CHENDU

COLLEGE OF ENGINEERING AND TECHNOLOGY

MADURANTAKAM.

KANCHEEPURAM-603311.

EC 6411

CIRCUITS AND SIMULATION INTEGRATED LABORATORY

LABORATORY MANUAL

Prepared by

C.KRITHIKA, ASSISTANT PROFESSOR

Faculty of Electronics & Communication Engineering

INDEX

HALF WAVE AND FULL WAVE RECTIFIER

EXP.NO : 1 **DATE:**

HALF WAVE RECTIFIER:

AIM:

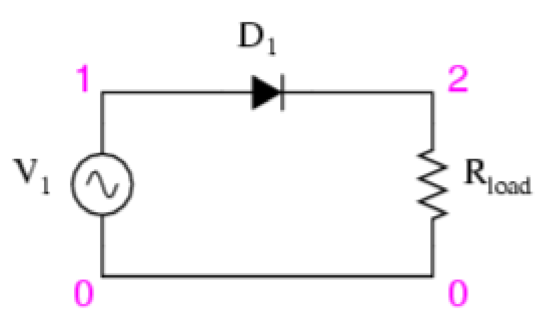
To construct a half wave rectifier with simple capacitor filter and to measure its ripple factor from the output waveforms.

APPARATUS REQUIRED:

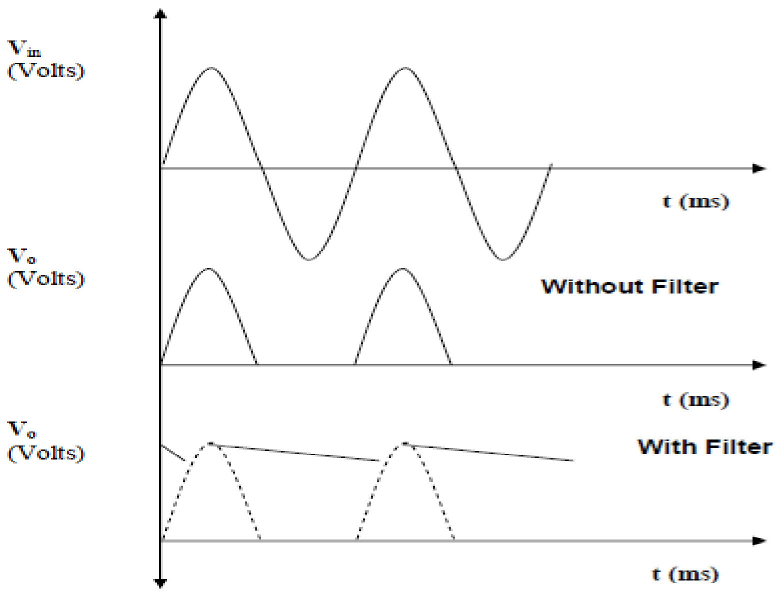
S NO	APPARATUS	RANGE	QUANTITY
1	RESISTOR	1K	1
2	CAPACITOR	470 μ f	1
3	DIODE	IN4007	1
4	STEP DOWN TRANSFORMER		1
5	BREAD BOARD		1
6	CONNECTING WIRES		FEW
7	CRO		1

CIRCUIT DIAGRAM:

Half wave rectifier with filter



MODEL GRAPH:



THEORY:

A Half wave rectifier is a device which converts ac voltage to pulsating dcvoltage using one PN junction diode.The ac voltage (230 V, 50 HZ) is connected to the primary of the transformer. Thetransformer steps down the ac voltage. Thus, with suitable turns ratio we get desired ac secondary voltage. The rectifier circuit converts this ac voltage in to a pulsating dcvoltage. Half wave rectifier conducts during positive half cycle and gives output in the form of positive sinusoidal pulses. Hence the output is called pulsating dc. A pulsating dc voltage containing large varying component called ripple in it. The capacitor filter is used after rectifier circuit, which reduces the ripple content in the pulsating dc. Thus filter converts pulsating dc in to pure dc.

Ripple Factor:
The output of the rectifier is of pulsating dc type. The amount of ac content in theoutput can be mathematically expressed by a factor called ripple factor.

TABULATION:

Parameter	Output with simple C filter
Vm (V)	
Vrms(V)	

CALCULATION:

Ripple Factor = R.M.S Value of ac component

~ Average dc component = Vrms/Vdc) 2

Where, Vrms = Vm/2 ; Vdc = Vm/Π

PROCEDURE:

1. The circuit connections are made as per the circuit diagram.
2. First without connecting the capacitive filter, note down the amplitude and time period of the rectified waveform.
3. Now connect the capacitive filter and note down the amplitude and time period of the rectified waveform.
4. Connect the CRO across the load and measure the full load voltage then remove The load and measure the no load voltage.
5. Plot the graph and calculate the efficiency.

RESULT:

Thus the half wave rectifier was constructed and input, output waveforms were drawn.

Ripple Factor r =

FULL WAVE RECTIFIER

AIM:

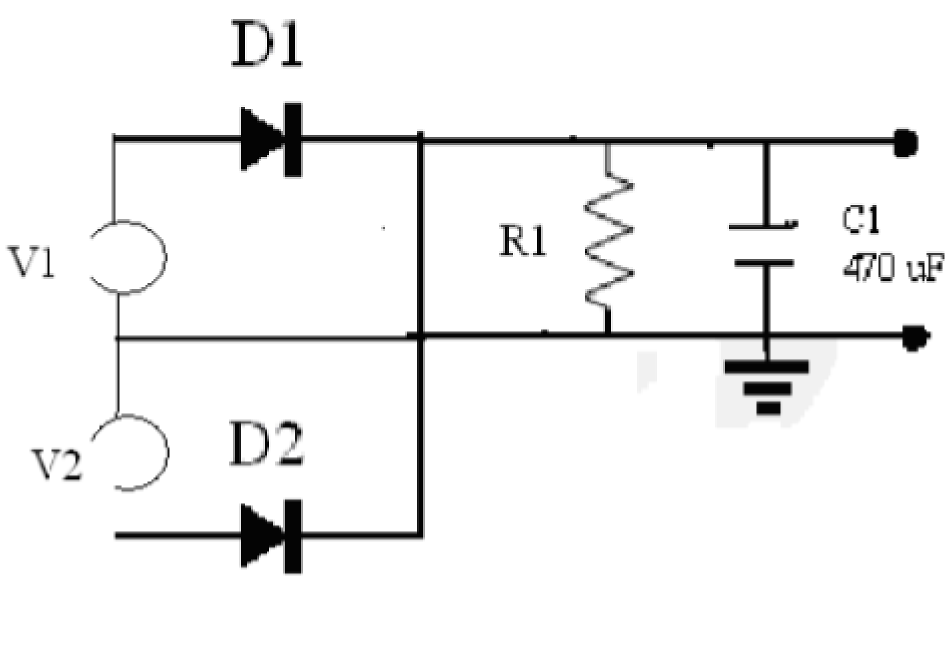
To construct a full wave rectifier with simple capacitor filter and to measure its ripple factor from the output waveforms

APPARATUS REQUIRED:

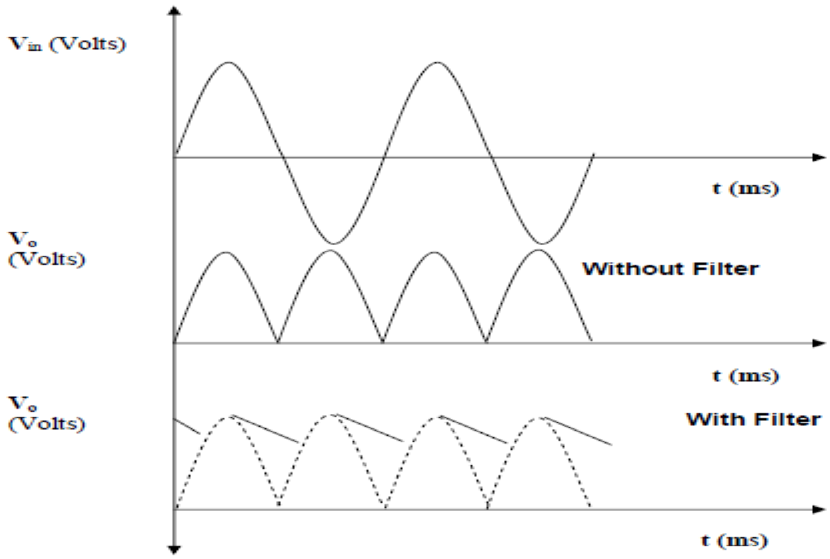
S NO	APPARATUS	RANGE	QUANTITY
1	RESISTOR	1K	2
2	CAPACITOR	100 µf	1
3	DIODE	IN4007	2
4	STEP DOWN TRANSFORMER		1
5	BREAD BOARD		1
6	CONNECTING WIRES		FEW
7	CRO		1

CIRCUIT DIAGRAM:

Full wave rectifier with filter:



MODEL GRAPH:



THEORY:

A full wave rectifier is a device which converts ac voltage to pulsating dc voltage using two PN junction diode.The ac voltage (230 V, 50 HZ) is connected to the primary of the transformer.The transformer steps down the ac voltage. Thus, with suitable turns ratio we get desiredac secondary voltage. The rectifier circuit converts this ac voltage in to a pulsating dc voltage. Full wave rectifier conducts during both positive and negative half cycle of input ac supply. Because two diodes are used in this cir circuit .It gives output in the form of positive sinusoidal pulses. Hence the output is called pulsating dc. A pulsating dc voltage containing large varying component called ripple in it.The capacitor filter is used after rectifier circuit, which reduces the ripplecontent in the pulsating dc. Thus filter converts pulsating dc in to pure dc.

TABULATION:

Parameter	Output with simple C filter
Vm (V)	
Vrms(V)	

CALCULATION:

Ripple Factor:

The output of the rectifier is of pulsating dc type. The amount of ac content in the output can be mathematically expressed by a factor called ripple factor.

Ripple Factor = R.M.S Value of ac component / Average dc component = Vrms/Vdc) 2

Where, Vrms = Vm/√2 ; Vdc = 2Vm/Π

PROCEDURE:

1. The circuit connections are made as per the circuit diagram.
2. First without connecting the capacitive filter, note down the amplitude and time period of the rectified waveform.
3. Now connect the capacitive filter and note down the amplitude and time period of the rectified waveform.
4. Connect the CRO across the load and measure the full load voltage then remove theload and measure the no load voltage.
5. Plot the graph and calculate the efficiency.

RESULT:

Thus the full wave rectifier was constructed and input, output waveforms were drawn.

Ripple Factor with filter =

FIXED BIAS AMPLIFIER CIRCUIT USING BJT

EXP.NO : 2 **DATE:**

AIM:

To construct a fixed bias amplifier circuit and to plot the frequency response characteristics.

APPARATUS REQUIRED:

S.No.	Name	Range	Quantity
1.	Transistor	BC107	1
2.	Resistor	10 kΩ,100 kΩ,680 Ω	1,1,1
3.	Regulated power supply	(0-30)V	1
4.	Signal Generator	(0-3)MHz	1
5.	CRO	30 MHz	1
6.	Bread Board		1
7.	Capacitor	47µ F	2

CIRCUIT DIAGRAM :

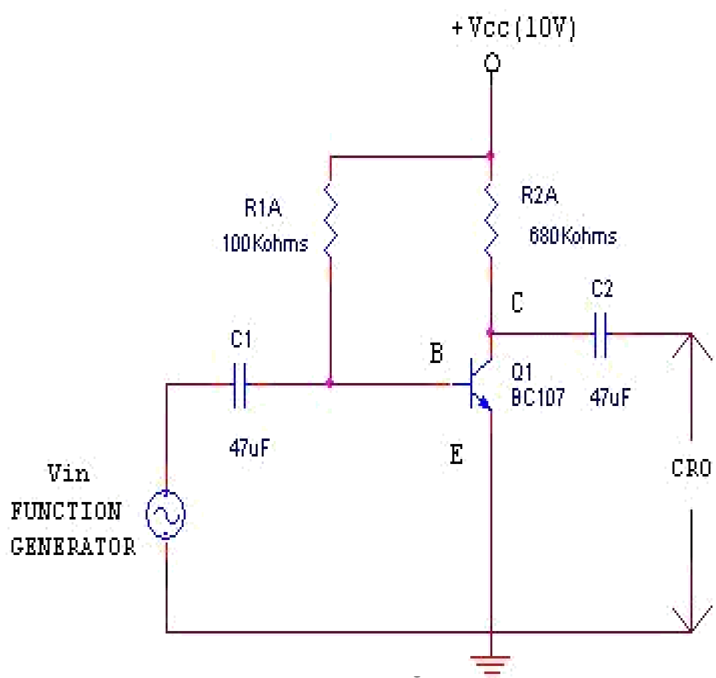
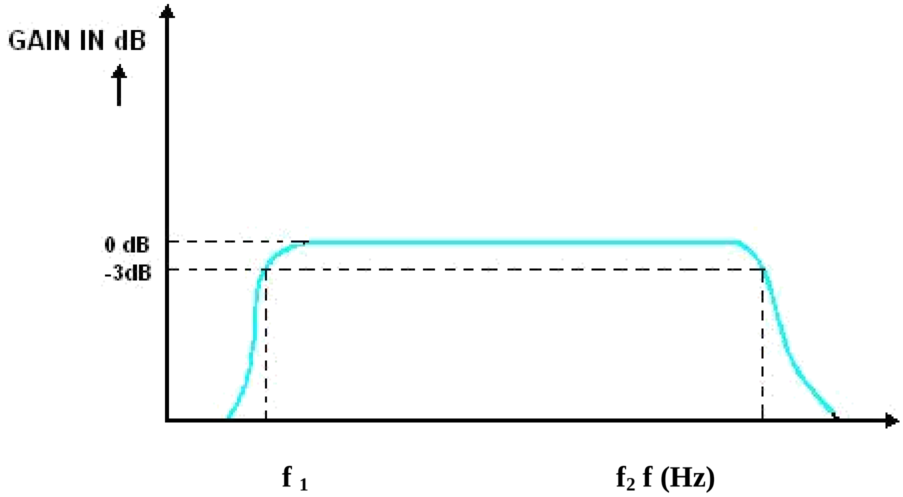


FIG.9.1

MODEL GRAPH :



FREQUENCY RESPONSE OF FIXED BIAS AMPLIFIER:

Keep the input voltage constant (V_{in}) =

Frequency (in Hz)	Output Voltage (in volts)	Gain = 20 log (V_o / V_{in}) (in dB)

FORMULA:

a) $\mathbf{R_2 / (R_1+R_2)}$ = voltage at which Class A, Class B or Class C operation takes place

b) $h_{fe} = I_c / I_b$

THEORY:

In order to operate the transistor in the desired region, we have to apply an external dc voltage of correct polarity and magnitude to the two junctions of the transistor. This is called biasing of the transistor.

When we bias a transistor, we establish certain current and voltage conditions for the transistor. These conditions are called operating conditions or dc operating point or quiescent point. This point must be stable for proper operation of transistor. An important and common type of biasing is called Fixed Biasing. The circuit is very simple and uses only few components. But the circuit does not check the collector current which increases with the rise in temperature.

PROCEDURE :

- Connections are made as per the circuit diagram.
- The waveforms at the input and output are observed for Class A, Class B and Class C operations by varying the input voltages.
- The biasing resistances needed to locate the Q-point are determined.
- Set the input voltage as 1V and by varying the frequency, note the output voltage.
- Calculate gain=20 log (V_o / V_{in})
- A graph is plotted between frequency and gain.

CALCULATIONS:

a) To determine the value of bias resistace $R_2 / (R_1+ R_2)$

b) $h_{fe} = \Delta IC/\Delta IB$

RESULT:

Thus, the Fixed bias amplifier was constructed and the frequency response curve is plotted. ThGain Bandwidth Product is found to be = ______________ .

BJT AMPLIFIER USING VOLTAGE DIVIDER BIAS
CE-CONFIGURATION

EXP.NO : 3 **DATE:**

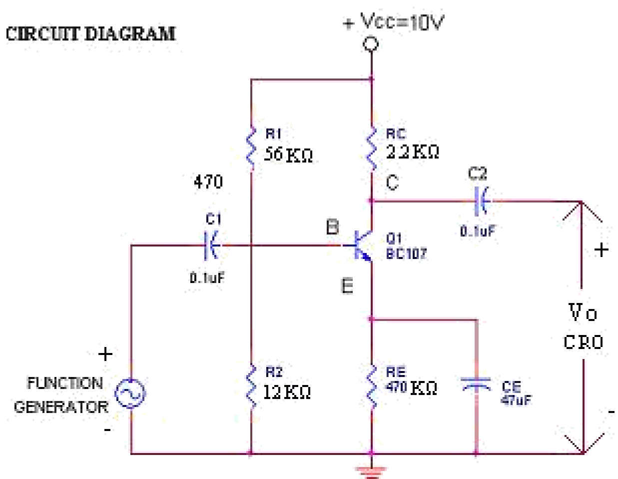
AIM:

To constant a voltage divider bias amplifier and measure input resistance and gain and also to plot the dc collector current as a function of collector resistance.

APPARATUS REQUIRED:

S.No.	Name	Range	Quantity
1.	Transistor	BC 107	1
2.	Resistor	56k ,12k ,2.2k ,470	1,1,1,1
3.	Capacitor	0.1µ F, 47µ F	2, 1
4.	Function Generator	(0-3)MHz	1
5.	CRO	30MHz	1
6.	Regulated power supply	(0-30)V	1
7.	Bread Board		1

CIRCUIT DIAGRAM:



MODEL GRAPH :

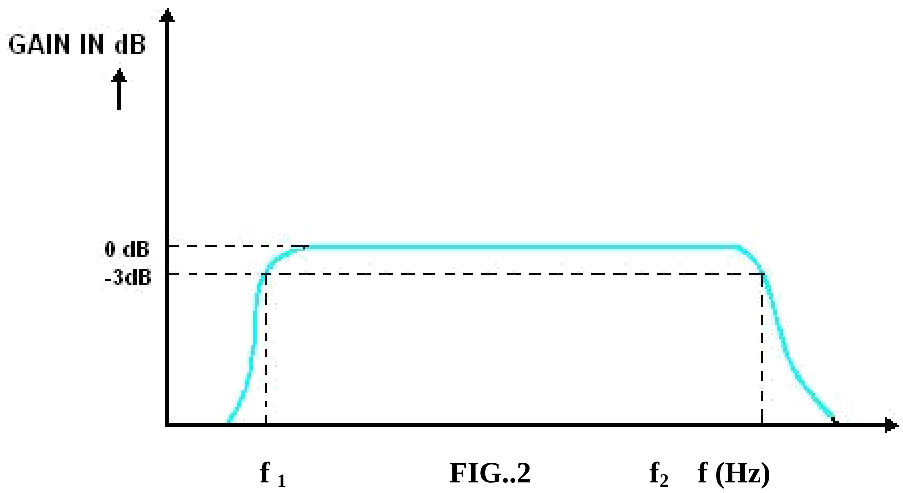


FIG..2

Keep the input voltage constant, Vin =

Frequency (in Hz)	Output Voltage (in volts)	Gain= 20 log(Vo/Vin) (in dB)

FORMULA:

a) $R_{in} = \beta * R_e$
b) $Gain = \beta * R_e/R_{in}$

THEORY:

This type of biasing is otherwise called Emitter Biasing. The necessary biasing is provided using 3 resistors: R1, R2 and Re. The resistors R1 and R2 act as a potential divider and give a fixed voltage to the base. If the collector current increases due to change in temperature or change in β, the emitter current I_e also increases and the voltage drop across R_e increases, reducing the voltage difference between the base and the emitter. Due to reduction in V_{be}, base current I_b and hence collector current I_c also reduces. This reduction in V_{be}, base current I_b and hence collector current I_c also reduces. This reduction in the collector current compensates for the original change in I_c.

The stability factor S= (1+β) * ((1/ (1+β)). To have better stability, we must keep R_b/R_e as small as possible. Hence the value of R1 R2 must be small. If the ratio R_b/R_e is kept fixed, S increases with β.

PROCEDURE:

1.Connections are given as per the circuit diagram.
2.Measure the input resistance as $R_{in}=V_{in}/I_{in}$ (with output open) and gain by plotting the frequency response.
3.Compare the theoretical values with the practical values.
4.Plot the dc collector current as a function of the collector resistance (ie) plot of V_{cc} and I_c for various values of Re.

RESULT:

Thus the voltage divider bias amplifier was constructed and input resistance and gain were determined. The Gain Bandwidth Product is found to be = ____________

DIFFERENTIAL AMPLIFIER USING BJT

EXP.NO : 4. **DATE:**

AIM:

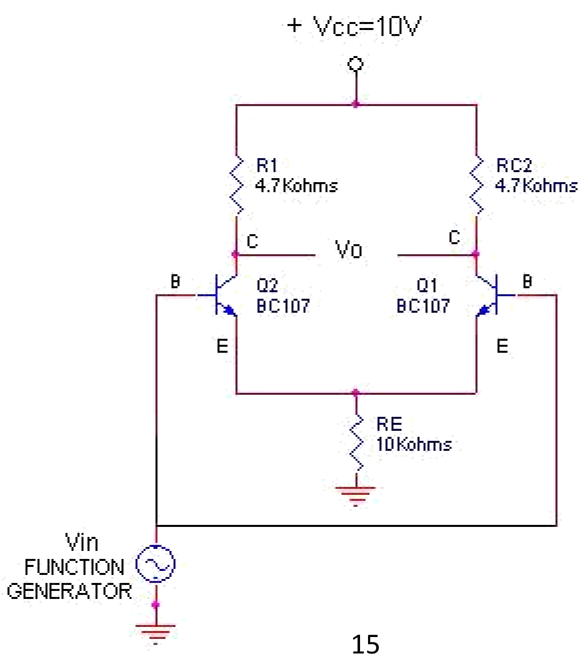
To construct a differential amplifier using BJT and to determine the dc collector current of individual transistors and also to calculate the CMRR.

APPARATUS REQUIRED:

S.No.	Name	Range	Quantity
1.	Transistor	BC107	2
2.	Resistor	4.7kΩ, 10kΩ	2,1
3.	Regulated power supply	(0-30)V	1
4.	Function Generator	(0-3) MHz	2
5.	CRO	30 MHz	1
6.	Bread Board		1

CIRCUIT DIAGRAM:

COMMONMODE OPERATION



OBSERVATION:

VIN =VO =AC = VO / VIN

FORMULA:

Common mode Gain (A_c) = V_O / V_{IN}

Differential mode Gain (A_d) = V_0 / V_{IN}

Where $V_{IN} = V_1 – V_2$

Common Mode Rejection Ratio (CMRR) = A_d/A_c

Where, A_d is the differential mode gain

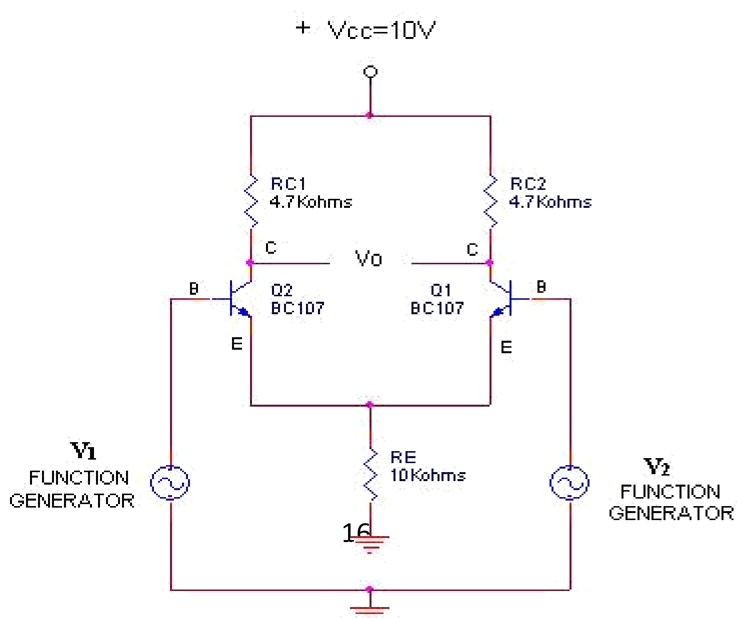
A_c is the common mode gain.

THEORY:

The differential amplifier is a basic stage of an integrated operational amplifier. It is used to amplify the difference between 2 signals. It has excellent stability, high versatility and immunity to noise. In a practical differential amplifier, the output depends not only upon the difference of the 2 signals but also depends upon the common mode signal.

Transistor Q1 and Q2 have matched characteristics. The values of R_{C1} and R_{C2} are equal. R_{e1} and R_{e2} are also equal and this differential amplifier is called emitter coupled differential amplifier. The output is taken between the two output terminals.

DIFFERENTIOL MODE OPERATION



OBSERVATION:

$\mathbf{V_{IN} = V_1 - V_2}$

$\mathbf{V_0 =}$

$\mathbf{A_d = V_0 / V_{IN}}$

For the differential mode operation the input is taken from two different sources and the common mode operation the applied signals are taken from the same source

Common Mode Rejection Ratio (CMRR) is an important parameter of the differential amplifier. CMRR is defined as the ratio of the differential mode gain, A_d to the common mode gain, A_c.

$$CMRR = A_d / A_c$$

In ideal cases, the value of CMRR is very high.

PROCEDURE:

1.Connections are given as per the circuit diagram.
2.To determine the common mode gain, we set input signal with voltage V_{in}=2V
and determine V_o at the collector terminals. Calculate common mode gain, $A_c=V_o/V_{in}$.
3.To determine the differential mode gain, we set input signals with voltages V_1 and V_2. Compute $V_{in}=V_1-V_2$ and find V_o at the collector terminals. Calculate differential mode gain, $A_d=V_o/V_{in}$.
4.Calculate the $CMRR=A_d/A_c$.
5.Measure the dc collector current for the individual transistors.

RESULT:

Thus, the Differential amplifier was constructed and dc collector current for the individual transistors is determined. The CMRR is calculated as _______________

DARLINGTON AMPLIFIER USING BJT

EXP.NO : 5. **DATE:**

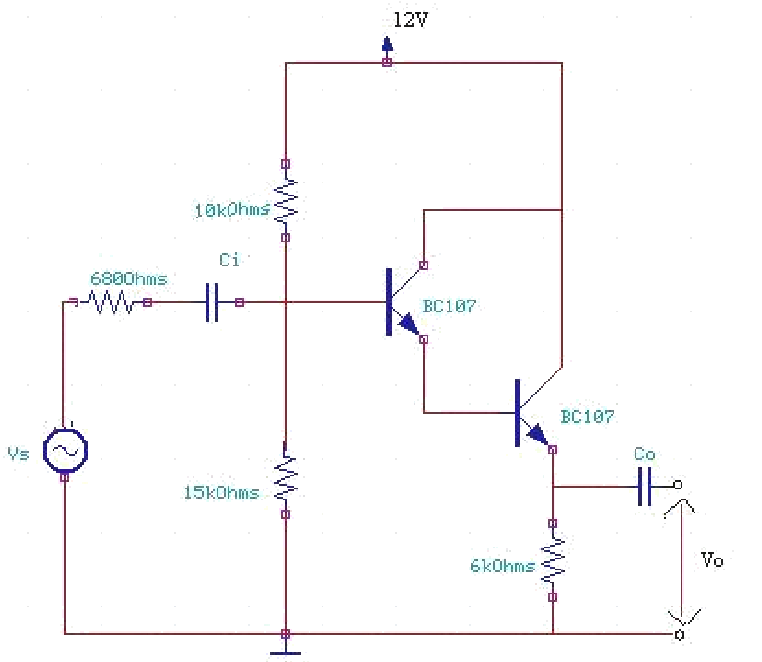
AIM:

To construct a Darlington current amplifier circuit and to plot the frequency response characteristics.

APPARATUS REQUIRED:

S.No.	Name	Range	Quantity
1.	Transistor	BC 107	1
2.	Resistor	15k ,10k ,680 ,6k	1,1,1,1
3.	Capacitor	0.1µ F, 47µ F	2, 1
4.	Function Generator	(0-3)MHz	1
5.	CRO	30MHz	1
6.	Regulated power supply	(0-30)V	1
7.	Bread Board		1

CIRCUIT DIAGRAM :



MODEL GRAPH

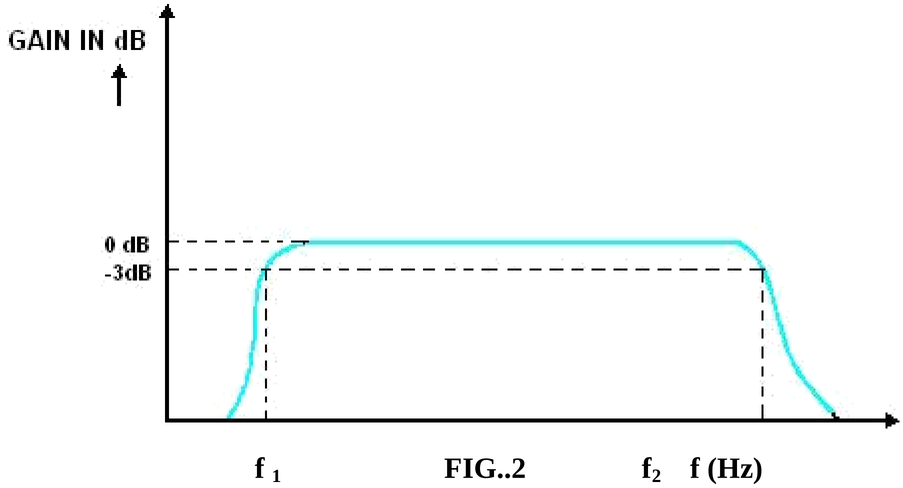


FIG..2

Keep the input voltage constant, Vin =

Frequency (in Hz)	Output Voltage (in volts)	Gain= 20 log(Vo/Vin) (in dB)

THEORY:

In Darlington connection of transistors, emitter of the first transistor is directly connected to the base of the second transistor .Because of direct coupling dc output current of the first stage is $(1+h_{fe})I_{b1}$.If Darlington connection for n transitor is considered, then due to direct coupling the dc output current foe last stage is $(1+h_{fe})^{n}$ times I_{b1} .Due to very large amplification factor even two stage Darlington connection has large output current and output stage may have to be a power stage. As the power amplifiers are not used in the amplifier circuits it is not possible to use more than two transistors in the Darlington connection.

In Darlington transistor connection, the leakage current of the first transistor is amplified by the second transistor and overall leakage current may be high, Which is not desired.

PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Set Vi =50 mv, using the signal generator.
3. Keeping the input voltage constant, vary the frequency from 0 Hz to 1M Hz in regular steps and note down the corresponding output voltage.
4. Plot the graph; Gain (dB) vs Frequency(Hz).
5. Calculate the bandwidth from the graph.

RESULT:

Thus, the Darlington current amplifier was constructed and the frequency response curve is plotted. . The Gain Bandwidth Product is found to be = _________________.

SOURCE FOLLOWER WITH BOOTSTRAPPED CIRCUIT

EXP.NO : 6. **DATE:**

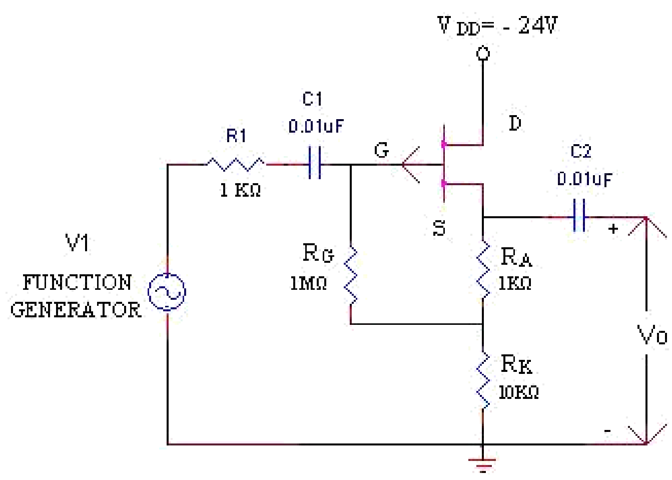
AIM:

To construct a source follower with bootstrapped gate resistance amplifier and plot its frequency response characteristics.

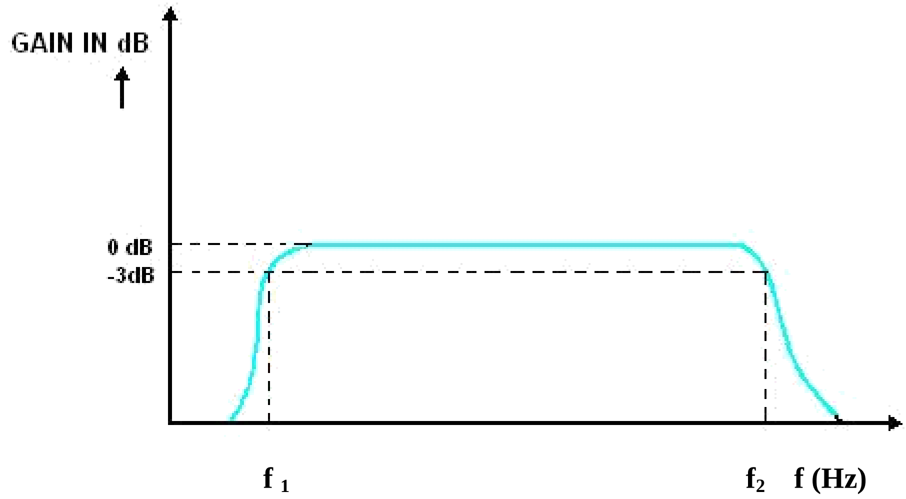
APPARATUS REQUIRED:

S.No.	Name	Range	Quantity
1.	Transistor	BC107	2
2.	Resistor	1kΩ,11 kΩ,1M kΩ	1,1,1
3.	Regulated power supply	(0-30)V	1
4.	Signal Generator	(0-3)MHz	1
5.	CRO	30 MHz	1
6.	Bread Board		1
7.	Capacitor	0.01µF	2

CIRCUIT DIAGRAM



MODEL GRAPH



Keep the input voltage constant (V_{in}) =

frequency (in Hz)	Output Voltage (in volts)	Gain = 20 log (V_o / V_{in}) (in dB)

THEORY:

Source follower is similar to the emitter follower(the output source voltage follow the gate input voltage),the circuit has a voltage gain of less than unity, no phase reversal, high input impedance, low output impedance. Here the Bootstrapping is used to increase the input resistance by connecting a resistance in between gate and source terminals. The resister R_A is required to develop the necessary bias for the gate.

PROCEDURE:

1. Connections are made as per the circuit diagram.
2. The waveforms at the input and output are observed for cascode operations by varying the input frequency.
3. The biasing resistances needed to locate the Q-point are determined.
4. Set the input voltage as 1V and by varying the frequency, note the output voltage.
5. Calculate gain=20 log (V_o / $V_{in.}$)
6. A graph is plotted between frequency and gain.

RESULT:

Thus, the Source follower with Bootstrapped gate resistance was constructed and the gain was determined.

CLASS - A POWER AMPLIFIER

EXP NO: 7 **DATE :**

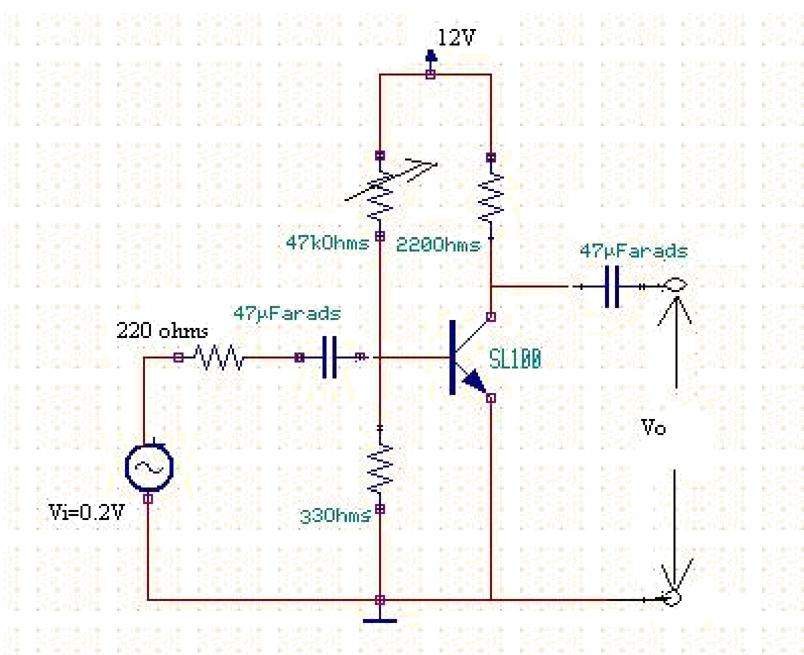
AIM:

To construct a Class A power amplifier and observe the waveform and to compute maximum output power and efficiency.

APPARATUS REQUIRED:

S.No.	Name	Range	Quantity
1.	Transistor	CL100, BC558	1,1
2.	Resistor	47k ,33 ,220Ω,	2,1
3.	Capacitor	47 µ F	2
4.	Signal Generator	(0-3)MHz	1
5.	CRO	30MHz	1
6.	Regulated power supply	(0-30)V	1
7.	Bread Board		1

CIRCUIT DIAGRAM



TABULATION

Keep the input voltage constant, Vin =

Frequency (in Hz)	Output Voltage (in volts)	Gain= 20 log(Vo/Vin) (in dB)

FORMULA

Maximum power transfer =P^o,max=$V_o^{\,2}/R_L$

Effeciency,η = $P_{o,max}/P_c$

THEORY:

The power amplifier is said to be Class A amplifier if the Q point and the input signal are selected such that the output signal is obtained for a full input signal cycle.

For all values of input signal, the transistor remains in the active region and never enters into saturation region. When an a.c signal is applied, the collector voltage varies sinusoidally hence the collector current also varies sinusoidally.The collector current flows for 360^0 (full cycle) of the input signal. i e the angle of the collector current flow is 360^0 .

PROCEDURE:

1. Connect the circuit as per the circuit diagram.

2. Set Vi =50 mv, using the signal generator.

3. Keeping the input voltage constant, vary the frequency from 10 Hz to 1M Hz in regular steps and note down the corresponding output voltage.

4. Plot the graph; Gain (dB) vs Frequency(Hz).

RESULT:

Thus the Class A power amplifier was constructed. The following parameters were calculated:

a) Maximum output power=
` b) Efficiency=

CLASS – B COMPLEMENTARY SYMMETRY POWER AMPLIFIER

EXP NO: 8 **DATE :**

AIM:

To construct a Class B complementary symmetry power amplifier and observe the waveforms with and without cross-over distortion and to compute maximum output power and efficiency.

APPARATUS REQUIRED:

S.No.	Name	Range	Quantity
1.	Transistor	CL100, BC558	1,1
2.	Resistor	4.7k ,15k	2,1
3.	Capacitor	100µ F	2
4.	Diode	IN4007	2
5.	Signal Generator	(0-3)MHz	1
6.	CRO	30MHz	1
7.	Regulated power supply	(0-30)V	1
8.	Bread Board		1

CIRCUIT DIAGRAM

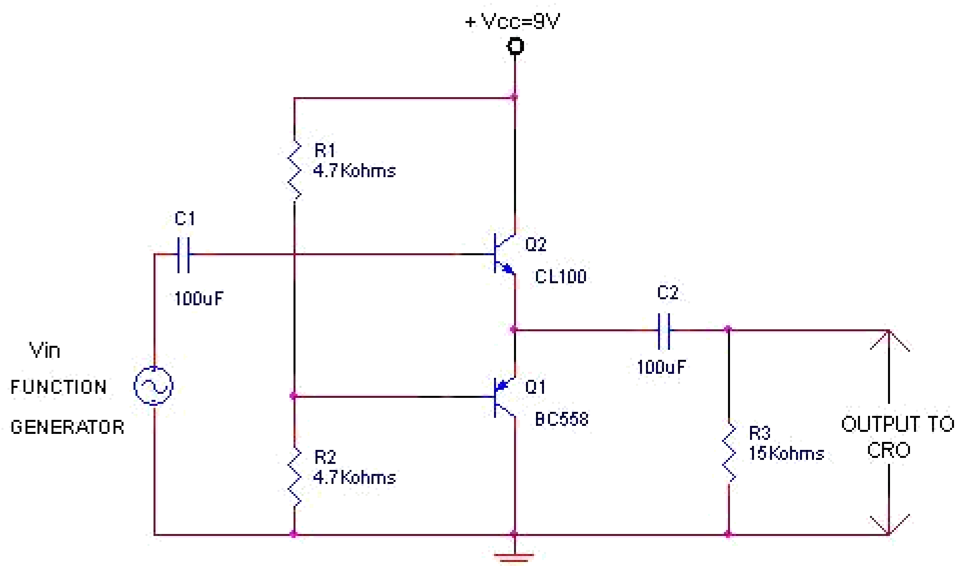


FIG.6.1

FORMULA:

Input power, $P_{in}=2V_{cc}I_m/\Pi$

Output power, $P_{out}=V_mI_m/2$

Power Gain or efficiency, $\eta=\pi/4(V_m/V_{cc})\ 100$

THEORY:

A power amplifier is said to be Class B amplifier if the Q-point and the input signal are selected such that the output signal is obtained only for one half cycle for a full input cycle. The Q-point is selected on the X-axis. Hence, the transistor remains in the active region only for the positive half of the input signal.

There are two types of Class B power amplifiers: Push Pull amplifier and complementary symmetry amplifier. In the complementary symmetry amplifier, one n-p-n and another p-n-p transistor is used. The matched pair of transistor are used in the common collector configuration. In the positive half cycle of the input signal, the n-p-n transistor is driven into active region and starts conducting and in negative half cycle, the p-n-p transistor is driven into conduction. However there is a period between the crossing of the half cycles of the input signals, for which none of the transistor is active and output, is zero

CIRCUIT DIAGRAM

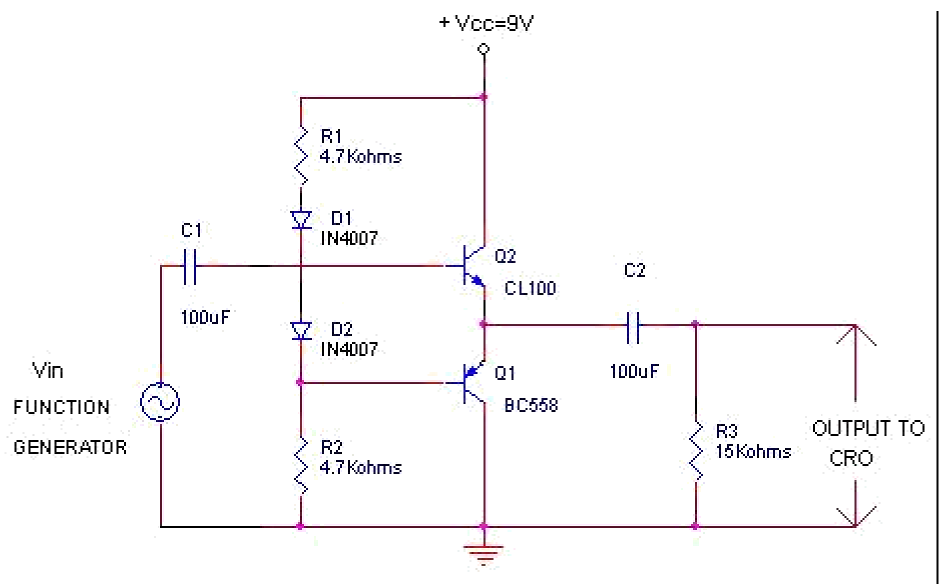


FIG.6.2

OBSERVATION

OUTPUT SIGNAL

AMPLITUDE :

TIME PERIOD :

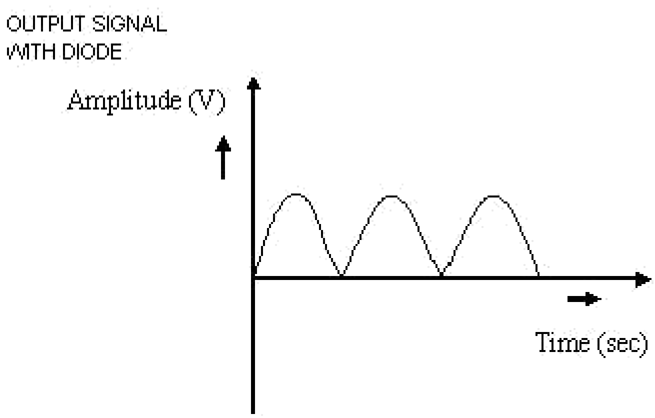
CALCULATION

POWER, P_{IN} $= 2V_{CC} I_m/л$

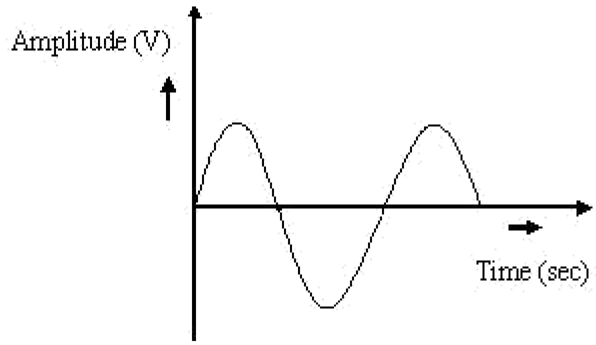
OUTPUT POWER, P_{OUT} $= V_m I_m/2$

EFFICIENCY, η $= (л/4)(V_m/V_{CC}) \times 100$

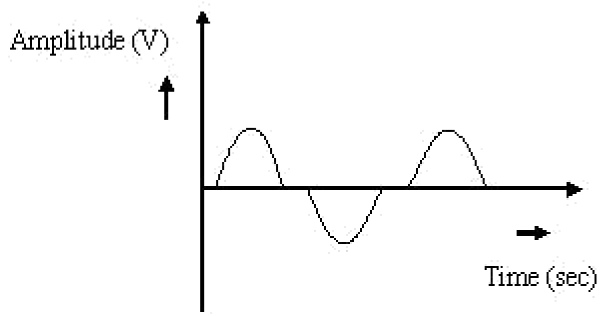
MODEL GRAPH



INPUT SIGNAL



OUTPUT SIGNAL
WITHOUT DIODE



PROCEDURE:

1. Connections are given as per the circuit diagram without diodes.
2. Observe the waveforms and note the amplitude and time period of the input signal and distorted waveforms.
3. Connections are made with diodes.
4. Observe the waveforms and note the amplitude and time period of the input signal and output signal.
5. Draw the waveforms for the readings.
6. Calculate the maximum output power and efficiency.

Hence the nature of the output signal gets distorted and no longer remains the same as the input. This distortion is called cross-over distortion. Due to this distortion, each transistor conducts for less than half cycle rather than the complete half cycle. To overcome this distortion, we add 2 diodes to provide a fixed bias and eliminate cross-over distortion.

RESULT:

Thus the Class B complementary symmetry power amplifier was constructed to observe cross-over distortion and the circuit was modified to avoid the distortion. The following parameters were calculated:

a)Maximum output power=
b)Efficiency=

SIMULATION USING PSPICE

FREQUENCY RESPONSE OF CE AMPLIFIER WITH EMITTER RESISTANCE AMPLIFIER

EXP.NO: 09 **DATE:**

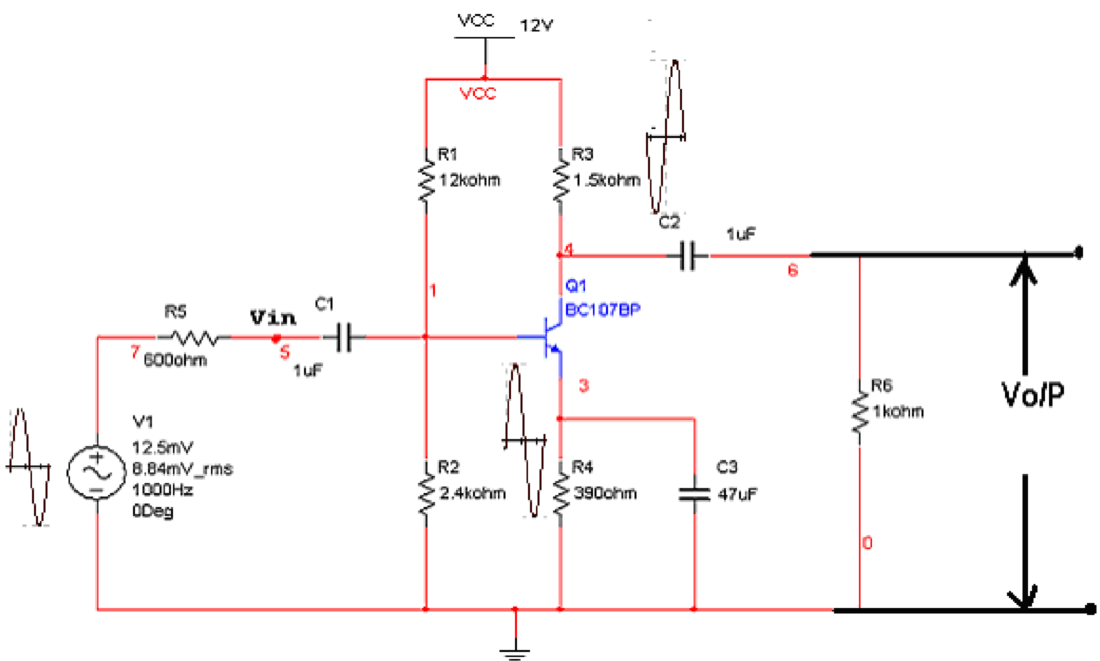
AIM :

To design and simulate the frequency response of common emitter amplifier for a gain of 50.

APPARATUS REQUIRED:

- PC with SPICE software

CIRCUIT DIAGRAM :



THEORY:

The CE amplifier provides high gain and wide frequency response. The emitter lead is common to both the input and output circuits are grounded. The emitter base junction is at forward biased .The collector current is controlled by the base current rather than the emitter current. The input signal is applied to the base terminal of the transistor and amplified output taken across collector terminal. A very small change in base current produces a much larger change in collector current. When the positive is fed to input circuit it opposes forward bias of the circuit which cause the collector current to decrease, it decreases the more negative. Thus when input cycle varies through a negative half cycle, increases the forward bias of the circuit, which causes the collector current increases .Thus the output signal in CE is out of phase with the input signal.

PROCEDURE

1. Select different components and place them in the grid.
2. For calculating the voltage gain the input voltage of 25mv (p-p) amplitude and 1KHz frequency is applied, then the circuit is simulated and output voltage is noted.
3. The voltage gain is calculated by using the expression

$$Av = Vo/Vi$$

4. For plotting frequency response, the input voltage is kept constant at 25mv(p-p) and frequency is varied.
5. Note down the output voltage for each frequency.
6. All readings are tabulated and Av in db is calculated using the formula

$$20 \text{ Log } Vo/Vi.$$

7. A graph is drawn by taking frequency on X-axis and gain in dB on Y-axis on a Semi log graph sheet.

SPICE FILE :

```
VIN 1 4 SIN(0 1.5V 2KHZ)
VB 4 0 2.3V
RL 3 0 15K
V1 2 0 15V
Q1 2 1 3 MOD1
.MODEL MOD1 NPN
.TRAN 0.02MS 0.78MS
.PROBE
.END
```

THEORITICAL CALCULATIONS :

$$V_B = \frac{R_2}{R_1 + R_2} \times V_{CC} = \frac{2400}{14400} \times 12 = 2V$$

$$V_E = V_B - V_{BE} = 2 - 0.7 = 1.3V$$

$$IE = \frac{VE}{RE} = \frac{1.3}{390} = 3.3mA$$

$$V_C = V_{CC} - I_C \times R_C = 12 - 0.0033 \times 1500 = 7.05V$$

To calculate A_v, $Z_{in(base)}$ and Z_{in} :

$$r'_e = \frac{25mV}{I_E} = \frac{25}{3.3} = 7.575\Omega$$

$$r_L = \frac{R_C \times R_L}{R_C + R_L} = \frac{1500 \times 1000}{1500 + 1000} = 600\Omega$$

$$A_V = \frac{r_L}{r'_e} = \frac{600}{7.575} = 79.20$$

$$Z_{in(base)} = \beta r'_e = 40 \times 7.575 = 303\Omega$$

$$Z_{in} = Z_{in(base)} \parallel R_1 \parallel R_2 = 303 \parallel 12000 \parallel 2400 = 263\Omega$$

$$V_b = \frac{Z_{in}}{R_G + Z_{in}} \times 25mV = \frac{263}{600 + 263} = 7.61mV$$

$$V_{out} = A_V \times V_b = 79.20 \times 7.61mV = 0.603V$$

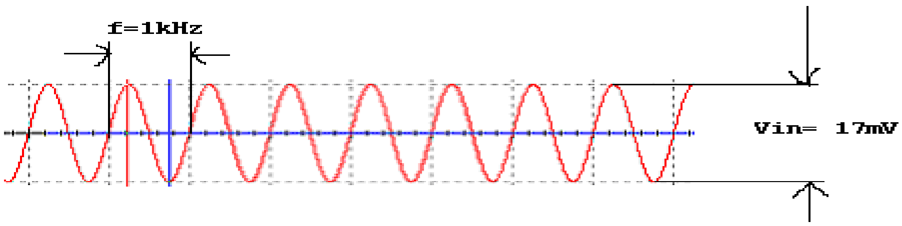
PRACTICAL CALCULATIONS :

$$V_{in} =$$

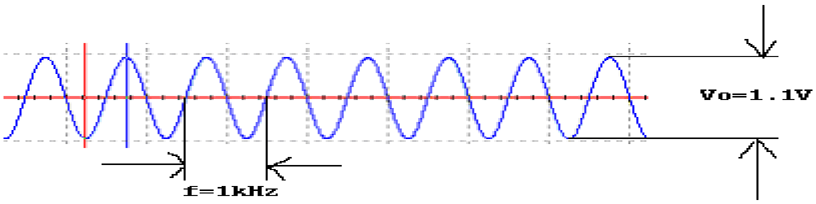
$$V_{out} =$$

$$A_V = \frac{V_{out}}{V_{in}} =$$

INPUT WAVE FORM :



OUT PUT WAVE FORM :



RESULT:

Thus the simulation of class-B amplifier using PSPICE was simulated successfully.

DC RESPONSE OF CS AMPLIFIER

EXP.NO: 10 **DATE:**

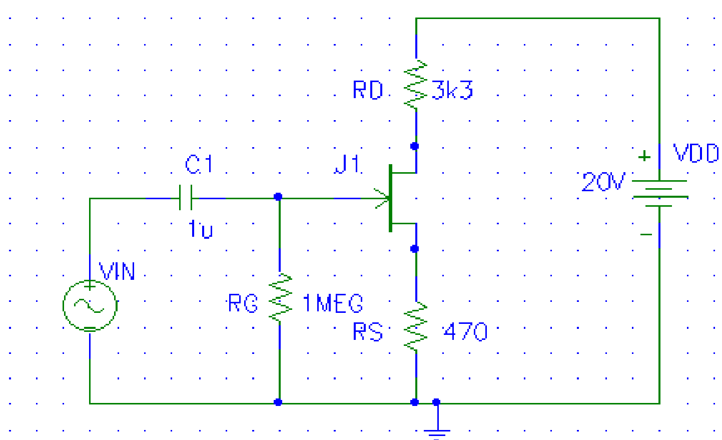
AIM:

To find the DC response of CS amplifier.

APPARATUS REQUIRED:

- PC with SPICE software

CIRCUIT DIAGRAM:



THEORY:

In electronics, a **common-source** amplifier is one of three basic single-stage field-effect transistor (FET) amplifier topologies, typically used as a voltage or transconductance amplifier. The easiest way to tell if a FET is common source, common drain, or common gate is to examine where the signal enters and leaves. The remaining terminal is what is known as "common". In this example, the signal enters the gate, and exits the drain. The only terminal remaining is the source. This is a common-source FET circuit. The analogous bipolar junction transistor circuit is the common-emitter amplifier.

The common-source (CS) amplifier may be viewed as a transconductance amplifier or as a voltage amplifier. (See classification of amplifiers). As a transconductance amplifier, the input voltage is seen as modulating the current going to the load. As a voltage amplifier, input voltage modulates the amount of current flowing through the FET, changing the voltage across the output resistance according to Ohm's law. However, the FET device's output resistance typically is not high enough for a reasonable transconductance amplifier (ideally infinite), nor low enough for a decent voltage amplifier (ideally zero). Another major drawback is the amplifier's limited high-frequency response. Therefore, in practice the output often is routed through either a voltage follower (common-drain or CD stage), or a current follower (common-gate or CG stage), to obtain more favorable output and frequency characteristics. The CS–CG combination is called a cascode amplifier.

PROCEDURE:

1.Measure the DC operating point of each transistor and compare your results with the calculated values.

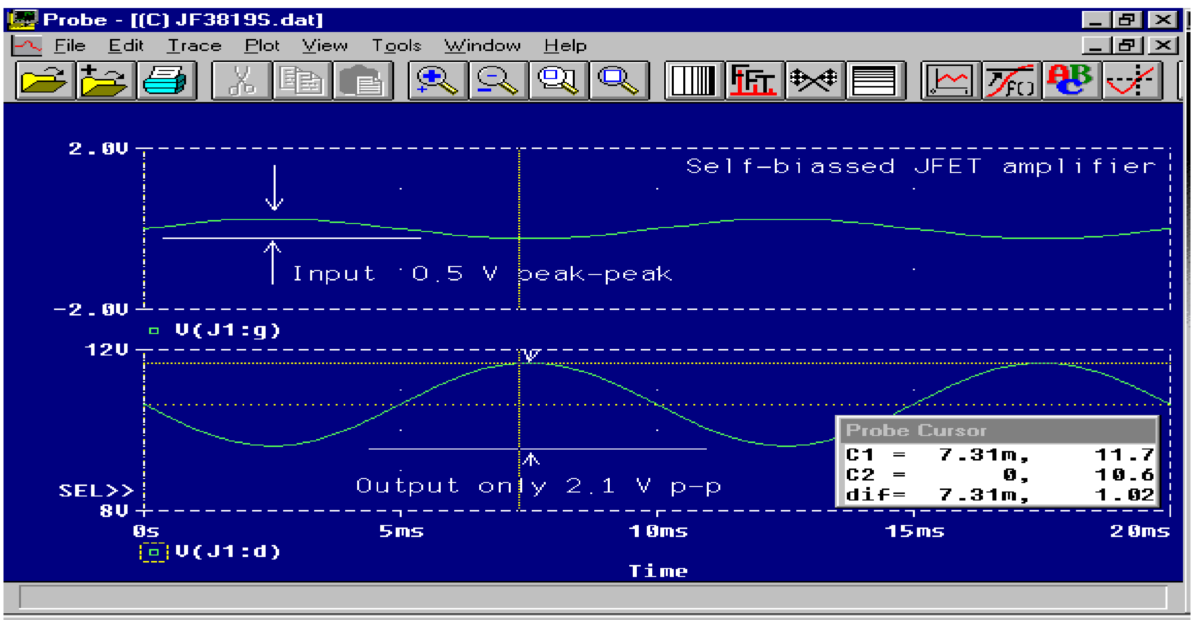
2.At a frequency of 5 KHz, measure the voltage gain, the input and the output resistance and compare your results withthe theoretical values. Calculate the power gain from both experimental and theoretical values.

3.Find the maximum peak-to-peak output voltage swing (i.e. the maximum swing without distortion).

4.Measure the frequency response of the circuit and comment on the change observed in comparison with a single stage common-emitter amplifier.

5.Simulate the circuit using Pspice. Compare the Pspice results with those obtained in the previous parts.

OUTPUT WAVEFORM:



RESULT:

Thus the simulation of DC response of CS amplifier was simulatdd successfully.

FREQUENCY RESPONSE OF CASCODE AMPLIFIER

EXP.NO: 11 **DATE:**

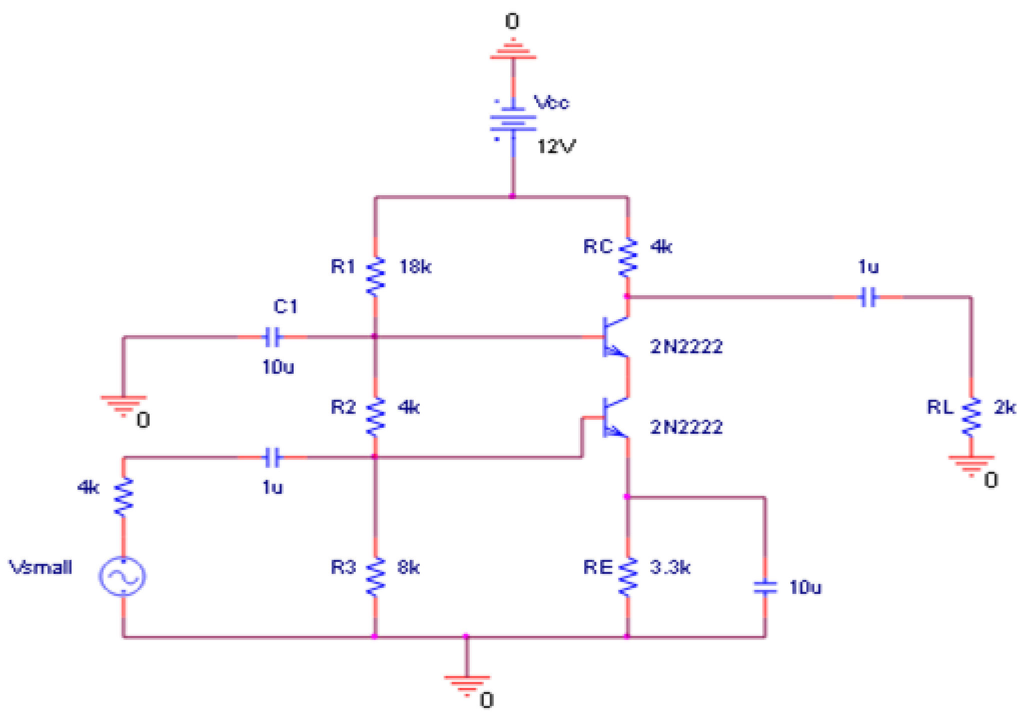
AIM:

To determine the frequency response of Cascode Amplifier.

APPARATUS REQUIRED:

- PC with SPICE software

CIRCUIT DIAGRAM:



THEORY:

Cascode amplifier is a two stage circuit consisting of a transconductance amplifier followed by a buffer amplifier. The word "cascode" was originated from the phrase "cascade to cathode". This circuit have a lot of advantages over the single stage amplifier like, better input output isolation, better gain, improved bandwidth, higher input impedance, higher output impedance, better stability, higher slew rate etc. The reason behind the increase in bandwidth is the reduction of

Miller effect. Cascode amplifier is generally constructed using FET (field effect transistor) or BJT (bipolar junction transistor). One stage will be usually wired in common source/common emitter mode and the other stage will be wired in common base/ common emitter mode.

Miller effect.

Miller effect is actually the multiplication of the drain to source stray capacitance by the voltage gain. The drain to source stray capacitance always reduces the bandwidth and when it gets multiplied by the voltage gain the situation is made further worse. Mulitiplication of stray capacitance increases the effective input capacitance and as we know, for an amplifier, the increase in input capacitance increases the lower cut of frequency and that means reduced bandwidth. Miller effect can be reduced by adding a current buffer stage at the output of the amplifier or by adding a voltage buffer stage before the input.

PROCEDURE:

1.Measure the DC operating point of each transistor and compare your results with the calculated values.

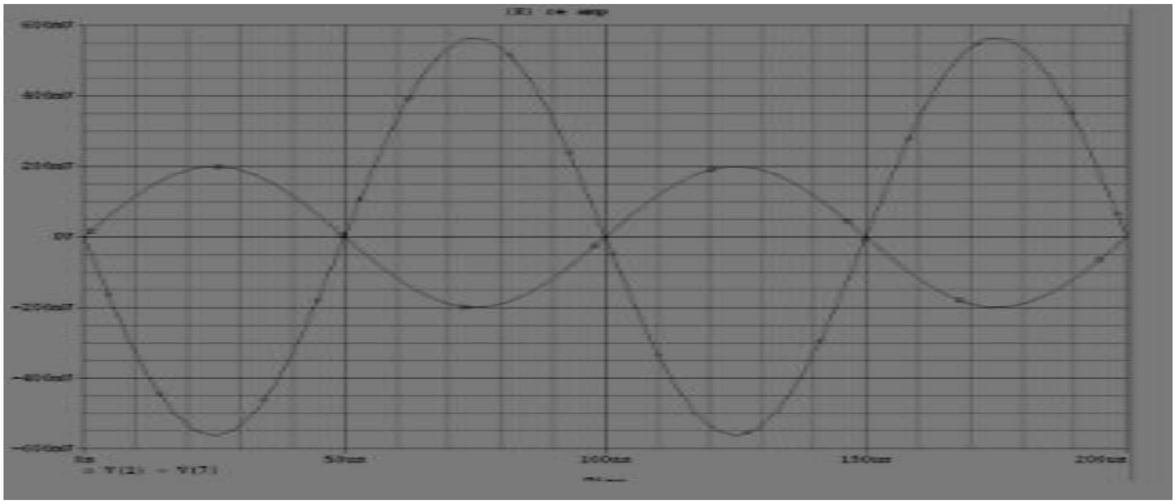
2.At a frequency of 5 KHz, measure the voltage gain, the input and the output resistance and compare your results withthe theoretical values. Calculate the power gain from both experimental and theoretical values.

3.Find the maximum peak-to-peak output voltage swing (i.e. the maximum swing without distortion).

4.Measure the frequency response of the circuit and comment on the change observed in comparison with a single stage common-emitter amplifier.

5.Simulate the circuit using Pspice. Compare the Pspice results with those obtained in the previous parts.

<u>MODEL GRAPH</u>



RESULT:

Thus she simulation of frequency response of Cascode Amplifier was simulated**.**

TRANSFER CHARACTERISTICS OF CLASS-B POWER AMPLIFIER

EXP.NO: 12 **DATE:**

AIM:

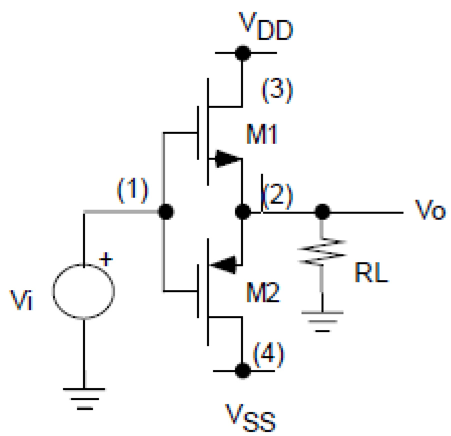
To simulate the Class-B amplifier by using PSICE.

APPARATUS REQUIRED:

- PC with SPICE software

CIRCUIT DIAGRAM:

Class B Amplifier



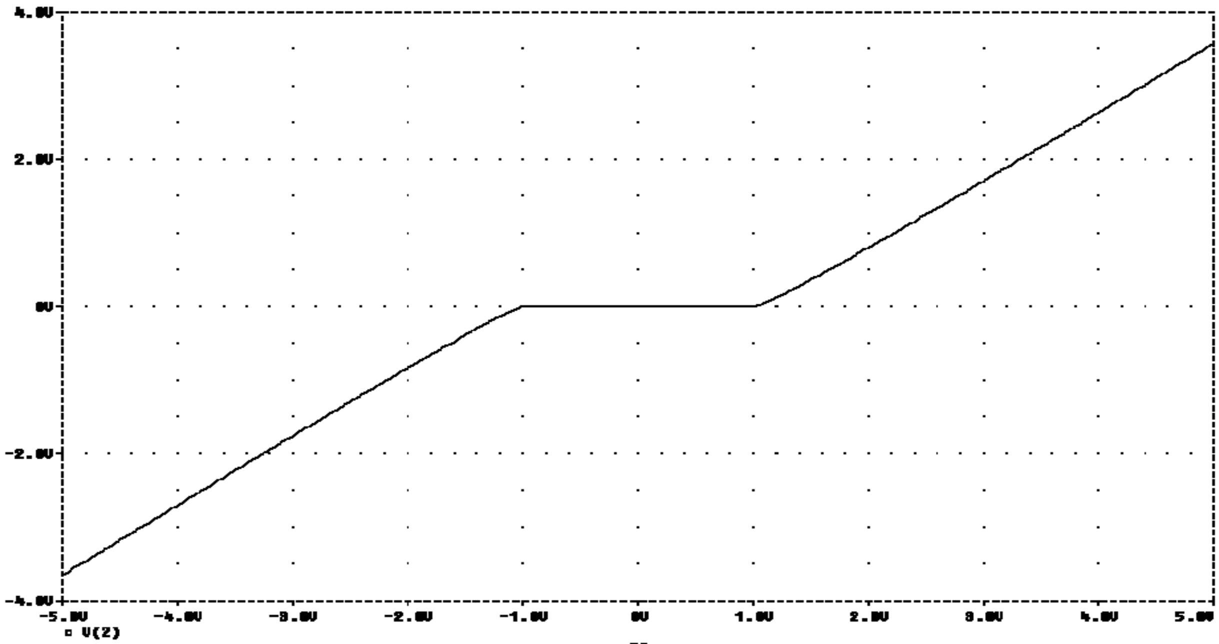
THEORY:

Class-B amplifiers improve the efficiency of the output stage by eliminating quiescent power dissipation by operating at zero quiescent current. This is implemented in Figure 2. As the input voltage V swings positive, M1 turns on when V exceeds the threshold voltage , and the output voltage follows the input on the positive swing. When the input voltage swings negative, M2 turns on when is less than threshold voltage , and the output voltage follows the input on the negative swing. There is a "dead zone" in the class-B voltage transfer characteristic, where both transistors are not conducting.

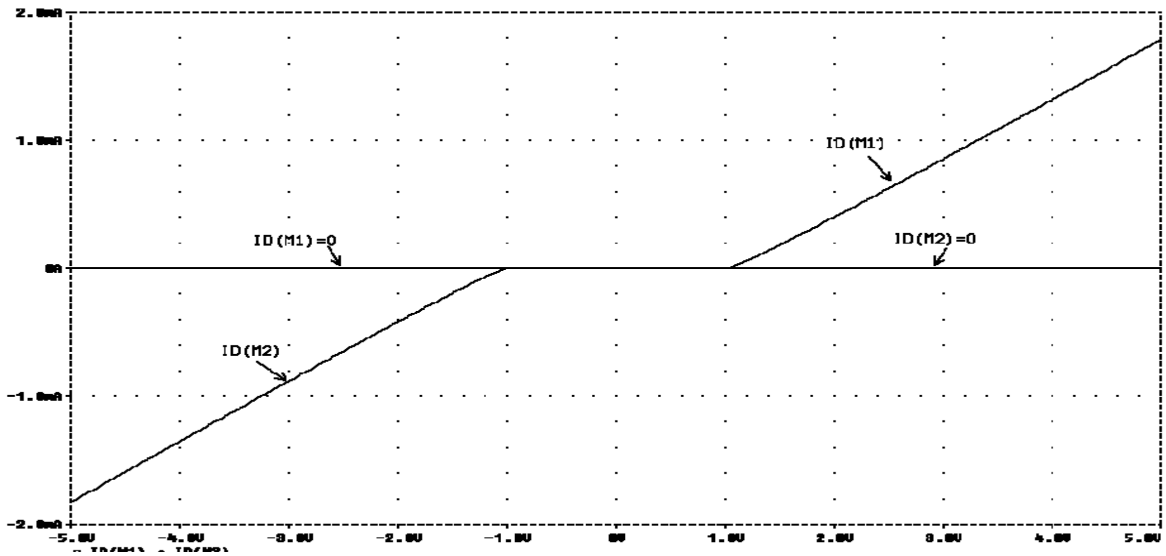
The class-B amplifier is simulated the netlist is shown below:
* Class B Amplifier
*Filename="classb.cir"

```
VI 1 0 DC 0 sin(0 5 1000)
VDD 3 0 DC 5
VSS 4 0 DC -5
M1 3 1 2 4 N1 W=1000U L=2U
M2 4 1 2 3 P1 W=4000U L=2U
RL 2 0 2K
.MODEL N1 NMOS VTO=1 KP=40U
.MODEL P1 PMOS VTO=-1 KP=15U
.OP
.DC VI -5 5 .05
.TF V(2) VI
.TRAN 1U 2M
.PROBE
.END
```

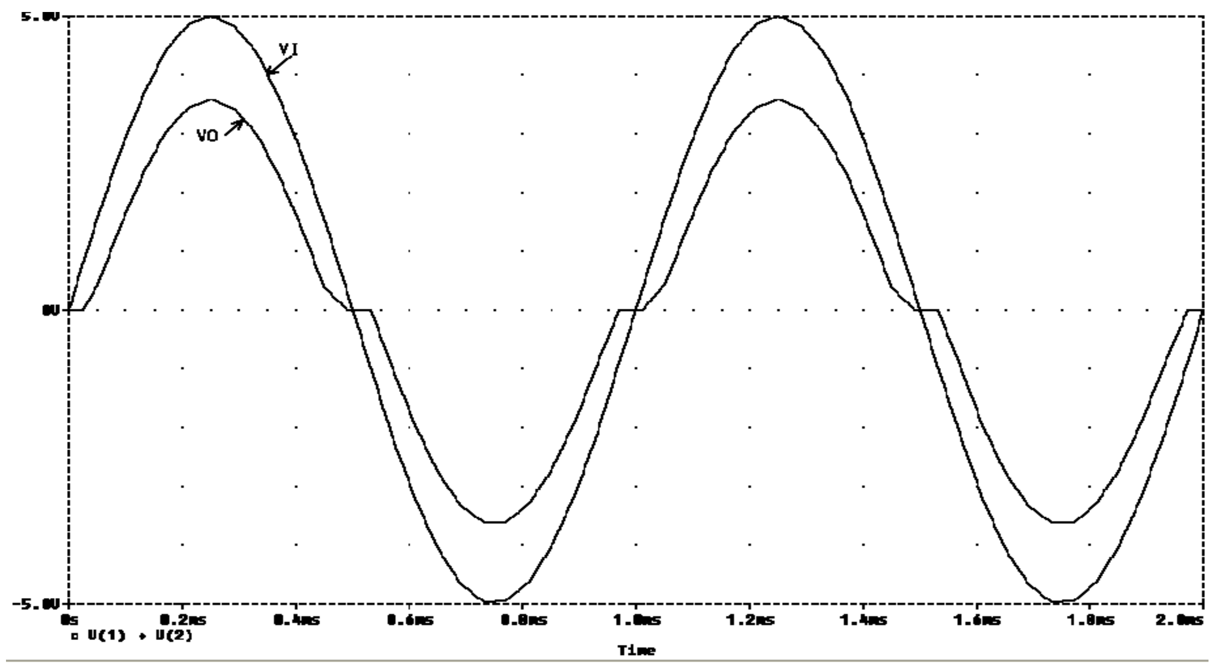
The Pspice simulation shows the presence of the dead zone in the output voltage transfer characteristic.



The current in each transistor conducts for less than half cycle. M1 conducts in the positive half cycle and M2 in the negative half cycle.



The transient analysis shows that the dead zone causes a distortion in the output waveform.



RESULT:

Thus the simulation of class-B amplifier using PSPICE was simulated successfully.

DESIGN OF A DC POWER SUPPLY USING RECTIFIER

EXP.NO: 13 **DATE:**

AIM:

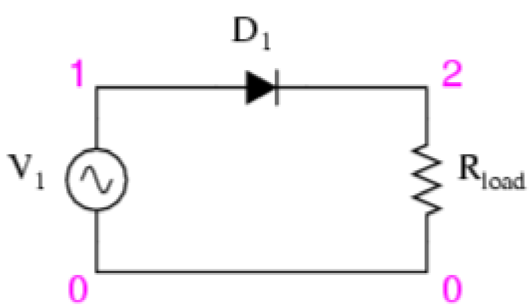
To simulate the design of DC power supply using by using PSICE.

APPARATUS REQUIRED:

- PC with SPICE software

CIRCUIT DIAGRAM:

HALF WAVE RECTIFIER:



THEORY:

In half wave rectification, either the positive or negative half of the AC wave is passed, while the other half is blocked. Because only one half of the input waveform reaches the output, it is very inefficient if used for power transfer. Half-wave rectification can be achieved with a single diode in a one-phase supply, or with three diodes in a three-phase supply. Half wave rectifiers yield a unidirectional but pulsating direct current.

SPICE FILE :

```
V1 1 0 SIN(0 10V 100HZ)
R 1 2 1K
DA 0 2 D1
.MODEL D1 D
.TRAN 0.01MS 20MS
.PROBE
.END
```

MODEL GRAPH:

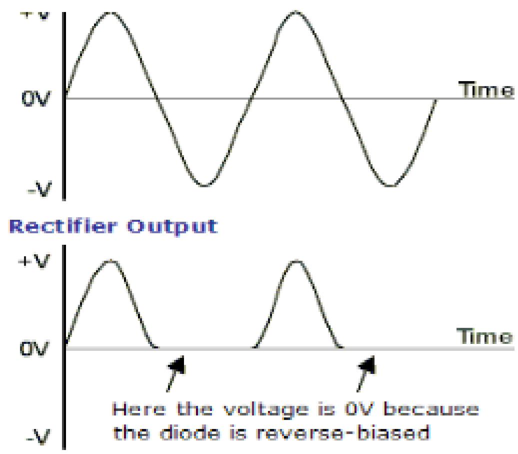
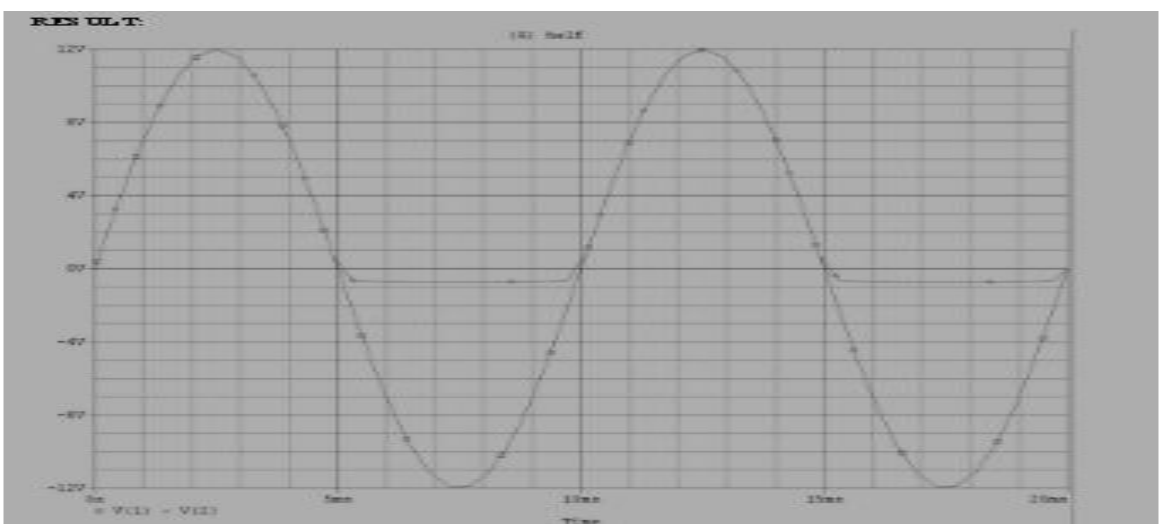


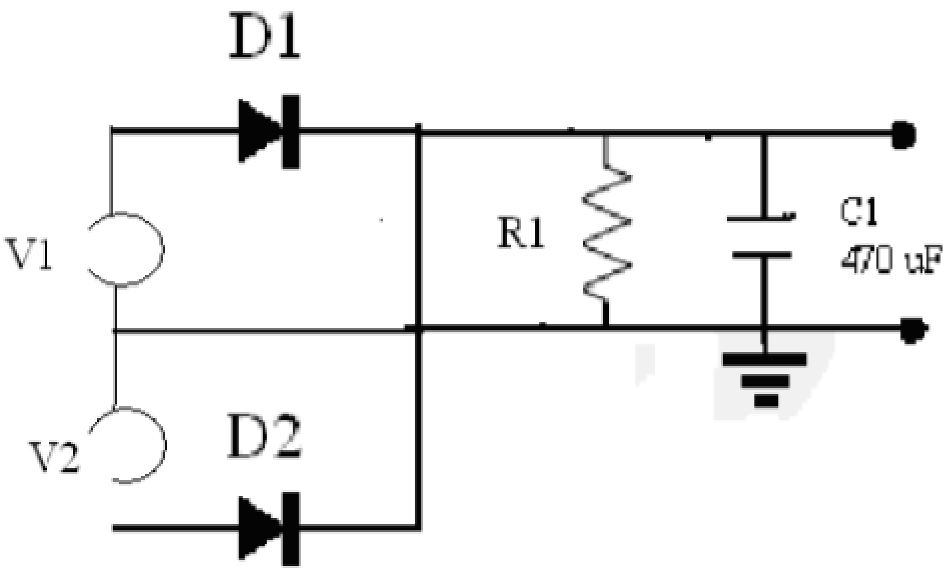
Figure : Half-wave rectification

OUTPUT:



FULLWAVE RECTIFIER:

CIRCUIT DIAGRAM:



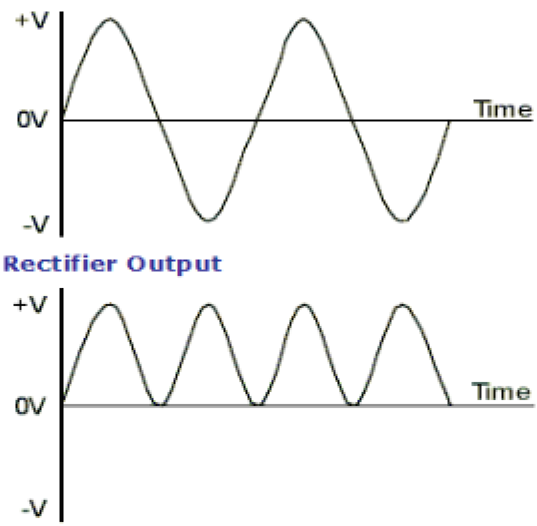
THEORY:

A full-wave rectifier converts the whole of the input waveform to one of constant polarity (positive or negative) at its output. Full-wave rectification converts both polarities of the input waveform to DC (direct current), and is more efficient.

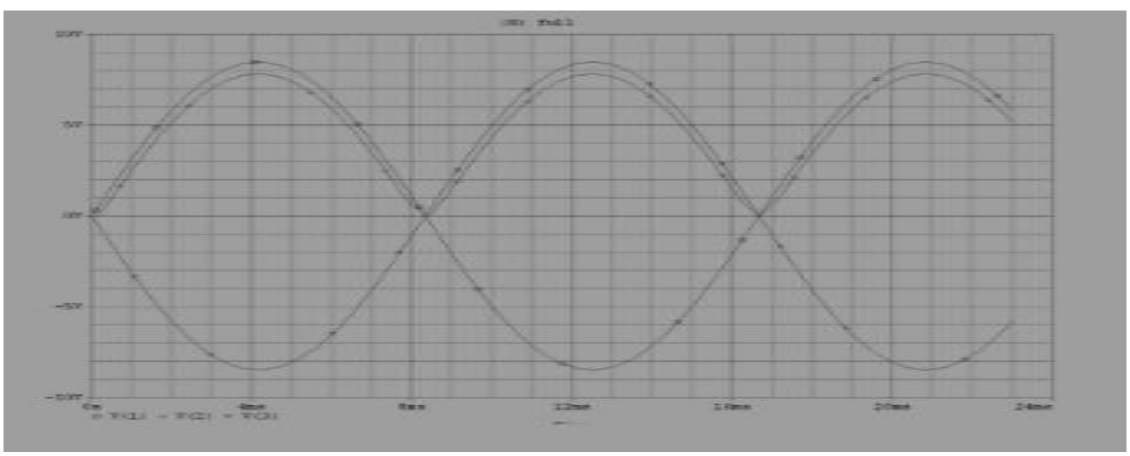
SPICE FILE :

V1 1 0 SIN(0 10V 100HZ)
R 2 3 1K
C 2 3 1N
D1 1 2 MOD1
D2 0 2 MOD1
.MODEL MOD1 D
.TRAN 0.01MS 20MS
PROBE
.END

GRAPH:



OUTPUT:



RESULT:

Thus the simulation of design of a DC power supply using Rectifier was simulated.