POORNIMA
COLLEGE OF ENGINEERING

Department of Electrical Engineering

<u>Analog Electronics Lab Manual</u>

Year: 2ndYr. /III SEM

Lab Code: - 3EE4-21

INDEX

POORNIMA COLLEGE OF ENGINEERING, JAIPUR

DEPARTMENT OF ELECTRICAL ENGINEERING

VISION

To be a model of excellence in Professional Education and Research by creating electrical engineers who are prepared for lifelong engagement in the rapidly changing fields and technologies with the ability to work in team.

MISSION

- ✓ To provide a dynamic environment of technical education wherein students learn in collaboration with others to develop knowledge of basic and engineering sciences.
- ✓ To identify and strengthen current thrust areas based upon informed perception of global societal issues in the electrical and allied branches.
- ✓ To develop human potential with intellectual capability who can become a good professional, researcher and lifelong learner.

POORNIMA COLLEGE OF ENGINEERING, JAIPUR
DEPARTMENT OF ELECTRICAL ENGINEERING

PROGRAM EDUCATIONAL OBJECTIVES (PEO's)

PEO 1: Graduates will have the ability to formulate, analyze and apply design process using the basic knowledge of engineering and sciences to solve complex electrical engineering problems.

PEO 2: Graduates will exhibit quality of leadership, teamwork, time management, with a commitment towards addressing societal issues of equity, public and environmental safety using modern engineering tools.

PEO 3: Graduates will possess dynamic communication and have successful transition into a broad range of multi-disciplinary career options in industry, government and research as lifelong learner.

PROGRAM OUTCOMES (PO's)

Engineering Graduates will be able to:

1. **Engineering knowledge**: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
2. **Problem analysis**: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
3. **Design/development of solutions**: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4. **Conduct investigations of complex problems**: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.

5. **Modern tool usage**: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6. **The engineer and society**: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7. **Environment and sustainability**: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8. **Ethics**: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and team work**: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10. **Communication**: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11. **Project management and finance**: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one’s own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
12. **Life-long learning**: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES (PSO's)

PSO1: Graduate possesses the ability to apply fundamental knowledge of basic sciences, mathematics and computation to solve the problems in the field of electrical engineering for the benefit of society.

PSO2: Graduate possesses the ability to professionally communicate and ethically solve complex electrical engineering problems using modern engineering tools.

PSO3: Graduate possesses sound fundamental knowledge to be either employable or develop entrepreneurship in the emerging areas of renewable and green energy, electric and hybrid vehicles and smart grids and shall be susceptive to life- long learning.

LAB OUTCOMES

LO1: Analyze the possible causes of discrepancies between experimental observations and theoretical results in amplifier, regulator and oscillator circuits. [ANALYZE].

LO2: Measure the output response of various amplifier, regulator and oscillator circuits on a circuit design software tool. [MANIPULATION]

LO3: Contribute efficiently in a team so as to achieve the desired response of amplifier, regulator and oscillator circuits. [RESPONDING]

LO4: Demonstrate the solution to a problem on amplifier, regulator and oscillator circuits. [PRECISION]

MAPPING OF LO WITH PO

LO	Lab outcome	PO											
		1	2	3	4	5	6	7	8	9	10	11	12
1	Student will able to **Analyze** the possible causes of discrepancies between experimental observations and theoretical results in amplifier, regulator and oscillator circuits. **[ANALYZE]**.	**-**	**3**	**-**	**-**	**-**	**-**	**-**	**-**	**-**	**-**	**-**	**-**
2	Student will able to **Measure** the output response of various amplifier, regulator and oscillator circuits on a circuit design software tool. **[MANIPULATION]**	**-**	**-**	**-**	**-**	**2**	**-**	**-**	**-**	**-**	**-**	**-**	**-**
3	Student will able to **Contribute** efficiently in a team so as to achieve the desired response of amplifier, regulator and oscillator circuits. **[RESPONDING]**	**-**	**-**	**-**	**-**	**-**	**-**	**-**	**-**	**2**	**-**	**-**	**-**
4	Student will able to **Demonstrate** the solution to a problem on amplifier, regulator and oscillator circuits. **[PRECISION]**	**-**	**-**	**-**	**-**	**-**	**-**	**-**	**-**	**-**	**2**	**-**	**-**

MAPPING OF LO WITH PSO

LO	Lab outcome	Pso1	Pso2	Pso3
1	Student will able to **Analyze** the possible causes of discrepancies between experimental observations and theoretical results in amplifier, regulator and oscillator circuits. **[ANALYZE]**.	**2**	-	-
2	Student will able to **Measure** the output response of various amplifier, regulator and oscillator circuits on a circuit design software tool. **[MANIPULATION]**	-	**3**	-
3	Student will able to **Contribute** efficiently in a team so as to achieve the desired response of amplifier, regulator and oscillator circuits. **[RESPONDING]**	-	**2**	-
4	Student will able to **Demonstrate** the solution to a problem on amplifier, regulator and oscillator circuits. **[PRECISION]**	**2**	-	-

LAB RULES

Always:

- Be punctual.
- Maintain decorum in the lab.
- Utilize lab hours for experiments.
- Bring lab records as per their lab schedule.

Never:

- Don’t bring any external material in the lab, except your lab record, copy and books.
- Don’t bring the mobile phones in the lab. If necessary then keep them in silence mode.
- Please be considerate of those around you, especially in terms of noise level.

SAFETY MEASURES

• First aid box and fire extinguishers are kept in each laboratory.

• Well trained technical supporting staff monitor the labs at all times.

• Damaged equipment's are identified and serviced at the earliest.

• Periodical calibration of the lab equipment's are regularly done

• A clean and organized laboratories are maintained

• The use of cell phones is prohibited.

• Appropriate storage areas are available.

• In order to create more space in the laboratories, a separate section has racks to store the belongings of the students.

LIST OF EXPERIMENTS

Max. Marks=50

1. Plot gain-frequency characteristics of BJT amplifier with and without negative feedback in the emitter circuit and determine bandwidths, gain bandwidth products and gains at 1 kHz with and without negative feedback.
2. Study of series and shunt voltage regulators and measurement of line and load regulation and ripple factor.
3. Plot and study the characteristics of small signal amplifier using FET.
4. Study of push pull amplifier. Measure variation of output power & distortion with load.
5. Study Wein bridge oscillator and observe the effect of variation in R & C on oscillator frequency.
6. Study transistor phase shift oscillator and observe the effect of variation in R & C on oscillator frequency and compare with theoretical value.
7. Study the following oscillators and observe the effect of variation of C on oscillator frequency: (a) Hartley (b) Colpitts.
8. To plot the characteristics of UJT and UJT as relaxation.

EVALUATION SCHEME

Name Of Exam	Conducted By	Experiment Marks	Viva Marks	Total
I Mid Term	PCE	15	5	20
II Mid Term	PCE	15	5	20
End Term	RTU	15	5	20

Name Of Exam	Conducted By	Performance Marks	Attendance Marks	Total
Sessional	PCE	15	5	20

DISTRIBUTION OF LAB RECORD MARKS PER EXPERIMENT

Attendance	Record	Performance	Total
2	3	5	10

LAB PLAN

Total number of experiment: 11

Total number of turns required: 12

NUMBER OF TURNS REQUIRED FOR

Experiment Number	Turns	Scheduled Day
Zero Lab	1	Turn 1
Exp. 1	1	Turn 2
Exp. 2	1	Turn 3
Exp. 3	1	Turn 4
Exp. 4	1	Turn 5
Exp. 5	1	Turn 6
Exp. 6	1	Turn 7
Exp. 7	1	Turn 8
Exp. 8	1	Turn 9
Exp. 9	1	Turn 10
Exp. 10	1	Turn 11
Exp. 11	1	Turn 12

DISTRIBUTION OF LAB HOURS

- Explanation of Experiment & Logic : 20 Minutes
- Performing the Experiment : 40 Minutes
- File Checking : 30 Minutes
- Viva/Quiz : 20 Minutes
- Solving of Queries : 10 Minutes

ROTOR PLAN

Rotor I

1. Plot gain-frequency characteristics of BJT amplifier with and without negative feedback in the emitter circuit and determine bandwidths, gain bandwidth products and gains at 1 kHz with and without negative feedback.
2. Study Wien bridge oscillator and observe the effect of variation in R & C on oscillator frequency.
3. Study transistor phase shift oscillator and observe the effect of variation in R & C on oscillator frequency and compare with theoretical value.
4. Study the following oscillators and observe the effect of variation of C on oscillator frequency: (a) Hartley (b) Colpitts.
5. **To design the following circuits, assemble these on bread board and test them. Simulation of these circuits with the help of appropriate software**. **[BEYOND SYLLABUS]**
    a. **Op-Amp in inverting and non-inverting modes.**
    b. **Op-Amp as scalar, summer and voltage follower.**

Rotor II

6. Plot and study the characteristics of small signal amplifier using FET.
7. Study of push pull amplifier. Measure variation of output power & distortion with load.
8. To plot the characteristics of UJT and UJT as relaxation
9. Study of series and shunt voltage regulators and measurement of line and load regulation and ripple factor.
10. **To plot the characteristics of MOSFET and CMOS.** **[BEYOND SYLLABUS]**
11. **To design the following circuits, assemble these on bread board and test them. Simulation of these circuits with the help of appropriate software**. **[BEYOND SYLLABUS]**
    a. **Op-Amp as differentiator and integrator**

ZERO LAB

Introduction to Lab:

a). Relevance to Branch:

Electronics & Communication branch deals with use of electronic circuitry in communication

System which uses many different kind of electronic devices i.e. analog and digital. So this Subject tells us about analog electronic devices, their construction, working and application

b). Relevance to Society:

In our daily needs we confront with many electronic devices of analog nature like amplifiers Signal generators, and many others. So this subject helps us to get in the depth of their working.

And this innovation will help in bringing a tremendous change in the society.

c). Relevance to Self:

This subject moves us to the depth of knowledge. If one is interested in the practical applications then this is useful like in further studies, in projects. As this subject brushes up the thinking power so we come up to the most optimum utilization of the things and subjects.

.*d)* Pre- Requisites (Connection with previous year): -

1. Electronics' Device & Circuit (3EE1A)
2. Analog Electronics (4EE1A)
3. Power Electronics (5EE1A)

As in previous semester, the concepts learnt in Electronics Devices and Circuit was related to build the basics of the students. They understood how to solve any complex circuit and simplify them and get the same results as that of the complex one. Students will be able to understand electronic circuits and how they can be designed with components such as operational amplifiers and transistors, tuned amplifier also able to individually dimension, simulate, build and test Low Frequency and High Frequency. This subject is the basic building block of electrical engineering enabling students in easy grasping of various electronics concepts.

EXPERIMENT NO: 1 (BJT AMPLIFIER)

AIM: - Plot gain-frequency characteristics of BJT amplifier with and without negative feedback in the emitter circuit and determine bandwidths, gain bandwidth products and gains at 1 kHz with and without negative feedback.

APPARATUS REQUIRED

Transistor (BC107), Resistor, Capacitor, AFO, CRO and RPS

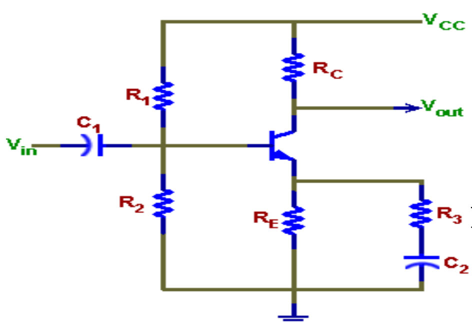
THEORY

The circuit diagram of CE Amplifier with current series feedback is shown below. The resistor RF in emitter is the feedback element. The voltage drop V_f across RF constitutes the feedback signal while the current I_c forms the sampled signal. Hence, this forms a current series feedback. Due to negative feedback, though the voltage gain of the amplifier is decreased, it improves stability and increases the bandwidth. This is the advantage of negative feedback. Using h-parameter model for ac analysis the amplifier parameters such as the voltage gain, bandwidth can be calculated. For this, following steps have to be followed.

1) To find the input circuit, set I_0=0, i.e. open the output loop. Hence RE appears in input side.

2) To find the output circuit set I_1=0, i.e. open the input loop. Hence RE appears in output loop.

CIRCUIT DIAGRAM OF CURRENT-SERIES FEEDBACK AMPLIFIER:-



PROCEDURE:-

Frequency Response of Current- Series Feedback Amplifier

1. Connect the circuit as shown in the figure.
2. Connect a sine- wave generator set at 1000Hz frequency and 50mV peak-to-peak signal voltage at the input of the amplifier circuit.
3. Connect an oscilloscope across the output nodes. Observe the sine wave output on the oscilloscope.
4. Adjust the output of the sine-wave generator until undistorted. Maximum signal output is obtained.

5. Observe and measure the peak-to-peak amplitude of input and output signal and record the values in the tabulation provided.
6. Now, sweep the input signal frequency in the range 30Hz to 1 MHz by adjusting the sine wave generator output.
7. For each setting of input frequency, measure the output signal voltage.
8. Draw the frequency response curve on a semi-log graph sheet. From this plot, obtain the values of mid-band voltage gain, upper and lower cut-off frequency and BW (f_h-f_l).

Frequency Response of Amplifier without Negative Feedback

1. Remove R_f from the circuit and connect RE and CE directly to the emitter terminal.
2. Measure and record in the table, the frequency response of this circuit without R_f by repeating steps 5 through 6.
3. Draw the response curve on the same graph as before. Obtain the values of mid-band voltage gain, lower and upper cut-off frequency and BW. Comment on the differences between this response curve and the previous curve.

TABULATION 1

Measurement of frequency response of current series feedback amplifiers

V_{in}= 50 mV

Frequency (in Hz)	V0 (Volts)	Gain= V0/Vin	Gain (dB) = 20 log(Vo/Vin)

TABULATION 2

Measurement of frequency response of amplifier without feedback

V_{in}= 50 mV

Frequency (in Hz)	V0 (Volts)	Gain= Vo/Vin	Gain dB= 20 log(Vo/Vin)

RESULT:-The current series feedback amplifier was designed, constructed and its frequency response was plotted.

<u>VIVA QUESTIONS:</u>

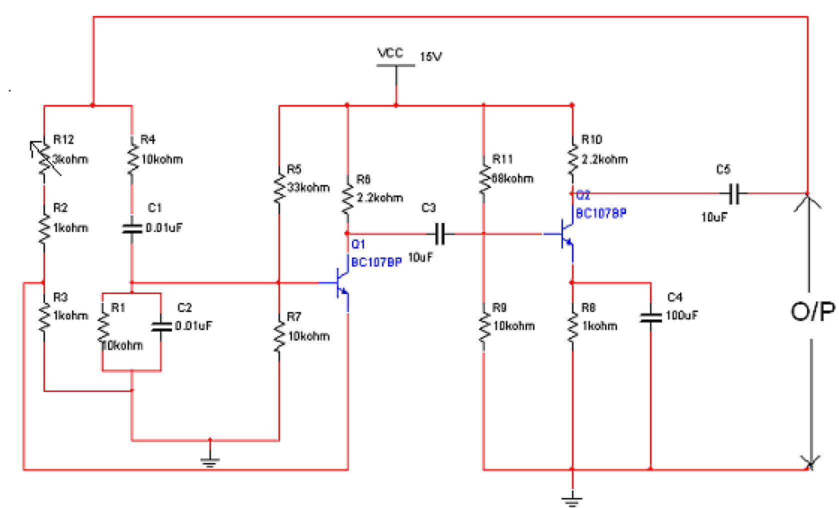
1. What are different types of feedback amplifier?
2. What is the phase shift between input and output in negative feedback amplifier?
3. In negative feedback amplifier gain and Bandwidth is increase or decreases.
4. Where used positive feedback amplifier.
5. What is mean by feedback?
6. Define the feedback factor β?
7. How does an oscillator differ from an amplifier?
8. Overall performance of an amplifier circuit can be improved by
9. In a degenerative feedback amplifier the phase difference between the input signal and feedback signal is
10. In amplifiers, the feedback is negative if

EXPERIMENT NO: 2 (WEIN BRIDGE OSCILLATOR)

AIM: - Study Wein bridge oscillator and observe the effect of variation in R & C on oscillator frequency.

APPARATUS:

Transistor	(BC 107)	2 No
Resistors	10K Ω	4 No
	1 K Ω	3 No
	2.2 K Ω	2 No
	33 K Ω	
	6.8 K Ω	
Capacitors	10 μF	2 No
	100 μF	
	0.01 μF	2 No
RPS	(0 – 30 V)	
Potentiometer		
Bread Boar		
CRO		
Connecting wires		



CIRCUITDIAGRAM:

THEORY:

The wein bridge oscillator is a standard circuit for generating low frequencies in the range of 10 Hz to about 1MHz.The method used for getting positive feedback in wein bridge oscillator is to use two stages of an RC-coupled amplifier. Since one stage of the RC-coupled amplifier introduces a phase shift of 180 deg, two stages will introduces a phase shift of 360 deg. At the frequency of oscillations *f* the positive feedback network shown in fig makes the input & output in the phase. The frequency of oscillations is given as

$$f = 1/2\pi\sqrt{R_1C_1R_2C_2}$$

In addition to the positive feedback

PROCEDURE:

1. Connections are made as per the circuit diagram
2. Feed the output of the oscillator to a C.R.O by making adjustments in the Potentiometer connected in the +ve feedback loop, try to obtain a stable sine Wave.
3. Measure the time period of the waveform obtained on CRO. & calculate the Frequency of oscillations.
4. Repeat the procedure for different values of capacitance.

OBSERVATION:

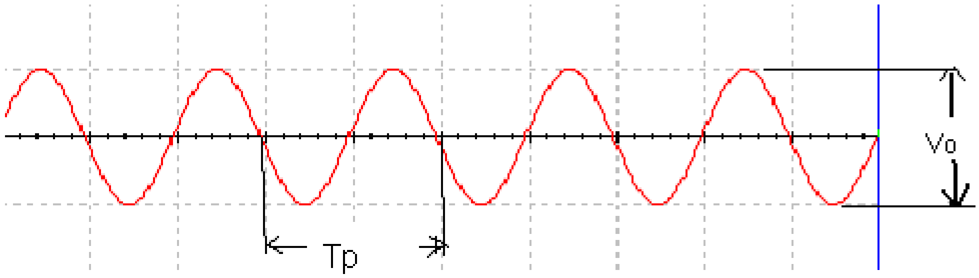
Given R=10kΩ, C=0.01μF

f_T = 1/ 2πRC

$f_P = \dfrac{1}{T} =$

Amplitude, V_0 =

MODEL WAVE FORM:



RESULT:

The frequency of the wein bridge oscillator is calculated and is verified

VIVA 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 Wein bridge oscillator?
5. which type of feedback is used in Wein bridge oscillator
6. What is gain of Wein bridge oscillator?
7. What are the application of Wein bridge oscillator
8. What is the condition for oscillations?
9. What is the difference between damped oscillations undammed Oscillations?
10. Wein bridge oscillator is either LC or RC oscillator.

EXPERIMENT NO: 3 (RC PHASE SHIFT OSCILLATOR)

AIM: - Study transistor phase shift oscillator and observe the effect of variation in R & C on oscillator frequency and compare with theoretical value.

COMPONENTS REQUIRED:-

1. Transistor BC 147 =1
2. Resistor 3.3KΩ =4,470Ω =1
3. Capacitor 0.01µF
4. CRO =1

THEORY:-

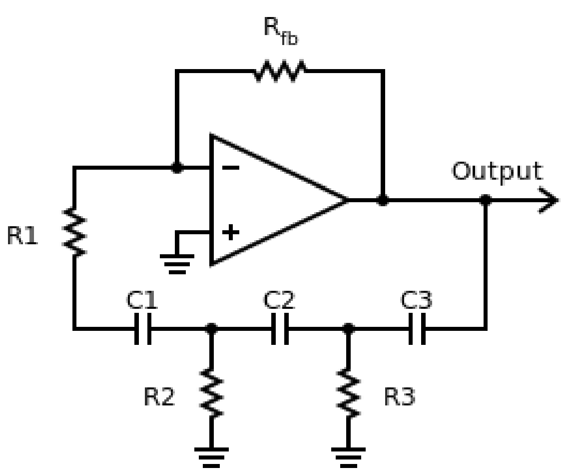
Definition: An Oscillator is an amplifier, which uses positive feedback and without any external input signal, generates an output waveform by energizing the DC signal. An Oscillator is a source of AC voltage. Oscillations are produced in the circuit when Barkhausen Criterion is satisfied.

Barkhausen Criterion

(1) The total phase shift in the closed loop is 0 or 3600.

(2) The magnitude of the loop gain of the amplifier (A) and the feedback factor β is unity. Aβ = 1.The frequency of Oscillation is determined by the frequency selective feedback network (R-C, L-C). In the RC phase shift Oscillator, the cascaded RC networks determine the frequency. It is an Audio frequency Oscillator.

CIRCUIT DIAGRAM OF RC PHASE SHIFT OSCILLATOR:-



Generating signals from 15Hz to 20 KHz. The frequency of Oscillations can be varied over a wide range by ganged tuning the capacitor.

The frequency of oscillation is given by the relation

$f_0 = f_{TH} = 1/ (2\pi RC\sqrt{} (6+(4RC/R))$

Where R = value of resistor in the phase shift network

C = value of capacitor in the phase shift network

For the loop gain to be greater than unity, the current gain of the transistor,

PROCEDURE:-

1. Connections are made as shown in the circuit diagram
2. The DC power supply is switched ON
3. The output waveform is displayed on the CRO
4. The peak to peak Amplitude and time period of the sine wave is noted
5. The graph of output waveform is drawn.

OBSERVATION

Time period T =

Amplitude V P-P =

f_0 = 1/T =

RESULT

Thus an RC phase shift oscillator is designed, constructed and tested.

Frequency of oscillation:-

Theoretical fT=

Practical fP=

Viva Questions:

1. What is an oscillator? List its types?
2. What are RC oscillators? What are its types?
3. What is the phase shift given by each RC section?
5. What is a phase shift oscillator?
6. What is the frequency of RC phase shift oscillator?
7. Why RC oscillators cannot generate high frequency oscillations?
8. What are the applications of RC phase shift oscillators?
9. what phase shift does RC phase shift oscillator produce?
10. Why we need a phase shift between input and output signal?

EXPERIMENT NO: 4 (HARTLEY & COLPITTS OSCILLATOR)

Aim: - Study the following oscillators and observe the effect of variation of C on oscillator frequency: (a) Hartley (b) Colpitts

Apparatus: - n-p-n transistor, Carbon resistors (as shown in circuit), two inductors, capacitors, dc power supply, CRO and connecting terminals.

Formulae: - The frequency of oscillation of the oscillator f=1/2πseq(*LC) Hz*

Where L = L1 + L2 = Resultant inductance of the series combination.

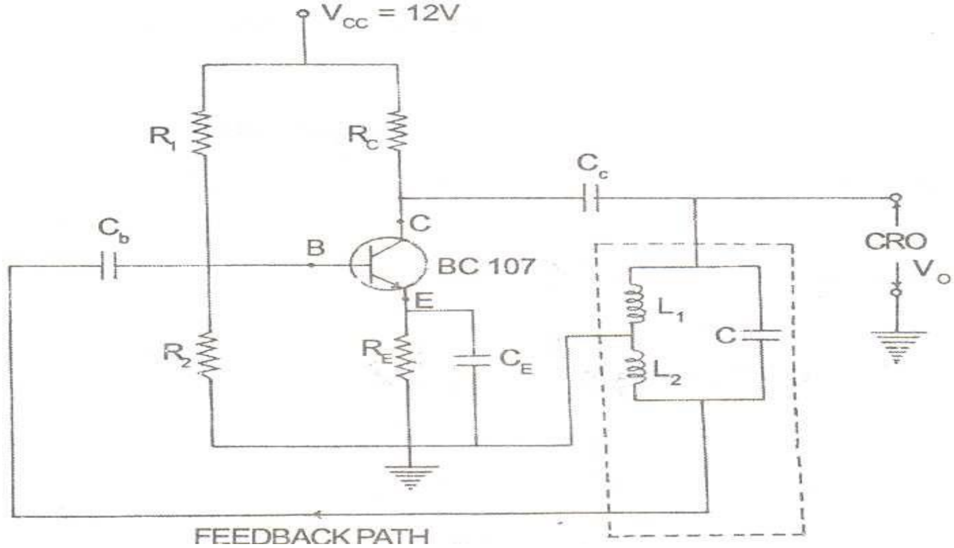
L1, L2 = Self inductances of the two coils (H)

C = Capacitance of the condenser (F)

Description:-The Hartley oscillator is designed for generation of sinusoidal oscillations in the R.F range (20 KHz - 30 MHz). It is very popular and used in radio receivers as a local oscillator. The circuit diagram of Hartley oscillator (parallel or shunt-fed) using BJT is shown in Figure. It consists of an R-C coupled amplifier using an n-p-n transistor in CE configuration. R1 and R2 are two resistors which form a voltage divider bias to the transistor. A resistor RE is connected in the circuit which stabilizes the circuit against temperature variations. A capacitor CE is connected in parallel with RE, acts as a bypass capacitor and provides a low reactive path to the amplified ac signal. The coupling capacitor CC blocks dc and provides an ac path from the collector to the tank circuit. The L-C feedback network (tank circuit) consists of two inductors L1, and L2 (in series) which are placed across a common capacitor C and the centre of the two inductors is tapped as shown in fig. The feedback network (L1, L2 and C) determines the frequency of Oscillation of the oscillator.

THEORY: - When the collector supply voltage Vcc is switched on, collector current starts rising and charges the capacitor C. When this capacitor is fully charged, it discharges through coils L1 and L2, setting up damped harmonic oscillations in the tank circuit. The oscillatory current in the tank circuit produces an a.c. voltage across L1 which *is* applied to the base emitter junction of the transistor and appears in the amplified form in the collector circuit. Feedback of energy from output (collector emitter circuit) to input (base-emitter circuit is) accomplished through auto transformer action. The output of the amplifier is applied across the inductor L1, and the voltage across L2 forms the feedback voltage. The coil L_1, is inductively coupled to coil L_2, and the combination acts as an auto-transformer. This energy supplied to the tank circuit overcomes the losses occurring in it. Consequently the oscillations are sustained in the circuit. The energy supplied to the tank circuit is in phase with the generated oscillations. The phase difference between the voltages across L1 and that across L2 is always 180° because the centre of the two is grounded. A further phase of 180° is introduced between the input and output voltages by the transistor itself.

Thus the total phase shift becomes 3600 (or zero), thereby making the feedback positive or regenerative which is essential for oscillations. So continuous undammed oscillations are obtained.



Procedure:-The circuit is connected as shown in figure. Connect the CRO across the output terminals of the oscillator. Switch on the power supply to both the oscillator and CRO. Select proper values of C, L1 and L2 in the oscillator circuit and get the sine wave form on the screen of CRO. The voltage (deflection) sensitivity band switch (Y-plates) and time base band switch (X-plates) are adjusted such that a steady and complete picture of one or two sine waveform is obtained on the screen. The horizontal length (l) between two successive peaks is noted. When this horizontal length (l), is multiplied by the time base (m) i.e. sec/div , we get the time-period (T = l x m).The reciprocal of the time-period(1/T) gives the frequency (f). This can be verified with the frequency, calculated theoretically by using the above formula. The experiment is repeated by changing C or L1 or L2 or all. The readings are noted in the table given.

Observation table:-

S.No	Capacitance (μF)	Inductance (mH)			Measurement of time period			Frequency (Hz)	
		L_1	L_2	L = L1 + L2	Peak to peak (Horiznal) length (Div) (l)	Timebase Sec/Div (m)	Timeperiod T = mxl Sec.	f=1/T	f=1/2π(seqLC)

COLPITT'S OSCILLATOR

Formulae: - The frequency of oscillation of the oscillator

f=1/2πseq(*LC) Hz*

Where L = Self inductance of the coil (H)

C = Capacitance of the condenser (F)

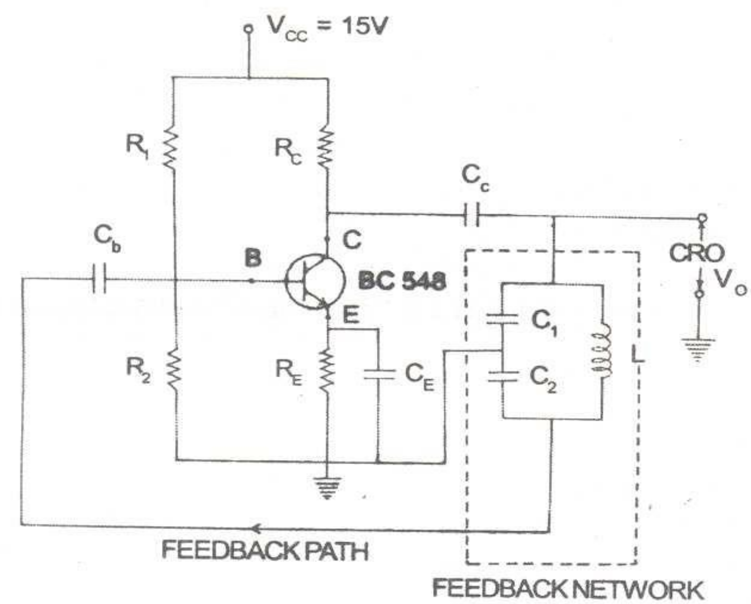
$C=C_1C_2/C_1+C_2$ = Resultant capacitance of the series combination.

C1, C2 = capacitances of the two capacitors in the tank circuit

Description: - The Colpitt's oscillator is designed for generation of high frequency sinusoidal oscillations (radio frequencies ranging from 10 KHz to100MHz). They are widely used in commercial signal generators up to100MHz.Colpitt's oscillator is same as Hartley oscillator except for one difference. Instead of using a tapped inductance, Colpitt's oscillator uses a tapped capacitance. The circuit diagram of Colpitt's oscillator using BJT is shown in Fig. It consists of an R-C coupled amplifier using an n-p-n transistor in CE configuration. R1 and R2 is two resistors which form a voltage divider bias to the transistor. A resistor RE is connected in the circuit which stabilizes the circuit against temperature variations. A capacitor CE is connected in parallel with RE, acts as a bypass capacitor and provides a low reactive path to the amplified ac signal. The coupling capacitor CC blocks dc and provides an a.c. path from the collector to the tank circuit. The feedback network (tank circuit) consists of two capacitors C1and C2 (in series) which placed across a common inductor L. The center of the two capacitors is tapped (grounded). The feedback network (C1, C2and L) determines the frequency of oscillation of the oscillator. The two series capacitors

C1, and C2form the potential divider led for providing the feedback voltage. The voltage developed across the capacitor C2provides regenerative feedback which is essential for sustained oscillations.

THEORY:-When the collector supply voltage V_{cc} is switched on, collector current starts rising and charges the capacitors $C_1$and C_2. When these capacitors are fully charged, they discharge through coil L setting up damped harmonic oscillations in the tank circuit. The oscillatory current in the tank circuit produces an a.c. voltages across C_1, C_2. The oscillations across C2 are applied to base-emitter junction of the transistor and appears in the amplified form in the collector circuit and overcomes the losses occurring in the tank circuit. The feedback voltage (across the capacitor C2) is 180° out of phase with the output voltage (across the capacitor C1), as the center of the two capacitors is grounded. A phase shift of 180° is produced by the feedback network and a further phase shift of 180° between the output and input voltage is produced by the CE transistor. Hence, the total phase shift is 360° or 0°, which is essential for sustained oscillations, as per, the Barkhausen criterion. So we get continuous undammed oscillations.



Procedure:-

The circuit is connected as shown in figure. Connect the CRO across the output terminals of the oscillator. Switch on the power supply to both the oscillator and CRO. Select proper values of L, C1and C2in the oscillator circuit and get the sine wave form on the screen of CRO. The voltage (deflection) sensitivity band switch (Y-plates) and time base band switch (X-plates) are adjusted such that a steady and complete picture of one or two sine waveform is obtained on the screen. The horizontal length (l) between two successive peaks is noted. When this horizontal length (l), is multiplied by the time base (m) i.e. sec/div, we get the time-period (T = l x m).The reciprocal of the time-period (1/T) gives the frequency (f). This can be verified with the frequency, calculated theoretically by using the above formula. The experiment is repeated by changing L or C1or C2or all. The readings are noted in the table given.

S.No	Inductance (mH)	**Capacitance (μF)**			**Measurement of time period**			**Frequency (Hz)**	
	L	**C_1**	**C_2**	**$C = C_1.C_2/C_1+C_2$**	**Peak to peak (Horiznal) length (Div) (l)**	**Timebase Sec/Div (m)**	**Time period T = mxl Sec.**	**f=1/T**	**f=1/2π(seqLC)**

Precautions: - 1) Check the continuity of the connecting terminals before going to connect the circuit.

2) Identify the emitter, base and collector of the transistor properly before connecting it in the circuit.

3) The horizontal length between two successive peaks should accurately be measured.

Result:-thus we have studied about the Hartley and Colpitt's oscillators and observe the effect of variation of C on oscillator frequency.

Viva Question

1. What is an Oscillator?
2. Which feedback used in oscillators?
3. What is the output of an oscillator if transistor is ideal?
4. How an oscillator generates oscillations without any input?
5. What are LC oscillators?
6. Why can't we use LC oscillator for low frequency oscillations?
7. What is Barkhausen criterion?
8. The frequency of Colpitts oscillator is expressed as
9. Colpitts oscillator uses which type of feedback
10. The gain device in the colpitts oscillator act as a
11. The improvement of Colpitts oscillator over Hately oscillator is, Colpitts oscillator's performance in

EXPERIMENT NO.:5

Op-Amp in inverting and non-inverting modes

INVERTING AMPLIFIER

Aim: To design and setup an inverting amplifier circuit with OP AMP 741C for a gain of 10, plot the waveforms, observe the phase reversal, measure the gain.

Objectives: After completion of this experiment, student will be able to design and setup an inverting amplifier using OP AMP. He/she will be able to design and implement OPAMP inverting amplifier circuit.

Equipments/Components:

Sl .No	Name and Specification	Quantity required
1	Dual power supply +/- 15V	1
2	Function generator (0 - 1MHz)	1
3	Oscilloscope	1
4	Bread board	1
5	IC 741C	1
6	Resistors	2
7	Probes and connecting wires	As required.

Principle: It is a closed loop mode application of opamp and employs negative feedback. The Rf and Ri are the feedback and input resistance of the circuit respectively. The input terminals of the opamp draws no current because of the large differential input impedance. The potential difference across the input terminals of an opamp is zero because of the large open loop gain. Due to these two conditions, the inverting terminal is at virtual ground potential. So the current flowing through Ri and Rf are the same.

Ii = If

That is Vin/Ri = - Vo /Rf

Therefore Vo/Vin = Av = - Rf/ Ri,

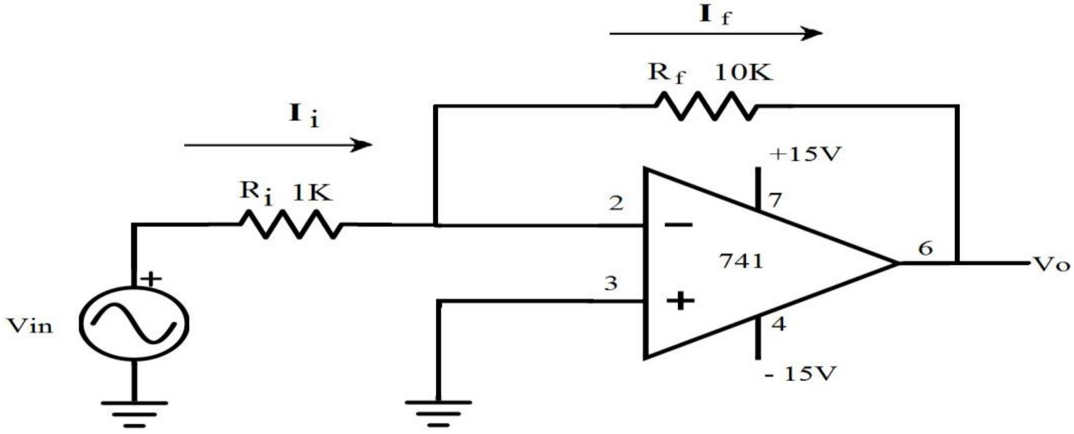
Here the –Ve sign indicates that the output will be an amplified wave with 1800 phase shift (inverted output). By varying the Rf or Ri, the gain of the amplifier can be varied to any desired value.

Procedure

1. Check the components.
2. Setup the circuit on the breadboard and check the connections.
3. Switch on the power supply.
4. Give 1 Vpp / 1 KHz sine wave as input.
5. Observe input and output on the two channels of the oscilloscope simultaneously.
6. Note down and draw the input and output waveforms on the graph.
7. Verify the input and output waveforms are out of phase.
8. Verify the obtained gain is same as designed value of gain.

Result:

Circuit Diagram:



Design:

Gain of an inverting amplifier Av = Vo/Vin = - Rf / Ri

The required gain = 10, That is Av = - Rf/ Ri = 10

Let Ri = 1KΩ, Then Rf = 10KΩ

Observations: Vin = 1 Vpp Vo=?

Gain, Av = Vo/Vin =?

Observed phase difference between the input and the output on the CRO =?

NON-INVERTING AMPLIFIER

Aim: To design and setup a non-inverting amplifier circuit with OPAMP IC 741C for a gain of 11, plot the waveform, observe the phase reversal, measure the gain.

Objectives: After completion of this experiment, student will be able to design and setup a non-inverting amplifier using OP AMP. He/she will acquire skill to design and implement OPAMP non-inverting amplifier circuit.

Equipments/Components:

Sl .No	Name and Specification	Quantity required
1	Dual power supply +/- 15V	1
2	Function generator (0 - 1MHz)	1
3	Oscilloscope	1
4	Bread board	1
5	IC 741C	1
6	Resistors	2
7	Probes and connecting wires	As required.

Principle:

It is a linear closed loop mode application of op-amp and employs negative feedback. The R_f and R_i are the feedback and input resistance of the circuit respectively. There will be no phase difference between the output and input. Hence it is called non-inverting amplifier.

$$Av = V_o / V_{in} = 1 + R_f / R_i ,$$

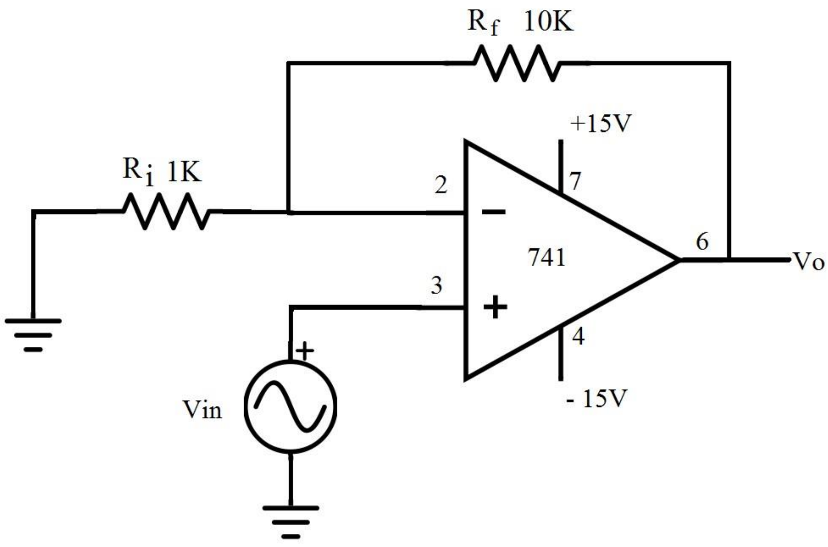
Here the +Ve sign indicates that the output will be an amplified wave in phase with the input. By varying the R_f or R_i, the gain of the amplifier can be varied to any desired value.

Procedure

1. Check the components.
2. Setup the circuit on the breadboard and check the connections.
3. Switch on the power supply.
4. Give 1 Vpp / 1 KHz sine wave as input.
5. Observe input and output on the two channels of the CRO simultaneously.
6. Note down and draw the input and output waveforms on the graph.
7. Verify the input and output waveforms are in phase.
8. Verify the obtained gain is same as designed value.

Result:

Circuit Diagram



Design:

Gain of an inverting amplifier $Av=V_o/V_{in} = 1+R_f/ R_i$,

Let the required gain be 11, Therefore

$Av= 1+R_f/ R_i= 11$

$R_f/ R_i = 10$

Take $R_i= 1K\Omega$, Then $R_f = 10K\Omega$

Observations:

Vin = 1Vpp

Vo = ?

Gain Av = Vo/Vin =?

Observed phase difference between the input and the output on the CRO =?

Viva Question

1. Can you explain the basic concept of an inverting amplifier using an operational amplifier (Op-Amp)?
2. What is the purpose of the feedback resistor in the inverting amplifier configuration?
3. How does the input voltage in an inverting amplifier relate to the output voltage?
4. What is the significance of the virtual short concept in the inverting amplifier circuit?
5. How does the closed-loop gain of an inverting amplifier change with the values of the input resistor (Rin) and feedback resistor (Rf)?
6. What are the advantages of using an inverting amplifier configuration in certain applications?
7. Can you discuss any potential drawbacks or limitations of an inverting amplifier circuit?
8. How does the inverting amplifier configuration handle negative and positive input voltages?
9. What role does the operational amplifier play in the performance of an inverting amplifier?
10. How does the frequency response of an inverting amplifier compare to other amplifier configurations?
11. Can you describe the input impedance of an inverting amplifier and its significance in practical applications?
12. Explain how changes in temperature may affect the performance of an inverting amplifier circuit.
13. What are some common applications of inverting amplifiers in electronic circuits?
14. How can you calculate the input bias current and input offset voltage in an inverting amplifier circuit?
15. In what scenarios would you prefer using an inverting amplifier over a non-inverting amplifier?

Experiment No.: 5(B)

Op-Amp as scalar, summer and voltage follower

SUMMING AMPLIFIER

Aim: To design and setup a summing amplifier circuit with OP AMP 741C for a gain of 2 and verify the output.

Objectives: After completion of this experiment, student will be able to design and setup a summing amplifier using OP AMP.

Equipments/Components:

Sl .No	Name and Specification	Quantity required
1	Dual power supply +/- 15V	1
2	DC power source 1.5V	2
3	Function generator (0- 1MHz)	1
4	Oscilloscope	1
5	Bread board	1
6	IC 741C	1
7	Resistor	3
8	Probes and connecting wires	As required.

Principle: Op-amp can be used to design a circuit whose output is the sum of several input signals. Such a circuit is called a summing amplifier or an adder. Summing amplifier can be classified as inverting & non-inverting summer depending on the input applied to inverting & non-inverting terminals respectively. Circuit Diagram shows an inverting summing amplifier with 2 inputs. Here the output will be amplified version of the sum of the two input voltages with 1800 phase reversal.

Vo = - (Rf/ Ri)(V1+V2)

Procedure

1. Check the components.
2. Setup the circuit on the breadboard and check the connections.
3. Switch on the power supply.
4. Give V1 =V2 = +1.5V DC with polarity as shown in fig.1.
5. Make sure that the CRO selector is in the D.C. coupling position.
6. Observe input and output on two channels of the oscilloscope simultaneously.
7. Note down and draw the input and output waveforms on the graph.
8. Verify that the output voltage is -6VDC
9. Repeat the procedure with V1 =1Vpp / 1 KHz sine wave and V2 = +1.5Vdc as shown in fig2.
10. Verify the output .

Result:

Circuit Diagram

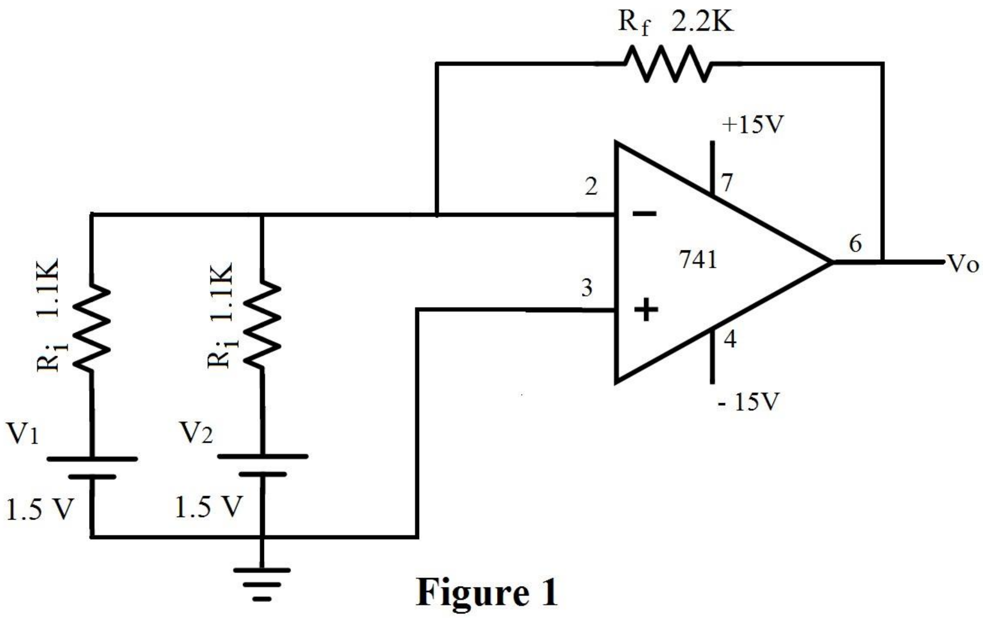


Figure 1

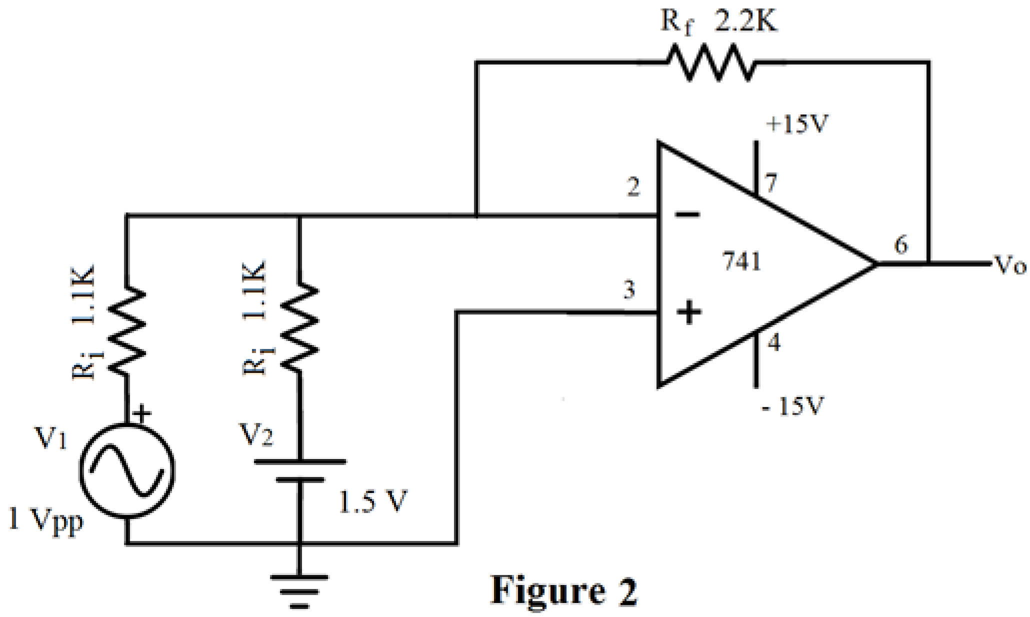


Figure 2

Design:

The output voltage of an inverting summing amplifier is given by Vo = -(Rf / Ri)(V1+V2) Let Ri = 1.1KΩ

Then Rf = 2.2KΩ Then Vo = -2(V1+V2) Observations:

Part1:

Part 2:

V1= 1.5 DC V2= 1.5 DC Then Vo=?

V1= 1Vpp sine wave

V2= 1.5 DC Then Vo=?

Graph:

Viva Question

1. What is the purpose of using an operational amplifier in a unity gain (buffer) configuration?
2. How does the voltage gain of an operational amplifier change when configured as a voltage follower (unity gain)?
3. Explain the significance of input and output resistors in a unity gain Op-Amp circuit.
4. What are the practical applications where using an Op-Amp as a voltage follower is advantageous?

Configuration Summing Amplifier:

5. Can you explain the fundamental principle behind a summing amplifier using operational amplifiers?
6. How do individual input voltages contribute to the overall output voltage in a summing amplifier circuit?
7. Discuss the role of input resistors in a summing amplifier, and how do their values impact the performance of the circuit?
8. Describe the concept of an inverting summing amplifier and provide examples of its applications.
9. How would you calculate the output voltage in a summing amplifier when multiple input signals are present?
10. What measures can be taken to ensure accuracy and minimize errors when using a summing amplifier?

Voltage Follower Configuration:

11. Explain the purpose of configuring an operational amplifier as a voltage follower.
12. How does the voltage follower configuration affect the input and output impedance of the Op-Amp?

EXPERIMENT NO: 6 (FET CHARACTERISTICS)

AIM:-Plot and study the characteristics of small signal amplifier using FET.

APPARATUS: FET (BFW-11)

Regulated power supply

Voltmeter (0-20V)

Ammeter (0-100mA)

Bread board

Connecting wires

THEORY:

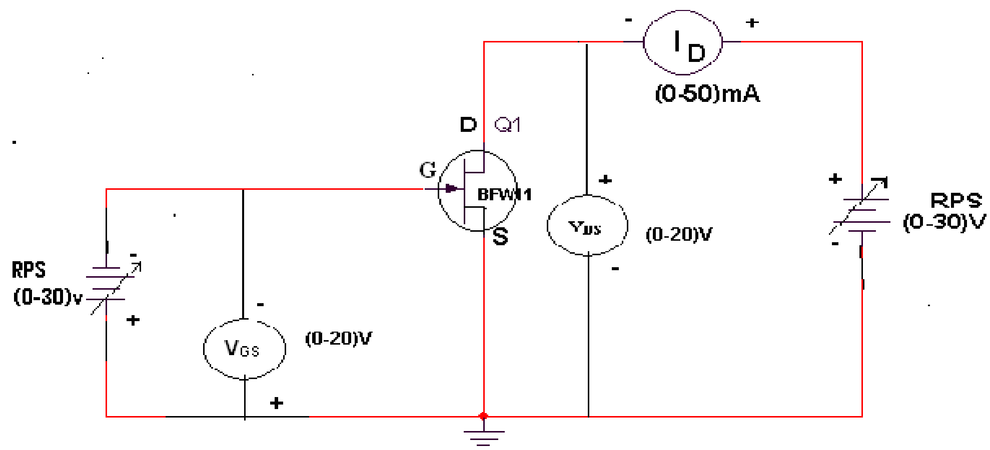
A FET is a three terminal device, having the characteristics of high input impedance and less noise, the Gate to Source junction of the FET s always reverse biased. In response to small applied voltage from drain to source, the n-type bar acts as sample resistor, and the drain current increases linearly with $V_{DS.}$ With increase in I_D the ohmic voltage drop between the source and the channel region reverse biases the junction and the conducting position of the channel begins to remain constant. The V_{DS} at this instant is called "pinch of voltage".

If the gate to source voltage (V_{GS}) is applied in the direction to provide additional reverse bias, the pinch off voltage ill is decreased.

In amplifier application, the FET is always used in the region beyond the pinch-off.

F_{DS}=I_{DSS}(1-V_{GS}/V_P)^2

CIRCUIT DIAGRAM



PROCEDURE:

1. All the connections are made as per the circuit diagram.
2. To plot the drain characteristics, keep V_{GS} constant at 0V.
3. Vary the V_{DD} and observe the values of V_{DS} and I_D.
4. Repeat the above steps 2, 3 for different values of V_{GS} at 0.1V and 0.2V.
5. All the readings are tabulated.
6. To plot the transfer characteristics, keep V_{DS} constant at 1V.
7. Vary V_{GG} and observe the values of V_{GS} and I_D.
8. Repeat steps 6 and 7 for different values of V_{DS} at 1.5 V and 2V.
9. The readings are tabulated.
10. From drain characteristics, calculate the values of dynamic resistance (r_d) by using the formula

$$r_d = \Delta V_{DS}/\Delta I_D$$

11. From transfer characteristics, calculate the value of transconductace (g_m) By using the formula

$$G_m = \Delta I_D/\Delta V_{DS}$$

12. Amplification factor (μ) = dynamic resistance. Tran conductance

$$\mu = \Delta V_{DS}/\Delta V_{GS}$$

OBSERVATIONS:

DRAIN CHARACTERISTICS:

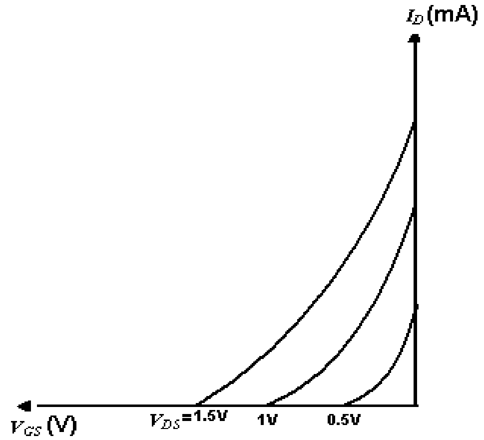
S.NO	**V_{GS}=0V**		**V_{GS}=0.1V**		**V_{GS}=0.2V**	
	V_{DS}(V)	**I_D(mA)**	**V_{DS}(V)**	**I_D(mA)**	**V_{DS}(V)**	**I_D(mA)**

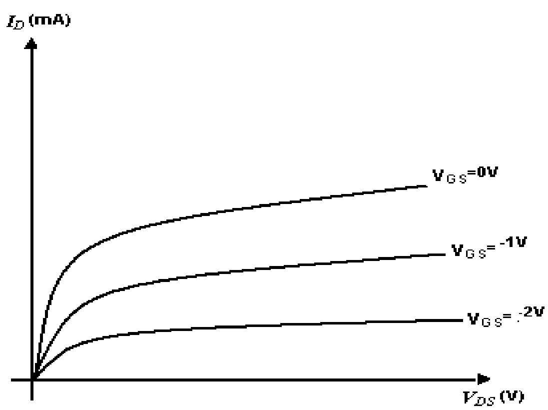
TRANSFER CHARACTERISTICS:

S. NO	**V_{DS} =0.5V**		**V_{DS}=1V**		**V_{DS} =1.5V**	
	V_{GS} (V)	**I_D(mA)**	**V_{GS} (V)**	**I_D(mA)**	**V_{GS} (V)**	**I_D(mA)**

MODEL GRAPH:

TRANSFER CHARACTERISTICS **DRAIN CHARACTERISTICS**





PRECAUTIONS:

1. The three terminals of the FET must be carefully identified
2. Practically FET contains four terminals, which are called source, drain, Gate, substrate.
3. Source and case should be short circuited.
4. Voltages exceeding the ratings of the FET should not be applied.

RESULT:

1. The drain and transfer characteristics of a given FET are drawn
2. The dynamic resistance (r_d), amplification factor (μ) and Tran conductance (g_m) of the given FET are calculated.

VIVA QUESTIONS:

1. What are the advantages of FET?
2. Different between FET and BJT?
3. Explain different regions of V-I characteristics of FET?
4. What are the applications of FET?
5. What are the types of FET?
6. Draw the symbol of FET.
7. What are the disadvantages of FET?
8. What are the parameters of FET?

EXPERIMENT NO: 7 (PUSH PULL AMPLIFIER)

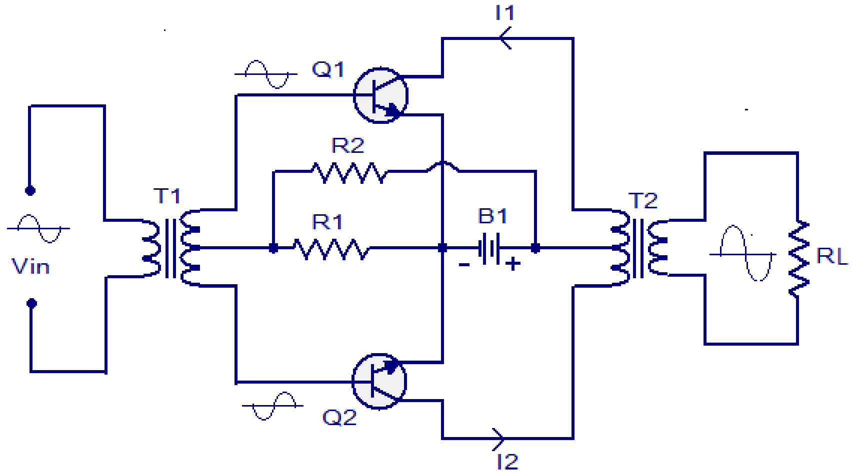
Aim: - Study of push pull amplifier & Measure variation of output power & distortion with load.

Apparatus:-Signal Generator, CRO (dual trace) and Connecting Wires

Components Used:- Transistor Q1 & Q2= bc 148 Driver, T1 & T2 = 6-0-6 v

Theory:-A push pull amplifier is an amplifier which has an output stage that can drive a current in either direction through the load. The output stage of a typical push pull amplifier consists of two identical BJTs or MOSFETs one sourcing current through the load while the other one sinking the current from the load. Push pull amplifiers are superior over single ended amplifiers (using a single transistor at the output for driving the load) in terms of distortion and performance. A single ended amplifier, how well it may be designed will surely introduce some distortion due to the nonlinearity of its dynamic transfer characteristics. Push pull amplifiers are commonly used in situations where low distortion, high efficiency and high output power are required. The basic operation of a push pull amplifier is as follows: The signal to be amplified is first split into two identical signals 180° out of phase. Generally this splitting is done using an input coupling transformer. The input coupling transformer is so arranged that one signal in applied to the input of one transistor and the other signal is applied to the input of the other transistor. Advantages of push pull amplifier are low distortion, absence of magnetic saturation in the coupling transformer core, and cancellation of power supply ripples which results in the absence of hum while the disadvantages are the need of two identical transistors and the requirement of bulky and costly coupling transformers.

Class A push pull amplifier.

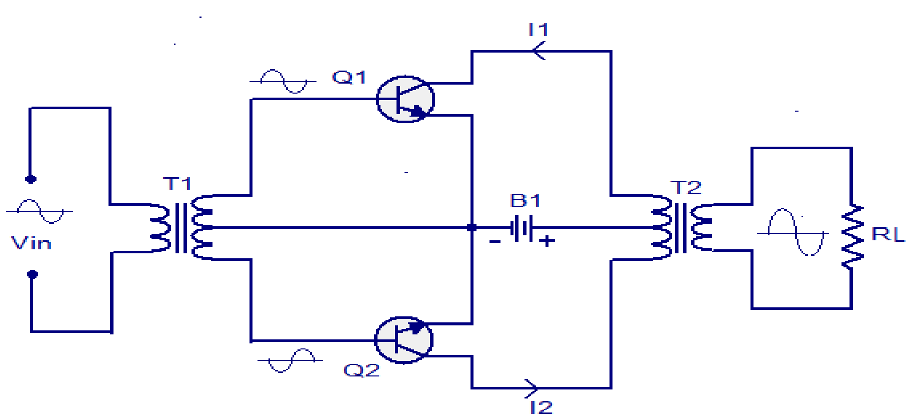


Class A push pull amplifier

A push pull amplifier can be made in Class A, Class B and Class AB or Class C configurations. The circuit diagram of a typical Class A push pull amplifier is shown above. Q1 and Q2 are two identical transistor and their emitter terminals are connected together. R1 and R2 are meant for biasing the transistors. Collector terminals of the two transistor are connected to the respective ends of the primary of the output transformer T2. Power supply is connected between the center tap of the T2 primary and the emitter junction of the Q1 and Q2. Base terminal of each transistor is connected to the respective ends of the secondary of the input coupling transformer T1. Input signal is applied to the primary of T1 and output load RL is connected across the secondary of T2.Quiescent current of Q2 and Q1 flows in opposite directions through the corresponding halves of the primary of T2 and as a result there will be no magnetic saturation. From the figure you can see the phase splatted signals being applied to the base of each transistors. When Q1 is driven positive using the first half of its input signal, the collector current of Q1 increases. At the same time Q2 is driven negative using the first half of its input signal and so the collector current of Q2 decreases. From the figure you can understand that the collector currents of Q1 and Q2 i.e.; I1 and I2 flows in the same direction trough the corresponding halves of the T2 primary. As a result an amplified version of the original input signal is induced in the T2 secondary. It is clear that the current through the T2 secondary is the difference between the two collector currents. Harmonics will be much less in the output due to cancellation and this is results in low distortion.

Class B push pull amplifier.

The Class B push pull amplifier is almost similar to the Class A push pull amplifier and the only difference is that there is no biasing resistors for a Class B push pull amplifier. This means that the two transistors are biased at the cutoff point. The Class B configuration can provide better power output and has higher efficiency (up to 78.5%). Since the transistor are biased at the cutoff point, they consumes no power during idle condition and this adds to the efficiency. The advantages of Class B push pull amplifiers are, ability to work in limited power supply conditions (due to the higher efficiency), absence of even harmonics in the output, simple circuitry when compared to the Class A configuration etc. The disadvantages are higher percentage of harmonic distortion when compared to the Class A, cancellation of power supply ripples is not as efficient as in Class A push pull amplifier and which results in the need of a well regulated power supply. The circuit diagram of a classic Class B push pull amplifier is shown in the diagram below.



Procedure:-

1. Study the circuit diagram provided on the front panel of the kit.
2. Connect the sine wave generator to the input terminals.
3. Connect Dual Trace CRO at input & output side,
4. Switch 'ON ' the power supply.
5. Apply a sine wave I/P from signal generator at I/P Vin of circuit.

6. Now keep the amplitude Vin constant. Vary the I/P frequency Fi from 100 Hz to 200KHz in steps. Observe & note the corresponding O/P voltage VO. Calculate the voltage gain AV= VO / Vin.

7. Repeat step 6 for different frequencies.

8. Plot a graph between I/P frequency Fi (on X-axis) & voltage gain AV (on Y-axis).Find out the bandwidth from graph.

Observations:-

Vin = --------V (constant)

S.No.	Input Frequency Fi	Output voltage Vo	AV = VO / Vin.

Formulas: - $P_{oac} = I^2 RL$

Result:-

The frequency response of Class- B Push pull amplifier is studied having bandwidth

BW = --------- Hz.

<u>Viva Question:</u>

1. Which amplifiers are normally operated in a push-pull configuration in order to produce an output that is a replica of the input.
2. Which type of power amplifier is biased for operation at less than 180º of the cycle?
3. What is the maximum temperature rating for silicon power transistors?
4. Which of the push-pull amplifiers is presently the most popular form of the class B power amplifier?
5. Which operation class is generally used in radio or communications?
6. The maximum efficiency of a class B amplifier is ________ percent.

EXPERIMENT NO: 8 (UJT CHARACTERISTICS)

AIM:-To plot the characteristics of UJT and UJT as relaxation.

APPARATUS:-

Regulated Power Supply (0-30V, 1A) - 2Nos

UJT 2N2646

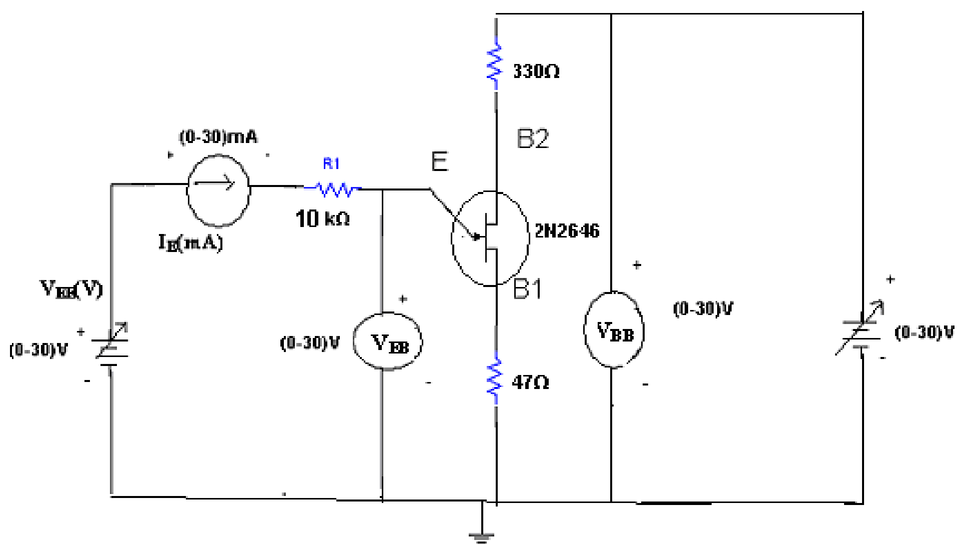
Resistors 10kΩ, 47Ω, 330Ω

Millimeters - 2Nos

Breadboard

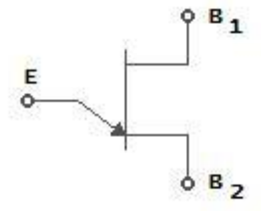
Connecting Wires

CIRCUIT DIAGRAM



THEORY:

A Uni junction Transistor (UJT) is an electronic semiconductor device that has only one junction. The UJT Uni junction Transistor (UJT) has three terminals and emitter (E) and two bases (B1 and B2). The base is formed by lightly doped n-type bar of silicon. Two ohmic contacts B1 and B2 are attached at its ends. The emitter is of p-type and it is heavily doped. The resistance between B1 and B2, when the emitter is open-circuit is called inter base resistance. The original unit junction transistor, or UJT, is a simple device that is essentially a bar of N type semiconductor material into which P type material has been diffused somewhere along its length. The 2N2646 is the most commonly used version of the UJT.



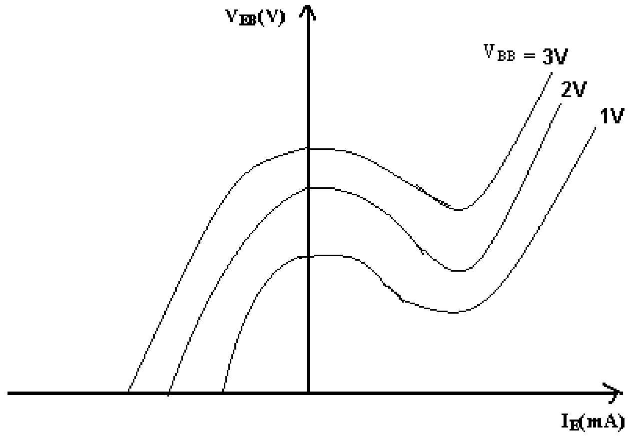
Circuit symbol

The UJT is biased with a positive voltage between the two bases. This causes a potential drop along the length of the device. When the emitter voltage is driven approximately one diode voltage above the voltage at the point where the P diffusion (emitter) is, current will begin to flow from the emitter into the base region. Because the base region is very lightly doped, the additional current (actually charges in the base region) causes (conductivity modulation) which reduces the resistance of the portion of the base between the emitter junction and the B2 terminal. This reduction in resistance means that the emitter junction is more forward biased, and so even more current is injected. Overall, the effect is a negative resistance at the emitter terminal. This is what makes the UJT useful, especially in simple oscillator circuits. When the emitter voltage reaches V_p, the current starts to increase and the emitter voltage starts to decrease. This is represented by negative slope of the characteristics which is referred to as the negative resistance region, beyond the valley point R_{B1} reaches minimum value and this region, V_{EB} proportional to $I_{E.}$

PROCEDURE:

1. Connection is made as per circuit diagram.
2. Output voltage is fixed at a constant level and by varying input voltage corresponding emitter current values are noted down.
3. This procedure is repeated for different values of output voltages.
4. All the readings are tabulated and Intrinsic Stand-Off ratio is calculated using $\eta = (V_p - V_D) / V_{BB}$
5. A graph is plotted between V_{EE} and I_E for different values of $V_{BE.}$

MODEL GRAPH:



OBSEVATIONS:

V_{BB}=1V		V_{BB}=2V		V_{BB}=3V	
V_{EB}(V)	I_E(mA)	V_{EB}(V)	I_E(mA)	V_{EB}(V)	I_E(mA)

CALCULATIONS:

$V_P = \eta V_{BB} + V_D$

$\eta = (V_P - V_D) / V_{BB}$

$\eta = (\eta_1 + \eta_2 + \eta_3) / 3$

RESULT: The characteristics of UJT are observed and the values of Intrinsic Stand-Off Ratio is calculated.

VIVA QUESTIONS

1. What is the symbol of UJT?
2. Draw the equivalent circuit of UJT?
3. What are the applications of UJT?
4. Formula for the intrinsic standoff ratio?
5. What does it indicates the direction of arrow in the UJT?
6. What is the difference between FET and UJT?
7. Is UJT is used an oscillator? Why?
8. What is the Resistance between B_1 and B_2 is called as?
9. What is its value of resistance between B_1 and $B_{2?}$
10. Draw the characteristics of UJT?

EXPERIMENT NO: 9 (SERIES VOLTAGE REGULATOR)

AIM: - Study of series and shunt voltage regulators and measurement of line and load regulation and ripple factor.

EQUIPMENTS AND COMPONENTS:

APPARATUS

1. CRO (Dual channel) DC-20 MHz 1 No
2. Bread Board - ! No. .
3. Regulated power supply- 0-30v 1 A, 1 No.
4. DMM 3 ½ Digit LCD hand held 1No

COMPONENTS:

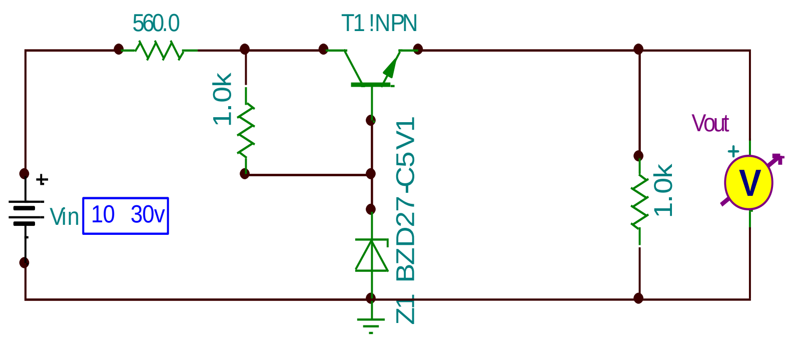
1. 1kΩ Resistor – 1 No.
2. 560Ω Resistor – 1 No.
3. 1k , 2k , 4.7k, 10k (load resistors) – 1 No each.
4. Zener diode – 1 No.
5. Transistor – SL100 – 1 No.

All resistors are carbon / metal film ¼ W 5% unless otherwise specified.

THEORY: Voltage regulator is a device designed to maintain the output voltage as nearly constant as possible. It monitors the output voltage and generates feed back that automatically increases are decreases the supply voltage to compensate for any changes in output voltage that might occur because of change in load are changes in load voltages.

In transistorized series voltage regulator the control element is a transistor which is in series with load must be operated in reverse break down region, where it provides constant voltage irrespective of changes in applied voltages. The output voltage of the series voltage regulator is $V_o = V_z - V_{be}$. Since, V_z is constant, any change in Vo must cause a change in V_{be} in order to maintain the above equation. So, when V_o decreases V_{be} increases, which causes the transistor to conduct more and to produce more load current, this increase in load causes an increase in Vo and makes Vo as constant. Similarly, the regulation action happens when Vo increases also.

CIRCUIT DIAGRAM:



PROCEDURE:

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

ii. Apply the input voltage from power supply.

iii. Measure base, emitter and collector D.C voltages and compare against estimated values.

	Estimated voltages	Observed voltages
Vb1 ,Vc1, Ve1		
Vz		

iv. For a specific value of load resistor, vary the input voltage from 10 to a maximum of 20 volts and not the values of output voltage.

v. Change the load resistor and repeat steps 2 and 3.

vi. Remove the load resistor and note down the voltage at no load.

vii. Find percentage regulation.

Percentage regulation = $\frac{V_{NL} - V_{FL}}{V_{FL}} x100$

viii. Plot the graph for load regulation and line regulation.

OBSERVATIONS:

S.no	Vin	***Output voltage***		
		R_L=	**R_L=**	**R_L=**

CALCULATIONS:

Percentage load regulation = $\frac{V_{NL} - V_{FL}}{V_{FL}} x100 =$

Percentage Line Regulation = (change in output) / (change in input) X 100

APPLICATIONS:

1. Low current applications.
2. Fixed voltage applications
3. Extension of zener regulator for higher currents.

SHUNT VOLTAGE REGULATOR

THEORY:

A voltage regulator is a device or a combination of devices, design to maintain the output voltage of a power supply as nearly constant as possible even if there are changes in load or in input voltage. In shunt voltage regulator transistor Q1 acts as control element, which is in shunt with load voltage.

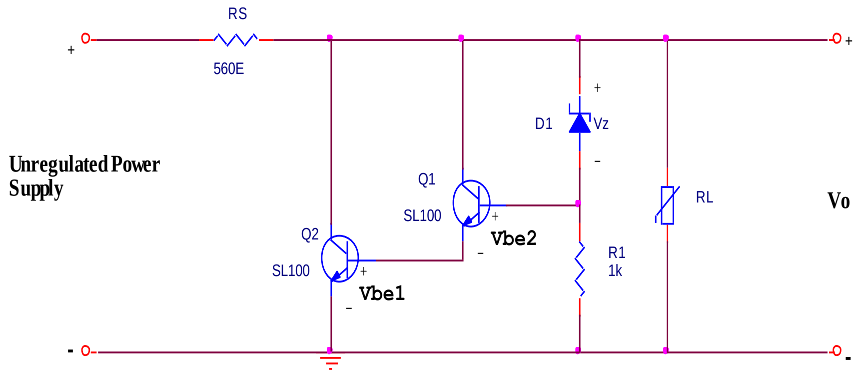
The output voltage is given as

$$Vo = Vz + V_{R1} = Vz + Vbe1 + Vbe2$$

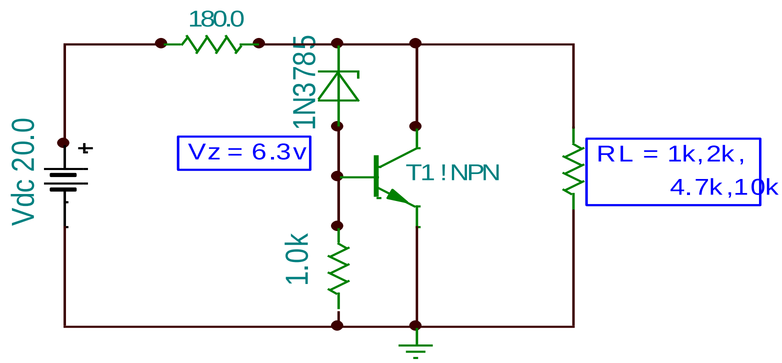
The regulation action of the circuit is explained below :

Since Vz is constant, any changes in output voltage reflects a propositional change in R1. If the output voltage decreases, voltage across R1 decreases which in turn decreases the base voltage of Q2. As a result the base current of Q1 decreases which allows the load voltage to rise and makes it constant the same regulation action follows even if the output voltage increases.

CIRCUIT DIAGRAM:



ALTERNATE CIRCUIT:



PROCEDURE:

i. Connect the circuit as shown in the circuit diagram. Apply the input voltage from power supply.

ii. Measure base, emitter and collector D.C voltages and compare against estimated values.

	Estimated voltages	Observed voltages
Vb1 ,Vc1, Ve1		
Vb2 ,Vc2, Ve2		
Vz		

iii For a specific value of load resistor, vary the input voltage from zero to a maximum of 20 volts and note the values of output voltage.

iv. Change the load resistor and repeat steps 2 and 3.

v. Remove the load resistor and note down the voltage at no load.

vi. Find percentage regulation.

Percentage regulation = $\frac{V_{NL} - V_{FL}}{V_{FL}} x100$

vii. Plot the graph for load regulation and line regulation.

OBSERVATIONS:

VOLTAGE AT NO-LOAD =

S.no	Vin	***Output voltage***		
		R_L=	**R_L=**	**R_L=**

CALCULATIONS:

Percentage regulation = $\frac{V_{NL} - V_{FL}}{V_{FL}} x100$

RESULT:

for series regulator:-

For RL = ----------------, Regulating range is_____________

For RL = ----------------, Regulating range is_____________

For RL = ----------------, Regulating range is_____________

For shunt regulator:-

For RL = ----------------, Regulating range is_____________

For RL = ----------------, Regulating range is_____________

For RL = ----------------, Regulating range is_____________

PRECAUTIONS:

i. Proceed on the experiment only after obtaining expected DC voltages do not apply more than 20 V without connecting load on the output as this would result in maximum current in shunt transistors.

ii. Shorting the output will result in overheating series resistors which may burn at high voltage.

iii. Reversing the zener may not damage the circuit but result in output voltage to drop 2 V or less.

APPLICATIONS:

1. Low current applications.
2. Fixed voltage applications.

Viva Question:

1. What is meant by regulation? Why is it required?
2. Define line regulation and load regulation?
3. .What are the different types of voltage regulators and which type is best, why?
4. What is a "load-line"? What is its significance? Differentiate between a.c. load line and d.c. load line.
5. Define line and load regulation, what are the ideal values

EXPERIMENT NO: 10 (MOSFET CHARACTERISTICS)

AIM: - To plot the characteristics of MOSFET and CMOS

APPARATUS: - MOSFET characteristics Tr.kit

Connecting Wires

Power Supply

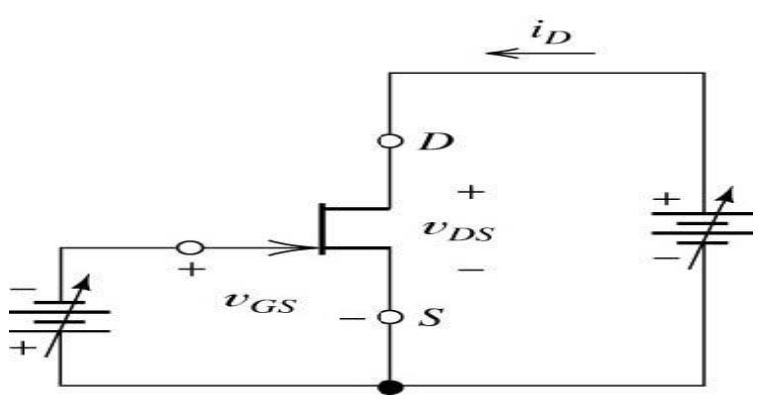
THEORY:-

Metal Oxide FET - MOSFET

As well as the Junction Field Effect Transistor (JFET), there is another type of Field Effect Transistor available whose Gate input is electrically insulated from the main current carrying channel and is therefore called an Insulated Gate Field Effect Transistor or IGFET. The most common type of insulated gate FET which is used in many different types of electronic circuits is called the Metal Oxide Semiconductor Field Effect Transistor or MOSFET for short.

The MOSFET is a voltage controlled field effect transistor that differs from a JFET in that it has a "Metal Oxide" Gate electrode which is electrically insulated from the main semiconductor N-channel or P-channel by a thin layer of insulating material usually silicon dioxide (commonly known as glass). This insulated metal gate electrode can be thought of as one plate of a capacitor. The isolation of the controlling Gate makes the input resistance of the MOSFET extremely high in the Mega-ohms (MΩ) region thereby making it almost infinite.

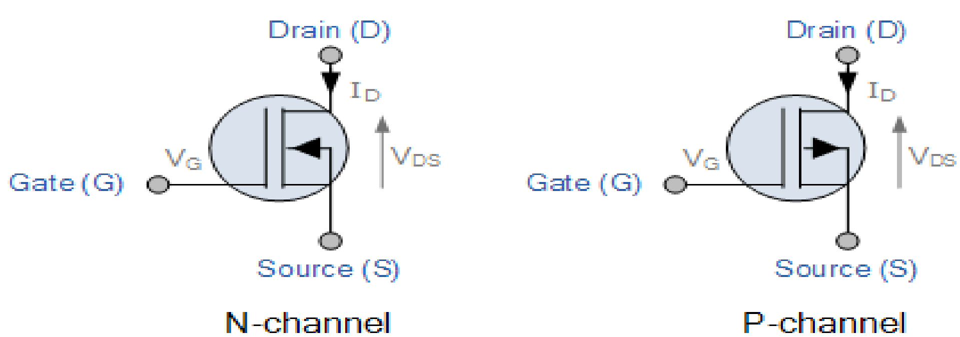
.**CIRCUIT DIAGRAM:-**



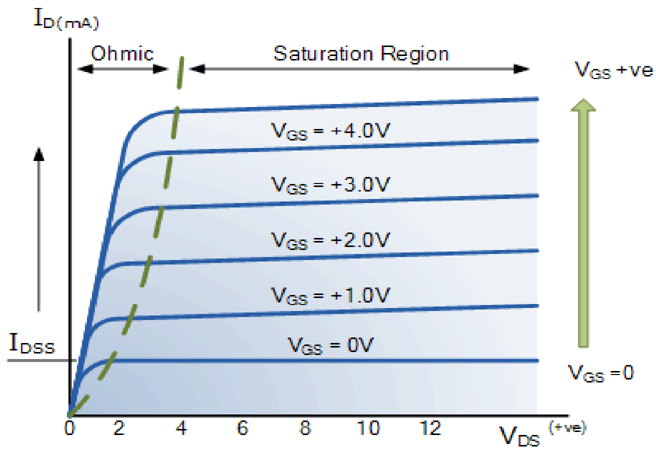
As the Gate terminal is isolated from the main current carrying channel "NO current flows into the gate" and just like the JFET, the MOSFET also acts like a voltage controlled resistor were the current flowing through the main channel between the Drain and Source is proportional to the input voltage. Also like the JFET, this very high input resistance can easily accumulate large amounts of static charge resulting in the MOSFET becoming easily damaged unless carefully handled or protected.

Like the previous JFET tutorial, MOSFETs are three terminal devices with a Gate, Drain and Source and both P-channel (PMOS) and N-channel (NMOS) MOSFETs are available. The main difference this time is that MOSFETs are available in two basic forms:

- 1. **Depletion Type** - the transistor requires the Gate-Source voltage, (V_{GS}) to switch the device "OFF". The depletion mode MOSFET is equivalent to a "Normally Closed" switch.
-
- 2. **Enhancement Type_**- the transistor requires a Gate-Source voltage, (V_{GS}) to switch the device "ON". The enhancement mode MOSFET is equivalent to a "Normally Open" switch.



CHARACTERISTICS



RESULT: - The characteristics of MOSFET

Viva Question:

1. What are the comparisons and differences between a BJT and a JFET?
2. What is meant by a unipolar device? Why is a JFET known as a Unipolar Device?
3. Draw the symbols of JFET, MOSFET?
4. What are the typical applications of a JFET?
5. What is a MOSFET? What are the possible types in a MOSFET?
6. .What is the need for the capacitor C_S?
7. What is meant by thermal stabilization?
8. Which MOSFET allows the flow of drain current even with zero gate to source voltage just due to existence of channel between drain and source terminals?
9. The enhancement type basically termed as normally-OFF MOSFET works only with
10. Which property of MOSFET distinguishes it from JFET regarding to voltage application in addition to operational strategies and mechanisms?

EXPERIMENT NO.: 11

Op-Amp as differentiator and integrator

Aim: To design and setup a Differentiator circuit using OP AMP 741C and plot their pulse response.

Objectives: After completion of this experiment, student will be able to design and setup a differentiator circuit using OP AMP.

Equipments/Components:

Sl. No	Name and Specification	Quantity required
1	Dual power supply +/- 15V	1
2	Function generator (0- 1MHz)	1
3	Oscilloscope	1
4	Bread board	1
5	IC 741C	1
6	Resistor	1
7	Capacitor	1
8	Probes and connecting wires	As required.

Principle:

It is an opamp circuit which performs the mathematical operation of differentiation. That is the output waveform is the derivative or differentia l of the input voltage. That is Vo= - $R_f C d(V_{in})/dt$. The differentiator circuit is constructed from basic inverting amplifier by replacing the input resistance R_i with capacitor C. This circuit also works as high pass filter.

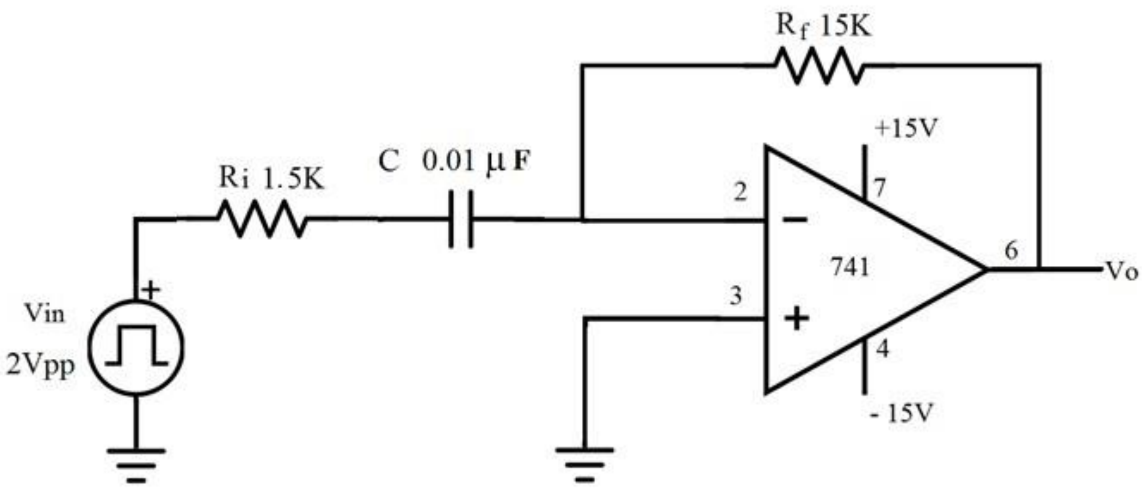
Procedure:

1. Check the components.
2. Setup the circuit on the breadboard and check the connections.

3. Switch on the power supply.
4. Keep the oscilloscope in AC coupling mode.
5. Give V_i= 2Vpp, 1KHz square wave.
6. Observe input and output on two channels of the oscilloscope simultaneously.
7. Note down and draw the input and output waveforms on the graph.

Result:

Circuit Diagram:



Design:

Given f = 1 KHz

So T = 1/f = 1ms

Design equation is $T = 2\pi R_f C$ Let C = 0.01μF

Then Rf = 15KΩ

Let Ri = Rf/10 = 1.5KΩ

INTEGRATOR

Aim: To design and setup an integrator circuit using OP AMP 741C and plot its pulse response.

Objectives: After completion of this experiment, student will be able to design and setup an integrator circuit using OP AMP.

Equipments/Components:

Sl. No	Name and Specification	Quantity required
1	Dual power supply +/- 15V	1
2	Function generator (0-1MHz)	1
3	Oscilloscope	1
4	Bread board	1
5	IC 741C	1
6	Resistor	1
7	Capacitor 0.01µF	1
8	Probes and connecting wires	As required.

Principle:

It is a closed loop op-amp circuit which performs the mathematical operation of integration. That is the output waveform is the integral of the input voltage and is given by Vo = $(-1/R_fC) \int V_{in}dt$. The integrator circuit is constructed from basic inverting amplifier by replacing the feedback resistance R_f with capacitor C. This circuit also works as low pass filter.

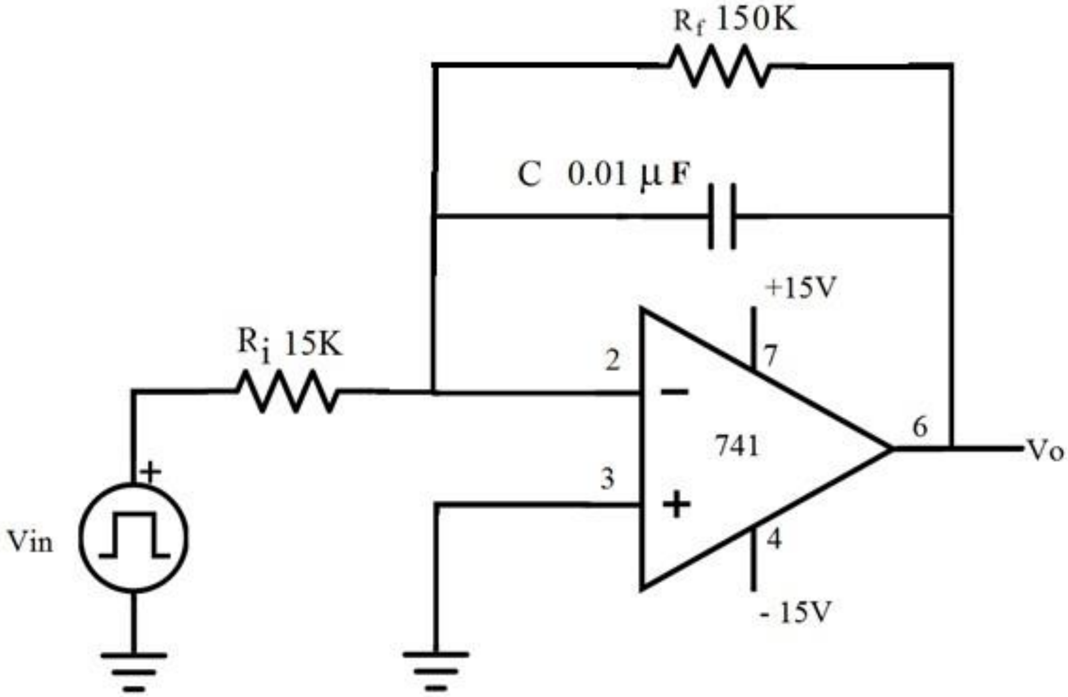
Procedure:

1. Check the components.
2. Setup the circuit on the breadboard and check the connections.
3. Switch on the power supply.
4. Give V_i= 2Vpp, 1KHz square wave.
5. Keep the oscilloscope in AC coupling mode.
6. Observe input and output on two channels of the oscilloscope simultaneously.

7. Draw the input and output waveforms on the graph.

Result:

Circuit Diagram:



Design:

Given f =1 KHz

So T = 1/f = 1ms

Design equation is $T = 2\pi R_i C$ Let

C = 0.01µF

Then R_i = 15KΩ

Take R_f = $10R_i$ = 150KΩ

Graph:

Viva Question:

Basics of Op-Amp as Differentiator:

1. What is the primary function of an operational amplifier in a differentiator circuit?
2. Explain the concept of differentiation in the context of an Op-Amp circuit.
3. How does the capacitor in a differentiator circuit affect the output response?

Differentiator Circuit Characteristics:

4. Describe the transfer function of an Op-Amp differentiator circuit.
5. How does the amplitude and phase response of the output signal change with different input frequencies in a differentiator?

Design and Calculations for Differentiator:

6. What considerations should be taken into account while selecting resistor and capacitor values for a differentiator circuit?
7. Can you derive the expression for the output voltage of an Op-Amp differentiator circuit?

Practical Aspects of Op-Amp Differentiator:

8. Discuss any potential challenges or limitations associated with using Op-Amp differentiator circuits.
9. How does the presence of noise in the input signal impact the performance of a differentiator circuit?

Basics of Op-Amp as Integrator:

10. What is the primary function of an operational amplifier in an integrator circuit?
11. Explain the concept of integration in the context of an Op-Amp circuit.