SRM VALLIAMMAI ENGINEERING COLLEGE

(An Autonomous Institution)

SRM NAGAR,
KATTANKULATHUR

DEPARTMENT OF ELECTRICAL & ELECTRONICS ENGINEERING

Lab Manual

III SEMESTER

1905306-ANALOG ELECTRONICS LABORATORY

Academic year 2022-2023
ODD SEMESTER
(Regulation 2019)

Prepared by

Mr. S. Balaji, Assistant Professor (O.G)
Ms. P. Bency, Assistant Professor (O.G)

General Instructions to students for EEE Lab courses

- Be punctual to the lab class.
- Attend the laboratory classes wearing the prescribed uniform and shoes.
- Avoid wearing any metallic rings, straps or bangles as they are likely to prove dangerous at times.
- Girls should put their plait inside their overcoat
- Boy's students should tuck in their uniform to avoid the loose cloth getting into contact with rotating machines.
- Acquire a good knowledge of the surrounding of your worktable. Know where the various live points are situated in your table.
- In case of any unwanted things happening, immediately switch off the mains in the work table.
- This must be done when there is a power break during the experiment being carried out.
- Before entering into the lab class, **you must be well prepared for the experiment** that you are going to do on that day.
- You must bring the related text book which may deal with the relevant experiment.
- Get the circuit diagram approved.
- Prepare the list of equipment and components required for the experiment and get the indent approved.
- Plan well the disposition of the various equipment on the worktable so that the experiment can be carried out.
- **Make connections as per the approved circuit diagram and get the same verified. After getting the approval only supply must be switched on.**
- Get the reading verified. Then inform the technician so that supply to the worktable can be switched off.
- You must get the observation note corrected within two days from the date of completion of experiment. Write the answer for all the discussion questions in the observation note. If not, marks for concerned observation will be proportionately reduced.
- Submit the record note book for the experiment completed in the next class.
- If you miss any practical class due to unavoidable reasons, intimate the staff in charge and do the missed experiment in the repetition class.
- Such of those students who fail to put in a minimum of 75% attendance in the laboratory class will run the risk of not being allowed for the University Practical Examination. They will have to repeat the lab course in subsequent semester after paying prescribed fee.
- **use isolated supply for the measuring instruments like CRO in analog electronics Laboratory experiments.**

SYLLABUS

1905306	ANALOG ELECTRONICS LABORATORY	L	T	P	C
		0	0	4	2

OBJECTIVES:

To impart knowledge on the following Topics

1 To be familiar with the structure of basic electronic devices.

2 To be exposed to the operation and application of electronic devices and their circuits.

3 To analyze circuit characteristics with signal analysis using Op-amp ICs.

4 To design and construct application circuits with ICs as Op-amp, 555 etc.

5 To study internal functional blocks and the applications of special ICs like Timers, DAC/ADCs

I: EXPERIMENTS ON BASIC ELECTRONIC DEVICES:

1. Introduction to circuit simulation package by:
   i) PN junction characteristics
   ii) Transistor (CE conf) characteristics
   iii) JFET characteristics.
2. Frequency response of transistor amplifier circuit.
3. Line and load regulation of Zener regulator.
4. UJT – relaxation oscillator circuit.
5. Wien bridge oscillator.
6. Transistorized Differential amplifier.

II: EXPERIMENTS USING LINEAR INTEGRATED CIRCUITS (ICS):

7. OPAMP based amplifier circuits:
   i) Inverting amplifier.
   ii) Non-inverting amplifier and voltage follower
   iii) Differential amplifier and Instrumentation amplifier.
8. Design of Adder-subtractor circuits.
9. Square wave oscillator/ Tri-angular wave oscillator.
10. OPAMP based RC –phase shift oscillator.
11. 555 – timer IC based astable multi-vibrator.
12. OPAMP based precision rectifier circuit/ clipper circuits.

TOTAL: 60 PERIODS

LIST OF EXPERIMENTS

Exp. No	CYCLE- I
	BASIC ELECTRONIC DEVICES
1.	Introduction to circuit simulation package by: a) PN junction characteristics b) Transistor (CE conf) characteristics c) JFET characteristics.
2.	Frequency response of transistor amplifier circuit.
3.	Line and load regulation of Zener regulator.
4.	UJT – relaxation oscillator circuit.
5.	Wien bridge oscillator.
6.	Transistorized Differential amplifier.
Exp. No	**CYCLE- II**
	LINEAR INTEGRATED CIRCUITS (ICS)
7.	OPAMP based amplifier circuits: a) Inverting amplifier. b) Non-inverting amplifier and voltage follower c) Differential amplifier and Instrumentation amplifier.
8.	Design of Adder-subtractor circuits.
9.	Square wave oscillator/ Tri-angular wave oscillator
10.	OPAMP based RC –phase shift oscillator.
11.	555 – timer IC based astable multi-vibrator
12.	OPAMP based precision rectifier circuit/ clipper circuits
	ADDITIONAL EXPERIMENTS
13.	Integrator
14.	Study of CRO for frequency and phase measurements

S.No	Date	List Of Experiments	Signature

S.No	Date	List Of Experiments	Signature

Ex. No.: 1

Date:

SIMULATION PACKAGE

STUDY OF BASIC PSPICE:

I. Opening PSpice

II. Drawing the circuit

A. Getting the Parts
B. Placing the Parts
C. Connecting the Circuit
D. Changing the Name of the Part
E. Changing the Value of the Part
F. Making Sure You Have a GND
G. Voltage and Current Bubbles

III. Voltage Sources

A. VDC
B. VAC
C. VSIN
D. VPULSE

IV. Analysis Menu

A. DC Sweep
B. Bias Point Detail
C. Transient

V. Probe

A. Before you do the Probe
B. To Start the Probe
C. Graphing
D. Adding/Deleting Traces
E. Finding Points

VI. Measuring DC Analysis

I. Opening PSpice

Find PSpice on the C-Drive. Open Schematics or you can go to PSpice A_D and then click on the schematic icon  .

You will see the window as shown in Figure 1.

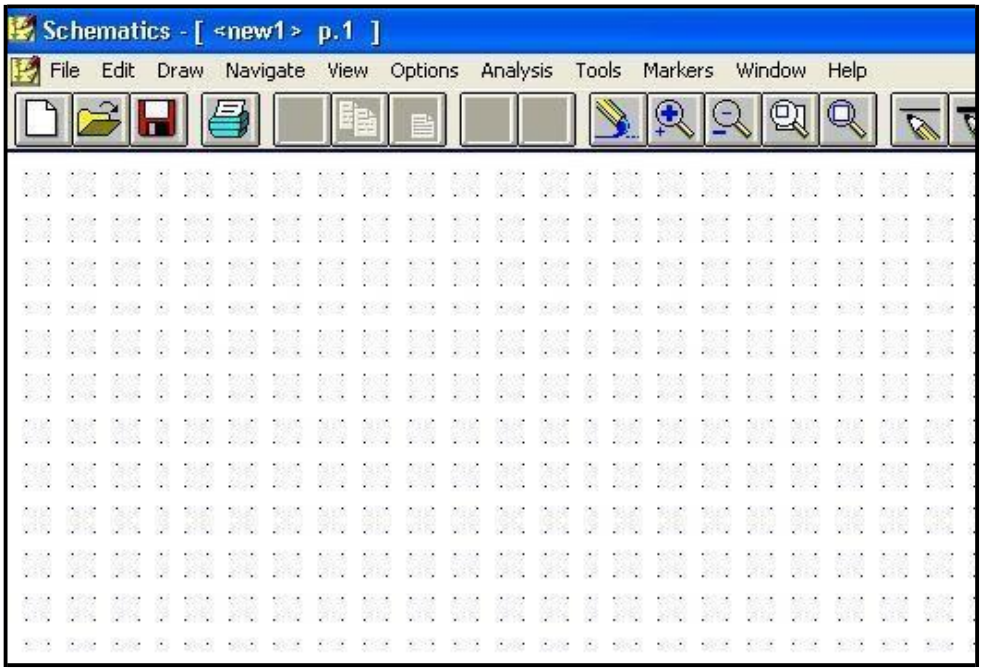


Figure 1

II. Drawing the circuit

A. Getting the Parts:

The first thing that you have to do is get some or all of the parts you need.
This can be done by

- Clicking on the 'get new parts' button  , or
- Pressing "Control+G", or
- Going to "Draw" and selecting "Get New Part..."

Once this box is open, select a part that you want in your circuit. This can be done by typing in the name (part name) or scrolling down the list until you find it.

Figure 2

An important prerequisite to building a schematic is the availability of the necessary parts (in the form of symbols) for assembly. *Schematics* have an extensive symbol libraries and a fully integrated symbol editor for creating your own symbols or modifying existing symbols. For the labs you will be using the existing symbols.

- Some common parts are:
- R - resistor
- C - capacitor
- L - inductor
- D - diode
- GND_ANALOG or GND_EARTH -- this is very important, you MUST have a ground in your circuit
- VAC and VDC
- Q2N – bipolar transistor
- VSIN –Transient sine voltage source

Upon selecting your part (you will also see description of the part below part name and you can see the symbol of that part when you click on ***advanced*** in the above figure), click on the place button (you will see the part attached to the mouse pointer) then click where you want it placed (somewhere on the white page with the blue dots), if you need multiple instances of this part click again, once you have selected that part right click your mouse the part will not be attached to the mouse pointer. Don't worry about putting it in exactly the right place, it can always be moved later.

If you want to take a part and close then you just select the part and click on ***place & close***.

Once you have all the parts you think you need, close that box. You can always open it again later if you need more or different parts. (The parts you have selected will be listed on the menu bar for quick access)

B. Placing the Parts

You should have most of the parts that you need at this point.

Now, all you do is put them in the places that make the most sense (usually a rectangle works well for simple circuits). Just select the part (It will become Red) and drag it where you want it.

To rotate parts so that they will fit in you circuit nicely, click on the part and press "Ctrl+R" (or Edit "Rotate"). To flip them, press "Ctrl+F" (or Edit "Flip").

If you have any parts left over, just select them and press "Delete".

C. Connecting the Circuit

Now that your parts are arranged well, you'll have to attach them with wires.

Go up to the tool bar and

- Select "Draw Wire"  or
- "Ctrl+W" or
- Go to "Draw" and select "Wire".

With the pencil looking pointer, click on one end of a part, when you move your mouse around, you should see dotted lines appear. Attach the other end of your wire to the next part in the circuit.

Repeat this until your circuit is completely wired.

If you want to make a node (to make a wire go more than one place), click somewhere on the wire and then click to the part (or the other wire). Or you can go from the part to the wire.

To get rid of the pencil, right click.

If you end up with extra dots near your parts, you probably have an extra wire, select this short wire (it will turn red), then press "Delete".

If the wire doesn't go the way you want (it doesn't look the way you want), you can make extra bends in it by clicking in different places on the way (each click will form a corner).

D. Changing the Name of the Part

You probably don't want to keep the names C1, C2 etc., especially if you didn't put the parts in the most logical order. To change the name, double click on the present name (C1, or R1 or whatever your part is), and then a box will pop up (Edit Reference Designator) see Figure 3. In the top window, you can type in the name you want the part to have.

Figure 3

Note that if you double click on the part or its value, a different box will appear.

E. Changing the Value of the Part

If you only want to change the value of the part (if you don't want all your resistors to be 1K ohms), you can double click on the present value and a box called "Set Attribute Value" will appear see Figure 4. Type in the new value and press OK. Use u for micro as in uF = microFarad.

Figure 4

F. Making Sure You Have a GND

This is very important. You cannot do any simulation on the circuit if you don't have a ground. If you aren't sure where to put it, place it near the negative side of your voltage source.

G. Voltage and Current Bubbles

These are important if you want to measure the voltage at a point or the current going through that point.

- To add voltage or current bubbles, go to the right side of the top tool bar and select "Voltage/Level Marker" (Ctrl+M)  or "Current Marker"  . To get either of these, go to "Markers" and either "Voltage/Level Marker" or "Current Marker".

III. Voltage Sources

A. VDC

This is your basic direct current voltage source that simulates a simple battery and allows you to specify the voltage value.

B. VAC

A few things to note about the alternating current source, first PSpice takes it to be a sine source, so if you want to simulate a cosine wave you need to add (or subtract) a 90° phase shift. There are three values which PSpice will allow you to alter, these being:

- **ACMAG** which is the RMS value of the voltage.
- **DC** which is the DC offset voltage.
- **ACPHASE** which is the phase angle of the voltage.

Note that the phase angle if left unspecified will be set by default to 0°.

C. VSIN

The SIN type of source is actually a damped sine with time delay, phase shift and a DC offset (see Figure 7). If you want to run a transient analysis you need to use the VSIN see how AC will effect your circuit over time. Do not use this type of source for a phasor or frequency sweep analysis, VAC would be appropriate for that.

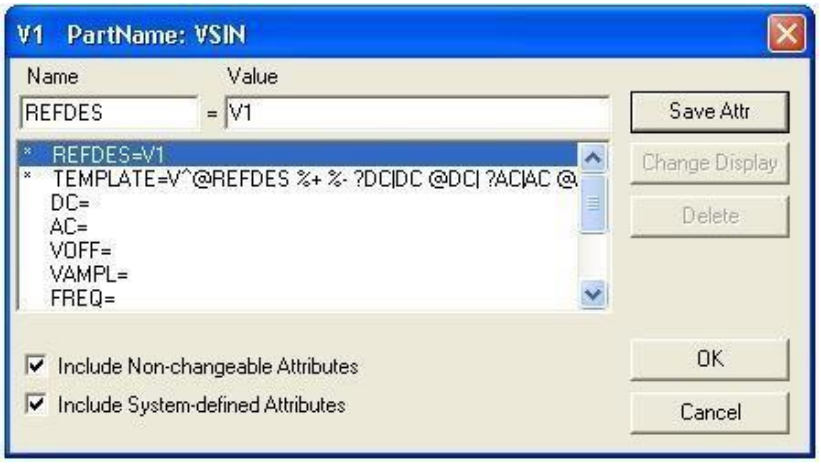


Figure 7

- **DC** the DC component of the sine wave.
- **AC** the AC value of the sine wave.
- **VOFF** is the DC offset value. It should be set to zero if you need a pure sinusoid.
- **Vamplitude** is the undamped amplitude of the sinusoid; i.e., the peak value measured from zero if there were no DC offset value.
- **FREQ** is the frequency in Hz of the sinusoid.
- **TD** is the time delay in seconds. Set this to zero for the normal sinusoid.
- **DF** is the damping factor. Also set this to zero for the normal sinusoid.
- **PHASE** is the phase advance in degrees. Set this to 90 if you need a cosine wave form.

Note that the normal usage of this source type is to set **VOFF**, **TD** and **DF** to zero as this will give you a 'nice' sine wave.

D. VPULSE

The VPULSE is often used for a transient simulation of a circuit where we want to make it act like a square wave source. It should never be used in a frequency response study because PSpice assumes it is in the time domain, and therefore your probe plot will give you inaccurate results. Details of VPULSE are (see Figure 8):

- **DC** the DC component of the wave.
- **AC** the AC component of the wave.
- **V1** is the value when the pulse is not "on." So for a square wave, the value when the wave is 'low'. This can be zero or negative as required. For a pulsed current source, the units would be "amps" instead of "volts."
- **V2** is the value when the pulse is fully turned 'on'. This can also be zero or negative. (Obviously, V1 and V2 should not be equal.) Again, the units would be "amps" if this were a current pulse.
- **TD** is the time delay. The default units are seconds. The time delay may be zero, but not negative.
- **TR** is the rise time of the pulse. PSpice allows this value to be zero, but zero rise time may cause convergence problems in some transient analysis simulations. The default units are seconds.
- **TF** is the fall time in seconds of the pulse.
- **TW** is the pulse width. This is the time in seconds that the pulse is fully on.
- **PER** is the period and is the total time in seconds of the pulse.

This is a very important source for us because we do a lot of work on with the square wave on the wave generator to see how various components and circuits respond to it.

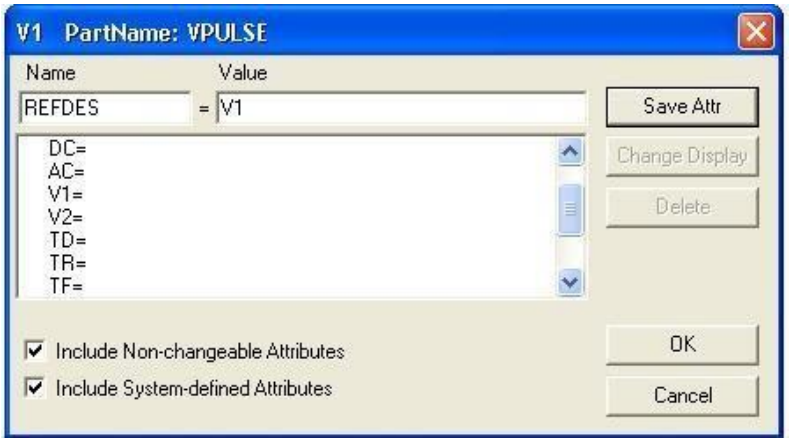


Figure 8

IV. Analysis Menu

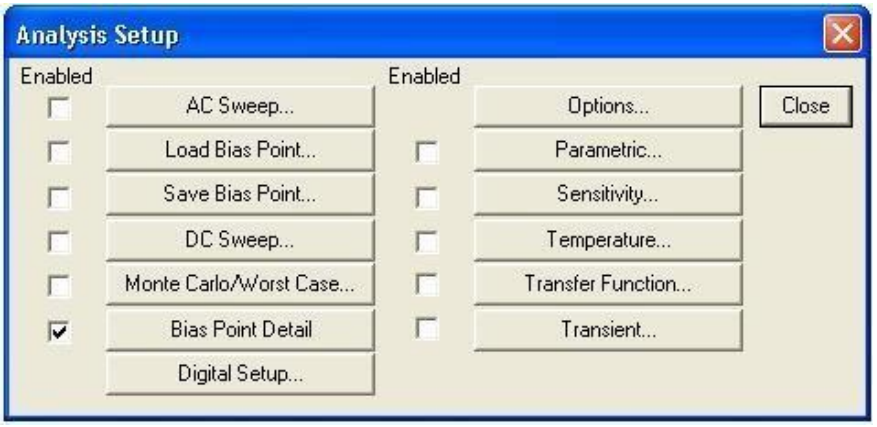


Figure 5

To open the analysis menu click on the  button.

A. DC Sweep

The DC sweep allows you to do various different sweeps of your circuit to see how it responds to various conditions.

For all the possible sweeps,

- o voltage,
- o current,
- o temperature, and
- o parameter and global

You need to specify a start value, an end value, and the number of points you wish to calculate.

For example you can sweep your circuit over a voltage range from 0 to 12 volts. The main two sweeps that will be most important to us at this stage are the voltage sweep and the current sweep. For these two, you need to indicate to PSpice what component you wish to sweep, for example V1 or V2.

Another excellent feature of the DC sweep in PSpice, is the ability to do a nested sweep.

A nested sweep allows you to run two simultaneous sweeps to see how changes in two different DC sources will affect your circuit.

Once you've filled in the main sweep menu, click on the nested sweep button and choose the second type of source to sweep and name it, also specifying the start and end values. (Note: In some versions of PSpice you need to click on **enable nested sweep**). Again you can choose Linear, Octave or Decade, but also you can indicate your own list of values, example: 1V 10V 20V. **DO NOT** separate the values with commas.

B. Bias Point Detail

This is a simple, but incredibly useful sweep. It will not launch Probe and so give you nothing to plot. But by clicking on **enable bias current display** or **enable bias voltage display**, this will indicate the voltage and current at certain points within the circuit.

C. Transient

The transient analysis is probably the most important analysis you can run in PSpice, and it computes various values of your circuit over time

Choose **Analysis…Setup** from the menu bar, or click on the Setup Analysis button  in the toolbar. The Analysis Setup dialog box opens.

Click on the **Transient** button in the Analysis Setup dialog box. The Transient dialog box opens.

Two very important parameters in the transient analysis are (see Figure 6):

- o **Print step.**
- o **Final time**.

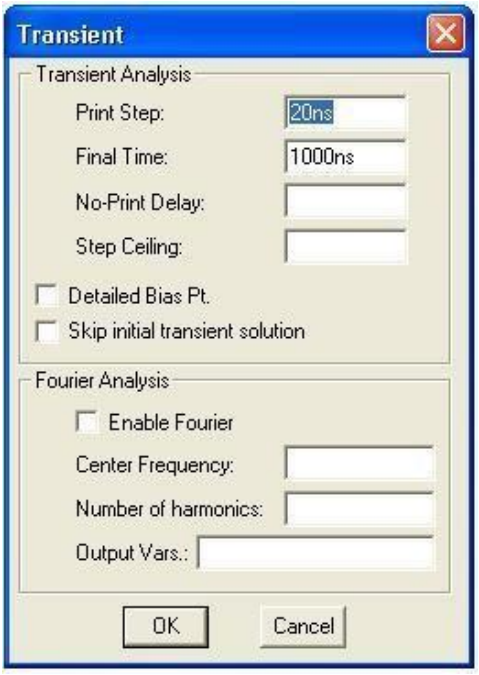


Figure 6

The ratio of **final time: print step** (Keep print step atleast $1/100^{th}$ of the final time) determines how many calculations PSpice must make to plot a wave form. PSpice always defaults the start time to zero seconds and going until it reaches the user defined final time. It is incredibly important that you think about what print step you should use before running the simulation, if you make the print step too small the probe screen will be cluttered with unnecessary points making it hard to read, and taking extreme amounts of time for PSpice to calculate. However, at the opposite side of that coin is the problem that if you set the print step too high you might miss important phenomenon that are occurring over very short periods of time in the circuit. Therefore play with step time to see what works best for your circuit.

You can set a step ceiling which will limit the size of each interval, thus increasing calculation speed. Another handy feature is the Fourier analysis, which allows you to specify your fundamental frequency and the number of harmonics you wish to see on the plot. PSpice defaults to the 9th harmonic unless you specify otherwise, but this still will allow you to decompose a square wave to see it's components with sufficient detail.

V. Probe

A. Before you do the Probe

You have to have your circuit properly drawn and saved.
There must not be any floating parts on your page (i.e. unattached devices).
You should make sure that all parts have the values that you want.
There are no extra wires.
It is very important that you have a ground on your circuit.

Make sure that you have done the Analysis Setup and that only the things you want are enabled.

B. To Start the Probe:

Click on the Simulate button on the tool bar  (or Analysis, Simulate, or F11).

It will check to make sure you don't have any errors. If you do have errors, correct them.

Then a new window will pop up. Here is where you can do your graphs.

C. Graphing:

If you don't have any errors, you should get a window with a black background to pop up.

If you did have errors, in the bottom, left hand side, it will say what your errors were (these may be difficult to understand, so go To "View - Output File").

D. Adding/Deleting Traces:

PSpice will automatically put some traces in. You will probably want to change them.

Go to Trace - Add Trace or  on the toolbar. Then select all the traces you want.

To delete traces, select them on the bottom of the graph and push Delete.

E. Finding Points:

There are Cursor buttons that allow you to find the maximum or minimum or just a point on the line. These are located on the toolbar (to the right).

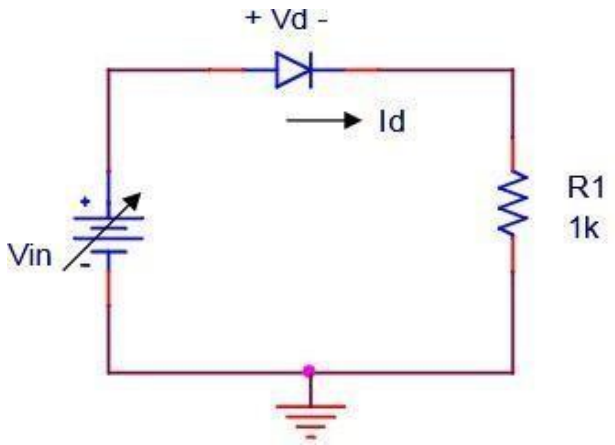
Select which curve you want to look at and then select "Toggle Cursor" . 

Then you can find the max, min, the slope, or the relative max or min ( is find relative max).

VI. Measuring DC Analysis

If you want to measure DC levels you can use two parts to view these levels. These parts are placed on the schematic drawing the same way any other part is placed. VIEWPOINT is a voltage viewing point, which will show the value after the circuit is simulated. You place VIEWPOINT on a node. IPROBE is a current probe, which will show the value after the circuit is simulated. You need to put this part between two parts, so that current flowing in that branch can be measured. If you have measurements that are time-varying (i.e. a sinusoid) then you need to run *Probe*.

CIRCUIT DIAGRAM:



PARTS TO BE USED IN THE PSPICE:

Part Name	Pspice Name	Quantities
Resister	R	1
Diode	D1N4148	1
DC source	VDC	1
Electrical Ground	EGND	1

Ex. No.: 1 (a)

Date:

SIMULATION OF PN JUNCTION DIODE

AIM:

To simulate PN junction Diode and Obtain VI Characteristics in PSPICE – Software.

APPARATUS REQUIRED:

A PC with PSPICE package.

THEORY:

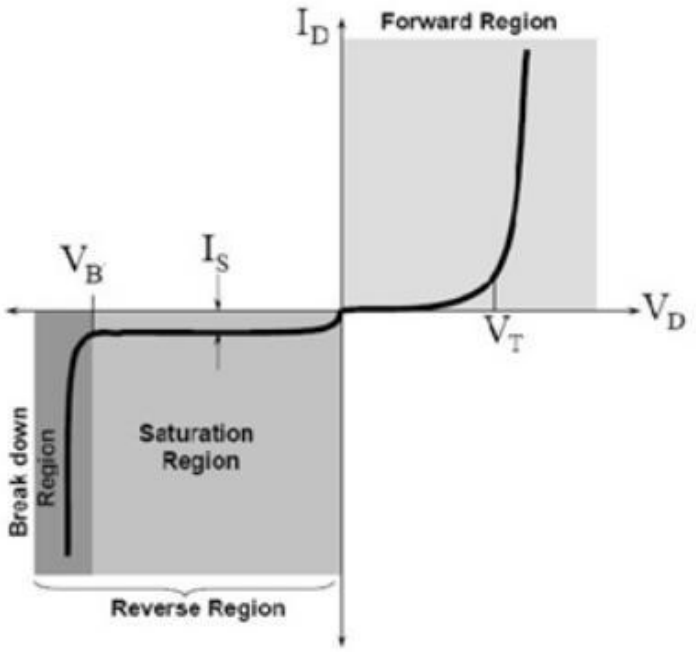
A diode is a PN junction formed by a layer of P type and layer of N type Semiconductors. Once formed the free electrons in the N region diffuse across the junction and combine with holes in P region and so a depletion Layer is developed. The depletion layer consists of ions, which acts like a barrier for diffusion of charged beyond a certain limit. The difference of potential across the depletion layer is called the barrier potential. At 2.5degree the barrier potential approximately equal 0.7v for silicon diode and 0.3v for germanium diode.

When the junction is forward bias, the majority carrier acquired sufficient energy to overcome the barrier and the diode conducts. When the junction is reverse biased the depletion layer widens and the barrier potential increases. Hence the Majority carrier cannot cross the junction and the diode does not conduct. But there will be a leakage current due to minority carrier. When diode is forward biased, resistance offered is zero, and when reverse biased resistance offered is infinity. It acts as a perfect switch.

PROCEDURE:

1. Click on *Start →Program →MicroSim Eval 6.3 →Schematic*
2. Open the *Draw* menu by clicking once on the Draw menu. Choose *Get New Part*, and then *Browse*.
3. Get part DC battery VDC from the source.slb library
4. Get part resistance R from the analog.slb library
5. Get part diode D1N4002 from the eval.slb library.
6. Get part earth ground AGND from the port.slb library.
7. To rotate the part first select it, then press Ctrl-R.
8. Draw and complete the diode circuit shown in Figure. You can click the left mouse on the device or element and choose Attributes from the Edit menu. Alternatively, you can change the attributes of any devices or elements by double clicking the left mouse and giving new values.
9. Analyze the circuit of Figure by choosing *Analysis* from Schematic menu.
   Click once on the *Analysis* menu and then choose *Setup* menu.
   Choose the analysis type – *DC Sweep* and give the sweep information: sweep name VDD, start value – 0V, Sweep end value – 0.8V, and Sweep increment – 0.01.
   Run the simulation by choosing *Simulate* from the Analysis menu.
   After successful simulation, PSPICE will automatically run Probe and move to Probe menu. Choose *Add* from the *Trace* menu of Probe and select the plot variable, the diode current. e.g. I(D1).

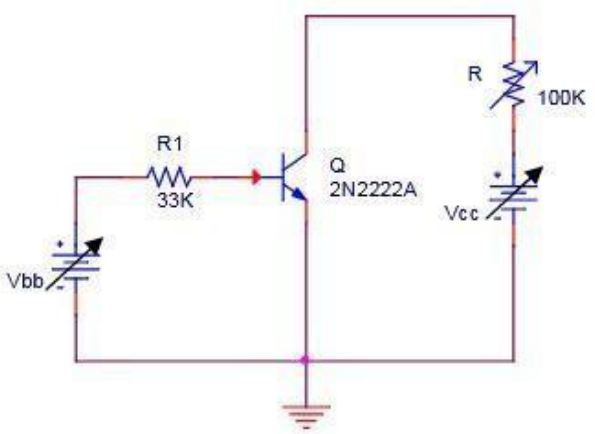
V-I CHARACTERISTICS:



RESULT:

Thus the simulation of PN Junction Diode model is done and V-I Characteristics is obtained using PSPICE Software.

CIRCUIT DIAGRAM:



PARTS TO BE USED IN THE PSPICE:

Part Name	Pspice Name	Quantities
Resister	R	2
NPN-BJT	2N2222A	1
DC source	VDC	2
Electrical Ground	EGND	1

Ex. No.: 1 (b)

Date:

SIMULATION OF TRANSISTOR COMMON EMITTER CONFIGURATION

AIM:

To simulate transistor common emitter configuration and Obtain input / output Characteristics in PSPICE – Software.

APPARATUS REQUIRED:

A PC with PSPICE package.

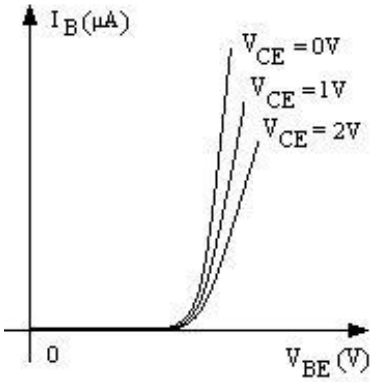
THEORY:

A Bipolar Junction Transistor or BJT is a three terminal device having two PN-junctions connected together in series. Each terminal is given a name to identify it and these are known as the Emitter (E), Base (B) and Collector (C). There are two basic types of bipolar transistor construction, NPN and PNP, which basically describes the physical arrangement of the P-type and N-type semiconductor materials from which they are made. Bipolar Transistors are "CURRENT" Amplifying or current regulating devices that control the amount of current flowing through them in proportion to the amount of biasing current applied to their base terminal. The principle of operation of the two transistors types NPN and PNP, is exactly the same the only difference being in the biasing (base current) and the polarity of the power supply for each type. In CE configuration, Emitter is common to both the input and output as shown in figure. The direction of the arrow in the symbol shows current flow between the base and emitter terminal, pointing from the positive P-type region to the negative N-type region, exactly the same as for the standard diode symbol. For normal operation, the emitter-base junction is forward- biased and the collector-base junction is reverse-biased.

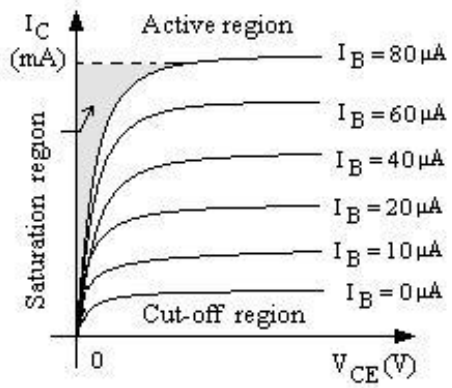
PROCEDURE:

1. Click on *Start →Program →MicroSim Eval 6.3 →Schematic*
2. Open the *Draw* menu by clicking once on the Draw menu. Choose *Get New Part*, and then *Browse*.
3. Get part DC battery VDC from the source.slb library
4. Get part resistance R from the analog.slb library
5. Get part NPN-BJT 2N2222 from the eval.slb library.
6. Get part earth ground AGND from the port.slb library.
7. To rotate the part first select it, then press Ctrl-R.
8. Draw and complete the circuit shown in Figure. You can click the left mouse on the device or element and choose Attributes from the Edit menu. Alternatively, you can change the attributes of any devices or elements by double clicking the left mouse and giving new values.
9. Analyze the circuit of Figure by choosing *Analysis* from Schematic menu.
   Click once on the *Analysis* menu and then choose *Setup* menu.
   Choose the analysis type – *DC Sweep* and give the sweep information: sweep name VDD, start value – 0V, Sweep end value – 0.8V, and Sweep increment – 0.01.
   Run the simulation by choosing *Simulate* from the Analysis menu.
   After successful simulation, PSPICE will automatically run Probe and move to Probe menu. Choose *Add* from the *Trace* menu of Probe and select the plot variable, the diode current. e.g. I(D1).

CHARACTERISTICS:



CE I/P Characteristics

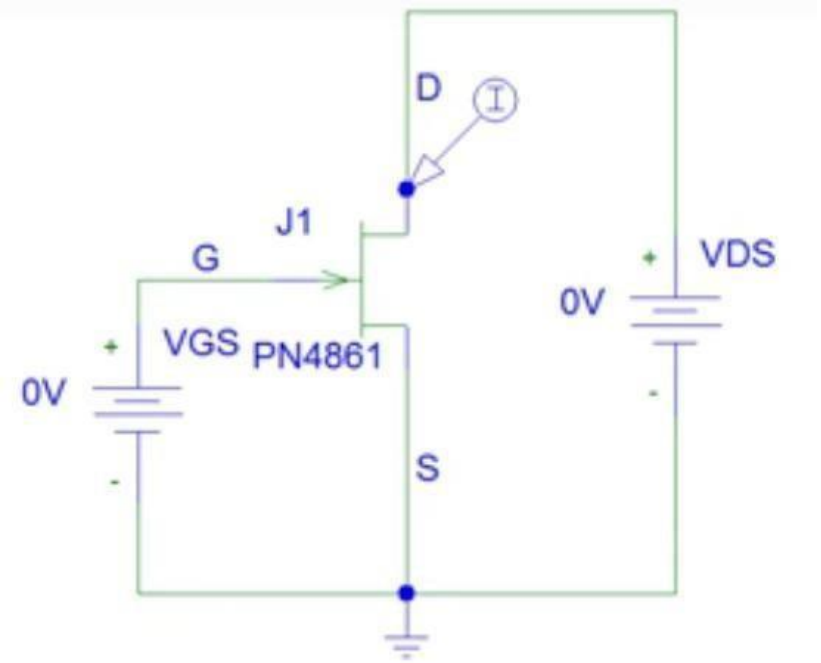


CE O/P Characteristics

RESULT:

Thus the simulation of transistor common emitter configuration is done and the input / output Characteristics is obtained using PSPICE Software.

CIRCUIT DIAGRAM:



PARTS TO BE USED IN THE PSPICE:

Part Name	Pspice Name	Quantities
JFET	PN4861	1
DC source	VDC	2
Electrical Ground	EGND	1

Ex. No.: 1 (c)

Date:

SIMULATION OF JFET

AIM:

To simulate JFET and obtain Drain and transfer Characteristics in PSPICE – Software.

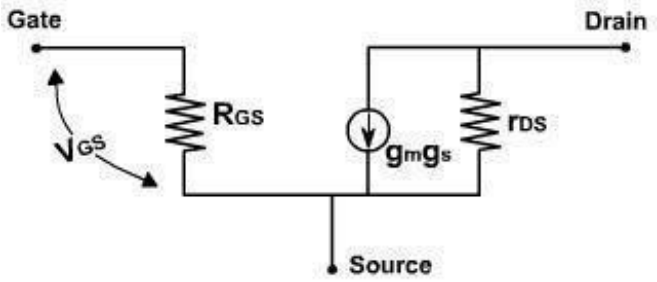
APPARATUS REQUIRED:

A PC with PSPICE package.

THEORY:

The Field Effect Transistor (FET) is a three terminal device. Three terminals are Drain (D), Source (S), Gate (G) and fourth terminal is substrate/body/shield/Bulk but it is not used. In FET, current flow is due to only one type of charge particles, either electrons or holes. So FET is known as unipolar device. The name "field effect" is derived from the fact that the current is controlled by an electric field set up in the device by an externally applied voltage. Thus FET is a voltage controlled device while bipolar transistor is current controlled device.

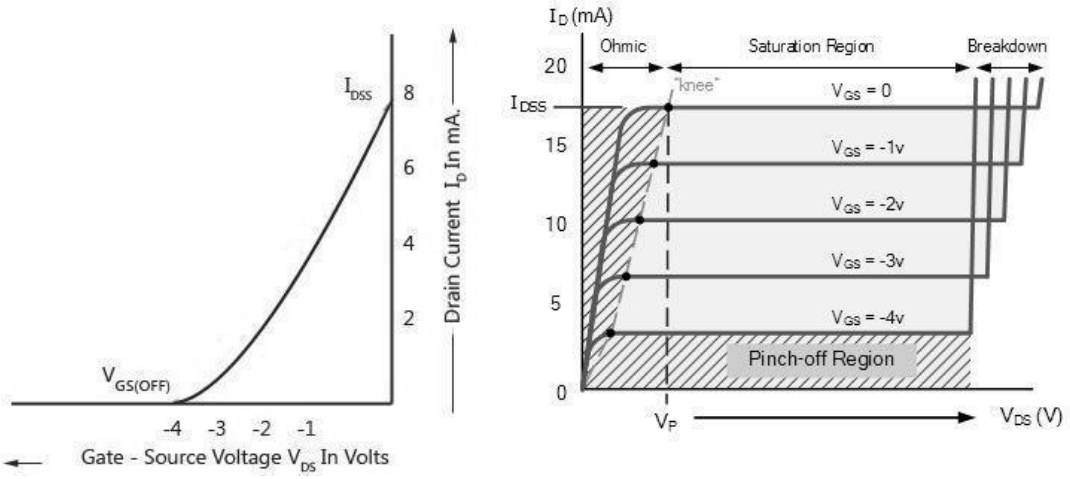
JFET AC Equivalent Circuit:



PROCEDURE:

1. Click on *Start →Program →MicroSim Eval 6.3 →Schematic*
2. Open the *Draw* menu by clicking once on the Draw menu. Choose *Get New Part,* and then *Browse.*
3. Get part DC battery VDC from the source.slb library
4. Get part resistance R from the analog.slb library
5. Get part JFET PN4861 from the eval.slb library.
6. Get part earth ground AGND from the port.slb library.
7. To rotate the part first select it, then press Ctrl-R.
8. Draw and complete the circuit shown in Figure. You can click the left mouse on the device or element and choose Attributes from the Edit menu. Alternatively, you can change the attributes of any devices or elements by double clicking the left mouse and giving new values.
9. Analyze the circuit of Figure by choosing *Analysis* from Schematic menu.
   Click once on the *Analysis* menu and then choose *Setup* menu.
   Choose the analysis type – *DC Sweep* and give the sweep information: sweep name VDD, start value – 0V, Sweep end value – 0.8V, and Sweep increment – 0.01.
   Run the simulation by choosing *Simulate* from the Analysis menu.
   After successful simulation, PSPICE will automatically run Probe and move to Probe menu. Choose *Add* from the *Trace* menu of Probe and select the plot variable, the diode current. e.g. I(D1).

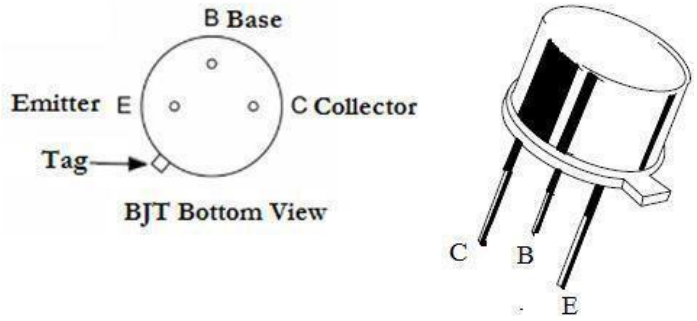
CHARACTERISTICS:



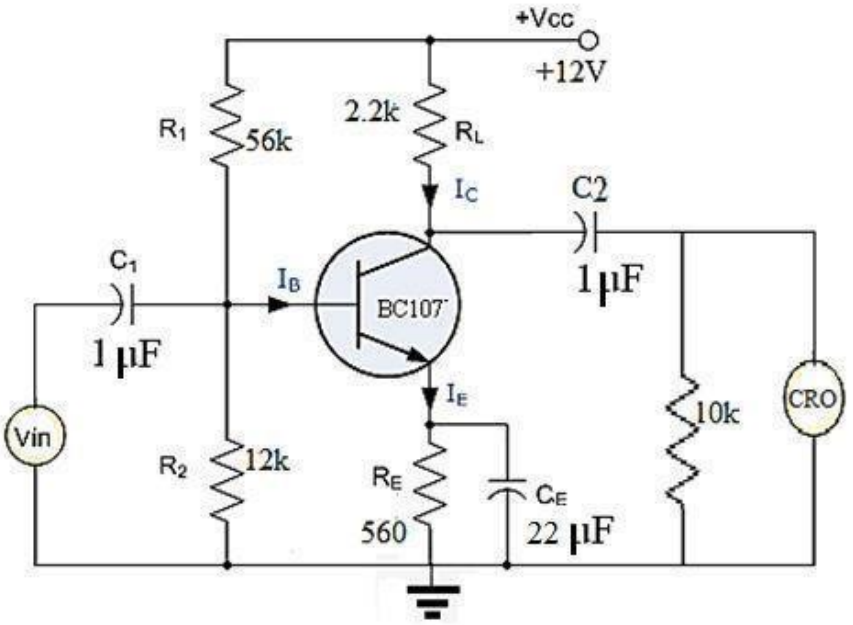
RESULT:

Thus the simulation of JFET is done and the Drain & Transfer Characteristics is obtained using PSPICE Software.

PIN DIAGRAM:



CIRCUIT DIAGRAM FOR CE AMPLIFIER:



Ex. No.: 2

Date:

FREQUENCY RESPONSE OF TRANSISTOR AMPLIFIER CIRCUIT

Ex. No: 2 (a)

FREQUENCY RESPONSE CHARACTERISTICS OF A COMMON EMITTER AMPLIFIER

AIM:

To design and construct BJT CE Amplifier using voltage divider bias and to obtain its frequency response

APPARATUS REQUIRED:

Sl. No	Components / Equipments	Specifications	Quantity
1	Transistor BC107	Max Rating: 50V 1A, 3W	1
2	Resistors	56kΩ,12kΩ,2.2kΩ, 10kΩ,560Ω	Each One
3	Capacitor	1μF, 22μF	2,1
4	Dual Regulated Power Supply	(0-30) V	1
5	CRO	(0-30) MHz	1
6	Function Generator	(0 – 1) MHz	1
7	Bread Board		1
8	Connecting Wires		Few
9	BNC CRO Probe		2

THEORY:

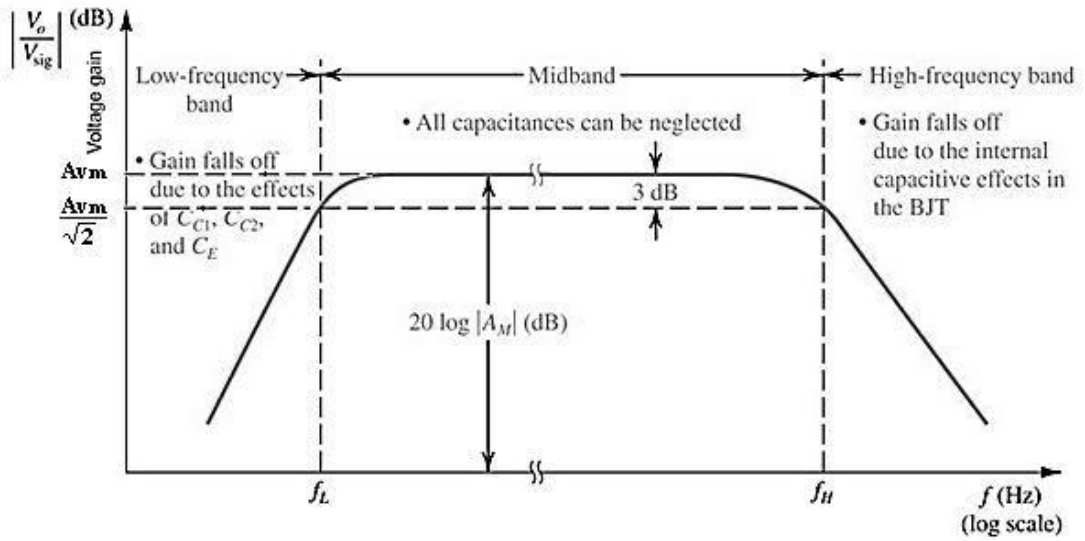
Common Emitter amplifier has the emitter terminal as the common terminal between input and output terminals. The emitter base junction is forward biased and collector base junction is reverse biased, so that transistor remains in active region throughout the operation. When a sinusoidal AC signal is applied at input terminals of circuit during positive half cycle the forward bias of base emitter junction V_{BE} is increased resulting in an increase in I_B, The collector current Ic is increased by β times the increase in I_B, V_{CE} is correspondingly decreased. i.e. output voltage gets decreased. Thus in a CE amplifier a positive going signal is converted into a negative going output signal i.e. 180° phase shift is introduced between output and input signal and it is an amplified version of input signal.

TABULATION:

Input Voltage (V_i) = 1V

Frequency (Hz)	O/P Voltage, Vo (V)	Voltage Gain Av =Vo/Vi	Av in dB =20 log(Av)

MODEL GRAPH:



DESIGN:

Vcc =12 V, Ic = 2mA, R_B = 10kΩ, hfe =1 00, hie =2kΩ

Vce = Vcc /2 = 12/2 = 6V

V_E = Vcc /10 =12/10 =1.2 V $I_E \approx$

Ic = 2mA

$R_E = V_E / I_E$ = 1.cV /2mA = 600Ω

Choose, R_E = 560Ω

$V_{cc} = I_c R_c + V_{ce} + I_E R_E$

$R_c = (V_{cc} – V_{ce} – I_E R_E) / I_c$

Rc = (12 – 6 – 1.2) / 2mA Rc

= 2.4k Ω

Choose, Rc = 2.7k Ω

$V_{BE} = V_B – V_E$

$V_B = V_{BE} + V_E$

V_B = 0.7 + 1.2 = 1.9V

$V_B = V_{cc} (R2/(R1+R2))$ =1.9V

R2 /(R1+R2) =1.9V / 12V = 0.158

R_B = 10k = R1R2/(R1+R2) =R1(0.158) R1

=63.157 kΩ

R2 /(R1+R2) = 0.158 => R2 = 11.58kΩ

Choose, R1 =56 kΩ and R2 = 12kΩ

$X_E = R_E/10$ =600/10 = 60Ω

$C_E = 1/(2\pi f X_E)$ = 26.5μF (for f=1kHz),

Choose Coupling capacitors C1 =C2 =1μF

PROCEDURE:

1. Connect the circuit as per the circuit diagram
2. Set V_{in} = 2V in the signal generator. Keeping input voltage constant, vary the frequency from 1Hz to 3MHzin regular steps.
3. Note down the corresponding output voltage.
4. Plot the graph: Gain in dB Vs Frequency in Hz.
5. Calculate the Bandwidth from the Frequency response graph.

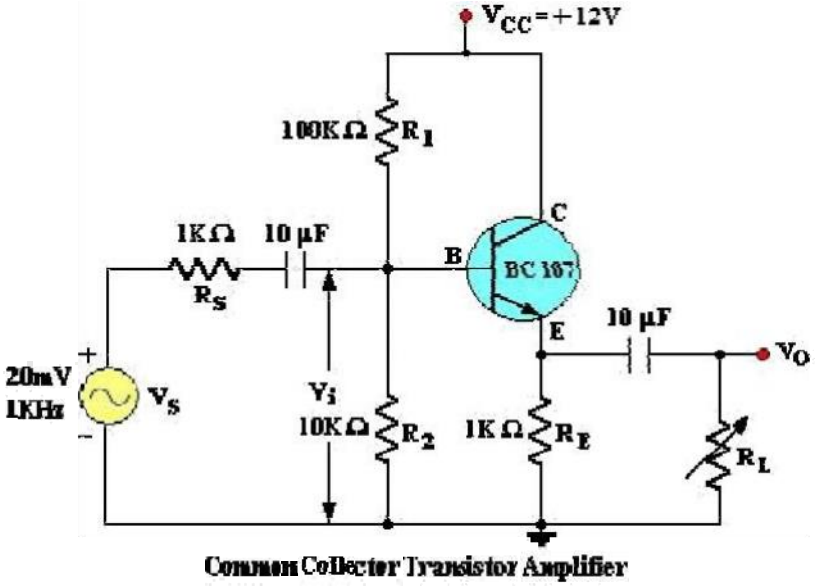
REVIEW QUESTIONS:

1. What are the operating modes of BJT with reference to junction biasing?
2. Why CE configuration is preferred over CB configuration?
3. Write some applications of CE amplifier.
4. What will be the input and output impedance of CE amplifier?
5. What is the voltage and current gain of CE amplifier?

RESULT:

Thus a BJT CE Amplifier with voltage divider bias was designed and plotted the frequency response curve.

CIRCUIT DIAGRAM FOR CC AMPLIFIER



Common Collector Transistor Amplifier

Ex. No: 2(b)
Date:

FREQUENCY RESPONSE CHARACTERISTICS OF A COMMON COLLECTOR AMPLIFIER

AIM:

To find the frequency response of a Common Collector Transistor Amplifier and to find the Bandwidth from the Response, Voltage gain, Input Resistance, output resistance.

APPARATUS REQUIRED:

Sl. No	Components / Equipments	Specifications	Quantity
1	Transistor BC107	Max Rating : 50V 1A, 3W	1
2	Resistors	100kΩ,10kΩ,1kΩ,	1,1,2
3	Capacitor	10μF	2
4	Dual Regulated Power Supply	(0-30)V	1
5	CRO	(0-30)MHz	1
6	Function Generator	(0 – 1)MHz	1
7	Bread Board		1
8	Connecting Wires		Few
9	BNC CRO Probe		2

THEORY:

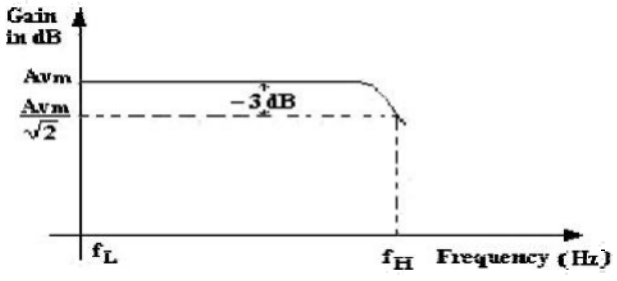
Common Emitter amplifier has the emitter terminal as the common terminal between input and output terminals. The emitter base junction is forward biased and collector base junction is reverse biased, so that transistor remains in active region throughout the operation. When a sinusoidal AC signal is applied at input terminals of circuit during positive half cycle the forward bias of base emitter junction V BE is increased resulting in an increase in IB, The collector current Ic is increased by β times the increase in IB, VCE is correspondingly decreased. i.e. output voltage gets decreased. Thus in a CE amplifier a positive going signal is converted into a negative going output signal i.e. 180° phase shift is introduced between output and input signal and it is an amplified version of input signal.

TABULATION:

Vi=1Volt

Frequency (Hz)	O/P Voltage, Vo (V)	Voltage Gain Av =Vo/Vi	Av in dB =20 log(Av)

MODEL GRAPH:



PROCEDURE:

1. Connect the circuit as per the Fig., Apply Vcc of 12 Volts DC.
2. Apply I/P Voltage of 20mV at 1KHz from the Signal Generator and observe the O/P on CRO.
3. Vary the frequency from 100 Hz to 1MHz in appropriate steps and note down the corresponding O/P Voltage Vo in a tabular form .
4. Calculate the Voltage Gain Av = Vo/Vi and note down in the tabular form.
5. Plot the frequency (f) Vs Gain (Av) on a semi-log Graph sheet
6. Draw a horizontal line at 0.707 times Av and note down the cut off points and the Bandwidth is given by B.W = $f_2 - f_1$.

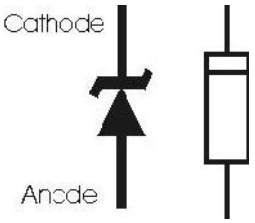
REVIEW QUESTIONS:

1. What is the other name for CC Amplifier?
2. What are the uses of CC Amplifier?
3. Why this amplifier has got the name Emitter Follower?
4. What is the maximum Voltage gain of an Emitter Follower?
5. Why it is used as a Buffer amplifier?

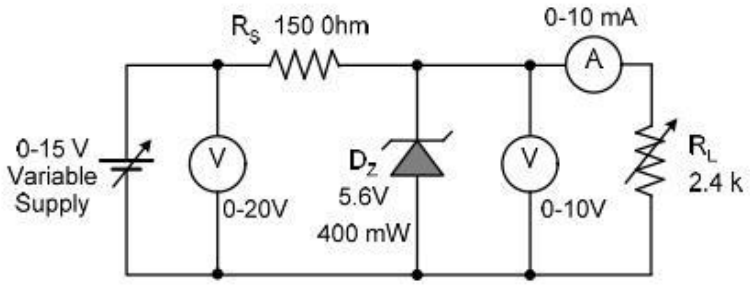
RESULT:

Thus a BJT CC Amplifier with voltage divider bias was designed and plotted the frequency response curve.

SYMBOL & PIN DIAGRAM:



CIRCUIT DIAGRAM:



Ex. No.: 3

Date:

LINE AND LOAD REGULATION OF ZENER REGULATOR

AIM:

To study a zener diode shunt regulator and to plot its line and load regulation Characteristics.

APPARATUS REQUIRED:

Sl. No	Components / Equipments	Specifications	Quantity
1	Diode- Zener	FZ3.2, 5.6V, 400mW	1
2	Resistors	150Ω, 2.4kΩ	Each One
3	Dual Regulated Power Supply	(0-30)V	1
4	Voltmeters	MC (0-10)V, (0-20)V	Each One
5	Ammeters	MC (0-10)mA	1
6	Bread Board	-	1
7	Connecting Wires		Few

THEORY:

A zener diode functions as an ordinary diode when it is forward biased. It is a specially designed device to operate in the reverse bias. When it is in the reverse breakdown region, the zener voltage V_Z remains almost constant irrespective of the current I_Z through it. A series resistor R_S is used to limit the zener current below its maximum current rating. The current through R_Z is given by the expression $I_S = I_Z + I_L$, where I_L is the current through the load resistor R_L. The value of R_S must be properly selected to fulfil the following condition requirements.

When the input voltage, V_I increases I_L remains the same, I_S and I_Z increases. Similarly if input voltage decreases, I_L remains the same, I_S and I_Z decreases. But if I_Z falls lower than the minimum zener current enough to keep the zener in the breakdown region, the regulation will cease and output voltage decreases. A low input voltage can cause the regulator fail to regulate. The series resistance should be selected between R_{Smax} and R_{Smin} which are given by the expressions,

$$R_{smin} = \frac{V_{Imax} - V_Z}{I_{Zmax}}$$

$$R_{smax} = \frac{V_{Imin} - V_Z}{I_{Zmin}\, I_Z}$$

TABULAR COLOUMNS:

LINE REGULATION

Keeping load current constant at I_L= 5mA, The input voltage is varied from 8 V to 14V and corresponding observations are made.

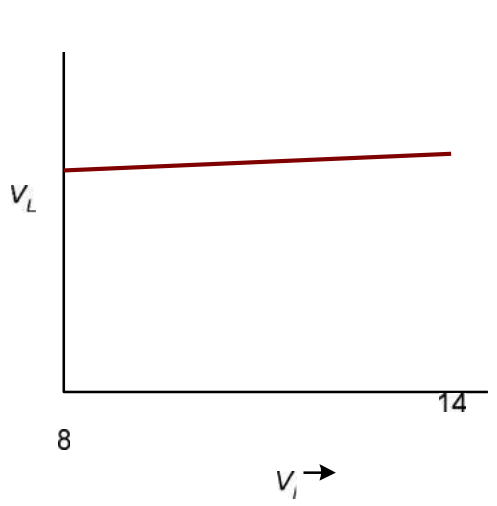
Vin (volts)	Vo (volts)

LOAD REGULATION

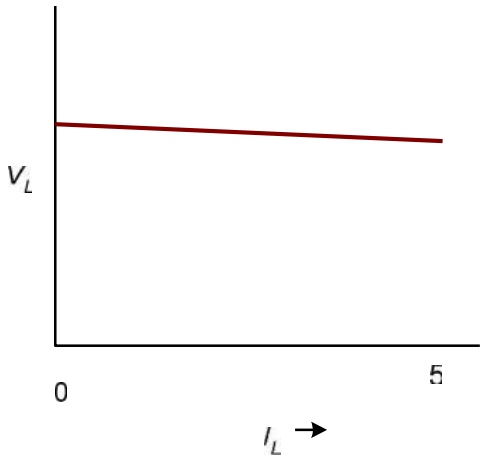
Keeping input voltage at 10V, the load current is varied from 0 to 5 mA and observations are made. For taking reading corresponding to no load (I_L = 0), the loading rheostat may be disconnected.

I_L(mA)	Vo(volts)

EXPECTED OUTPUT PLOTS



Line Regulation



Load Regulation

PROCEDURE:

1. Wire up the circuit on the bread board after testing all the components.
2. Keep the load constant. Note down the output voltage varying input from 8V to 14V in steps of 1V. Plot the line regulation graph with V_i along x-axis and Vo along y-axis. Calculate percentage line regulation using the expression $(\Delta V_o/\Delta V_i)$ x100%
3. Keep the input voltage constant (say 10V) and note down the output voltage for various values of load current starting from 0 to 5 mA, by varying R_L using a rheostat. Plot the load regulation graph with I_L along x-axis and Vo along y-axis.
4. To calculate percentage load regulation, mark V_{NL} and V_{FL} on y-axis on the load regulation graph. V_{NL} is the output voltage in the absence of load resistor and V_{FL} is the output voltage corresponding to rated I_L (here, 5 mA). Calculate the percentage load regulation V_R as per the equation,

$$V_R = \frac{V_{NL} - V_{FL}}{V_{NL}}$$

DESIGN:

Assume Vo=5.6V, I_{Lmax}=5mA input voltage in the range of 8-14V.

Select 5.6V Zener [Po = 400mW, V_Z=5.6V, r_d=8Ω at I_Z=10mA]

Use 2.4k rheostat as load resistance load current can be varied from 2.4mA and upwards.

$$I_{Zmax} = \frac{P_{max}}{V_Z} = \frac{0.4}{5.6} = 71.42mA$$

$$I_{Zmin} = 10\ \%\ of\ I_{zmax} = 0.1\ x\ 71.42 = 7.142mA$$

$$R_{smin} = \frac{V_{Imax} - V_Z}{I_{Zmax}} = \frac{14 - 5.6}{71.42mA} = 117.6$$

$$R_{smax} = \frac{V_{Imin} - V_Z}{I_{Zmin}\ I_Z}\ \frac{8 - 5.6}{7.142mA} = 197.6$$

Select Rs=150Ω

Power rating of Rs

Maximum current through Rs

$$I_m = \frac{V_{Imax} - V_Z}{R_S} = \frac{14 - 5.6}{150} = 56mA$$

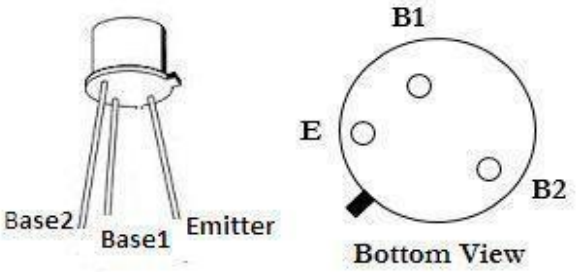
Power rating of Rs= $I_m^2 Rs$

$0.4704W$ >> Select 150Ω, 0.5W resistor

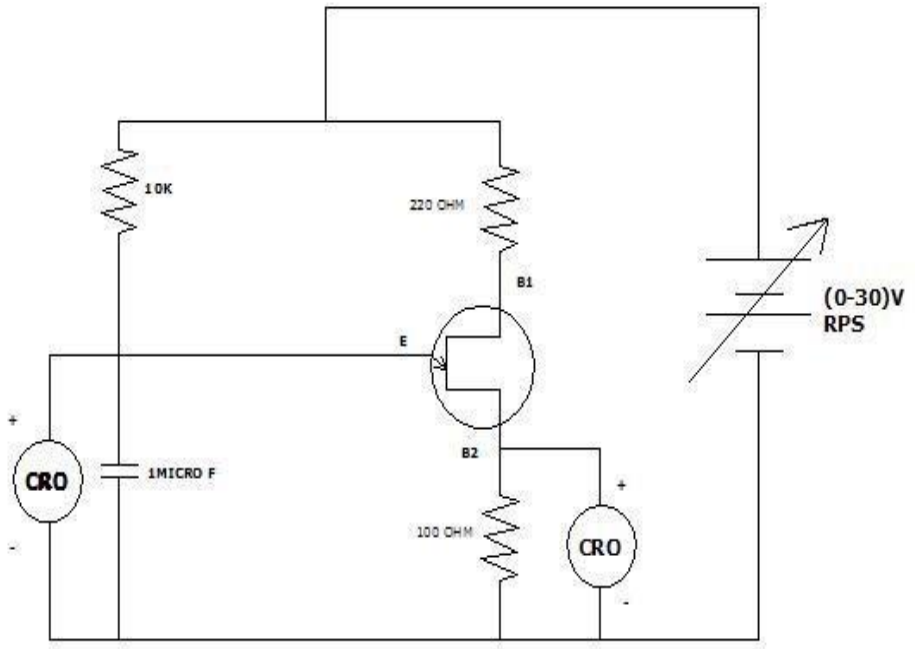
RESULT:

Thus a study a Zener diode shunt regulator and plotted the line and load regulation Characteristics curve.

PIN DIAGRAM



CIRCUIT DIAGRAM:



Ex. No.: 4

Date:

UJT RELAXATION OSCILLATOR

AIM:

To construct a relaxation oscillator using UJT & trace the wave form & also measure the frequency.

APPARATUS REQUIRED:

Sl. No	Apparatus Name	Range	Quantity
1	RPS	(0-30V)	1
2	CRO	(0-20 MHZ)	1
3	UJT	2N 2646	1
4	Capacitor	1µF	1
5	Resistor	DRB 10K 100K 220K	1 1 1 1
6	Bread board	-	1
7	Connecting Wires	-	Few

THEORY:

When the battery VBB is turned on, the capacitor C charges through resistor R1. During the charging period, the voltage across the capacitor rises in an exponential manner until it reaches the peak point voltage. Now, the UJT goes to conducting mode and the capacitor is discharged between E and B1. As the capacitor voltage reached the valley point (VV), the UJT is switched off. The next cycle then begins, allowing the capacitor C to charge again.

PROCEDURE:

1. Rig up the circuit as per the circuit diagram.
2. Set VBB = 10 V (say) (Supply voltage normally used lies in the range 10 to 35 V)
3. Connect CRO across capacitor.
4. Switch on the power supply set VBB = 10 V & observe the wave form on the CRO
5. Note down the practical frequency.
6. Switch off the supply
7. Connect CRO across R1 switch on the supply & observe the wave forms.
8. Note down the practical frequency.

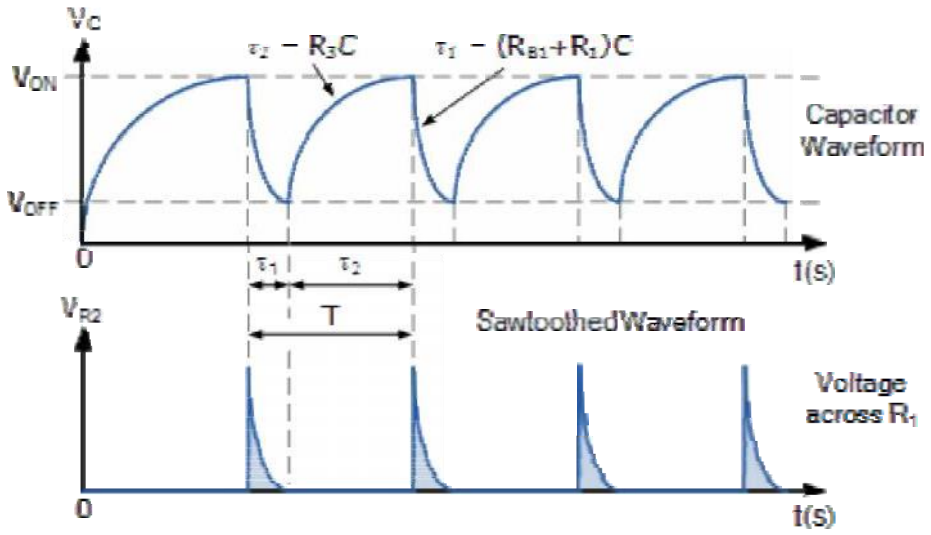
TABULATION (First CRO)

Peak voltage (V)	Time in sec	Frequency = 1/t Hz

TABULATION (Second CRO)

Peak voltage (V)	Time in sec	Frequency = 1/t Hz

OUTPUT WAVEFORM:



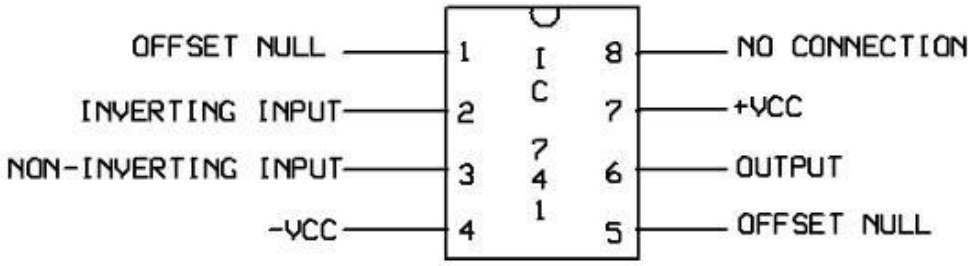
REVIEW QUESTIONS:

1. What is the function of oscillator?
2. Which portion of the characteristics used in a relaxation oscillator?
3. Which oscillator is very suitable for audio range applications?
4. Which oscillator is suitable for RF range application?
5. How does oscillation start in oscillators?
6. What is the difference between oscillator & amplifier?
7. How many classifications of oscillators?
8. What are types of oscillators based on frequency generator?
9. What is crystal oscillator?
10. Define intrinsic stand - off ratio?
11. How the frequency of oscillation determined in theoretical?
12. How many terminals in the UJT?
13. Which Condition of region the UJT is off state?
14. When the UJT is ON? Which region?
15. What is negative resistance region?

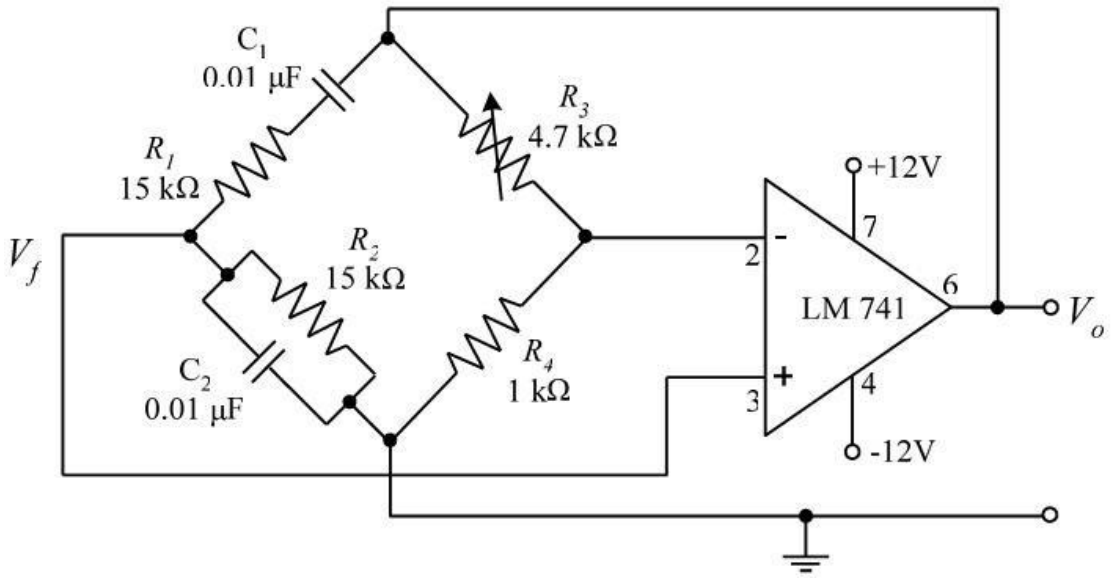
RESULT:

Thus the UJT relaxation oscillator circuit was constructed & its waveforms were drawn.

PIN DIAGRAM



CIRCUIT DIAGRAM:



Ex. No.: 5

Date:

WIEN BRIDGE OSCILLATOR USING OP-AMP

AIM:

To design a Wien Bridge oscillator using op-amp for a given frequency of 1kHz.

APPARATUS REQUIRED:

Sl. No	Apparatus Name	Range	Quantity
1	OPAMP	LM741	1
2	CRO	(0-20 MHZ)	1
3	POT Resistor	(0-10)K	1
4	Capacitor	0.01µ F	2
5	Resistor	15K 1K 4.7K	2 1 1
6	Bread board	-	1
7	Connecting Wires	-	Few

THEORY:

An oscillator is a circuit that produces periodic electric signals such as sine wave or square wave. The application of oscillator includes sine wave generator, local oscillator for synchronous receivers etc. An oscillator consists of an amplifier and a feedback network.

1. 'Active device' i.e. op-amp is used as an amplifier.
2. Passive components such as R-C or L-C combinations are used as feedback network.

To start the oscillation with the constant amplitude, positive feedback is not the only sufficient condition. Oscillator circuit must satisfy the following two conditions known as Barkhausen conditions:

1. Magnitude of the loop gain ($A_V \beta$) =1, where, A_V = Amplifier gain and β = Feedback gain.
2. Phase shift around the loop must be 360° or 0°.

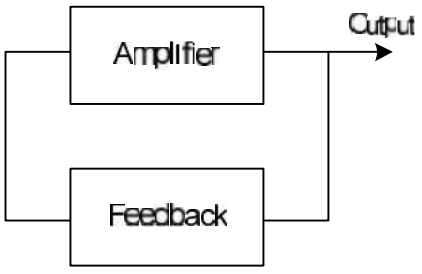


Fig 1. Basic oscillator block diagram

Wien bridge oscillator is an audio frequency sine wave oscillator of high stability and simplicity. The feedback signal in this circuit is connected to the non-inverting input terminal so that the op-amp is working as a non-inverting amplifier. Therefore, the feedback network need not provide any phase shift. The circuit can be viewed as a Wien bridge with a series Combination of R_1 and C_1 in one arm and parallel combination of R_2 and C_2 in the adjoining arm. Resistors R_3 and R_4 are connected in the remaining two arms. The condition of zero phase shift around the circuit is achieved by balancing the bridge.

DESIGN:

Frequency of oscillation, $f = \dfrac{1}{2\pi\sqrt{R_1C_1R_2C_2}}$

Let, $R_1 = R_2 = R$ and $C_1 = C_2 = C$

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

Given frequency, $f = 1$ kHz. assume C = 0.01μF

$$1000 = \frac{1}{2\pi R \times 0.01 \times 10^{-6}}$$

then $R = 15.9\ \text{k}\Omega$

Take $R_1 = R_2 = 15\ \text{k}\Omega$ (nearest standard value)

Also, $\dfrac{R_3}{R_4} = 2$

Let $R_4 = 1\ \text{k}\Omega$, then $R_3 = 2\ \text{k}\Omega$. (Use 4.7kΩ potentiometer for fine corrections).

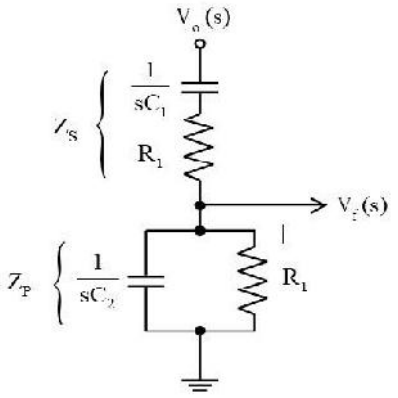


Fig 2 Circuit diagram of Wien bridge oscillator using opamp

PROCEDURE:

1. Test the op-amp by giving a sine wave at the inverting terminal, ground at the non-inverting terminal to obtain a square wave at the output.
2. Set up the circuit as shown in the figure.
3. Obtain the sine wave at the output. Check for the frequency obtained.

The series and parallel combination of RC network form a lead-lag circuit. At high frequencies, the reactance of capacitor C1 and C2 approaches zero. This causes C1 and C2 appears short. Here, capacitor C2 shorts the resistor R2. Hence, the output voltage Vo will be zero since output is taken across R2 and C2 combination. So, at high frequencies, circuit acts as a 'lag circuit'. At low frequencies, both capacitors act as open because capacitor offers very high reactance. Again, output voltage will be zero because the input signal is dropped across the R1 and C1 combination. Here, the circuit acts like a 'lead circuit'. But at one particular frequency between the two extremes, the output voltage reaches to the maximum value. At this frequency only, resistance value becomes equal to capacitive reactance and gives maximum output. Hence, this frequency is known as oscillating frequency (f).

.

Consider the feedback circuit shown in fig 2 on applying voltage divider rule,

$$V_f(s) = \frac{V_o(s) \times Z_P(s)}{Z_P(s) + Z_S(s)}$$

where, $Z_S(s) = R_1 + \frac{1}{sC_1}$ and $Z_P(s) = R_2 \| \frac{1}{sC_2}$

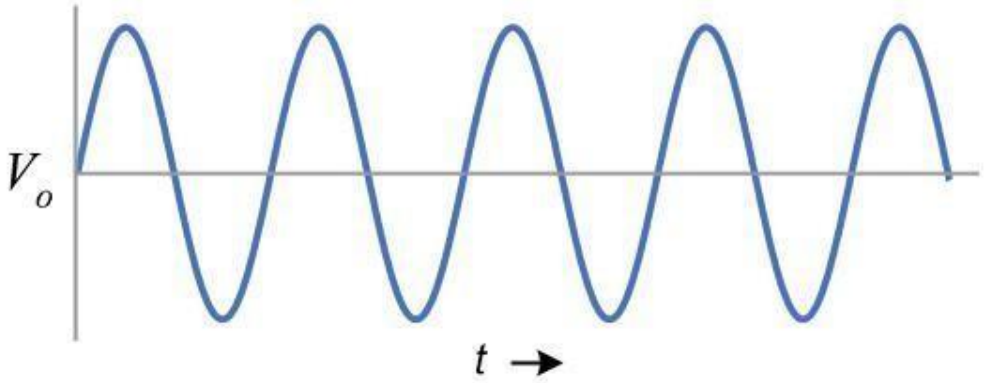
Let, $R_1 = R_2 = R$ and $C_1 = C_2 = C$. On solving,

$$\text{feedback gain, } \beta = \frac{V_f(s)}{V_o(s)} = \frac{RsC}{(RsC)^2 + 3RsC + 1} \quad (1)$$

Since the op-amp is operated in the non-inverting configuration the voltage gain,

$$A_v = \frac{V_o(s)}{V_f(s)} = 1 + \frac{R_3}{R_4} \quad (2)$$

OUTPUT:



f = 1 kHz

Applying the condition for sustained oscillations, $A_v\beta = 1$

Substitute equations (1) & (2), we get,

$$\left(1+\frac{R_3}{R_4}\right)\left(\frac{RsC}{(RsC)^2+3RsC+1}\right)=1$$

Substitute $s = j\omega$

$$\left(1+\frac{R_3}{R_4}\right)\left(\frac{j\omega RC}{-R^2C^2\omega^2+3j\omega RC+1}\right)=1$$

$$\left(1+\frac{R_3}{R_4}\right)j\omega RC=\left(-R^2C^2\omega^2+3j\omega RC+1\right)$$

$$j\omega\left[\left(1+\frac{R_3}{R_4}\right)RC-3RC\right]=1-R^2C^2\omega^2$$

To obtain the frequency of oscillation equate the real part to zero.

$$1-R^2C^2\omega^2=0$$

$$\omega=\frac{1}{RC}$$

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

To obtain the condition for gain at the frequency of oscillation, equate the imaginary part to zero.

$$j\omega\left[\left(1+\frac{R_3}{R_4}\right)RC-3RC\right]=0$$

$$j\omega\left(1+\frac{R_3}{R_4}\right)RC = j\omega 3RC$$

$$\left(1+\frac{R_3}{R_4}\right)=3 \quad (gain\ of\ the\ amplifier)$$

$$\frac{R_3}{R_4}=2$$

Therefore, $R_3 = 2\ R_4$ is the required condition.

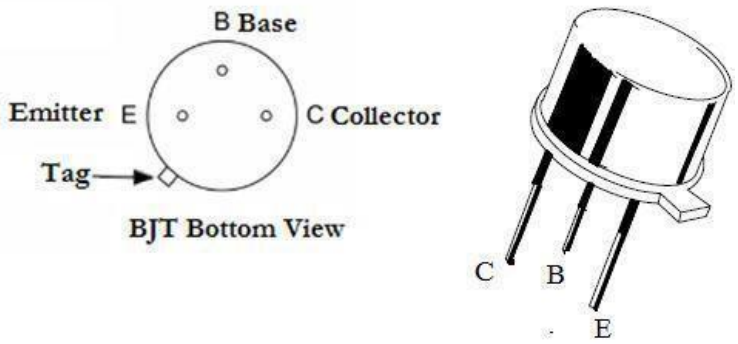
REVIEW QUESTIONS:

1. Give the formula for frequency of oscillations?
2. What is the condition for wien bridge oscillator to generate oscillations?
3. What is the total phase shift provided by the oscillator?
4. What is the function of lead-lag network in Wien bridge oscillator?
5. Which type of feedback is used in Wien bridge oscillator?
6. What is gain of Wien bridge oscillator?
7. What are the application of Wien bridge oscillator?
8. What is the condition for oscillations?
9. What is the difference between damped oscillations undamped Oscillations??

RESULT:

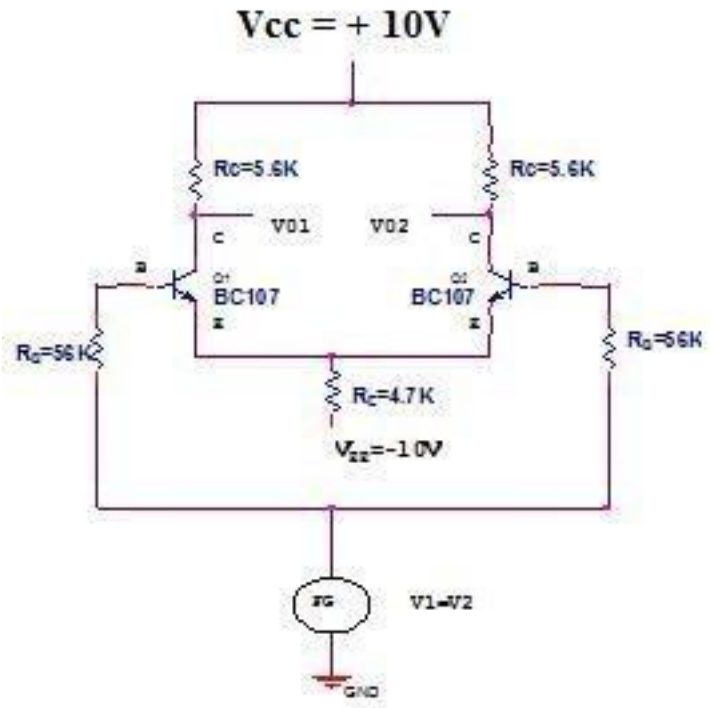
A Wien bridge oscillator was designed and setup for a frequency of 1kHz and the output waveform is observed.

PIN DIAGRAM:



CIRCUIT DIAGRAM:

COMMON MODE



Ex. No.: 6

Date:

DIFFERENTIAL AMPLIFIER USING BJT

AIM:

i. To design and construct Differential amplifier using BJT BC107.
ii. To calculate Common mode gain and Differential mode gain.
iii. To calculate common mode rejection ratio (CMRR).

APPARATUS REQUIRED:

S.No	Name	Range	Quantity
1.	Resistors	5.6KΩ,56KΩ,4.7KΩ	2,2,1
2.	Transistor	BC107	2
3.	Function Generator	-	1
4.	Multimeter	-	1
5.	Dual power supply	(0-30) V	1
6.	Bread board	-	1
7.	Connecting Wires	-	Few

THEORY:

The differential amplifier amplifies the difference between two input voltage signals. Hence it is also called difference Amplifier. In an ideal differential amplifier, the output voltage Vo is proportional to the difference between the two input signals. Hence we can write, $V_O = A_d$ (V1-V2) Where Ad refers to differential gain, which amplifies the difference between two input signals.

$$Vo = A_d v_d ;$$

$$A_d = Vo/V_d$$

Generally the differential gain is expressed in its decibel (dB) value as, $Ad=20 \log_{10} (A_d)$ in dB. An average level of the two input signals is called common mode signal denoted as Vc, Vc= (V1+V2)/2 The gain with which it amplifies the common mode signal to produce the output is called common mode gain of the differential amplifier denoted as Ac.

$$V_0 = AcVc ;$$

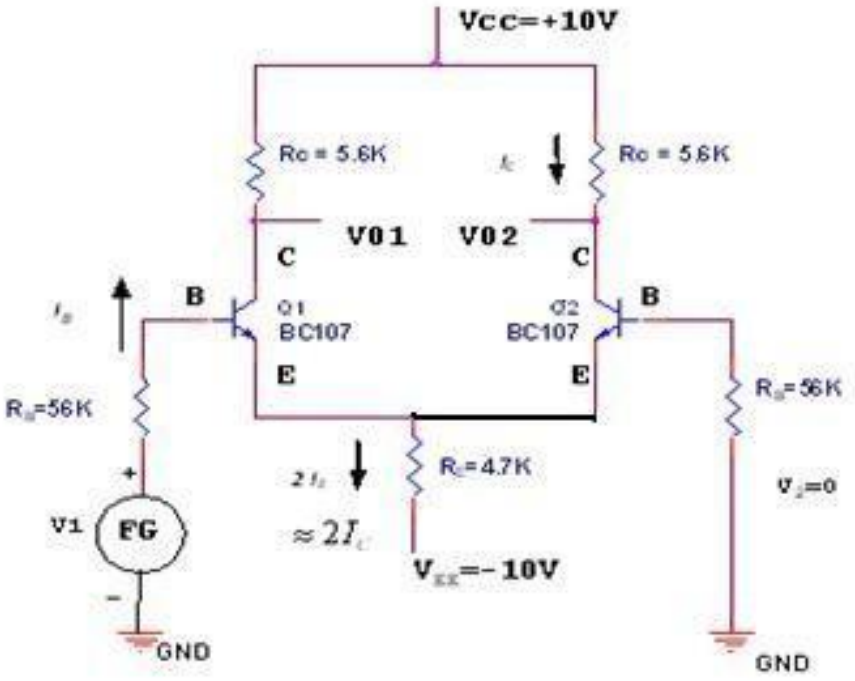
$$Ac = V_O/Vc$$

Therefore total output of any differential amplifier can be expressed as,

$$Vo = A_dV_d + AcVc$$

CIRCUIT DIAGRAM:

DIFFERENTIAL MODE:



TABULATION:

COMMON MODE:

$V_1=V_2$ (Volts)	$V_{IN} = (V_1 + V_2) / 2$ (Volts)	V_0 (Volts)	$A_{C=} V_0/V_{in}$

PRACTICAL CALCULATION:

CMRR=Ad/Ac; CMRR (dB)= 20 log(Ad/Ac)

DESIGN:

$I_{CQ} = 1mA \approx I_E$; $V_{CC} = 10V = V_{EE}$; $\beta = 250$

hie =1.2k

$$I_B = \frac{Ic}{\beta} = \frac{1*10^3}{250} = 4\mu A$$

Choose R_B as 57k

Applying KVL at input side with AC input as 0v

$$-I_B R_B - V_{BE} - 2I_E R_E + V_{EE} = 0$$

$$V_{EE} - V_{BE} - I_B R_B = 2I_C R_E$$

$$R_E = \frac{V_{EE} - V_{BE} - I_B R_B}{2I_E} = \frac{10 - 0.7 - (4\mu A * 57 * 10K)}{2*1*10^{-3}} = 4.53K$$

Assume $V_{CE} = \frac{V_{CC}}{2} = 5V$

Applying KVL at Output side

$$V_{CC} - (-V_{EE}) = V_{EE} + V_{CC} = I_C R_C + V_{CE} + 2I_E R_E$$

$$R_C = \frac{V_{EE} + V_{CC} - V_{CE} - 2I_E R_E}{I_C} = \frac{10 + 10 - 5 - 2(1*10^{-3} * 4.5*10^{-3})}{1*10^{-3}} = 6K$$

THEORETICAL CALCULATION:

$$A_d = \frac{h_{fe} R_c}{Rs + hie}$$

$$A_C = \frac{hfeRc}{Rs + hie + 2R_E(1 + hfe)}$$

TABULATION:

DIFFERENTIAL MODE:

V_1 (Volts)	V_2 (Volts)	$V_{IN} = V_1 - V_2$ (Volts)	V_0 (Volts)	$A_{d=} V_0/V_{IN}$

PROCEDURE:

1. Connect the circuit as based on the designed values (differential mode, common mode).
2. Verify the KVL at both input and output side of the circuit.
3. Set V_{IN} =50mV at 1KHz in the function generator. Keeping input voltage as constant, for both transistors. (at common mode)
4. To find output voltages V_{01} and V_{02} and also find output voltage V_0.
5. Calculate the common Gain A_C in dB using the formula mentioned.
6. Set V_1 =50mV at 1KHz in the function generator input for transistor Q_1 and Set V_2 =100mV at 1KHz in the function generator input for transistor Q_2 (at Differential mode)
7. To find output voltages V_{01} and V_{02} and also find output voltage V_0.
8. Calculate the Differential mode Gain A_d in dB using the formula mentioned.
9. Calculate the Common Mode Rejection Ratio (CMRR) = 20 log (Ad/Ac) in dB.

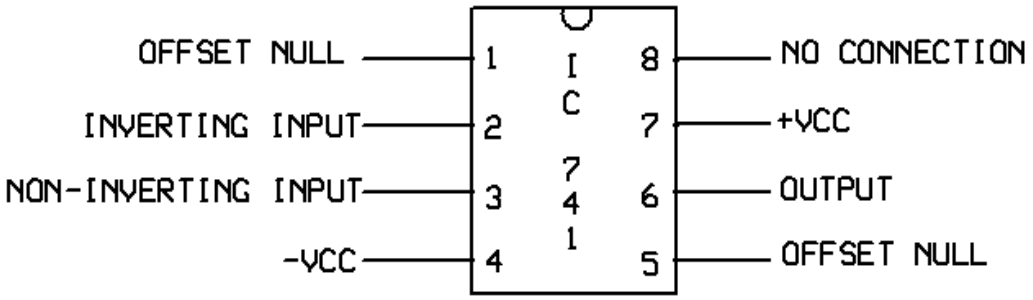
REVIEW QUESTIONS:

1. What is Differential amplifier?
2. What is difference between amplifier and Differential amplifier?
3. Sketch the circuit diagram of Differential amplifier using BJT.
4. List out the applications of Differential amplifier.
5. What is the mode of operations in Diff. Amp?
6. What is Common mode gain and difference mode gain?
7. What is the use of CMRR?
8. What is gain?
9. What is the unit of current and voltage gain?
10. Compare the application of Differential amplifier using FET with Diff. Amp using BJT

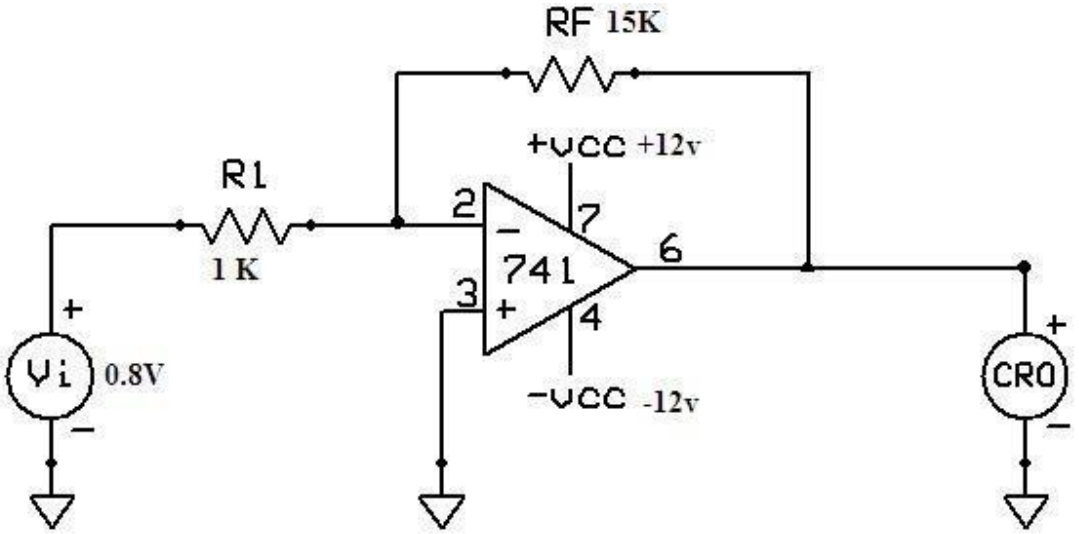
RESULT:

(i) Thus, the Differential amplifier is designed and constructed using BJT.
(ii) Common mode gain and Differential mode gain are calculated.
(iii)The CMRR of Differential Amplifier is _____ dB.

PIN DIAGRAM:



CIRCUIT DIAGRAM OF INVERTING AMPLIFIER:



DESIGN:

We know for an inverting Amplifier ACL = RF / R1
Assume R1 (approx. 10 KΩ) and find Rf
Hence VO (theoretical) = - ACL VI

OBSERVATIONS:

S.No.	Amplitude (No. of div x Volts per div)	Time period (No. of div x Time per div)
Input		
Output	Theoretical -	
	Practical -	

OPAMP BASED AMPLIFIER CIRCUITS

Ex. No: 7(i)
Date:

i. INVERTING AMPLIFIER

AIM:

To design an Inverting Amplifier for the given specifications using Op-Amp IC 741.

APPARATUS REQUIRED:

S.No	Name of the Apparatus	Range	Quantity
1.	Function Generator	3 MHz	1
2.	CRO	30 MHz	1
3.	Dual RPS	0 – 30 V	1
4.	Op-Amp	IC 741	1
5.	Bread Board	-	As required
6.	Resistors	1KΩ, 15KΩ	Each 1
7.	Connecting wires and probes	Single Strand Wire	As required

THEORY:

The input signal V_i is applied to the inverting input terminal through R_1 and the non- inverting input terminal of the op-amp is grounded. The output voltage V_o is fed back to the inverting input terminal through the R_f - R_1 network, where R_f is the feedback resistor. The output voltage is given as,

$$V_o = - A_{CL} V_i$$

Here the negative sign indicates that the output voltage is 180^0 out of phase with the input signal.

PRECAUTIONS:

1. Output voltage will be saturated if it exceeds ± 15V.

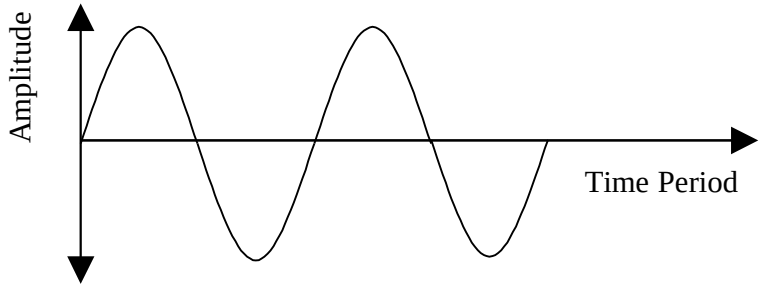
PROCEDURE:

1. Connections are given as per the circuit diagram.
2. + V_{CC} and - V_{CC} supply is given to the power supply terminal of the Op-Amp IC.
3. By adjusting the amplitude and frequency knobs of the function generator, appropriate input voltage is applied to the inverting input terminal of the Op-Amp.
4. The output voltage is obtained in the CRO and the input and output voltage waveforms are plotted in a graph sheet.

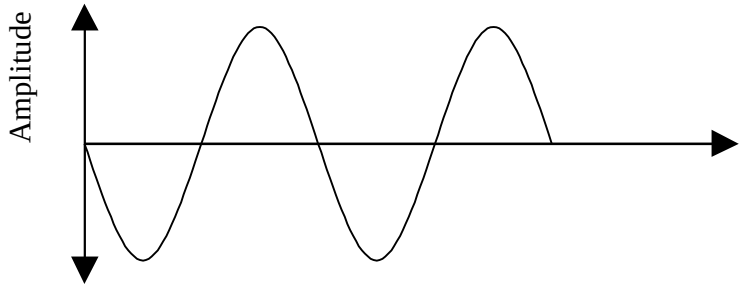
MODEL GRAPH:

INVERTINGAMPLIFIER:

INPUT SIGNAL:



OUTPUT SIGNAL:



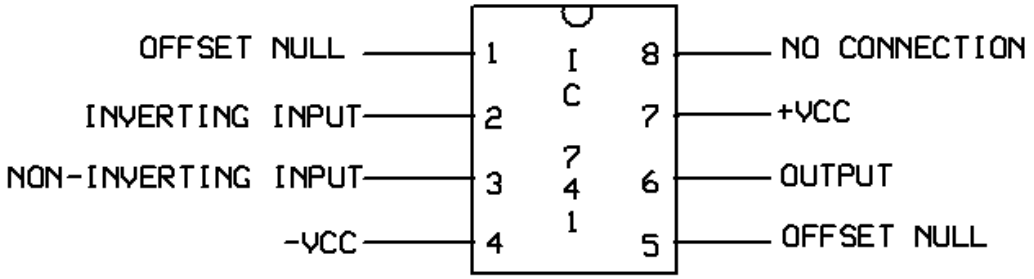
RESULT:

The design and testing of the inverting amplifier is done and the input and output waveforms were drawn

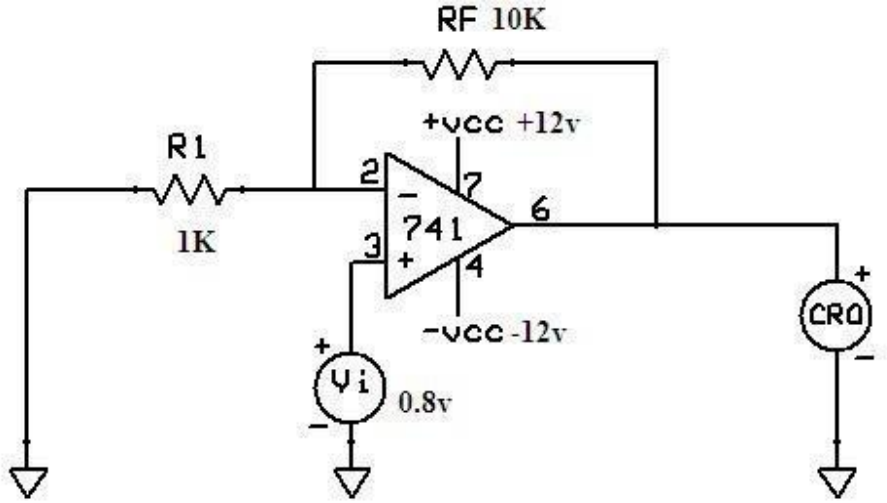
DESIGN:

We know for a Non-inverting Amplifier $A_{CL} = 1 + (R_F / R_1)$
Assume R_1 (approx. 10 KΩ) and find R_f
Hence $V_o = A_{CL}\ V_i$

PIN DIAGRAM:



CIRCUIT DIAGRAM OF NON INVERITNG AMPLIFIER:



OBSERVATIONS:

S.No.	Amplitude (No. of div x Volts per div)	Time period (No. of div x Time per div)
Input		
Output	Theoretical -	
	Practical -	

Ex. No: 7(ii)
Date:

ii. NON - INVERTING AMPLIFIER

AIM:

To design a Non-Inverting Amplifier for the given specifications using Op-Amp IC 741.

APPARATUS REQUIRED:

S.No	Name of the Apparatus	Range	Quantity
1.	Function Generator	3 MHz	1
2.	CRO	30 MHz	1
3.	Dual RPS	0 – 30 V	1
4.	Op-Amp	IC 741	1
5.	Bread Board	-	1
6.	Resistors	1KΩ, 10KΩ	Each 1
7.	Connecting wires and probes	Single Strand Wire	As required

THEORY:

The input signal V_i is applied to the non - inverting input terminal of the op-amp. This circuit amplifies the signal without inverting the input signal. It is also called negative feedback system since the output is feedback to the inverting input terminals. The differential voltage V_d at the inverting input terminal of the op-amp is zero ideally and the output voltage is given as,

$$V_o = A_{CL} V_i$$

Here the output voltage is in phase with the input signal.

PRECAUTIONS:

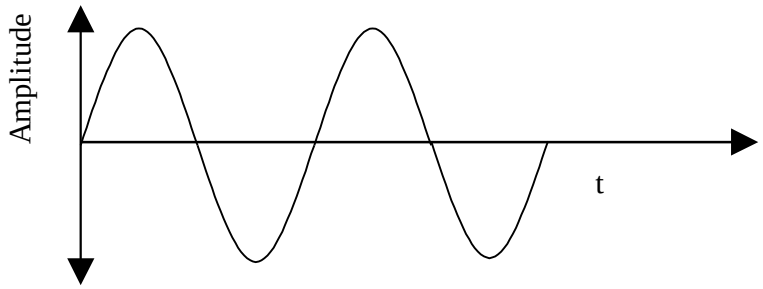
1. Output voltage will be saturated if it exceeds ± 15V.

PROCEDURE:

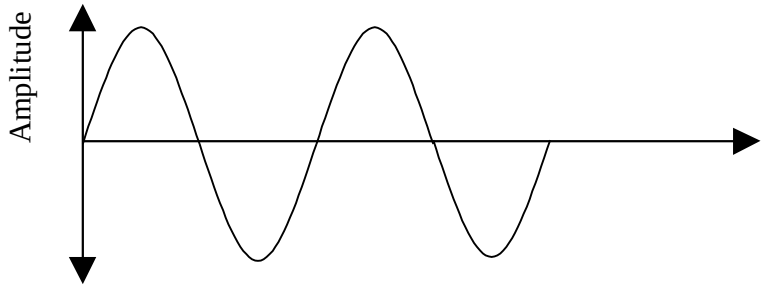
1. Connections are given as per the circuit diagram.
2. $+ V_{cc}$ and $- V_{cc}$ supply is given to the power supply terminal of the Op-Amp IC.
3. By adjusting the amplitude and frequency knobs of the function generator, appropriate input voltage is applied to the non - inverting input terminal of the Op-Amp.
4. The output voltage is obtained in the CRO and the input and output voltage waveforms are plotted in a graph sheet.

MODEL GRAPH:
NON-INVERTINGAMPLIFIER:

INPUT SIGNAL:



OUTPUT SIGNAL:



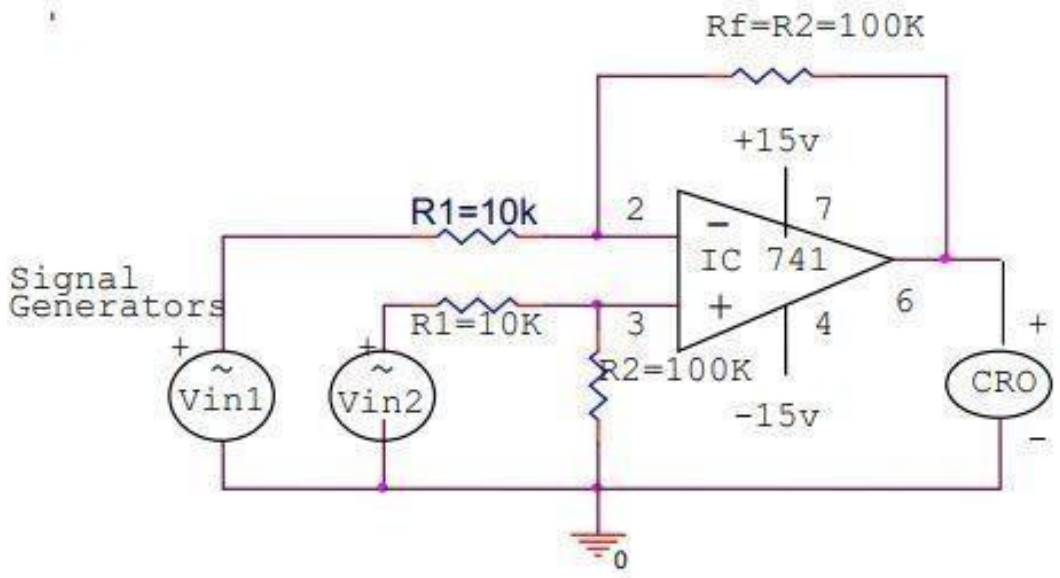
DISCUSSION QUESTIONS:

1. What do you mean by linear circuits?
2. Define an IC?
3. What is an inverting amplifier?
4. What is the type of feedback employed in the inverting op-amp

RESULT:

The design and testing of the Non-inverting amplifier is done and the input and output waveforms were drawn

DIFFERENTIAL AMPLIFIER:- CIRCUIT DIAGRAM:-



TABULATION:

S.No	V_{in1} (Volts)	V_{in2} (Volts)	$V_{in2} - V_{in1}$ (Volts)	V_0 (Volts)	Theoretical Gain $A = -R_f / R_1$	Practical Gain $A=V_0 / (V_{in2} - V_{in1})$

Ex. No: 7(iii.a)
Date:

iii.a DIFFERENTIAL AMPLIFIER

AIM:

To design a Differential Amplifier for the given specifications using Op-Amp IC 741.

APPARATUS REQUIRED:

S.No	Name of the Apparatus	Range	Quantity
1.	Function Generator	3 MHz	1
2.	CRO	30 MHz	1
3.	Dual RPS	0 – 30 V	1
4.	Op-Amp	IC 741	1
5.	Bread Board	-	1
6.	Resistors	10KΩ, 100KΩ	Each 2
7.	Connecting wires and probes	Single Strand Wire	As required

THEORY:

A configuration which combines inverting & non-inverting configuration with both input terminals are supplied with Vin1 & Vin2, then it is called as Differential Amplifier configuration. This circuit amplifies the difference between the two inputs. Differential amplifier with a single op-amp has the exact gain of an inverting amplifier and it is given as

$$A_D(\text{Using one OP-AMP}) = A_{VCL} = \mathbf{V_0/(V_{in2} - V_{in1}) = -R_f / R_1}$$

A differential amplifier with two op-amps has the exact gain of a non-inverting amplifier and it is given as:

$$A_D(\text{Using two OP-AMP}) = A_{VCL} = \mathbf{V_0/(V_{in2} - V_{in1}) = 1+(R_f / R_1)}$$

PROCEDURE:

1. Select the value of R1, R2, R3 & Rf such that R1=R2 and R3=Rf.

2. Connect the circuit as per as the circuit diagram.

3. Provide constant input voltage Vin1 to Non-inverting terminal of op-amp through R1 & constant input voltage Vin2 to inverting terminal of op-amp through R2.

4. Measure the output voltage using CRO.

5. Calculate the theoretical gain and compare it with practical gain.

6. Practical gain & theoretical gain should be approximately equal.

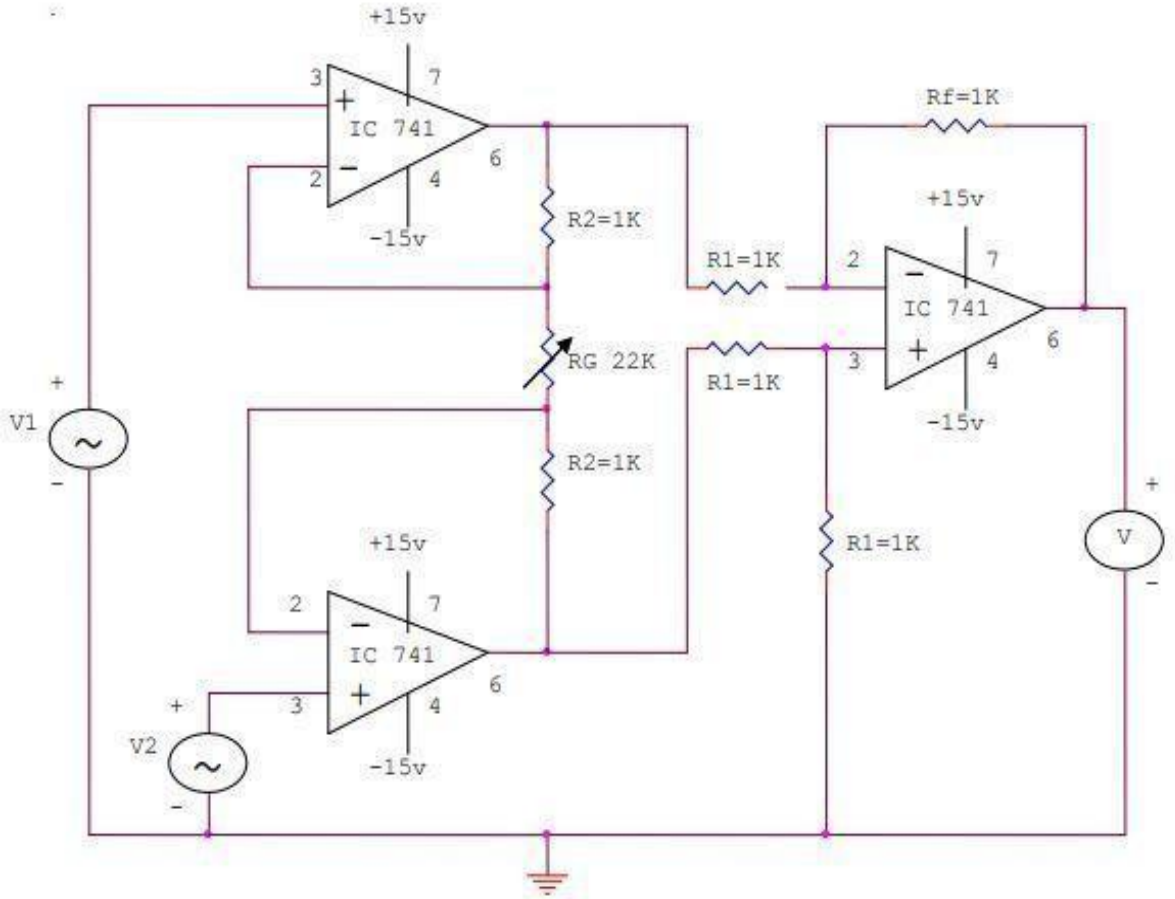
DISCUSSION QUESTIONS:

1. Give the closed loop gain of an inverting amplifier?
2. What is the gain of a non-inverting amplifier?

RESULT:

The design and testing of the Differential amplifier is done and the input and output waveforms were drawn

CIRCUIT DIAGRAM:



Ex. No: 7(iii.b)
Date:

iii.b.INSTRUMENTATION AMPLIFIER

AIM:

To construct and test the CMRR of an instrumentation amplifier using op-amp IC741.

APPARATUS REQUIRED:

S.No	Name of the Apparatus	Range	Quantity
1	Op-Amp	IC 741	03
2	Resistors	1KΩ & 22KΩ	6 & 1
3.	Digital trainer kit	--	1
4	Signal generator	(0-3) MHz	02
5	CRO	(0-30)MHz	1
6	Connecting wires	Single Strand Wire	few

THEORY:

An instrumentation amplifier is the intermediate stage of a instrumentation system. The signal source of the instrumentation amplifier is the output of the transducer. Many transducers output do not have the ability or sufficient strength to drive the next following stages. Therefore, instrumentation amplifiers are used to amplify the low-level output signal of the transducer so that it can drive the following stages such as indicator or displays. The major requirements of a instrumentation amplifier are precise, low-level signal amplification where low-noise, low thermal and time drifts, high input resistance & accurate closed-loop gain, low power consumption, high CMRR & high slew rate for superior performance.

TABULATION:

COMMON MODE GAIN CALCULATION A_C

S.No	R_G (KΩ)	V_1 VOLTS	V_2 VOLTS	V_0 VOLTS	$A_C=V_0/[(V_1+V_2)/2]$

DIFFERENTIAL MODE GAIN A_D & CMRR CALCULATION.:

S.No	R_G (KΩ)	V_1 VOLTS	V_2 VOLTS	V_0 VOLTS	$A_D=V_0/(V_1-V_2)$	$CMRR=20\log(\lvert A_D\rvert/\lvert A_C\rvert)$ DB

PROCEDURE:

1. Select the entire resistor with same value of resistance R. Let R_G be the gain varying resistor with different values of resistance for simplicity let R_G, be a constant value.
2. Connect the circuit as shown in the circuit diagram.
3. Give the input V_1 & V_2 to the non-inverting terminals of first & second op amp respectively.
4. By varying the value of R_G, measure the output voltage for common mode and differential mode operation. Since R_G is selected as constant value, provide different input value of V_1 & V_2.
5. Calculate the differential mode gain A~~D~~ and common mode gain A~~C~~ to calculate the CMRR as $20\log(|A_D|/|A_C|)$ dB.

DISCUSSION QUESTIONS:

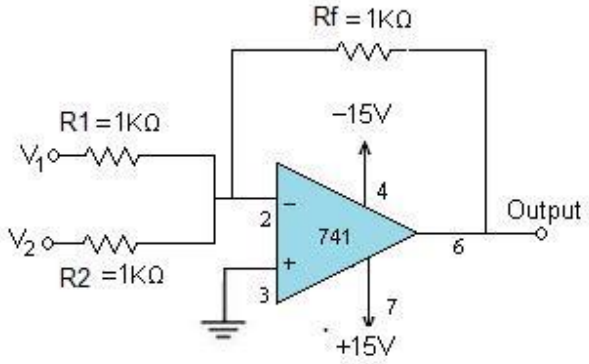
1. What is a voltage follower?
2. Define a non-inverting amplifier?

RESULT:

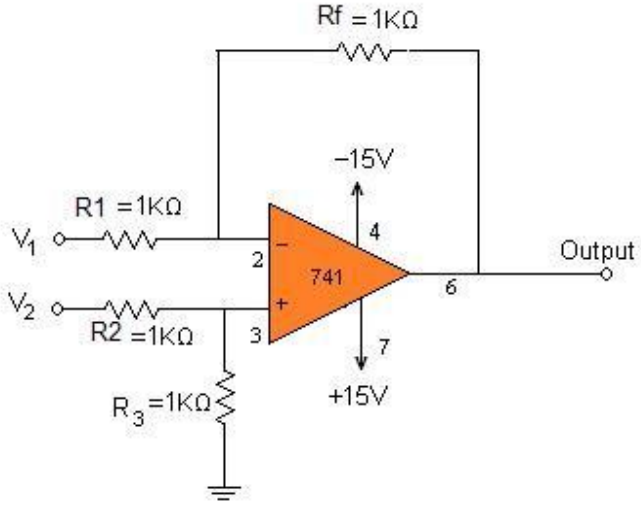
Thus an instrumentation amplifier was constructed and CMRR is tested using op-amp IC 741.

CIRCUIT DIAGRAM:

Adder:



Subtractor:



EX.NO:8
DATE:

DESIGN OF ADDER/SUBTRACTOR CIRCUITS

AIM:

To study the applications of IC 741 as adder and Subtractor.

APPARATUS REQUIRED:

S.NO	APPARATUS NAME	SPECIFICATIONS	QUANTITY
1.	Op-amp	IC741	1
2.	Resistor	1K	4
3.	Patch chords	-	few
4	Function generator	3 MHz	1
5	IC Bread board trainer	-	1
6	CRO	(0-30)MHz	1
7	CRO Probe	-	few

THEOR:

ADDER:

Op-Amp may be used to design a circuit whose output is the sum of several input signals such as circuit is called a summing amplifier or summer. We can obtain either inverting or non inverting summer.

The circuit diagrams shows a two input inverting summing amplifier. It has two input voltages $V_1$and V_2, two input resistors R_1 ,R_2 and a feedback resistor R_f.

Assuming that op-amp is in ideal conditions and input bias current is assumed to be zero, there is no voltage drop across the resistor R_{comp} and hence the non inverting input terminal is at ground potential.
By taking nodal equations.

$V_1/R_1 + V_2/R_2 + V_0/R_f = 0$
$V_0 = - [(R_f/R_1)\ V_1 + (R_f/R_2)\ V_2]$ And
here $R_1 = R_2 = R_f = 1K$
$V_0 = -(V_1 + V_2)$

Thus output is inverted and sum of input.

SUBTRACTOR:

The subtractor circuit, input signals can be scaled to the desired values by selecting appropriate values for the resistors. When this is done, the circuit is referred to as scaling amplifier. However in this circuit all external resistors are equal in value. So the gain of amplifier is equal to one. The output voltage Vo is equal to the voltage applied to the non-inverting terminal minus the voltage applied to the inverting terminal; hence the circuit is called a subtractor.

PROCEDURE:

ADDER:

1. connections are made as per the circuit diagram.
2. Apply input voltage 1) V_1= 5v,V_2=2v
   2) V_1= 5v,V_2=5v
   3) V_1=5v,V_2=7v.
3. Using Millimeter measure the dc output voltage at the output terminal.
4. For different values of V_1 and V_2 measure the output voltage.

SUBTRACTOR:

1. Connect the circuit as per the diagram.
2. Apply the supply voltages of +15V to pin7 and pin4 of IC741 respectively.
3 Apply the inputs V1 and V2.
4. Apply two different signals (DC/AC) to the inputs.
5. Vary the input voltages and note down the corresponding output at pin 6 of the IC 741 subtractor circuit.
6. Notice that the output is equal to the difference of the two inputs.

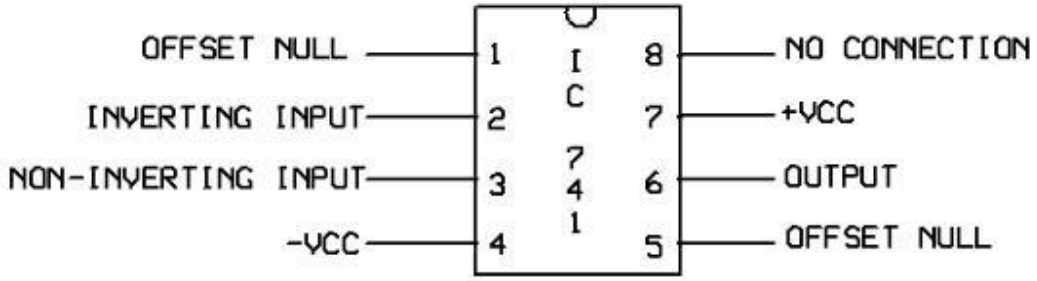
DISSCUSSION QUESTION:

1. What is the use of an op amp?
2. Why op amp has high gain?

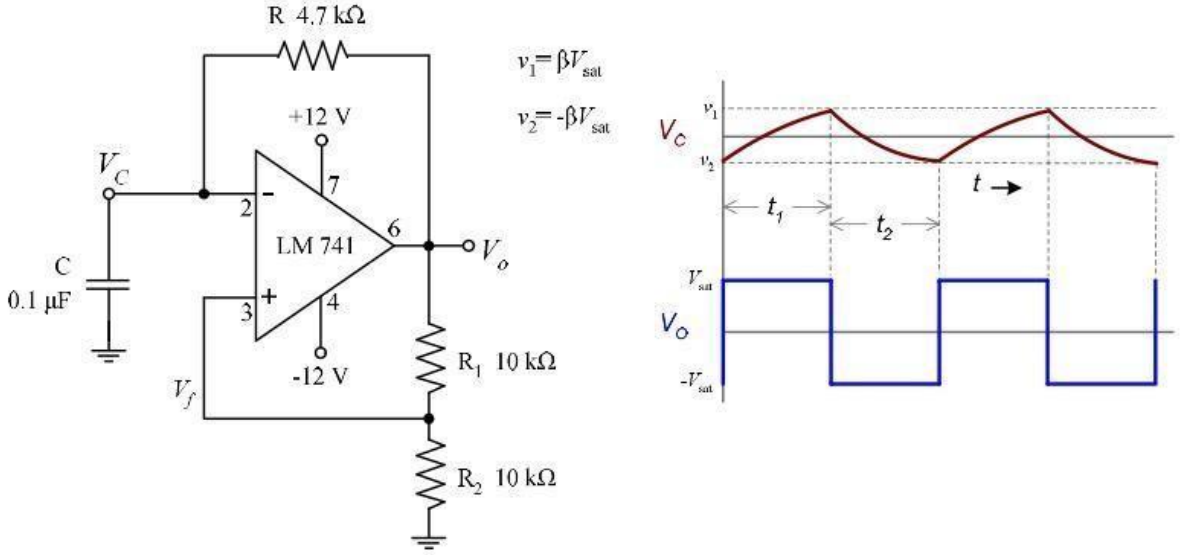
RESULT:

Thus the Adder and Subtractor circuit was studied.

PIN DIAGRAM



SQUARE WAVE GENERATOR AND WAVEFORMS



DESIGN:

DESIGN OF SQUARE WAVE GENERATOR:

Let the frequency of oscillation be 1 kHz

Take $\beta = 0.5$ and $R_1 = R_2 = 10$ kΩ.

Frequency, $f = \dfrac{1}{2RC \ln 3}$ Assume $C = 0.1$ µF

Then, $R = \dfrac{1}{2Cf \ln 3} = \dfrac{1}{2 \times 0.1 \times 10^{-6} \times 1000 \times \ln 3} = 4.55$ kΩ

Select standard value of 4.7 kΩ for R.

Ex. No.: 9

Date:

SQUARE WAVE OSCILLATOR/ TRIANGULAR WAVE OSCILLATOR

AIM:

To set up and study square waveform, triangular waveform generator using Op-Amp.

APPARATUS REQUIRED:

S.No	Components / Equipments	Specifications	Quantity
1.	Resistors	10 KΩ 12KΩ 4.7KΩ 1KΩ	2 1 1 1
2.	OP-Amp	LM741	2
3.	CRO	(0-30)MHz	1
4.	Capacitor	0.1µ F	1
5.	Dual power supply	(0-30)v	2
6.	Bread board	-	1
7.	Connecting Wires	Single Strand Wire	Few

THEORY:

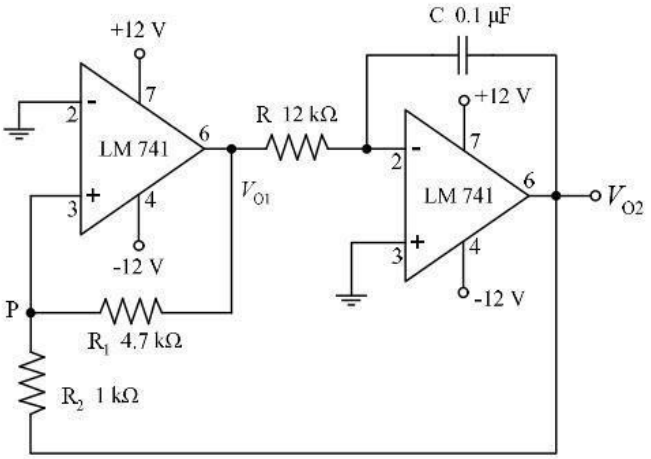
Square wave oscillator:

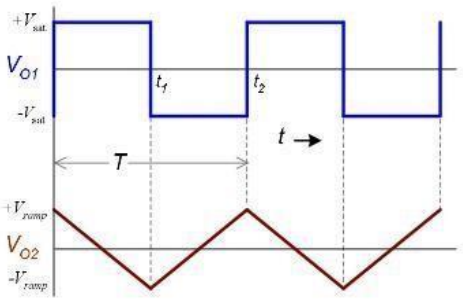
The basic square wave oscillator is based on the charging and discharging of a capacitor. Op-amps inverting input is the capacitor voltage and the non-inverting input is a portion of the output fed back through resistors R1 and R2. When the circuit is first turned on, the capacitor is uncharged, and thus the inverting input is at 0V. This makes the output a positive maximum, and the capacitor begins to charge towards voltage at VO through resistor R. When the capacitor voltage reaches a value equal to the feedback voltage (Vf) on the non-inverting input, the op-amp switches to the maximum negative state. At this point, the capacitor begins to discharge from +Vf towards –Vf. When the capacitor voltage reaches –Vf, the op-amp switches back to the maximum positive state. This action repeats and a square wave output voltage is obtained.

Expression for period is

$$T = 2RC\ln\frac{1+\beta}{1-\beta} \qquad \text{where } \beta = \frac{R_2}{R_1 + R_2}$$

TRIANGULAR WAVE GENERATOR AND WAVEFORMS





DESIGN:

DESIGN OF TRIANGULAR WAVE GENERATOR:

Let the frequency of oscillation be 1 kHz

We have $f = \dfrac{R_1}{4RCR_2}$ and $V_{O(pp)} = 2\dfrac{R_2}{R_1}V_{sat}$

Since supply voltage is ±12 V, V_{sat} will be approximately 10 V

Let $V_{O(pp)}$ be 5 V; Assume $R_2 = 1$ kΩ.

Then $R_1 = \dfrac{2V_{sat}}{V_{O(pp)}}R_2 = \dfrac{2\times10}{5}\times1\times10^3 = 4\ \text{k}\Omega$

Select standard value, $R_1 = 4.7$ kΩ

Assume C = 0.1 µF

$$R = \frac{R_1}{4fCR_2} = \frac{4.7\times10^3}{4\times1000\times0.1\times10^{-6}\times1\times10^3} = 11.7\ \text{k}\Omega$$

Select standard value, $R = 12$ kΩ

If $R_1 = R_2$, the equation for period reduces to $T = 2RC \ln 3$

The frequency of oscillation, $f = \dfrac{1}{2RC \ln 3}$

Triangular-wave oscillator:

This circuit (figure 2) uses two operational amplifiers. Op-amp A1 functions as a comparator and the op-amp A2 as an integrator. Comparator compares the voltage at point P continuously with respect to the voltage at the inverting input; which as at ground potential. When the voltage at P goes slightly below zero, the output of A1 will switch to negative saturation. Suppose the output of A1 is at positive saturation $+V_{sat}$. Since this voltage is the input of the integrator, the output of A2 will be a negative going ramp. Thus, one end of the voltage divider R1-R2 is at $+V_{sat}$ and the other at the negative going ramp. At time t = t1, when the negative going ramp attains value of $-V_{ramp}$ the effective voltage at point P becomes slightly less than 0 V. This switches output of A1 from positive saturation to negative saturation level $-V_{sat}$. During the time when the output of A1 is at $-V_{sat}$, the output of A2 increases in positive direction. At the instant t = t2, the voltage at point P becomes just above 0 V, thereby switching the output of A1 from $-V_{sat}$ to $+V_{sat}$. The cycle repeats and generates a triangular waveform

At $t = t_1$ $\quad \dfrac{-V_{ramp}}{R_2} = -\dfrac{+V_{sat}}{R_1}$ $\quad$ ie. $\quad -V_{ramp} = -\dfrac{R_2}{R_1}(+V_{sat})$

Similarly, at $t = t_2$ $\quad +V_{ramp} = -\dfrac{R_2}{R_1}(-V_{sat})$

The peak to peak output of the triangular wave is

$$V_{O(pp)} = +V_{ramp} - (-V_{ramp}) = 2\frac{R_2}{R_1}V_{sat}$$

During the period 0-t_1, The integrator functions as below.

$$V_{O(pp)} = \frac{1}{RC}\int_0^{T/2}(-V_{sat})\,dt = \left(\frac{V_{sat}}{RC}\right)\left(\frac{T}{2}\right)$$

Then, $\quad T = 2RC\left(\dfrac{V_{O(pp)}}{V_{sat}}\right)$

Substituting for $V_{O(pp)}$ $\quad T = \dfrac{4RCR_2}{R_1}$

Then, frequency of oscillation, $f = \dfrac{R_1}{4RCR_2}$

REVIEW QUESTIONS:

1. Do we require any input signal for a square wave generator give reason.
2.What is the combination of OP-AMP circuits used for generation of Triangular wave generator?
3. What is the O/P of Integration if positive unit step signal is applied?
4. Define Comparator circuit.
5. What are the applications of Square &Triangular wave generator circuit?
6. How can we increase the amplitude of the o/p square and triangular wave?
7. State the two conditions for oscillations.
8. Define an oscillator?

PROCEDURE:

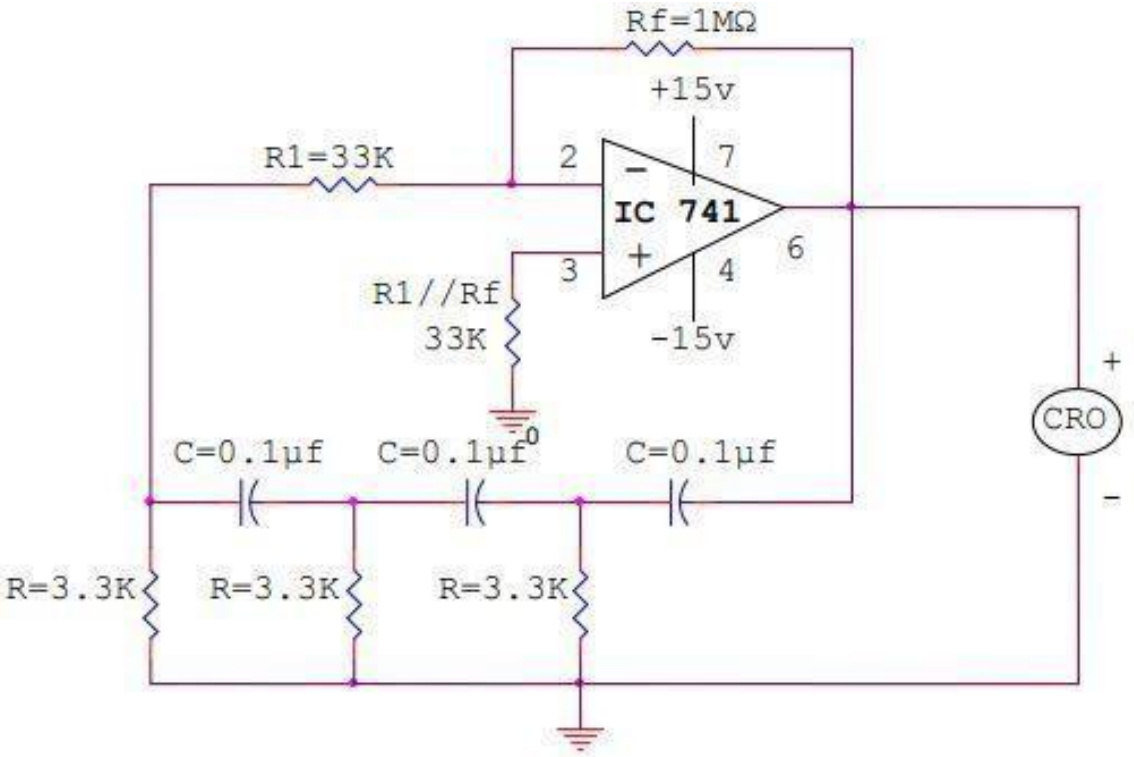
1. Set up the circuit after testing the components.
2. Set up the square wave generator as shown in figure and observe the output waveform and note down their amplitudes and frequencies.
3. Set up the triangular wave generator as shown in figure and observe the variation in frequencies of output waveform by varying the values of resistances ,
4. Move the wiper of the potentiometer in both directions and observe the changes taking place in the waveform.

RESULT:

Circuits of square wave generator and triangular wave generator wave generator are designed, setup and waveforms observed.

RC PHASE SHIFT OSCILLATOR:-

CIRCUIT DIAGRAM:-



TABULATION:

OBSERVED OUTPUT WAVEFORM			Design Frequency (Hz)
Amplitude (volts)	Time period (ms)	Frequency (Hz)	

Ex. No: 10
Date:

OP AMP BASED RC – PHASE SHIFT OSCILLATOR

AIM:

To design RC Phase Shift Oscillator using op-amp IC 741 and to test its performance.

APPARATUS REQUIRED:

S.No	Name of the Apparatus	Range	Quantity
1.	Op-Amp	IC741	1
2.	Resistors	3.3KΩ, 33KΩ	3,2
		1MΩ	1
3.	Capacitors	0.1μf	3
4	Digital Trainer Kit	--	1
5	CRO	(0-30)MHz	1
6	Connecting Wires	Single Strand Wire	FEW

THEORY:

RC phase shift oscillator produces 360° of phase shift in two parts. Firstly, each and every RC pair in the feedback network produces 60° phase shift and totally there were three pairs, thus producing 180° Phase shift and secondly, the feedback input is given to the inverting terminal of op-amp to produce another 180° phase shift and a total phase shift of 360°. The frequency of oscillation is given by $f_0 = 1/(2\pi\sqrt{6}RC$; If an inverting amplifier is used, the gain must be atleast equal to 29 to ensure the oscillations with constant amplitude that is, $|A_V \beta| < 1$. Otherwise the oscillation will die out.

DESIGN PROCEDURE:

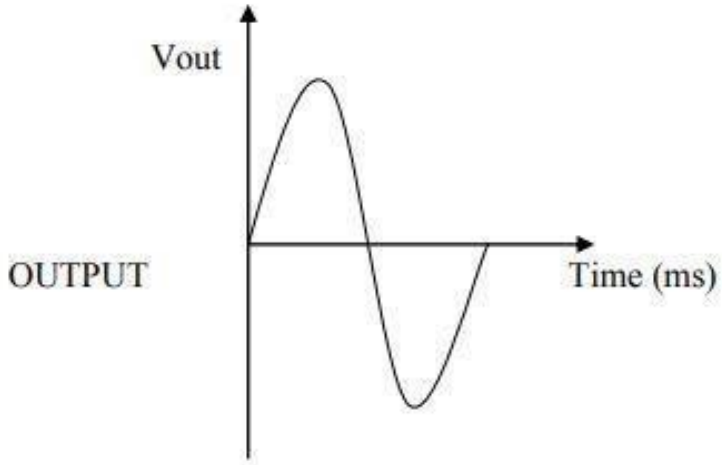
Design a RC phase shift oscillator to oscillate at 200Hz.

1. Select $f_o = 200$Hz.
2. Assume C = 0.1μf & determine R from f_o.

   $f_o = 1/(2\pi\sqrt{6}RC) = R = 1/(2\pi\sqrt{6}f_{Oc}) = 3.3$ K.

3. To prevent the loading of amp because it is necessary that R1>>10R.
   Therefore $R_1 = 10R = 33$K.
4. At this frequency the gain must be atleast 29 (i.e)Rf / R1 =29.
   Therefore $R_f = 29R_1$.
   $R_f = 29$ (33K) = 957KΩ.
   Therefore use R_f = 1MΩ.

MODEL GRAPH:



PROCEDURE- (RC PHASE SHIFT):-

1. Select the given frequency of oscillation f_0 = 200Hz.
2. Assume either R or C to find out the other using formula $f_0 = 1/(2\pi\sqrt{6}RC)$.
3. The gain is selected such that R_f / R_1 = 29K. Assume R_f or R_1 to find the other.
4. Connect the circuit as per as the circuit diagram.
5. Measure the amplitude frequency of the output signal plot the graph.

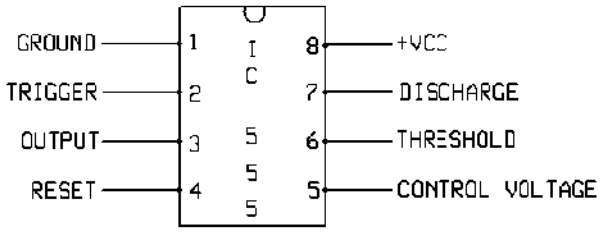
DISCUSSION QUESTIONS:

1. What is RC phase shift oscillator?
2. What is the frequency of RC phase shift oscillator?
3. What are the advantages of RC phase shift oscillator?

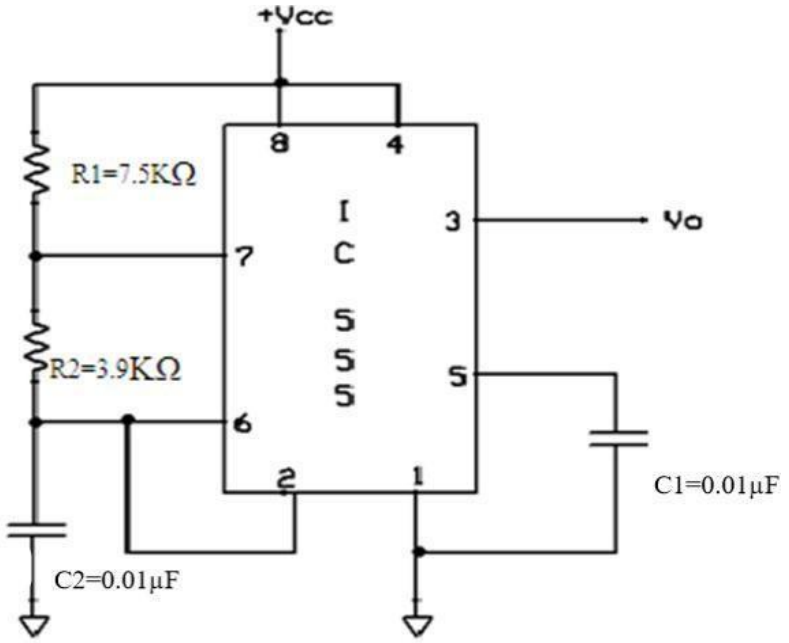
RESULT:

Thus RC Phase Shift Oscillator were designed and tested using op-amp IC 741.

PIN DIAGRAM:



CIRCUIT DIAGRAM OF ASTABLE MULTIVIBRATOR:



DESIGN:

Given f= 11.11 KHz and duty cycle = 23%
Therefore, Total time period, $T = 1/f = 90 \times 10^{-6}$ s

We know, duty cycle = t_d / T

$23 / 100 = t_d / 90 \times 10^{-6}$, $t_d = 0.23 \times 90 \times 10^{-6}$

Therefore, $t_d = 20.7 \times 10^{-6}$ s
and $t_c = T - t_d = 90 \times 10^{-6} - 20.7 \times 10^{-6} = 69.3 \times 10^{-6}$ s

We also know for an astable multivibrator $t_d = 0.69 (R_2) C$

Ex. No: 11
Date:

555- TIMER IC BASED ASTABLE MULTIVIBRATOR

AIM:

To design an astable multivibrator circuit for the given specifications using 555 Timer IC.

APPARATUS REQUIRED:

S. No	Name of the Apparatus	Range	Quantity
1.	Function Generator	3 MHz	1
2.	CRO	30 MHz	1
3.	Dual RPS	(0 – 30) V	1
4.	Timer IC	IC 555	1
5.	Bread Board	-	1
6.	Resistors	7.5KΩ, 3.9 KΩ	Each 1
7.	Capacitors	0.01µ F	2
8.	Connecting wires and probes	Single Strand Wire	As required

THEORY:

An astable multivibrator, often called a free-running multivibrator, is a rectangular- wave-generating circuit. This circuit do not require an external trigger to change the state of the output. The time during which the output is either high or low is determined by two resistors and a capacitor, which are connected externally to the 555 timer. The time during which the capacitor charges from 1/3 V_{cc} to 2/3 V_{cc} is equal to the time the output is high and is given by,

$$t_c = 0.69 (R_1 + R_2) C$$

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

$$t_d = 0.69 (R_2) C$$

Thus the total time period of the output

waveform is, $T = t_c + t_d = 0.69 (R_1 + 2 R_2) C$

The term duty cycle is often used in conjunction with the astable multivibrator. The duty cycle is the ratio of the time t_c during which the output is high to the total time period T. It is generally expressed in percentage. In equation form,

$$\% \text{ duty cycle} = [R_2 / (R_1 + 2 R_2)] \times 100\% \text{ or } t_d / t_c \times 100\%$$

Assume C = 0.01 x 10^{-6} F,
$R_2 = t_d /(0.69 \times C) = 20.7 \times 10^{-6} / (0.69 \times 0.01 \times 10^{-6})$
Therefore, $R_2 = 3K\Omega$

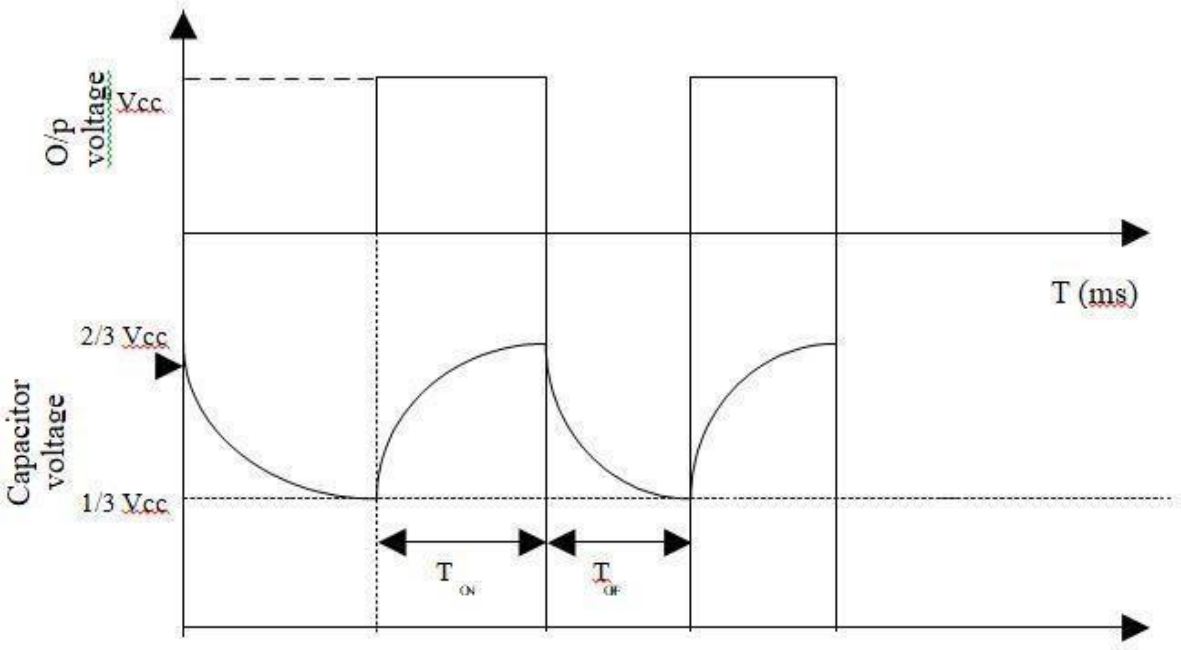
$t_c = 0.69 (R_1 + R_2) C$

$R_1 = (t_c / (0.69 \times C)) – R_2$

$R_1 = (69.3 \times 10^{-6} / (0.69 \times 0.01 \times 10^{-6})) - 3000$

Therefore, $R_1 = 7\ K\Omega \approx 6.8\ K\Omega \approx 7.5\ K\Omega$

MODEL GRAPH:



OBSERVATIONS:

S.No	Waveforms	Amplitude (No. of div x Volts per div)	Time period (No. of div x Time per div)	
			t_c	t_d
1.	Output Voltage , V_o			
2.	Capacitor voltage , V_c			

PROCEDURE:

1. Connections are given as per the circuit diagram.
2. + 5V supply is given to the + V_{cc} terminal of the timer IC.
3. At pin 3 the output waveform is observed with the help of a CRO
4. At pin 6 the capacitor voltage is obtained in the CRO and the V_0 and V_c voltage waveforms are plotted in a graph sheet.

DISCUSSION QUESTIONS:

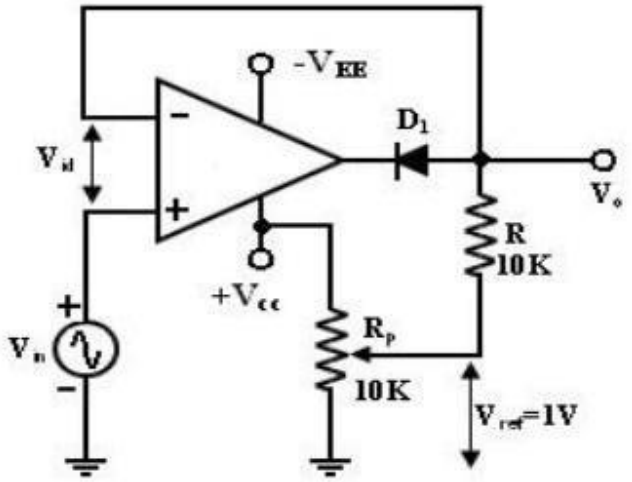
1. Define Offset voltage.
2. Define duty cycle.
3. Mention the applications of IC555.
4. Give the methods for obtaining symmetrical square wave.
5. What is the other name for monostable multivibrator?
6. Explain the operation of IC555 in astable mode..
7. Why negative pulse is used as trigger?

RESULT:

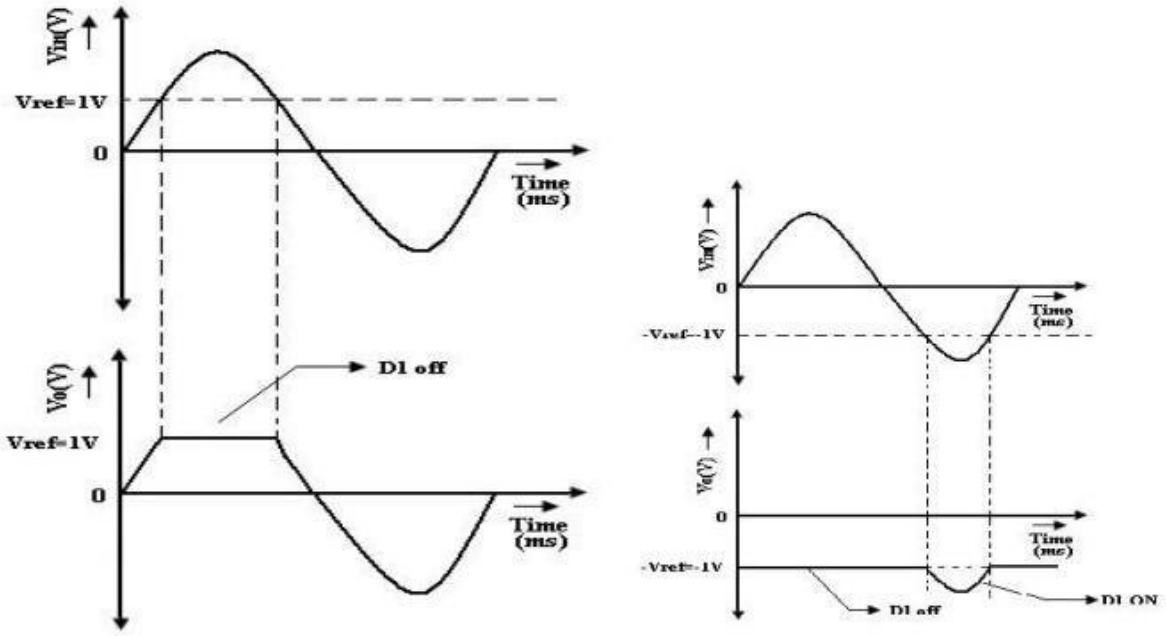
The design of the Astable multivibrator circuit was done and the output voltage and capacitor voltage waveforms were obtained.

Positive Clipping Circuit:

Circuit Diagram:



Waveforms:



Ex. No: 12
Date:

OP AMP BASED PRECISION RECTIFIER CIRCUIT/CLIPPER CIRCUITS

AIM:

To design and test Op-Amp based clipping circuits for peak clipping.

COMPONENTS REQUIRED:

-Power Supply
-IC741
-Diodes IN4007or BY127
-Resistors 10k(2 No.)

PROCEDURE:

1. Make the Connections as shown in the circuit diagram
2. Apply sinusoidal input Vi of 1 KHz and of amplitude 8V P-P to the circuit
3. Observe the output signal in the CRO and verify it with given waveforms.
4. Apply Vi and Vo to the X and Y channel of CRO and observe the transfer characteristic waveform and verify it.

To find the value of R:

Given: $R_f = 100\Omega$, $R_r = 100K\Omega$

R_f - Diode forward resistance

R_r - Diode reverse resistance

$R = \sqrt{R_f R_r} = \sqrt{100 \times 100 \times 10^3} \approx 3.16K\Omega$

Choose R as 10 KΩ

Let the output voltage be clipped at +3V

∴ $V_{omax} = 3V$

From the circuit diagram,

$V_{omax} = V_r + V_{ref}$

Where V_r is the diode drop = 0.6V

∴ $V_{ref} = V_{omax} - V_r$

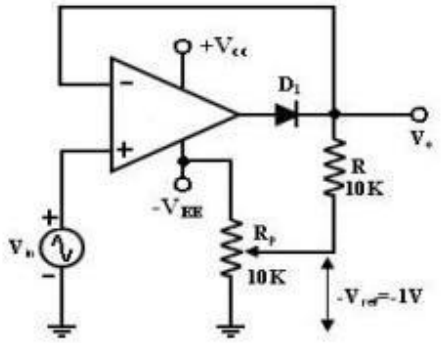
$= 3 - 0.7$

$V_{ref} = 2.3$ V

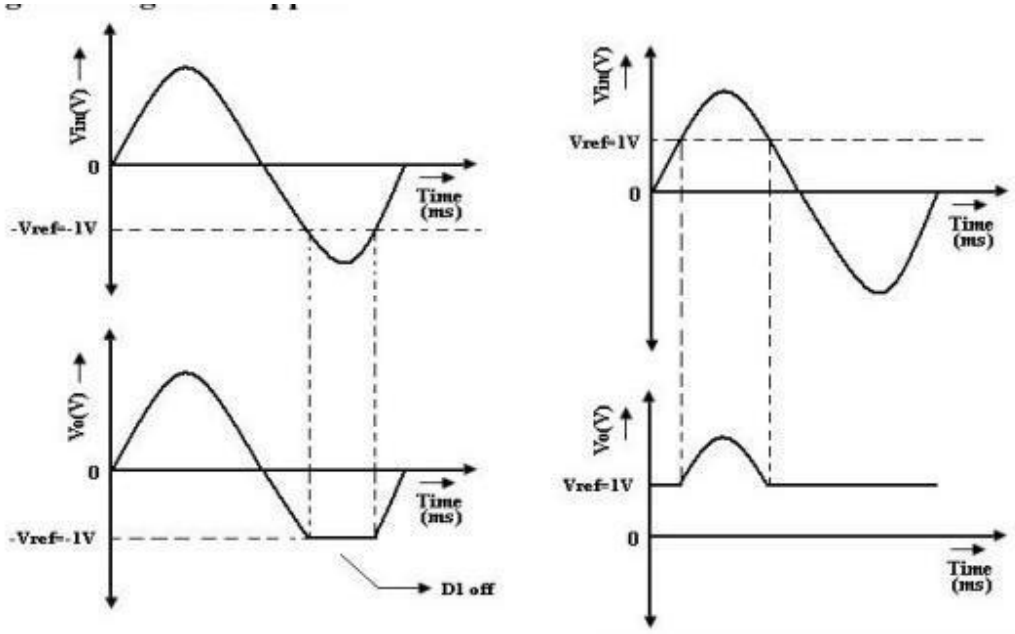
Negative clipping

circuit: Circuit

diagram:



Waveforms:



Let the output voltage be clipped at -3V
Vomin = -3V
Vomin = -Vr+Vref
Vref = Vomin+Vr = -3 + 0.7
Vref = -2.3V

Let the output voltage be clipped at 2V

$V_{omax} = V_{ref} = 2V$

Let the output voltage be clipped at -2V

$V_{omin} = V_{ref} = -2V$

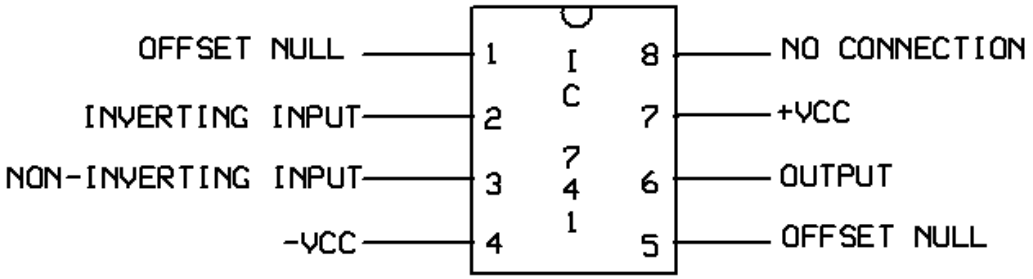
DISCUSSION QUESTIONS:

1. What is clipping circuit?
2. Give the types of clipping circuit.

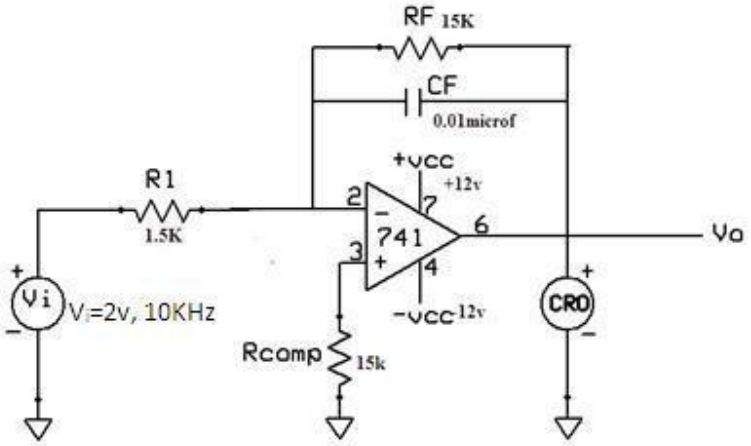
RESULT:

Thus the clipping circuits are designed and its performace are verified.

PIN DIAGRAM:



CIRCUIT DIAGRAM OF INTEGRATOR:



DESIGN:

Given f_a = 10KHz

We know the frequency at which the gain is 0 dB,
$f_a = 1 / (2\pi R_1 C_f)$
Therefore f_b = 10KHz Since $f_b = 0.1 f_a$, and
also the gain limiting frequency $f_b = 1 / (2\pi R_f C_f)$, assume $C_f = 0.01\mu F$

$R_f = 1 /(2\pi (1000)(0.01 \times 10^{-6}) = 15.9\ K\Omega \approx 15K\Omega$

$R_1 = 1/(2 \pi (10000) (0.01 \times 10^{-6}) = 1.59\ K\Omega \approx 1.5K\Omega$

Additional experiment

Ex. No: 13
Date:

INTEGRATOR

AIM:

To design an Integrator circuit for the given specifications using Op-Amp IC 741.

APPARATUS REQUIRED:

S.No	Name of the Apparatus	Range	Quantity
1.	Function Generator	3 MHz	1
2.	CRO	30 MHz	1
3.	Dual RPS	0 – 30 V	1
4.	Op-Amp	IC 741	1
5.	Bread Board		1
6.	Resistors	1.5KΩ	1
7.	Resistors	15KΩ	2
8.	Capacitors	0.01µ F	1
9.	Connecting wires and probes	As required	

THEORY:

A circuit in which the output voltage waveform is the integral of the input voltage waveform is the integrator. Such a circuit is obtained by using a basic inverting amplifier configuration if the feedback resistor R_f is replaced by a capacitor C_f. The expression for the output voltage is given as,

$$V_o = - (1/R_f C_1) \int V_i \, dt$$

Here the negative sign indicates that the output voltage is 180^0 out of phase with the input signal. Normally between f_a and f_b the circuit acts as an integrator. Generally, the value of $f_a > f_b$. The input signal will be integrated properly if the Time period T of the signal is larger than or equal to $R_f C_f$. That is,

$$T \geq R_f$$
$$C_f$$

The integrator is most commonly used in analog computers and ADC and signal-wave shaping circuits.

PROCEDURE:

1. Connections are given as per the circuit diagram.
2. $+ V_{cc}$ and $- V_{cc}$ supply is given to the power supply terminal of the Op-Amp IC.
3. By adjusting the amplitude and frequency knobs of the function generator, appropriate input voltage is applied to the inverting input terminal of the Op-Amp.
4. The output voltage is obtained in the CRO and the input and output voltage waveforms are plotted in a graph sheet.

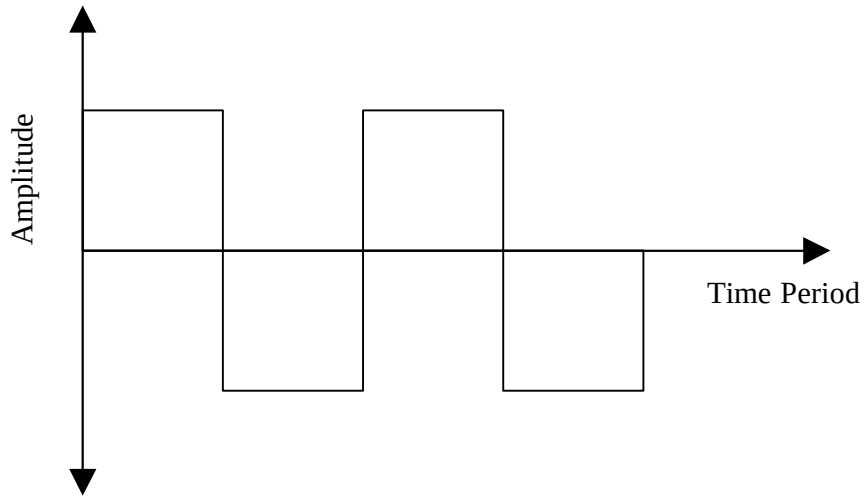
OBSERVATIONS:

S.No.	Amplitude (No. of div x Volts per div)	Time period (No. of div x Time per div)
Input		
Output		

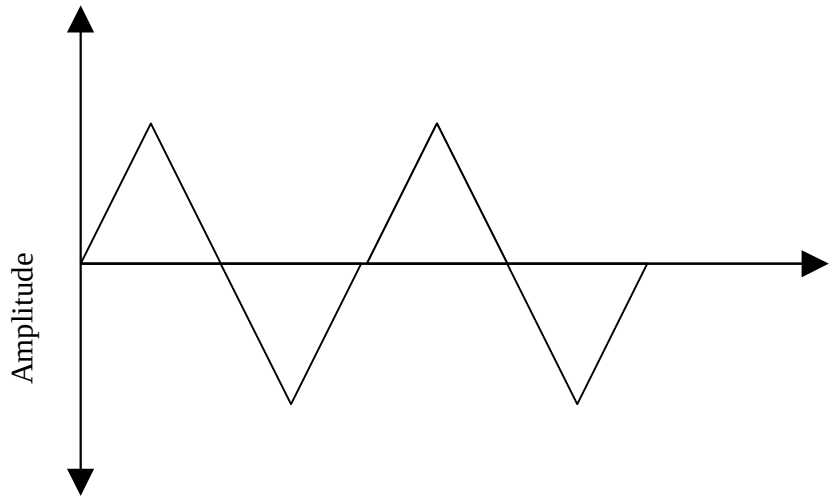
MODEL GRAPH:

INTEGRATOR:

INPUT SIGNAL:



OUTPUT SIGNAL:



DISCUSSION QUESTIONS:

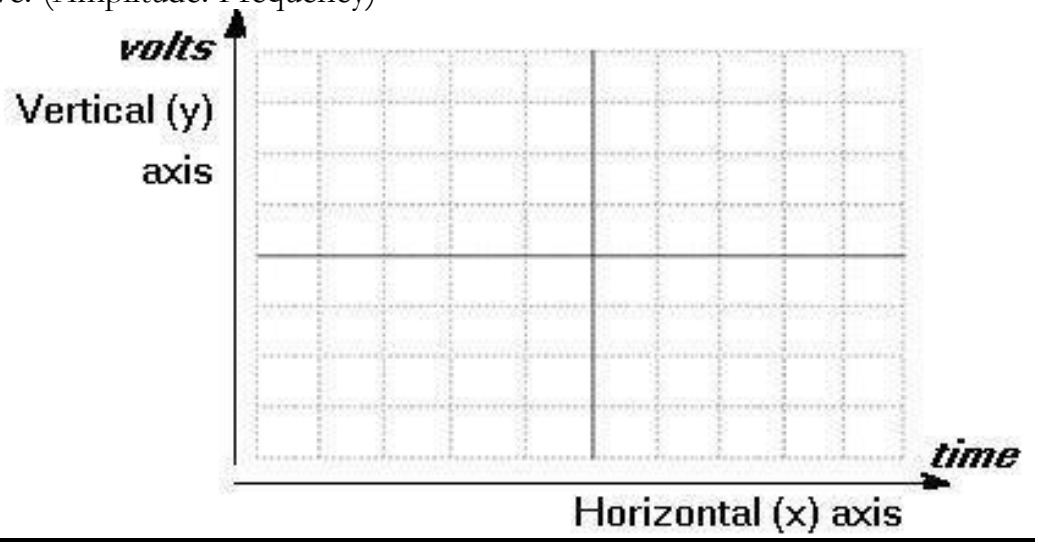
1. What is integrator?
2. Write the disadvantages of ideal integrator?
3. Write the application of integrator?
4. Why compensation resistance is needed in integrator and how will you find it values?
5. What is differentiator?
6. Write the disadvantages of ideal differentiator.
7. Write the application of differentiator?
8. Why compensation resistance is needed in differentiator and how will you find it values?
9. Why integrators are preferred over differentiators in analog comparators?

RESULT:

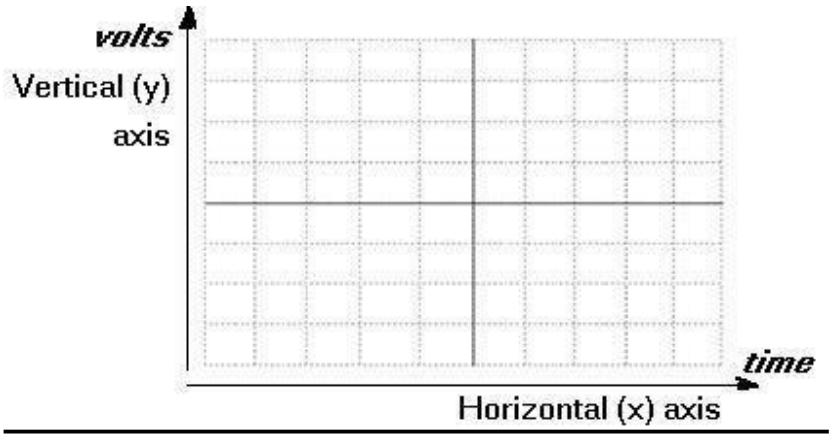
The design of the Integrator circuit was done and the input and output waveforms were obtained.

Draw observed waveforms

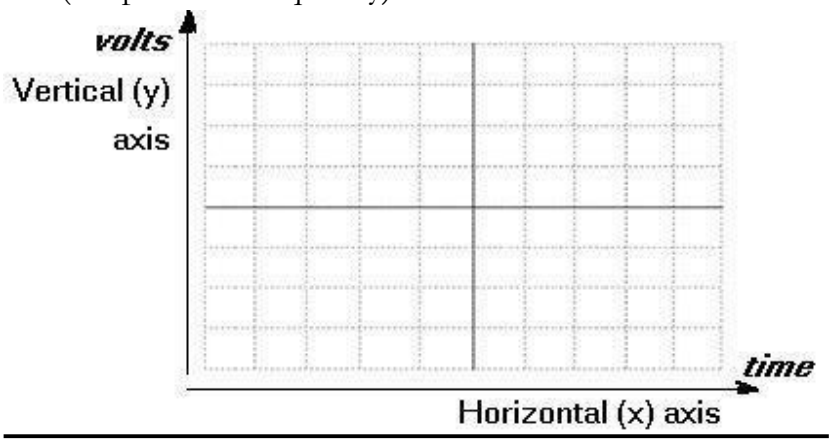
Sine wave: (Amplitude: Frequency)



Square Wave: (Amplitude: Frequency)



Triangular Wave: (Amplitude: Frequency)



Additional experiment

Ex. No: 14

Date:

STUDY OF CRO FOR FREQUENCY AND PHASE MEASUREMENTS

Aim:

To observe sine wave, square wave and triangular waveforms on the Cathode Ray Oscilloscope and to measure amplitude, frequency and Phase of the waveforms.

APPARATUS REQUIRED:

Sl. No	Components / Equipments	Specifications	Quantity
1	CRO-Cathode Ray Oscilloscope	20 MHz	1
2	Function Generator	1MHz	1
3	BNC CRO Probes		3

THEORY:

CRO-(Cathode Ray Oscilloscope) is the instrument which is used to observe signal waveforms. Signals are displayed in time domain i.e. variation in amplitude of the signal with respect to time is plotted on the CRO screen. X-axis represents time and Y-axis represents amplitude. It is used to measure amplitude, frequency and phase of the waveforms. It is also used to observe shape of the waveform. C.R.O. is useful for troubleshooting purpose. It helps us to find out gain of amplifier, test oscillator circuits.

A CRO is a versatile instrument that can be used to measure voltage, time intervals, and the phase angle between two sinusoidal voltages of the same frequency. There are 8 vertical divisions and 10 horizontal divisions indicated with grid lines or graticules. A standard screen size is 8 cm by 10 cm. The screen is coated with phosphor that emits light when struck by the electron beam. We can measure following parameters using the CRO:

AC or DC voltage.
Time (t=1/f).
Phase relationship
Waveform calculation: Rise time; fall time; on time; off-time Distortion, etc.

Latest digital storage oscilloscope display voltage and frequency directly on the LCD and does not require any calculations. It can also store waveform for further analysis.

Major blocks:

Cathode ray tube (CRT)
- Electron gun assembly
- Deflection plate unit
- Screen.

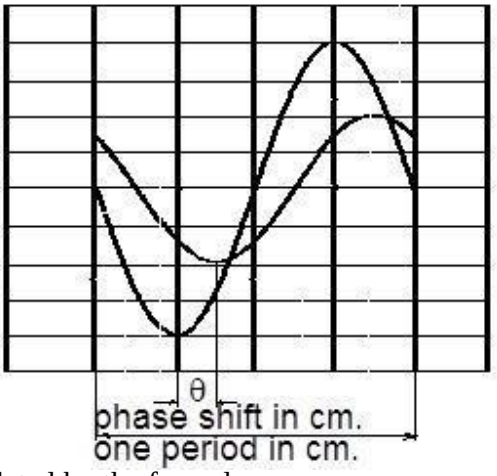
Vertical amplifier
Horizontal amplifier
Sweep generator
Trigger circuit
Associated power supply.

FORMULA:

Amplitude = No. of vertical divisions * Volts/div. Time period
= No. of horizontal divisions * Time/div. Frequency= (1/T) Hz

Measurement of Phase

The calibrated time scales can be used to calculate the phase shift between two sinusoidal signals of the same frequency. If a dual trace or beam CRO is available to display the two signals simultaneously (one of the signals is used fo r synchronization), both of the signals will appear in proper time perspective and the amount of time difference between the waveforms can be measured. This, in turn can be utilized to calculate the phase angle θ, between the two signals.



the phase shift can be calculated by the formula
θ=(Phase shift in cm/ One period in cm .)*360^{o}
The frequency relationship between the horizontal and vertical inputs is given by
F_h/f_v= (No. of tangencies (vertical)/ No. of tangencies (horizontal))
from which fv, the unknown frequency can be calculated.

PROCEDURE:

1. Connect function generator output at the input of C.R.O. at channel 1 or at channel 2.
2. Select proper channel i.e. if signal is connected to channel 1 select CH1 and if signal is connected to channel 2 select CH2.
3. Adjust Time /Div. knob to get sufficient time period displacement of the wave on the CRO screen.
4. With fine tuning of time/Div. make the waveform steady on screen.
5. Use triggering controls if waveform is not stable.
6. Keep volt/div knob such that waveform is visible on the screen without clipping.
7. Measure P-P reading along y-axis. This reading multiplied with volt/div gives peak to peak amplitude of the ac i/p wave.
8. Measure horizontal division of one complete cycle. This division multiplied by time/div gives time period of the i/p wave.
9. Calculate frequency using formula f = 1/T.
10. Note down your readings in the observation table.
11. Draw waveforms of sine, square, ramp and triangular in the given space.

REVIEW QUESTIONS:

1. What is the use of CRO & mention the Manufactures.
2. Define the terms offset error, peak value and peak to peak value.
3. What is the highest frequency that can be measured by CRO available in your laboratory?
4. What is highest voltage that can be measured by CRO available in your laboratory?
5. What you will do to measure voltage which is greater than voltage limit of the CRO?
6. What do you mean by dual channel CRO?
7. What type of deflection mechanism used in CRO to deflect electron beam?
8. How to test whether CRO probe is in working condition or not?
9. How do you measure the frequency and phase angle in CRO?
10. What is the use of AC/DC input coupling push-button switch, Volt/Div. and Time/Div. knob in CRO?

RESULT:

Thus the operation of CRO has been studied along with the measurement of frequency and phase of a signal.