Ahsanullah University of Science and Technology
Department of Electrical and Electronic Engineering

LABORATORY MANUAL

FOR

ELECTRICAL AND ELECTRONIC SESSIONAL COURSES

Student Name :

Student ID :

Course no : EEE - 2204

Course Title : Electronic Circuits – II Lab

For the students of

Department of Electrical and Electronic Engineering

2nd Year, 2nd Semester

Experiment No: 01

Name of the experiment: Study of Class A Power Amplifier.

Theory:

An amplifier receives a signal from some pickup transducer or other input source. This signal is generally small and needs to be amplified sufficiently to operate an output device. At first, the input voltage level is improved using voltage amplifier and this is then fed to power amplifier to obtain sufficient power at the output.

In fact, a power amplifier does not amplify power. It only takes power from the dc power supply connected to the output circuit and converts it into useful ac signal power.

Depending upon the amount of the output signal variation over one cycle of operation for a full cycle of input signal, power amplifiers are grouped into various classes like Class A, Class B, Class AB, Class C, Class D, etc. In class A power amplifier, the transistor conducts for the entire cycle of the input signal and hence the output signal varies for a full 360° of the cycle. To achieve this, the Q point on the DC Load Line is approximately half of the supply voltage. This facilitates the swing of the output to vary up and down without being cut-off.

The average dc input power to the amplifier is given by,

$$P_i\ (dc) = V_{CC} I_{CQ} \tag{1}$$

Where, I_{CQ} is the collector current at Q point and V_{CC} is the supply voltage.

The ac power delivered to the load can be calculated using any of the following equations:

$$P_O\ (ac) = V_{CE}(rms)\ .\ I_C(rms)$$

$$P_O\ (ac) = \frac{V_{CE}^2\ (rms)}{R_L} \tag{2}$$

$$P_O\ (ac) = \frac{\left(V_{CE_{max}} - V_{CE_{min}}\right)\left(I_{C_{max}} - I_{C_{min}}\right)}{8} \tag{3}$$

Where symbols have their usual meanings.

The efficiency of a power amplifier is a measure of how well it converts dc power into useful ac output power. Conversion efficiency of a power amplifier is defined as,

$$\%\ \eta = \frac{P_0\ (ac)}{P_i\ (dc)} \times 100\%$$

The maximum value of efficiency of a Class A power amplifier is only 25%.

Objective:

To study Class A power amplifier.

Equipments:

Serial no.	Component Details	Specification	Quantity
1.	Transistor	BD 135	1 piece
2.	Resistor	1 KΩ, 470 Ω, 10 KΩ, 2.2 KΩ ,15 KΩ	2 pieces, 1 piece each for the rest
3.	Capacitors	10 µF	3 pieces
4.	POT	10 KΩ	1 piece
5.	Oscilloscope		1 unit
6.	DC Power Supply		1 unit
7.	AC Signal generator		1 unit
8.	Digital Multimeter		1 unit
9.	Trainer Board		1 unit

Circuit Diagram:

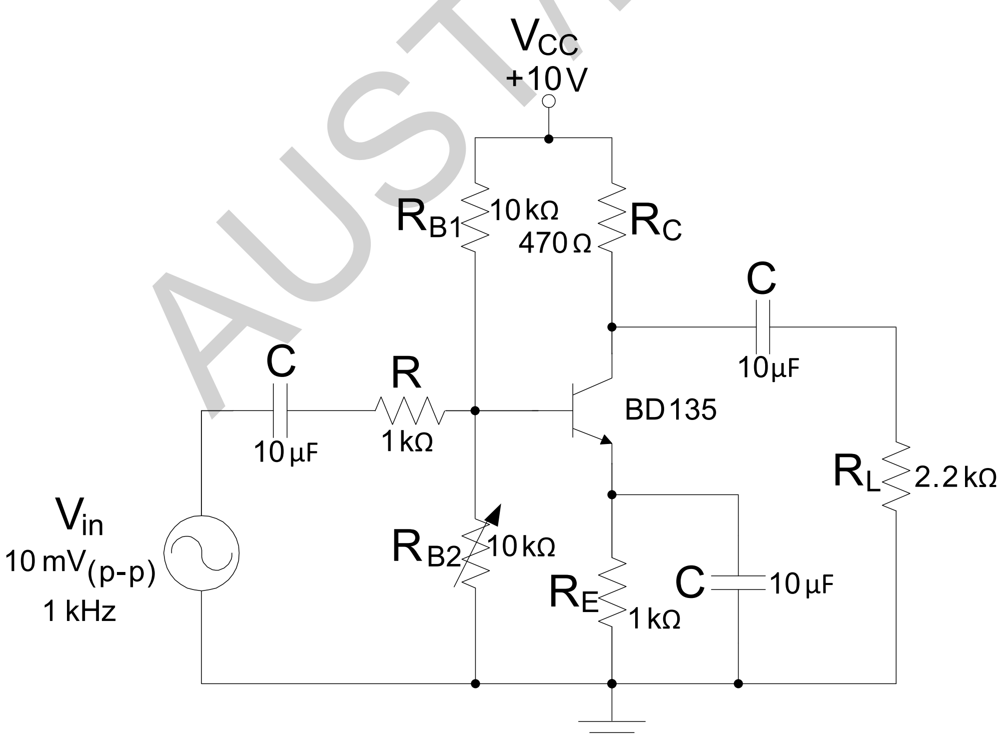

Fig: 1.1 Class A amplifier

Procedure:

1) Realize the circuit of figure 1.1
2) Without any ac input measure V_{CE}, I_C, I_B. Vary the pot resistance to select the Q point such that the circuit works as a Class A power amplifier.
3) Apply 1 KHz 10 $mV_{(p-p)}$ signal at the input.
4) Measure V_{CE} and R_L with multimeter.
5) Observe I_C and V_{CE} with oscilloscope and take the readings of their maximum and minimum values. Draw the wave shapes.
6) Observe I_B and V_{CE} simultaneously and note the phase relationship between them. Draw the wave shapes.

Reports:

1) Present the observed wave shapes drawn to scale and to phase.
2) Draw ac and dc load lines.
3) Explain the phase relationships between I_B and V_{CE}.
4) Using equations (2) and (3), calculate P_O (ac) and compare them.
5) Calculate efficiency.
6) Discuss the effect of using an inductor in place of R_C in the circuit of figure 1.1.

References:

1) Basic Electronics and Linear Circuits- N N Bhargava

2) Electronic Devices and Circuit Theory-Robert Boylestead

Experiment No: 02

Name of the experiment: Study of Class B and Class C Power Amplifier

Theory:

Class B Power Amplifier:

A class B power amplifier circuit provides an output signal varying over one half the input cycles. The dc bias point for class B is at 0V with the output varying from this bias point for a half cycle. Obviously, the output is not a faithful reproduction of the input if only one half cycle is present. Two class B operations, one to provide output on the positive half cycle and another on the negative half cycle, are combined to obtain an output for a full 360° of operation.

In the circuit of Fig.2.1, the voltage across 560Ω resistor is approximately a quarter of 1V. This is the voltage across two base-emitter junctions of Q1 and Q2. Thus base and emitter of each transistor is effectively short. This biases the transistors at cut-off region that ensures class B operation. For positive half cycle of input signal, Q1 conducts and for negative half cycle Q2 conducts. These two class B operations are combined to an output for full 360° of operation.

Class B power amplifier have many advantages over class A. Since transistors are biased at cut-off, greater voltage swing can be accommodated without being cut-off. This results in greater power output. Moreover, there is negligible collector current and hence negligible power loss when no signal is applied whereas for class A operation maximum power is dissipated at no input. This results in higher efficiency for class B amplifier.

One of the most important features for this circuit is crossover distortion. For low values of input, the transfer characteristics of transistors are not linear. Therefore, noticeable output is not obtained until the input exceeds the cut-in voltage of the base-emitter junction of each transistor. The effect of crossover distortion is most pronounced when the input signal is small.

The dc power input to class B power amplifier can be calculated using $P_i\,(dc)=\frac{2V_{CC}I_{max}}{\pi}$; where I_{max} is the peak value of current drawn from power supply.

The ac power output is given by, $P_O(ac)=\frac{V_{O_{max}}}{2R_L}$

Then conversion efficiency is given by, $\eta=\frac{P_O\,(ac)}{P_i\,(dc)}$

Maximum value of theoretical efficiency is 78.5%. Class B power amplifiers are used where the power supply is limited.

Class C power amplifier:

For class C operation, the transistor conducts for an interval shorter than a half cycle. The result is periodically pulsating current waveform. To obtain a sinusoidal output voltage, this current is passed through a parallel LC circuit. This circuit acts as band pass filter and provides an output voltage proportional to the amplitude of the fundamental component in the Fourier series representation of the current waveform. The resonant frequency of the LC combination is $f_O=1/(2\,\pi(LC))$; where, L and C are the inductance and capacitance respectively of the LC combination. The Quality Factor $(Q=X_L/R_L)$ of the tank circuit is assumed to be high. Voltage gain at resonant frequency is a maximum while it drops on either side of resonance.

The average collector current for class C operation is much less and as a result, the collector losses are less and efficiency is very high. Class C operation is used with resonant or tuned circuits as for example, in radio and television transmitters where efficiency is of utmost importance.

Objective:

To study the operation of class B and class C power amplifier.

Equipments:

Serial no.	Component Details	Specification	Quantity
1.	Transistors	BD135, BD136, C828	1 piece each
2.	Resistor	10 KΩ, 1 KΩ, 560 Ω, 120 Ω	2 piece, 1 piece each for the rest
3.	Capacitors	2.2 µF , 0.01µF, 10 µF, 22 µF, 100 µF	2 piece, 1 piece each for the rest
4.	Inductor	2.7mH	1 unit
5.	Oscilloscope		1 unit
6.	AC Signal generator		1 unit
7.	DC Power Supply		1 unit
8.	Digital Multimeter		1 unit
9.	Trainer Board		1 unit

Circuit Diagram:

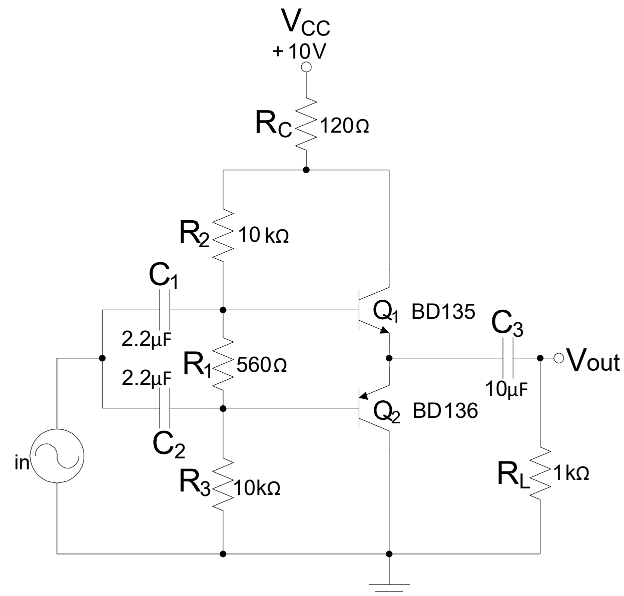

Fig 2.1: Class B Amplifier

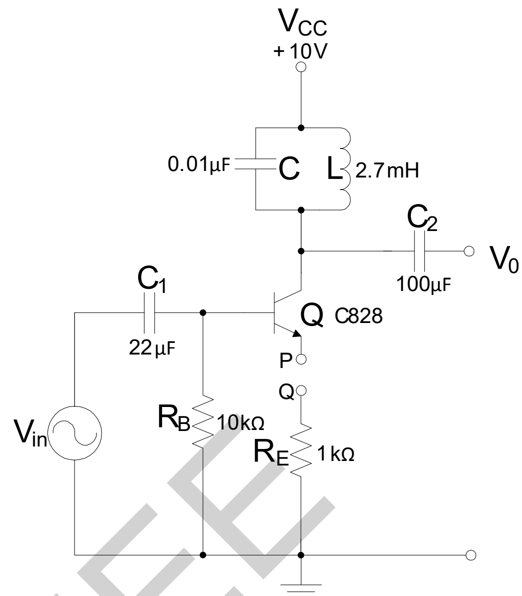

Fig 2.2: Class C Amplifier

Procedure:

Class B power amplifier:

1. Connect the circuit as shown in figure 2.1. Apply a sinusoidal signal at the input of frequency 1 kHz. The peak-to-peak value is 2V.
2. Observe the output. If distorted, vary the frequency and the amplitude of the input signal to obtain undistorted output. Crossover distortion should occur as the output switches from positive cycle to negative cycle and vice versa. Draw the waveshape.
3. Vary the frequency keeping the amplitude fixed and observe its effect on crossover distortion.
4. Vary the amplitude keeping the frequency fixed and observe its effect on crossover distortion.
5. For any undistorted output take reading of peak output voltage. This is V_{0max}.
6. Disconnect the oscilloscope probe from the output as well as from the input and ground and then connect one of the channels of the oscilloscope across R_{CC}. Draw its waveshape and take readings of its maximum value. Divide it by R_{CC} to obtain I_{max}.
7. Repeat steps 2, 3 and 4 for a square wave input.

Class C power amplifier:

1. Connect the circuit as shown in figure 2.2. Connect point P to ground.
2. Apply a sinusoidal signal having a peak-to-peak value of 80mV. Vary the frequency until a distortion free output is obtained. Draw the output wave shape.
3. Vary the frequency above and below cut-off. Take readings of output voltages at different frequencies.
4. Disconnect P from ground and connect it to Q. Observe the current wave shape by observing the voltage wave shape across R_E. Draw the wave shape.
5. Apply a square wave signal of resonant frequency at the input. Observe the output.

Reports:

Class B:

1) Submit the wave shapes drawn at steps 2, 6 and 7.
2) Calculate efficiency.
3) Explain why crossover distortion is absent for square wave input.

Class C:

1) Submit the wave shapes drawn at steps 2 and 4 and explain why voltage waveform is sinusoidal but current waveform is pulsating.
2) Calculate the resonant frequency. Plot Output vs. Frequency and verify that the maximum output is obtained at resonant frequency.

References:

1) Basic Electronics and Linear Circuits- N N Bhargava

2) Electronic Devices and Circuit Theory-Robert Boylestead

3) Integrated Electronics: Analog and Digital Circuits and Systems- Millman and Halkias

4) Microelectronic Circuits-Sedra and Smith

Experiment No: 03

Name of the experiment: Study of a voltage series feedback amplifier circuit.

Theory:

Feedback is the process of taking a part of output signal and feeding it to the input circuit. If this fed back signal adds to the input, the phenomenon is called positive feedback and if it subtracts from the input, this is called negative feedback.

Negative feedback decreases gain. Still it is an important technique that is applied in many circuits in order to obtain desirable characteristics. Negative feedback improves the stability of amplifier gain, reduces distortion due to noise, and increases input resistance and decreases output resistance. Moreover, for negative feedback, the lower cut-off frequency reduces and higher cut-off frequency increases. This results in higher bandwidth since bandwidth,

$$W = f_H - f_L$$

Where, f_H and f_L are higher and lower cut-off frequencies respectively. An interesting feature is to note that bandwidth increases by the same factor by which gain is reduced. Thus gain-bandwidth product remains constant.

Depending upon what (voltage or current) is sampled from output and how (in series or in parallel) it is fed back to input, there are four feedback topologies namely, voltage series, voltage shunt, current series and current shunt feedback.

For voltage series feedback, a part of output voltage is sampled and it is fed back (negative) in series with the input. Voltage gain with feedback (A_{Vf}) in terms of voltage gain without feedback (A_V) is given by,

$$A_{Vf} = \frac{A_V}{D}$$

The expression for output resistance with feedback ($R_{0f=}$) in terms of that without feedback (R_0) is as follows:

$$R_{0f} = \frac{R_O}{D}$$

Where, $D=1+\beta A_V$ with β=feedback voltage/output voltage.

For figure 3.1,

$$\beta = \frac{R_E}{R_O + R_E}$$

If an external resistance is connected to the output terminal, it will be parallel with the output resistance of the circuit. Thus, if the externally connected resistance equals the output resistance, then equivalent output resistance is halved which results in an output voltage that is half of the open circuit output voltage. This phenomenon is applied to measure output resistance experimentally.

Objective:

To study the voltage gain, bandwidth and output resistance under voltage series feedback condition and without feedback conditions for a single stage CE amplifier configuration.

Equipments:

Serial no.	Component Details	Specification	Quantity
1.	Transistor	C828	1 piece
2.	Resistor	100 Ω, 1 KΩ, 100 KΩ	1 pieces, 1 piece, 2 pieces
3.	POT	1 KΩ, 10 KΩ	1 piece each
4.	Capacitors	10 µF, 100 µF	3 pieces, 1 piece
5.	Oscilloscope		1 unit
6.	AC Signal generator		1 unit
7.	DC Power Supply		1 unit
8.	Digital Multimeter		1 unit
9.	Trainer Board		1 unit

Circuit Diagram:

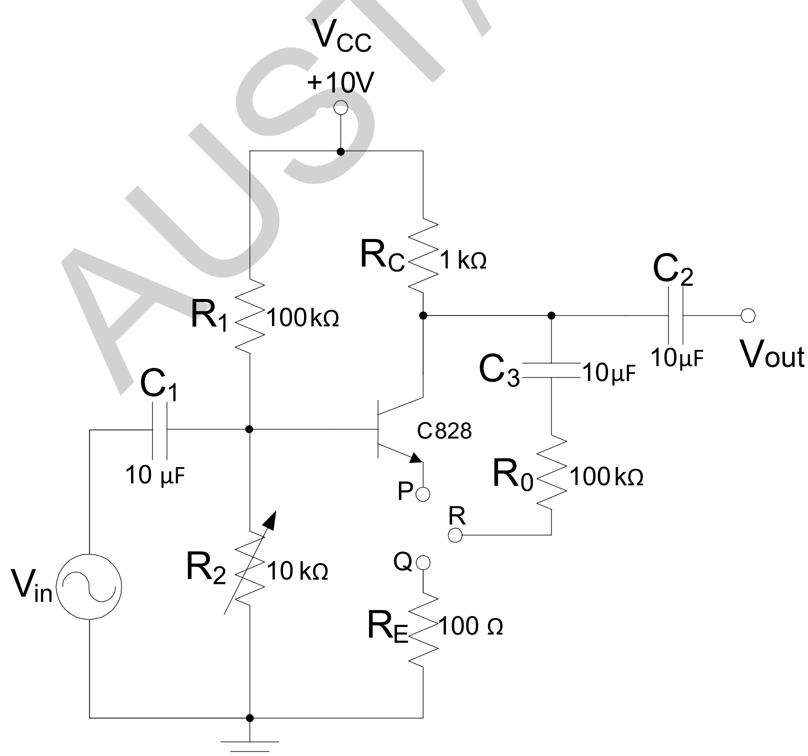

Fig 3.1: Study of a voltage series feedback amplifier circuit

Procedure:

1) Connect point P to ground.

2) Without applying V_{in} vary 10K POT so that V_{CE} =5V.

3) Apply V_{in} with $1KH_Z$ and vary its amplitude to obtain undistorted output. Keep V_{in} constant during the experiment.

4) Calculate voltage gain without feedback, $A_V = V_0/V_{in}$. V_0 and V_{in} can be measured by multimeter in ac mode. Also readings for V_0 and V_{in} can be taken from the oscilloscope.

5) Connect 1K or 10K POT at the output terminal and vary it until the voltage is half of the open circuit voltage. Now disconnect the POT and measure its value. This is output resistance without feedback, R_0.

6) Keeping input constant vary its frequency from $10H_Z$ to $1MH_Z$ measure V_0. Remember that you have to vary frequency in step like 10, 100, 1K, 2K,……, 10K, 20K,…., 100K, 200K,…, 1M etc. so that you can plot these on semilog paper.

7) Apply voltage series feedback by connecting P, Q, R, and repeat steps 4, 5 and 6 sequentially.

Reports:

1) Plot gain vs. frequency on semi log paper and calculate bandwidth for both cases.

2) Calculate β and D. Then calculate A_{Vf} and R_{0f} using this value of D and compare them with those obtained experimentally.

3) Justify the statement ‘’Gain-bandwidth product is constant’’ from your experimental data.

References:

1) Basic Electronics and Linear circuits-N N Bhargava

2) Integrated Electronics: Analog and Digital Circuits and Systems- Millman and Halkias

Experiment No: 04

Name of the experiment: Study of a current series feedback amplifier circuit.

Theory:

For current series feedback, a part of output current is sampled after multiplication with the feedback factor β, it is converted in voltage and this is then fed back (negative) in series with the input.

For current series feedback, β is given by,

$$\beta = \frac{V_f}{I_O} = -\frac{I_O R_E}{I_O} = -R_E$$

$$G_M = \frac{I_O}{V_{in}} = \frac{V_O}{R_L V_{in}} = \frac{A_v}{R_L}$$

$$D = 1 + \beta G_M$$

Then output resistance with feedback is given by,

$$R_{of} = R_O D$$

Objective:

To study the voltage gain, bandwidth and output resistance under voltage series feedback condition and without feedback conditions for a single stage CE amplifier configuration.

Equipments:

Serial no.	Component Details	Specification	Quantity
1.	Transistor	C828	1 piece
2.	Resistor	100 KΩ, 33 KΩ, 2.2 KΩ, 1 KΩ, 2.2 Ω	1 piece each
3.	Resistor	10 KΩ	2 pieces
4.	POT	10 KΩ	1 piece
5.	Capacitor	10 µF	4 pieces
6.	Oscilloscope		1 unit
7.	AC Signal generator		1 unit
8.	DC Power Supply		1 unit
9.	Digital Multimeter		1 unit
10.	Trainer Board		1 unit

Circuit Diagram:

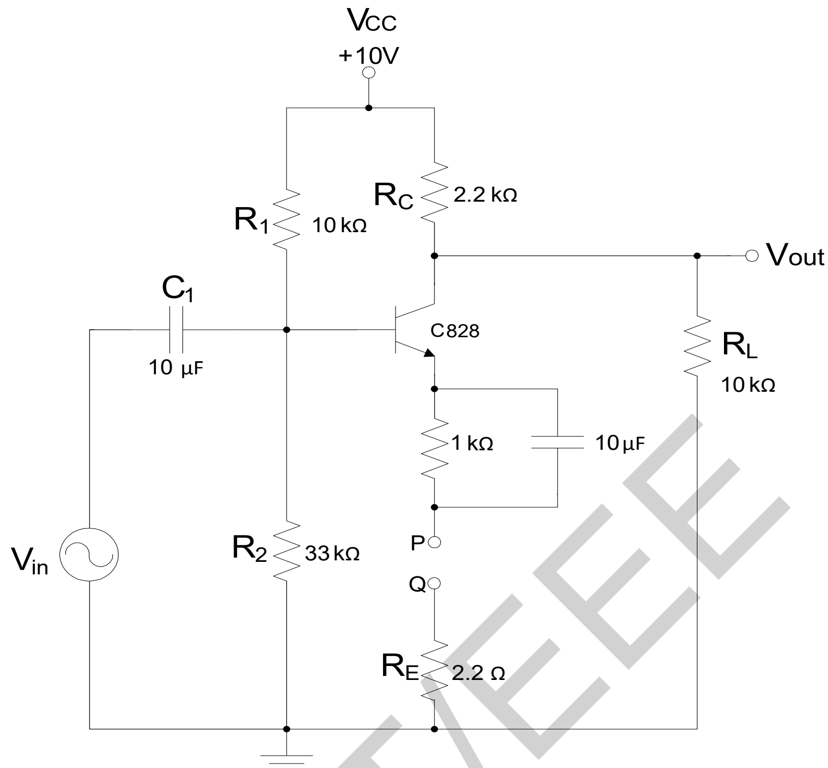

Fig 4.1: Study of current series feedback amplifier.

Procedure:

1) Connect point P to ground.
2) Apply V_{in} with $10kH_Z$. V_{in} should be in between 10mV(p-p) and 20mV(p-p).
3) Keep V_{in} constant during the experiment.
4) Calculate voltage gain without feedback, $A_V=V_0/V_{in}$. V_0 and V_{in} can be measured by oscilloscope. Also readings for V_0 and V_{in} can be taken from the oscilloscope.
5) Connect 1k or 10k POT at the output terminal and vary it until the voltage is half of the open circuit voltage. Now disconnect the POT and measure its value. This is output resistance without feedback, R_0.
6) Keeping input constant vary its frequency from $10H_Z$ to $1MH_Z$ and measure V_0. Remember that you have to vary frequency in step like 10, 100, 1K, 2 K,……., 10K, 20K,……, 100K, 200K,….,1M etc. so that you can plot these on semi-log paper.
7) Apply current series feedback by connecting P, Q and repeat steps 4, 5 and 6 sequentially.

Reports:

1) Plot gain vs. frequency on semi-log paper and calculate bandwidth for both cases.
2) Calculate β and D. Then calculate A_{Vf} and R_{of} using this value of D and compare them with those obtained experimentally.
3) Justify the statement ''Gain-bandwidth product is constant'' from your experimental data.

References:

1) *Basic Electronics and Linear Circuits- N N Bhargava*
2) *Integrated Electronics: Analog and Digital Circuits and Systems- Millman and Halkias*

Experiment No: 05

Name of the experiment: Introduction to OPAMPs.

Introduction:

The operational amplifier (abbreviated as OPAMP) is a direct-coupled high-gain amplifier to which feedback is added to control its overall response characteristic. It has very high (ideally ∞) input impedance, very low (ideally 0) output impedance and large bandwidth and its characteristics do not drift with temperature. It offers all the advantages of monolithic integrated circuits: Small size, high reliability, reduced cost, temperature tracking and low offset voltage and current. For these reasons, it has gained wide acceptance as a versatile, predictable and economic system building block.

An OPAMP may be used to perform many mathematical operations. Some of these basic applications are studied in this experiment

Inverting adder adds up the signals at its inverting input terminal and produces the inverse of this summation at the output, provided, the value of the feedback resistance and the resistance in series with the input signals are chosen correctly.

Voltage follower produces almost the same output as is applied to its input. The input resistance of a voltage follower circuit is very high (several mega ohms). Therefore, it draws negligible current from a signal source. Thus it works as a voltage buffer that provides a means of isolating an input signal from a load.

An important feature that must be taken into account while designing circuits with OPAMPs is Slew rate. It is the time rate of change of the closed loop amplifier output voltage under large signal conditions. It tells how fast the output voltage of an OPAMP changes and limits the output frequency f_{max} for distorted output. The relation between f_{max} and slew rate is expressed as

$$f_{max} = \frac{slew\ rate}{2\pi \times V_0}$$

Where, V_0 is the maximum distorted output voltage in volts, f_{max} is the maximum operable frequency in Hz and the slew rate is in volts per microsecond.

Equipments:

Serial no.	Component Details	Specification	Quantity
1.	OPAMP	741	1 piece
2.	Resistor	2.2 MΩ,100 KΩ, 10kΩ	1 piece, 1piece, 4 pieces
3.	Oscilloscope		1 unit
4.	AC signal generator		1 unit
5.	DC Power Supply		3 units
6.	Digital Multimeter		1 unit
7.	Trainer Board		1 unit

Circuit Diagram:

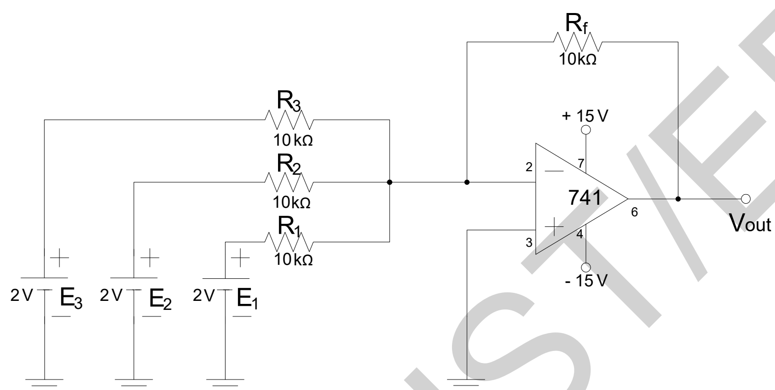

Fig 5.1

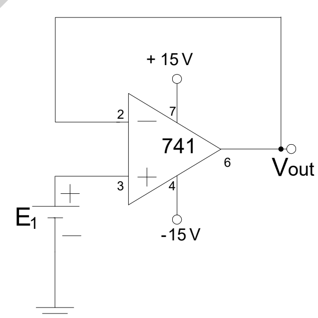

Fig 5.2

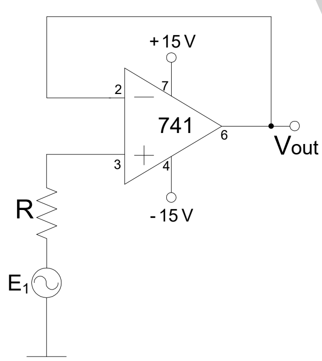

Fig 5.3

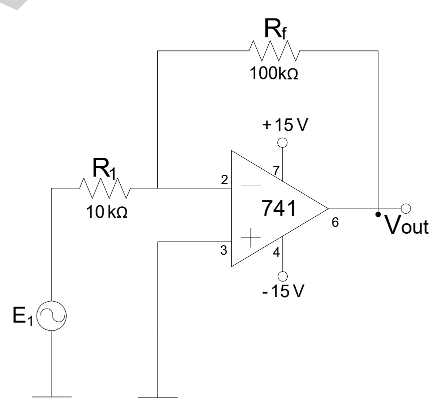

Fig 5.4

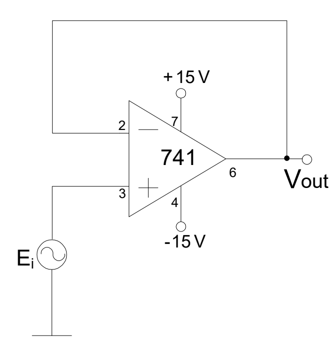

Fig 5.5

Procedure:

a) Study of inverting adder:

1) Connect the circuit of figure 5.1 and obtain values of V_0 for

- $E_1=E_2=E_3=2V$
- $E_1=3V$, $E_2=-3V$, $E_3=0V$
- $E_1=5V$, $E_2=-0V$, $E_3=2V$

2) Put R_1= 10K, R_2= 20K, R_3= 50K, R_f= 100K and repeat step 1.

b) Study of the voltage follower:

1) Connect the circuit of figure 1.2 and measure V_0 for E_1=4V and -4V.

2) To measure the input resistance of a voltage follower, change E_i to 5V rms at 100 Hz (sine wave) and connect R= 2.2MΩ in series with the source as shown in figure 1.3.

c) Measuring frequency response of small signal amplifier:

1) Connect the circuit of figure 5.4.

2) Adjust the input voltage E_2 to some convenient value, say 100mV (p-p)

3) Adjust the frequency to obtain undistorted maximum output. Measure the midband output voltage and calculate the voltage gain.

4) Keeping E_i constant, reduce the frequency until the output drops to 0.707 of its midband value and read the lower cut-off frequency, f_L. Next increase the frequency until the output again drops to 0.707 of its midband value and read upper cut-off frequency f_H.

5) Now connect E_1 to non-inverting terminal of figure 1.4 and following the above procedure find f_L and f_H at this condition.

d) Slew Rate:

1) Connect the circuit of figure 5.5 and apply 10V (p-p) sinusoidal input E_i and increase the frequency until the output voltage, V_0 is distorted. Take readings for f_{max} and V_0.

2) Do the same with 1V (p-p) sinusoidal input.

Reports:

1) For inverting adder, do the experimental results support theory?
2) Calculate input resistance of voltage follower.
3) Calculate bandwidth for both cases in the experiment of frequency response measurement. Use Bandwidth B, f_H, f_L.
4) For OPAMP 741, slew rate is 0.5v/µS. Calculate maximum frequency for undistorted output with inputs 10V (p-p) and 1V (p-p). Compare them with the experimental results.

References:

1) *Operational Amplifiers and Linear Integrated Circuits –Coughlin and Driscoll*
2) *Electronic Devices and Circuit Theory-Robert Boylestead*
3) *Integrated Electronics-Millman and Halkias*

Experiment No: 06

Name of the experiment: Application of OPAMPs.

Theory:

Integrator and Differentiator:

OPAMPs can be used to perform mathematical operations like integrations and differentiation.

The ideal performance equation for integrator is:

$$V_O = -(\frac{1}{RC})\int V_{in}dt$$

And that of differentiator is:

$$V_O = -RC\,\frac{d(V_{in})}{dt}$$

In this experiment, a square wave is at first integrated to produce a triangular wave of same frequency. This is then differentiated to reproduce the square wave (fig.6.2).

Free Running Multivibrator:

OPAMP circuits can be used to generate signals like the square wave, triangular wave, saw tooth wave, sine wave etc. accordingly, the signal generator is classified by the shape of the wave it generates. However, some circuits are so widely used that they have been assigned a special name. One shot or monostable multivibrator, free running or astable multivibrator, etc. are of this category.

A free running or astable multivibrator is a square wave generator. The circuit of fig.6.3 is a multivibrator circuit. The resistor R_1 and R_2 form a voltage divider to feedback a fraction of the output (V_0) to the (+) input. When V_0 is at $+V_{sat}$, the feedback voltage is called the upper-threshold voltage V_{UT}, where,

$$V_{UT} = (\frac{R_2}{R_1+R_2})\ (+V_{sat})$$

Resistor R_f provides a feedback path to the (-) input. When V_0 is at $+V_{sat}$, current flows through R_f to charge the capacitor C towards V_{UT}. As long as the capacitor voltage V_C is less than V_{UT}, the output voltage remains at $+V_{sat}$.

When V_C charges to a value slightly greater than V_{UT}, the (-) input goes positive with respect to the (+) input. This switches the output from $+V_{sat}$ to $-V_{sat}$. The (+) input is now held negative with respect to the ground because the feedback voltage is negative ad given by

$$V_{LT} = (\frac{R_2}{R_1+R_2})\ (-V_{sat})$$

Now V_C discharges to V_0 and recharges to V_{LT}. When V_C becomes slightly more negative than the feedback voltage V_{LT}, output voltage V_0 switches back to $+V_{sat}$ and the process repeats.

Frequency of Oscillation:

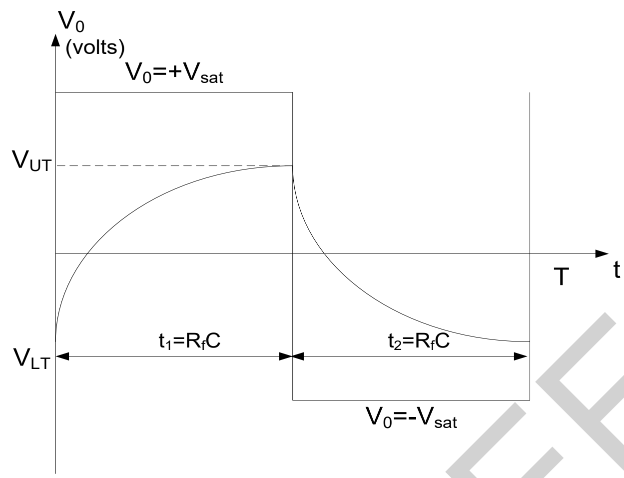

Fig 6.1

The capacitor and output voltage waveform for free running multivibrator are shown in fig 6.1.

The period of oscillation, T, is the time needed for one complete cycle. Since T is the sum of t_1 and t_2,

$$T = 2RfC \text{ for } R_2 = 0.86R_1$$

The frequency of oscillation,

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

Objective:

1) Study of Integrator and Differentiator.

2) Study of Free Running Multivibrator.

Equipments:

Serial no.	Component Details	Specification	Quantity
1.	OPAMP	741	2 pieces
2.	Resistor	100KΩ, 86KΩ, 10kΩ	2 pieces, 1 piece, 1 piece
3.	POT	10KΩ	1 piece
4.	Capacitors	0.1μF, 0.01 μF	1 piece each
5.	Oscilloscope		1 unit
6.	AC Signal generator		1 unit
7.	Digital Multimeter		1 unit
8.	Trainer Board		1 unit

Circuit Diagram:

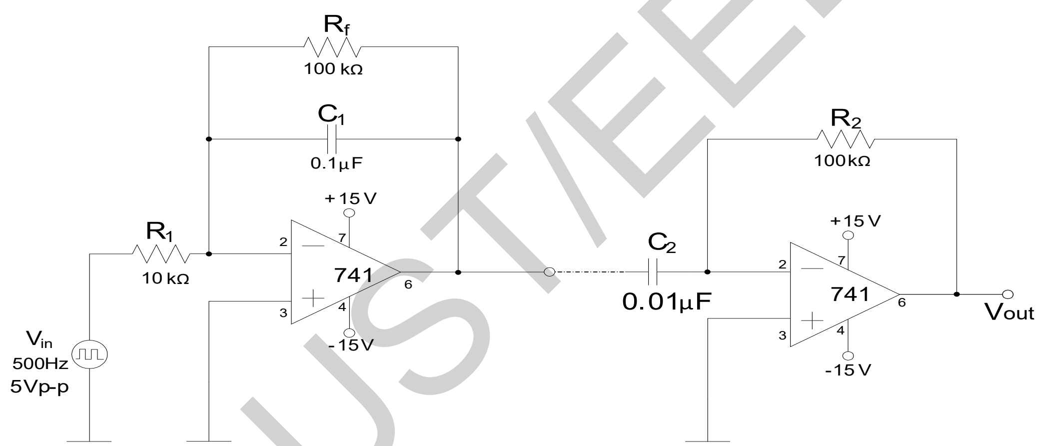

Fig 6.2

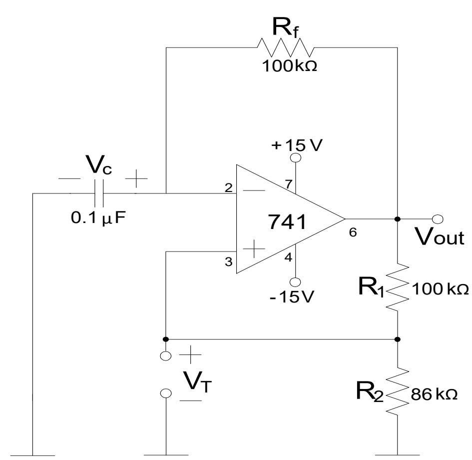

Fig 6.3

Procedure:

Integrator & Differentiator:

1) Make the connections as shown in figure 6.2.

2) Apply a square wave of 500Hz to the input of the integrator. Observe the wave shape at point A. This is the integrated output. The integrated output at point A should be a triangular wave. Take reading of its amplitude. Draw the wave shape.

3) Observe the output (of the differentiator) and input (to the integrator) at dual mode of oscilloscope. Notice the slightly damped output. Take reading of amplitudes. Draw the wave shapes.

4) Now insert a potentiometer in series with the input capacitor of the differentiator (at point A).

5) Adjust the value of the potentiometer to just prevent the overshoot and ringing in the differentiator output signal. Measure the resistance that has been introduced into the input path.

Free Running Multivibrator:

1) Complete the circuit of figure 6.3.

2) Observe the wave shapes of output (V_0) and voltage across the capacitor (V_C). Draw the wave shapes. Take reading for T_1, V_{LT} and V_{UT}.

Report:

Integrator & Differentiator:

1) Submit the wave shapes drawn at different steps of procedure.

2) Calculate the amplitudes of differentiator and integrator using the values of R and C. Compare them with those obtained from the experiment.

3) Why the output of the differentiator is slightly damped? Describe the measures taken to prevent damping.

Free Running Multivibrator:

1) Submit the wave shapes drawn at step 2.

2) Compute frequency (f) and threshold voltages (V_{LT} & V_{UT}) of the experimental multivibrator using the values of C, R_1, R_2 and R_f. Compare them with those obtained from the experiment.

3) How can you modify the free running multivibrator circuit to get a square wave of 18 $V_{(p-p)?}$

Experiment No: 07

Name of the experiment: Design and study of a -40dB/decade low pass Butterworth filter.

Theory:

A filter is a circuit that is designed to pass signals of a specified band of frequencies while attenuating all signals outside this band. There are four types of filters: low pass, high pass, band pass and band elimination filters. A low pass filter is a circuit that has a constant output voltage from dc up to a cut-off frequency f_C. High pass filters attenuate the output voltage for all frequencies below f_C. Above f_C, the magnitude of the output voltage is constant. Band pass filters attenuate the output voltage for all frequencies outside the band. Band elimination filters perform in an exactly opposite way; i.e. they reject a specified band of frequencies while passing all frequencies outside the band.

A popular application uses OPAMPs to build active filters. A filter circuit may be constructed using passing components: resistors, capacitors etc. An active filter additionally uses an amplifier to provide voltage amplification and signal isolation or buffering. OPAMPs are a good choice for this purpose.

For ideal filters, the voltage gain inside the pass band should be constant and outside the pass band it should be zero. But this is not true for practical filters. Actually, output does not reduce to zero instantly rather it decreases slowly. This is called roll-off. As the roll-off becomes steeper, the filter approaches the ideal filter more closely. Many applications require steeper roll-offs after the cut-off frequency. One common filter configuration that gives steeper roll-offs is the Butterworth filter. For a -40dB/decade Butterworth filter, above higher cut-off frequency and below lower cut-off frequency, gain reduces by 40dB for every 10 times change in frequency.

Equipments:

Serial no.	Component Details	Specification	Quantity
1.	Transistor	C828	1 piece
2.	Resistor	100 Ω, 10 KΩ, 1 kΩ	2 pieces, 1 piece, 1 piece
3.	Capacitors	As chosen for the design (100 pF to 0.1 μF), 30 μF	1 piece each
4.	Oscilloscope		1 unit
5.	AC Signal generator		1 unit
6.	Digital Multimeter		1 unit
7.	Trainer Board		1 unit

Circuit Diagram:

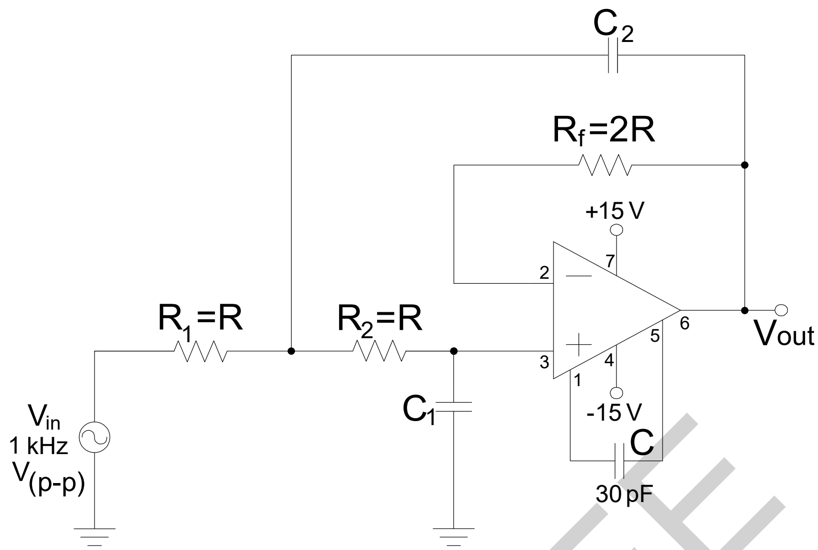

Fig 7.1

Design:

1) Choose the cut-off frequency ω_C and f_C(assume f_C = $1\ kHz$).

2) Pick C_1 ; choose a convenient value between 100 pF and 0.1 µF.

3) Make C_2 = $2C_1$

4) Calculate, R = $0.707/\ (\omega_C C1)$

5) Choose R_f = $2R$

Experiment:

1) Assemble the circuit with the obtained components.

2) Apply a sinusoidal voltage of frequency 1 kH$_Z$. Adjust the amplitude to obtain undistorted output.

3) Observe the wave shapes with the oscilloscope in dual mode. Take readings of amplitudes of input & output so that you can calculate gain. Also take readings of phase angle difference between input & output.

4) Keeping amplitude fixed, vary frequency of the input above and below f_C and take readings of output and phase angle difference at different frequencies. (Frequency should be varied at least up to 3dB point i.e. when output amplitude reduces to at least 0.707 of its midband value).

Reports:

1) Obtain "Gain vs. Frequency" curve.

2) Obtain "phase angle vs. frequency" curve.

3) What are the differences between active and passive filters?

4) What are the main characteristics of a Buttterworth filter?

Experiment No: 08

Name of the experiment: Study of Wien Bridge Oscillator

Theory:

Any circuit that generates an alternating voltage is called an oscillator. To generate ac voltage, the circuit is supplied energy from a dc source. Oscillators those deliver sinusoidal output waveform even without input signal excitation, are called sinusoidal Oscillators.

Sinusoidal oscillators use positive feedback with a unity loop gain. For unity loop gain, the feedback signal can be used as input since its amplitude; phase and frequency will not be change.

The behavior of sinusoidal oscillators is governed by the **Barkhausen Criterion**. It states that the frequency at which a sinusoidal oscillator will operate is the frequency for which the total shift introduced, as a signal proceed from the input terminals, through the amplifier and feedback network, and back again to the input, is precisely zero. Stated more simply, the frequency of a sinusoidal oscillator is determined by the condition that the loop gain phase shift is zero. However, in every practical oscillator the loop gain is slightly larger than unity, and the amplitude of the oscillations is limited by the onset of nonlinearity.

An oscillator circuit in which a balanced bridge is used as the feedback network is the Wien Bridge Oscillator. In this circuit, the active element is an OPAMP, which has a very large positive voltage gain, negligible output resistance and very high input resistance. For figure 8.1, to satisfy the Barkhausen Criterion, Z_1 and Z_2 should have the same phase angle. This occurs at the frequency,

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

The frequency of oscillation is precisely the null frequency of the balanced bridge. For the balanced bridge, if a null is desired, R_1 and R_2 should be chosen so that V_i =0. This condition is satisfied for R_1 =R_2. Frequency of oscillation can be varied by varying R or C. For practical circuit, changes in frequency range are accomplished by switching in different values for the two identical resistors R and continuous variation of frequency is obtained by varying simultaneously the two capacitors (ganged variable air capacitor).

Equipments:

Serial no.	Component Details	Specification	Quantity
1.	OPAMP	741	1 piece
2.	Resistor	10 KΩ, 20 KΩ, 50 KΩ, 100 KΩ	3 pieces, 1 piece, 1 piece, 1 piece
3.	Capacitors	10 µF	2 pieces
4.	Oscilloscope		1 unit
5.	Trainer Board		1 unit

Circuit Diagram:

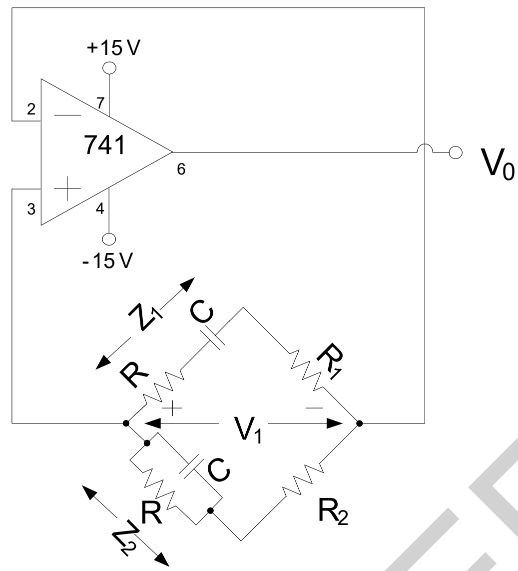

Fig: 8.1

Procedure:

1) Connect the circuit as shown in figure 6.1. Choose C=0.022µF, R=10k, R_1=22k, R_2=10k.

2) Observe the output. Draw the wave shapes. Take reading of time period (T) from the oscilloscope.

3) Assemble the circuit with R_1=100K, R_2=47K. (If required, two 100K can be paralleled to obtain 50K). Then repeat step 2.

Reports:

1) For both cases, calculate frequency using $f_0=1/T$. This is practical frequency of oscillation. Compare them with those calculated theoretically using

$$f_0=\frac{1}{2\pi RC}$$

2) How amplitude of Wien Bridge Oscillator can be stabilized against temperature variation?

References:

1) Integrated Electronics: Analog and Digital Circuits and Systems- Millman and Halkias.
2) Basic Electronics and Linear circuits-N N Bhargava