Analog Electronics

1. STUDY OF WAVEFORMS ON CRO

OBJECTIVES

1. To understand the function of each control knob on the front panel of the double-beam Oscilloscope.
2. To measure the output sinusoidal voltage of the signal generator.
3. To measure the frequency of a sine-wave voltage obtained from the signal generator.
4. To study the square wave-form and voltage pulse shape.

APPARATUS REQUIRED:

Double beam C.R.Oscilloscope, Function Generator.

THEORY

. The main purpose of a CRO is to display waveshapes and make measurements of the characteristics of the waveshape like the peak voltage, time period of wave and thus the frequency of the wave, pulse-shape, pulse width and pulse repetition rate etc. To operate the CRT, the oscilloscope has a sweep generator (saw- tooth oscillator), deflection voltage amplifiers (horizontal and vertical), power supply circuits and a number of controls, switches and input and output terminals on the front panel. An electron beam, produced by the electron gun in the CRT strikes the fluorescent screen. By applying voltages to the horizontal and vertical deflection plates in the CRT, the beam is deflected in any desired direction. To display a waveform, it is connected to the vertical input of the scope, which is usually a BNC connector. To the horizontal deflection plates , a saw tooth wave voltage is applied internally. The trigger circuit of the sweep generator is activated either by the input signal or by an external trigger signal. The impedance offered by the CRO to the signal at the input terminals is of the order of a Megohm shunted by a capacitor (about 40 pF).

WAVEFORM DISPLAY:

For a usual display of a signal waveform on the CRT screen, a time base generator is needed. This generator gives a sawtooth voltage waveform. Each cycle of this periodic waveform has a linearly increasing voltage part followed by a fast decreasing part (called the flyback time or return time). Since the sweep voltage increases linearly with time, the spot moves constantly with constant velocity across the screen from left to right. At the end of the sweep the spot quickly returns to the starting position on n the left.

This cycle is repeated. To measure time, the horizontal displacement on the CRT screen is calibrated in time. The calibration of the horizontal axis is read from the front panel control marked TIME/DIV. The vertical axis is calibrated in volts to measure the signal voltage. The calibration is read from the front panel control marked VOLTS/DIV. The sweep and the signal voltages are amplified before application to the deflecting plates.

APPLICATIONS:

(i) Visual display and qualitative study of signal waveform: To display a signal on the oscilloscope, the signal is applied to the vertical input terminals. The time variation of the signal is visualized by means of the sweep generator displacing the spot in proportion to time in the horizontal direction. The nature of the signal can be qualitatively studied from the trace on the screen.

(ii) Measurement of the voltage: The calibration of the vertical scale gives the voltage corresponding to the vertical deflection of the spot on the screen,. Thus the magnitude of the applied D.C. voltage or the voltage at different times of a time-varying signal can be measured.

(iii) Measurement of Frequency: The calibration of the horizontal scale (time base) helps to determine the frequency of a time varying signal displayed on the screen,. If N complete cycles of the a.c. signal are found to appear in a time interval t, the time period of the signal is T=t/N and the frequency of the signal is f = 1/T.

(iv) Measurement of phase: The two signals whose phase difference is to be determined are applied to the tow channels of the double bean oscilloscope. The same trigger signal is used for the two sweep voltages. The phase difference between the two waveforms can be found from the time base. If the two signals of time period T are found to attain the same phase at times t_1 and t_2, respectively, the phase difference between them is

$\phi = (2\pi/T)(t_1-t_2)$.

PROCEDURE:

1. Switch on the CRO and the function Generator.
2. Connect a sinusoidal signal of frequency 1000 Hz from the function generator to one of the two Y terminals of the CRO with the help of a cable with BNC terminals and observe the signal on the CRT screen.
3. Adjust the attenuator of the signal generator and bring the peak-to-peak voltage of the sinusoidal waveform to 1 volt. Adjust the frequency to 1kHz. Now adjust the sensitivity of the vertical section to 1 V/cm and the time base to 1 ms/cm. Adjust the trigger level. You will observe stationary display of the waveform.
4. To measure the voltage of the input signal, adjust the vertical amplifier sensitivity suitably, so as to get a sufficiently large display. Read the calibrated graticule, the vertical length of the display. This corresponds to

peak-to-peak value of the signal. Multiply this length to the sensitivity (in V/cm). Dividing this result by 2√2 gives the RMS value of the signal voltage. Repeat the measurement procedure for five values of the output signal voltages.

5. To measure the frequency of the signal, adjust the time base control of the CRO suitably so as to get about 2 to 3 cycles of the signal displayed on the screen. Rotate the vernier control to CAL position. Read the length of one cycle on the calibrated graticule on the screen. Multiply this by the time base setting (in msec/cm or μsec/cm). This gives the time period of the signal. Taking inverse of the time period gives the frequency of the signal.
6. Make measurement of the frequency of sinusoidal waveform at 100 Hz, 1kHz, 10kHz and 30 kHz,
7. Adjust the function generator to give square wave signal output.
8. Measure the width of the square waveform at above frequencies.
9. Connect s pulse signal of about 10 μsec pulse width and pulse-repetition frequency of 1000 from the pulse generator to the Y terminal. Measure the pulse width, peak voltage of the pulse and its prf.
10. Connect an unknown pulse waveform and measure
    1 Width of the pulse (in μsec.)
    2 Peak pulse voltage amplitude (in voltrs).
    3 Pulse repetition frequency (in Hz).

OBSERVATIONS AND RESULT:

1. Sinusoidal wave shape.

FREQUENCY	VOLTAGE P-P	TIME PERIOD	FREQUENCY
1000 Hz	4 Volts	1 msec.	1 kHz.
5000 Hz			
10,000 Hz			
30,000 Hz			

2.Square wave-shape

FREQUENCY	PEAK VOLTAGE	WIDTH	TIME PERIOD / FREQUENCY
500 Hz		.	

3.Pulse wave-shape

PULSE WIDTH	PEAK VOLTAGE	TIME BETWEEN. PULSES.	PULSE-REP. FREQUENCY

b
2

2

SEMICONDUCTOR DIODES CHARACTERISTICS

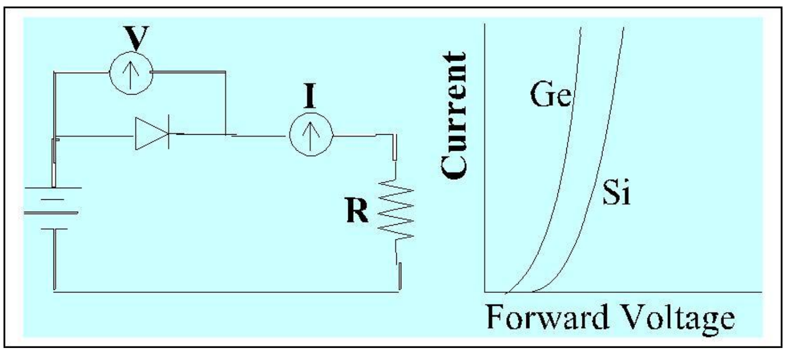
OBJECTIVES

1. To connect the circuit on the experimental board as per the circuit diagram.
2. To measure forward Voltage and Current characteristics of Silicon diode and plot the result.
3. To measure forward Voltage and Current characteristics of Germanium diode and plot the result.
4. To calculate dynamic resistance of the diodes.

APPARATUS REQUIRED:

A circuit board consisting of variable DC voltage supply, resistors and the silicon and Germanium diodes., voltmeter and milliammeter.

CIRCUIT DIAGRAM

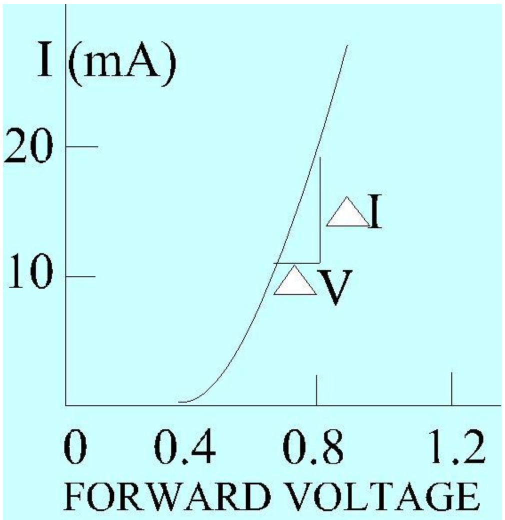


THEORY:

A semiconductor diode conducts in the forward bias condition. As the barrier potential at the junction decreases, majority carriers diffuse through the junction and current flows. A plot of voltage and current through the diode in the forward direction is called the forward characteristic of the diode. The static and dynamic resistance is determined at a fixed operating point of the diode.

$R_D = V / I$

dynamic $r_d = V / I$



Semiconductor Diode- forward characteristics.

PROCEDURE

1. Connect a milliammeter of 25 mA range and a voltmeter of 20 volt range in the silicon diode circuit with a series resistance of 1000 ohms.
2. Switch on the variable d.c. power supply and connect it in the forward direction. Increase the voltage slowly and measure the voltage and current. through the Silicon diode. Plot the result of a graph paper.
3. Repeat the experiment with Germanium diode and plot the result.
4. At a suitable operating points determine the static and dynamic resistance of the two diodes. .

OBSERVATIONS:

	SILICON DIODE		GERMANIUM DIODE	
Sr..	VOLTAGE	CURRENT	VOLTAGE	CURRENT
1				
2				
3				
4				
5				
6				

RESULT:

1. Plot of V-I characteristivs of Silicon and Germanium diodes separately.
2. Measure Static and Dynamic resistance values.C

Q: Why are the V-I characteristics of Silicon and Germanium diodes shifted in voltage scale ?

C

3 ZENER DIODE CHARACTERISTICS

OBJECTIVES

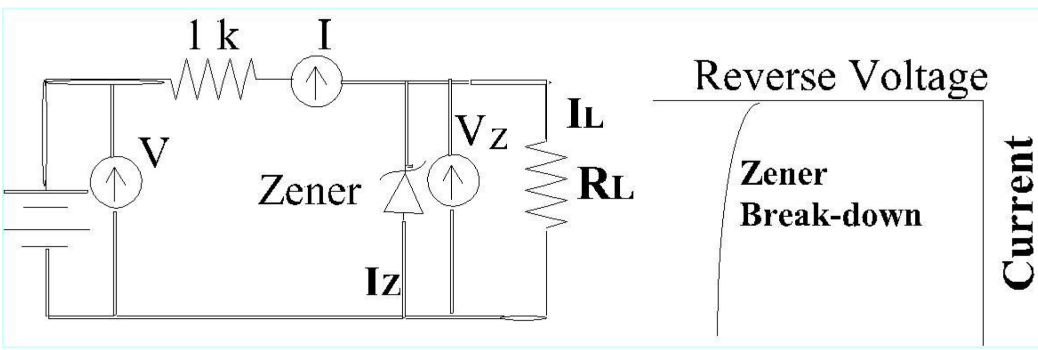
1. To connect the circuit on the experimental board as per the circuit diagram.
2. To measure reverse Voltage and Current characteristics of zener diode and plot the result..
3. To calculate dynamic resistance of the diode

.

APPARATUS REQUIRED:

A circuit board consisting of variable DC voltage supply, resistors and the silicon and Germanium diodes., voltmeter and milliammeter.

CIRCUIT DIAGRAM



THEORY

A p-n junction diode does not normally conduct in the reverse bias condition but at the break-down it starts conducting heavily. Zener breakdown occurs in heavily doped diodes having a narrow depletion region. For lightly doped diodes the breakdown voltage is greater and avalanche multiplication is predominant. In order that a large current does not damage the diode, a protecting resistance of 1 kilo-ohm value is connected in series with the diode. Once Zener diode starts conducting, it maintains constant voltage across its terminals whatever may be the current through it. It has very low dynamic resistance .

In the diagram, the supply voltage V and the series resisrance are so chosen that the diode current I_Z is within the specified range and the diode operates ij the Zener breakdown region.. The voltage V_Z across the load R_L remains constant although the supply voltage V and the load current I_L can vary. By Kirchoff's law:

$$I = I_L + I_Z \text{ and } V_Z = V - IR \text{ also } V_Z = I_L R_L .$$

In ideal case V_Z remains constant. If the load cuuent increases, the Zener current decreases so as to keep the supply current I constant. If, on the other hand, the lead resistance R $_L$ is kept constant and the supply voltage is changed, the supply current I and the Zener current I_Z change equally to maintain the load current I_L at a constant value.

PROCEDURE:

1. Connect a milliammeter and a voltmeter of suitable range. Connect a voltage supply to the diode in the reverse direction.
2. Switch on the power supply and increase slowly the supply d.c.voltage. Measure the input voltage and the voltage across the Zener and the current through it. Once break-down occurs, voltage across the diode remains fairly constant even though the current increases.
3. Plot graph between the voltage and the current .
4. Calculate the dynamic resistance of the diode.

OBSERVATIONS:

Take measurements on three zener diodes.

Sr.No.	INPUT VOLTAGE	ZENER VOLTAGE	ZENER CURRENT

RESULT:

1. Draw graph between voltage and current for three Zener diodes.
2. Calculation of the dynamic resistance (V / I) of each diode..

Q: Why do you call it a Zener and not Avalanche diode ?

d

4 HALF-WAVE DIODE RECTIFIER CIRCUIT

OBJECTIVES:

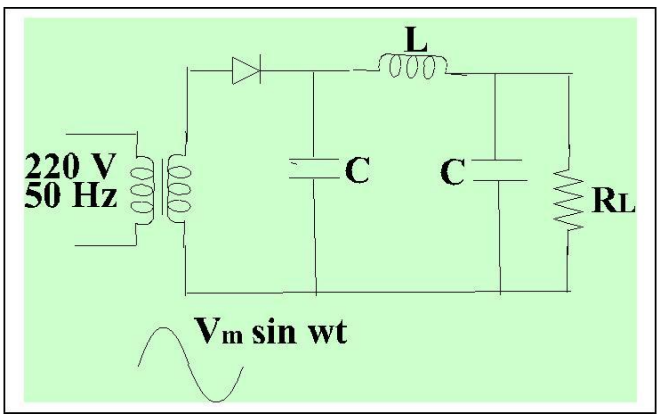
1. To connect the circuit as per the circuit diagram.
2. To draw electrical wave-shape at the input and the output points (as observed on CRO.)
3. To make measurements of AC voltages at the input and output terminals and the output DC voltage. To verify the theoretical expressions of V_{DC} and Ripple Factor.
4. To make measurements of the output DC and AC voltages with and without a shunt capacitor, a series inductor and π filter.To verify the theoretical expression for DC voltage and the ripple factor.,

APPARATUS REQUIRED:

A circuit board consisting of AC voltage supply, resistors and semiconductor diode.

Electronic multimeters and oscilloscope.

CIRCUIT DIAGRAM:



BRIEF THEORY:

In a half-wave rectifier circuit, the diode gets forward biased in the positive half cycle and it conducts. During the negative half cycle, it gets reverse biased and it does not conduct. The DC current flows through the load only during one half cycle. The DC voltage developed at the output is given by $V_{DC} = V_m / \pi$. and the ripple factor (the ratio of the AC to DC voltages at the output) is 1.21..

The output of the half wave rectifier contains an appreciable amount of AC voltage in addition to the DC voltage. It is desirable to filter out or smoothen out the AC variations from the output with the help of filters.

In a shunt capacitor filter, a high value capacitor is connected in shunt with the load. Capacitor offers a low impedance path to the AC components and most of the AC current passes through the capacitor and the DC current passes through the load. A series inductor filter offers high impedance to the AC component and low impedance to DC. As a result the output has very low AC component. A π filter utilizes the properties of both the inductor and the capacitor and the rectified output is almost free from AC components.

PROCEDURE OF EXPERIMENT:

1. Observe the waveshape of the input voltage and the output ripple voltage on a CRO and compare them.

2 Measure the AC voltage at the input and convert is to peak voltage.

3..Measure AC & DC voltages at the output of the half-wave rectifier unit across the load resistor.

4.Verify the DC voltage at the output using the expression $V_{DC}=V_m/\pi$.. and the value of the Ripple Factor of the half wave rectifier (1.21).

5.Measure AC and DC output voltages with and without a shunt capacitor filter. Repeat the same with the series inductor filter and the Pi filter.

OBSERVATIONS AND RESULT:

I Wave shape of the input voltage.

Waveshape of the output voltage.

II Input voltage (rms value) =

Output voltage (rms value) =

Output D.C. voltage =

III Verification of the formula:

OUTPUT DC VOLTAGE	V_M/π =	MEASURED VALUE
Ripple Factor	1.21	
Efficiency	Max.40.6%	

IV.FILTERS

:

NO FILTER	V_{AC} = V	V_{DC} = V
Shunt capacitor		
series inductor		
Pi filter		

The DC voltage approaches the peak input AC voltagee

5 FULL-WAVE BRIDGE RECTIFIER CIRCUIT

OBJECTIVES:

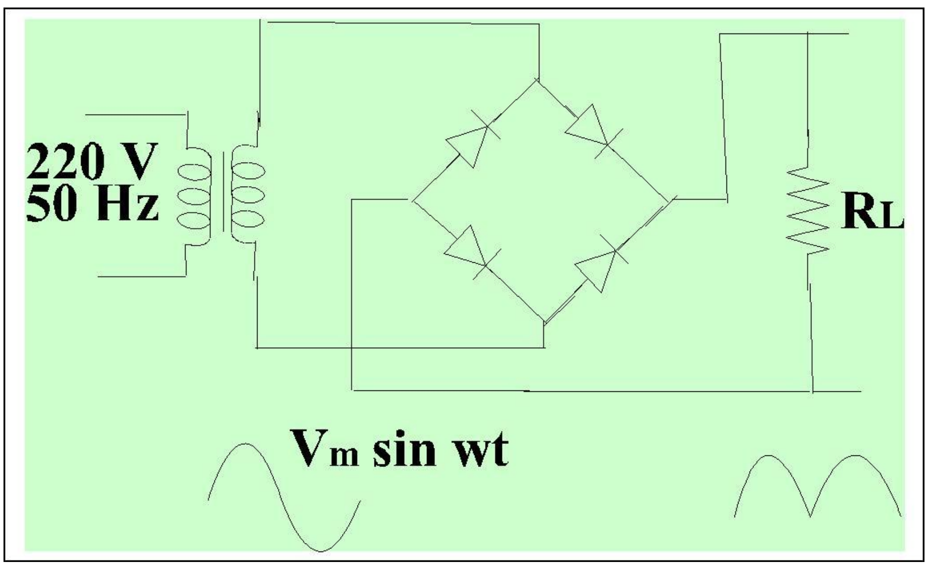
1. To connect the circuit as per the circuit diagram.
2. To draw electrical waveshape at the inpuyt and the output points (as observed on CRO.)
3. To make the following measurements using digital multimeter.
   a. AC voltage at the input terminals.
   b. AC voltage at the output terminals.
   c. DC voltage at the output terminals.
4. To verify the theoretical expression for DC voltage, Ripple factor and efficiency.

APPARATUS REQUIRED:

A circuit board consisting of AC voltage supply, resistors and four semiconductor diodes.

Electronic multimeters and oscilloscope.

CIRCUIT DIAGRAM:



BRIEF THEORY:

Full wave rectification is also achieved with a bridge circuit consisting of four semiconductor diodes. Only two diodes are involved in the rectification of one half-cycle of the input ac wave while the other two are reverse biased. In the next half cycle the other two diodes conduct simultaneously. During both the half cycles of the input voltage, $V_m \sin \omega t$, the load current flows in the same direction. It is unidirectional but fluctuating. The average value of the dc load voltage is $V_{DC} = (1/\pi) \int_0^{\pi} V_m \sin \omega t \, d(\omega t) = 2 V_m/\pi$, while the rms value of of the load voltage is $V_m / \sqrt{2}$, hence $V_{rms} / V_{DC} = 1.11$. The fluctuating component of the output D.C. is the ripple.The ripple factor has a value 0.482 and the conversion efficiency from ac to dc is 81.2% maximum.

The advantages of the bridge circuit over the conventional full wave rectifier are a) the input transformer does not require a centre tap terminal in the secondary output b) Peak Inverse Voltage (PIV) across the diodes is half c) bridge rectifier circuit is available as a block from manufacturers and is cheap.

PROCEDURE OF EXPERIMENT:

1. Observe the wave-shape of the input voltage and the output ripple voltage on a CRO and compare them.
2. Measure the AC voltage at the input of the bridge and convert is to peak voltage.
3. Measure AC and DC voltages at the output of the bridge rectifier unit across the load resistor.
4. Calculate the DC voltage at the output using the expression $V_{DC}=2V_m/\pi$. Compare this value with the measured value.
5. Calculate the value of the Ripple Factor using the measured V_{rms} and $V_{DC.}$ and compare with its value (0.482).
6. Calculate the value of the conversion efficiency and compare with the maximum efficiency of 81.2%

OBSERVATIONS AND RESULT:

I Wave shape of the input voltage.:
Waveshape of the output voltage :

II Input voltage (rms value) = V_m =
Output voltage (rms value) =
Output D.C. voltage =

III Verification of the formula:

OUTPUT DC VOLTAGE	$2 V_M/\pi$ =	MEASURED VALUE
Ripple Factor	0.482	Measured value
Efficiency	Max.81,2%	Measured value

AA

6

COMMON BASE TRANSISTOR CHARACTERISTICS

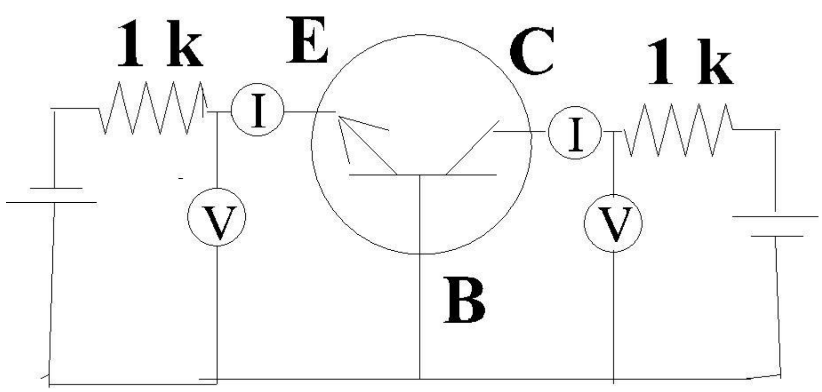
OBJECTIVES

1. To connect the circuit on the experimental board as per the circuit diagram.
2. To make measurements of the variation of emitter current with the emitter-base voltage at constant collector voltage and calculate input resistance.
3. To make measurements of the variation of collector current with collector-base voltage at constant emittercurrent and calculate the output resistance and α.

APPARATUS REQUIRED:

A circuit board consisting of AC voltage supply, resistors and the transistor., two 0-20 mA meters and multimeters.

CIRCUIT DIAGRAM:



THEORY:

INPUT CHARACTERISTICS:

This is a graphical presentation of the variation of emitter current with the emitter- base voltage at different fixed values of collector voltage, which are similar to the characteristics of a forward biased diode. The input characteristics are used to determine the input dynamic resistance of the transistor.

$$r_i = V_{EB} / I_E \text{ at constant } V_{CE}$$

OUTPUT CHARACTERISTICS:

A graphical plot of the collector current and the collector-base voltage at fixed values of emitter current represents the output performance of the transistor. The active region of the graph presents a picture of a constant current source pointing to a high output resistance. This can be used to determine the output dynamic resistance of the transistor:

$r_o = V_{CB} / I_C$ at constant I_E

The collector current $I_C = I_E - I_B + I_{CBO}$. When the output side is open and the emitter current is zero, there is a small collector reverse saturation current I_{CBO} flowing in the circuit.
The current gains of the transistor in CB mode are defined as
dc current gain = $\alpha_{DC} = I_C / I_E$ and ac current gain = $\alpha = I_C / I_E$ at constant V_{CE}

PROCEDURE OF EXPERIMENT:

1. Make the circuit connections as shown in the circuit diagram. Connect small dc milliammeters in the circuit and use multimeter for voltage measurements.
2. Set V_{CB} at zero volt. Vary V_{EB} in small steps (of 0.2 V). Measure and tabulate the values of V_{EB} and Emitter current.
3. Repeat the set of readings for V_{CB} = 1 volt. Take about four or five sets at different fixed values of collector-base voltages.
4. To draw output characteristics, set emitter current at 1 mA.
5. Vary collector-base voltage in steps and note down the values of collector current.
6. Change the value of emitter current and repeat the readings as above. Take four/five sets of readings.
7. Plot the tabulated readings of input characteristics and output characteristics of the transistor. Determine the input and output dynamic resistances; dc and ac current gains.
8. Determine the h parameters of the transistor.

OBSERVATIONS AND RESULT:

INPUT CHARACTERISTICS:

$V_{CB} = 2$		$V_{CB} = 5$		$V_{CB} =$	
V_{EB}	I_E	V_{EB}	I_E	V_{EB}	I_E

OUTPUT CHARACTERISTICS:

I_E = | I_E = | I_E =

V_{CB}	I_C	V_{CB}	I_C	V_{CB}	I_C

RESULT:

1. Input dynamic resistance =
2. Output dynamic resistance : =
3. DC Current Gain: =
4. AC Current Gain: =

F

7 COMMON EMITTER TRANSISTOR CHARACTERISTICS

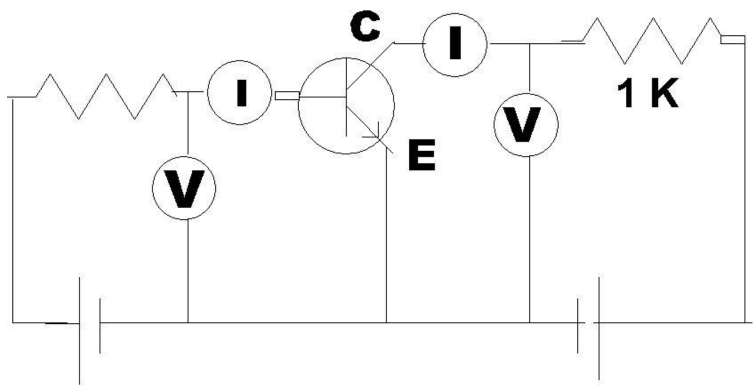
OBJECTIVES

1. To connect the circuit on the experimental board as per the circuit diagram.
2. To make measurements of the variation of base current with the base-emitter voltage at constant collector voltage. and calculate input resistance.
3. To make measurements of the variation of collector current with collector-emitter voltage at constant base current and calculate the output resistance and β.

APPARATUS REQUIRED:

A circuit board consisting of AC voltage supply, resistors and the transistor., mA meters and multimeters.

CIRCUIT DIAGRAM:



BRIEF THEORY:

In a transistor, the emitter-base junction is forward biased so that the impedance of the emitter circuit is low and a current flows between the emitter and the base. A change in the emitter current produces a change in the collector current, giving the transistor action. In the common emitter mode of operation, the emitter terminal is common between the input and output circuits. For the CE mode, I_B is the input current, V_{BE} is the input voltage and V_{CE} is the output voltage. The input characteristics constitute a graph of I_B plotted against V_{BE} with V_{CE} as a parameter. The characteristics are similar to that of a forward biased p-n diode.

The output characteristics for the common emitter mode refer to the plot of the collector current I_C versus the collector-emitter voltage V_{CE} with the base current I_B as a parameter. The curves in the active region are nearly horizontal because for a fixed $I_{B,}$, $| I_C |$ increases with $| V_{CE} |$ due to the base width modulation or the Early effect. The important parameters derived from the input and output characteristic curves are:

1. Input dynamic resistance, r_i = . V_{BE} / I_B at constant V_{CE}
2. Output ac resistance, r_o = V_{CE} / I_C at constant I_B.
3. DC Current Gain, $\beta_{dc} = I_C / I_B$
4. AC Current Gain, β = I_C / I_B at constant V_{CE}

:PROCEDURE OF EXPERIMENT:

1. Make the circuit connections as shown in the circuit diagram. The resistor in the Base circuit is 100 k. Connect small dc meters in the circuit and use multimeter for measurement of base current (micro-amps.).
2. For input characteristics measure Base-Emitter voltage and current at constant collector voltage. Take two sets of readings:
3. Set base current at 10 micro-amps and measure collector voltage and current. Repeat the measurements a different base currents. Take three sets of readings.
   Plot the output characteristics of the transistor,
4. Determine the input and output dynamic resistances; dc and ac current gains.

OBSERVATIONS AND RESULT:

INPUT CHARACTERISTICS:

V_{CE} =		V_{CE} =		V_{CE} =	
V_{BE}	I_{BE}	V_{BE}	I_{BE}	V_{BE}	I_{BE}

OUTPUT CHARACTERISTICS:

I_{BE} =		I_{BE} =		I_{BE} =	
V_{CE}	I_C	V_{CE}	I_C	V_{CE}	I_C

RESULT: 1. Input dynamic resistance = 2
Output dynamic resistance : =
3. DC Current Gain: =
4. AC Current Gain: =

a

8

CHARACTERISTICS OF A FIELD EFFECT TRANSISTOR.

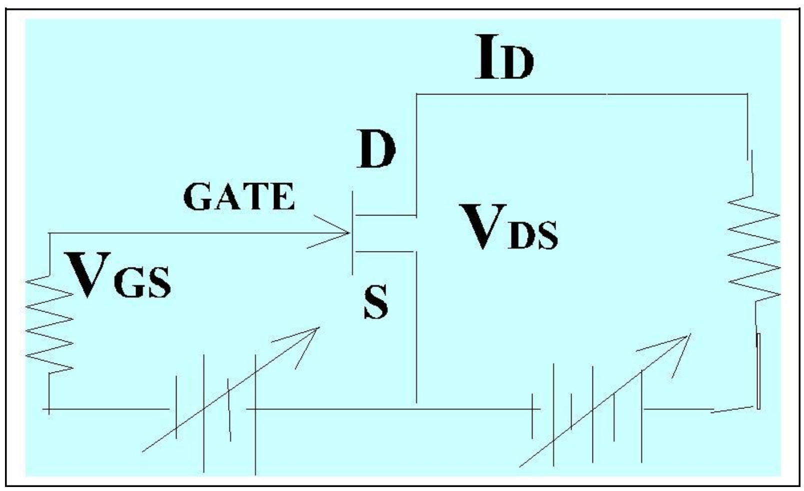
OBJECTIVE:

1. To trace the circuit diagram.
2. To plot the static drain characteristics of FET.
3. To calculate the FET parameters (r_d, g_m, μ) at a given operating point.

APPARATUS REQUIRED:

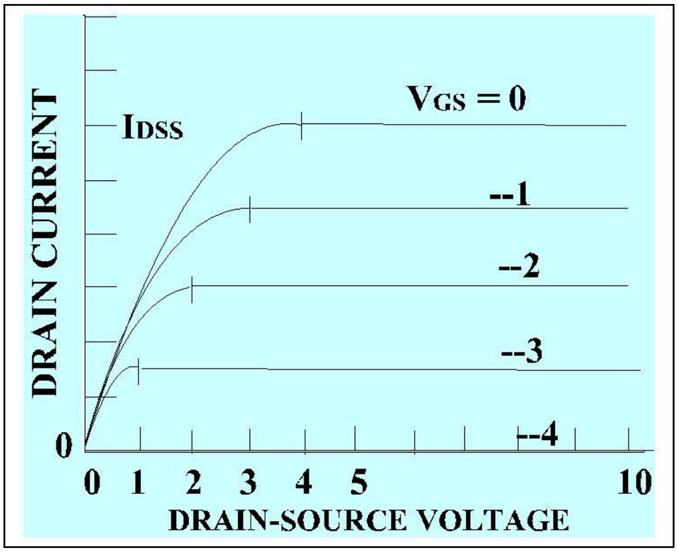
Experimental Board, two voltmeters and milliammeters..

CIRCUIT DIAGRAM



THEORY

A field effect transistor is a unipolar three terminal device like an ordinary bipolar transistor. It functions only with one type of carrier. It can be either an n-channel or a p-channel FET. The family of FET consists of J-FET, MOSFET (IGFET), MESFET and HEMT. The present experiment involves an n-channel Junction Field Effect Transistor type BFW 11. This comprises of an n-type semiconductor (silicon) bar; the two ends of which make the source and the drain terminals. On the two sides of the bar, p-n junctions are made. The p -regions make GATEs. The two gates are joined together to form a single gate. The drain terminal is given a positive potential with respect to Source and the Gate is reverse biased with respect to Source. The output I-V characteristics of the device are shown in the following diagram at constant gate-source bias voltage.



The characteristics give us the following information:

1. If V_{GS} is kept at a constant value (zero or negative). the drain current I_D initially increases with V_{DS} till the voltage approaches the channel pinch-off voltage, when the current becomes almost constant. As V_{GS} is kept constant at progressively higher negative values, the corresponding values of V_P also decreases.
2. If V_{DS} is kept at a fixed value and V_{GS} is biased more and more negative, I_D decreases till it is reduced to zero at a certain value of V_{GS} ,called V_{GS} Off.

Since the gate voltage controls the channel current, JFET is called a voltage controlled device. The static characteristics of JFET are

a) drain V-I characteristics and
b) transfer characteristics (V_{DS} vs. I_D)..

SMALL SIGNAL J-FET PARAMETERS:

1. AC Drain Resistance, r_d : : It is the ac resistance between the drain and source terminals when J-FET is operating in the pinch off region.

It is given by r_d = [change in V_{DS} / change in I_D] at constant V_{GS}

= [V_{DS} / I_D] at constant V_{GS}

This is also called dynamic drain resistance

2. Transconductance, g_m: :It is the slope of transfer characteristics and is called forward transconductance..

g_m = [I_D / V_{GS}] at constant V_{DS}

3. Amplification Factor: It is given by μ = [V_{DS} / V_{GS}] at const. I_D

The three parameters of J-FET are related by the equation:

$$\mu = g_m \times r_d$$

DC LOAD LINE:

DC load line for JFET can be easily drawn between two extreme points:

1) At $I_D = 0$, $V_{DS} = V_{DD}$

2). At $v_{DS} = 0$, $I_D = V_{DD} / R_L$

The Q point is generally situated at the middle of the load line.

PROCEDURE:

1. Make the circuit connections as shown in the circuit diagram.
2. Start the experiment by fixing $V_{GS} = 0$.
   Increase drain voltage in steps and note down the values of the drain current at each step.
3. Change the value of V_{GS} to a low negative value and repeat the observations as in 2 above.
4. Take five sets of observations.
5. Plot the Drain voltage-current characteristics on a graph paper.
6. Calculate the FET parameters.

OBSERVATIONS:

DRAIN CURRENT

V_{DS}	$V_{GS} = 0$	$V_{GS} = -1$	$V_{GS} = -2$	$V_{GS} =$	$V_{GS} =$
1 V					
2 V					

Plot the observations.

CALCULATIONS: and RESULT

At the Q point, $V_{DS} =$ V and $V_{GS} = -$ V

1. $r_d =$ kΩ, 2. $g_m =$ mS 3. μ=

iI

9 STUDY OF HARTLEY OSCILLATOR CIRCUIT

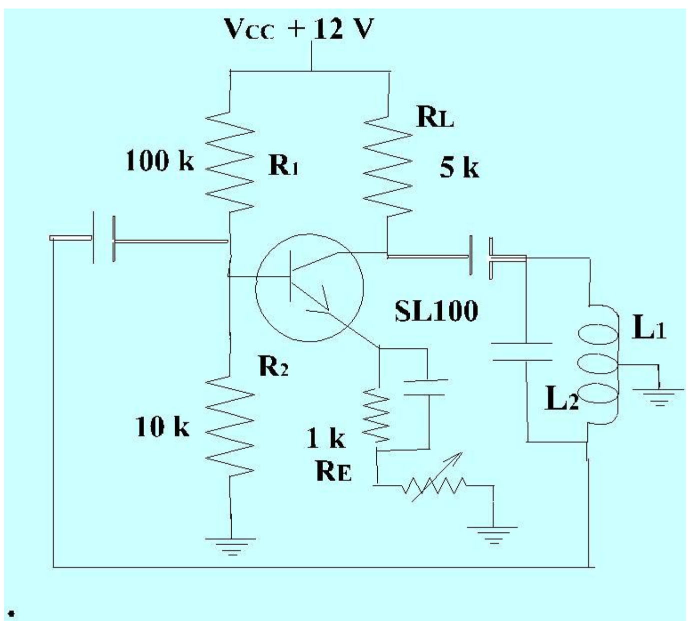
OBJECTIVE: 1. To trace the circuit diagram.

2. To measure the voltages and currents in the circuit and determine the Q point.
3. To measure the frequency of the oscillator with one set of L and C on CRO.
4. Measure the oscillator frequency with the other set of L and C on CRO voltage
5. Compare the calculated and observed values of frequency.

APPARATUS REQUIRED:

Hartley oscillator circuit, Multimeter, CRO.

CIRCUIT DIAGRAM



THEORY:

The circuit diagram of a Hartley oscillator is shown in the fig. It consists of a transistor amplifier (CE configuration) and a positive feedback loop. The open loop gain of the amplifier is $A = V_o/V_1$. If V_o' is the output voltage with feedback, then a fraction $\beta V_o'$ is fed back to the input making input voltage as $V_1+ \beta V_o'$. This positive feedback is in phase with the input voltage. The output will then be $A(V_1+ \beta V_o') = V_o'$. The new gain of the amplifier is $A' = A/(1+ \beta A)$. Since the positive feedback increases the amplifier gain, it is called regenerative feedback and the amplifier becomes an oscillator. The conditions are 1) feedback should be positive and 2) the feedback factor is unity.

Hartley oscillator is a tuned collector oscillator. It uses a single tapped coil having two parts L_1 and L_2. One side of L_2 is connected to base via C and the other to emitter via ground and C. One end of L_1 is connected to collector via C and the other end to common emitter terminal. L_1 is in the output (collector-emitter circuit) and whereas L_2 is in base- emitter circuit (input). The two parts are inductively coupled and form a split tank circuit. Feedback between output and input circuit is accomplished with phase reversal of 180^o. Since the transistor amplifier in CE configuration itself introduces a phase shift of 180^o , the total phase shift becomes 360^o or 0^o making the feedback positive and regenerative, so essential for oscillator. The feedback factor is given by the ratio of turns in L_2 and L_1 by N_{22}/N_1. This value is about 0.1 to 0.5.

The DC operating point on the active region of the transistor is established by R_1, R_2, R_E and V_{CC}. The load line is defined by V_{CC} and $I_{C\,Sat} = V_{CC}/(R_C + R_E)$ The oscillation frequency of the Harley circuit is given by the resonant circuit: $f_o = 1/2\pi\sqrt{LC}$

PROCEDURE:

1. Study the circuit and determine the operating point of the transistor.
2. The Hartley oscillator oscillates in two frequencies depending on the value of inductance of the tank circuit. There are two sets of inductors L_1 - L_2 and L_1' – L_2' . The net inductance of the first set is 2 mH and that of the other set is 20 mH.
3. Connect L_1 - L_2 inductors in the resonant circuit.
4. Observe the output on CRO and determine the frequency of oscillations.
5. Adjust R to get undistorted output.
6. Calculate the resonance frequency $f_c = 1/[2\pi\sqrt{LC}\,]$ where L= 2 mH and C = 0.01 μF. Compare the two values.
7. Disconnect L_1 - L_2 and connect L_1' - L_2'.
8. Calculate the frequency and observe the output on CRO. Compare the two values.

OBSERVATIONS AND RESULT:

SET	TIME PERIOD MEASURED ON CRO	FREQUENCY	CALCULATED FREQUENCY.
1			
2			

DISCUSSIONS:

J

10 STUDY OF COLPITTS OSCILLATOR CIRCUIT

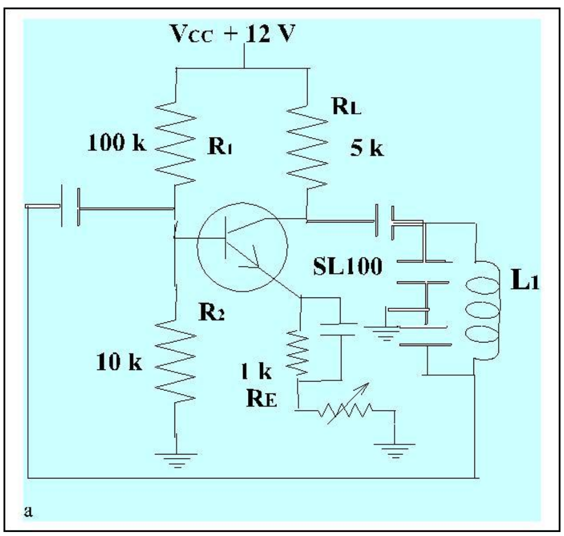
OBJECTIVE:

1. To trace the circuit diagram.
2. To measure the voltages of the circuit and determine the Q point
3. To measure the frequency of the oscillator with one set of L and C on CRO
4. Measure the oscillator frequency with the other set of L and C on CRO
5. Compare the calculated and observed values of frequencies.

APPARATUS REQUIRED:

Colpitts oscillator circuit,
Two sets of capacitors, One inductance 20 mH
Multimeter, CRO.

CIRCUIT DIAGRAM



THEORY

The circuit diagram of a Colpitts oscillator is shown in the fig. It consists of a transistor amplifier (CE configuration) and a positive feedback loop. The open loop gain of the amplifier is $A = V_o/V_1$. If V_o' is the output voltage with feedback, then a fraction βV_o' is fed back to the input making input voltage as $V_1+ \beta V_o$'. This positive feedback is in phase with the input voltage. The output will then be $A(V_1+ \beta V_o') = V_o'$. The new gain of the amplifier is $A' = A/(1+ \beta A)$. Since the positive feedback increases the amplifier gain, it is called regenerative feedback and the amplifier becomes an oscillator. The conditions are 1) feedback should be positive and 2) the feedback factor is unity.

Colpitts oscillator is a tuned collector oscillator. It uses a single tapped capacitorl having two parts C_1 and C_2. One side of C_2 is connected to base via L and the other to emitter via ground and L. One end of C_1 is connected to collector via L and the other end to common emitter terminal. C_1 is in the output (collector-emitter circuit) and whereas C_2 is in base-emitter circuit (input). The two parts are capacitatively coupled and form a split tank circuit. Feedback between output and input circuit is accomplished with phase reversal of 180^o. Since the transistor amplifier in CE configuration itself introduces a phase shift of 180^o , the total phase shift becomes 360^o or 0^o making the feedback positive and regenerative, so essential for oscillator.

The DC operating point on the active region of the transistor is established by R_1, R_2, R_E and V_{CC}. The load line is defined by V_{CC} and $I_{C\ Sat} = V_{CC}/(R_C + R_E)$ The oscillation frequency of the Colpitts circuit is given by the resonant circuit:

$$f_o = 1/2\pi\sqrt{LC}$$

PROCEDURE:

1. Study the circuit and determine the operating point of the transistor.
2. The Colpitts oscillator oscillates at two frequencies depending on the value of capacitance of the tank circuit. There are two sets of capacitors C_1 - C_2 and C_1' – C_2' . The inductance is 20 mH.
3. Connect C_1 - C_2 capacitors to the Colpitts oscillator circuit.
4. Observe the output on CRO and determine the frequency of oscillations.
5. Adjust the emitter resistance R to get undistorted output.
6. Calculate the frequency and observe the output on CRO. Compare the two values.

$$f_o = 1/2\pi\sqrt{LC}$$

L = 20 mH

$C = C_1C_2 / [C_1 + C_2] = 0.01 \times 10^{-6} \times 0.01 \times 10^{-6} /[0.02 \times 10^{-6}] = 0.005 \times 10^{-6}$ $f_o = 1/2\pi\sqrt{LC} = 16$ kHz.

7. Now disconnect C_1, C_2 and connect C_1', C_2' and calculate the frequency of the output waveform.
8. Compare the practical frequency with the theoretical frequency.

OBSERVATIONS AND RESULT:

SET	L	C	FREQ.	TIME PERIOD ON CRO.	FREQUENCY.
1					
2					

11 STUDY OF PHASE-SHIFT OSCILLATOR CIRCUIT

OBJECTIVE:

1. To trace the circuit diagram.
2. To measure the voltages of the circuit and determine the Q point
3. To measure the frequency of the oscillator with phase shift circuit connected in the feedback loop.
4. Measure the oscillator frequency.
5. Compare the calculated and observed value of frequency..

APPARATUS REQUIRED:

Phase shift oscillator circuit,
Multimeter, CRO

PROCEDURE:

1. connect the circuit as indicated in the circuit board.
2. Display the frequency waveform on CRO.
3. Measure the frequency and compare with the theoretically calculated value from the values of R and C.

EXPERIMENT AND RESULT:

1. Time period of the waveform on CRO.
2. Measured frequency.
3. Calculated frequency.

12. STUDY OF LOGIC GATES "NOT"; "OR"; "AND" AND VERIFY THEIR PERFORMANCE.

AIM:

1. To study logic gate "NOT" in IC-7404 and verify its performance.
2. To study logic gate "OR" in IC-7432 and verify its performance.
3. To study logic gate "AND" in IC-7408 and verify its performance.

EQUIPMENT REQUIRED

BREAD-BOARD WITH LEDs, 5V POWER SUPPLY, IC-7404. IC-7408, IC-7432. , FUNCTION GENERATOR, DOUBLE BEAM OSCILLOSCOPE...

CIRCUIT DIAGRAM: PCM KIT

THEORY:

Logic gates are the basis building blocks for forming digital electronic circuitry. A logic gate has an output and one two or more input terminals. The output of the gate would be HIGH (1) or LOW (0) depending on the digital levels at input terminals. The logic gates are used to design digital systems that will evaluate digital input levels and produce a specific output response based on that logic circuit d3sign.

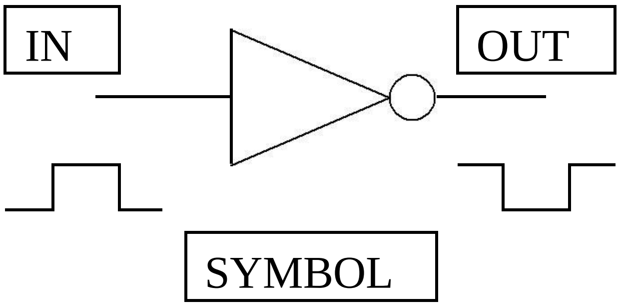
We shall study the following GATES:

1. INVERTER GATE (NOT GATE)
2. OR GATE
3. AND GATE

The INVERTER, OR & AND GATES are considered to be the basic logic gates in the field of digital electronics. In the experiment, +5 V represents HIGH and 0 V represents LOW levels.

"INVERTER" GATE:

The INVERTER is also known as NOT GATE. It is a logic circuit that performs INVERSION operation. It changes one logic level to the opposite logic level i.e. from HIGH to LOW or from LOW to HIGH level. In terms of bits, it changes a 1 to 0 and a 0 to 1.The logic gate symbol and the associated TRUTH TABLE are given below:



TRUTH TABLE

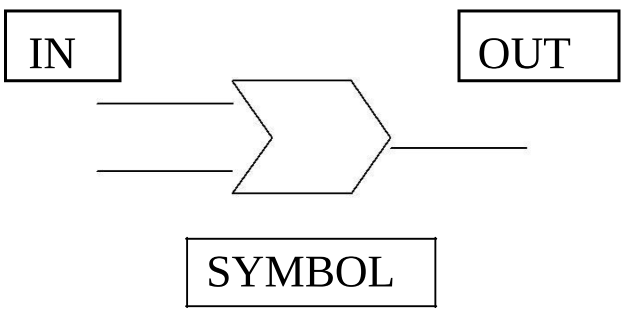
A	B
0	1
1	0

Boolean expression for the INVERTER, if A is the input variable and Y is the output variable, is Y = A'

"OR" GATE:

The OR is also known as the 'INCLUSIVE OR' Gate. It is a logic circuit that produces an output of 1 if either or both the inputs of thecate are 1 or any of the input is 1. In case both the inputs are 0 , the output is 0. The function is denoted by +. Thus A+B is stated as A or B.

The logic gate symbol and the associated TRUTH TABLE are given below:



Y = A + B
OR GATE

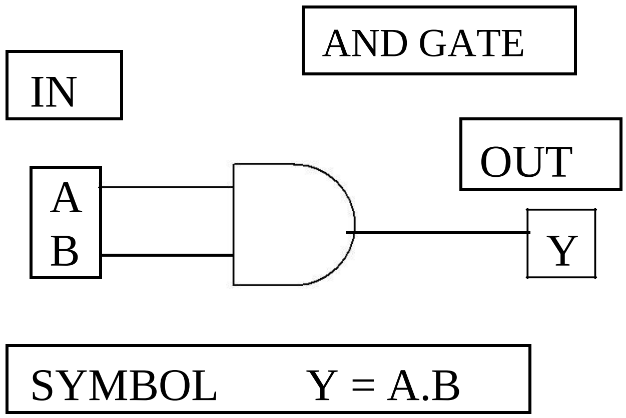
TRUTH TABLE

INPUT		OUTPUT
A	B	Y
0	0	0
0	1	1
1	0	1
1	1	1

"AND" GATE:

The AND GATE has two binary inputs and it produces one output. The output is 1 only when both the inputs are 1 otherwise the output is 0.

The logic gate symbol and the associated TRUTH TABLE are given below:



TRUTH TABLE

A	B	Y
0	0	0
0	1	0
1	0	0
1	1	1

PROCEDURE

1. Select the appropriate IC and fix it on the breadboard. The notch of the IC is used to identify the numbering of the different pins.

2. Connect +5 Volt (V_{CC}) to pin no. 14 and the pin no.7 is grounded.

3. Check the various gates of the IC.
4. For each gate except the NOT gate, pins 1 and 2 are used for input and pin 3 for output.
5. NOT gate uses pin no.1 for input and pin no. 2 for output.
6. Input 0 is provided by connecting to ground.
7. Input 1 is provided by connecting to pin.14.
8. For each gate, design the circuit using connecting wires and verify the output corresponding to the TRUTH TABLE. of the respective GATE. The output is shown by LEDs.
9. High output LED glows RED

   LOW output LED glow GREEN.

PRECAUTIONS:

1. The ICs should be checked before use.
2. Tight connections should be made.
3. Overheating should be avoided.
4. TTL ICs require +5 V and ground supply.

OBSERVATIONS AND RESULT:

It has been checked and found that all the three LOGIC GATES follow the TRUTH TABLE.

NOT GATE

INPUT A	OUTPUT LED GLOW B

OR GATE

INPUT A	INPUT B	OUTPUT LED GLOW Y

AND GATE

INPUT A	INPUT B	OUTPUT LED GLOW Y

13

SINGLE STAGE C. E. TRANSISTOR AMPLIFIER.

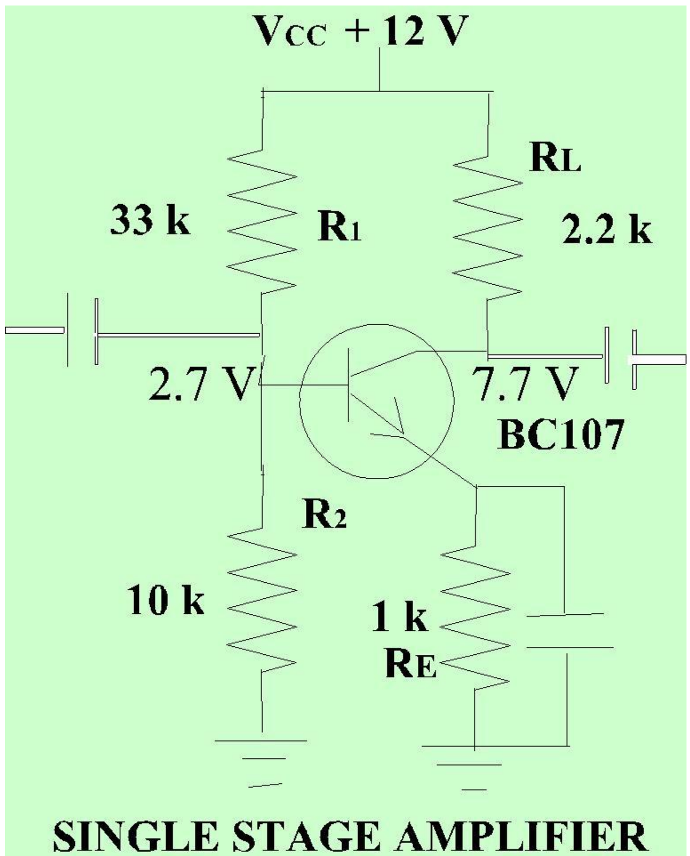
OBJECTIVES: 1. To trace the circuit diagram.

2. To measure the voltages and currents in the circuit and determine the Q point.
3. Measure the voltage gain at 1 kHz.
4. Plot the frequency response.

APPARATUS REQUIRED:

Amplifier, Multimeter, CRO.

CIRCUIT DIAGRAM

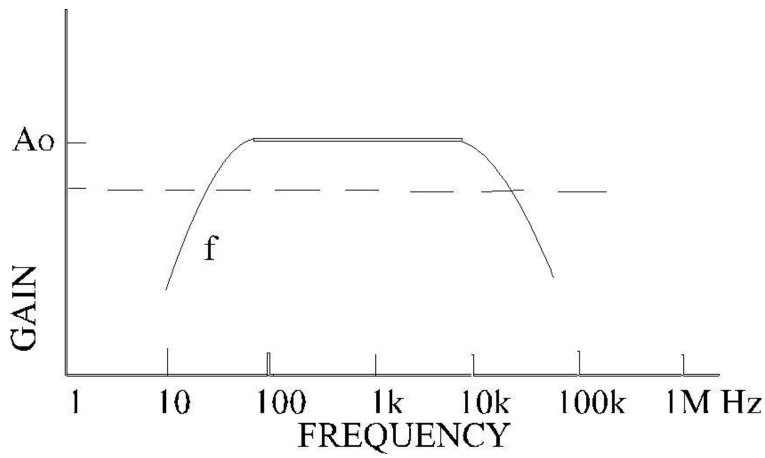


THEORY

The circuit of the single stage amplifier is based on NPN silicon transistor BC107 and voltage divider biasing circuit, The operating point (Q point) of the amplifier is fixed by the resistors R_1, R_2 , R_L and $R_{E.}$

The resistor R_E stabilizes the circuit against temperature variations. Capacitor across emitter resistor bypasses the A.C. signal. In a multi -stage amplifier, two stages can be coupled by RC coupling scheme. This is the most widely used method in audio small signal amplifiers. The voltage gain of an RC coupled amplifier is maximum around 1 kHz. As frequency decreases gain starts falling mainly because of coupling capacitor C_C, which offers sufficiently high impedance. and bypass capacitors at low frequencies are no longer effective shortcircuits. Because of this an ac.current passes through R_E . This gives rise to negative feedback and the voltage gain reduces. The voltage gain also decreases at high frequencies because of i) the shunt capacitances made up of junction and wiring capacitances and ii) the decrease of β at such frequencies.

The frequencies where voltage gain falls by 3 dB or becomes 70.7% of the maximum value are called cut off frequencies. These frequencies are determined from the frequency response curve.as shown in the Fig.



PROCEDURE:

1. Draw a neat circuit diagram.
2. Connect V_{CC} : +12 V. Measure the d.c. voltage.
3. For the measurement of the quiescent collector current, measure the voltage of the collector terminal (V_c). Calculate the collector current from the formula $I_c = (V_{cc} – V_c) / R_c$.. Note down the voltages and current on the circuit diagram and the Q point..
4. Feed 1 kHz signal to the input of the amplifier as well as to Channel 1 of the CRO. The output of the amplifier is connected to channel 2 of CRO. Observe both the input and output waveforms on the oscilloscope screen and note the phase-shift between the input and output waveshape.
5. Keep the input signal in the range of millivolts. Increase the input signal till the output waveform shows distortion. Measure the input signal strength. This is the maximum signal that the amplifier can amplify without giving distorted output.
6. Note down the input and output voltages and determine the voltage gain of the amplifier at 1 kHz frequency.

7. Measure the frequency response of the amplifier by measuring the output voltage at different frequencies from 100 Hz to 300 kHz. Keep the input voltage constant throughout the measurement.
8. Plot the frequency response on a graph paper and determine the lower and higher cut-off frequencies.

OBSERVATIONS

1. Q Point of the transistor.
   V_{CC} = 12 Volts.
   V_{CE} = ...
   I_C = ...
2. (a) Input signal to the amplifier for undistorted output =
   (b) Voltage gain at 1 kHz. =
   (c) Approximate value of f_1 = Hz.
   (d) Approximate value of f_2 = kHz.

RESULTS:

1. A plot of the frequency response of the amplifier. 2.Values of voltage gain, lower and upper frequency cut-off .

QUESTIONS:

1.Why do you use decoupling capacitor at the input and output?
2.What is the purpose of capacitor across emitter resistor?
3.What will happen if a) R_1 is shorted b) R_1 is open, c) R_2 is shorted and d) R_2 is open circuited ?

GURU GOBIND SINGH
INDRAPRASTHA UNIVERSITY
Kashmere Gate, Delhi – 110 006

Guru Gobind Singh
Indraprastha University

Laboratory Experiments

Electronic Devices and Circuits

M.Tech.(Integrated) Semester III Code IT-251

Name:....................................

Roll. No.................................

School of Information Technology
Aug – Nov 2009

Laboratory Experiments

M.Tech.(Integrated) Semester III Code IT-251
Electronic Devices and Circuits

LIST OF EXPERIMENTS

Dr. PREM SWARUP ___________________