Department of Electronics and CommunicationEngineering

ANALOG CIRCUITS LABORATORY MANUAL

Subject Code: 18ECL48

Prepared by

Mrs. Aishwaryaa LK
Assistant Professor, Dept. Of ECE

Checked By
Dr. H B Bhuvaneswari
HOD, Dept. Of ECE

Affiliated to Visvesvaraya Technological University,Belagavi, Karnataka - 590018

2019-2023

<table>
<tr><td colspan="4">ANALOG CIRCUITS LABORATORYSEMESTER
– III (EC/TC)
[As per Choice Based Credit System (CBCS) scheme</td></tr>
<tr><td>Laboratory Code</td><td>21ECL35</td><td>CIE Marks</td><td>50</td></tr>
<tr><td>Number of Lecture Hours/Week</td><td>02 Hr. Tutorial(Instructions)
+ 02 Hours Laboratory</td><td>SEE Marks</td><td>50</td></tr>
<tr><td>RBT Level</td><td>L1, L2, L3</td><td>Exam Hours</td><td>03</td></tr>
<tr><td colspan="4">CREDITS – 01</td></tr>
<tr><td colspan="4">Course objectives: This laboratory course enables students to
Understand the electronic circuit schematic and working
Realize and test amplifiers and oscillator circuits for the given specifications
Realize and opamp circuits for the applications such as DAC , implement mathematical functions and precision rectifiers.
Study the static characteristics of SCR and test the RC triggering circuits
Design and test the combinational and sequential logic circuits for their functionalities.
Use the suitable ICs based on the specifications and functions.</td></tr>
<tr><td colspan="4"></td></tr>
<tr><td colspan="4">Experiments</td></tr>
<tr><td colspan="4">1. Design and set up the BJT common emitter voltage amplifier with and without feedback and determinethe gain- bandwidth product, input and output impedances.</td></tr>
<tr><td colspan="4">2. Design and set-up BJT/FET: i) Colpitts Oscillator, and ii) Crystal Oscillator iii) RC phase shift oscillator</td></tr>
<tr><td colspan="4">3. Design and setup the circuits using opamp: i) Adder , ii) Integrator, iii) Differentiator and iv) Comparator</td></tr>
<tr><td colspan="4">4. Obtain the static characteristics of SCR and test SCR controlled HWR and FWR using RC triggering circuits.</td></tr>
<tr><td colspan="4">5.Test the precision rectifiers using opamp: i)Half wave rectifier, ii) full wave rectifier</td></tr>
<tr><td colspan="4">6. Design and test Monostable and Astable Multivibrator using 555 Timer.</td></tr>
</table>

Course Outcomes: On the completion of this laboratory course, the students will be able to: • Design analog circuits using BJT/FETs and evaluate their performance characteristics. • Design analog circuits using OPAMPs for different applications • Simulate and analyze analog circuits that uses ICs for different electronic applications.
Conduct of Practical Examination: • All laboratory experiments are to be included for practical examination. • Students are allowed to pick one experiment from the lot. • Strictly follow the instructions as printed on the cover page of answer script for breakup of marks. • Change of experiment is allowed only once and Marks allotted to the procedure part to be made zero
Reference Books: 1. David A Bell, “Fundamentals of Electronic Devices and Circuits Lab Manual, 5th Edition, 2009, Oxford University Press.

INDEX

EXPERIMENT: 1

COMMON EMITTER AMPLIFIER

Aim: Design and set up the BJT common emitter amplifier using voltage divider bias with and without feedback and determine the gain- bandwidth product from its frequency response.

Components required: Transistor – SL100, Resistors - 470 Ω, 1KΩ, 10KΩ - 2nos, and 33KΩ, Capacitors 100µf, 0.22µf and 0.47µf, Power Supply, 10Hz – 3MHz Signal generator, CRO, Connecting wires and Bread board/Spring board with spring terminals.

Theory:
Common Emitter Amplifier Configuration, the Emitter of a BJT is common to both the input and output signal.

Design: Transistor: **SL100**

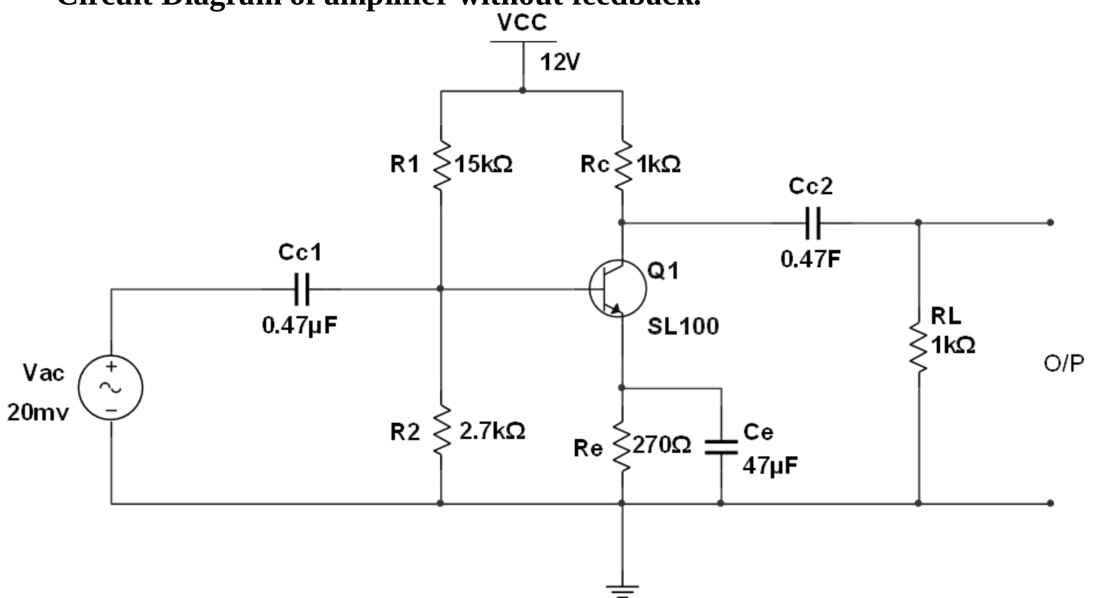
Let V_{CC} = 12V; I_C = 4.5 mA; V_E = 1.2V; V_{CE} = 6V; h_{FE} = 100.

Given V_E = 1.2V. Therefore $R_E = V_E / I_E$ Ω V_E / I_C =266.67Ω; R_E =270Ω
Writing KVL for the Collector loop we get, $V_{CC} = I_C R_C + V_{CE} + V_E$
$R_C = (V_{CC} – V_{CE} – V_E)$; I_C = (12-6-1.2)V/4mA=1.06K Ω; R_C = 1 K Ω
hFE RE = 10R2

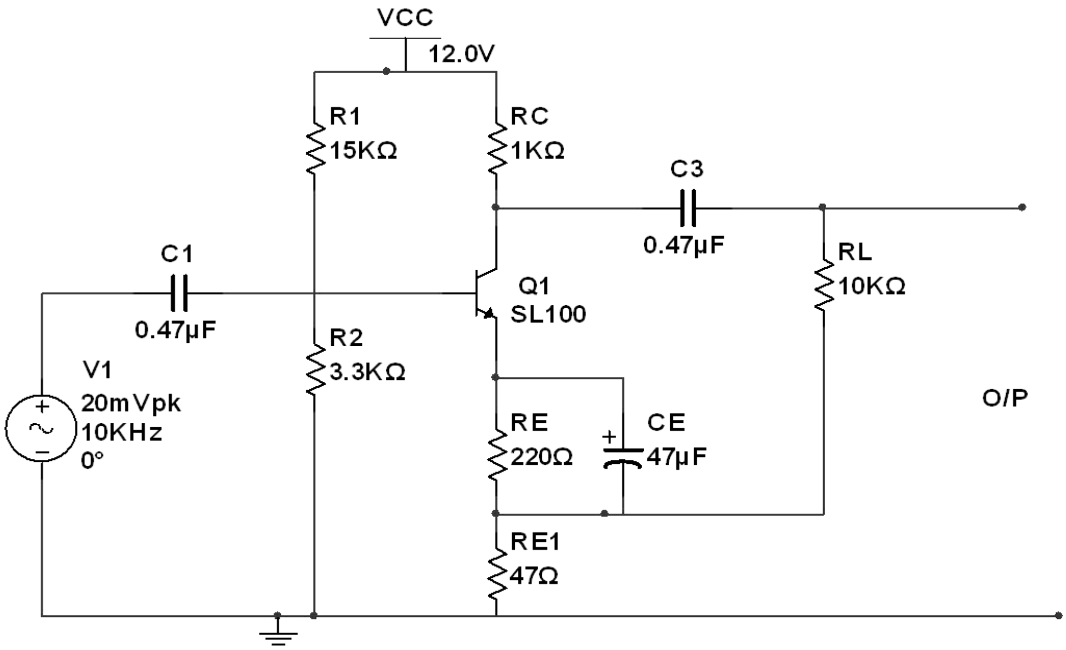
Assume R_2=2.7KΩ,
$V_B = (Vcc \times R_2) / (R_1 + R_2)$
Hence R_1 = 14.14 K Ω ; R_1 = 15 K Ω

Use C_{C1} = 0.47µF
Use C_{C2} = 0.47µF
Use C_E =47µF

Circuit Diagram of amplifier without feedback.



Circuit Diagram of amplifier with feedback. (Introduce a resistor in the emitter circuit)



Procedure: Follow the same procedure for both circuits

After making the connections, switch on the D.C. power supply and check the D.C. conditions without any input signal and record in table below

Parameter	V_{RC}	V_{CE}	V_E	I_{CQ}	V_{BE}
Assumed	4.8V	6 V	1.2V	4.5 mA	0.6 V
Practical					

1. Select sine wave input and set the input signal frequency ≥10f1 (Say = 10 KHz. This will be a convenient 'Mid – frequency').
2. Observe the input wave form and output wave form on a dual channel CRO.
3. Adjust the input amplitude such that the output waveform is just undistorted (or in the verge of becoming distorted). Measure the amplitude of the Input Signal now. This amplitude is the Maximum Signal Handling Capacity of your amplifier.
4. Decrease the input voltage to a convenient value such that the output is undistorted. Say 20mV. Measure the corresponding o/p voltage. Calculate mid-band gain, $A_M = V_O$ (p-p) / V_{in} (p-p).
5. Keeping the input voltage constant, go on reducing the frequency until the output voltage reduces to 0.707 times its value at 10 KHz. The frequency at which this happens gives you the Lower Cut-off frequency (*f1*).
6. Keeping the input voltage constant, go on increasing the frequency until the output voltage decreases to 0.707 times its value at 10 KHz. The frequency at which this happens gives you the Upper Cut-off frequency (*f2*).
7. Thus you have pre-determined *f1* and *f2*. Find the amplifier band width, BW = *f2* −*f1*
8. Determine Gain Bandwidth product (GBW product) which is a Figure of Merit of

your amplifier as GBW = AM x BW.

9. Now repeat the experiment by recording values of output voltage versus frequency keeping the input voltage at a constant value convenient to you. You should take at least 5 readings below *f1* and 5 readings above *f1*, at least 5 readings in the mid band, at least 5 readings below *f2* and 5 readings above *f2*.

10. Plot graphs of A_V versus Frequency, *f* and /or M, dB versus Frequency, *f* on a *semi log graph paper*. From the graph determine: Mid –band - gain, Lower and Upper Cut-off frequencies and Band width. Compute the GBW product and verify with answer obtained earlier.

observation: Use the tabular column separately for each circuit

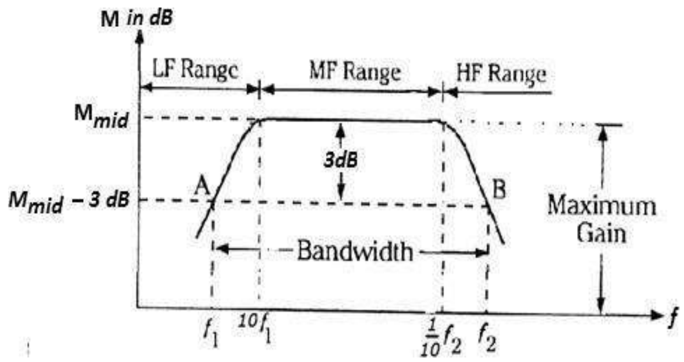
$V_{in(P-P)}$ = ...V

$A_V = V_{O\ (P-P)} / V_{in\ (P-P)}$

M = 20log (A_V), dB

Frequency In Hz	100	200	300	350	400	500	600	700	800	1K	2K	3K	5K	8K
$V_{O(P-P)}$ in Volts														
A_V														
M, dB, (A_V in dB)														
Frequency In Hz	10 K	20K	30K	50 K	100 K	200 K	300 K	400 K	500 K	600 K	700 K	800 K	900 K	1M
$V_{O(P-P)}$ in Volts														
A_V														
M, dB, (A_V in dB)														

Expected Graph:



EXPERIMENT: 2

.

COLPITTS OSCILLATOR ,CRYSTAL OSCILLATOR and RC PHASE SHIFT OSCILLATOR

Aim: Design and set-up the following oscillators circuits using BJT, and determine the frequency of oscillation.

i) Colpits Oscillator

ii) Crystal Oscillator

iii) RC phase shift oscillators

Components required: Transistor SL 100, Resistors 470Ω, 1KΩ 10KΩ and 33 KΩ; Capacitors 0.1µf - 3nos, Discrete inductances 100 µH – 2 nos, Capacitor 470 pF – 2nos, Power supply, CRO, Connecting wires etc.

Theory:

i) Colpits Oscillator: LC oscillators are generally used as RF oscillators since they generally used to create high frequency oscillations. In Colpitts oscillator an LC tank circuit is used for selection of frequency of oscillation. A voltage divider biased common emitter amplifier is used as amplifier. The amplifier and tank circuit together provides a phase shift of 360 degrees to satisfy Barkhausen criterion.

Design:

BJT- Amplifier design is same as given in Common Emitter Amplifier.

Tank Circuit Design: $f = \frac{1}{2\pi\sqrt{LC_{eq}}}$ Where $C_{eq} = \frac{C_1 C_2}{C_1 + C_2}$

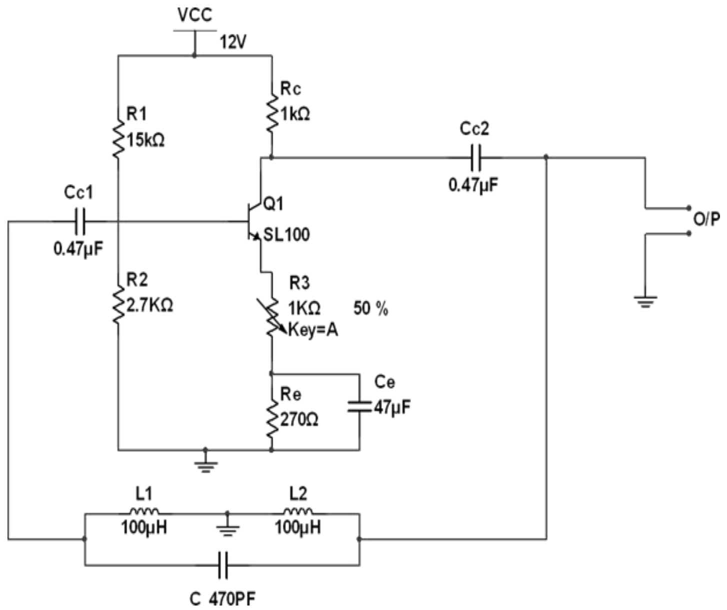
Given Oscillation frequency $f = 1$ MHz

Assume $C_1 = C_2 = 470$ pF ∴ $C_{eq} = 235$ pF $= 2.35 * 10^{-10}$F

Then, $L = \frac{1}{4\pi^2 (f^2) C} = 119$ µH

Use L = 100 µH, For this value of L, $f = 1.04$ MHz

Circuit Diagram:



Procedure:

1. Switch on the Power Supply and check the D.C conditions by removing the coupling capacitor CC1 or CC2.
2. Connect the coupling capacitors and obtain an output waveform on the CRO. If the o/p is distorted adjust 1- KΩ Potentiometer (R3) to get perfect SINE wave.
3. Measure the period of oscillation and calculate the frequency of oscillation.
4. Compare the measured frequency with re-computed theoretical value for the component values connected.

Observation

Parameter	VRC	VCE	VE	ICQ = VRC / RC	VBE	VB
Assumed	4.8V	6 V	1.2V	4.5 mA	0.6 V	1.8 V
Practical						

CRYSTAL OSCILLATOR

Theory:

Crystal oscillators are used in order to get stable sinusoidal signals despite of variations in temperature, humidity, transistor and circuit parameters. A piezo electric crystal is used in this oscillator as resonant tank circuit. Crystal works under the principal of piezo-electric effect. i.e., when an AC signal applied across the crystal, it vibrates at the frequency of the applied voltage. Conversely if the crystal is forced to vibrate it will generate an AC signal. Commonly used crystals are Quartz, Rochelle salt etc.

Components required: Transistor SL 100, Crystal – 2MHz, Resistors 470Ω, 1KΩ 10KΩ and 33 KΩ; Capacitors 0.1µf - 2nos, Power supply, CRO, Connecting wires etc.

Design:

Let V_{CC} = 12V; I_{CQ} = 4mA;

V_E = (1/10) V_{CC} to (1/5) V_{CC};

V_{CE}= 6V;

h_{FE} = 100.

To find R_E: Let us choose V_E = 2 V

$R_E = V_E / I_E \approx V_E / I_C$ =2 V/ 4 mA =500 Ω; let **RE =470Ω**

VCC = ICRC + VCEQ + VEQ

$R_C = (V_{CC} - V_{CEQ} - V_{EQ}) / I_{CQ}$ = 4.0 V/ 4mA =1.0K Ω; **RC= 1K Ω**

Assume **R2=10kΩ**.

$V_B = V_E + V_{BE} = 2 + 0.6 = 2.6V$

I_2 = Current through R_2= V_B /R_2 = 0.26mA or 260µA

The base current $I_B = I_C / h_{FE}$= 4mA / 100= 0.04mA= 40µA

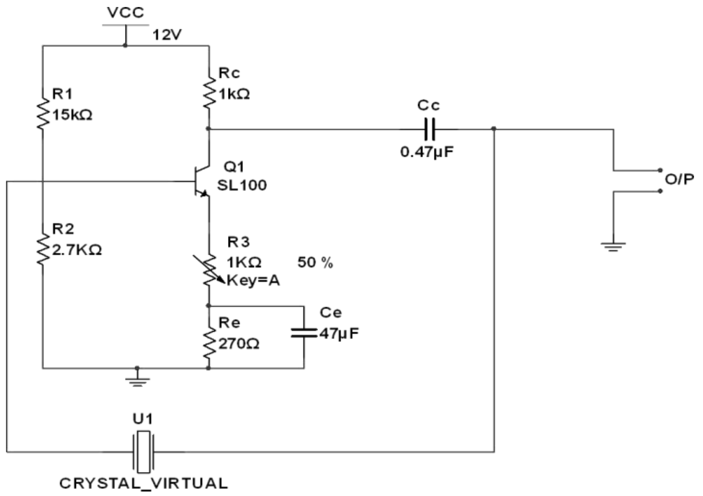
($h_{FE} = \beta_{DC}$ = 100, a working value; It varies from 50 to 280 for SL 100) I1 = Current through $R_1 = I_B+I_2$ = 300 µA

$V_{R1}=V_{CC}- V_B$=12 – 2.6 =9.4V

$R_1 = V_{R1}/ I_1$= 9.4V/300 µA =9400/300 KΩ =31.33KΩ R1 = 33K Ω

$C_E = C_C$ =0.1µF (Arbitrary, any value which gives a reactance < 10 Ω at Crystal frequency may be used. reactance of a Capacitor $X_C = (1/2\pi fC)$; For C = 0.1 µF, XC = 0.8Ω at 2 MHz)

Circuit Diagram



Procedure:

1. Switch on the Power Supply and before inserting the crystal check the D.C conditions by removing the coupling capacitor CC1 or CC2.
2. Insert the crystal and the coupling capacitors and obtain the output waveform on the CRO. If the o/p is distorted vary 1- KΩ Potentiometer (R3)to get perfect SINE wave.
3. Measure the period of oscillation and calculate the frequency of oscillation.
4. Compare with frequency marked on the crystal

Observation:

Parameter	VRC	VCE	V_E	ICQ = VRC / RC	VBE	V_B
Assumed	4.8V	6 V	1.2V	4.5 mA	0.6 V	1.8 V
Practical						

RC PHASE SHIFT OSCILLATOR

Theory

A Phase Shift Oscillator is an electronic oscillator circuit which produces sine wave output. It can either be designed by using transistor or by using an Op-amp as inverting amplifier. Generally, these phase shift oscillators are used as audio oscillators. In RC phase shift oscillator, 180 degree phase shift is generated by the RC network and another 180 degree is generated by the Op-amp, so the resulting wave is inverted by 360 degree.

Apart from generating the sine wave output they are also used to provide significant control over the phase shifting process

Circuit diagram

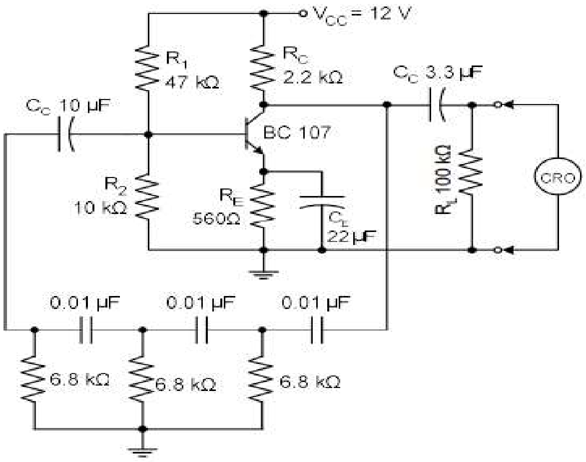


Fig1 g. Circuit Diagram of RC Phase shift Oscillator

DESIGN

From the transistor data sheet, for BC107,

$h_{fe} = \beta = 110$, $I_{c\,max} = 100$ mA, $V_{CE\,max} = 45$V

Let $V_{CC} = 12$V, $I_c = 2$mA. Since the quiescent point is in the middle of the load line for the amplifier, $V_{CE} = 50\%$ of $V_{CC} = 6$V.

$V_{RE} = 10\%$ of $V_{CC} = 1.2$ V

Assuming $I_C = I_E$, $\quad V_{RE} = I_C R_E = I_E R_E$

$1.2 = 2\times10^{-3} \times R_E$

$R_E = \frac{1.2}{2\times10^{-3}} = 600\ \Omega$ Select standard value of resistance 560 Ω.

Voltage across collector resistance, $V_{RC} = V_{CC} - V_{CE} - V_{RE}$

$= 12 - 6 - 1.2 = 4.8$ V

$R_C = \frac{V_{RC}}{I_C} = \frac{4.8}{2\times10^{-3}} = 2.4\ k\Omega$ Select standard value of 2.2 kΩ

Base current, $I_B = \frac{I_C}{\beta} = \frac{2\times10^{-3}}{110} = 18.2\ \mu A$

Take $I_2 = I_B$ then $I_1 = 10I_B + I_B = 11I_B$

Base voltage, $V_B = V_{RE} + V_{BE} = 1.2 + 0.6 = 1.8$ V

$R_2 = \frac{V_B}{I_2} = \frac{1.8}{10\times18.2\times10^{-6}} = 9.9\ k\Omega$ Select standard value of 10 kΩ

$$R_1 = \frac{V_{CC} - V_B}{I_1} = \frac{12 - 1.8}{11 \times 18.2 \times 10^{-6}} = 51\ \text{k}\Omega$$ Select standard value of 47 kΩ

Design of coupling capacitors C_{C1} and C_{C2}

X_{C1} should be less than the input impedance of the transistor. Here, R_{in} is the series impedance.

Then $X_{C1} \le \frac{R_{in}}{10}$

Here $R_{in} = R_1 \parallel R_2 \parallel h_{FE}\ r_E = 47\text{k}\Omega \parallel 10\ \text{k}\Omega \parallel 110 \times 12.5\ \Omega = 1.17\ \text{k}\Omega$

We get R_{in}=1.17 kΩ. Then $X_{C1} \le 117\ \Omega$.

For a lower cut off frequency of 200 Hz, $C_{C1} = \frac{1}{2\pi f X_{C1}} = \frac{1}{2\pi \times 200 \times 117} = 6.8\ \mu\text{F}$

Select standard value of 10 µF for C_{C1}

Similarly, $X_{C2} \le \frac{R_{out}}{10}$ where R_{out}=R_C. Then $X_{CE} \le 220\Omega$.

So, $C_{C2} = \frac{1}{2\pi f X_{C2}} = \frac{1}{2\pi \times 200 \times 220} = 3.6\ \mu\text{F}$

Select standard value of 3.3 µF for C_{C2}

Design of bypass capacitors C_E

To bypass the lowest frequency (say 200 Hz), X_{CE} should be much less than or equal to the resistance R_E.

$$X_{CE} \le \frac{R_E}{10}$$

$$X_{CE} \le \frac{560}{10} \qquad \text{ie. } X_{CE} \le 56$$

Apply value of f such that the amplifier has good gain at a lower cutoff frequency of 200 Hz

$$C_E \ge \frac{1}{2\pi f X_{CE}} = \frac{1}{2\pi \times 200 \times 56} = 14.2\ \mu\text{F}$$

Select standard value of 22 µF for C_E

Design of Feedback network

The circuit consists of an amplifier stage and a feedback network to provide an additional 180° phase shift, approximately depending upon the frequency of operation. The RC phase shift network must provide 180° or an average of 60° phase shift/lag of RC network

RC phase shift factor, $k = V_S/V_0$.

For one stage of RC network

$$V_o = I\,R \qquad V_C = I\,X_C$$

$$\tan\Phi = \frac{V_C}{V_o} = \frac{I\,X_C}{I\,R} = \frac{1}{(2\pi f\,C)R}$$

$$f = \frac{1}{2\pi RC\ \tan\Phi}$$

If these are three sections, each must give, approximately $\Phi = 60^o$ then $\tan 60 = \sqrt{3}$

$$f = \frac{1}{2\pi RC\sqrt{3}}$$

This gives the approximate frequency of oscillation of a phase shift with three RC sections.

In the above phase relationship, between voltage and current in the RC network, the additional current I that flow through C for the other sections so that V_C is larger than the value indicated which means that f is smaller than the value obtained in the above equation. A transfer function analysis of the three-stage network would give a more accurate expression for frequency of oscillation as below.

$$\boxed{f = \frac{1}{2\pi RC\sqrt{6}}}$$

Assume $f = 1000$ Hz and $C = 0.01$ µF

$$1000 = \frac{1}{2\pi R \times 0.1 \times 10^{-6} \times \sqrt{6}}$$

$$R = \frac{1}{2\pi \times 1000 \times 0.01 \times 10^{-6} \times \sqrt{6}} = 6.5\ \text{k}\Omega$$

Select nearest standard value of 6.8 kΩ for R

The three-stage feedback network would have an attenuation of $\left(\frac{1}{29}\right)$ and to satisfy the Barkhausen criterion for oscillation, the amplifier should have a gain of 29 or more. If the gain is just above 29, a pure sine wave will be generated. If the gain is too high, there may be distortions in the output waveform. The circuit in figure 1 is having gain of more than 29 by default. A quick method to adjust the voltage gain is to adjust the load resistance R_L.

Procedure:

1. Switch on the Power Supply and check the D.C conditions by removing the coupling capacitor CC1 or CC2.
2. Connect the coupling capacitors and obtain an output waveform on the CRO. If the o/p is distorted adjust 1- KΩ Potentiometer (R3) to get perfectSINE wave.
3. Measure the period of oscillation and calculate the frequency of oscillation.
4. Compare the measured frequency with re-computed theoretical value forthe component values connected.

Observation

Parameter	VRC	VCE	VE	ICQ = VRC / RC	VBE	VB
Assumed	4.8V	6 V	1.2V	4.5 mA	0.6 V	1.8 V
Practical						

Result: The frequency of

- **i.** Colpits Oscillator is…….
- **ii.** Crystal Oscillator is……..
- **iii.** RC phase shift oscillators is ……

EXPERIMENT: 3

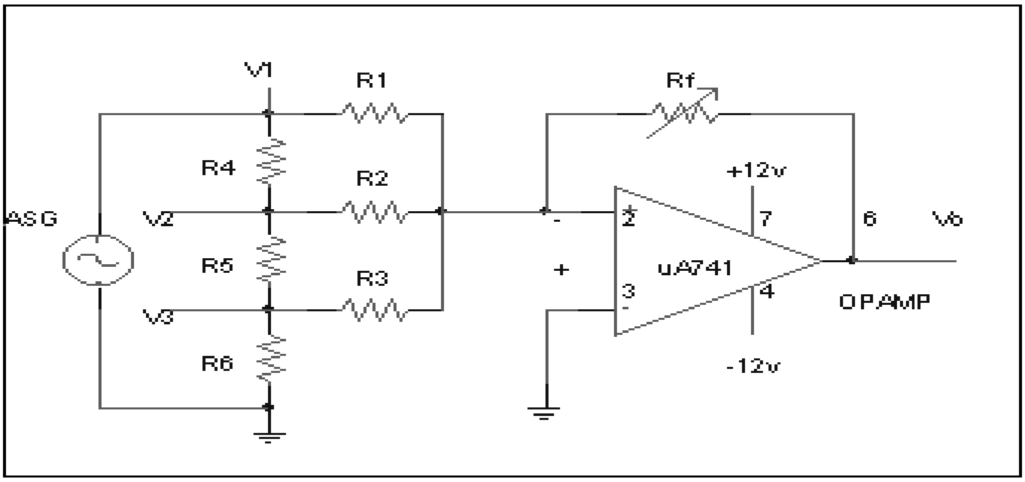
OPAMP AS AN ADDER, INTEGRATOR, DIFFERENTIATOR and COMPARATOR

Aim: To Design Adder, Integrator , Differentiator and comparator circuits using Op-Amp

Components: Op-Amp µA 741, Resistors, Capacitor, Signal Generator, CRO, Fixed Power supply +12V,0,-12V.

A. Op-Amps as summer(Inverting)

Circuit Diagram:



Design :

Design a summing amplifier to obtain a output of –6V. Since inputs are given to Inverting terminal, output is given by,

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

Now,

$$\frac{R_f}{R_1} = \frac{R_f}{R_2} = \frac{R_f}{R_3} = \frac{1k\Omega}{3k\Omega} = \frac{1}{3}$$

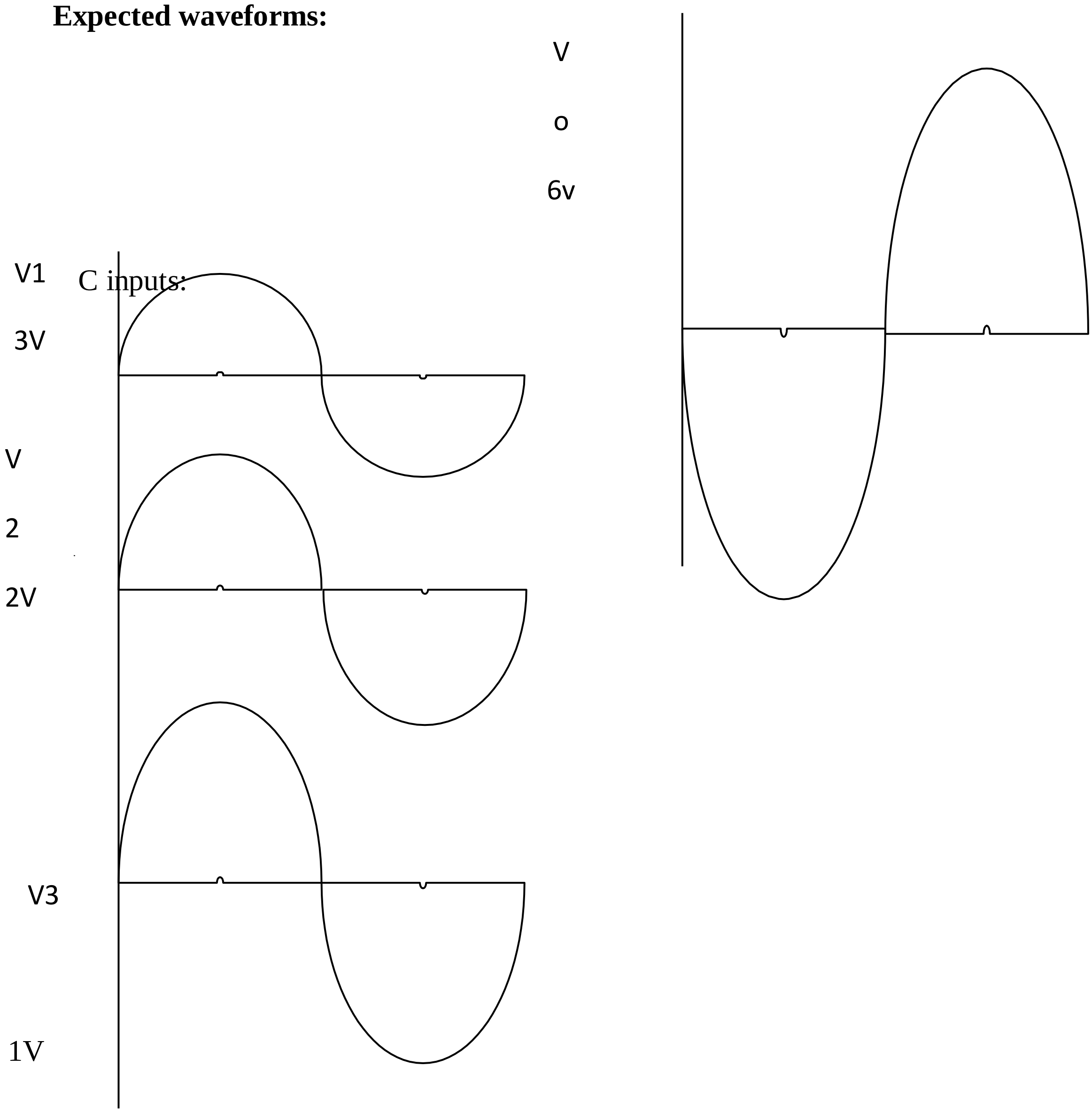
$$V_{out} = -(\frac{V_1 + V_2 + V_3}{3})$$

Procedure :

1. Connections are made as shown in the circuit diagram.

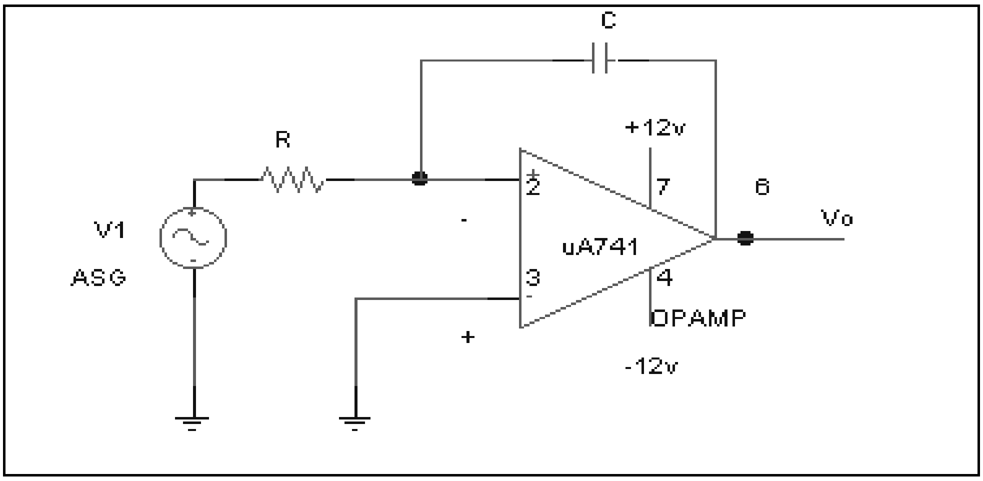
2. With the chosen values of R_f, R_i, and R_3, provide DC voltages V_1, V_2, and V_3 from VRPS.
3. Measure the output voltage and compare it with the designed value.
4. Repeat the above procedure providing AC sinusoidal signal of frequency 1KHz for V_1, V_2 and V_3 and observe the output waveform.

Expected waveforms:



B. Op-amp as an integrator:

Circuit Diagram:



Design :

Design a integrator circuit for different values of R and C. Since input is given to Inverting terminal, the output of integrator is given by

$$V_0 = -\frac{1}{RC}\int V_i \, dt$$

Note: Requirement for integration is RC>>T, where T is the time period of input signal.

Consider input square wave of frequency 1 KHz

$\therefore$ T = $\frac{1}{f}$ = 1 ms

Since RC >> T, let RC = 10 T = 10ms

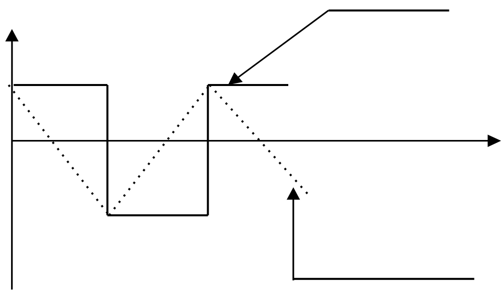
For C = 0.1 μf, R = $\frac{10\times10^{-3}}{0.1\times10^{-6}}$ = 100KΩ

Choose,R = 100KΩ and C = 0.1 μf

Procedure:

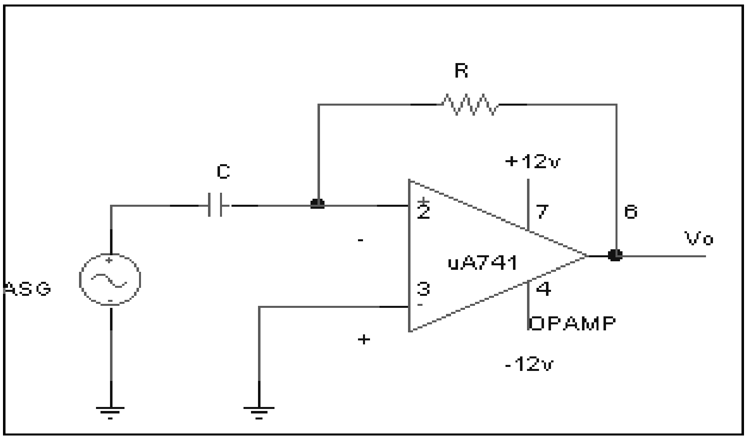
1. Connections are made as shown in the circuit diagram.
2. The input square wave signal (V_i) is set to 4V(p-p) of 1KHz frequency.
3. For the chosen values of Rand C, Observe the output Waveform (V_o) on the CRO and verify it with the expected waveforms.
4. Repeat the experiment for different values of R and C (RC = 10T, RC = T, RC=0.1T).

Expected waveforms: **vi**



C. Op-amp as a differentiator:

Circuit Diagram:



Design :

Design a differentiator circuit for different values of R and C. Since input is given to inverting terminal, the output of differentiator is given by

$$V_o = -RC\frac{dVi}{dt}$$

Note: Requirement for integration is RC<< T, where T is the time period of input signal.

Consider input square wave of frequency 1 KHz

$$T = \frac{1}{f} = 1 \text{ ms}$$

Since RC << T, let RC = $\frac{1}{10}T$ = 0.1 T

for C = 0.1 μf

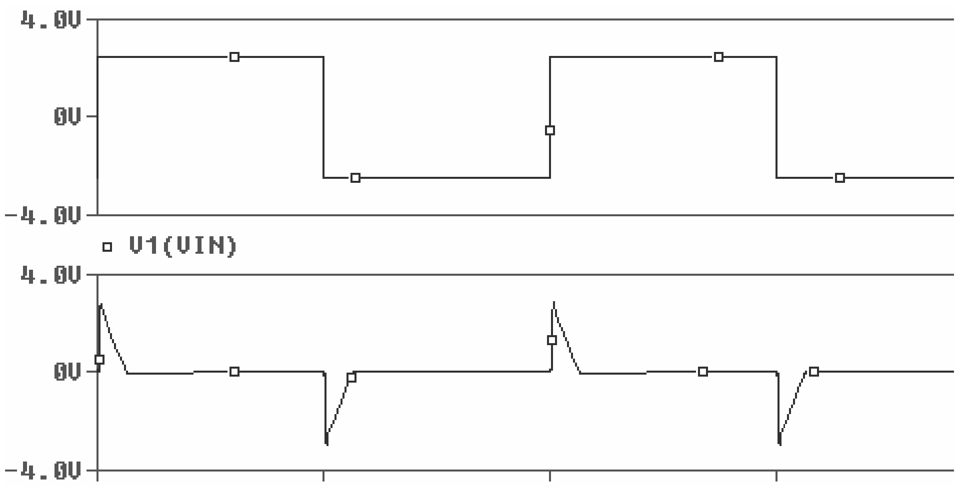
$$\therefore R = \frac{0.1\times10^{-3}}{0.1\times10^{-6}} = 1K\Omega$$

Choose, R = 1KΩ and C = 0.1 μf

Procedure :

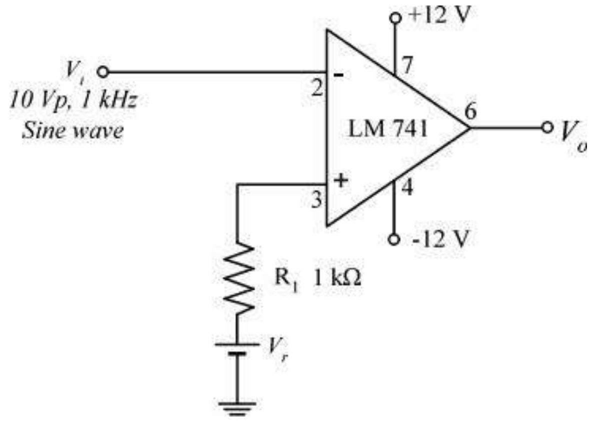
1. Connections are made as shown in the circuit diagram.
2. The input square wave signal (V_i) is set to 4V(p-p) of 1KHz frequency.
3. For the chosen values of Rand C, Observe the output Waveform (V_o) on the CRO and verify it with the expected waveforms.
4. Repeat the experiment for triangular waveform.
5. Repeat the experiment for different values of R and C (RC = 10T, RC = T, RC=0.1T).

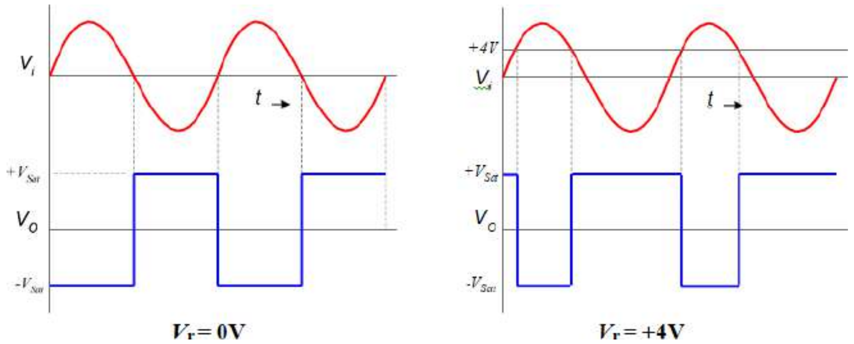
Expected waveforms:



Comparator A voltage comparator is a two-input circuit that compares the voltage at one input to the voltage at the other input. Usually one input is a reference voltage and the other input a time varying signal. If the time varying input is below or above the reference voltage, then the comparator provides a low or high output accordingly (usually the plus or minus power supply voltages, since the op-amp is used in the open loop configuration, a small difference (−) makes the output to saturate). For the comparator circuit shown in Figure 1, the output will be at its negative saturation value when the input is greater than the reference and at its positive saturation value when the input is less than the reference. If Vr is zero, the comparator can be used as a zero-crossing detector. If Vr is not zero, the comparator can be referred to as a level detector. One problem encountered with the simple comparator is the instability of its output resulting from noise when the input is in the neighborhood of Vr.

Circuit Diagram and waveforms of Comparator





Procedure

1. Set up circuit as shown in the connection diagram
2. Set the input voltage 20 V peak to peak, 1 kHz in function generator, and apply input signal to the circuit.
3. Observe the output waveform in CRO.
4. Obtain the response for different V_r (for comparator circuit only).

Result

Adder, integrator, differentiator and Comparator circuits were designed and set up. And the output waveform isobserved on CRO

EXPERIMENT: 4

SCR controlled HWR and FWR using RCTriggering circuit

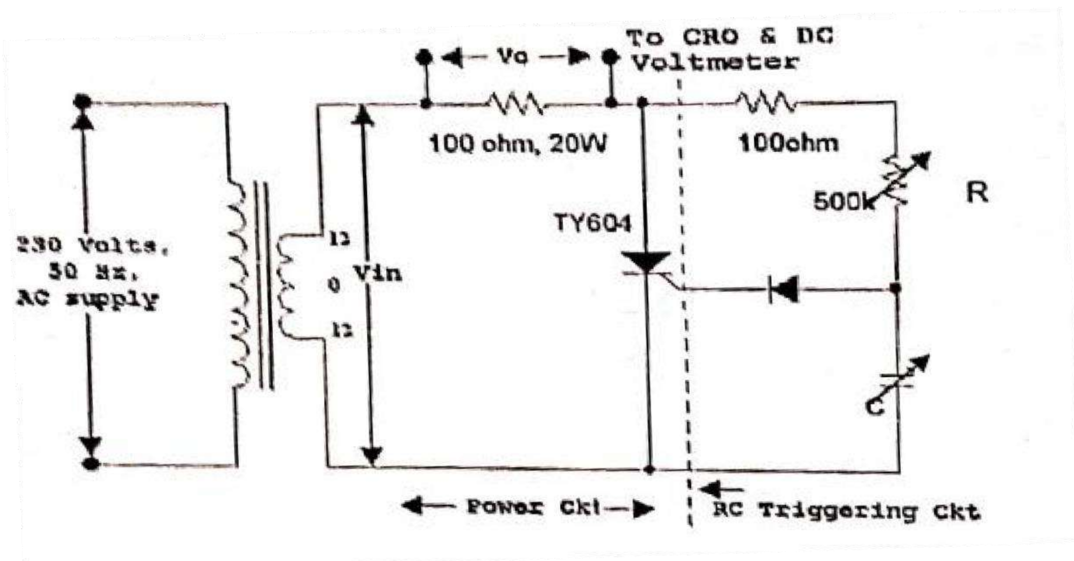
AIM: To study the performance and waveforms of HWR & FWR by using RC triggering circuit

APPARATUS:

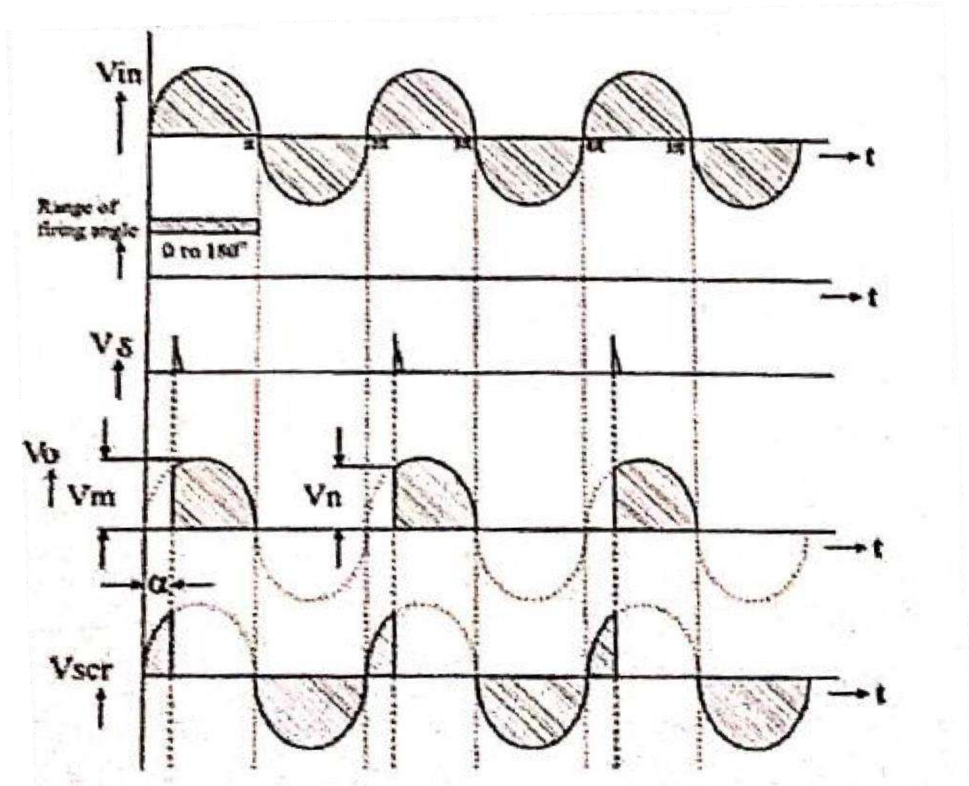
SL No	Components	Range	Quality
1	DC Power Supply	0-30V	2
2	Resistor	100Ω	1
3	Transformer	230V, 50Hz	1
4	SCR Diode	TY604 BY127	1
5	DC voltmeter	(0-20)V	1
6	DC ammeter	(0-200)mA (0-200)µA	1
7	Connecting wires		

CIRCUIT DIAGRAM:

Half wave rectifier:

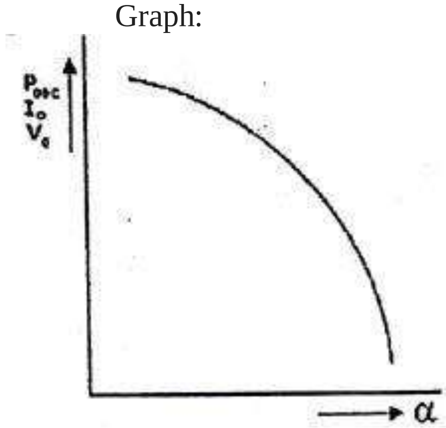


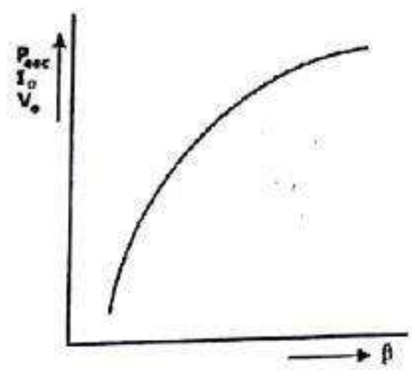
Waveforms:



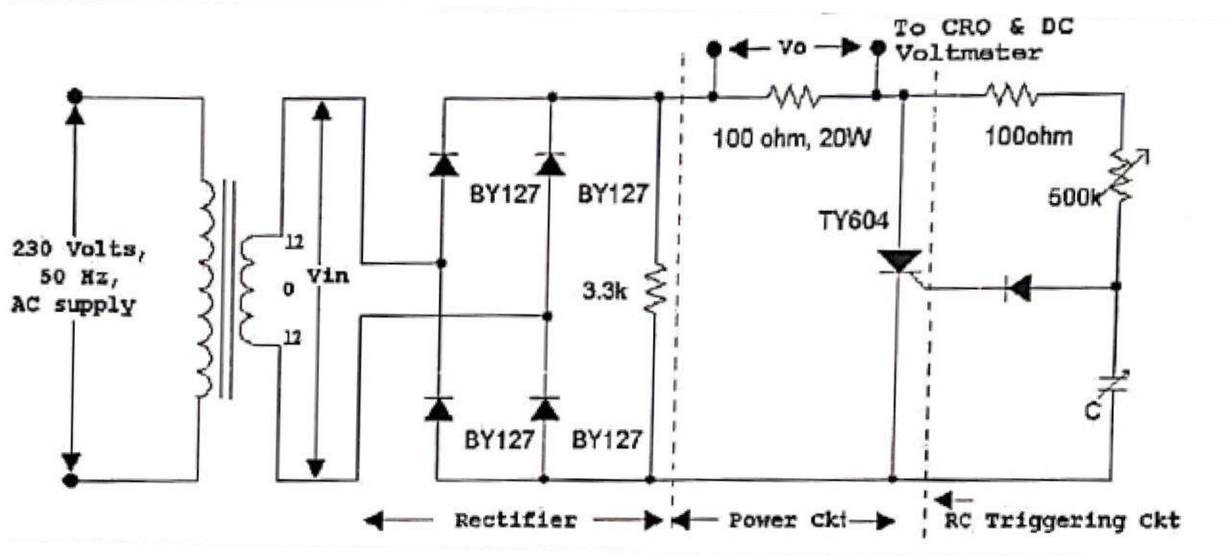
Where α = firing angle

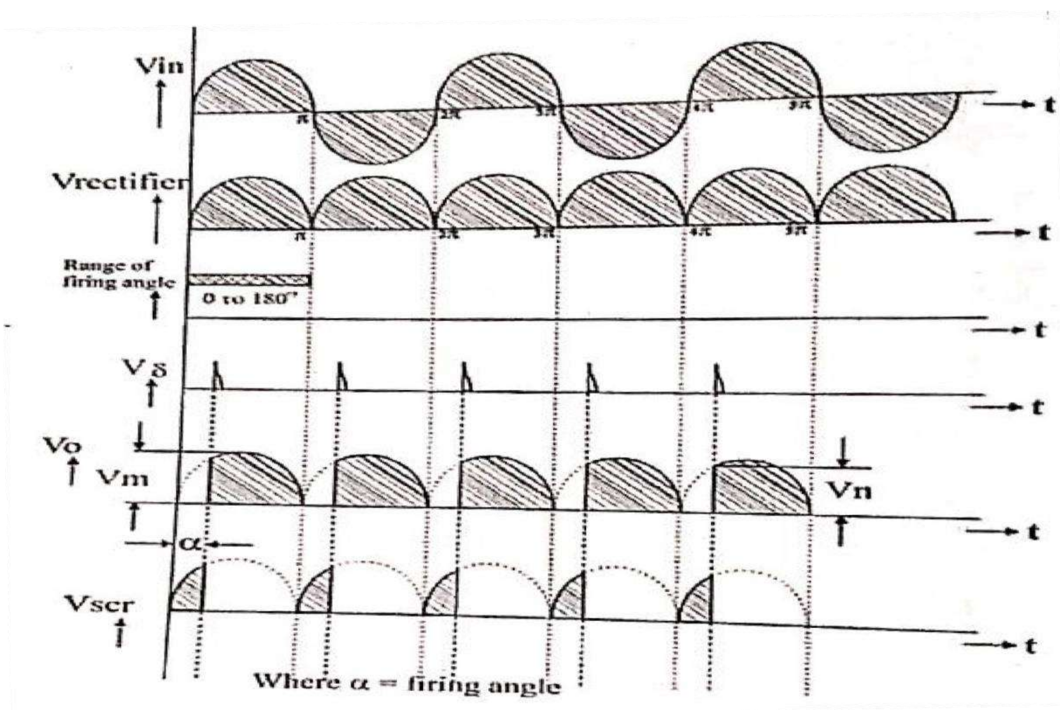
Graph:





Full wave rectifier:





PROCEDURE:

Half wave rectifier:

1. Connections are made as shown in the circuit diagram.
2. By varying a resistance R gradually step by step, note down the corresponding values of V_n& V_m from CRO and $V_{o\ dc}$ from the DC voltmeter. The readings are tabulated in the tabular column.
3. If the firing angle ranges from 0 to 90°, then the firing angle α is calculated by using a formula $\alpha = \sin^{-1}(V_n / V_m)$ in degrees.
4. The conduction angle β is calculated by using a formula, β = 180- α.
5. The current and power is calculated by
$I_{o\ dc} = (V_{o\ dc}/R)$ in A & $P_{o\ dc} = V^2_{odc}/R$ in watts respectively.
6. A graph of Vo v/s α, Vo v/s β, Io v/s α, lo v/s β, $P_{o\ dc}$ v/s α, $P_{o\ dc}$ v/s β to be plotted.
7. Compare practical output voltage with theoretical output voltage,
$V_{oth}=V_m(1+\cos\alpha)/2\pi$ volts where $V_m=\sqrt{2}V_{rms}$

Full wave rectifier:

1. Repeat the above said procedure for full wave rectifier.
$V_{oth}=V_m(1+\cos\alpha)/\pi$ volts where $V_m=\sqrt{2}V_{rms}$

TABULAR COLUMN:

Half wave rectifier

SL No	**V_n**	**V_m**	**α < 90 degree**	**α> 90 degree**	**Vodc**	**Voth**
			$\alpha=\sin^{-1}(V_n/V_m)$	$\alpha=180-\sin^{-1}(V_n/V_m)$		

Full wave rectifier

SL No	**V_n**	**V_m**	**α < 90 degree**	**α> 90 degree**	**Vodc**	**Voth**
			$\alpha=\sin^{-1}(V_n/V_m)$	$\alpha=180-\sin^{-1}(V_n/V_m)$		

Result:

The performance and waveforms of HWR & FWR by using RC triggering circuit is verified

EXPERIMENT 5

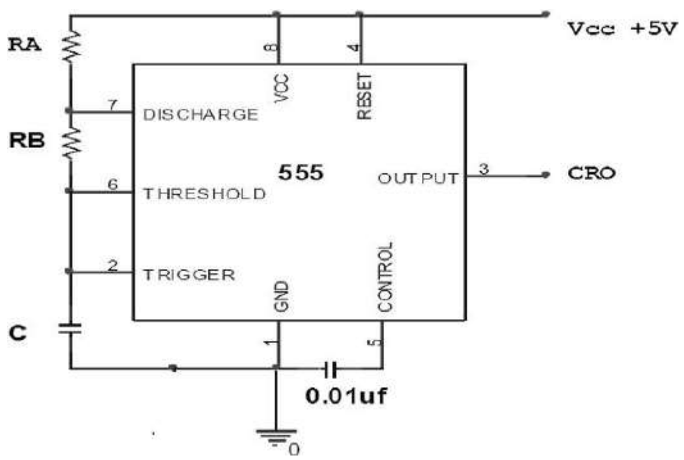
MONOSTABLE AND ASTABLE MULTIVIBRATOR USING 555 TIMER

Aim: To Design of Monostable and Astable Multivibrator using 555 Timer

Components required: 555 Timer, Resistor, Capacitor, Power supply +5V, 0, CRO.

a) Astable Multivibrator (free running oscillator):

Circuit:



Design:

Output time period of oscillations = $T=T_{on} + T_{off}$

Charging time Ton = 0.693 (R_A+R_B) C1

Discharge time T_{off} = 0.693 R_B.C1

Duty cycle D = $\frac{Ton}{Ton + Toff} = \frac{Ton}{T}$ =0.75

Let T = $\frac{1}{f} = 1msce,$

T_{off} = 0.693 R_B.C1

$0.25x10^{-3}$ =0.693 R_B $x0.1x10^{-6}$

R_B = 3.6 KΩ choose as **3.3 KΩ**

Ton = 0.693 (R_A+R_B) C1

$0.75x10^{-3}$ =0.693 (R_A +3.3 K) $x0.1x10^{-6}$

R_A=**7.2 KΩ choose as 6.8 KΩ**

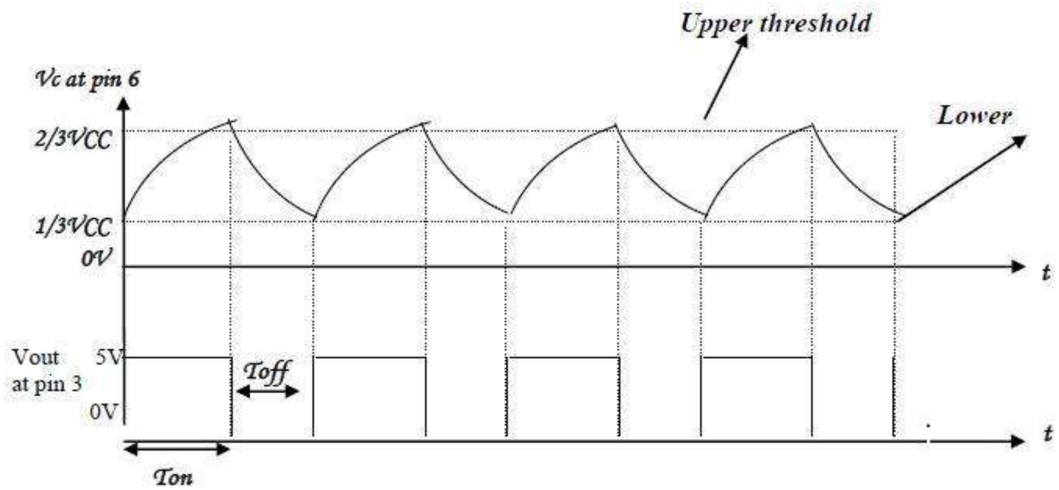
Here V_{LT} = Lower threshold voltage = $V_{cc}/3$ = 1.66V

V_{UT} = Upper threshold voltage = $\frac{2}{3}V_{CC} = \frac{2}{3} \times 5 = 3.33V$

Procedure:

1. Connections are made as shown in the circuit diagram
2. Switch on the DC power supply unit
3. Observe the wave form on CRO at pin 3 and measure the o/p pulse amplitude and also measure Ton & Toff.
4. Observe the waveforms across C1 (Vc) & measure the max & min voltage levels & verify V_{UT} & V_{LT}.
5. Compare the capacitor voltage Vc with the output waveform Vo & note that capacitor charges & Vc rises exponentially when output is high, the capacitor C1 discharges through R_B & the discharge transistor & Vc falls exponentially when output is low.
6. Calculate the duty cycle & the output frequency f & verify with the designed values.

Waveform:



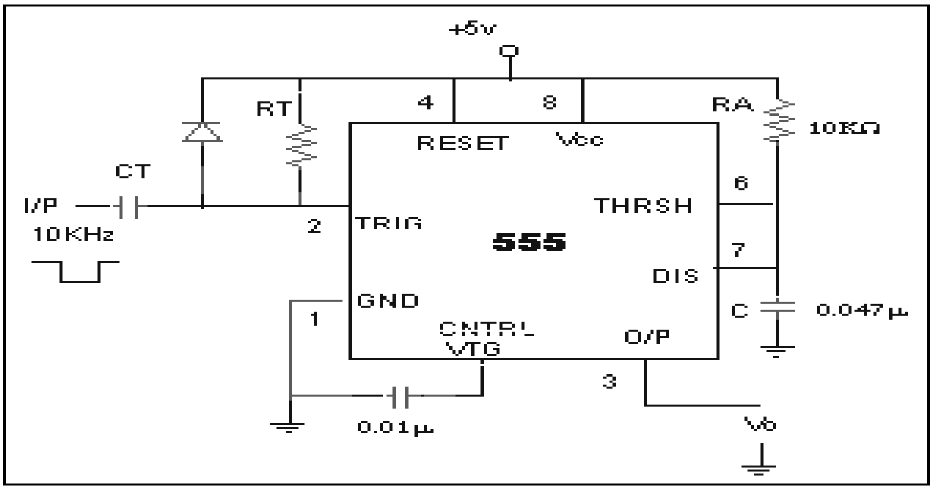
Tabular Column:

Parameters	Theoretical	Practical
Frequency		
Duty cycle		
Ton		
Toff		
$V_{LT} = V_{cc}/3$		
$V_{UT} = \frac{2}{3}V_{cc}$		

b) Monostable Multivibrator circuit Using 555 Timer.

Components required: Op-Amp μA 741, Capacitor, Function Generator , Fixed Powersupply +12V,0,-12V, CRO.

Circuit:



Design:

Let output pulse width = Delay time t_d 0.5 msec

Output delay time $T_d = 1.1\ R_A.C$

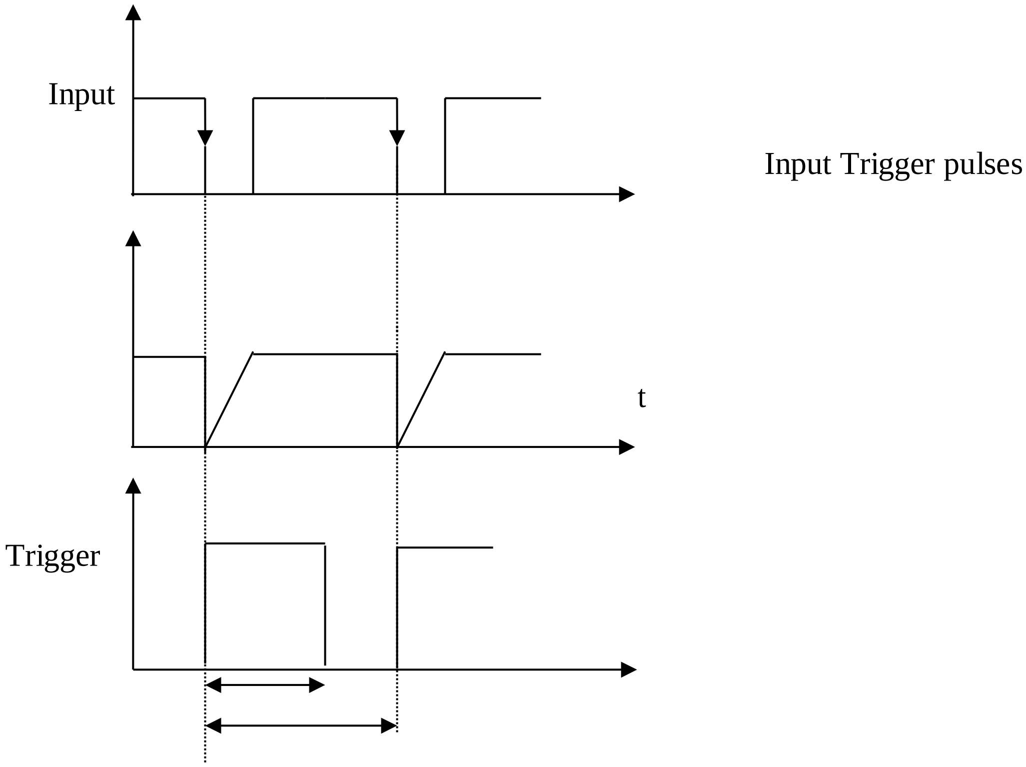
Let $R_A = 10\ K\Omega$

$$\therefore C = \frac{T_d}{1.1 \times R_A} = \frac{0.5 \times 10^{-3}}{1.1 \times 10 \times 10^{3}} = 0.045\mu$$

Choose C = 0.047 μF

Here **V_{UT}** = Upper threshold voltage = $\frac{2}{3} V_{cc}$

Waveforms:



Procedure:

1. Connections are made as shown in Figure
2. Switch on the power supply and observe the output waveforms on CRO at pin no 3 and measure the output delay time T_d and verify with the designed values and also observe the waveforms at pin no 2 – trigger input terminal & at pin no 7, also measure the voltage levels.

Result:

Output delay time of monostable and astable vibrators are verified.

EXPERIMENT 6

PRECISION HALF WAVE AND FULL WAVE RECTIFIER

AIM: To verify precision half wave and full wave rectified using opamp .

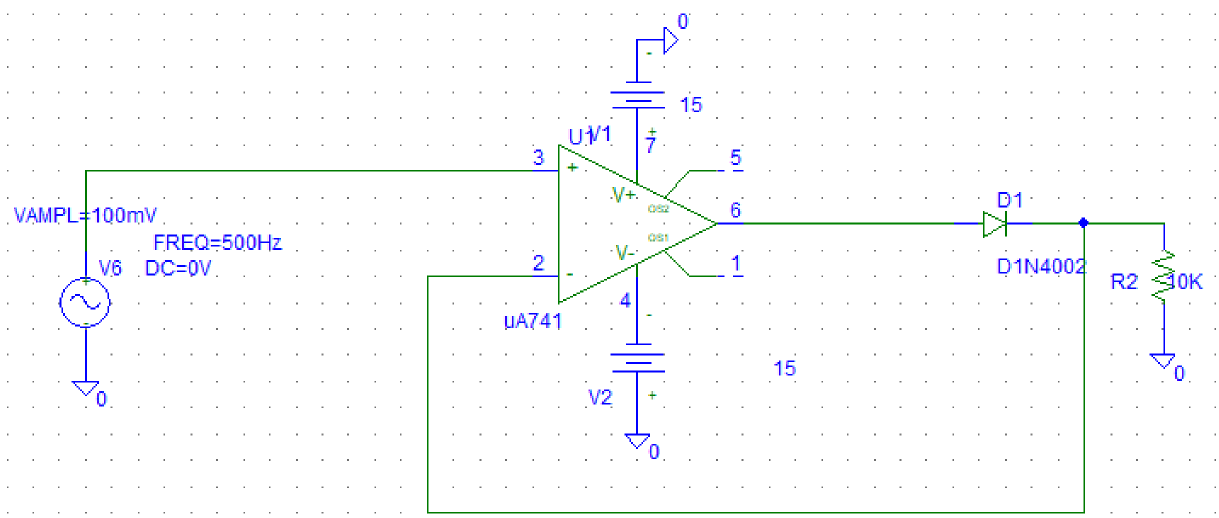
THEORY:

Precision Rectifier:Rectifiers are often called into action to measure signal strength. Rectify an AC signal, pass it through a low-pass filter and the resulting DC level represents some measure of the signal's magnitude. Although the series diode is the classic rectifier, it can't rectify signals smaller that it own forward voltage! But what if your expected amplitude can be as low as 100 mV? Op amps to the rescue! The advantage of op amp circuits lies in their ability to compensate for non-linear devices in the feedback loop. Combining the rectifying action of a diode with the accuracy of an op amp, this circuit creates a precision rectifier.

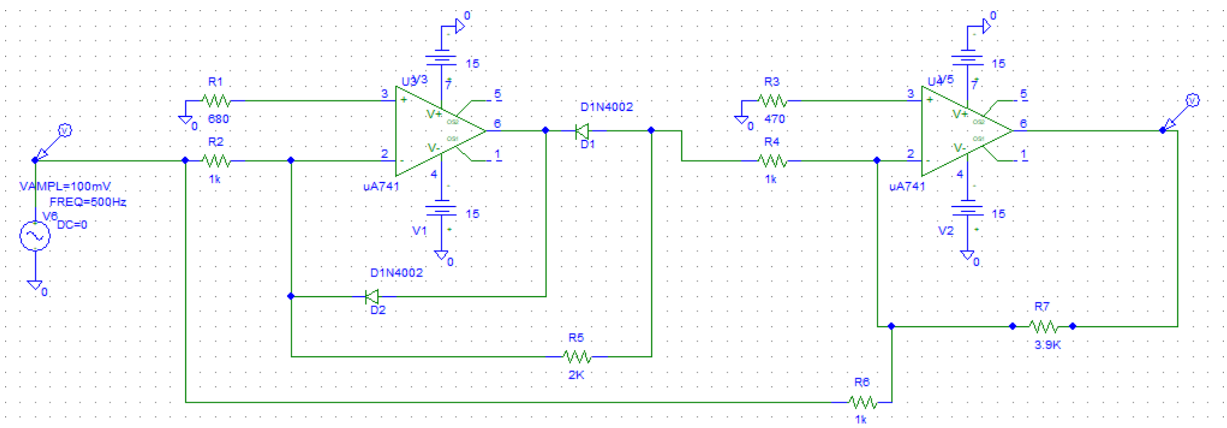
Precision Half wave rectifier: A half wave precision rectifier is implemented using an op amp, and includes the diode in the feedback loop. This effectively cancels the forward voltage drop of the diode, so very low level signals (well below the diode's forward voltage) can still be rectified with minimal error.

Precision Full Wave Rectifier circuits accept an ac signal at the input, inverts either the negative or the positive half, and delivers both the inverted and noninverted halves atthe output, as shown in the Fig

SIMULATION CIRCUIT DIAGRAM (Precision HWR)

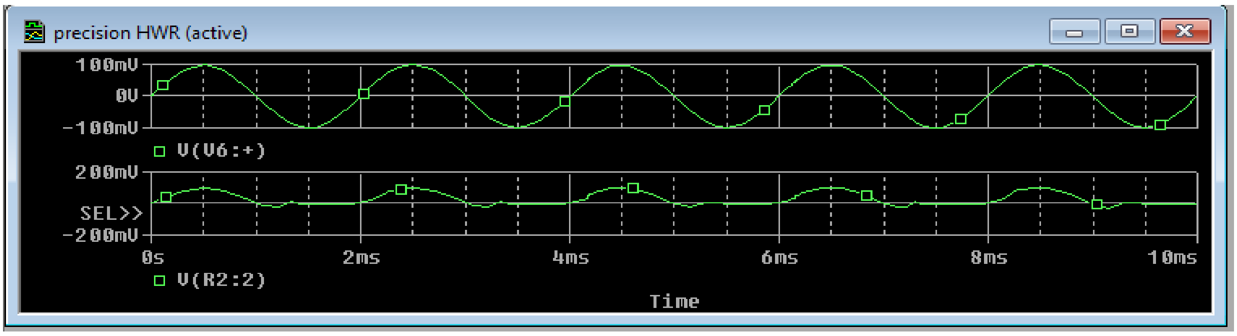


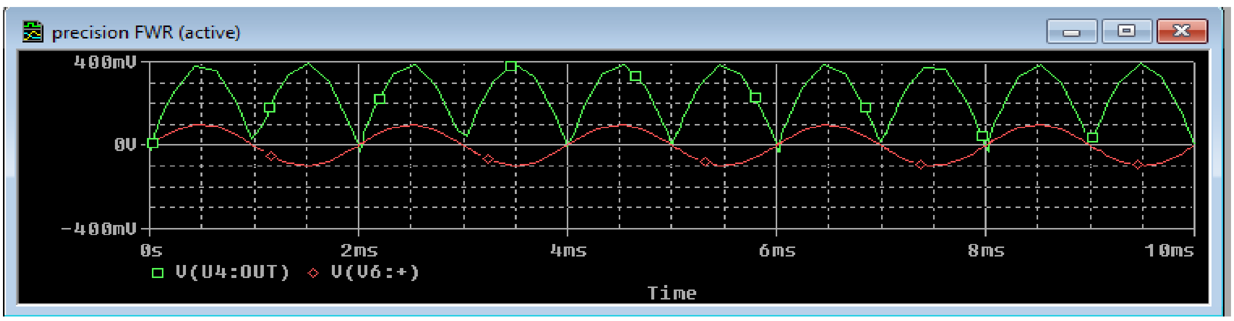
SIMULATION CIRCUIT DIAGRAM (Precision FWR)



Procedure (Same for both HWR & FWR)

1. Connections are made as shown in the circuit diagram.
2. The input square wave signal (V_i) is set to 100mV(p-p) of 500Hz frequency.
3. For the chosen values of Rand C, Observe the output Waveform (V_o) on the CROand verify it with the expected waveforms.
4. Repeat the experiment for different values of R and C





Result:

Precision half wave and full wave rectified using opamp is verified