Laboratory Journal

of

ELECTRONIC DEVICES AND CIRCUITS-II

For completion of term work of 5th *semester*

curriculum program

Bachelor of Technology

In

ELECTRONICS AND TELECOMMUNICATION ENGINEERING

DEPARTMENT OF ELECTRONICS AND TELECOMMUNICATION

ENGINEERING

Dr. BABASAHEB AMBEDKAR TECHNOLOGICAL UNIVERSITY

Lonere-402 103, Tal. Mangaon, Dist. Raigad (MS)

INDIA

INDEX

EXPERIMENT NO.1

Aim: To design single stage CE amplifier & study its frequency response.

Apparatus: Resistor, capacitor, transistor, CRO, Function Generator

Theory: Frequency response of CE amplifier.

Frequency response of amplifier is a graph of amplifier gain & frequency. It shows the variation in gain with respect to frequency. The practical frequency can be divided into three major regions as shown in diagram.

Various capacitors are responsible for reduction in gain in high or low frequency region.

The three regions are:

1. Low frequency region:

The amplifier gain in low frequency region decreases due to coupling and bypass capacitors in the amplifier.

2. Mid frequency region:

In this region gain & output voltage remains constant.

3. High frequency region:

In this region output voltage & gain will decrease due to the transistors, internal capacitors & stray capacitance. The lower and higher cut-off frequency f_H & f_L respectively corresponds to 3dB reduction is the mid-band region as in figure. The difference between f_H & f_L is known as bandwidth.

Low Frequency Amplifier Response Using BJT:

A typical capacitively coupled single stage amplifier in CE configuration is studied.The BJT amplifier has 3 high pass RC circuits. One each associated with C_{in} , C_E & C_{out}. These circuits will affect the amplifier in the low frequency region.

The RC network that decide the low frequency amplifier of BJT are as follows:

1. R_E & C_E
2. R_{in} & C_{in}
3. R_C & C_{out} (R_L too added if present)
4.

Steps to design single stage CE amplifier:

Assume suitable values for stability & frequency (min) if not provided.

1. To determine R_C

   $|A_V| = (\frac{hfe(min)RC}{hie+\Delta hRc}$

   $\Delta h = h_{ie}h_{oe}+h_{re}h_{fe}$

2. To find operating point:

   $I_{CQ}=\frac{V(op)}{Rc}$

   $V_{CE} \geq V_{o\,(p)} + V_{(sat)} + 1$

Q(V_{CE} , I_{CQ}) is operating point.

3. To find V_{CC}

   $V_{CC} \geq V_{CEQ} + V_{(O)P} + 1.5V$

5. To find R_1,R_2,R_E

   $V_E = 10\%V_{CC}$

   $I_E = I_{CQ}$

   i. $R_E = \frac{VE}{IE}$

      $R_{B\,=}$ (s-1)R_E

   ii. $R_{1\,=}\frac{Rb\ Vcc}{VB}$ $\quad V_B = V_{BE} + V_E$

   iii. $R_2 = R_1R_B /R_1 - R_B$

6. To determine capacitors :

   i. $C_{in} = 1/(2\pi F_{low}Z_i)$

      $Z_i = R_1 \,||\, R_E \,||\, h_{ie}$

   ii. $C_E = 1/2\pi(0.1)R_E$

   iii. $C_{out} = 1/(2\pi F_{low}Z_o)$

Procedure:

i. Design the circuit as per the given specification.
ii. Give input from function generator.
iii. Observe output from CRO & note down readings for specified frequency range.
iv. Draw the frequency response graph.

Conclusion:

We have studied design of single stage CE amplifier & frequency response of amplifier.

Circuit diagram:

Experiment No. 2

Aim: Study and design of effect of negative feedback on gain of RC–Coupled Amplifier

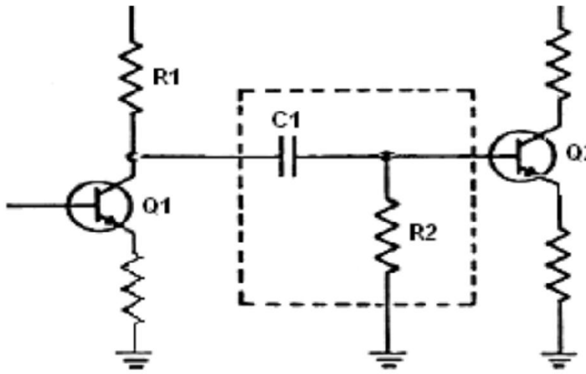
Apparatus:

1. Analog board AB64
2. DC power supply +12V from external source or ST2612 Analog Lab
3. Function Generator ST4064
4. Oscilloscope
5. 2mm patch cords

Theory:

` Single amplifier circuits, such as a common base, common emitter and common collector amplifiers are seldom found alone, as a single stage amplifier, in any system. Generally, at least two or more than two stages are connected in cascade combination. If the output of one amplifier is connected (coupled) to the input of another amplifier the stages are said to be connected in "cascade". The advantage of cascaded amplifiers is to develop an output voltage larger than either stage alone can develop. In fact, the overall gain of the cascaded amplifiers (called system gain) is the product of each individual stage gain. Because of this the gain of a single stage is not as important as the system gain. Designers usually set individual stage gains relatively low to reduce signal distortion.

One of the very important requirements to cascade one stage of amplifier to another is the impedance matching. When the output impedance of previous stage matches with the input impedance of its next stage, maximum power is transferred. One of the coupling methods to couple the two stages is RC-Coupling. RC-Coupling has the advantages of wide frequency response and relatively small cost and size. RC coupled amplifier is simple & low cost circuit .In these circuit voltage divider biasing is used for drive the transistors BC547. Here in the circuit coupling capacitor is using before an input. Since the impedance of the capacitor is inversely proportional to frequency, the capacitor effectively blocks DC voltage and transmits AC voltage. When the frequency is high enough, the capacitive reactance is much smaller than the résistance. So capacitor used for this purpose is called a coupling capacitor.



An RC-Coupled is the coupling network. C1 is the coupling capacitor which connects the output of Ql to the input of Q2. R2 will develop the signal to be applied to the base of Q2. Cl acts as a limiting factor at low frequencies because its reactance increases with a decrease in frequency and some point will be reached when a voltage drop will appear across it.

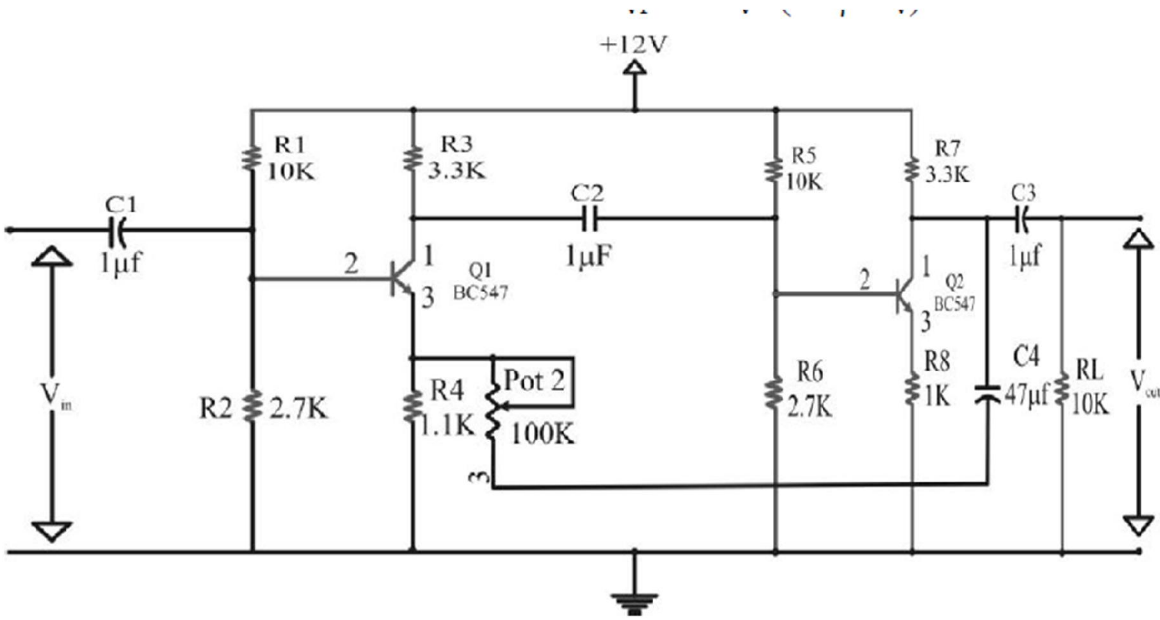
This will reduce the size of the signal being applied to Q2. At medium frequencies the reactance of Cl is so small that it can be considered a short to the signal. C1 will also isolate any DC voltage developed at the collector of Q1 from the DC bias developed at the base of Q2.

In the figure shown below RC feedback amplifier figure 2 we have used two-stage amplifier with the negative feedback. There are different feedback types for constant gain purpose. Feedback is of two types, positive feedback and negative feedback. Positive feedback used in Oscillators and Negative feedback used in amplifier circuits. Without any feedback, output changes with atmospheric conditions. That's why feedback is used for constant output. In the given circuit feedback from output is connected at emitter of the first stage through a feedback capacitor and resistance, capacitor is just a coupling capacitor butresistance is providing the amount of the feedback, means the voltage appeared at the emitter of first stage. Here the emitter voltage VE1 depends upon the output voltage variation because of the feedback resistance RF, voltage (VE1) is RE1.VOUT/ (RE1+ RF). Here gain is directly proportional to the feedback resistance. With using feedback gain is decrease. The feedback factor is,

$$ß = RE1/ (RE1+RF) \quad \ldots\ldots\ldots(1)$$

Feedback Factor range is 0.01 to 1 and Gain with feedback is

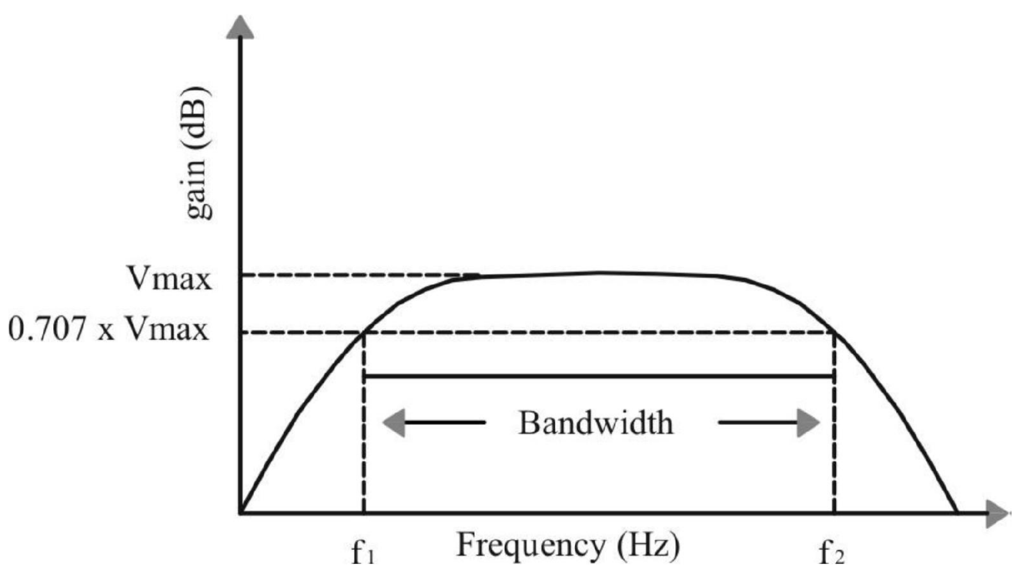
$$AVF = AV / (1+ ß\, AV) \qquad \ldots\ldots\ldots(2)$$



RC coupled amplifier with feedback

Bandwidth of an RC-Coupled Amplifier:

Bandwidth is a term used to describe the band of frequencies at which a particular amplifier amplifies the given input effectively.



The f1 and f2 points are known as half power points. The half power points are the points at which the signal amplitude dropped to 0.707 (3 dB) of the maximum signal amplitude. Any frequency below f 1 or above f 2 point is not considered a usable output from the amplifier.

The bandwidth of the amplifier is the difference between the f 1 and f 2 points. It is generally accepted that in an RC-Coupled Amplifier the fl point is established by the coupling capacitor and by-pass capacitors and the f 2 point is set by the "shunt" or "stray wire" capacitance. Using Negative feedback bandwidth increases.

The bandwidth of RC-Coupled Amplifier is given by,

Bandwidth (B) = fH – fL ...……….(3)

Procedure:

1. Connect +/-12V variable DC power supply at the indicated position from external source or ST2612 Analog Lab.
2. Switch 'On' the power supply.
3. Connect 2Vp-p, 2 KHz Sine wave signal at the input (between points Vin and ground) of amplifier ofAB64 board and observe the same on Oscilloscope CH I.
4. Observe the output waveform between points Vout and ground on Oscilloscope CH II and note output voltage (VOUT1) peak to peak.
5. Calculate the Voltage Gain without feedback (AV1).
6. Vary the potentiometer RF and adjust it to 60K and calculate feedback factor ß using equation (1).
7. Connect point 'a' to point 'b' to make a negative feedback.
8. Observe the output waveform between points Vout and ground on Oscilloscope CH II and note output voltage (VOUT2) peak to peak.
9. Calculate the Voltage Gain with feedback (AVF).
10. Compare the gain obtained by the feedback and without feedback.

Result:

Voltage Gain of amplifier without feedback [VOUT1/ VIN] =…………

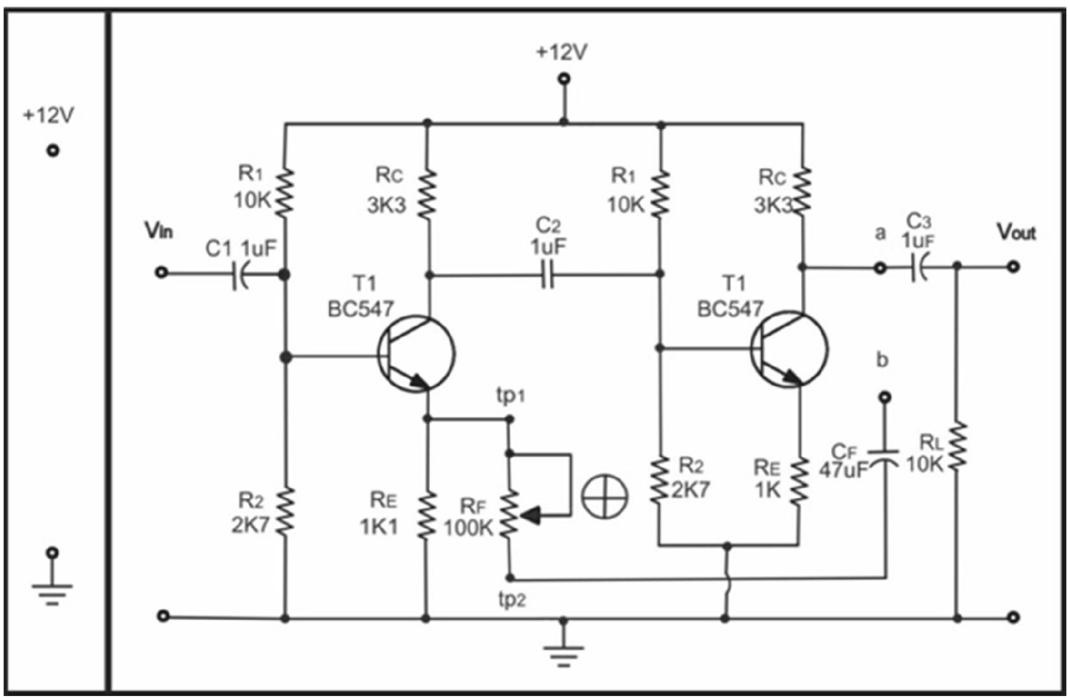
Voltage Gain of amplifier with feedback [VOUT2/ VIN] =…………

Feedback factor is……………

Conclusion:

We can see that using feedback overall gain decreases

Circuit diagram:



Observation Table:

Vin=..............

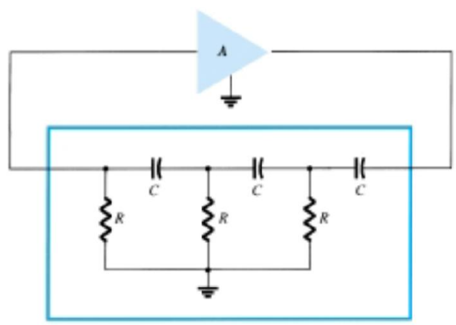
Frequency	**Vout**	**Gain=20*log(Vout/Vin)**

Experiment No.: 3

Aim: to study operation of RC phase shift oscillator using op-amp.

Theory:

An idealized version of phase shift oscillator is shown in Fig. 18.20. Recall that the requirements for oscillation are that the loop gain, _*A*, is greater than unity *and* that the phase shift around the feedback network is 180° (providing positive feedback). In the present idealization, we are considering the feedback network to be driven by a perfect source (zero source impedance) and the output of the feedback network to be connected into a perfect load (infinite load impedance). The idealized case will allow development of the theory behind the operation of the phase-shift oscillator. Practical circuit versions will then be considered.



Feedback network

Concentrating our attention on the phase-shift network, we are interested in the attenuation of the network at the frequency at which the phase shift is exactly 180°.

Using classical network analysis, we find that

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

$$\beta=1/29$$

and the phase shift is 180°.

For the loop gain βA to be greater than unity, the gain of the amplifier stage must be greater than $1/\beta$ or 29:

$$A > 29$$

When considering the operation of the feedback network, one might naively select the values of *R* and *C* to provide (at a specific frequency) 60°-phase shift per section for three sections, resulting in a 180° phase shift, as desired. This, however, is not the case, since each section of the *RC* in the feedback network loads down the previous one. The net result that the *total* phase shift be 180° is all that is important. The frequency given by Eq.is that at which the *total* phase shift is 180°. If one measured the phase shift per *RC* section, each section would not provide the samephase shift (although the overall phase shift is 180°). If it were desired to obtain exactly a 60° phase shift for each of three stages, then emitter-follower stages would be needed for each *RC* section to prevent each from being loaded from the following circuit._

If a transistor is used as the active element of the amplifier stage, the output of the feedback network is loaded appreciably by the relatively low input resistance (*hie*) of the transistor. Of course, an emitter-follower input stage followed by a common-emitter amplifier stage could be used. If a single transistor stage is desired, however, the use of voltage-shunt feedback more suitable. In this connection, the feedback signal is coupled through the feedback resistor *R*_ in *series* with the amplifier stage input resistance (*Ri*).

Analysis of the ac circuit provides the following equation for the resulting oscillator frequencies.

Conclusion:

Thus we have studied RC-Phase shift oscillator using op-amp.

CIRCUIT DIAGRAM:

CALCULATION:

Experiment no. 4

Aim: to study operation of RC phase shift oscillator using BJT.

Software: multisim

Theory:

In an RC phase shift oscillator three identical basic RC phase shifting network is cascaded as shown in the figure 1. Each RC network is designed to introduce a phase shift of 60^0. So total storage is180^0. This means output of network leads input at precisely 1 frequency required for oscillation.

RC phase shift oscillator using transistor

A typical phase shift oscillator using transistor is an active design . It consists of single stage CE amplifier and the RC phase shift ladder. Resistors R1,R2 and Re are connected for transistor biasing, CE is the emitter bypass capacitor.

Operation:

1. As shown in the fig 2, the output of Vo of the single stage amplifier has been connected as input to the RC phase shifting networks.
2. The output of network is given to the input of the amplifier.
3. As the CE amplifier introduces a phase shift of 180^0, the network add an additional 180^0. And makes the phase shift will be 00 only if frequency precisely equals to required frequency f. If gain of an amplifier is A and network factor is β they are adjusted properly to have loop gain $|A\beta| \geq 1$, to have sustained sinusoidal oscillations.
4. In this case, R_3 of RC ladder is slightly different and selected in such a way that R3 ≠Zi =R where Zi=input impedance.

   $=R_1 \parallel R_2 \parallel hie$
5. Frequency of oscillations of transistorized RC phase shift oscillator is given by $f=1/(2\pi RC\sqrt{6+4(\frac{Rc}{R})})$
6. Feedback factor of circuit is given as β= 4Rc/R +23+ 29/(Rc/R)
7. A typical RC phase shift oscillator has range from about 20Hz to 20KHz.

Advantages:

1. Simplicity of circuit
2. Useful for frequency in audio range.

3. Sine wave output can be obtained.

Disadvantages:

1. Poor frequency stability that keeps changing depending upon values of R,C, temperature, ageing etc.
2. Very difficult to get variable frequency output.

Steps for designing RC phase shift oscillator using BJT amplifier

To get R and C

i. Select R such that R>hie & R=Rc

ii. Now,

$F=1/(2\pi Rc\sqrt{10})$

$C=1/(2\pi RF\sqrt{10})$

Calculate C

To find oprating point

i. If $V_{O\,(p)}$ is given,

$Ic=V_{O(p)}/Rc$

Or

re=Rc/Av

& IE=26mv/re

ii. $VCE \geq VCE(sat)+V_{O(P)}+1$

$\geq 0.3+Vo(p)+1$

To find RE

$V_{CC} \geq Vo(p)+V_{CEQ}+1.5$

$V_E=10\%Vcc$

$V_E=I_ER_E$

4.To find R1 &R2

RB=(S-1)RE

$R1=R_BV_{CC}/V_B$

$V_B=V_{BE}+V_E$

$R2=R1.R_B/(R1-R_B)$

Assume suitable value for stability factor

5.To find all values of capacitor

Assume suitable value for fmin

CE=$\frac{1}{2\pi fmin*0.1*RE}$

Cin=1/(2πfmin Zi)

Zi=R1 || R2||hie

6. to find R3

R3= R-Zi

= R-(R1||R2||hie)

Procedure:

1. Make all connections as per circuit diagram and simulate it.
2. Give required DC supply Vcc.
3. Observe 60^0 phase shift in each mode.

Conclusion:

Hence we have designed RC phase shift oscillator and simulated it.

Circuit diagram:

Calculation:

EXPERIMENT NO.: 5

AIM: To study the operation of Colpitt oscillator.

Equipments Needed:

1. Analog board AB67.
2. DC power supplies +12 V from external sources or ST2612 Analog Lab.
3. Oscilloscope.
4. 2mm Patch cords.

THEORY

Oscillators are circuits that produce specific, periodic waveforms such as square, triangular, sawtooth, and sinusoidal.They can be made from some form of active or passive device like transistor, FETs and op-apm in association with devices such as resistors, capacitors, and inductors, to generate the output. .There are two main classes of oscillator : relaxation and sinusoidal. Relaxation oscillators generate the triangular, sawtooth and other nonsinusoidal waveforms. Sinusoidal oscillators consist of amplifires with external components used to generate oscillations, or crystals that internally generate the oscillations. The focus here is on sine wave oscillators. Sine wave oscillators are used as references or test waveforms by many circuits.

An oscillator is a type of feedback amplifier in which part of the output is fed back to the input via a feedback circuit. If the signal feedback is of proper magnitude and phase, the circuit produce alternating current or voltages. Two requirement for oscillations are

1. The magnitude of the loop gain Av.β must be atleast 1, and
2. The total phase shift of the loop gain Av.β must be equal to 0° or 360°. If the amplifier causes a phase shift of 180°, the feedback circuit must provide an additional phase shift of 180 degree so that the total phase shift around the loop is 360°.

Colpitt Oscillator:

The colpitt oscillator is one of the simplest and best known oscillators and is used extensively in circuits, which work at radio frequencies.Fig.1 shows the basic Colpitt oscillator circuit configuration. The transistor is in voltage divider bias which sets up Q-point of the circuit. In the circuit note that Vout is actually the ac voltage across C2.This voltage is fed back to the base and sustains oscillations developed across the tank circuit provided there is enough voltage gain at the oscillations frequency.

The resonant frequency of the colpitt oscillator can be calculated from the tank circuit used. We can calculate the approximate resonant frequency as

Fr = $1 \div 2\pi\sqrt{LC}$ ……………………………………………(1)

Here the capacitance used is the equivalent capacitance the circulating current passes through. In colpitt oscillator the circulating current passes through series combination of C1 and C2.Therefore equivalent capacitance is,

Ceqv = (C1 * C2)÷ (C1 + C2)

Procedure:

1. .Connect the patch cord between a and b terminal another one between ground and d.
2. Switch on the power supply.
3. Connect the oscilloscope between V out and ground.
4. Record the output frequency on CRO.
5. Calculate resonant frequency using equation.
6. Compare measured frequency and theoretical frequency.
7. Switch off the power supply.
8. Connect the patch cord between a and c.
9. Connect patch cord between e and ground.

Result:

1. For the combination of C_1, C_2, L
2. Resonanat frequency observed=
3. C_{eqe}=
4. Resonanat frequency calculated=
5. For the combination of C_3, C4, L
6. Resonanat frequency observed=
7. C_{eqe}=
8. Resonanat frequency calculated

Conclusion:

"Hence we studied the operation of colpitts oscillator"

Circuit Diagram:

Calculation:

EXPERIMENT NO.:6

AIM: To design colpitt oscillator using BJT

SOFTWARE: Multisim 11.0

Designing Steps:

$A_V=\beta=C_2/C_1$

To find frequecy:

$f=1/2\pi\sqrt{LC_{eq}}$

where $C_{eq}=(C_1.C_2/C_1+C_2)$

2. To determine R_C

   $|A_V| = (\frac{hfe(min)RC}{hie+\Delta hRc}$

   $\Delta h= h_{ie}h_{oe}-h_{re}h_{fe}$

2. To find operating point:

   $I_{CQ}=\frac{V(op)}{Rc}$

   $V_{CE}\geq V_{o\,(p)} + V_{CE\,(sat)} + 1$

$Q(V_{CE}, I_{CQ})$ is operating point.

3. To find V_{CC}

   $V_{CC} \geq V_{CEQ} + V_{(O)P} + 1.5V$

7. To find R_1,R_2,R_E

   $V_E = 10\%V_{CC}$

   $I_E = I_{CQ}$

   iv. $R_E = \frac{VE}{IE}$

   $R_B = (s-1)R_E$

   v. $R_1 = \frac{Rb.Vcc}{VB}$ $V_B = V_{BE} + V_E$

   vi. $R_2 = R_1R_B/R_1 - R_B$

8. To determine capacitors :

   iv. $C_{in} = 1/(2\pi F_{low}Z_i)$

   $Z_i = R_1 \,||\, R_E \,||\, h_{ie}$

   v. $C_E = 1/2\pi(0.1)R_E$

   vi. $C_{out} = 1/(2\pi F_{low}Z_o)$

Procedure:

1. Open the multisim software.
2. Select the components.
3. Make the connections as per circuit diagram.
4. Select CRO.
5. Observe the waveform on CRO.
6. Measure the output frequency 0n CRO.

Result:

For the combination of C1, C_2, L

Resonanat frequency observed=

C_{eqe}=

Conclusion: Hence we studied the operation of colpitts oscillator.

Circuit diagram:

Experiment No.-7

AIM: Study of the operation of Hartley Oscillator.

Equipments Needed:

1. Analog board of AB68.
2. DC power supplies +12V from external source or ST2612 Analog Lab.
3. Oscilloscope Caddo 802 or equivalent
4. 2 mm patch cords.

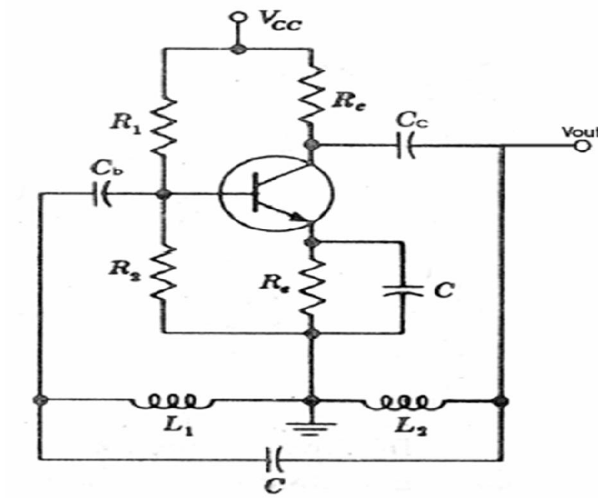
THEORY:

Oscillators are circuits that produce periodic waveforms without input other than perhaps a trigger. They generally use some form of active device, lamp, or crystal, surrounded by passive devices such as resistors, capacitors, and inductors, to generate the output. There are two main classes of oscillator: Relaxation and Sinusoidal. Relaxation oscillators generate the triangular, saw tooth and other non-sinusoidal waveforms. Sinusoidal oscillators consist of amplifiers with external components used to generate oscillation, or crystals that internally generate the oscillation. The focus here is on sine wave oscillators, created using operational amplifiers Op-Amps. Sine wave oscillators are used as references or test waveforms by many circuits.

An oscillator is a type of feedback amplifier in which part of the output is fed back to the input via a feedback circuit. If the signal fed back is of proper magnitude and phase, the circuit produces alternating currents or voltages. Two requirements for oscillation are:

1. The magnitude of the loop gain AVB must be at least 1, and
2. The total phase shift of the loop gain AVB must be equal to 0° or 360°. If the amplifier causes a phase shift of 180°, the feedback circuit must provide an additional phase shift of 180° so that the total phase shift around the loop is 360°.

Hartley Oscillator:



Hartley Oscillator

The Hartley oscillator is one of the simplest and best known oscillators and is used extensively in circuits, which work at radio frequencies. Figure 1 shows the basic Hartley oscillator circuit configuration. The transistor is in voltage divider bias which sets up Q-point of the circuit. The output voltage is fed back to the base and sustains oscillations developed across the tank circuit, provided there is enough voltage gain at the oscillation frequency.

The resonant frequency of the Hartley oscillator can be calculated from the tank circuit used. We can calculate the approximately resonant frequency as 1 &2

Fr p LTC= …………………………… (1)

Here, the Inductor used is the equivalent Inductance. In Hartley oscillator the circulating current passes through the series combination of L1 and L2. Therefore equivalent Inductance is,

LT = L1 + L2 + 2 M……….…………… (2)

Where M is the mutual inductance between two inductors

M = K √ L1 L2………………………… (3)

Where, K is the coefficient of coupling, lies between 0 to 1.The coefficient of coupling gives the extent to which two inductors are couple.Starting condition for oscillations is AB > 1 Where, B is approximately equal to L2/L1.

The feedback should be enough to start oscillations under all conditions as different transistor, temperature, voltage, etc. but it should not be much that you lose more output than necessary. The resonant frequency can be changed by either changing the value of inductor or changing the value of capacitor but the combination of the three components should satisfy the above given two conditions for oscillation

Procedure:

•Connect +12V DC power supplies at their indicated position from external source or ST2612 Analog Lab.

1. Connect a patch cord between points a and b and another patch cord between point d and e.
2. Switch on the Power Supply.
3. Connect Oscilloscope between Vout and ground on AB68 board.
4. Record the value of output frequency on oscilloscope.
5. Calculate the resonant frequency using equation 1.
6. Compare measured frequency with the theoretically calculated value.
7. Switch off the supply.
8. Remove the patch cord connected between points a and b and connect it between points a and c.
9. Remove the patch cord connected between points d and e and connect it between point d and f.
10. Follow the procedure from point 4 to 7.

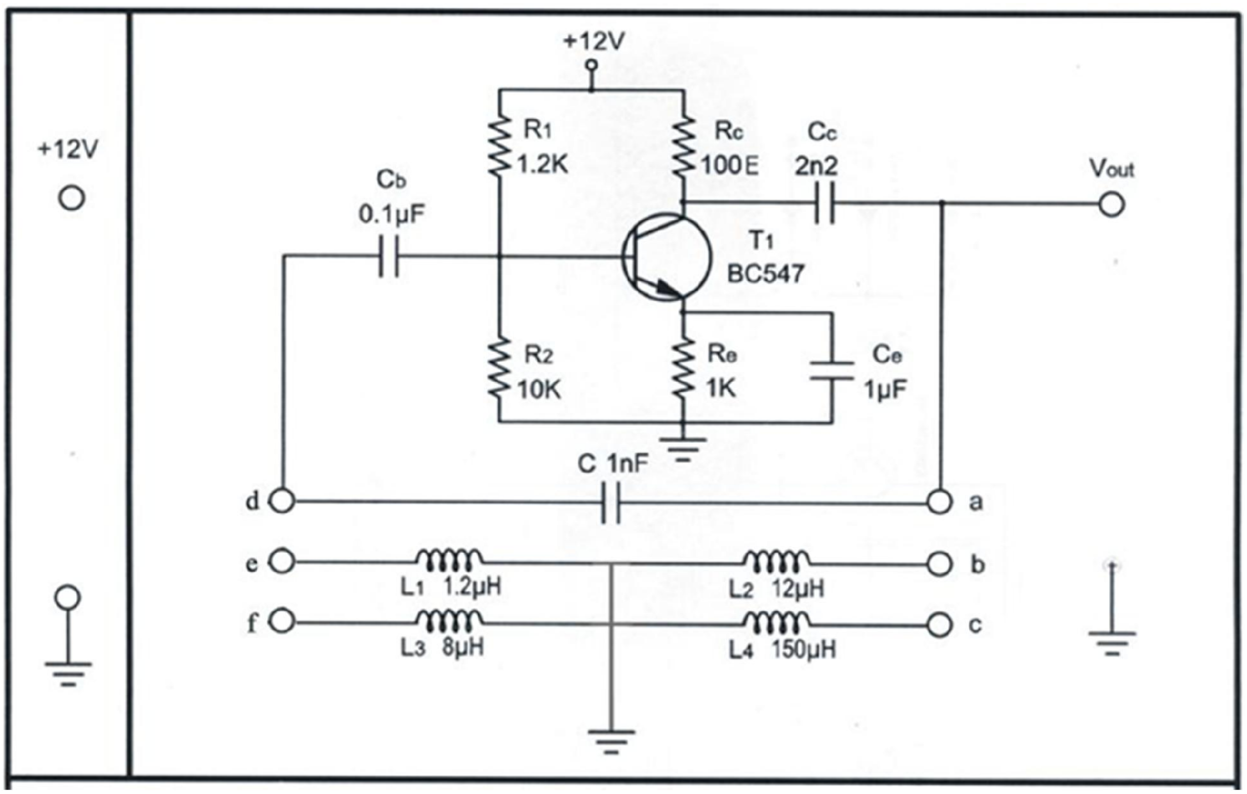
Result:

1. When patch cord connected across a and b Practically calculated Output frequency (on CRO): ....
2. Resonant frequency (fr) : ………………. (use equation l)
3. Output voltage amplitude: ……………... Vp-p
4. When patch cord connected across a and c practically calculated Output frequency (on CRO): ……….........................
5. Resonant frequency (fr) : ………………. (use equation l)
6. Output voltage amplitude: ……………... Vp-p

Conclusion:

Hence we studied the operaion of operation of Hartley Oscillator.

Circuit diagram:



Calculation:

Experiment No.:8

Aim: To design wein bridge oscillator and measure its frequency of oscillations using multisim.

Software: Multisim

Theory:

The wein bridge circuit:

1. The wein bridge circuit is used as a feedback network in wein bridge oscillator as shown in figure.

2. The amplifier shown can be used as FET or transistor or op-amp as a amplifying device.

3. Wein bridge of fig. has four arms the arm AD consist of series combination of R1& C1 arm AB consist of parallel combination of R2 & C2 are called as frequency selective arms. This is because components connected in this arms decides oscillator frequency.

4. Resistors R3 & R4 are used to generate a reference voltage which remains constant independent of frequency.

5. The AC input voltage is applied between point A & B of bridge. When wein bridge is used in oscillator circuit the feedback voltage is applied between this two points.

6. The AC output is obtained between C & D. when used in oscillator circuit these points are connected to input of amplifier.

7. The feedback network used in wein bridge oscillator is also called as load log network. At low frequency it acts as load & at high frequency it acts as log network. the phase shift introduce at output is 0^o.

Steps to design wein bridge oscillator:

Assume s and F_{min} if not given.

1. To determine the values of bridge components:
   Let R1 =R2=R & R ≥ hie
   C1=C2=C
   C= 1/(2πFoR)
   R3=2R4
2. To determine operating points
   $V_{CEQ} \geq V_{O(P)} + V_{CE\,(sat)} + 1$
   Let I_{CQ} = 1mA
   $Q(V_{CE}, I_{CQ})$ is operating points.

3. To find V_{CC}:

$$V_{CC} \geq V_{CEQ} + V_{O(P)} + 1.5V$$

4. To find R7,R5 & R_E :

$$V_E = 10\%\ V_{CC}$$

$$R_E = V_E / I_{CQ}$$

$$RE1 = RE2 = R_E$$

$$R_B = (S-1) R_E$$

$$R5 = R_B . V_{CC} / V_B$$

$$V_B = V_{BE} + V_{BE}$$

Let R5=R7

$$R6 = R8 = R5 . R_B / (R5 - R_B)$$

5. To find R_C:

$$R_C = (V_{CC} - V_{CEQ} - I_E . R_E) / I_{CQ}$$

Let $R_C1 = R_C2 = R_C$

6. To find capacitors:

i. $C_E1 = C_E2 = 1/(2\pi F_{low}\, 0.1 R_E)$

ii. $C_C = 1/(2\pi F_{low}(R_C1 + R7 \parallel R8 \parallel hie))$

iii. $Cin = 1/(2\pi F_{low}(R5 \parallel R6 \parallel hie))$

iv. $Cout = 1/(2\pi F_{low} R_C2)$

Procedure:

1. Design a circuit as per given specifications.
2. Make the connections as per circuit diagram.
3. Observe output & note down readings for specified frequency range.

Conclusion:

Thus we have designed wein bridge oscillator & observed its frequency of oscillations.

Circuit diagram:

Calculation:

Experiment No. 9

Aim: To study the operation of Class A amplifier

Equipment needed:

1. Analog board of AB-21
2. Variable DC power supply +12V from external source or ST-2612 Analog Lab.
3. Function Generator
4. Oscilloscope.

Theory:

The power amplifiers are the amplifiers which deliver maximum undistorted symmetrical output voltage swing to the low impedance load. Generally, any system (like a stereo, radio or television) consists of several stages of amplification. When the signal passes through these stages, the power level of signal rises so much that the later stages require high power handling circuit elements such as power transistors. Also as the load impedances of these later stages is very small (of the order of 8ohm of stereo amplifier speakers), heavy collector current flows. To handle this, transistors having power rating of 1W or more are used in power amplifiers.

Power amplifiers are classified as:

1. Class A (Voltage Amplifier)

2. Class B(Push-Pull Emitter Follower)

3. Class C Tuned Amplifier

Class A Amplifier:

Class A Amplifier is basically a voltage amplifier in which transistor operates in active region for the entire cycle of input AC signal. In other words the collector current flows for 360^0 of AC signal. For class A amplifiers the Q point is located somewhere near the middle of the AC load line and thus offers maximum amplification of the input signal as shown below. The collector current $I_c$is non-zero even when the input signal is zero i.e. the I_c flows for 100% of the time. This leads to power dissipation even in quiescent condition.

Procedure:

- Connect +12V variable DC power supplies at their indicated position from external source or ST-2612 Analog lab.
- Connect 2V $_{p-p}$ AC signal (1KHz) at the V_{in} input of the AB-21

1. Connect oscilloscope at output terminals of AB-21 and observe the output waveform.
2. Gradually increase the input signal up to the value before the signal just get clipped or decrease the DC supply voltage to the value before the signal just get clipped. This is the maximum amplification of Class A Amplifier AB-21.
3. Calculate the peak-to-peak value of the output signal.

Results:

Input AC signal amplitude (V_{in}): ___________________V_{p-p}

Output AC signal amplitude (V_{out}):___________________V_{p-p}

Voltage gain($A = V_{out} / V_{in}$): ___________________

Conclusion:

Hence we studied the operation of class A amplifier

Circuit Diagram:

Observation Table:

Frequency(HZ)	Output Voltage	Gain=$20\log_{10}$ Vo/Vi

Experiment No. 10

Aim: Study of the operation of Class B Amplifier.

Equipment Needed:

1. Analog board of AB22
2. Variable DC power supplies +5V and -5V from external source or ST2612 Analog Lab
3. Function Generator
4. Oscilloscope Caddo 802 or equivalent

Theory:

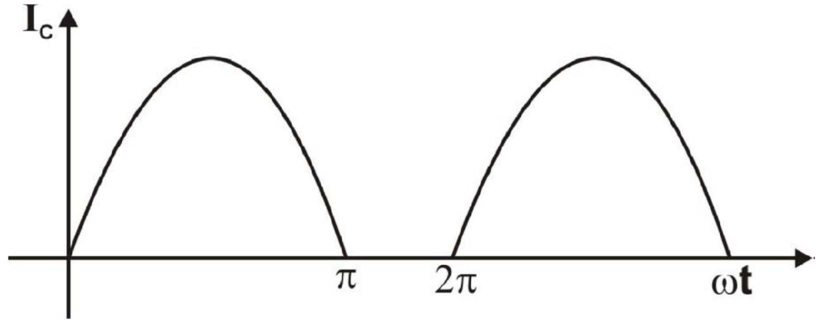
The power amplifiers are the amplifiers which deliver maximum undistorted symmetrical output voltage swing to the low impedance load. Generally any system (like a stereo, radio or television) consists of several stages of amplification. When the signal passes through these stages, the power level of signal rises so much that the later stages require high power handling circuit elements such as power transistors. Also as the load impedance of these later stages is very small (of the order of 8 ohm for stereo amplifier speakers), heavy collector current flows. To handle this, transistors Raving power rating of 1W or more are used in power amplifiers.

Power amplifiers are broadly classified as :

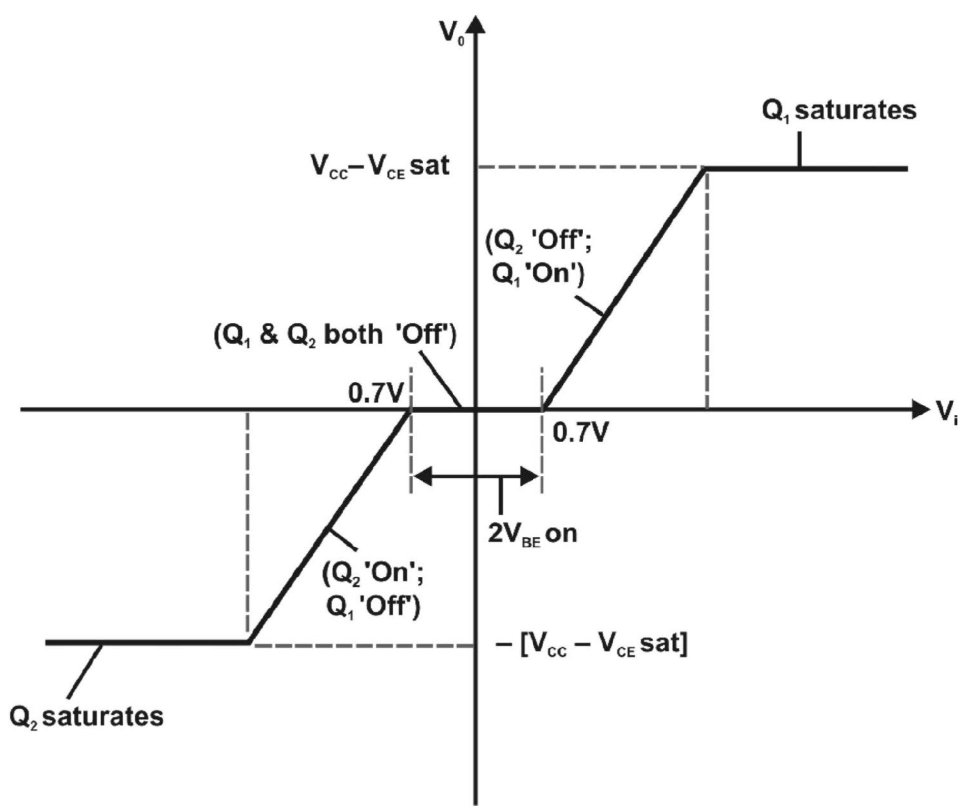
1. Class A (Voltage Amplifier)
2. Class B (Push-Pull Emitter Follower)
3. Class C

Class B Amplifier:

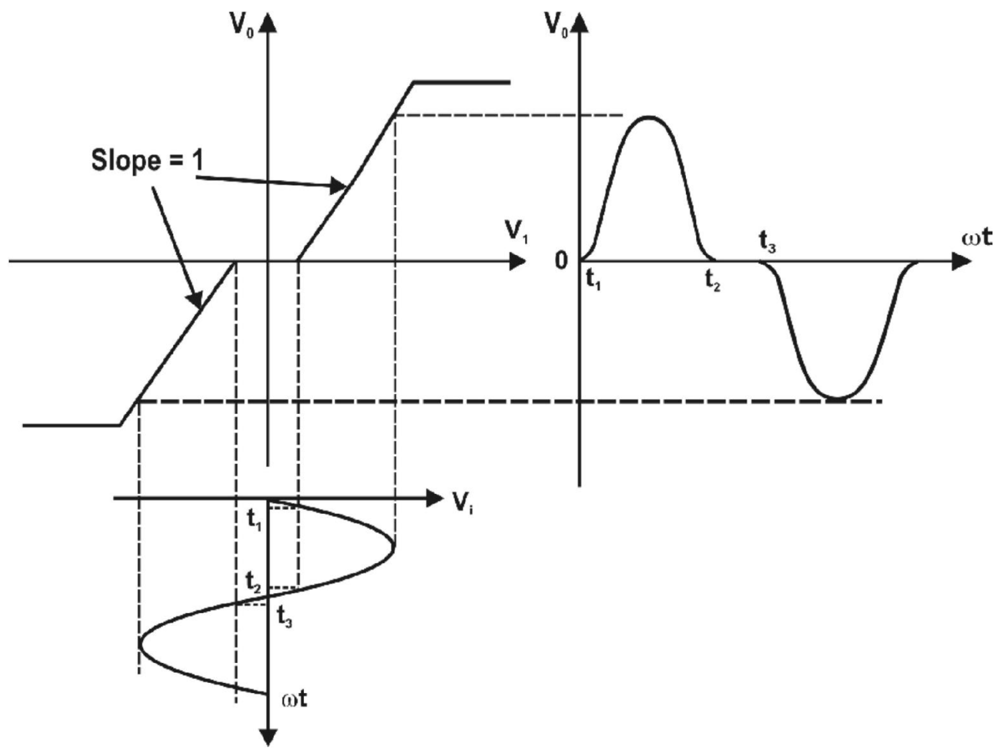
Class B amplifier is a circuit in which transistor conducts (collector current flows) for only 180– of input AC signal. When a signal is applied, one half cycle will forward bias the base-emitter junction and IC will flow. The other half cycle will reverse bias the base-emitter junction and IC will be cut off.



For class B amplifiers the Q point is located near the cutoff point of the AC load line. Thus, to amplify entire input AC signal a combination of two Class - B amplifiers are used. One of which amplifies positive half cycle of input AC signal and the other amplifies negative half cycle of input AC signal. This amplifier configuration is known as push-pull or complementary symmetry. In the push-pull configuration it is important to match the two transistors carefully for the proper amplification of boththe halves. While the input signal being amplified through class B amplifier the input signal has to rise to about 0.7V to overcome the barrier potential of amplifying transistor. During this period no current flows through the circuit and output is zero. The action is similar for both the transistors. Thus, following characteristic is obtained for input and output voltages:



The output signal no longer remains sine wave and gets distorted. Since the clipping occurs between the time when one transistor cuts off and at the time when other one comes on. We call it crossover distortion. Input and output waveforms illustrating the zone/crossover distortion to remove the crossover distortion a slight forward bias is applied to each emitter diode i.e. we locate the Q point of both the transistors slightly above the cutoff. Thus collector current in both the transistors flows for more than 180° but less than 360°. Sometimes we call such an amplifier as Class AB amplifier.

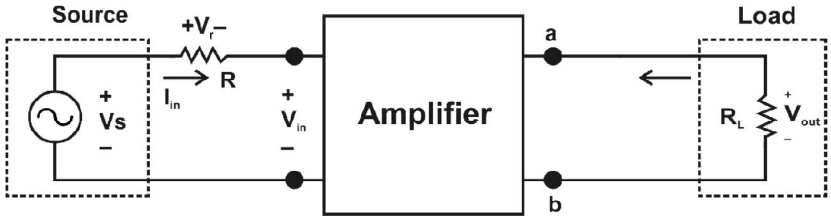


Operating parameters of Class B Amplifier:

Voltage Gain:

It is the ratio of output voltage (Vout) obtained to input voltage (Vin).

Av = Vout / Vin.



It is the ratio of Input voltage (Vin) to Input Current (Ii).

Zin = Vin / Ii

To measure the input impedance a known resistor (Rs) is placed in series before the input coupling capacitor and the impedance could be calculated using the equation.

Zin = RS / [AV / (AV'-I)]

Where,

AV = voltage gain without the resistor (RS)

AV' = voltage gain with the resistor (RS)

Output Impedance:

It is the ratio of Output Voltage (Vout) to Output Current (IO).

Zout = Vout / Io

To measure the output impedance a known resistor (RS) is placed from output to ground and the output impedance could be calculated using the equation.

Zout = [AV / (AV'-I)] * RS

Where,

AV = voltage gain without the resistor (RS)

AV' = voltage gain with the resistor (RS)

Current gain:

It is the ratio of output current (Io) to Input current (Ii).

Ai = Io / Ii

The Current gain could be calculated using the equation

Ai = -AV * Zin / RL

Power Gain:

It is the ratio of output AC power (Po) to input AC power (Pi).

2 / RL

2 / (8 * RL) = Vrms

PO = Vo(P-P)

Pi = Vin (p-p)2 / (8 * Zin) = Vin(rms)2 / Zin

The Power Gain can be calculated using the equation

Ap = Po / Pi

Procedure:

• Connect +5V and -5V DC power supplies at their indicated position from external source or ST2612 Analog Lab.

• Connect 2Vp-p AC signal (1 KHz) at the Vin input of the AB22.

1. Put the potentiometer P1 to its minimum position i.e. rotate it fully anticlockwise. (This is the condition when no bias voltage is applied to the emitter diodes of both the transistors.)
2. Connect Oscilloscope at the output terminals of AB22 and observe the output waveform. The crossover distortion can be clearly observed on the oscilloscope.
3. Gradually increase the bias voltage by increasing bias resistance (i.e. rotate the potentiometer in clockwise direction) up to the value when the crossover distortion is completely removed and maximum amplification of the input signal is obtained.
4. Connect the input AC signal to the Class B amplifier through 100 ohms series resistance.
5. Measure the input and output voltage amplitude and also input and output impedance as described earlier.
6. Measure the power gain of class B amplifier using the operating parameters as described in class B theory.

Results:

Input AC Voltage Amplitude (Vin) : ....Vp-p

Input Impedance (Zin) : ....Ohms

Input Current (Ii) : ....A

Output AC Voltage Amplitude (Vout) : .....Vp-p

Output Impedance (RL) : ....ohms

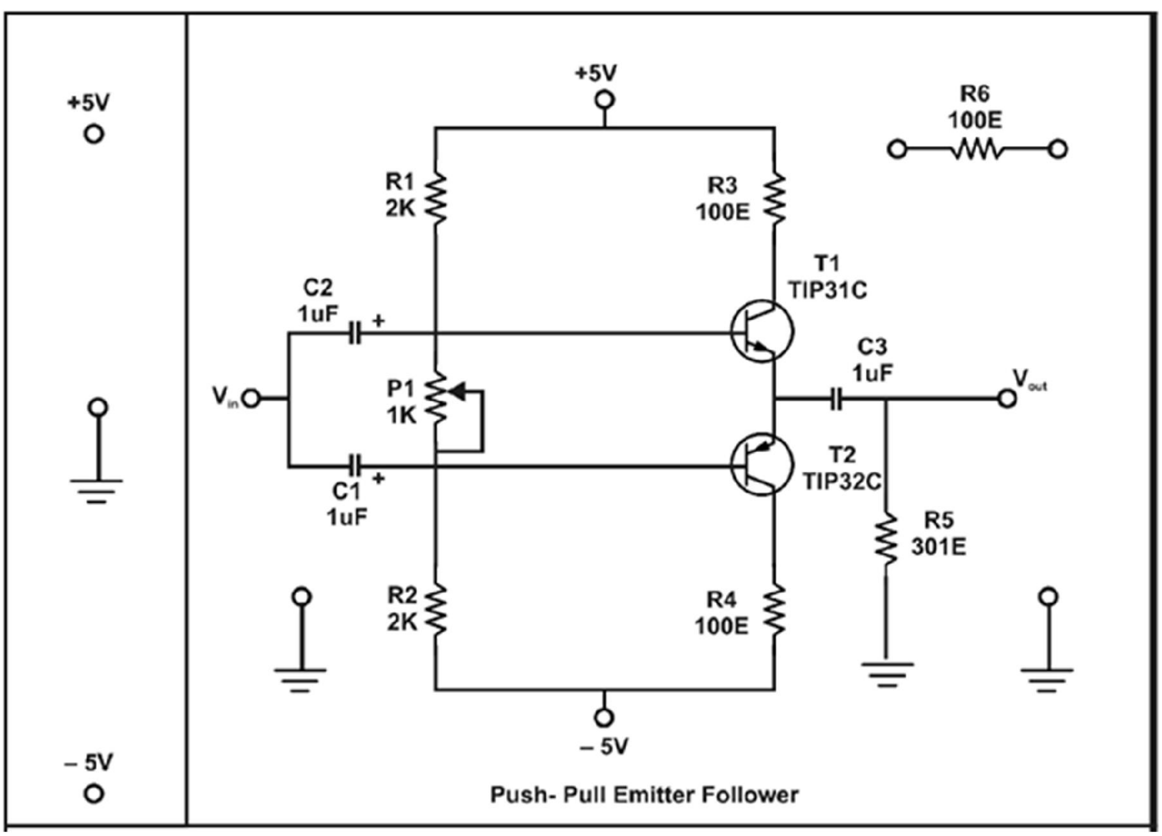
Output Current (Io) : ....A

Power Gain (Ap) : ....

Conclusion :

Hence we studied the operation of class B amplifie

Circuit diagram :



Calculation:

Experiment No: 11

Aim: Study of Transistor Shunt Voltage Regulator, when Input Voltage Vin is fixed while

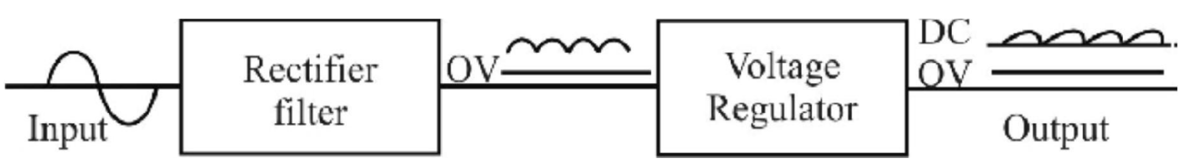
. Load resistance RL is variable

Equipment Needed:

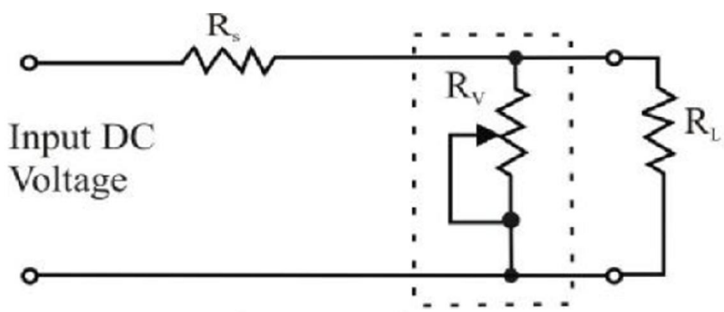
1. Analog board of AB32.
2. DC power supply +12V external source or ST2612 Analog Lab.
3. Digital Multimeters (2 numbers).
4. 2 mm patch cords.

Theory:-

Circuits that maintain power supply voltages or current output within specified limits, or tolerances are called Regulators. They are designated as DC voltage or DC current regulators, depending on their specific application. Voltage regulator circuits are additions to basic power supply circuits, which are made up of rectifier and filter sections (figure 1). The purpose of the voltage regulator is to provide an output voltage with little or no variation. Regulator circuits sense changes in output voltages and compensate for the changes.



There are two types of voltage regulators. Basic voltage regulators are classified as either Series or Shunt, depending on the location or position of the regulating element(s) in relation to the circuit load resistance. Figure 2 illustrates these two basic types of voltage regulators. Broken lines have been used in the figure to highlight the difference between the series and shunt regulators.



Shunt Regulator

The schematic drawing in view A is that of a shunt regulator. It is called shunt regulator because the regulating device is connected in shunt or in parallel with the load resistance. Figure 2 illustrates the principle of shunt voltage regulation. From the Figure it is clear that the regulator is in shunt with the load resistance (RL). In a shunt voltage regulator, as shown in Figure 2, Output voltage regulation is determined by parallel resistance of the regulating device, the Load resistance (RL), and the series resistor Rs.

If the load resistance RL increases/decreases, the regulating device decreases/increases its resistance to compensate for the change.

The schematic for a typical series voltage regulator is shown in Figure 3. It employs the NPN transistor in shunt configuration in place of the variable resistor found in Figure 2. Since AB is in parallel across VL, we have VL -VZ -VBE = 0 or VBE = VL -VZ (kirchoff's Voltage Law)

Also VL = VZ + VBE

i.e. The output voltage is close to the sum of the voltage across Zener and the voltage at the base-emitter junction of transistor

Procedure:

1. Connect + 12V DC power supply at their indicated position from external source or ST2612 Analog Lab.
2. Connect one voltmeter between test point 1 and ground to measure input voltage Vin.
3. Connect ohmmeter between test point 4 and ground and set the value of load resistance RL at some fixed value [full load (1.1K), 1K, 500....]
4. Connect a 2mm patch cord between test point 2 and 3.
5. Connect voltmeter between test point 4 and ground to measure output voltage Vout .
6. Switch on the power supply.

7. Vary the potentiometer P1 to set fixed value of input voltage Vin = 9V and measure the corresponding values of

a) Output voltage Vout between test points 4 and ground.

b) Zener voltage VZ between test points 5 and ground.

c) Forward bias voltage VBE of transistor between test point 5 and 4.

8. Disconnect the 2mm patch cord between test point 2 and 3.

9. Repeat the procedure from step 3 for different sets of load resistance RL and note the results in an observation Table 1.

Results:

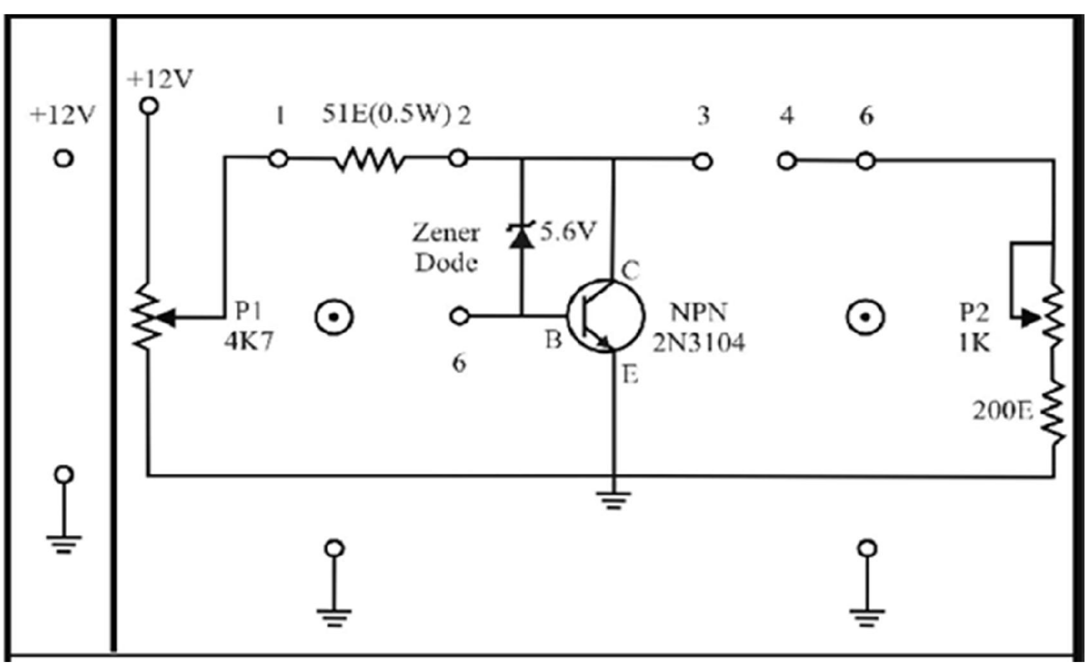
1. The result of Experiment 1 reveal that for the network of figure 4 with a fixed input voltage the output voltage will remain close to 4.9 V for a range of load resistance that extends . from______ to _______

2. Percentage regulation = _______%

Conclusion:

Hence we studied the Operation of Study of Transistor Shunt Voltage Regulator, when Input Voltage Vin is fixed while Load resistance RL is variable.

Circuit Diagram:



Observation Table:

Load regulation:

Sr.No	Load Resistance RL	Voltage Across Zener VZ	Forward bias voltage	Output voltage at constant Input voltage Vin=9 volt

Line regulation:

Sr.No	Load Resistance RL	Voltage Across Zener VZ	Forward bias voltage	Output voltage at constant Input voltage Vin=9 volt

Calculations:

Percentage regulation is given by formula

% Regulation = [(VNL - VFL) / VFL] * 100

VNL = no-load or open-circuit terminal voltage.

VFL = full-load terminal voltage

EXPERIEMENT NO: 12

Aim: To study voltage series feedback and its frequency response with and without feedback.

Apparatus: Breadboard, CRO, Function Generator, DC Power Supply, Multimeter, Connecting wires etc.

Components: Transistor BC 547B,

Resistors: 22K, 6.8K, 2.2K, 560Ω, 1K, 10K.

Capacitors: 3.3μF, 100μF, 4.7μF.

Theory:

Feedback is defined as process in which part of output signal (current or voltage) is returned back to input. There are two types of feedback series- positive feedback and negative feedback.

We know that input signal $\frac{V}{S}$, feedback signal V_f can be mixed in two different ways i.e., series mixing and shunt mixing. We can combine the sampling and mixing technique to yield following negative feedback configuration,

1. Voltage series feedback (series shunt feedback)
2. Voltage shunt feedback (shunt shunt feedback)
3. Current shunt feedback (shunt series feedback)
4. Current series feedback (series series feedback)

Figure shows block diagram of voltage series feedback amplifier. Here we use combination of voltage sampling or current sampling (shunt sampling and series mixing)

$$A = \frac{V_0}{V_i}$$

$$\beta = \frac{V_f}{V_0}$$

$$A_{Vf} = \frac{V_0}{V_i} = \frac{A V_i}{V_S} = \frac{A(V_S - V_f)}{V_S}$$

$$A_{V_f} = \frac{A(V_S - \beta V_0)}{V_S}$$

$$\frac{A_{V_f}}{A} = 1 - \frac{\beta V_0}{V_S}$$

$$\beta \frac{V_0}{V_s} = 1 - \frac{A_{Vf}}{A}$$

$$\frac{V_0}{V_s} = \frac{1}{\beta}\left[1 - \frac{A_{Vf}}{A}\right]$$

$$= \frac{A - A_{Vf}}{A\beta}$$

$$A_{Vf} = \mathrm{A} - \frac{A_{Vf}}{A\beta}$$

$$[1 + A\beta]A_{Vf} = A$$

$$A_{Vf} = \frac{A}{1+A\beta} \ldots\ldots\ldots\ldots\ldots\ldots \text{\{Gain with feedback\}}$$

Input impedance with feedback =

$$Z_i = \frac{V_i}{I_i}$$

$$I_f = \frac{V_i}{Z_i} = \frac{V_s - V_f}{Z_i}$$

$$I_f = \frac{V_s - \beta V_0}{Z_i}$$

$$I_f Z_i = V_s - \beta V_0$$

As there is series mixing input resistance of amplifier will increase. Due to shunt sampling output resistance of circuit will decrease.

One of the most advantage of feedback is that it increases the bandwidth of amplifier. For any amplifier product of voltage gain & Bandwidth always remains constant.

Gain X B.W = Constant

With negative feedback gain reduces therefore to keep the value of product constant bandwidth of amplifier increases proportionally. It is possible to obtain desired bandwidth by selecting power feedback factorβ. The increase in bandwidth due to negative feedback is shown in fig.

Advantages of Negative feedback:

1. Stabilize the gain & increase B.W.

2. Distortion in output are reduced.

3. Can increase input resistance & decrease output resistance for certain feedback Configuration.

Applications of Negative feedback:

1. Regulated power supply.

2. Wide band amplifier.

3. Special circuit like bool strapping.

Analysis of voltage series feedback:

Consider voltage gain of amplifier without feedback is ‘A’ and feedback factor is ‘β’.

$$I_f Z_f = V_s - \beta A V_i$$

$$Z_f = \frac{V_s}{I_f} - \beta A \frac{V_i}{I_f}$$

$$\beta A \frac{V_i}{I_f} = \frac{V_s}{I_f} - Z_i$$

$$\beta A Z_i = \frac{V_s}{V_i} - Z_i$$

$$[1 + \beta A] Z_i = Z_i f$$

$$Z_i f = Z_i [1 + \beta A]$$

Input impedance is increases due to negative feedback.

Output Impedance:

Applying KVL to output port,

$$V_0 - Z_0 I_0 = A V_i$$

$$V_0 = A V_i + Z_0 I_0$$

$$V_0 = A(V_S - V_f) + Z_0 I_0$$

$$\frac{V_0}{I_0} = \frac{A(V_S - V_f)}{I_0} + Z_0$$

$$= \frac{-A V_f}{I_0} + Z_0$$

$$= {}^{-A\beta} \frac{V_0}{I_0} + Z_0$$

$$[1 + \beta A] Z_0 f = Z_0$$

$$Z_0 f = \frac{Z_0}{1 + A\beta}$$

i.e. output impedance decreases due to voltage series feedback.

Procedure:

1. Make connections as per circuit diagram.
2. Set 20mV peak to peak at 1 KHz frequency and give these sine waves as input to amplifier.
3. Then observe the output waveform on CRO without feedback circuit.
4. Calculate the voltage gain without feedback.
5. Then connect the feedback circuit to network of input of main current & observe the output.
6. Calculate voltage gain with feedback.
7. Then observe the frequency response with feedback.

Conclusion:

Thus we have studied voltage series feedback amplifier as result of feedback, network bandwidth will increase and gain will reduce.

Circuit diagram:

OBSERVATION TABLE:

Without feedback

Vin:……

Frequency	**Vout**	**Gain=20*log(Vout/Vin)**

With feedback

Vin:…….

Frequency	**Vout**	**Gain=20*log(Vout/Vin)**

EXPERIMENT NO: 13

Aim: To study current series feedback amplifier and to plot its frequency response with and without feedback.

SOFTWARE: MULTISIM11.0

THEORY:

NEGATIVE FEEDBACK-When feedback signal result in decrease in magnitude of output signal it is called as negative feedback.

CURRENT SERIES FEEDBACK-The block diagram of current series feedback amplifier is shown as fig.a here we use the combination of current sampling and series mixing :

Current series feedback=current sampling +series sampling

Current series feedback is present in transconductance amplifier .

Analyzer of current series feedback amplifier.The CE transistor amplifier with the unbypassed RE is shown in fig.

For this ckt sampled signal on the output side is Io and feedback signal Vf is vtg across RE .

The h-parameter equiqvalent is shown as fig.

1.to calculate gain with feedback (Af) –we have gain feedback as:

$A=-I0/Vi$

$A=-hfe.Ib/hie.Ib$

$A=-hfe/hie$

The feedback factor B is given as

$\beta=Vf/io$

$\beta=-i0.RE/Io$

$\beta=-RE$

We know that $AF=A/1+A\beta$

$Af=-hfe.hie/hie.hie+hfe.re$

$=-hfe/hie+hfe.re$

2.to calculate voltage gain with feedback

$AVf=Vo/Vs$

$AVF=-hfe.Ib.RC/hie.ib+(1+hfe)Ib.re$

$=-hfe.rc/hie+(1+hfe)re$

3.INPUT IMPEDANCE(Zif)-

Zi=hie

Zif=zi(1+Aβ)

Zif=hie(1+hfe/hie.re)

Zif=hie+hfe.re

4. OUTPUT IMPEDANCE-

Zo=Rc

Zof=zo(1+Aβ)

Zof=Rc(hie+hfe.re)/hie

PROCEDURE:

1.Make the connections as per ckt dig

2.give the specified input to the ckt using function generator

3.observe the output on cro for without feedback and with feedback network

4.plot the frequency response for ckt with and without fceedback

Conclusion:

Thus we have studied the current series feedback ampligfier and plotted its frequency response and without feedback

CIRCUIT DIAGRAM:

OBSERVATION TABLE:

Without feedback

Vin:……

Frequency	**Vout**	**Gain=20*log(Vout/Vin)**

With feedback

Vin:…….

Frequency	**Vout**	**Gain=20*log(Vout/Vin)**