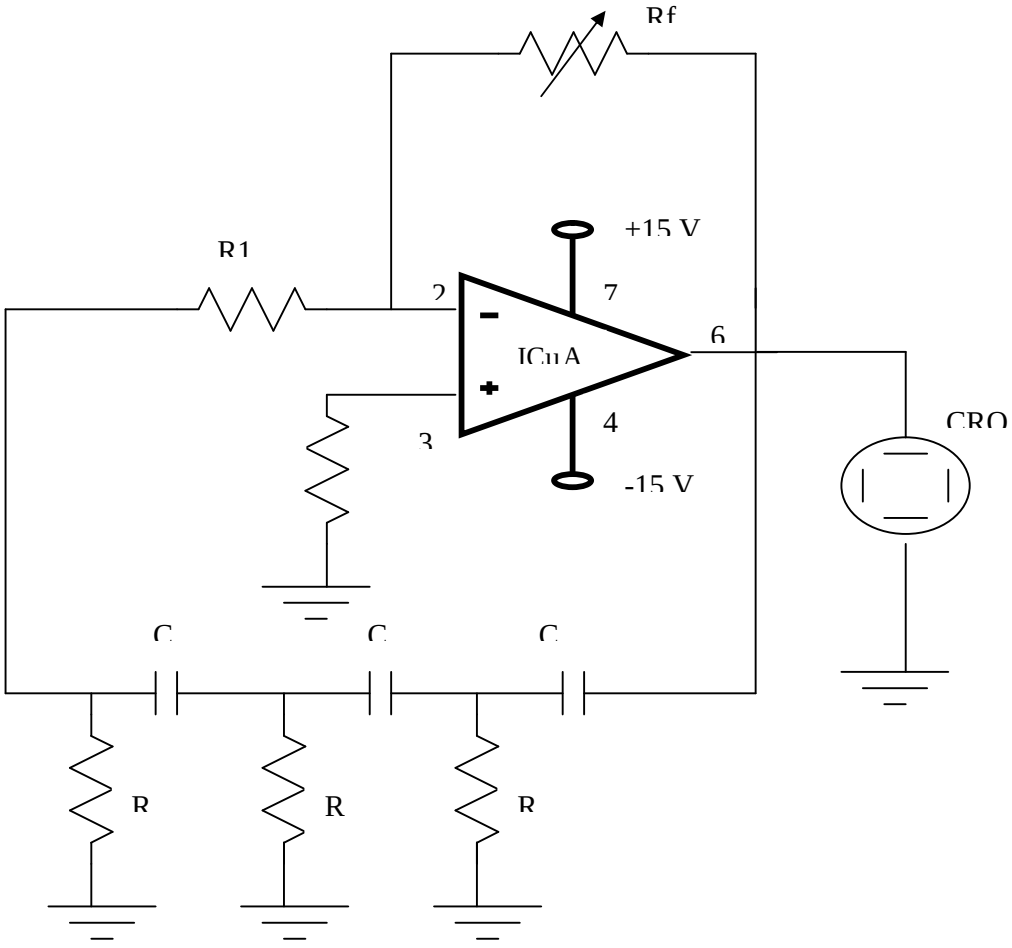
MODEL GRAPH:



RESULT:

Thus the Astable & MonostableMultivibrators and Schmitt Trigger using PSPICE was simulated and tested

CIRCUIT DIAGRAM-(RC PHASE SHIFT OSCILLATOR):



Ex. No :17	**SIMULATION OF PHASE SHIFT AND WEIN BRIDGE OSCILLATORS USING PSPICE**
DATE :	

AIM:

To simulate and analyze the Phase Shift and Wein Bridge Oscillators using op-amp using PSPICE.

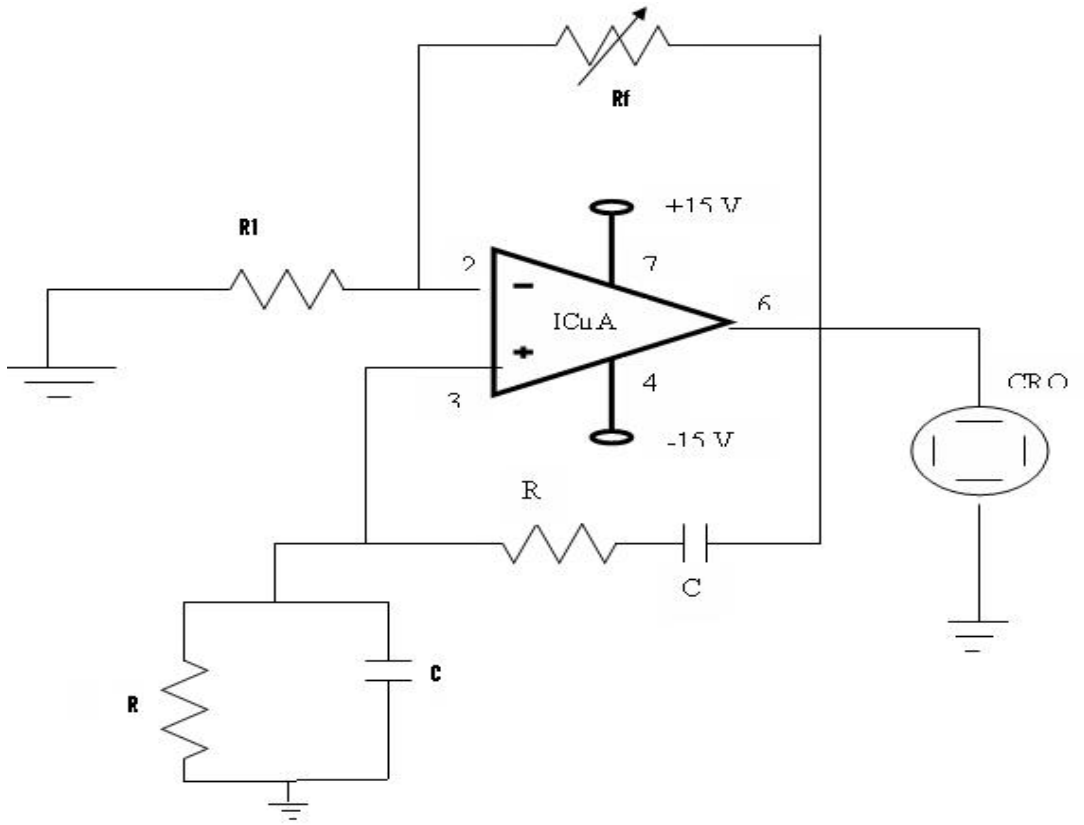
SOFTWARE REQUIRED:

Or CAD software.

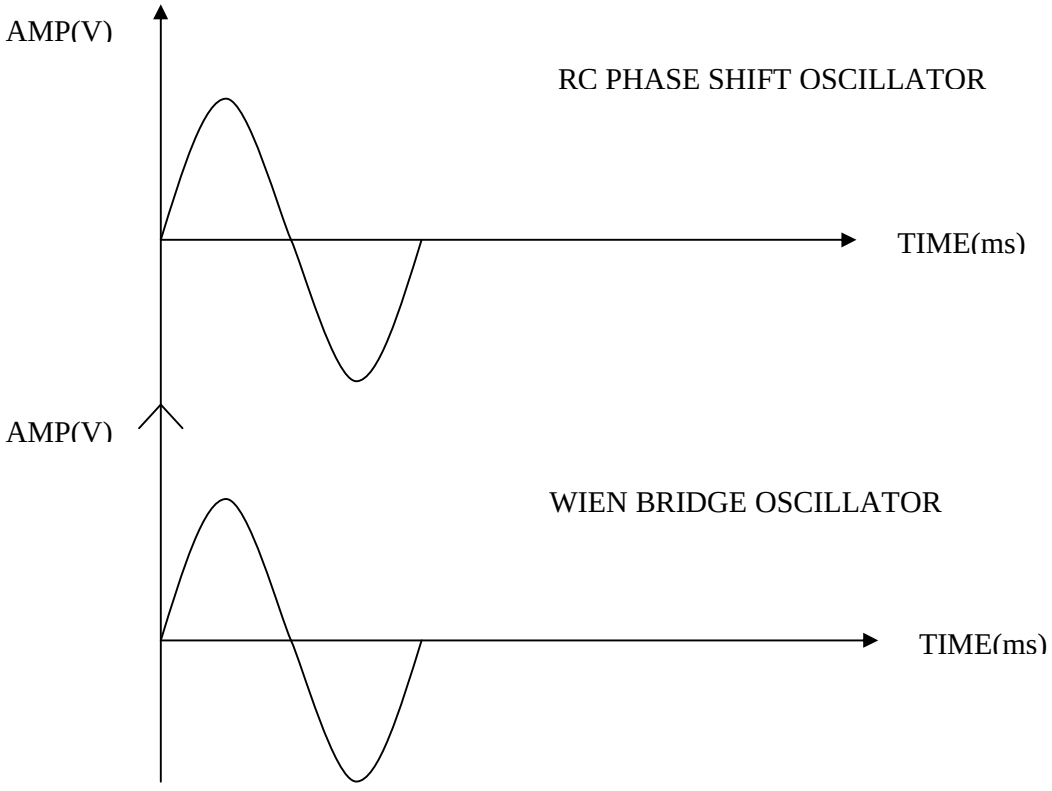
PROCEDURE:

- Switch on the computer and select ORCAD PSPICE ion.
- Open a new project to design a circuit in the file menu.
- Select the required components from the library.
- Draw the circuit as shown in Fig (1) &(2) tool bar.
- After completing save the project and go to simulation tool bar
- Verify the simulated output and take a print out.

CIRCUIT DIAGRAM-(WEIN BRIDGE OSCILLATOR):



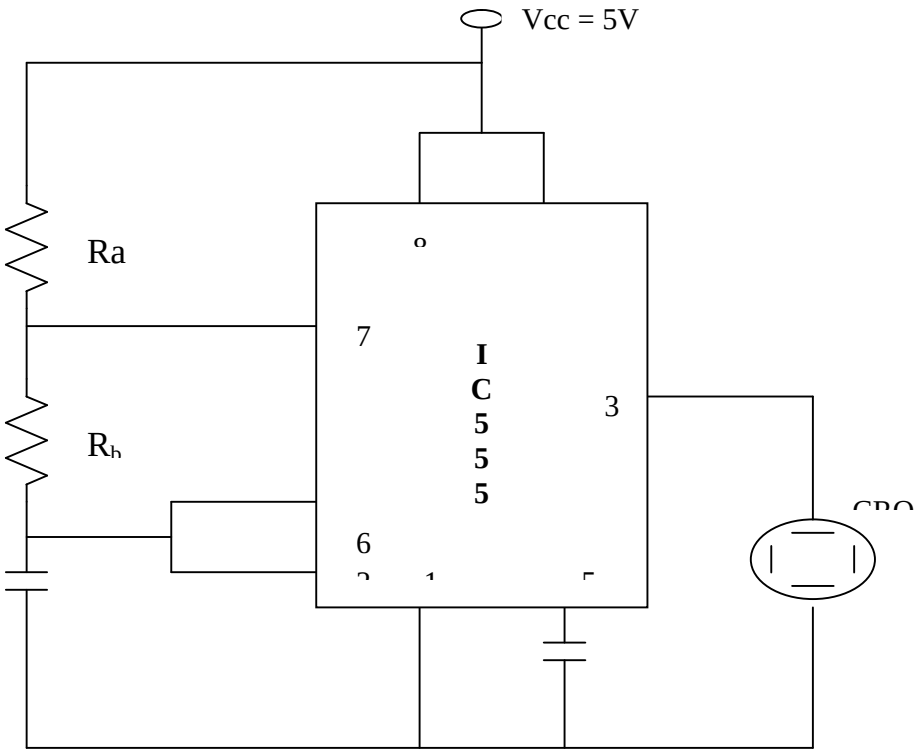
MODEL GRAPH:



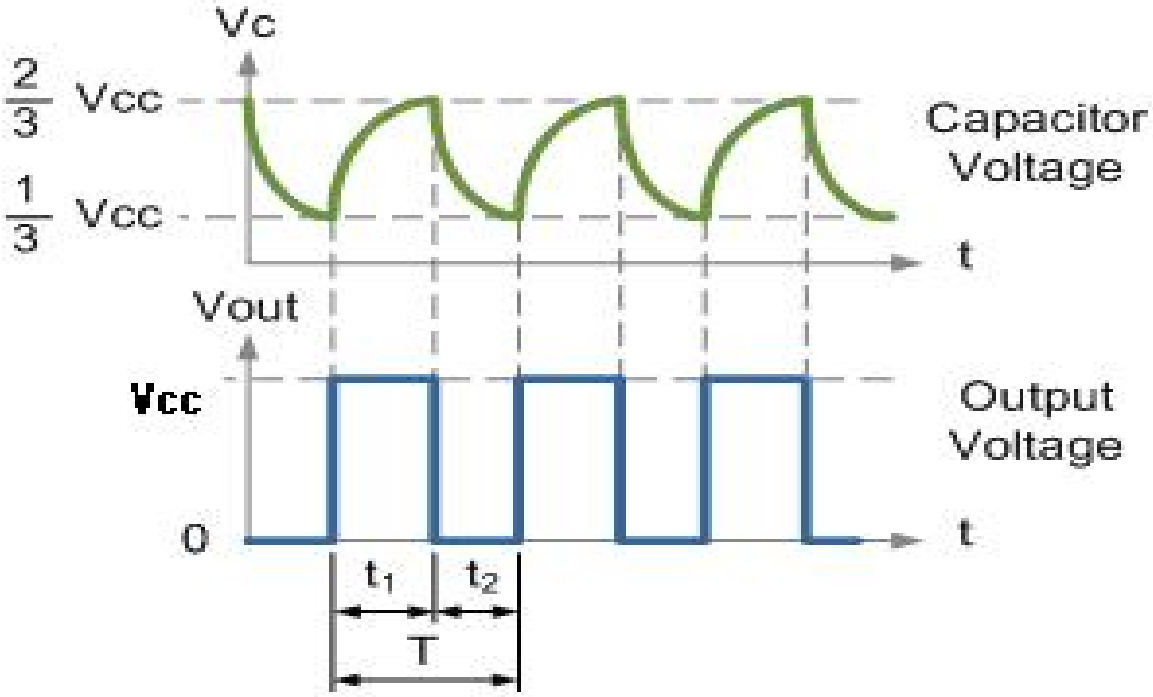
RESULT:

Thus the Phase Shift and Wein Bridge Oscillators using op-amp using PSPICE was simulated and tested.

CIRCUIT DIAGRAM-(ASTABLE MULTIVIBRATOR):



MODEL GRAPH:



Ex. No :18	**SIMULATION OF ASTABLE AND MONOSTABLE MULTIVIBRATORS (USING NE 555 TIMER) USING PSPICE**
DATE :	

AIM:

To simulate and analyze the Astable and monostable multivibrators using NE555 Timer using PSPICE.

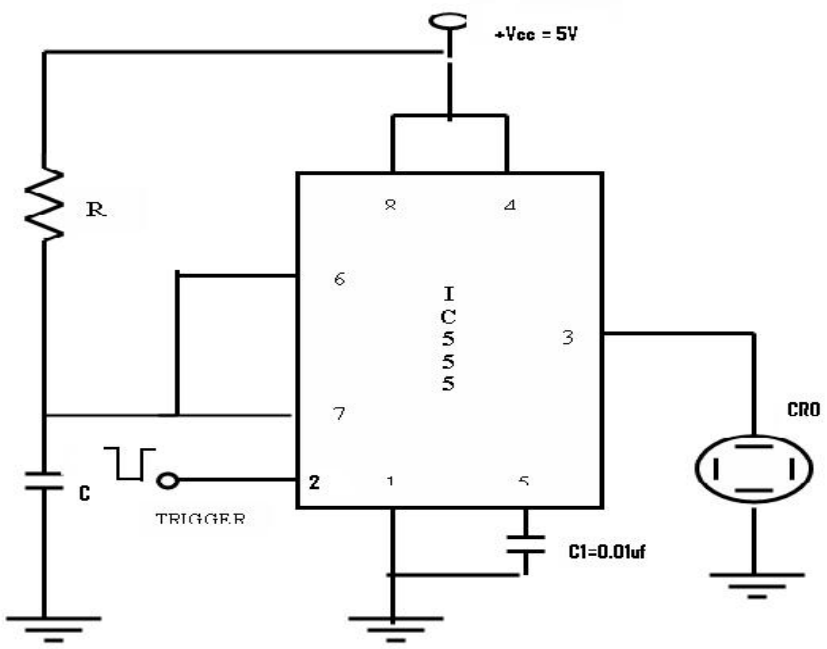
SOFTWARE REQUIRED:

Or CAD software.

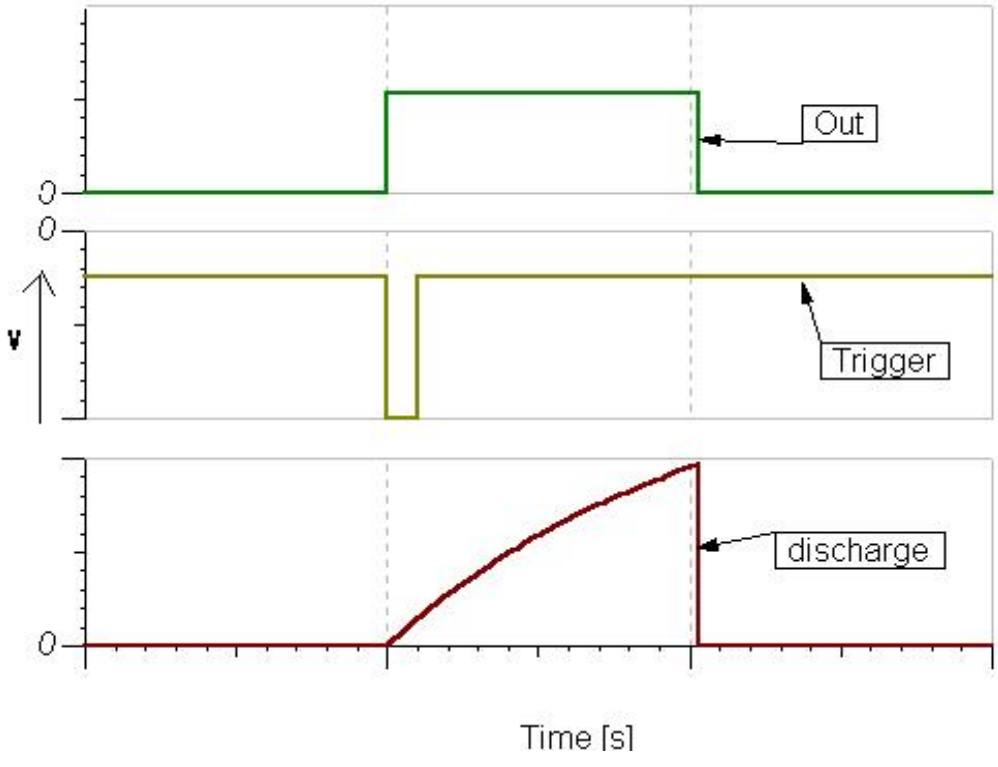
PROCEDURE:

- Switch on the computer and select ORCAD PSPICE ion.
- Open a new project to design a circuit in the file menu.
- Select the required components from the library.
- Draw the circuit as shown in Fig (1) &(2) tool bar.
- After completing save the project and go to simulation tool bar
- Verify the simulated output and take a print out.

CIRCUIT DIAGRAM-(MONOSTABLE MULTIVIBRATOR):



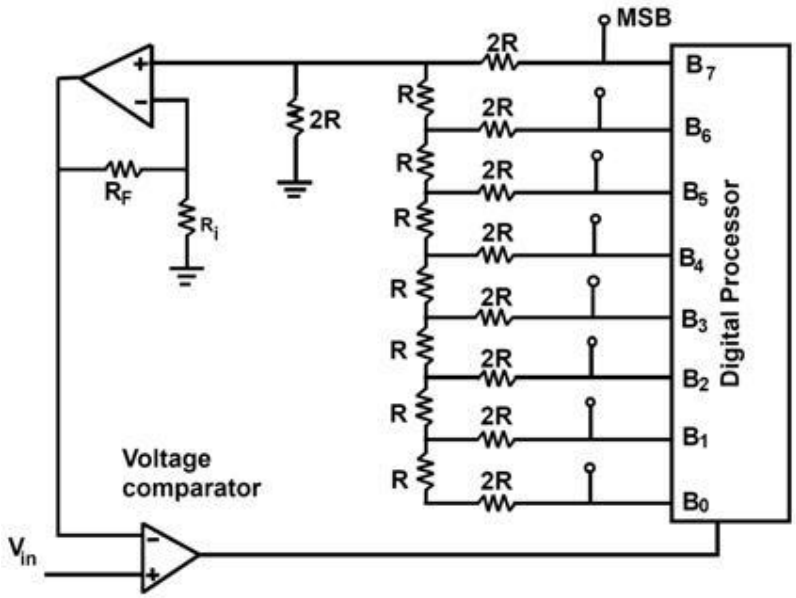
SIMULATION OUTPUT:



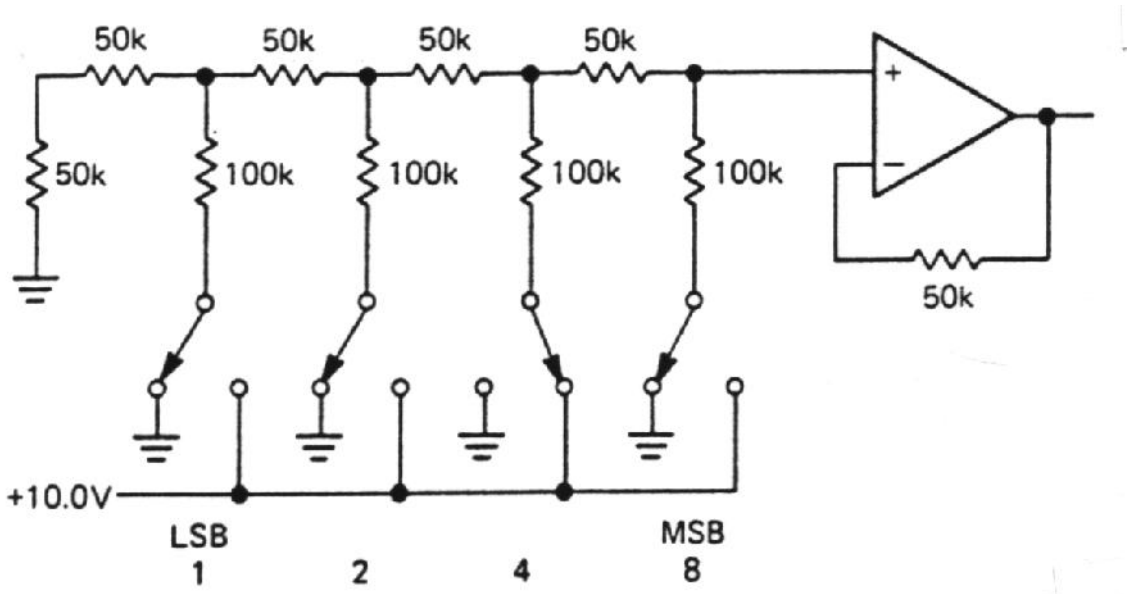
RESULT:

Thus the Astable and Monostable Multivibrators using NE555 Timer using PSPICE was simulated and tested.

CIRCUIT DIAGRAM-(ANALOG TO DIGITAL CONVERTER):



CIRCUIT DIAGRAM-(DIGITAL TO ANALOG CONVERTER):



Ex. No :19	**SIMULATION OF ADC, DAC AND ANOLOG MULTIPLIER USING PSPICE**
DATE :	

AIM:

To simulate and analyse the ADC, DAC& Anolog Multiplier using PSPICE.

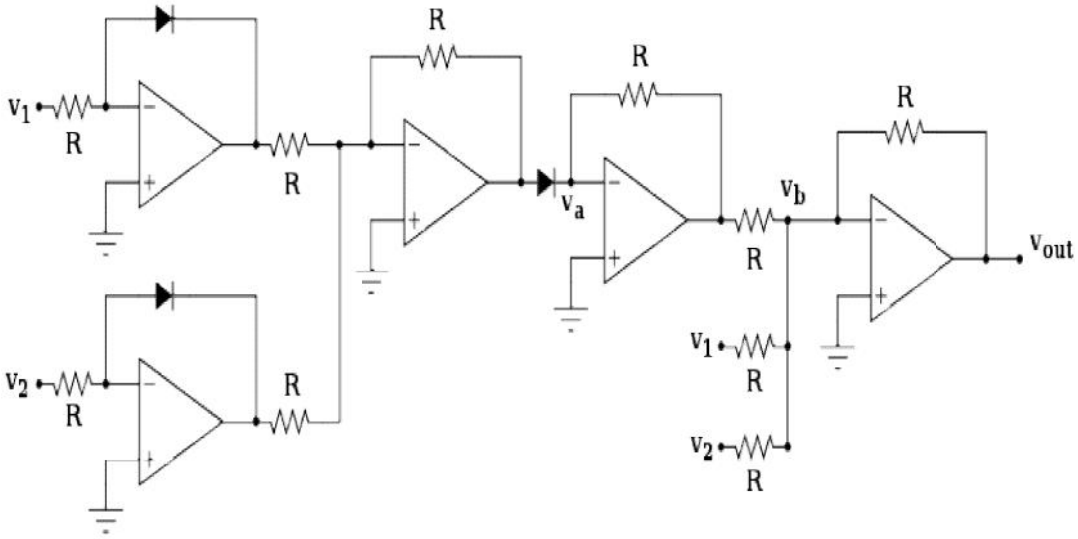
SOFTWARE REQUIRED:

Or CAD software.

PROCEDURE:

- Switch on the computer and select ORCAD PSPICE ion.
- Open a new project to design a circuit in the file menu.
- Select the required components from the library.
- Draw the circuit as shown in Fig (1) &(2) tool bar.
- After completing save the project and go to simulation tool bar
- Verify the simulated output and take a print out.

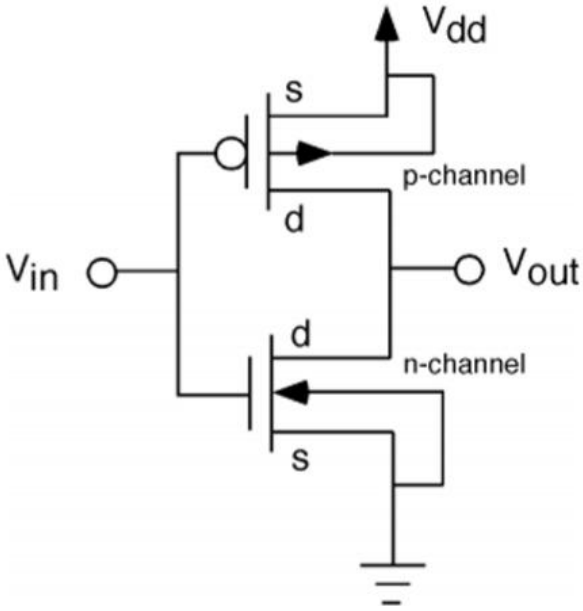
ANALOG MULTIPLIER:



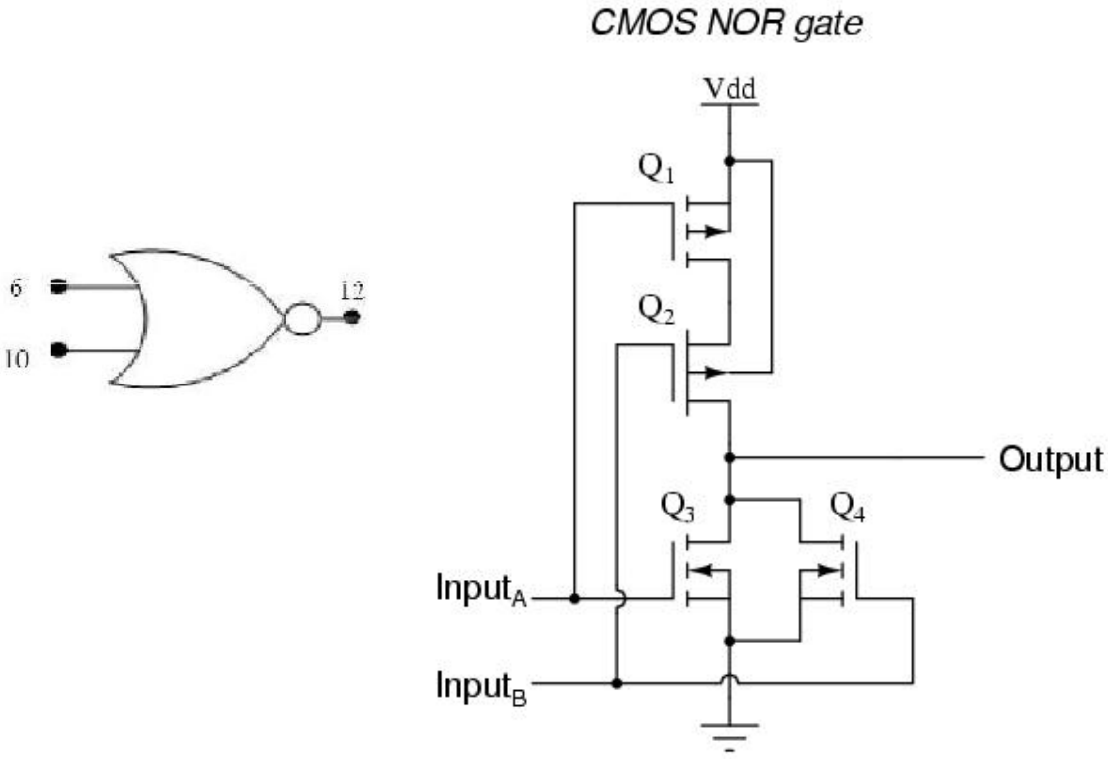
RESULT:

Thus the ADC, DAC, & Anolog Multiplier using PSPICE was simulated and tested.

CIRCUIT DIAGRAM –(CMOS INVERTER):



CIRCUIT DIAGRAM-(CMOS NOR)



Ex. No :20	**SIMULATION OF CMOS INVERTER,NAND, AND NOR USING PSPICE**
DATE :	

AIM:

To simulate and analyze the CMOS inverter,NAND,NOR using PSPICE.

SOFTWARE REQUIRED:

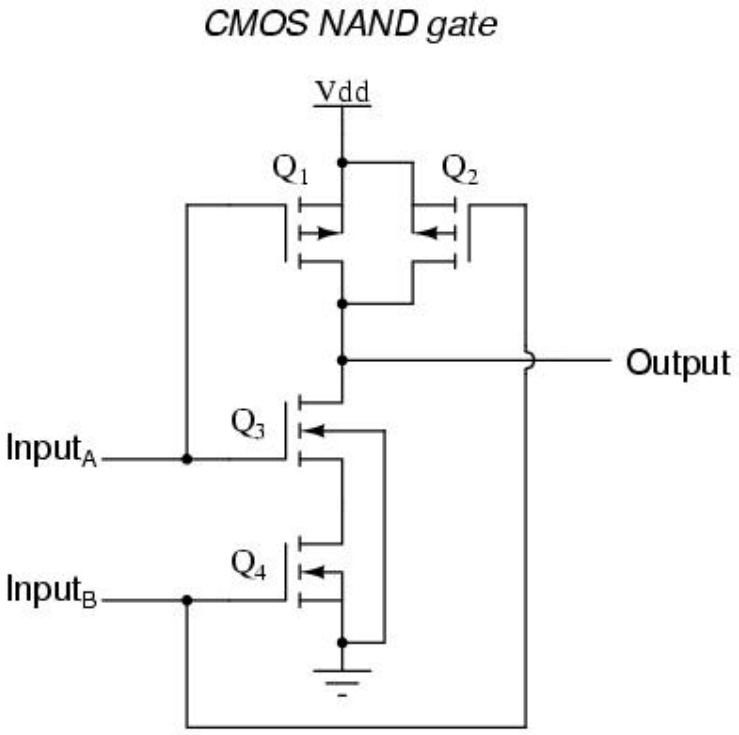
Or CAD software.

PROCEDURE:

- Switch on the computer and select ORCAD PSPICE ion.
- Open a new project to design a circuit in the file menu.
- Select the required components from the library.
- Draw the circuit as shown in Fig (1) &(2) tool bar.
- After completing save the project and go to simulation tool bar

Verify the simulated output and take a print out

CIRCUIT DIAGRAM-(CMOS NAND)



RESULT:

Thus the CMOS inverter, NAND, NOR using PSPICE was simulated and tested.