LENDI INSTITUTE OF ENGINEERING AND TECHNOLOGY

(Approved by AICTE, New Delhi and Affiliated to JNTU, Kakinada)

Department of Electronics & Communication Engineering

ELECTRONIC CIRCUIT ANALYSIS LAB

Lab Manual

Prepared by

LIST OF EXPERIMENTS

A) TESTING IN THE HARDWARE LABORATORY:

.

1. TWO STAGE RC COUPLED AMPLIFIER.

2. RC PHASE SHIFT OSCILLATOR USING TRANSISTORS.

3. CLASS A POWER AMPLIFIER.(SERIES FED)

4. CLASS B COMPLEMENTARY SYMMETRY AMPLIFIER.

5. CURRENT SHUNT FEEDBACK AMPLIFIER.

6. SINGLE TUNED VOLTAGE AMPLIFIER.

7.HARTELY OSCILLATOR

8.VOLTAGE SERIES FEED BACK AMPLIIER

9.CLASS –A TRANSORMER COUPLED POWER AMPIFIER

10.CLASS-B PUSHPULL POWER AMPLIFER

B) DESIGN AND SIMULATION USING PSPICE SOTWARE:

1. TWO STAGE RC COUPLED AMPLIFIER.

2. RC PHASE SHIFT OSCILLATOR USING TRANSISTORS.

3. CLASS B COMPLEMENTARY SYMMETRY AMPLIFIER

4.WEIN BRIDGE OSCILLATOR

5. CLASS A POWER AMPLIFIER.(SERIES FED)

6. SINGLE TUNED VOLTAGE AMPLIFIER.

7. CLASS –A TRANSORMER COUPLED POWER AMPIFIER

8.VOLTAGE SERIES FEED BACK AMPLIIER

9. CURRENT SHUNT FEEDBACK AMPLIFIER.

10.CLASS-B PUSHPULL POWER AMPLIFER

1. TWO STAGE RC COUPLED AMPLIFIER

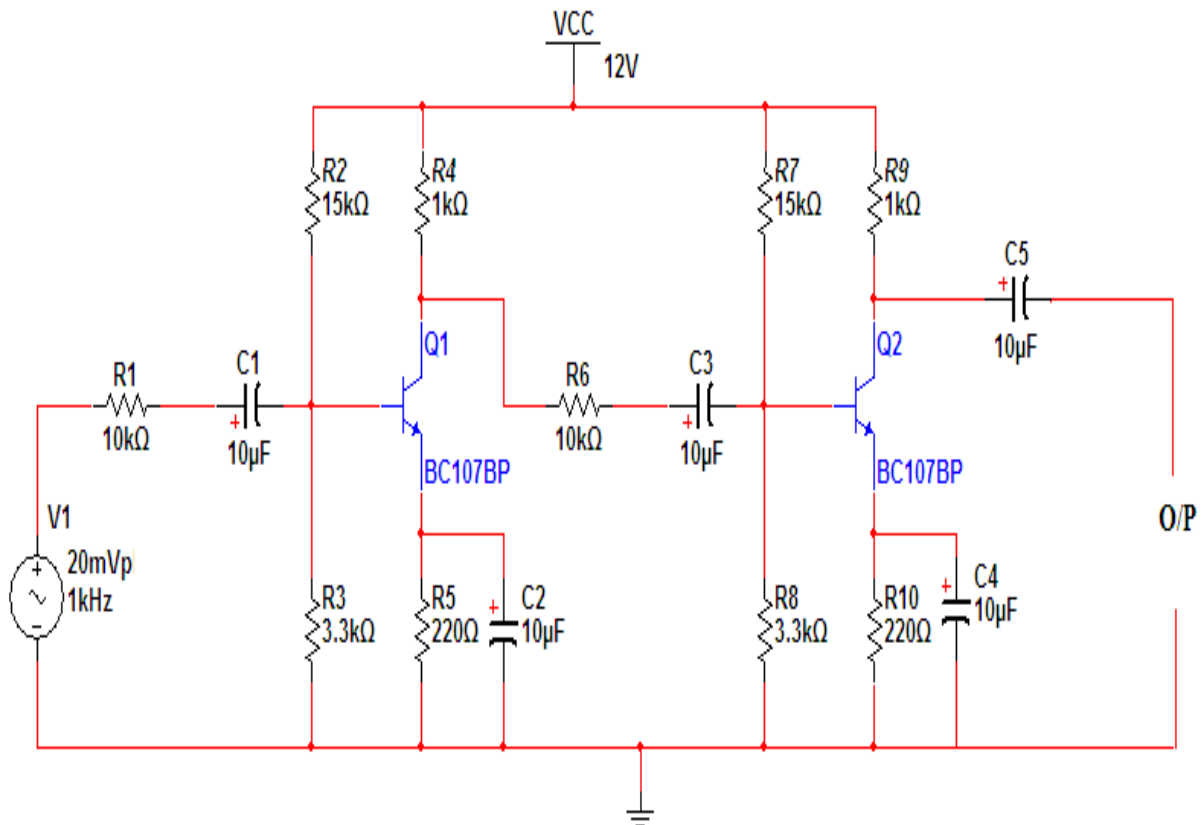**AIM:** To obtain the Voltage gain for two stage RC coupled Amplifier and also to observe the frequencyResponse.
.

EQUIPMENT REQUIRED:

      1. cathode ray oscilloscope   1No

      2.Regulated power supply   1No

      3. function generator   1No

      4.bread board   1No

      5.connecting wires

COMPONENTS REQUIRED:

    1. Resistors:

        15k-2no
        10k-2no
        1k-2no
        3.3k-2no
        220Ω-2no

    2.Transistor :BC107-2no
    3.Capacitors-10µf -5no

CIRCUIT DIAGRAM:

THEORY: Whenever large amplification with very good impedance matching is required using an active device such as a transistor or a field effect transistor a single active device and its associated circuitry will not be able to cater to the needs. In such a case single stage amplifier is not sufficient and one requires more stages of amplification i.e., output of one stage is connected to the input of second stage of amplification circuit and the chain continues until the required characteristics of amplifier is achieved such an amplifier is called as multistage amplifier.In multistage amplifier, the output signal preceding stage is to be coupled to the input circuit of succeeding stage. For this interstage coupling different types of coupling can be employed. They are

1. RC coupling

2. Transformer coupling

3. Direct coupling

RC coupling is most popularly used type of coupling because it is cheap and provides excellent fidelity over a wide range of frequency .it is usually employed for voltage.

PROCEDURE:

1) Connect the circuit as shown in the figure.

2) Apply 1Khz frequencyand 20mv Vp-p Sine wave from function generator..

3) Observe input and output Waveforms simultaneously on C.R.O

4) Change the frequency of input signal from 10HZ to 1MHZ in steps and note amplitudes of input and output Waveforms(input signal should be maintained constant).

5) Calculate Voltage gain (A) for each (in db) verses frequency.

Observations:

S.No	Frequency (Hz)	Input Voltage	Output Voltage	Gain=(V_0/V_i)	Gain in db= $20 \times \log_{10} (V_0/V_i)$

PRECAUTIONS:

1. Check connections before switching ON power supply.

2. Don't apply over voltage

3. When you are not using the equipment switch them Off
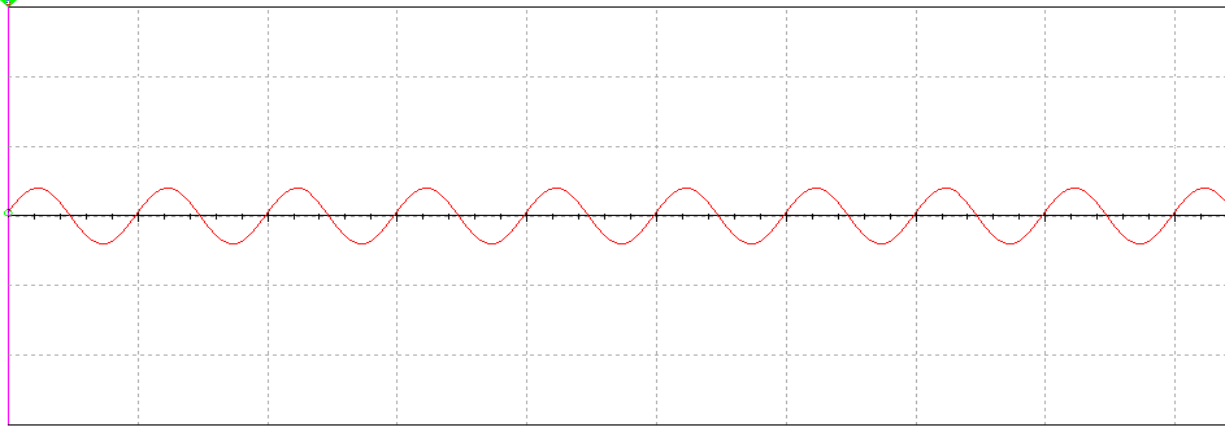
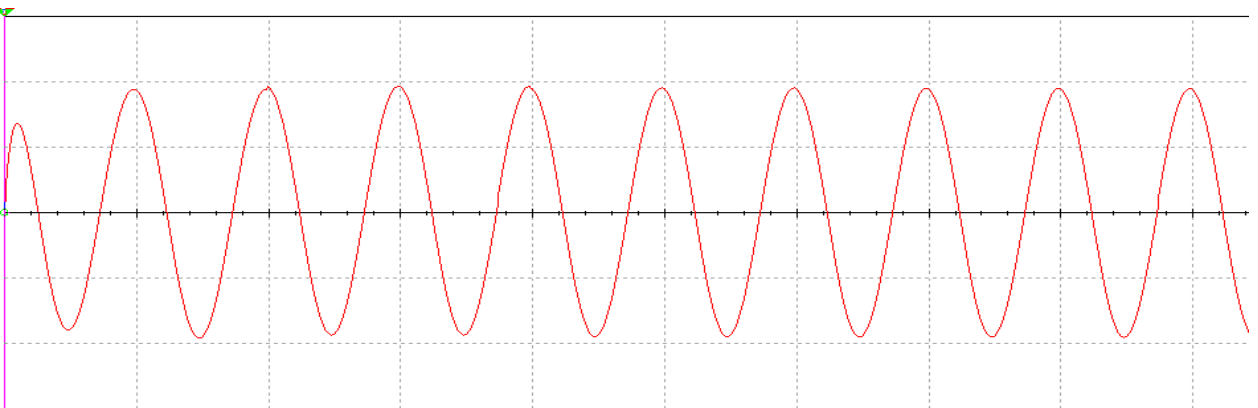## CALCULATIONS:

Maximum gain of the amp:

Upper cutoff frequency F2:

Lower cutoff frequency F1:

Band width=F2-F1:

EXPECTED GRAPH:
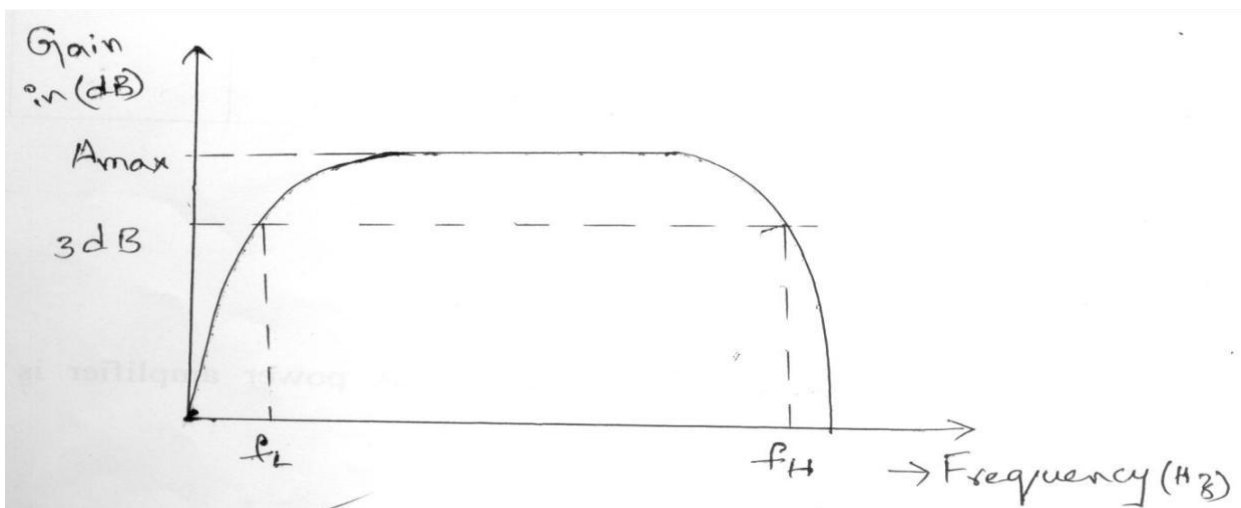
Input wave form

Output waveform

Frequeny response:

RESULT: -

1. Frequency response of Two stage RC coupled amplifier is plotted.
2. Gain = _____dB (maximum).
3. Bandwidth= fH--fL = _____Hz. At stage 2

VIVA QUESTIONS:
1. What is an amplifier?
2. what are the coupling elements of n this amplifier?
3. what are the applications of this amplifier?
4. why this type of amplifier will be used in AF mplifiers?
5. which type of biasing isused in this amplifier?
6. whatis the range of bandwidth in this amplifier?

Redraw the circuit diagram

2. RC PHASE SHIFT OSCILATOR USING TRANSISTORS

AIM: To calculate the frequency of the RC phase shift oscillator & to measure the phase angles at different RC sections.

APPARATUS:

     1. Transistor BC107

     2. Resistors: $10K\Omega$ -3Nos
                         $8K\Omega$ or $10K\Omega$
                         $22K\Omega$
                         $1.2K\Omega$
                         $100K\Omega$
     3. Capacitors: $0.001\mu f$ – 3 Nos
                        $10\mu F$ – 2Nos
                        $1\mu f$
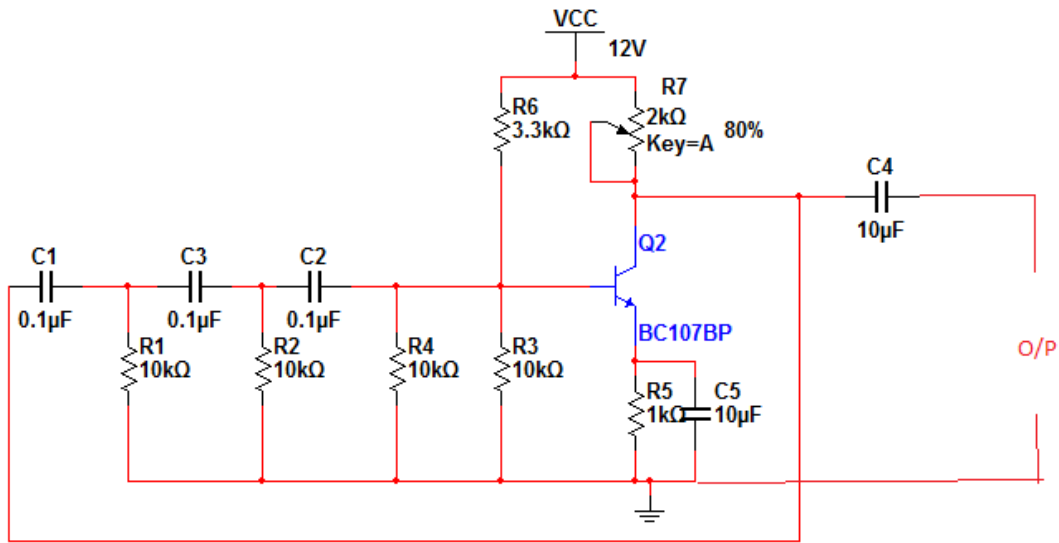   4. Regulated power Supply
     5. CRO

THEORY:

        RC-Phase shift Oscillator has a CE amplifier followed by three sections of RC phase shift feedback Networks the output of the last stage is return to the input of the amplifier. The values of R and C are chosen such that the phase shift of each RC section is 60º.Thus The RC ladder network produces a total phase shift of 180º between its input and output voltage for the given frequencies. Since CE Amplifier produces 180 º phases shift the total phase shift from the base of the transistor around the circuit and back to the base will be exactly 360º or 0º. This satisfies the Barkhausen condition for sustaining oscillations and total loop gain of this circuit is greater than or equal to 1, this condition used to generate the sinusoidal oscillations.

  The frequency of oscillations of RC-Phase Shift Oscillator is,

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

CIRCUIT DIAGRAM:

PROCEDURE:

1. Make the connection as per the circuit diagram as shown above.
2. Observe the output signal and note down the output amplitude and time period (T_d).
3. Calculate the frequency of oscillations theoretically and verify it practically ($f=1/T_d$).
4. Calculate the phase shift at each RC section by measuring the time shifts (T_p) between the final waveform and the waveform at that section by using the below formula.

OBSERVATIONS:

THEORITICAL CALCULATIONS: R = 10KΩ, C = 0.001 µf

$$f = \frac{1}{2\pi RC * \sqrt{6}} =$$
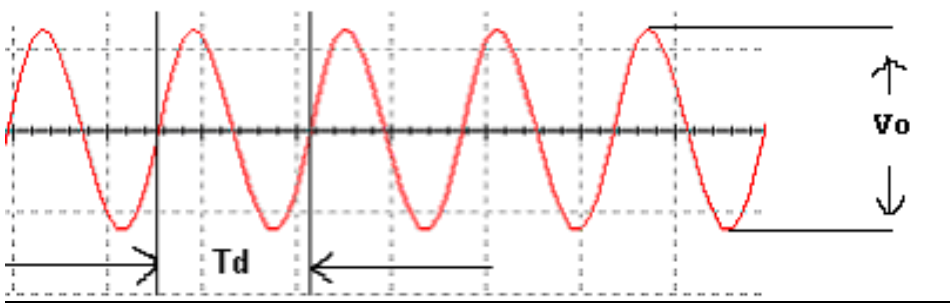
PRACTICAL CALCULATIONS:

$$T_d =$$

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

$$(1). \theta_1 = \frac{Tp_1}{T_d} * 360^0 =$$

$$(2). \; \theta_2 = \frac{Tp_2}{T_d} * 360^0 =$$
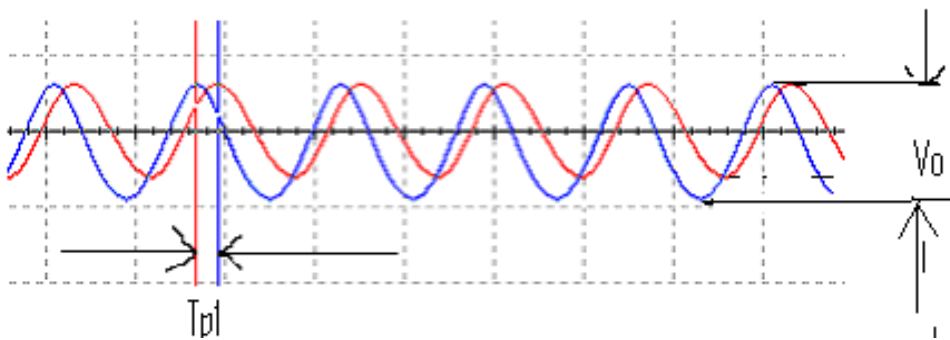
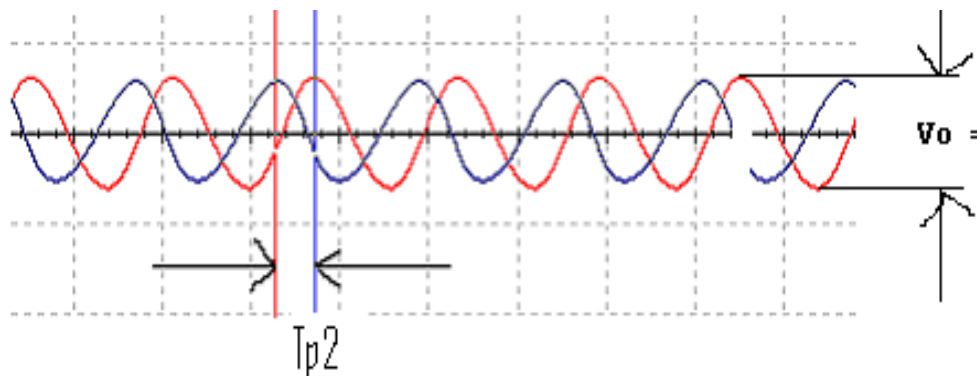$$(3). \; \theta_3 = \frac{Tp_3}{T_d} * 360^0 =$$
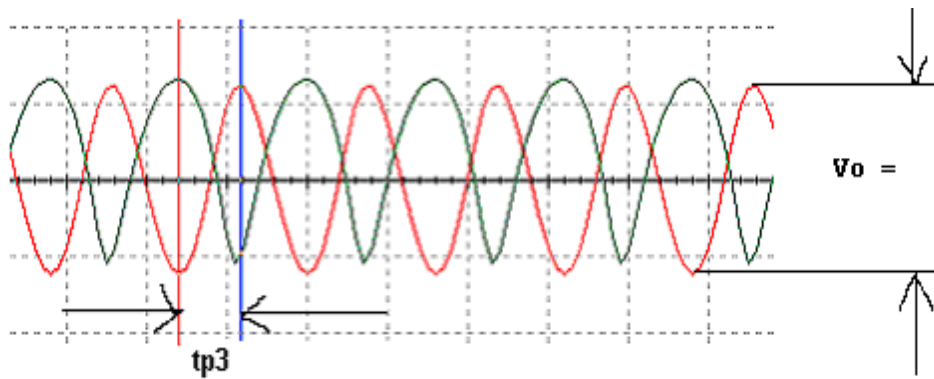
MODEL WAVE FORMS:

OUT PUT WAVE FORM :

OUT PUT WAVE FORM : $\theta = 60^0$

OUT PUT WAVE FORM : $\theta = 120^0$

OUT PUT WAVE FORM : $\theta = 180$

RESULT: The frequency of RC phase shift oscillator is calculated and the phase shift at different RC sections is noted.

$F_T =$ $F_P =$

VIVA QUESTIONS:

1. What are the conditions of oscillations?
2. Give the formula for frequency of oscillations?
3. What is the total phase shift produce by the RC ladder network?
4. Whether the oscillator is positive feedback or negative feedback?
5. What are the types of oscillators?
6. What is the gain of RC phase shift oscillator?
7. What is the difference between damped oscillations undamped oscillations?
8. What are the applications of RC oscillations?
9. How many resistors and capacitors are used in RC phase shift network
10. How the Barkhausen criterion is satisfied in RC phase shift oscillator
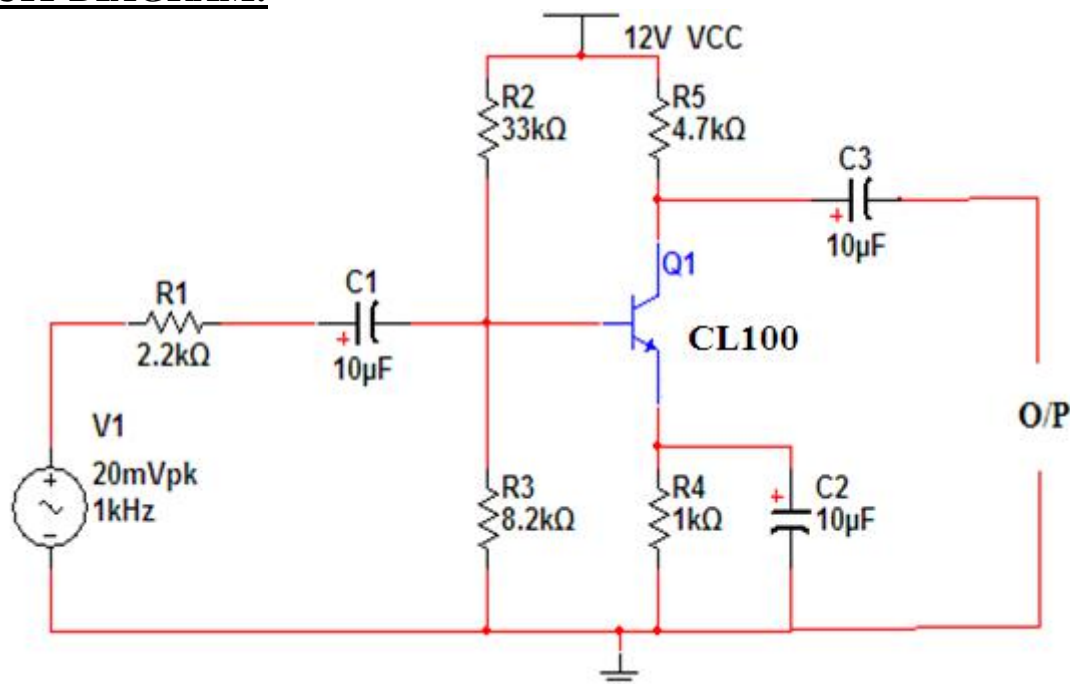
REDRAW THE CIRCUIT DIAGRAM:

3.CLASS A POWER AMPLIFIER(SERIES FED)

AIM: To design and test the class A power amplifier

APPARATUS:
1. Class A power amplifier trainer kit
2. Function Generator
3. CRO
4. BNC Probes and connecting wires

CIRCUIT DIAGRAM:

THEORY:

The amplifier is said to be class A power amplifier if the q point and the input signal are selected such that the output signal is obtained for a full input cycle. For this class the position of q point is approximately y at the midpoint of the load line. For all the values of input signal the transistor remains in the active region and never entire into the cutoff or saturation region. The collector current flows for 3600 (life cycle) of the input signal in other words the angle of the collector current flow is 3600 the class a amplifiers or furthers classified as directly coupled and transformer coupled and transformer coupled amplifiers in directly coupled type .The load is directly connected in the collector circuit while in the transformer coupled type, the load is coupled to the collector using the transformer.

Advantages:

1. Distortion analysis is very important

2. It amplifies audio frequency signals faithfully hence they are called as audio amplifiers

Disadvantages:

1. H parameter analysis is not applicable

2. Due to large power handling the transistor is used power transistor which is large in size and having large power rating.

PROCEDURE:

1. Switch ON Class –A power amplifier trainer

2. Set Vs (say 250 to 300 mV), at 10 KHz using signal generator.

3. Connect milli ammeter to the ammeter terminals

4. By keeping the input voltage constant, vary the frequency from 0 to 1MHz

   in regular steps .

5. Note down the corresponding output voltage from CRO

6. Calculate the DC input power using the formula $Pdc = V_{cc}I_c$

7. Calculate the AC output power using the formula $Pac = V_O^2/8R_L$

8. Calculate the efficiency $\eta = Pac /Pdc$

9. Plot the graph between Gain (db) and frequency.

10. Calculate bandwidth from the graph.

PRECAUTIONS:

1. Check connections before switching ON power supply

2. Don't apply over voltage

3. When you are not using the equipment switch them OFF.
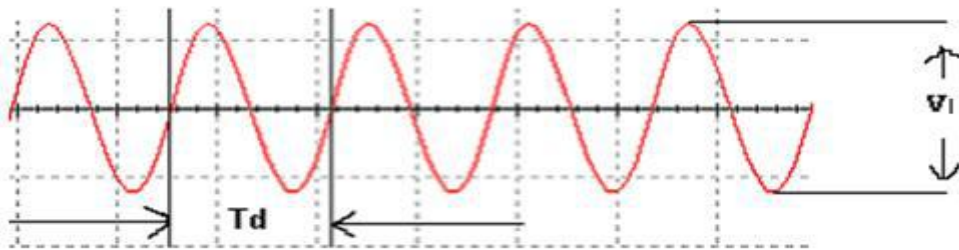
4. Handle all equipment carefully.

CALCULATIONS:

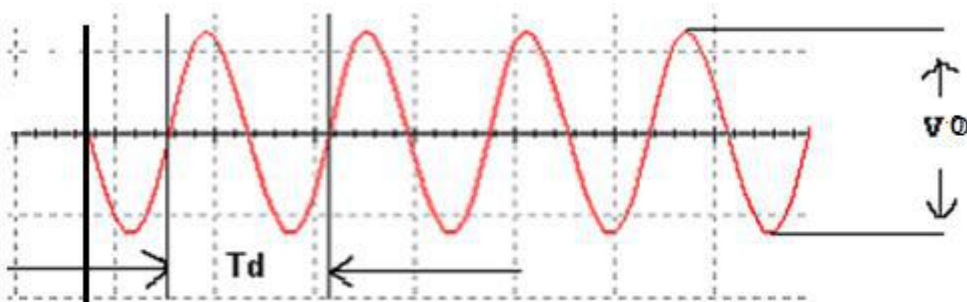Input power: $Pdc = V_{cc}I_c =$

Outpower: $Pac = VPP^2/8R_L =$

η = Efficiency=output power/input power*100 = $Pac /Pdc*100 =$

EXPECTED GRAPH:

I/P:

O/P:

RESULT: Gain and frequency as observed of Class A power amplifier.

η = Efficiency=output power/input power*100 = Pac /Pdc*100 =

VIVA QUESTIONS:

1.On which criteria based the power amplifiers are classified?

2.what is the operating point of class-A power ampliier?

3.what is meant by efficiency?

4.howmuch efficiency will b obtained in classs-A power amplifier?

5. what are the applications of power amplifier?

REDRAW THE CIRCUIT DIAGRAM:

4. CURRENT SHUNT FEEDBACK AMPLIFIER

AIM: To observe the performance of a current shunt feedback amplifier and obtain its bandwidth.

EQUIPMENT RQUIRED:
> Power supply 0-30V- 1 No.
> CRO 20MHz - 1 No.
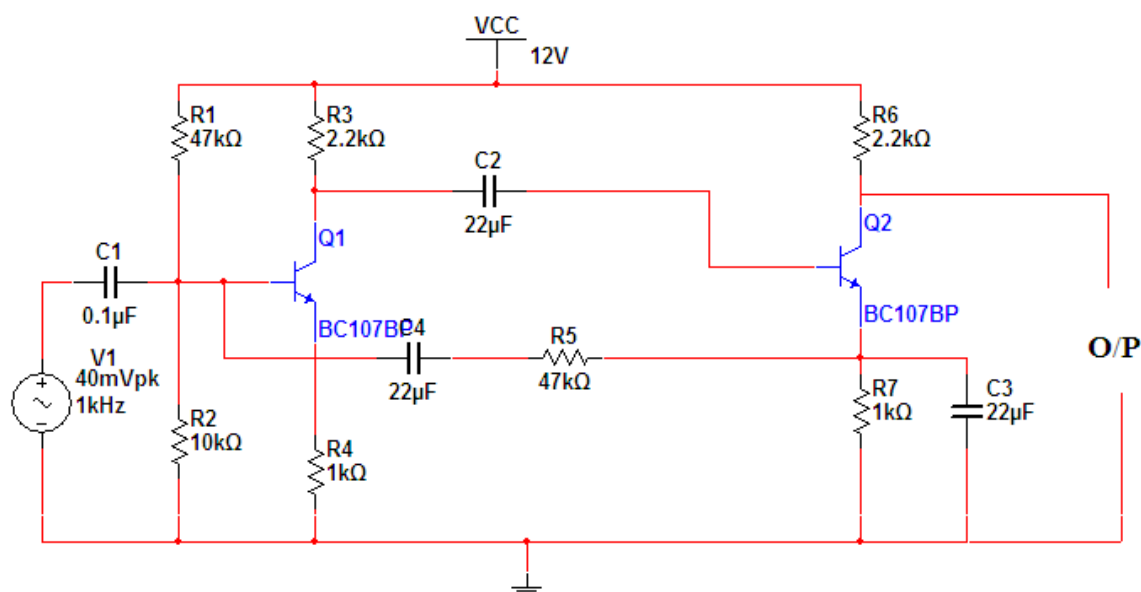> Digital multimeter - 1 No.
> Signal generator 1Hz - 1MHz - 1 No.

COMPONENTS:
> Resistors : 47KΩ - 2 Nos.
>   2.2KΩ - 2 Nos.
>   1KΩ   - 2 No
>   10KΩ - 1No.
> Capacitors 22µF   - 3Nos.
>   0.1µF - 1No.
> Transistors BC 107-2No.

THEORY:

Current shunt feedback circuit shows two transistor in cascade with feedback from the second emitter to the first base through the resistor R_F. we verify that this connection produces negative feedback. The voltage V_{i2} is much larger than V_{i1} because of the voltage gain of Q_1. Also V_{i2} is 180^0 out of phase with V_{i1}. Because of emitter follower action V_{e2} is only slightly smaller than V_{i2}, and these voltages are in phase. Hence V_{e2} is larger in magnitude than V_{i1} and is 180^0 out of phase with V_{i1}. If the input signal increases so that I_S increases, If also increases, and $I_i = I_S - I_f$ is smaller than it would be their were no feedback. This action is characteristics of negative feedback.

CIRCUIT DIAGRAM:

PROCEDURE:

1. Connect the circuit as shown in the figure
2. The operating points V_{CEQ}, I_{EQ} and V_{BE} are measured.
3. Connect the signal generator with a sine wave of 1KHz frequency to the input and increase the input to such a level that the output waveform of the signal as observed on CRO is not distorted.
4. Measure the input and output voltages and calculate the gain of the amplifier. Av = ($V_{O/P}$ / $V_{I/P}$).
5. To measure the input impedance, find the voltage drop across the known resistance R_S. The input current therefore is measured as the voltage across Rs / Rs value. Input impedance Zi = Vi / Ii
6. To measure the input impedance, measure the output signal voltage $V_{O/P}$ without any load. Connect a resistive load and then adjust the load until the new output signal $V_{O/P}$ equal to the one half of the original signal. Remove the R_{OUT} from the circuit and measure its value. The measured value is the output impedance of the circuit.
7. To measure the current gain A_I, note down the output signal voltage when Ro is connected and divide it by Ro to get the output current. Now current gain = output current / input current. The power gain is the product of voltage gain and current gain.
8. Vary the frequency of the input signal from 50Hz to 1MHz in suitable steps and calculate gain at each step. Plot the graph between voltage gain Vs frequency. Note down the half power points and find the bandwidth of the amplifier.
9. Repeat the above steps by connecting (disconnecting) the emitter bypass capacitor C_E. The readings with C_E give the response of the amplifier without out feedback. The readings without the C_E give the performance of the amplifier in current series feedback mode.

MODEL GRAPHS:
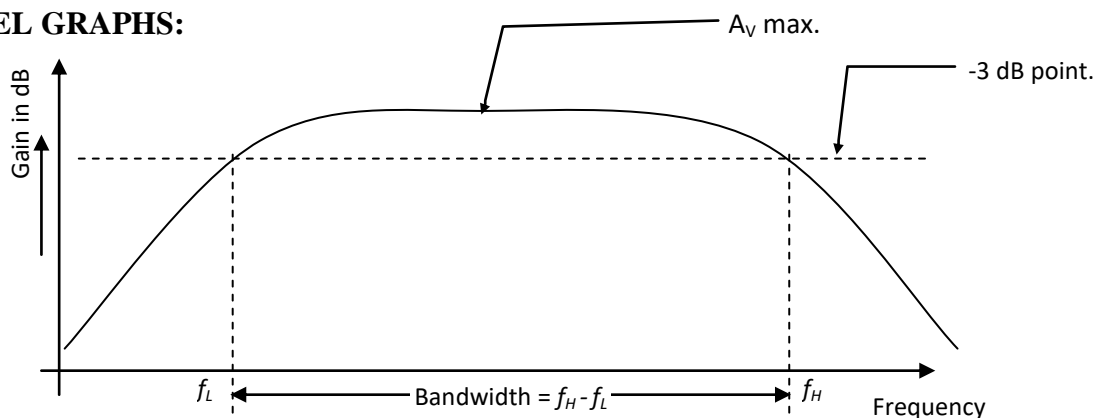
Tabulation:

Without feedback:

S.No	Frequency (Hz)	Input Voltage	Output Voltage (V)	Gain(V_0/V_i)	$20 \times \log_{10}(V_0/V_i)$

With feedback:

S.No	Frequency (Hz)	Input Voltage	Output Voltage (V)	Gain(V_0/V_i)	$20 \times \log_{10}(V_0/V_i)$

RESULTS:

Without feedback
Input voltage (V_i) =
Input frequency =
Output voltage (V_0) =
Voltage gain =
Gain in dB = ($20 \times \log_{10}(V_0/V_i)$) =

With feedback
Input voltage (V_i) =
Output voltage (V_0) =
Voltage Gain =
Gain in dB = ($20 \times \log_{10}(V_0/V_i)$) =

CONCLUSIONS:
        Conclusions can be drawn on comparing the voltage gain of the amplifier with and without feedback, bandwidth obtained from the frequency response plot.

VIVA QUESTIONS:

1. What is the relationship between the transfer gain with feedback A_f and that without feedback A.
2. Define negative feedback.
3. Define the amount of feedback in decibels.
4. State the three fundamental assumptions which are made in order that the expression $A_f = A/(1+A\beta)$ be satisfy exactly.

REDRAW THE CIRCUIT DIAGRAM:

5.SINGLE TUNED VOLTAGE AMPLIFIER

AIM: To study single tuned voltage Amplifier and to calculate

      1. Resonant Frequency.

      2. Q factor.

      3. Bandwidth and

      4. Impedance

APPARATUS:

      1. Tuned RF Amplifier trainer Kit.

      2. Function Generator.

      3. CRO.

      4. BNC probes and connecting wires

CIRCUIT DIAGRAM:

THEORY:

        It is usually required to use a number of tuned amplifier stages in cascade in order to obtain large overall gain. These cascade tuned amplifiers may be put into the following three categories:

      1. Single tuned amplifiers.

      2. Double tuned amplifiers.

      3. Stagger-tuned amplifiers.

Single tuned amplifiers use one parallel tuned circuit as the load impedance in each stage and all these tuned circuits in different stages are tuned to the same frequency.Double tuned amplifier uses two inductively coupled tuned circuits per stage, both the tuned circuits being tuned to the same frequency.Staggered tuned amplifier uses a number of single tuned stages in cascade, the successive tuned circuits being tuned to slightly different frequencies.

Single tuned amplifiers may again be put into following two categories:

        a) Capacitance coupled single tuned amplifiers and

b) Transformer coupled or inductively coupled single tuned amplifiers
RESONANT FREQUENCY:
Depending upon the frequency of the source voltage Vs, the circuits may behave either as inductive or capacitive. However, at a particular frequency when the inductive reactance Xl equals the capacitive reactance Xc, then the circuit behaves as a purely resistive circuit. This phenomenon is called resonance: and the corresponding frequency is called resonant frequency. The resonant frequency (fr) can be found by equating the two reactance values.

PROCEDURE:
1. Connect ions should be made as per the circuit diagram.
2. Connect the AC signal source from function generator (above AF range) to input of the trainer kit.
3. Keep the input voltage constant , vary the frequency in regular steps and down the corresponding output voltage
4. Calculate the resonant frequency.
5. Plot the graph: gain (db) Vs frequency
6. Find the input and output impedance
7. Calculate the bandwidth and Q factor

PRECAUTIONS:
1. Check connections before switching ON power supply
2. Don't apply over voltage
3. When you are not using the equipment switch them OFF.
4. Handle all equipment carefully

EXPECTED GRAPH:
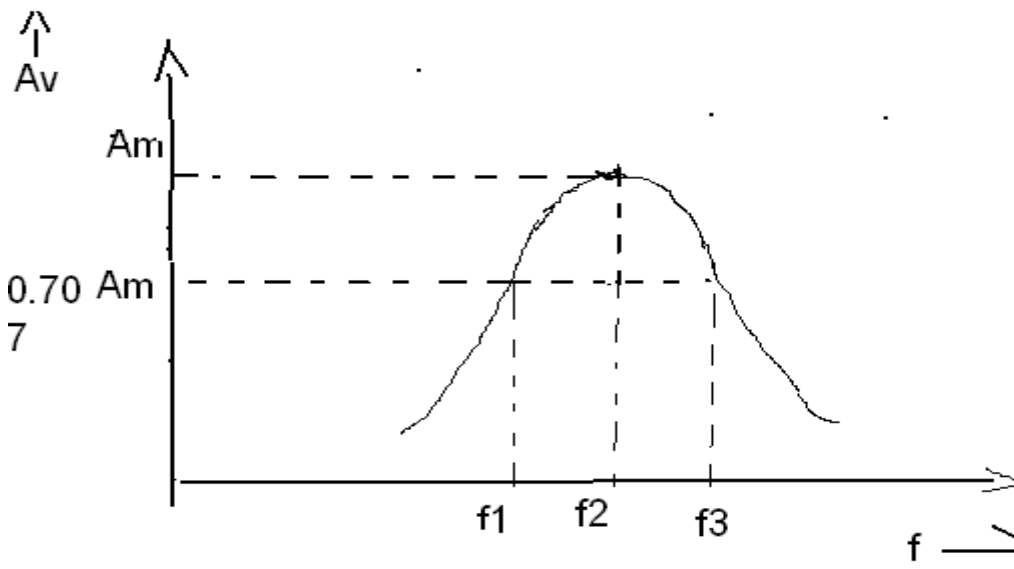
Observations:

Input Voltage=   (Constant)

S.NO	Input Frequency (Hz)	OutputVoltage(V_o)	Gain $A = V_o/V_i$	Gain in dB $20\log(V_o/V_i)$

RESULT: Gain and frequency as observed of single tuned voltage amplifier.

$F_T =$ $F_P=$

VIVA QUESTIONS:
1. What are tuned amplifiers?
2. What is tuning?
3. Explain how the tuned amplifiers work?

REDRAW THE CIRCUIT DIAGRAM:

7.CLASS B COMPLEMENTARY SYMMETRY AMPLIFIER

AIM: To observe the input and output waveforms and to calculate the efficiency of Class B Complimentary symmetry power amplifier.

APPARATUS:
EQUIPMENT REQUIRED:
    Power supply 0-30V- 1 No.
    CRO 20MHz - 1 No.
    Digital multimeter - 1 No.
    Signal generator 1Hz - 1MHz - 1 No.
COMPONENTS:
    Resistors : 2.2KΩ - 2 Nos.
             100Ω   - 2 Nos.
             1KΩ   - 1 No
     Transistors CL100- 1No , CK100 -1NO

CIRCUIT DIAGRAM:

THEORY:
        An amplifying system consists of several stages in cascade. The input and the intermediate stages amplify small signal excitations to a value large enough to drive the final device .The out put stage feeds the final device .The output stage feeds a transducer such as a CRO,loudspeaker or servomotor. Thus the final stage must be capable of delivering a large voltage or current or appreciable amount of power. This requires an amplifier which is referred as a power amplifier

In class B complimentary symmetry class _B amplifier one n-p-n and p-n-p is used. Hence the circuit is called class-B complimentary symmetry amplifier. This circuit ifis transformer less circuit .But with common emitter configuration it becomes power transfer without output impedance for maximum power transfer without an output transformers. Hence the matched pair of complementary transistors are used in common collector configuration This is because in common collector configuration has lowest output impedance and hence the impedance matching is possible.

PROCEDURE:
1.conncet the circuit s per the circuit diagram
2.apply 4v p-p with 1KHZ frequency using function generator
3.observe the output in CRO .
4.note the cross over distortion in output.(outputVp-p)
5.remove the collector connection and put ammeter.
6.note the Idc value in the ammeter.
7.using Pdc and Pac formulas find the efficiency.

OBSERVATION:

Pac=$V_m2/2RL$ =

Pdc=Vcc*Idc =

VO =

VCC =

RL =

EFFICIENCY: η =Pac/Pdc*100=

MODEL GRAPHS: I/P WAVEFORM
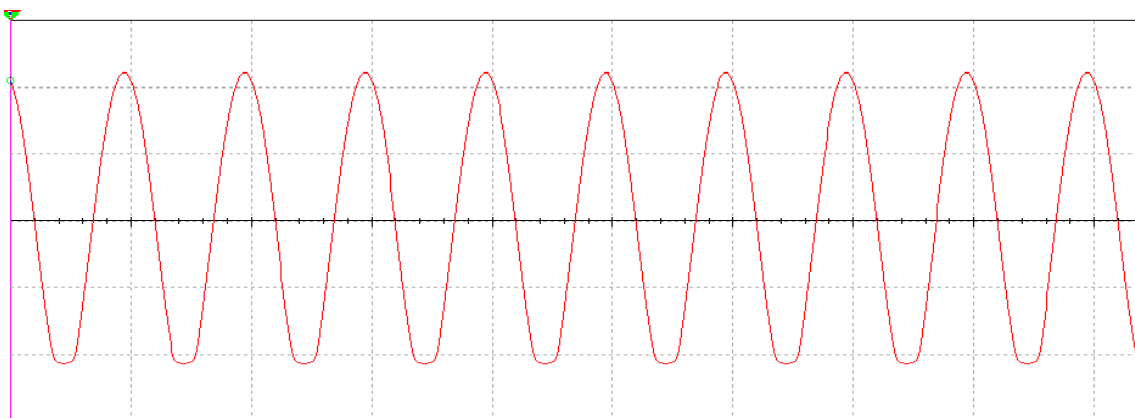
O/P WAVEFORM
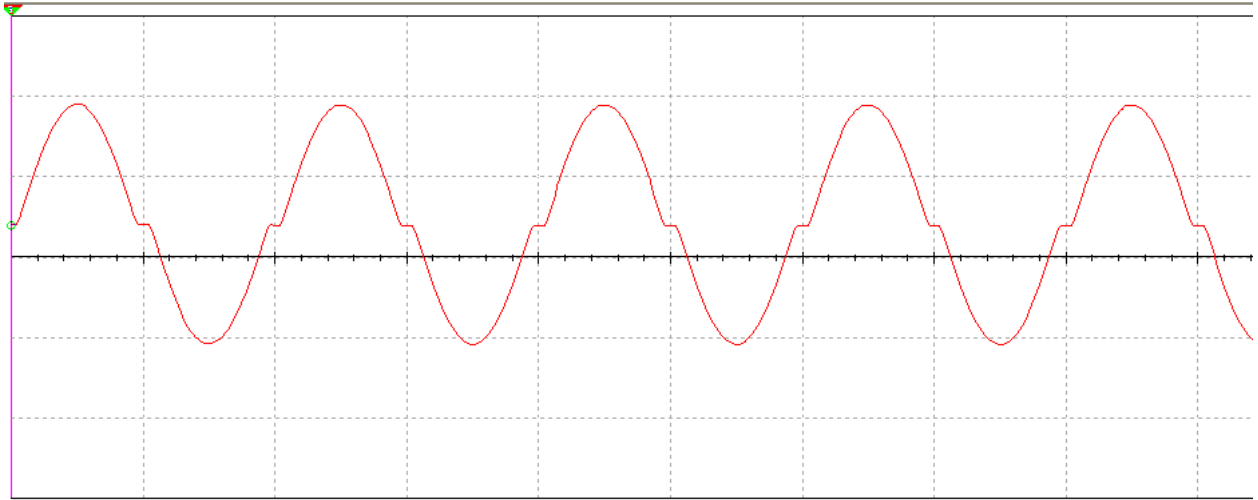
Vivavoice:

1.In classB power amplifiers why crossover distortion will be occurred?

2.what are he different methods to eliminate distortion?

3..howmuch efficiency will b obtained in classs-B power amplifier?

4.where is the location of operating point in class-B power amplifier?

REDRAW THE CIRCUIT DIAGRAM:

7. HARTLEY OSCILLATOR

AIM: To study and calculate frequency of oscillations of Hartley oscillator. Compare the frequency of oscillations, theoretically and practically.

APPARATUS:

Hartley Trainer kit

Connecting wires

CIRCUIT DIAGRAM:
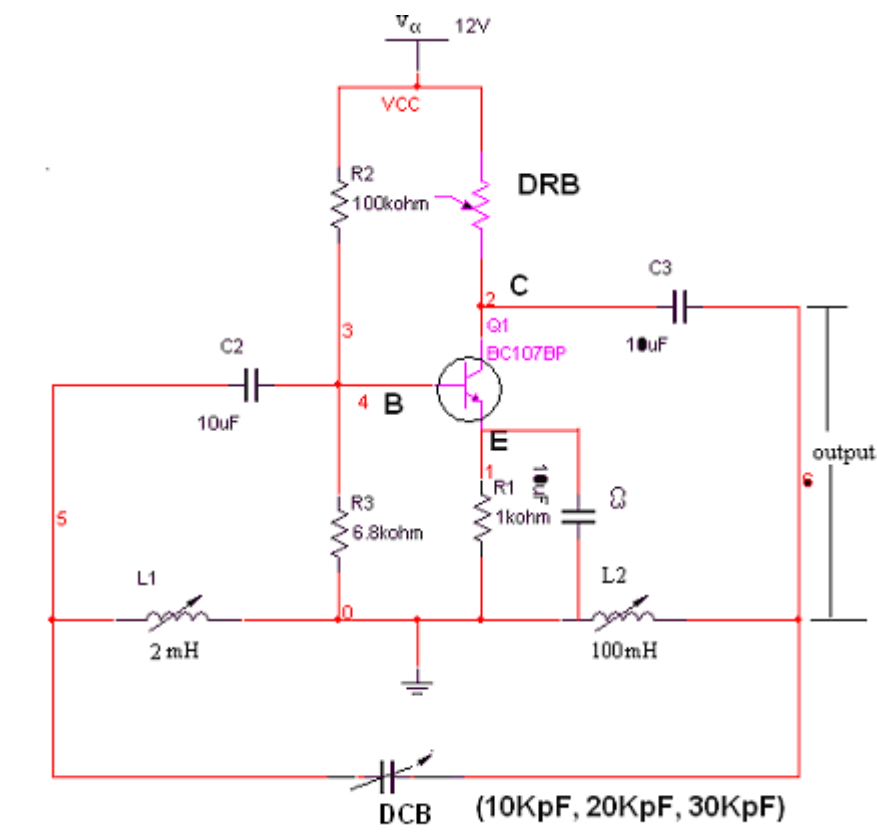
THEORY:

Hartley oscillator is very popular and is commonly used as a local oscillator in radio receivers. It has two main advantages viz... Adaptability to wide range of frequencies and easy to tune. The tank circuit is made up of L1, L2, and C1. The coil L1 is inductively coupled to coil L2, the combination functions as auto transformer. The resistances R2 and R3 provide the necessary biasing. The capacitance C2 blocks the d.c component. The frequency of oscillations is determined by the values of L1, L2 and C1 and is given by,

$$F=1/(2\pi(C1(\sqrt{L1+L2})))$$

The energy supplied to the tank circuit is of correct phase. The auto transformer provides 180˚ out of phase. Also another 180˚ is produced By the transistor. In this way, energy feedback to the tank circuit is in phase with the generated oscillations.

PROCEDURE:

1. Connections are made as per the circuit diagram.

2. Connect CRO at output terminals and observe wave form.

3. Calculate practically the frequency of oscillations by using the Expression.

$F=1/T$, Where T= Time period of the waveform

4. Repeat the above steps 2, 3 for different values of L1 and note Down practical values of oscillations of colpitts oscillator.

5. Compare the values of frequency of oscillations both theoretically And Practically.

OBSERVATIONS:

CAPACITANCE(µF), INDUCTANCE(mH)	Theoritical frequency (KHZ)	Practical frequency (KHZ)

MODEL GRAPH:
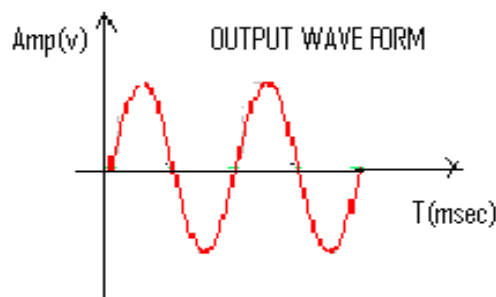
RESULT: Frequency of oscillations is calculated and compared with theoretical values.

$F(theoretical) = 1/2\pi\Gamma LC=$

$F(practicl) =$

VIVA QUESTIONS:

1. What are the applications of LC oscillations?

2. What type of feedback is used in oscillators?

3. What the expression for frequency of oscillations?

4. Whether an oscillator is dc to ac converter?

5. What is the loop gain of an oscillator?

6. What is the difference between amplifier and oscillator?

7. What is the condition for oscillations?

8. How many inductors and capacitors are used in Hartley Oscillator?

9. How the oscillations are produced in Hartley oscillator?

10. What is the difference between damped oscillations undamped oscillations?

REDRAW THE CIRCUIT DIAGRAM:

8. CLASS -A POWER AMPLIFIER(TRANSFORMER COUPLED)

AIM: To observe the input and output waveforms and to calculate the efficiency.

EQUIPMENT REQUIRED:

    APPARATUS REQUIRED:

        Power supply 0-30V- 1 No.

        CRO 20MHz - 1 No.

        Digital multimeter - 1 No.

        Signal generator 1Hz - 1MHz - 1 No.

    COMPONNTS REQUIRED:

        Resistors 33KΩ - 1N0

                5.6KΩ -2NO

                470Ω -1NO

        Capacitors 47uf -1NO

                2.2uf - 1NO

        TRANSFORMER -1NO

CIRCUIT DIAGRAM:

THEORY:

    The amplifier is said to be class A power amplifier if the q point and the input signal are selected such that the output signal is obtained for a full input cycle . For this class the position of q point is approximately y at the mid point of the load line. For all the values of input signal the transistor remains in the active region and never entire into the cutoff or saturation region. The collector current flows for 3600 (life cycle) of the input signal in other words the angle of the collector current flow is 3600 the claa a amplifiers or furthers classified as directly coupled and transformer coupled and transformer coupled amplifiers in directly coupled type .The load is directly connected in the collector circuit while in the transformer coupled type, the load is coupled to the collector using the transformer.

Advantages:

1. Distortion analysis is very important

2. It amplifies audio frequency signals faithfully hence they are called as audio amplifiers

Disadvantages:

1. H parameter analysis is not applicable

2. Due to large power handling the transistor is used power transistor which is large in size and having large power rating

PROCEDURE:

1. CONNCECT the circuit as per the ciecuit diadram

2. Set Vs (say 250 to 300 mV), at 10 KHz using signal generator.

3. Connect milli ammeter to the ammeter terminals

4. By keeping the input voltage constant, vary the frequency from 0 to 1MHz

   in regular steps .

5. Note down the corresponding output voltage from CRO

6. Calculate the DC input power using the formula $Pdc = V_{cc}I_c$

7. Calculate the AC output power using the formula $Pac = V_O{}^2/8R_L$

8. Calculate the efficiency $n = Pac /Pdc$

9. Plot the graph between Gain (db) and frequency.

10. Calculate bandwidth from the graph.

OBSERVATIONS:

VO = _____, VI = _____

VCC = _____

RL = _____

CALCULATIONS:

Efficiency (Pac/ Pdc) = _____

$P_{ac} = V_{cc}\ Ic$

$P_{DC} = V_m/2R_L = V^2{}_{pp}/8R_L$

$\%n = P_{ac}/ P_{DC} \times 100$

GRAPH: I/P
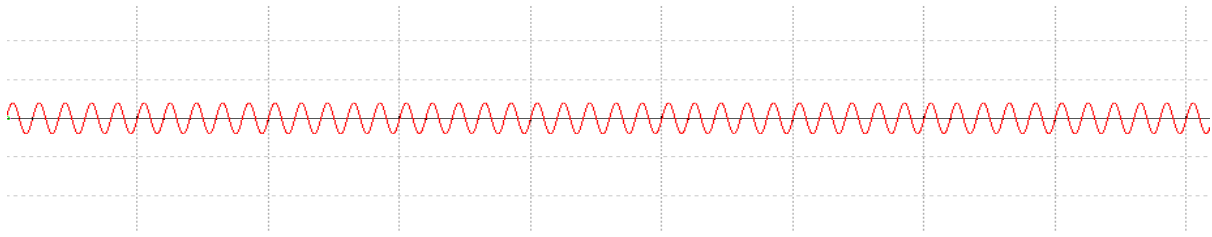
O/P
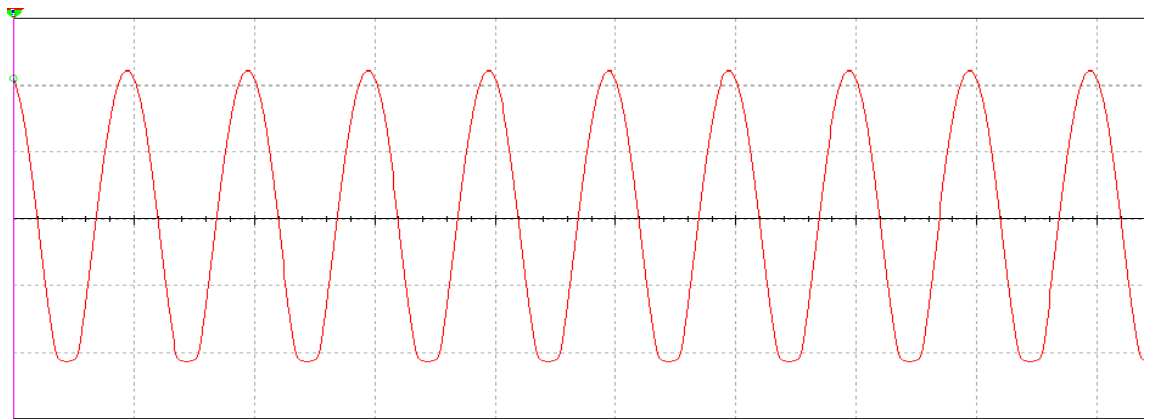
RESULT: Gain and frequency as observed of Class A power amplifier.

$$\%n = P_{ac}/ P_{DC} \text{ X } 100 =$$

VIVA QUESTIONS:

1.On which criteria based the power amplifiers are classified?

2.what is the operating point of class-A power ampliier?

3.what is meant by efficiency?

4.howmuch efficiency will b obtained in classs-A power amplifier?

5. what are the applications of power amplifier?

REDRAW THE CIRCUIT DIAGRAM:

9.VOLTAGE-SERIES FEEDBACK AMPLIFIER

AIM: To study the effect of voltage series feedback on Gain of the Amplifier.

APPARATUS:

Transistor BC 107 -1no

Breadboard

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

Function Generator

CRO (30 Mhz,dualtrace)

Resistors 33kΩ,3.3k Ω,1.5kΩ,1kΩ-2no

Capacitors   10µF - 3Nos

CIRCUIT DIAGRAM:

THEORY:

When any increase in the output signal results into the input in such a way as to cause the decrease in the output signal, the amplifier is said to have negative feedback. The advantages of providing negative feedback are that the transfer gain of the amplifier with feedback can be stablised against varations in the hybrid parameteresof the transistor or the parameters of the other active devices used in the circuit. The most advantage of the negative feedback is that by propere use of this , there is significant improvement in the frequency respponse and in the linearity of the operation of the amplifier.This disadvantage of the negative feedback is that the voltage gain is decreased.

In Voltage-Series feedback , the input impedance of the amplifier is decreased and the output impedance is increased.Noise and distortionsare reduced cosiderably.

PROCEDURE:

1. Connections are made as per circuit diagram.
2. Keep the input voltage constant at 20mV peak-peak and 1kHz frequency.For different values of load resistance, note down the output voltage and calculate the gain by using the expression

   $A_v = 20\log(V_0 / V_i)$ dB
3. Add the emitter bypass capacitor and repeat STEP 2.And observe the effect of Feedback on the gain of the amplifier
4. For plotting the frquency the input voltage is kept constant at 20mV peak-peak and the frequency is varied from 100Hz to 1MHz.
5. Note down the value of output voltage for each frequency. All the readings are tabulated and the voltage gain in dB is calculated by using expression $A_v = 20\log(V_0 / V_i)$ dB
6. A graph is drawn by takung frquency on X-axis and gain on Y-axis on semi log graph sheet
7. The Bandwidth of the amplifier is calculated from the graph using the expression Bandwidth B.W $= f_2 - f_1$.

Where f_1 is lower cutt off frequency of CE amplifier

   f_2 is upper cutt off frequency of CE amplifier

   The gain-bandwidth product of the amplifier is calculated by using the expression

   Gain-Bandwidth Product = 3-dB midband gain X Bandwidth.

OBSERVATIONS:

Frequency Response with feedback: $V_i = 20$mV

S.NO	Frequency (Hz)	OutputVoltage(V_o)	Gain A = V_o/V_i	Gain in dB $20\log(V_o/V_i)$

MODEL WAVEFORMS:

Result:

Bandwidth(with feed back)=f2-f1

Viva voice:

1. what is meant by feed back?

2. what are the types of feed back?

3. why we are using negative feed back in feedback amplifiers?

4.what are the different types of feedback amplifiers?

REDRAW THE CIRCUIT DIAGRAM:

10.CLASS B PUSH PULL POWER AMPLIFIER

AIM: To observe the input and output waveforms and to calculate the efficiency of Class B Complimentary symmetry power amplifier.

EQUIPMENT REQUIRED:
        APPARATUS REQUIRED:
            Power supply 0-30V- 1 No.
            CRO 20MHz - 1 No.
            Digital multimeter - 1 No.
            Signal generator 1Hz - 1MHz - 1 No.
        COMPONNTS REQUIRED:
            Resistors $10K\Omega$ - 1N0
                      $1K\Omega$ - 1NO
                      220Ω -1NO
            Capacitors 0.47uf -1NO
                      22uf - 2NO
            TRANSFORMER -2NO

CIRCUIT DIAGRAM:

THEORY:

An amplifying system consists of several stages in cascade. The input and the intermediate stages amplify small signal excitations to a value large enough to drive the final device .The out put stage feeds the final device .The output stage feeds a transducer such as a CRO,loudspeaker or servomotor. Thus the final stage must be capable of delivering a large voltage or current or appreciable amount of power. This requires an amplifier which is referred as a power amplifier

In class B complimentary symmetry class _B amplifier one n-p-n and p-n-p is used. Hence the circuit is called class-B complimentary symmetry amplifier. This circuit ifis transformer less circuit .But with common emitter configuration it becomes power transfer without output impedance for maximum power transfer without an output transformers. Hence the matched pair of complementary transistors are used in common collector configuration This is because in common collector configuration has lowest output impedance and hence the impedance matching is possible.

PROCEDURE:
1.conncet the circuit s per the circuit diagram
2.apply 4v p-p with 1KHZ frequency using function generator
3.observe the output in CRO .
4.note the cross over distortion in output.(outputVp-p)
5.remove the collector connection and put ammeter.
6.note the Idc value in the ammeter.
7.using Pdc and Pac formulas find the efficiency.

OBSERVATION:
VO =

VCC =

RL =

EFFICIENCY: $P_{ac}/P_{DC} = V_{mx}II/4 \times V_{CC}$ =

Vm = V pp/2

MODEL GRAPHS: I/P
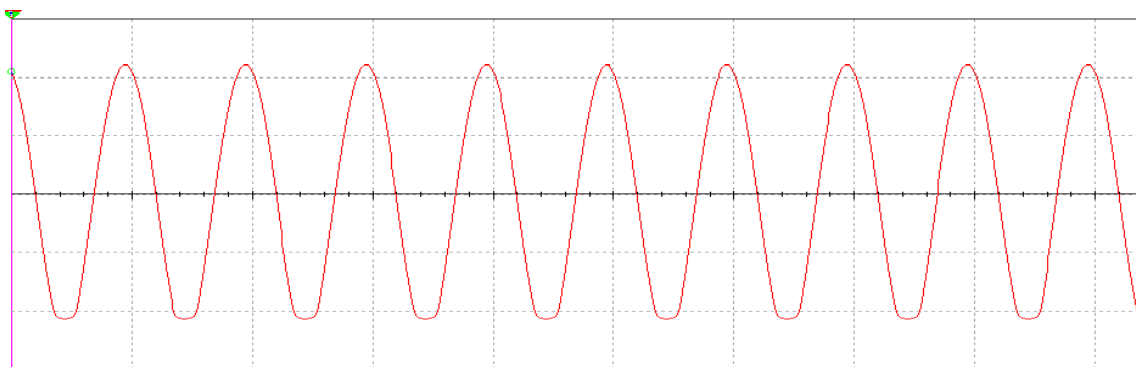
O/P
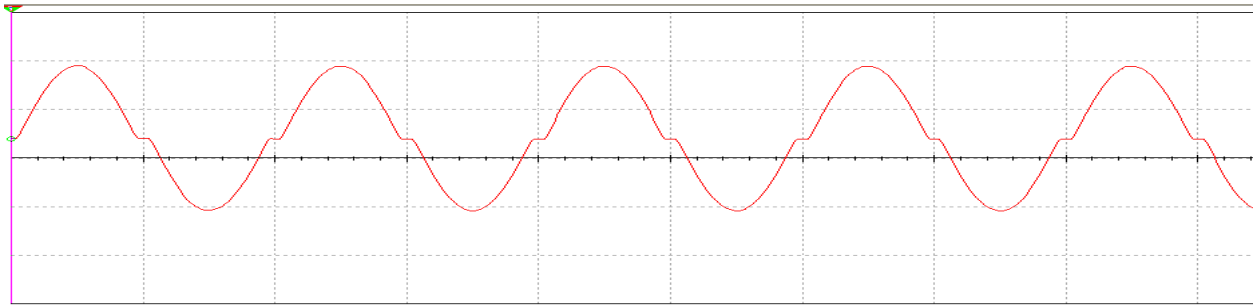
RESULT: The efficiency of class B complimentary symmetry power amplifier is obtained.

EFFICIENCY: $P_{ac}/P_{DC} = V_{mx} II /4 \times V_{CC} =$

VIVA QUESTIONS:
1.In classB power amplifiers why crossover distortion will be occurred?

2.what are he different methods to eliminate distortion?

3..howmuch efficiency will b obtained in classs-B power amplifier?

4.where is the location of operating point in class-B power amplifier?

REDRAW THE CIRCUIT DIAGRAM:

INTRODUCTION TO PSPICE:

PSPICE is abbreviated as Personal Simulation Program with Integrated Circuit Emphasis. **PSpice** is a <u>SPICE</u> <u>analog circuit</u> and digital logic <u>simulation</u> program.

SPICE (Simulated Program with Integrated Circuit Emphasis) is a general purpose software that simulates different circuits and can perform various analysis of electrical and electronic circuits including time domain response, small signal frequency response, total power dissipation, determination of nodal voltages and branch current in a circuit, transient analysis, determination of operating point of transistors, determinations of transfer functions etc. This software is designed in such a way so that it can simulate different circuit operations involving transistors, operational amplifiers (op – amp) etc. and contains models for circuit elements (passive as well as active).

SPICE was first developed in the University of California, Berkeley, USA in early 1970s. Subsequently an improved version SPICE 2 was available in the mid of 1970s especially to support computer aided designs. In due course of time this program (SPICE 2 has become so versatile in the industry that people used to call, this program itself as SPICE. PSpice is also the member of SPICE family and it is a commercial software product based on SPICE algorithm. It is useful for simulating all types of circuits in a variety of applications. In both SPICE and PSpice, the circuit is described by statements those are stored in a file (namely Circuit File). The SPICE simulator is assigned to read this file to run the simulation. In PSpice, the statements are self – contained and independent; obviously they do not interact with each other. The statements are also easy to learn and use. PSpice includes additional features that make the program more flexible and user friendly. Notably among other features is the graphics post processor probe which acts like a software oscilloscope and is capable of exhibiting various waveforms. PSpice has become one of the most popular circuit simulation programs. In order to draw the circuit and create a schematic file, schematic editor can be used in the PSpice simulation.

PSpice is a part of larger software package called the Design Lab, originally developed by MicroSim Corporation as the Design Centre. It is now marketed by Or CAD.

OVERVIEW:

Start a Project → Draw a schematic → Simulate circuit → Graph data

Each part will be discussed in more detail in the next four sections.

1.   STARTING A PROJECT

2.   DRAWING A SCHEMATIC

1. Summary of PSpice Parts

2. What your Schematic Needs

3. Adding Parts to your Circuit

4. Using Wires

5. Adding a Ground

6. Changing the Value of a Part

7. Other Notes: Unique Names and Labeling Nodes

3.   SIMULATING YOUR CIRCUIT

     1. General Instructions

     2. Bias Point (DC Calculations)

     3. DC Sweep on Input Source, V_i

     4. Parametric Sweep

     5. AC Sweep (Frequency Domain Simulation)

     6. Transient Analysis (Time Domain Simulation)

4.   GRAPHING IN PSPICE

     1. General Instructions

     2. Bode Plot

5.   MISCELLANEOUS ITEMS

     1. Creating an IV Plot

     2. Using the OpAmp

     3. Family of Curves Option for Resistance

     4. Using the Zener Diode

     5. Creating a Potentiometer

1. STARTING A PROJECT

1) Open OrCAD Capture

2) Go to File => New => Project…

3) Enter a name

4) Choose "Analog or Mixed A/D"
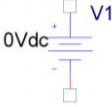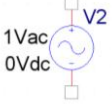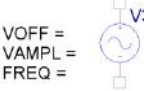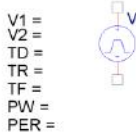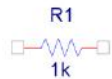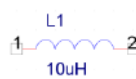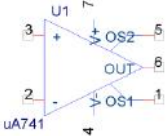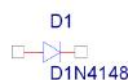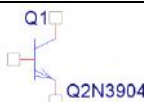
5) Set the location. (You should create a new directory for your project since PSpice will generate a bunch of project files in this folder.)

6) Click OK

7) Choose "Create blank project" and click OK

8) You should now see a window where you can draw the schematic (i.e., your circuit diagram).
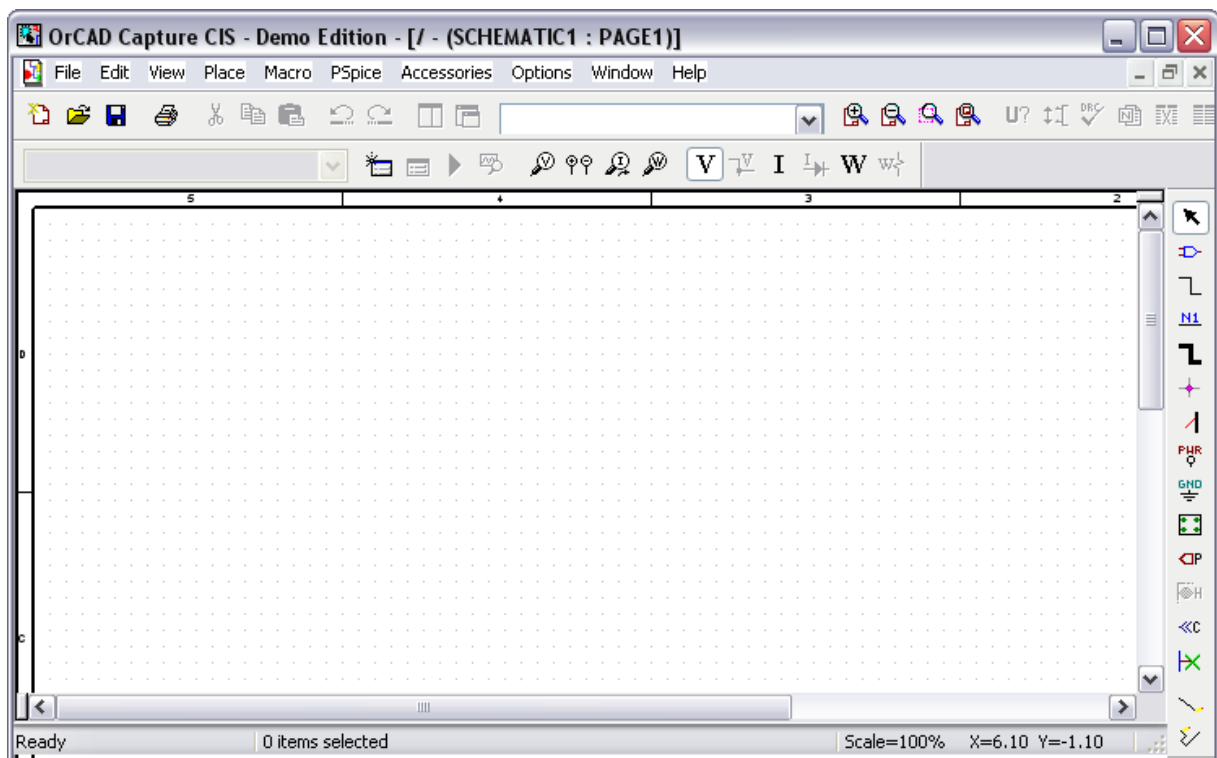
2. DRAWING A SCHEMATIC

2.1 Summary of PSpice Parts for ECE 65

PART	PART NAME	PICTURE	NOTES
DC Source	VDC / Source		
AC Source	VAC / SOURCE		

Sine Wave Source	VSIN / SOURCE	VOFF = VAMPL = FREQ =	See Section 4.6 for more instructions
Triangle Wave Source Square Wave Source	VPULSE / SOURCE	V1 = V2 = TD = TR = TF = PW = PER =	See Section 4.7 for more instructions
Ground (ref. voltage)	0 / SOURCE	0	
Resistor	R / ANALOG	R1 1k	
Capacitor	C / ANALOG	C1 1n	
Inductor	L / ANALOG	L1 10uH	
741 OpAmp	uA741 / EVAL	U1 uA741	See Section 6.2 for more instructions
Diode	D1N4148 / EVAL	D1 D1N4148	
Zener Diode	D1N5232 / EVAL	D2 D1N5232	See Section 6.4 for more instructions
npn BJT	Q2N3904 / BIPOLAR	Q1 Q2N3904	

2.2 What your Schematic Needs

All schematics you draw on PSpice will need the following: a voltage source, components, wires, and a ground. The next couple of sections will instruct you on how to draw a full circuit.

2.3 Adding Parts to your Circuit

      1) Go to Place => Parts
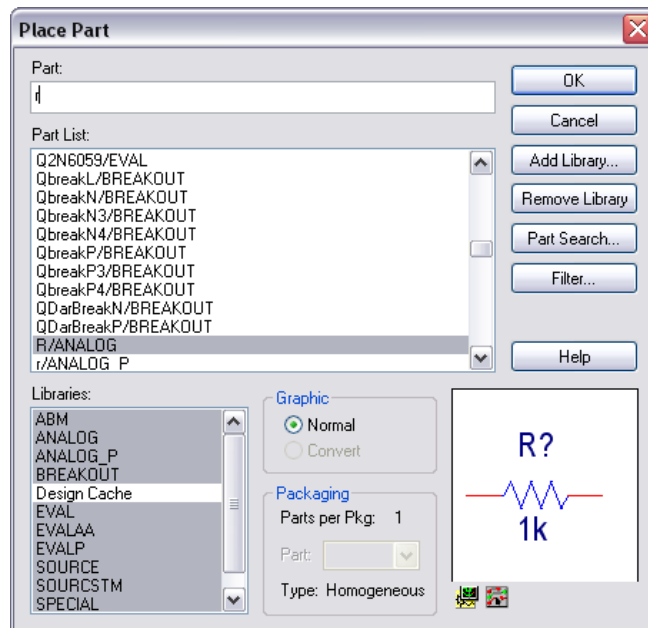
2) Click on the library you want to use, or select multiple libraries by holding Ctrl or dragging the mouse. In the parts window you should see at least the ANALOG, BIPOLAR, EVAL, SOURCE, and SPECIAL libraries. If you don't see these libraries already listed, you will need to add them:

a. Click Add Library…

b. Navigate to C:\Program Files\OrCad_Demo\Capture\Library\Pspice (This is the location in the PSpice lab computers. The location may be different if you install PSpice on your own computer, but find the ...\Capture\Library\Pspice folder)

c. Highlight all the *.olb files in this folder. You can hold Ctrl and click on the files, or drag the mouse to select multiple files.

d. Click Open. You should now see a list of libraries in the "Libraries:" section.

3) Find the part you want to add and press OK.

4) Click where you want to place the part on your schematic. (Press R to rotate the part by 90 degrees)

5) When you are finished with the part, right click and select End Mode to return to the pointer.

2.4 Using Wires

1) Select Parts => Wire. The pointer changes to a cross-hair.

2) Drag cursor from one connection point to another. Clicking on any valid connection will end the wire.

3) Continue connecting the rest of the circuit.

4) When you are finished, right click the mouse and select End Wire to return to the pointer. An example circuit from Lab 1 is shown below.
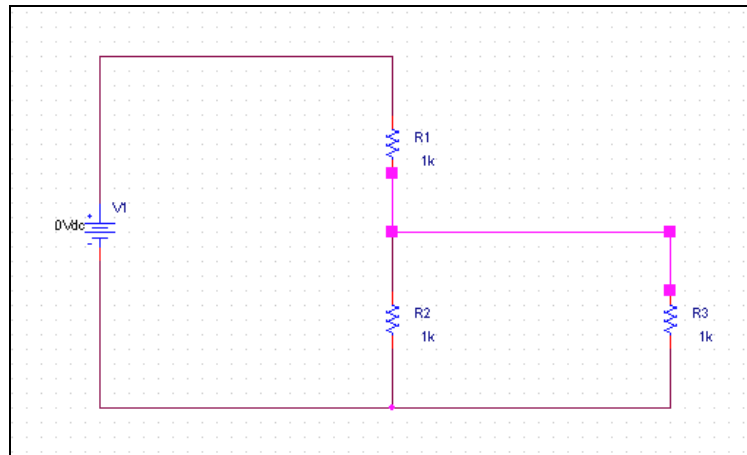
2.5 Adding a Ground

There are many types of grounds (common points in the circuit, noise reduction, etc.) PSpice uses node-voltage method for circuit simulation and, therefore, needs a reference node with "zero voltage". This is the **0/source** ground. You need to have it in your circuits! (It looks like a ground symbol with a zero.) If you don't, PSpice may complain of "floating nodes" even if you have a ground.

To place the ground on the circuit:

1) Go to Place => Ground. The ground you want to use is either listed as 0 or 0/source.
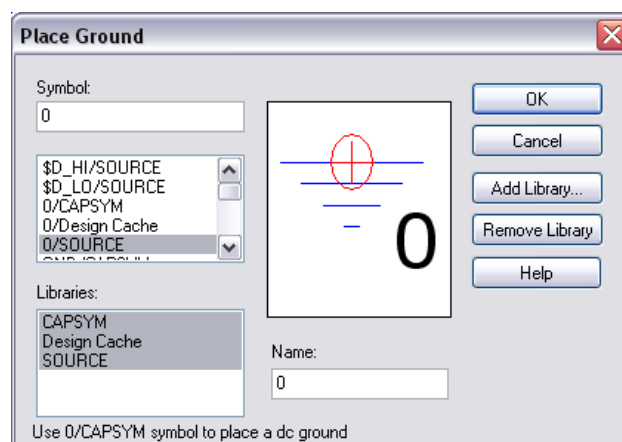
If you don't see the 0/source ground, you will need to add the "source" library:

a. Click Add Library…

b. Navigate to C:\Program Files\OrCad_Demo\Capture\Library\Pspice (This is the location in the PSpice lab computers. The location may be different if you install PSpice on your own computer, but find the ...\Capture\Library\Pspice folder)

c. Highlight source.olb.

d. Click Open. You should now see the "source" library and the 0/source ground.

2) Connect the ground to your circuit.
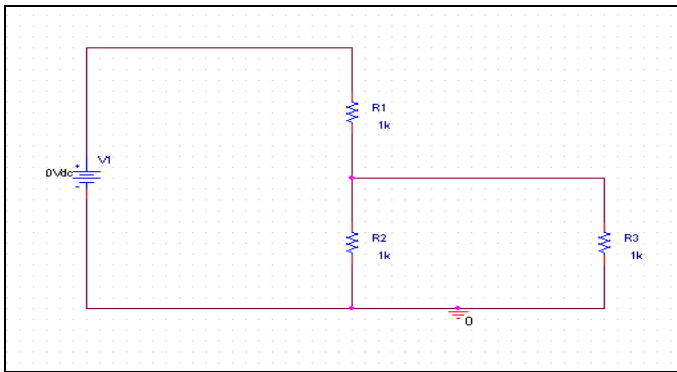
2.6 Changing the Value of a Part
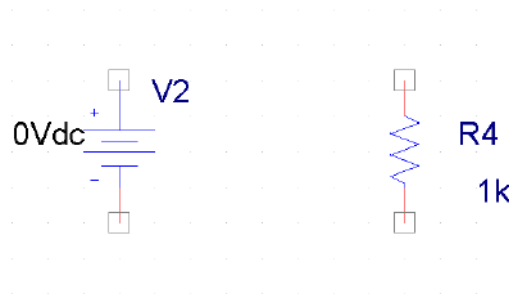
For the parts above, V2 and R4 are the *names* of the components, while 0Vdc and 1k are the *values*. To change a part's value, double-click the *value* of the part. A new window will pop up where you can type in the value you want.
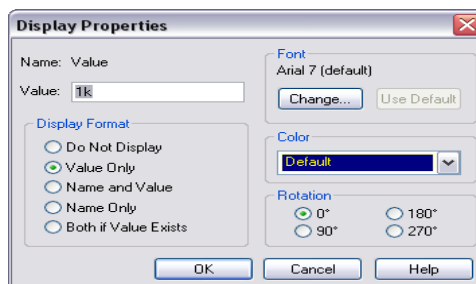
Special Characters	Meaning	Example	What to Type
M	milli (10^{-3})	10 mH	10m
K	kilo (10^{3})	1 kΩ	1k
MEG	mega (10^{6})	10 MΩ	10MEG

2.7 Other Notes

1) **All parts must have unique names.** You can't have two parts named "R1" in your circuit. If you are copying and pasting parts or circuits from another project, you will need to rename your parts because PSpice doesn't do this automatically.

2) **Labeling Nodes.** I recommend you use aliases to label your input and output nodes. This makes your node easier to find when you start plotting out your data. V(Vout) is simpler than finding V(R1:1)

    a. Go to Place => Net Alias

    b. Enter a name, *i.e.,* Vout or Vin

    c. Place the label on the wire connected to the node.

An example of labeling from Lab 1 is shown below.
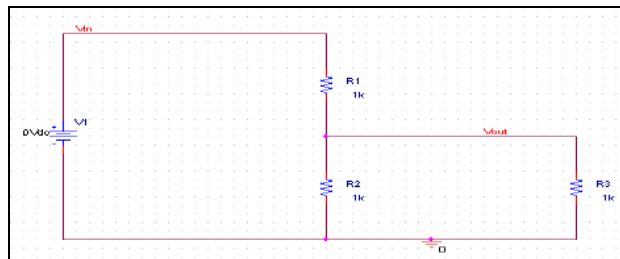
3. SIMULATING YOUR CIRCUIT

3.1 General Instructions

---
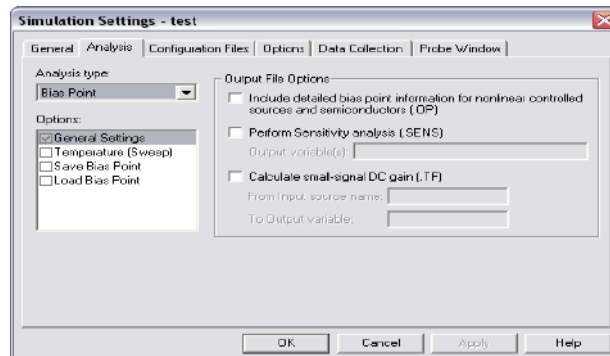
1) Go to PSpice => New Simulation Profile. Or if you already have a profile and would like to edit it, go to Edit Simulation Profile

2) Choose the analysis type from the drop down menu.

3) Adjust the settings on the right hand side. More instructions are given in the next four sections.

4) Press OK.

5) Go PSpice => Run. Or press the play button.

6) A new window (the simulation window) will pop up. Any errors from your circuit will be displayed on the bottom left text window. Fix those errors before you continue. If there are no errors, you are now ready to do one of two things: plot data on the simulation window or display the DC calculations on your schematic.

3.2 Bias Point (DC Calculations)

1) Analysis Type: Bias Point

2) Options: General Settings

    a. For Lab #7, you will also need to select Temperature (Sweep) to run the simulation at different temperatures.

3) Output File Options: None

Press OK and then simulate your circuit. To display DC bias voltages and currents on your circuit after you run the simulation, go to PSpice => Bias Points, and check Enable, Enable Bias Current Display, and/or Enable Bias Voltage Display. You should now see values on your circuit representing current and/or voltage.

3.3 DC Sweep on Input Source V_i

1) Analysis type: DC Sweep

2) Options: Primary Sweep

3) Sweep Variable: Voltage Source

4) Type in the name of the source you are sweeping.

5) Sweep Type: Linear (this is so you can sweep three a range of values)

6) Fill in the Start, End, and Increment Values. Type in 0.1V for the increment value to get a nice smooth plot.

3.4 Parametric Sweep for Resistance

You will need to make the following changes to your circuit first:

1) Change the **value** of the part (not the name!) to {RL} (use curly braces, name is arbitrary)

2) Go to Place => Part

3) Add the part PARAM/SPECIAL to your schematic

4) Double click on the PARAM part

5) Click "New Column..."

6) Set the name to RL (same name as in "a" but with no curly braces)

7) Set the value to something, *e.g.,* 1k (this is the value that is used in calculating DC bias values, choose somewhere in the range of your sweep).

8) Select the RL column (do not double click!) so that it is highlighted and then click Display...

9) Select "Name and Value" and press OK.

10) An example schematic from Lab 1 is shown below:
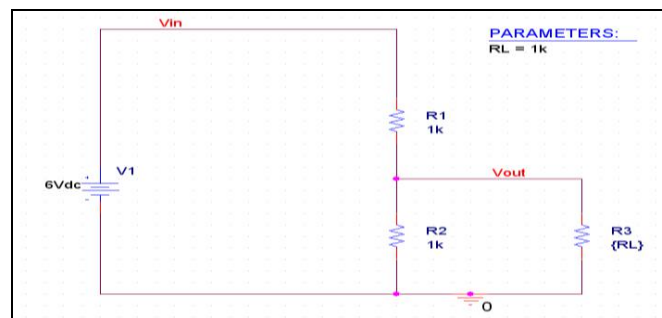
Simulation Settings:

1) Analysis type: DC Sweep

2) Options: Primary Sweep (not Parametric Sweep!)

3) Sweep variable: Global parameter

4) Parameter name: RL (or name of the parameter you used without curly braces)

5) Sweep Type: Linear

6) Fill in the Start, End, and Increment values. Note for resistance, the start value cannot be 0! Use 0.1 instead.

Press OK and simulate. The simulation window should now include a place for you to plot your data.

3.5 AC Sweep (Frequency Domain Simulation)

1) Set up your circuit with VAC voltage sources.

2) Go to PSpice => New or Edit Simulation Profile

3) Analysis Type: AC Sweep/Noise

4) Sweep Type: choose logarithmic and decade. Then select the frequency range of interest. Don't start frequency sweeps at 0!

5) Set the Points/Decade to be at least 20.

Press OK and simulate. The simulation window should now include a place for you to plot your data.

3.6 Transient Analysis (Time Domain Simulation)

1) For a sine wave, use **VSIN** for your voltage source instead of VAC (VOFF is the DC offset, VAMPL is the amplitude, and FREQ is the frequency of the sine wave).

2) For a square or triangular wave, use **VPULSE** . The values you type in for V1 and V2 will depend on the amplitude specified on the lab instructions. If a 5V amplitude signal is specified, then V1 = 5V and V2 = -5V.
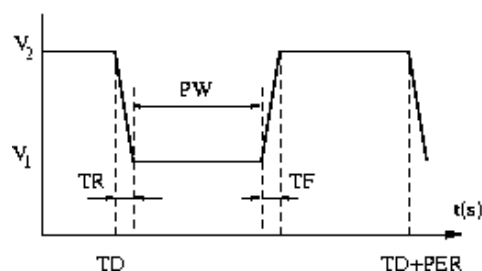
a. **Square Wave** is the VPLUSE function in the limit of TR = TF = 0 and PW = 0.5 * PER (PER is the period of the wave). This limit case, however, causes numerical difficulties in calculations. In any case, we can never make such a square function in practice. In reality, square waves have very small TR and TF. Typically, we use a symmetric function, i.e., we set TR = TF and PW = 0.5 * PER - 2 * TR. Thus, for a given frequency we can set up the square function if we choose TR. If we choose TR too large, the function does not look like a square wave. If we choose TR too small, the program will take a long time to simulate the circuit and for TR smaller than a certain value, the simulation will not converge numerically. A good choice for TR is to set it to be 1% of the PER (a period): TR = TF = 0.01 * PER, PW = 0.48 * PER. This usually results in a nice signal without a huge amount of computational need. Note that TR does not have to be exactly 1% of PER. You can choose nice round numbers for TR, TF, and PW.

b. **Triangular Wave** is the VPLUSE function in the limit of TR = TF = 0.5* PER and PW = 0 (convince yourself that this is the case). As before, the limit case of PW = 0 causes numerical difficulties in calculations. So we have to choose PW to be a reasonably small

value. A good choice for PW is to be set at 1% of the PER (period): PW = 0.01* PER, TR = TF = 0.49 * PER (and not TR = TF = 0.495 * PER so that we get a symmetric function). This usually results in a nice signal without a huge amount of computational need. Again, note that PW does not have to be exactly 1% of PER. You can choose nice round numbers for TR, TF, and PW.

3) Simulation settings:

   a. Analysis Type: Time Domain (Transient)

   b. Options: General Settings

   c. Enter a Run to time so that a few periods will be displayed. Remember that the period (seconds) = 1/frequency (Hz), i.,e, if you are using a 1kHz sine wave, it has a 1/1kHz=1ms period, so use a Run to time of 5ms for 5 periods

   d. **Set the Maximum step size to be much smaller than the period**. i.,e, for a 1kHz sine wave: It has a 1ms period, so set a maximum step size of approx .01ms. (This works out to 100 data points per period). If you don't set the maximum step size, PSpice may choose one which is too big, making your sine wave look angular and ugly.

Press OK and simulate. The simulation window should now include a place for you to plot your data.

4 GRAPHING IN PSPICE

4.1 General Instructions

On the simulation window,

1) Go to Trace => Add Trace

2) Select the variable you want to plot on the y-axis. Or type in an expression on the Trace Expression prompt at the bottom of the window. Press OK

3) To mark points:

   a. Click the "Toggle Cursor" button.   (Or go through the menu, Trace => Cursor => Display.) You will now be able to move the cursor along your plot.

   b. Click the "Mark Label" button to label that point. (Or go through the menu, Plot => Label => Mark.)

4.2 Bode Plots

1) For the magnitude plot, use the PSpice DB() function to convert the transfer function to decibels. For example, you could type in **DB(V(Vout)/V(Vin))** as your Trace Expression,

assuming you have labeled your output and input nodes with "Vout" and "Vin" aliases. Note that DB(Vout) is NOT the transfer function in dB.

2) Next, mark the cutoff frequency on the magnitude plot. To find the cutoff frequency, remember the cutoff frequency is 3dB below the highest point (NOT always at -3dB). Here are some instructions on how to label the cutoff frequencies.

a. Click the "Toggle Cursor" button. (Or go through the menu, Trace => Cursor => Display.)

b. Click the "Cursor Max" button to find the highest point. (Or go through the menu, Trace => Cursor => Max.)

c. Click the "Mark Label" button to label the max point. (Or go through the menu, Plot => Label => Mark.) This point is the center frequency f_o for a bandpass filter.

d. Click the "Cursor Search" button (Or go through the menu, Trace => Cursor =>Search Commands…)

e. Select 1 for Cursor To Move to search along the y-axis

f. To find the cutoff frequency f_c (or cutoff frequencies f_{cl} and f_{cu} for a bandpass filter), enter "search forward level (max-3)" (don't enter the quotation marks) to move the cursor to the right to the point which is 3dB below the max. Or enter "search back level (max-3)" (don't enter the quotation marks) to move the cursor to the left
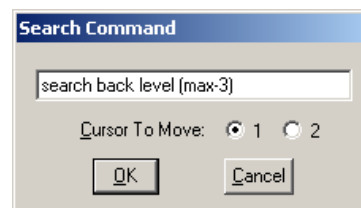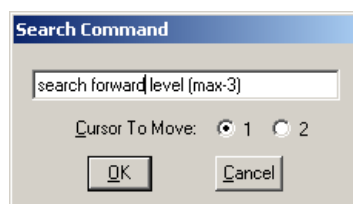
f. Click the "Mark Label" button to label that cutoff point.

- Unclick the Toggle Cursor button to disable the cursor so you can move the label.
- Double click on the label to edit the text (to add units, or to name the point)

3) Once you have completed the magnitude plot, you will now need to create a phase plot. To put the plot on the same window for convenience, go to Plot => Add Plot to Window. To graph the phase plot, use the PSpice P() function. For example, **P(V(Vout)/V(Vin))**.

4) To label the cutoff frequencies on the phase plot, simply search for the angles that correspond to each cutoff frequency. You can find these in the class lecture notes. For

example, for a passive lowpass filter, the cutoff frequency is located where the phase shift is -45 degrees. So on the plot, you would search for -45 and then label that point.

5) It may help to increase the width of the lines in the plot:

  a. The colored symbol at the bottom of the graph, or on the graph line.

  b. Note you can select all of the lines by going to Edit => Select All.

  c. Right click on the line. Make sure the selection list has Information, Properties, Cursor 1, and Cursor 2. (If it lists Settings and Properties, you clicked on the background, not on the line).

  d. Select Properties.

  e. You can change the width and other settings of that trace.

6) An example of a complete Bode plot with labels is shown below:
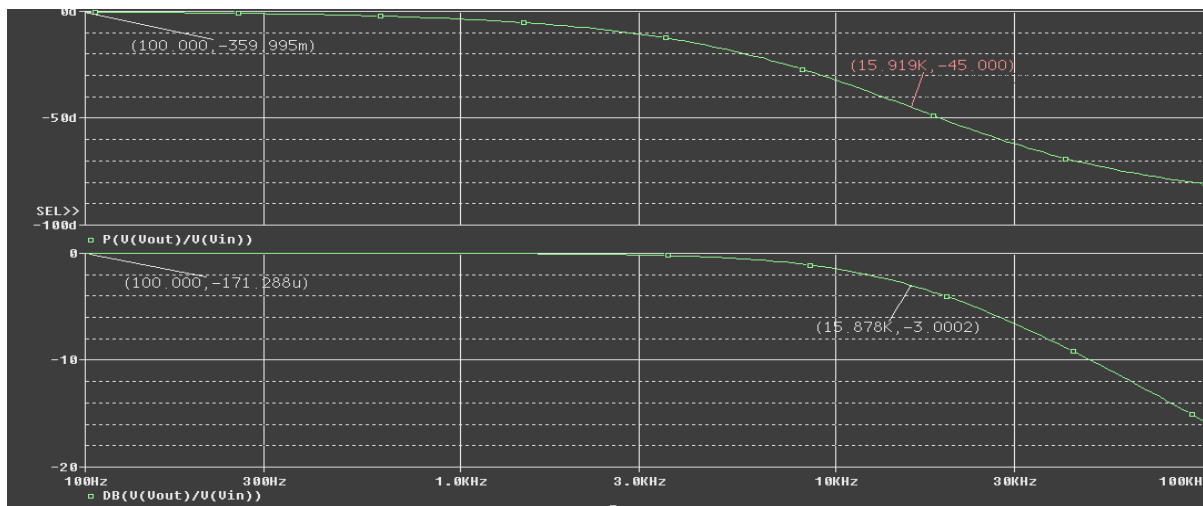
5. MISCELLANEUOUS ITEMS

5.1 Creating an IV Plot

Here, you will need to make a plot of current vs. voltage.

1) Use the same schematic and settings you made for a parametric sweep on the resistance.

2) Simulate the circuit.

3) On the simulation window, go to Trace => Add Trace.

4) Select the current through the load resistance. If you typed in RL as your parameter, the variable you will choose will be I(RL).

5) Add a negative sign in the trace expression, so that you have –I(RL). You will need to do this because you want the current going into the voltage divider circuit. Without the negative sign, you will be plotting the current coming out of the voltage divider circuit.

6) Change the x-axis variable from resistance to voltage:

   a. Go to Plot => Axis Setting

   b. Under the X Axis tab, click Axis Variable…

   c. Select the voltage across the load. This will probably be V2(RL) if RL is your parameter name.

7) Press OK. Your graph should now be a plot of I vs. V.

5.2 Using the OpAmp
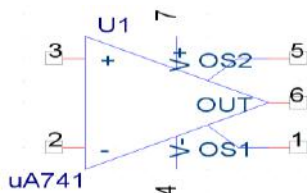
1) Connection points 2, 3, and 6 should be self-explanatory. See the circuit diagram in your lab instructions to figure what to connect to these points.

2) Connection point 7: attach a VDC to supply power to the OpAmp and change the value to 15V. Don't forget to ground the DC source by attaching the 0/source to the negative terminal.

3) Connection point 4: attach a VDC to supply power to the OpAmp and change the value to

-15V. Don't forget to ground the DC source by attaching the 0/source to the negative terminal.
4) Do not connect anything to connection points 1 and 5.

5) An example circuit is shown on the next stage:
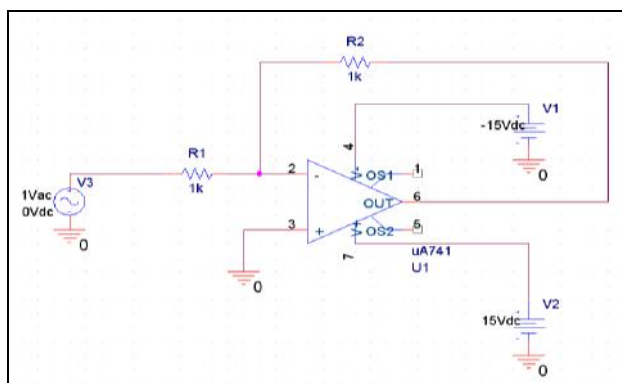
5.3 Family of Curves Option for Resistance

First, you will need to make changes to your circuit so that your resistance R2 is now a parametric variable. Follow the first part of the instructions in section 4.4 on how to do this. Do not follow the instructions for the simulation settings. You will need to follow these instructions:

1) Analysis type: AC Sweep

2) Options: General Settings and Parametric Sweep

3) For General Settings,

    a. AC Sweep Type: Logarithmic Decade

    b. Fill in Start, End, and Points/Decade

4) For Parametric Sweep,

    a. Sweep variable: Global parameter

    b. Parameter name: R (or name of the parameter you used on the schematic minus the curly braces)

    c. Sweep type: Value list

    d. Fill in the list box. Make sure to separate each value with a space and not a comma. For example, you would type in 1k 2k 3k and not 1k, 2k, 3k.

Once you have set up the Sweep Type, press OK and then simulate your circuit. The simulation window should now include a place for you to plot your data. See Section 5.

5.4 Using the Zener Diode

1) You need 2 files

    a. D1N5232.lib (PSpice library file)

    b. D1N5232.olb (Orcad Capture library file)

2) PSpice Instructions:

    a. Go to the menu: 'PSpice => Edit Simulation Settings'

    b. Go to the 'Libraries' tab. Click the 'Browse...' button. Open the D1N5323.lib file

    c. Click 'Add as Global'

    d. Press 'OK' to exit the simulation settings.

    e. Now go to the menu: 'Place => Part...'

f. Click 'Add Library'

g. Open the D1N5232.olb file

h. You should now see a part named D1N5232. Select it and press OK to use the part.

5.5 Creating a Potentiometer
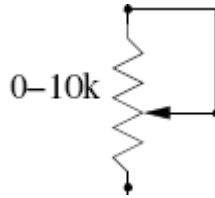
You will not need to use a special part for this. Simply perform a parametric sweep on a regular resistor

1.TWO STAGE RC COUPLED AMPLIFIER

AIM: To plot the frequency response of a two stage RC –coupled amplifier using BJT and calculate 3-db band width

APPARATUS:

      PSPICE Simulation Software

THEORY:

      Whenever large amplification with very good impedance matching is required using an active device such as a transistor or a field effect transistor a single active device and its associated circuitry will not be able to cater to the needs. In such a case single stage amplifier is not sufficient and one requires more stages of amplification i.e., output of one stage is connected to the input of second stage of amplification circuit and the chain continues until the required characteristics of amplifier is achieved such an amplifier is called as multistage amplifier In multistage amplifier, the output signal preceding stage is to be coupled to the input circuit of succeeding stage. For this interstage coupling different types of coupling can be employed. They are

1. RC coupling

2. Transformer coupling

3.   Direct coupling

RC coupling is most popularly used type of coupling because it is cheap and provides excellent fidelity over a wide range of frequency .it is usually employed for voltage amplification .A coupling capacitor is connected to the output of first stage to the base of the input of the second stage and so on.

CIRCUIT DIAGRAM:

DC SWEEP:

TRANSIENT:

AC SWEEP:

RESULT:

VIVA QUESTIONS:

1.What is an amplifier?
2.what are the coupling elements of n this amplifier?
3.what are the applications of this amplifier?
4.why this type of amplifier will be used in AF mplifiers?
5.which type of biasing isused in this amplifier?
6.whatis the range of bandwidth in this amplifier?

2.RC PHASE SHIFT OSCILLATOR

AIM: To calculate the frequency of the RC phase shift oscillator & to measure the phase angles at different RC sections.

APPARATUS:

PSPICE Simulation Software

THEORY:RC-Phase shift Oscillator has a CE amplifier followed by three sections of RC phase shift feed back Networks the out put of the last stage is return to the input of the amplifier. The values of R and C are chosen such that the phase shift of each RC section is 60º.Thus The RC ladder network produces a total phase shift of 180º between its input and output voltage for the given frequencies. Since CE Amplifier produces 180 º phases shift the total phase shift from the base of the transistor around the circuit and back to the base will be exactly 360º or 0º. This satisfies the Barkhausen condition for sustaining oscillations and total loop gain of this circuit is greater than or equal to 1, this condition used to generate the sinusoidal oscillations.   The frequency of oscillations of RC-Phase Shift Oscillator is,

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

CIRCUITDIAGRAM:

PROCEDURE:

1. Enter in to the Pspice software.
2. Customize the screen and then draw the circuit on the screen .
3. Start the simulation and observe the input and output waveforms.

SIMULATION RESULTS:

DC SWEEP:
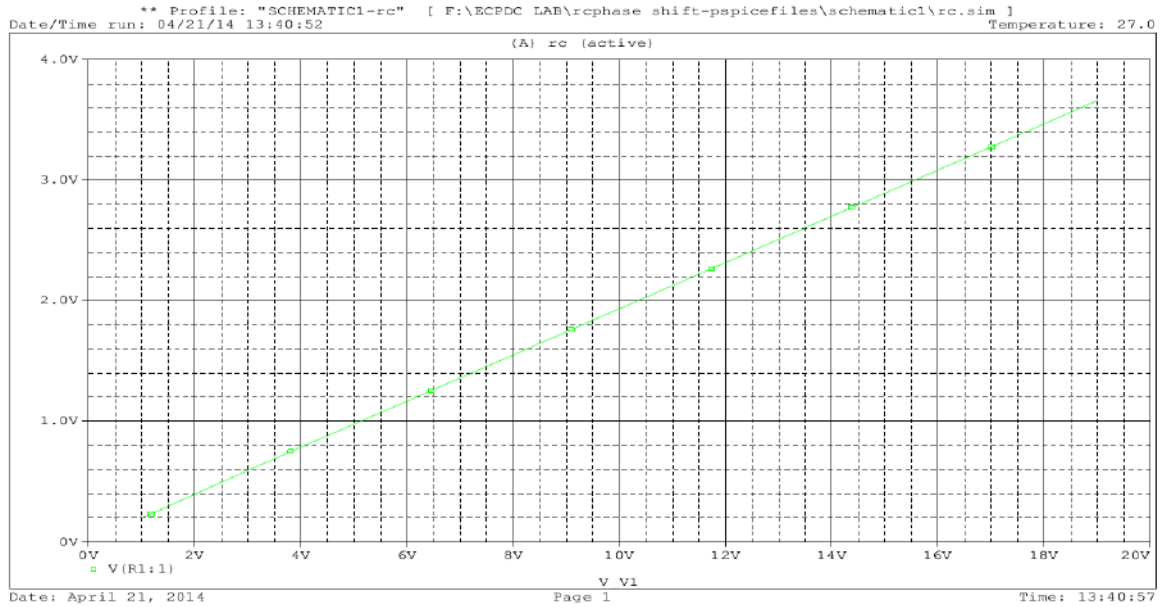
TRANSIENT ANALYSIS:
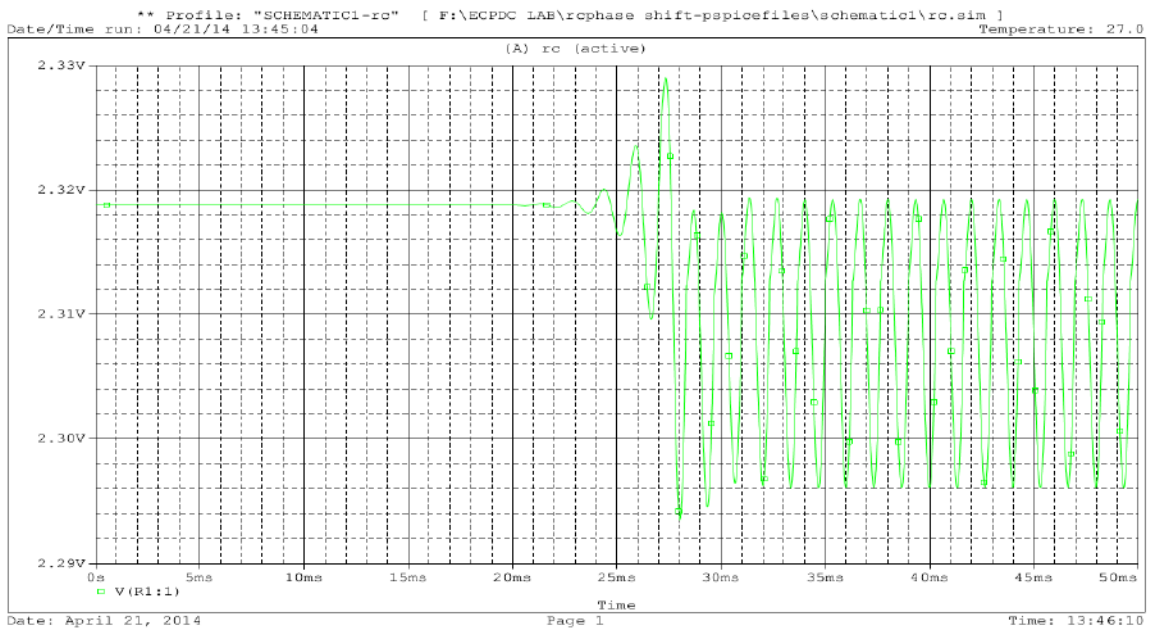
RESULT:

VIVA QUESTIONS:

1. What are the conditions of oscillations?
2. Give the formula for frequency of oscillations?
3. What is the total phase shift produce by the RC ladder network?
4. Whether the oscillator is positive feedback or negative feedback?
5. What are the types of oscillators?
6. What is the gain of RC phase shift oscillator?
7. What is the difference between damped oscillations undamped oscillations?
8. What are the applications of RC oscillations?
9. How many resistors and capacitors are used in RC phase shift network
10. How the Barkhausen criterion is satisfied in RC phase shift oscillator

3. CLASS B COMPLEMENTARY SYMMETRY AMPLIFIER

AIM: To observe the input and output waveforms and to calculate the efficiency of Class B Complimentary symmetry power amplifier.

EQUIPMENT REQUIRED: PSPICE Simulation Software

CIRCUIT DIAGRAM:

THEORY:

An amplifying system consists of several stages in cascade. The input and the intermediate stages amplify small signal excitations to a value large enough to drive the final device .The out put stage feeds the final device .The output stage feeds a transducer such as a CRO,loudspeaker or servomotor. Thus the final stage must be capable of delivering a large voltage or current or appreciable amount of power. This requires an amplifier which is referred as a power amplifier

In class B complimentary symmetry class _B amplifier one n-p-n and p-n-p is used. Hence the circuit is called class-B complimentary symmetry amplifier. This circuit ifis transformer less circuit .But with common emitter configuration it becomes power transfer without output impedance for maximum power transfer without an output transformers. Hence the matched pair of complementary transistors are used in common collector configuration This is because in common collector configuration has lowest output impedance and hence the impedance matching is possible.

PROCEDURE:

1. Enter in to the Pspice software.
2. Customize the screen and then draw the circuit on the screen .
3. Start the simulation and observe the input and output waveforms.
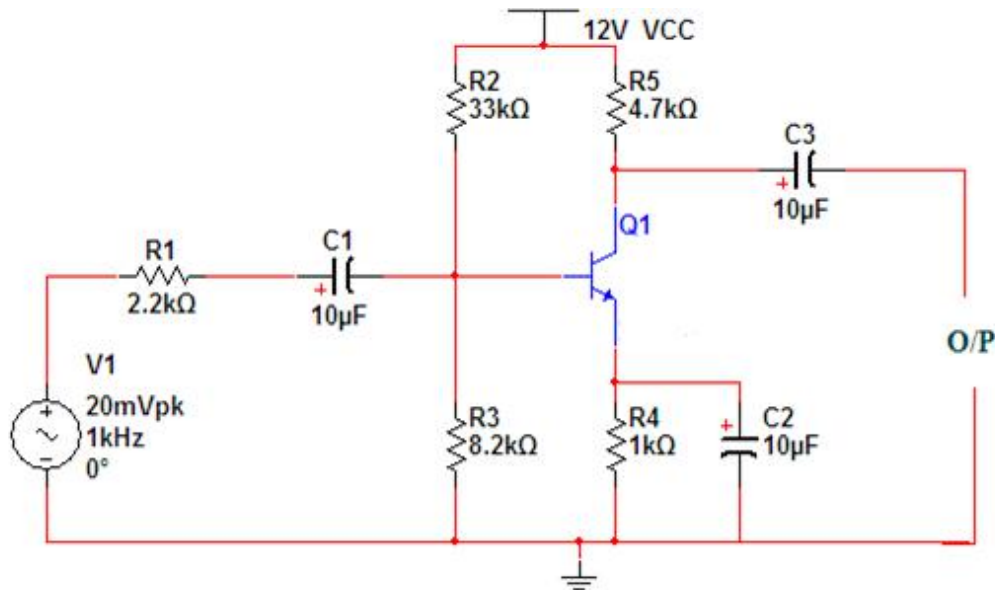
MODEL GRAPHS:
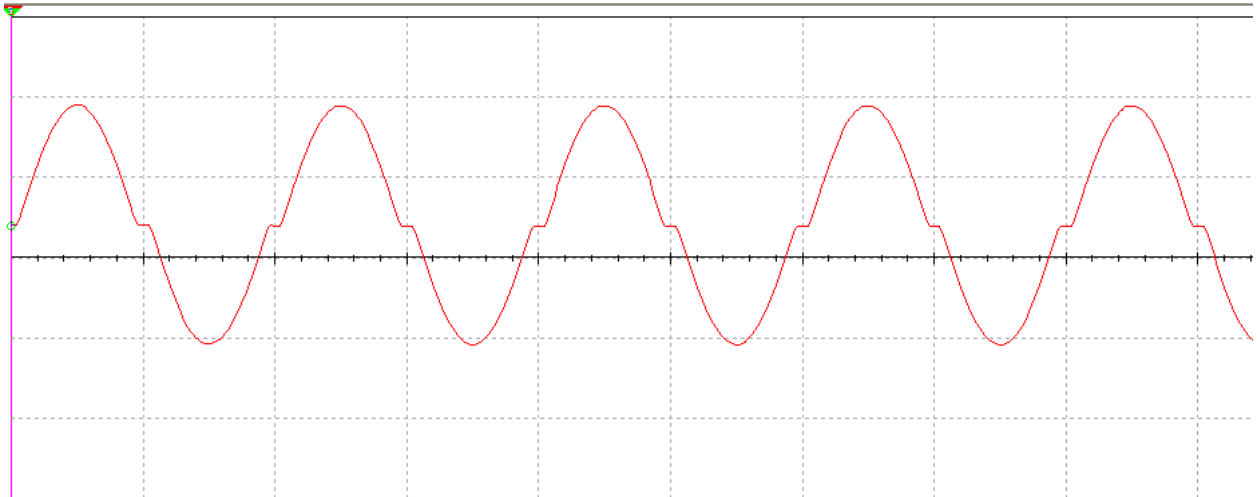
RESULT:.

VIVA QUESTIONS

1. What is the efficiency of Class B complimentary symmetry power amplifier?
2. What are the advantages of Class B complimentary symmetry power amplifier?
3. What are the disadvantages in Class B push pull power amplifier?

4. WEIN BRIDGE OSCILLATOR

AIM: To study and calculate frequency of Wein Bridge Oscillator.

APPARATUS:

Pspice software

CIRCUITDIAGRAM:
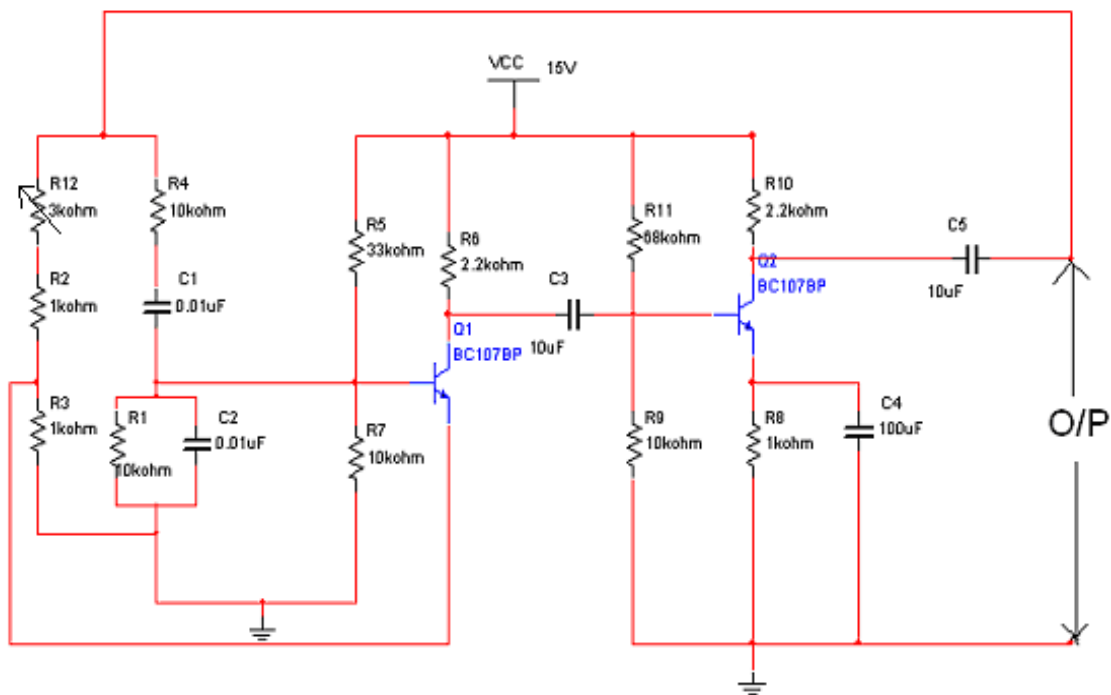
THEORY:

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 +ve 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 +ve feedback network shown in fig makes the input & output in the phase. The frequency of oscillations is given as

$$f = 1/2\pi\sqrt{R_1 C_1 R_2 C_2}$$

In addition to the positive feedback

PROCEDURE:

1. Enter in to the Pspice software.
2. Customize the screen and then draw the circuit on the screen .
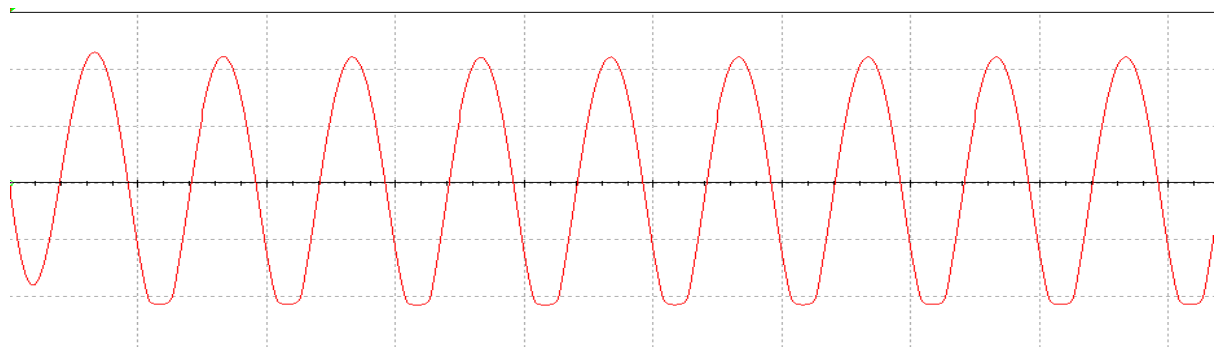3. Start the simulation and observe the input and output waveforms.

**MODEL WAVE FORM:**
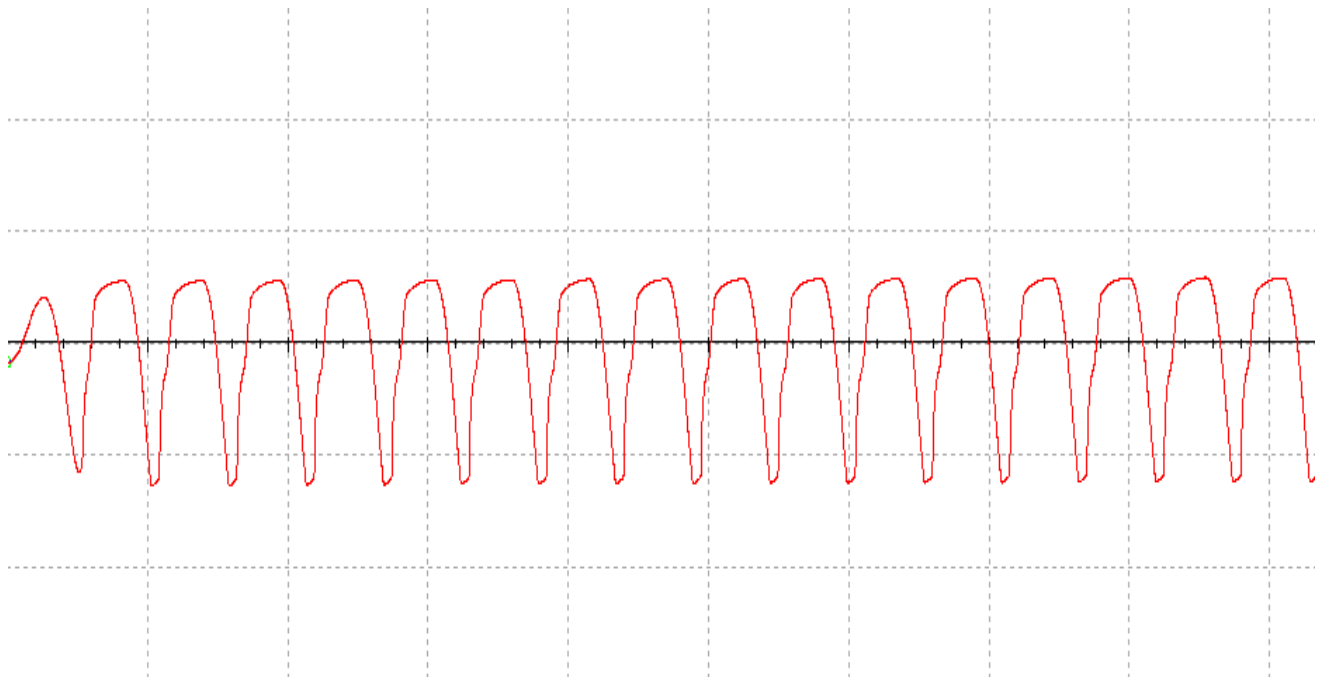
RESULT:

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 undamped   Oscillations?

5. CLASSA POWER AMPLIFIER

AIM: To observe the input and output waveforms and to calculate the efficiency of

CLASS A Power Amplifier.

EQUIPMENT REQUIRED: PSPICE Simulation Software

CIRCUIT DIAGRAM:

THEORY:

         The amplifier is said to be class A power amplifier if the q point and the input signal are selected such that the output signal is obtained for a full input cycle . For this class the position of q point is approximately y at the mid point of the load line. For all the values of input signal the transistor remains in the active region and never entire into the cutoff or saturation region. The collector current flows for 3600 (life cycle) of the input signal in other words the angle of the collector current flow is 3600 the claa a amplifiers or furthers classified as directly coupled and transformer coupled and transformer coupled amplifiers in directly coupled type .The load is directly connected in the collector circuit while in the transformer coupled type, the load is coupled to the collector using the transformer.

Advantages:

1. Distortion analysis is very important

2. It amplifies audio frequency signals faithfully hence they are called as audio amplifiers

Disadvantages:

1. H parameter analysis is not applicable

---

2. Due to large power handling the transistor is used power transistor which is large in size and having large power rating

PROCEDURE:

1. Enter in to the Pspice software.
2. Customize the screen and then draw the circuit on the screen .
3. Start the simulation and observe the input and output waveforms.

GRAPH:

I/P

O/P:

FREQUENCY RESPONSE:
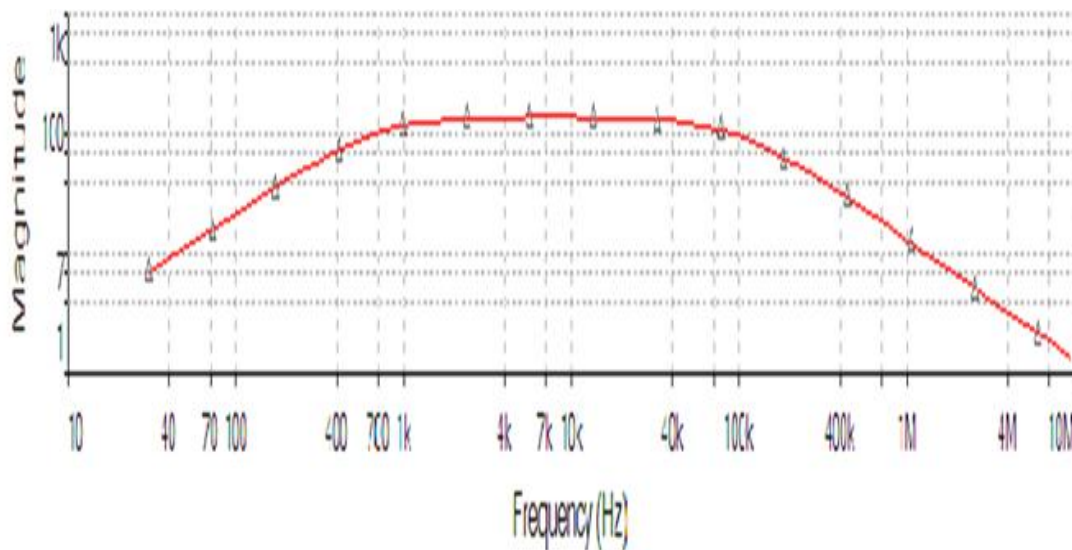
RESULT:.

VIVA QUESTIONS:

1.On which criteria based the power amplifiers are classified?

2.what is the operating point of class-A power ampliier?

3.what is meant by efficiency?

4.howmuch efficiency will b obtained in classs-A power amplifier?

5. what are the applications of power amplifier?

6.SINGLE TUNED VOLTAGE AMPLIFIER

AIM: To study single tuned voltage Amplifier and to calculate
1. Resonant Frequency.
2. Q factor.
3. Bandwidth and
4. Impedance

APPARATUS: **Pspice software**

CIRCUIT DIAGRAM:

THEORY:

It is usually required to use a number of tuned amplifier stages in cascade in order to obtain large overall gain. These cascade tuned amplifiers may be put into the following three categories:

1. Single tuned amplifiers.

2. Double tuned amplifiers.

3. Stagger-tuned amplifiers.

Single tuned amplifiers use one parallel tuned circuit as the load impedance in each stage and all these tuned circuits in different stages are tuned to the same frequency.Double tuned amplifier uses two inductively coupled tuned circuits per stage, both the tuned circuits being tuned to the same frequency.Staggered tuned amplifier uses a number of single tuned stages in cascade, the successive tuned circuits being tuned to slightly different frequencies.
Single tuned amplifiers may again be put into following two categories:

a) Capacitance coupled single tuned amplifiers and

b) Transformer coupled or inductively coupled single tuned amplifiers

RESONANT FREQUENCY:

Depending upon the frequency of the source voltage Vs, the circuits may behave either as inductive or capacitive. However, at a particular frequency when the inductive reactance Xl equals the capacitive reactance Xc, then the circuit behaves as a purely resistive circuit. This

phenomenon is called resonance: and the corresponding frequency is called resonant frequency. The resonant frequency (fr) can be found by equating the two reactance values.

PROCEDURE:

1. Enter in to the Pspice software.
2. Customize the screen and then draw the circuit on the screen .
3. Start the simulation and observe the input and output waveforms.

GRAPH:

RESULT:.

VIVA QUESTIONS:
1. What are tuned amplifiers?
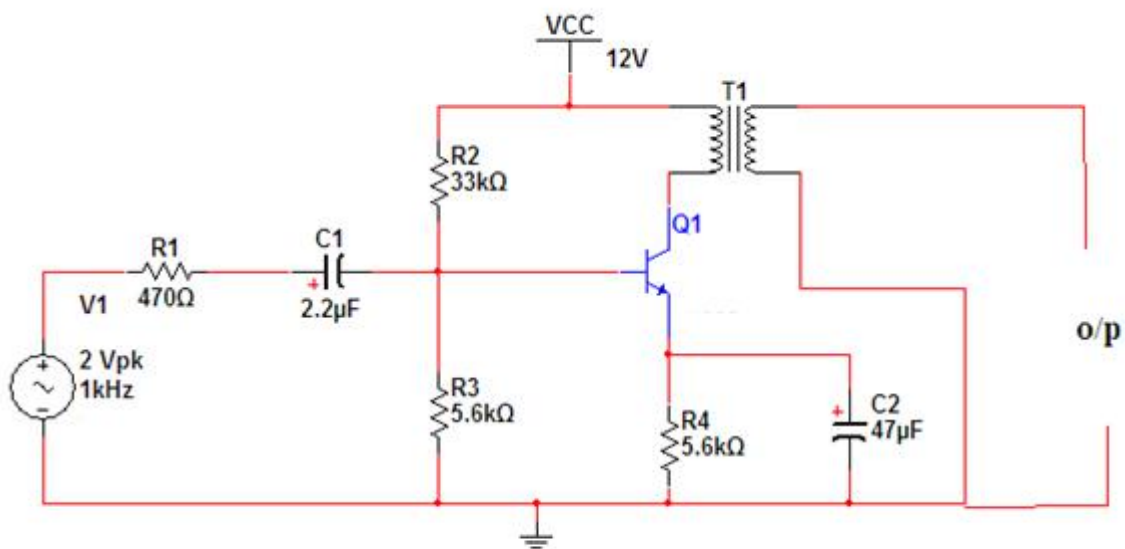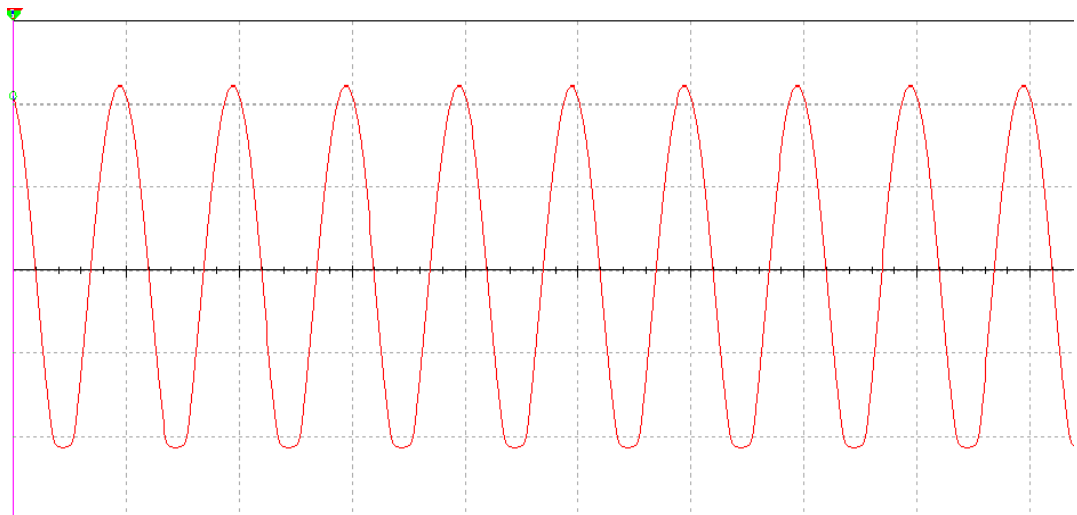2. What is tuning?
3. Explain how the tuned amplifiers work?

8. CLASS -A POWER AMPLIFIER(TRANSFORMER COUPLED)

AIM: **To observe the input and output waveforms and to calculate the efficiency of CLASS A Power Amplifier.**

EQUIPMENT REQUIRED: **Pspice software**

CIRCUIT DIAGRAM:

THEORY:

The amplifier is said to be class A power amplifier if the q point and the input signal are selected such that the output signal is obtained for a full input cycle . For this class the position of q point is approximately y at the mid point of the load line. For all the values of input signal the transistor remains in the active region and never entire into the cutoff or saturation region. The collector current flows for 3600 (life cycle) of the input signal in other words the angle of the collector current flow is 3600 the claa a amplifiers or furthers classified as directly coupled and transformer coupled and transformer coupled amplifiers in directly coupled type .The load is directly connected in the collector circuit while in the transformer coupled type, the load is coupled to the collector using the transformer.

Advantages:

1. Distortion analysis is very important

2. It amplifies audio frequency signals faithfully hence they are called as audio amplifiers

Disadvantages:

---

1. H parameter analysis is not applicable

2. Due to large power handling the transistor is used power transistor which is large in size and having large power rating

PROCEDURE:

1. Enter in to the Pspice software.
2. Customize the screen and then draw the circuit on the screen .
3. Start the simulation and observe the input and output waveforms.

GRAPH:

I/P

O/P:

RESULT:

VIVA QUESTIONS:

1.On which criteria based the power amplifiers are classified?

2.what is the operating point of class-A power ampliier?

3.what is meant by efficiency?

4.how much efficiency will b obtained in classs-A power amplifier?

5. what are the applications of power amplifier?

8.VOLTAGE-SERIES FEEDBACK AMPLIFIER

AIM: To study the effect of voltage series feedback and Gain of the Amplifier.

APPARATUS: Pspice Software

CIRCUIT DIAGRAM:

THEORY:

When any increase in the output signal results into the input in such a way as to cause the decrease in the output signal, the amplifier is said to have negative feedback. The advantages of providing negative feedback are that the transfer gain of the amplifier with feedback can be stablised against varations in the hybrid parameteresof the transistor or the parameters of the other active devices used in the circuit. The most advantage of the negative feedback is that by propere use of this , there is significant improvement in the frequency respponse and in the linearity of the operation of the amplifier.This disadvantage of the negative feedback is that the voltage gain is decreased.

In Voltage-Series feedback , the input impedance of the amplifier is decreased and the output impedance is increased.Noise and distortionsare reduced cosiderably.

PROCEDURE:

1. Enter in to the Pspice software.
2. Customize the screen and then draw the circuit on the screen .
3. Start the simulation and observe the input and output waveforms.

GRAPH:

o/p:
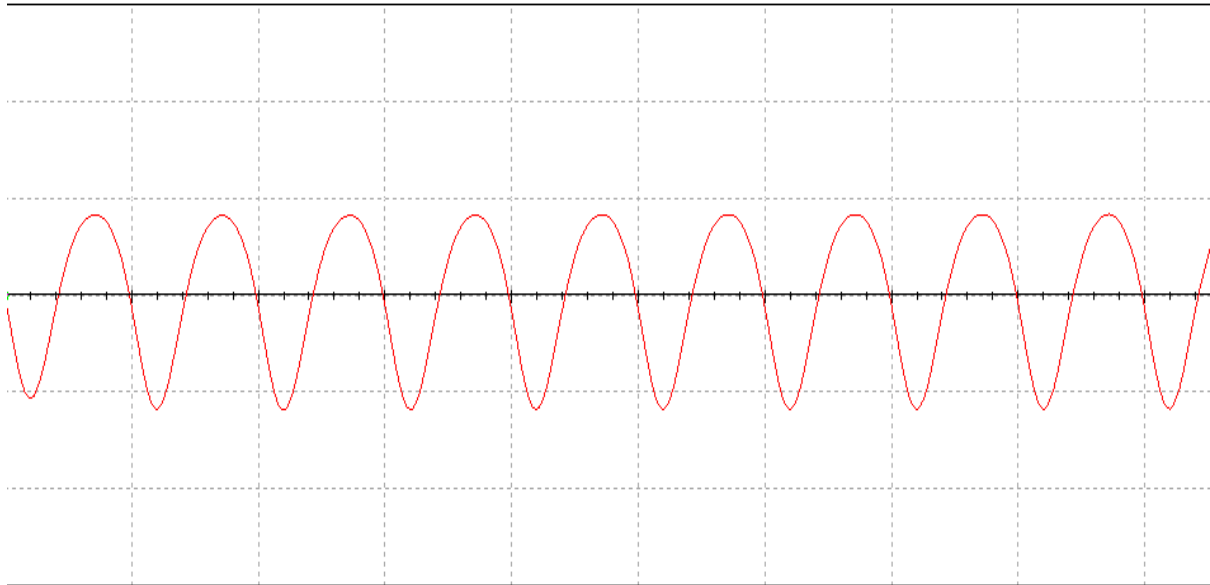
Frequency response:
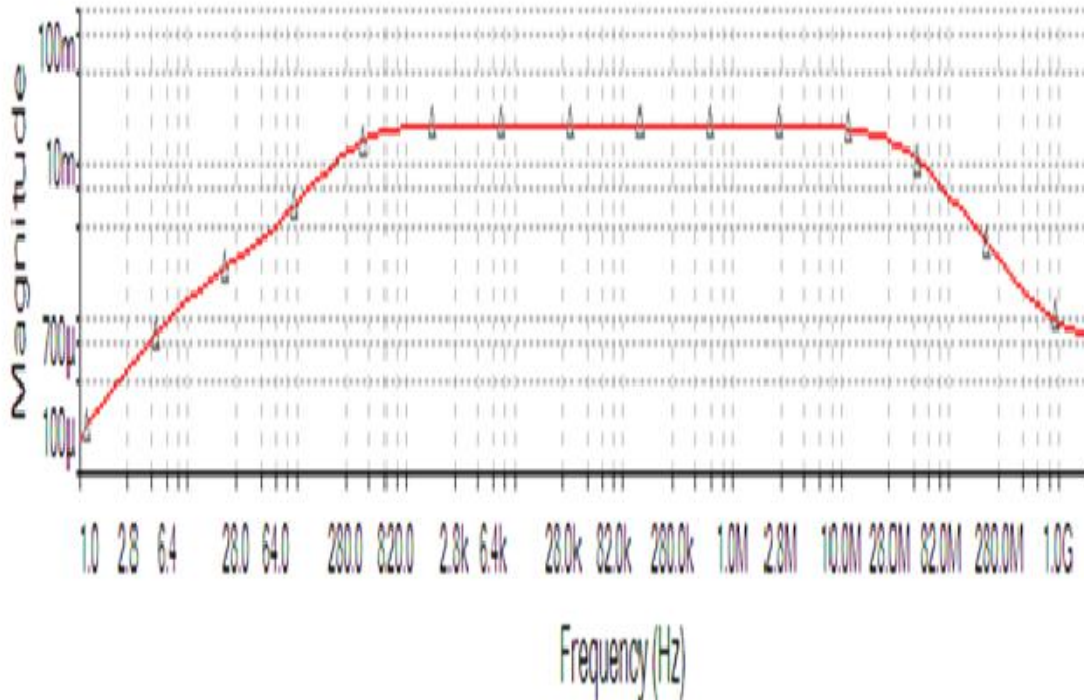
Result:

Viva voice:

1. what is meant by feed back?

2. what are the types of feed back?

3. why we are using negative feed back in feedback amplifiers?

4.what are the different types of feedback amplifiers?
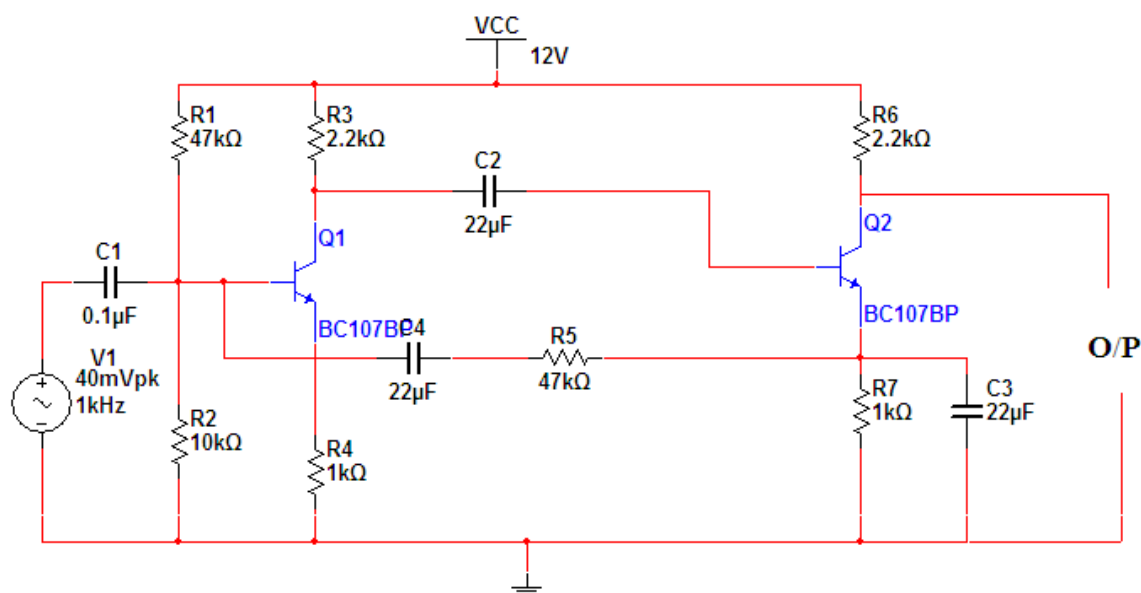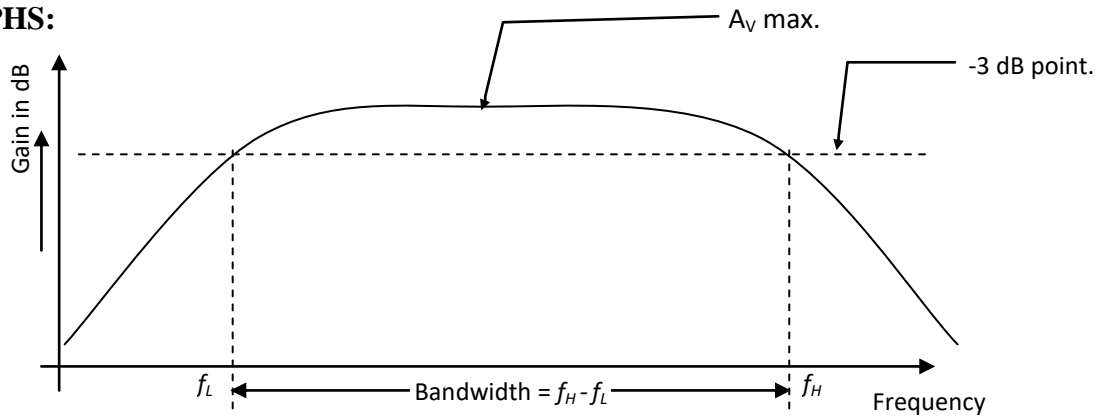
10. CURRENT SHUNT FEEDBACK AMPLIFIER

AIM: To observe the performance of a current shunt feedback amplifier and obtain its bandwidth.

EQUIPMENT RQUIRED: Pspice Software

THEORY:

Current shunt feedback circuit shows two transistor in cascade with feedback from the second emitter to the first base through the resistor R_F. we verify that this connection produces negative feedback. The voltage V_{i2} is much larger than V_{i1} because of the voltage gain of Q_1. Also V_{i2} is 180^0 out of phase with V_{i1}. Because of emitter follower action V_{e2} is only slightly smaller than V_{i2}, and these voltages are in phase. Hence V_{e2} is larger in magnitude than V_{i1} and is 180^0 out of phase with V_{i1}. If the input signal increases so that I_S increases, If also increases, and $I_i = I_S - I_f$ is smaller than it would be their were no feedback. This action is characteristics of negative feedback.

CIRCUIT DIAGRAM:

PROCEDURE:

1. Enter in to the Pspice software.
2. Customize the screen and then draw the circuit on the screen .
3. Start the simulation and observe the input and output waveforms.

GRAPHS:

RESULT:

CONCLUSIONS:

       Conclusions can be drawn on comparing the voltage gain of the amplifier with and without feedback, bandwidth obtained from the frequency response plot.
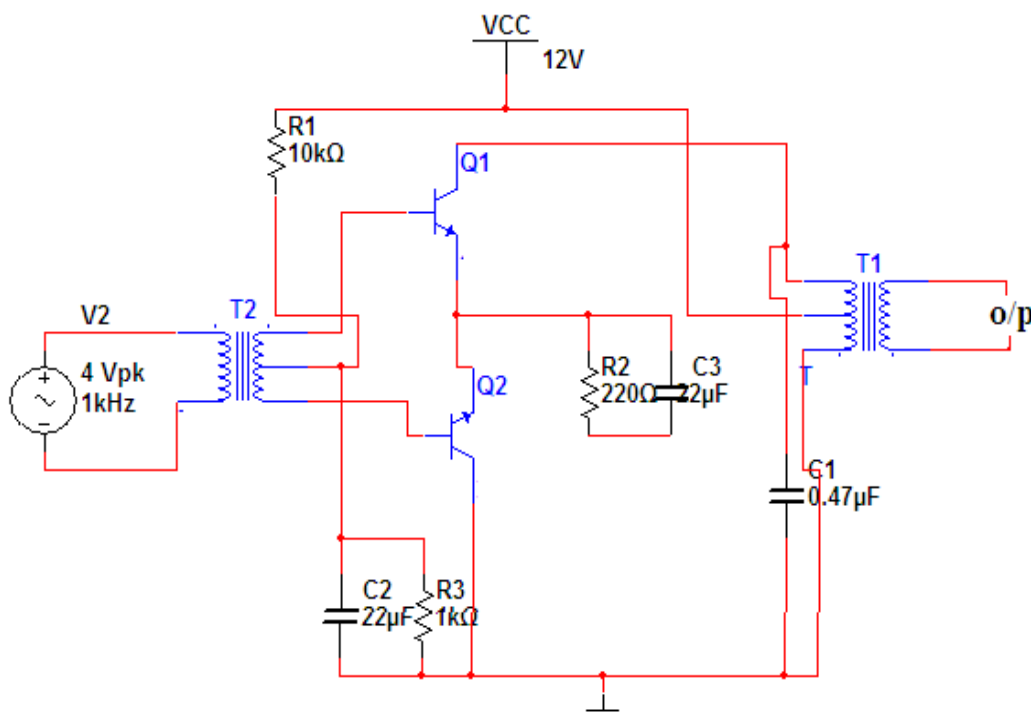
VIVA QUESTIONS:

5. What is the relationship between the transfer gain with feedback A_f and that without feedback A.
6. Define negative feedback.
7. Define the amount of feedback in decibels.
8. State the three fundamental assumptions which are made in order that the expression $A_f = A/(1+A\beta)$ be satisfy exactly.

10.CLASS B PUSH PULL POWER AMPLIFIER

AIM: To observe the input and output waveforms and to calculate the efficiency of Class B Complimentary symmetry power amplifier.

EQUIPMENT REQUIRED: Pspice software

CIRCUIT DIAGRAM:

THEORY:
An amplifying system consists of several stages in cascade. The input and the intermediate stages amplify small signal excitations to a value large enough to drive the final device .The out put stage feeds the final device .The output stage feeds a transducer such as a CRO,loudspeaker or servomotor. Thus the final stage must be capable of delivering a large voltage or current or appreciable amount of power. This requires an amplifier which is referred as a power amplifier In class B complimentary symmetry class _B amplifier one n-p-n and p-n-p is used. Hence the circuit is called class-B complimentary symmetry amplifier. This circuit ifis transformer less circuit .But with common emitter configuration it becomes power transfer without output impedance for maximum power transfer without an output transformers. Hence the matched pair of complementary transistors are used in common collector configuration This is because in common collector configuration has lowest output impedance and hence the impedance matching is possible.

 PROCEDURE:
1.conncet the circuit s per the circuit diagram
2.apply 4v p-p with 1KHZ frequency using function generator
3.observe the output in CRO .
4.note the cross over distortion in output.(outputVp-p)

5.remove the collector connection and put ammeter.

6.note the Idc value in the ammeter.

7.using Pdc and Pac formulas find the efficiency.

 GRAPHS:

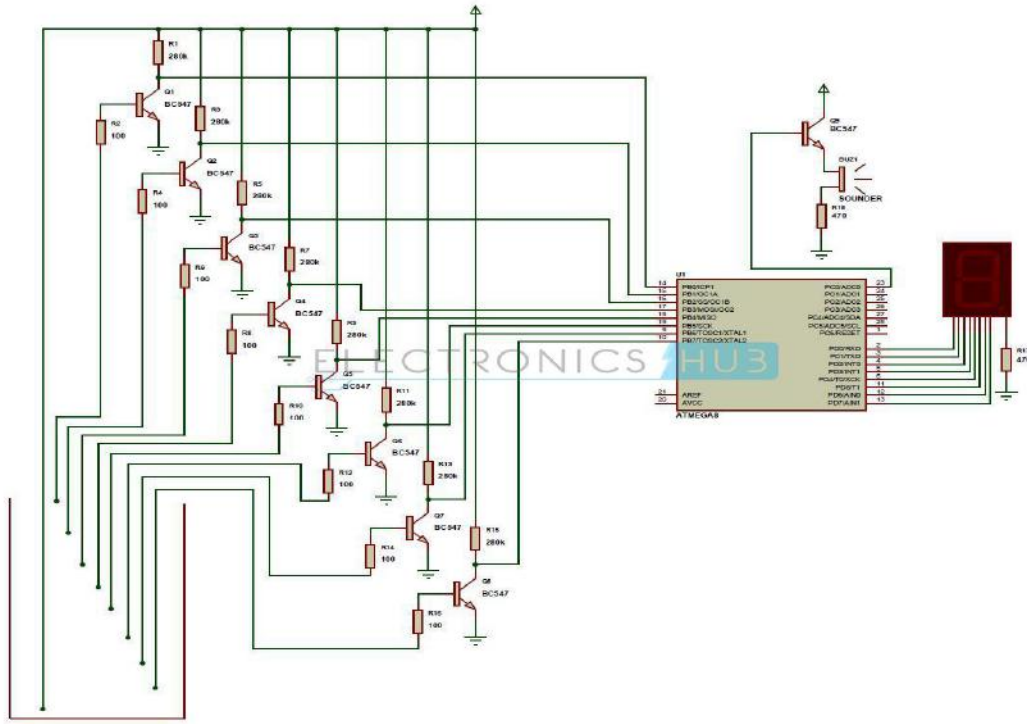## RESULT:

VIVA QUESTIONS:

1.In classB power amplifiers why crossover distortion will be occurred?

2.what are he different methods to eliminate distortion?

3..howmuch efficiency will b obtained in classs-B power amplifier?

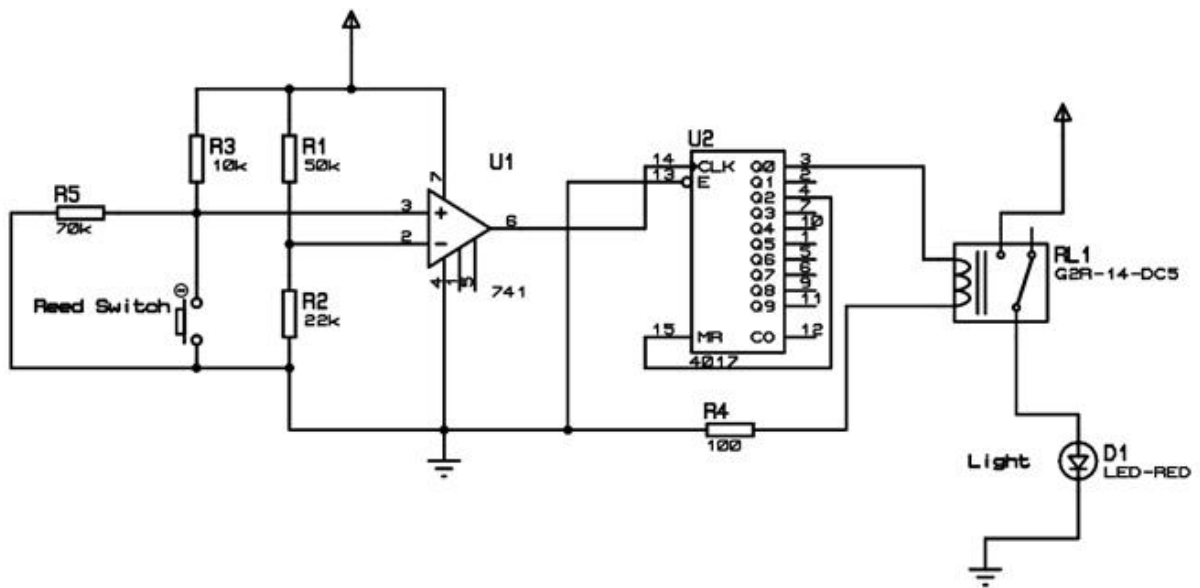4.where is the location of operating point in class-B power amplifier?

PROJECTS RELATED TO LAB

Water Level Indicator Circuit Diagram:

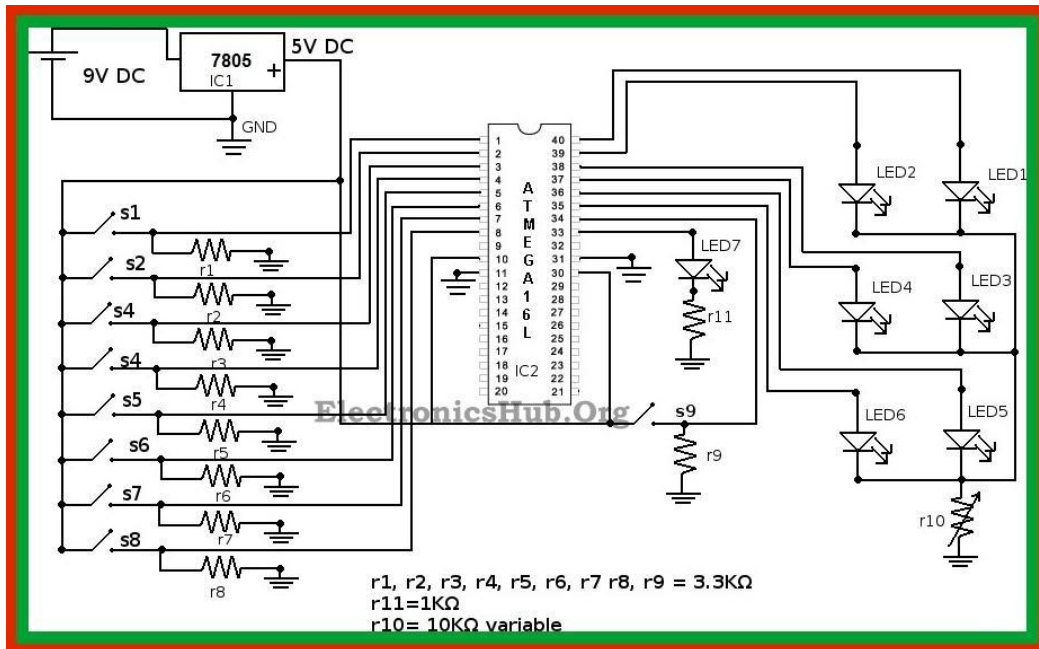Water Level Indicator Project Block Diagram:
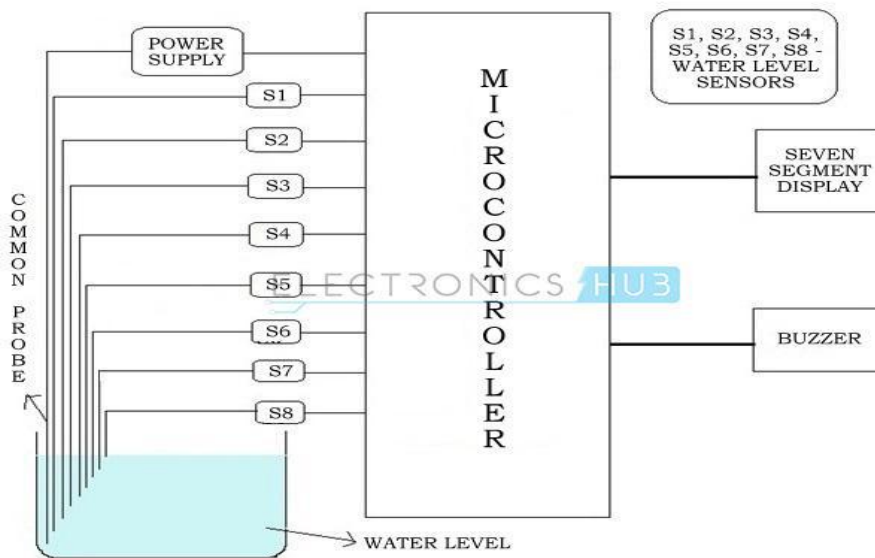
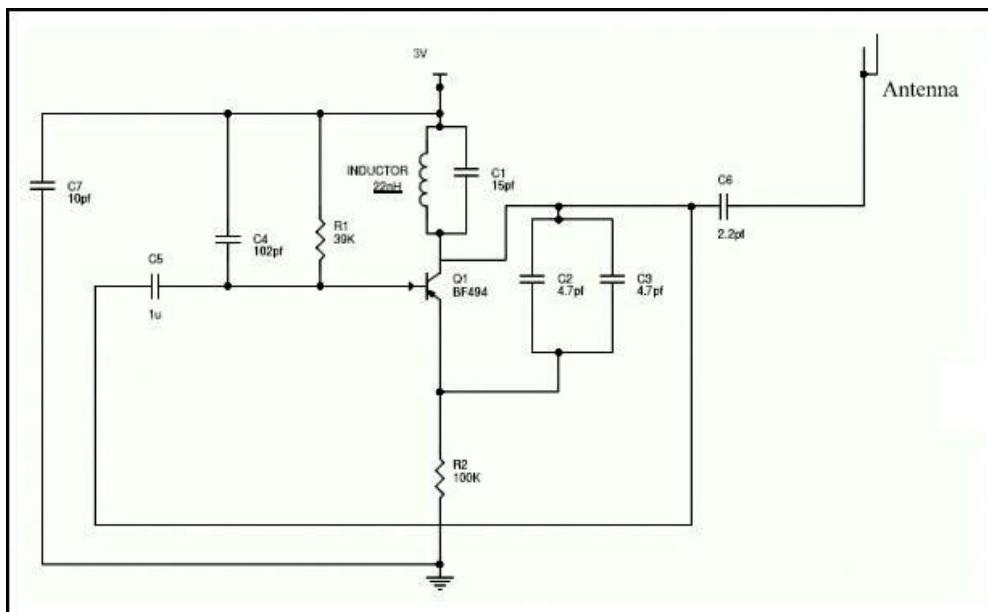Automatic Room Light Switch Circuit Diagram:

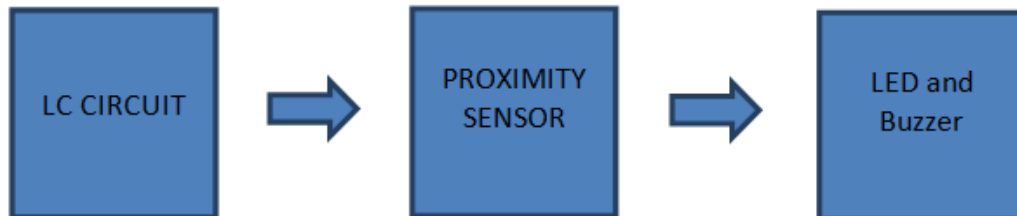Block Diagram of Boolean Algebra Calculator:
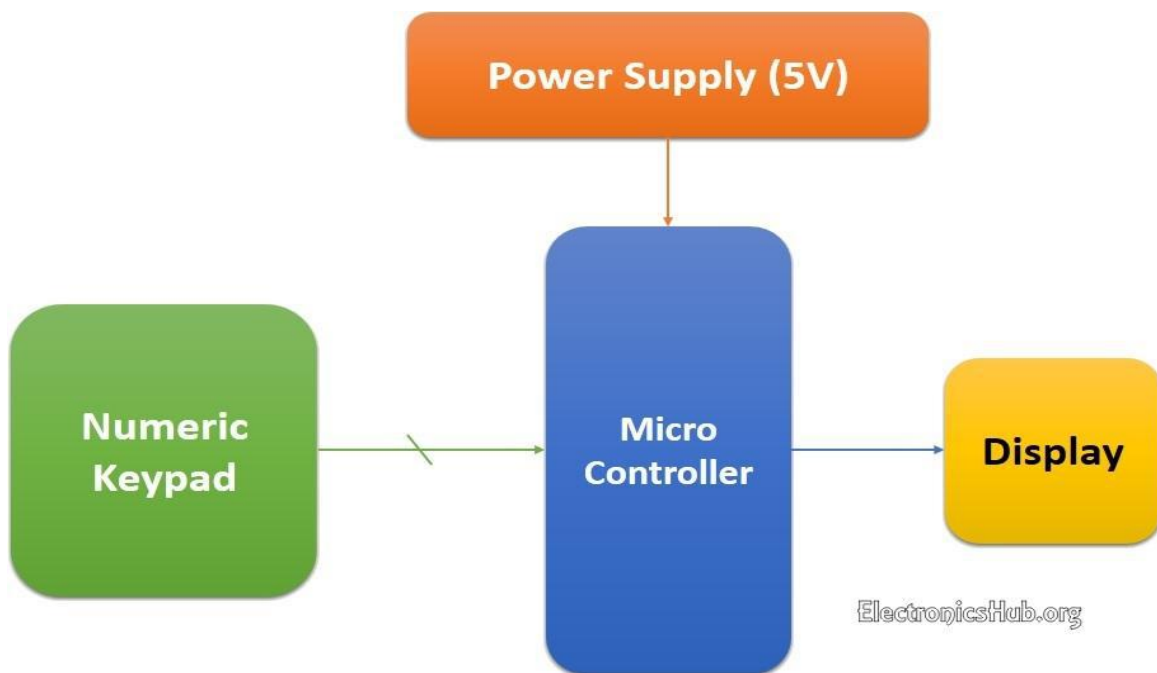
Boolean Algebra Calculator Circuit Diagram:

Simple Mobile Jammer Circuit Diagram:

Block Diagram of Metal Detector:

Circuit Diagram of Metal Detector: