Salahaddin University-Hawler

College of Education

Physics Department

Electronics Laboratory

Student Name:

Stage: Third

Address:

Electronic Lab.

(2014/2015)

Contents

No.	Experiment Name	Page No.
1.	Silicon diode, Plotting Characteristics Curves	2
2.	Zener diode, Plotting Characteristic Curves	4
3.	Half-wave Rectification and Filtering	6
4.	Centre-point connection for full-wave Rectification	8
5.	Bridge circuit for Full-wave Rectification and Filtering	10
6.	Voltage Doublers	11
7.	Clipping Circuits	12
8.	Clamping Circuits	13
9.	Characteristic curves of Transistor using Point-by-Point Method	14
10.	Common emitter amplifier	16
11.	Common base amplifier	19
12.	Common Collector Amplifier	23
13.	Two-stage amplification using RC connection	25
14.	Differential Amplifier	26
15.	RC phase-shift oscillator	28
16.	Colpitts LC Oscillator	30
17.	HARTLY LC OSCILLATOR	32
18.	Pulse Generator(IC 555)	34

Experiment (1)
Silicon diode, Plotting Characteristics Curves

General

Silicon diodes are semiconductor diodes. Since the maximum permissible temperature is 150°C, these diodes used for considerably higher powers than germanium diodes. Moreover, silicon diodes have a considerably higher reverse resistance. At a certain inverse voltage, the reversed biased silicon diode suddenly breaks down. The breakdown normally leads to the destruction of the diode. Silicon diode 1N4007, for example, have breakdown voltage 1000 V and permissible maximum current one A for the forward bias.

Circuit diagram

Figure 1 shows the circuit diagram. It shows with a voltage source of 15 V for the forward direction. It is regulating at potentiometer P1 by means of resistor R1 so that the current will not exceed a maximum of 70 mA.

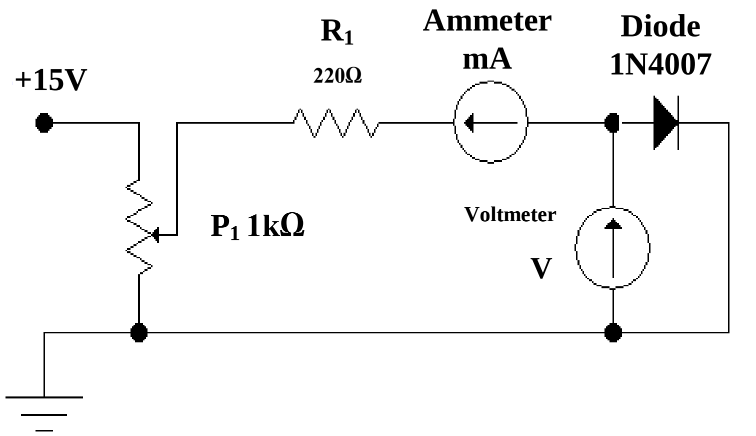


Fig.(1): Diagram of circuit in forward biased.

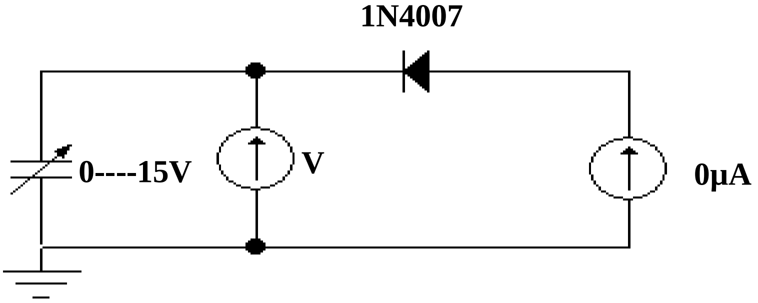


Fig.(2): Diagram of circuit in reverse biased.

Performance of the experiment

A- Forward Biases

1. set up the experiment as in fig.(1). Pay attention to the polarity of the diode and the 15 V power supply. Set the wiper of the potentiometer P1 to the left-hand stop (V=0) (check without diode being plugged in).
2. Set the wiper of P1 to the right-hand in steps and write the values of V and I. Plot these values as I=f(V).

B- Reverse Bias

1. set up the experiment as in fig.(2). Pay attention to the polarity of the diode and the 15 V power supply. Set the wiper of the potentiometer P1 to the left-hand stop (V=0).
2. Set the wiper of P1 to the right-hand in steps and write the values of V_R and I_R. Plot these values on the same graph paper as $I_R=f(V_R)$.

Discussion:

1. The current has very small values in the forward bias and then it increases toward very large values suddenly. Why?
2. The voltage reaches to nearly constant values in the forward bias. Why? Is this value varying by varying the type of the diode?
3. Does the current pass through in the reverse bias? Why? In addition, is these phenomena whether benefit or not? How?

Experiment (2)
Zener diode, Plotting Characteristic Curves

General

Zener diodes are special silicon diodes. In the forward direction, there is no difference between its characteristic and that of a normal silicon diode. When it operated in the reverse direction, a very small reverses current flows, which does not increase at first, even when the inverse voltage increased. However, if the inverse voltage exceeds a certain value, the Zener diode suddenly becomes conductive. Up to certain values, this effect is due to the Zener effect. What is more, the so-called avalanche effect, which also makes the diode capable of conducting in the reverse direction, then also manifests itself. This effect will not destroy the diode until a permissible rise in temperature has taken place and it is therefore use for stabilization.

The ratio of the change of voltage in reverse direction to the change of current due to this effect called the Dynamic resistance r_z. The breakdown point is determined by the doping, and the voltage at which that occurs called the Zener voltage V_z.

Aim of the Experiment:

To set up a given circuit, measure voltage and current and draw the characteristic curve of Zener diodes (ZF4.7 & ZX10) in the reverse direction.

The characteristics of the diodes used are:

Characteristics \ Diodes	ZF4.7	ZX10
V_z	4.4…5 V	9.4…10 V
r_z	7 Ω	4 Ω
I_z max	180 mA	105 mA
I_z max(with heat sink)	-	800 mA

Apparatus and Components

1. Mains power supply unit, DC +15 V
2. Potentiometer, 1 KΩ
3. Resistor, 100 Ω
4. Millimeters (Voltmeter, Ammeter)

Performance of the Experiment:

1. Zener Diode ZF4.7

1. Set up the circuit as in Fig.(1) and set the wiper of P1 to the left-hand stop.

2. Set the wiper of P1 to the right-hand to get currents as in the table(1) and write the corresponding voltages (Values have negative signs).

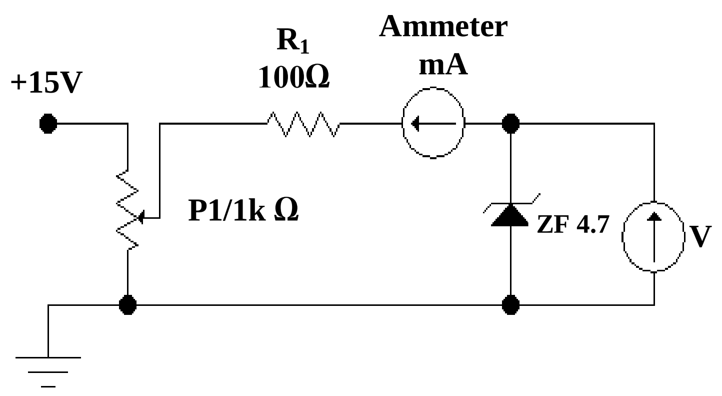


Fig.(1): Circuit diagram, which used to plot the characteristic Curve of a Zener diode.

Table (1): Measured values of a ZF4.7 Zener diode.

-I/mA	0.1	0.5	1	2	5	10	25	30	…	100
- V/v										

2. Zener Diode ZX10

Set the Zener diode ZX10 in circuit (Fig.(1)) and repeat the same steps for the ZF4.7, then write the results in a similar table.

Calculations:

1. Plot the two results on the same graph paper as I=f(V) (remember that the results I and V have negative values).
2. Calculate the dynamic resistance r_z from the relation $r_z=\Delta V/\Delta I$ for the following values of currents:

ZF4.7 5-------20 mA
20------35 mA
ZX10 15------45 mA

Discussion:

1. What is your conclusion about the curve?
2. What is your conclusion concerning dynamic resistance for ZF4.7 and ZX10?
3. It known that Zener diode used for voltage regulation. Why?
4. What is the major factor in which determine the quality of regulation from your results?

Experiment (3)

Half-wave Rectification and Filtering

Aim of the Exp.:

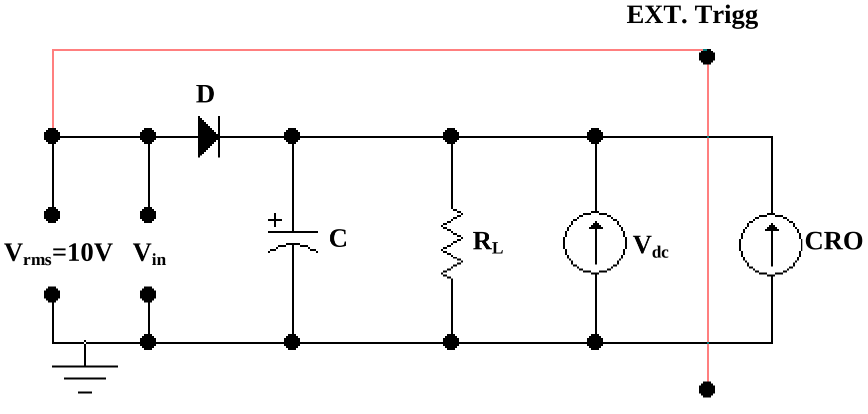
1. To be able to construct a circuit with a transformer, diode, load resistor and reservoir capacitor for half-wave rectification of an A.C. voltage.
2. To be able to measure the output voltage as a function of time with the Oscilloscope and as direct voltage with the moving coil meter V_{dc}.
3. To be able to recognize the dependence of the output voltage in respect of its curve shape and magnitude upon the size of the reservoir capacitor and load resistor.

Apparatus and Components

1. Resistors: 100 Ω, 1 kΩ
2. Electrolytic capacitors: 10 µF, 100 µF
3. Diode: 1N4007
4. Mains power supply, A.C.
5. Double-beam Oscilloscope
6. Multimeter, high resistance

Performance of Exp.

1. The circuit is to be set up as in Fig.(1). The resistor and capacitor removed initially.



$Y_{in\ put}$:	D.C
$V_{diflection\ sensitivity}$:	5 V/Cm
$T_{base\ sweep\ time}$:	5 m.sec/Cm
Triggering:	external, +

Fig.(1): Set-up of the experiment circuit.

To measure ripple voltage $V_{p\text{-}p}$ switch oscilloscope to A.C. and select a suitable Y deflection sensitivity.

2. The input and output voltages (without C and R_L) visible on the oscilloscope screen as a function of time must be drawn in on the graph. Then measure V_{p-p} from the CRO (and graph) and V_{dc} from the voltmeter.

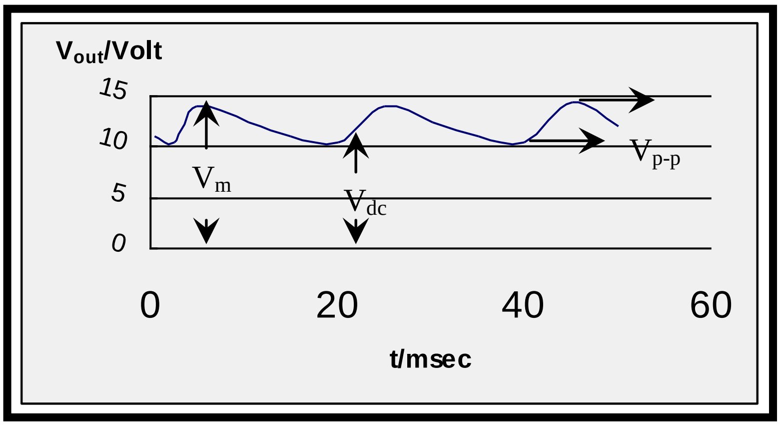
1. Repeat the previous step for the following values of C and R_L(Do not repeat the graph of input voltage):

R_L/ Ω	C/ µf	V_{p-p}/V	V_{dc}/V
100	**10**		
100	**100**		
1k	**10**		
1k	**100**		

Note:

Write the values of C, R_L, V_{dc} and Vp-p on the graphs.

R_L= 1KΩ
C=100µf
V_{p-p}= 3v
V_{dc} = 13v



Discussion:

1. What is the role of the capacitor C?
2. What is the effect of increasing each of C and R_L on V_{dc}? And what is the shape of your relation in your results?
3. How V_{dc} could calculate, approximately, from the graph instead of voltmeter measurements?
4. The frequency used in this experiment was equal to the mains frequency (50 Hz), if the frequency was 50 kHz, for example, what you suggest:
   a. Varying values of C and R_L.
   b. Changing electronic element.

Experiment (4)
Centre-point connection for full-wave Rectification

Aim of the Exp.

To be able to construct a circuit arrangement with a mains transformer with a centre-tapped winding, two diodes and a reservoir capacitor to produce a full-wave rectifier circuit.

Apparatus and Components

1. Two diodes: 1N4007; 2. Resistors: 100Ω, 1 KΩ; 3. Capacitors: 1 μF, 100 μF 4. A.C. Power supply: 12 V, 6 V, 0 V; 5. CRO, One Multimeter (high resistance)

Performance of the Exp.:

1. The circuit made up as in Fig. 1 and removes initially C, R_L.

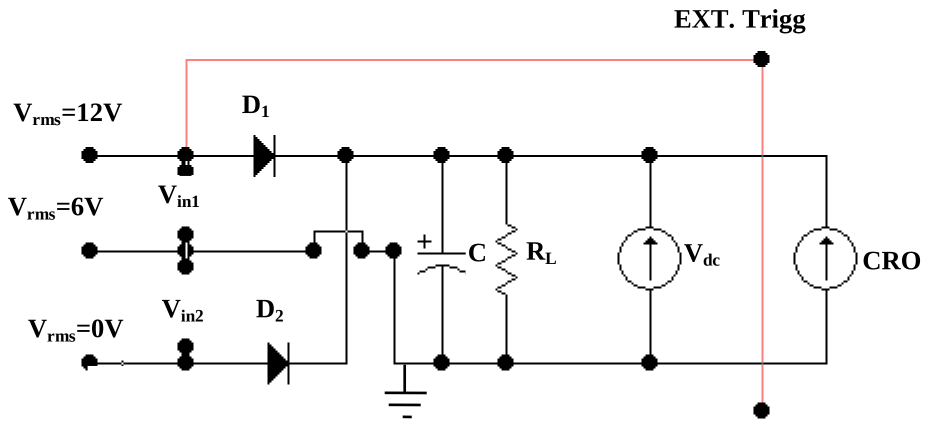


Fig. 1: Experiment set up of the circuit.

2. Set the input Y for CRO on D.C. and use EXT. Trigg.
   Note: Set the input of the CRO, for precise measuring of V_{p-p}(V_{ripple}), on A.C. and use suitable scale for sensitivity of Y channel.
3. Plot the V_{in1} and V_{in2} (the earth is 6 V).
4. Plot the output voltages then measure V_{DC} and V_{p-p}.
5. Set C and R_L according to the following table then repeat the step 4:

R_L/ Ω	C/ μf	V_{p-p}/V	V_{dc}/V
100	10		
100	100		
1k	10		
1k	100		

Discussion:

1. Discuss shortly the role of each: D_1, D_2, R_L, C.
2. Compare the differences in the results of this experiment with the previous one, (Rectification and Filtering half-wave), in terms of VDC, V-p and the output wave frequency?
3. Suppose you have a transformer which doesn't own central tapping, (only two polar), and you are asked to rectify a full wave using two diodes, what must you add to the circuit to operate normally?

Experiment (5)
Bridge circuit for Full-wave Rectification and Filtering

Aim of the Exp.:

To be able to connect a bridge circuit for full-wave rectification with a source of alternating voltage, four diodes and a load resistor or a filter chain.

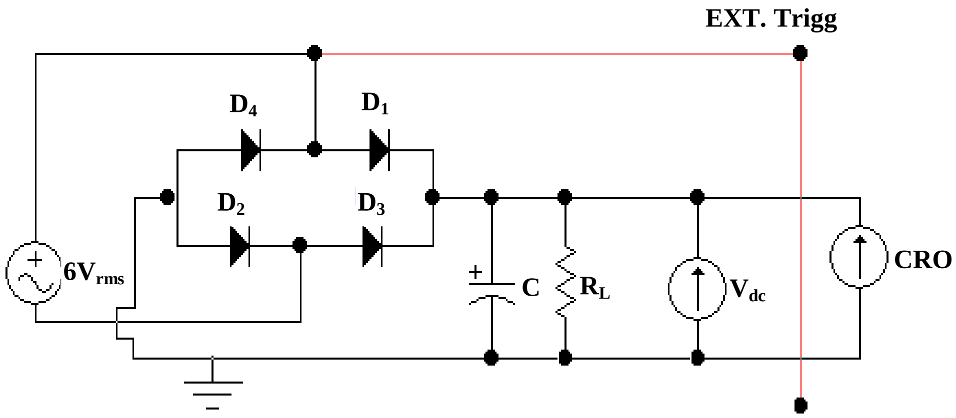


Fig. 1: Experiment set up of the circuit.

Apparatus and Components See the above figure.

Performance of the Exp.: As in the previous experiment.

Discussion:

1. Discuss the role of D1 and D2 in both the first half and second half of the wave (cycle)?
2. Compare most important differences between this experiment and the previous one, in terms of V_{AC}, V_{DC}.
3. What are the reason for the following:
   a. The bridge method is preferred on the rectifier method for a centre-tap transformer.
   b. The bridge method not recommended for a low-amplitude wave rectification.

Experiment (6)
Voltage Doublers

Aim of the Exp.:

To be able to built up a circuit with a source of alternating voltage, two diodes and two capacitors in such a way that a doubling of the voltage occurs.

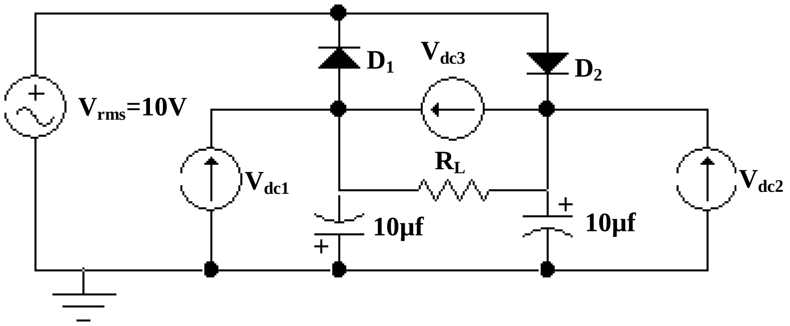


Fig. 1: Experiment set up of the circuit.

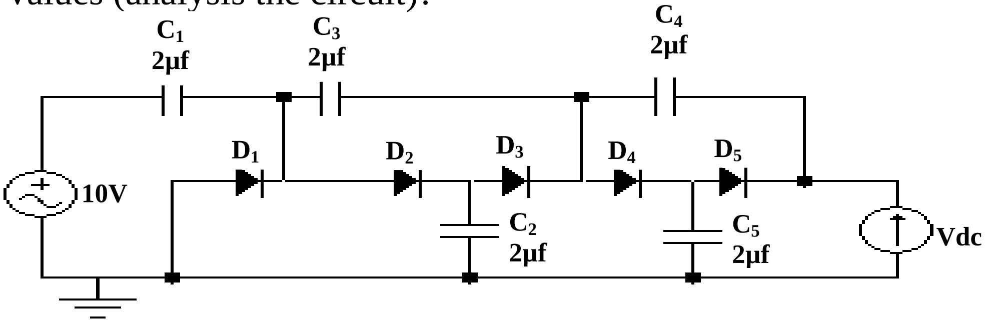
Apparatus and Components see the figure.

Performance of the Exp.:

1. The circuit in Fig. 1 made up. The load resistor R_L left out (running off-load, removed).
2. The three component voltages V_{DC1}, V_{DC2}, and V_{DC3} are measured (without R_L) with voltmeter.
3. Set the values of R_L to 100 Ω, 220 Ω, 470 Ω, 1 KΩ, 2.2 KΩ, and 10 KΩ respectively, then measure values of V_{DC3}, and finally plot a graph where $V_{DC3}=f(R_L)$.

Discussion:

1. Summarize operating both D_1 and D_2 in Fig.1, and what is the basic characteristic in connecting C_1 and C_2 (in parallel or series)?
2. What is the effect of R_L, and what is the relation of each V_{DC1}, and V_{DC2} with V_{DC3} usually?
3. What is the output voltage for the following circuit if V_{rms}=10 V? How obtain this output values (analysis the circuit)?



Experiment (7)
Clipping Circuits

Aim of the Exp.:

To be able to build up a circuit with AC and DC power supplies, diode and resistor in such a way that clipping the wave (clipping the positive and negative part, and both parts together).

Apparatus and Components:

1. Two diodes: 1N4007
2. AC power supply, DC power supply
3. Voltmeter, CRO

Performance of the Exp.:

A. Clipping positive part of the Sine wave:

1. Set up the experiment as in Fig. 1.
2. Set V_{in}= 8 V_{p-p}, and plot the input wave.
3. Set E=2 V and plot the output wave.

B. Clipping negative part of the Sine wave:

1. Set up the experiment as in Fig. 2.
2. Repeat the previous steps and plot the output voltage.

C. Clipping both parts of the Sine wave

1. Set up the experiment as in Fig. 3.
2. Repeat the previous steps.

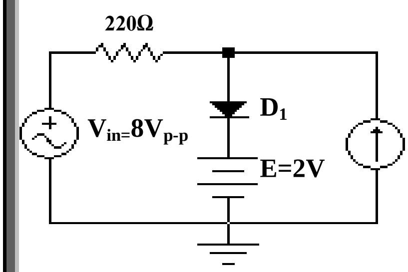


Fig. 1

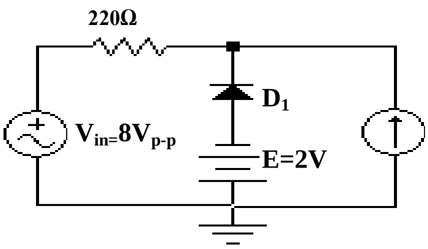


Fig. 2

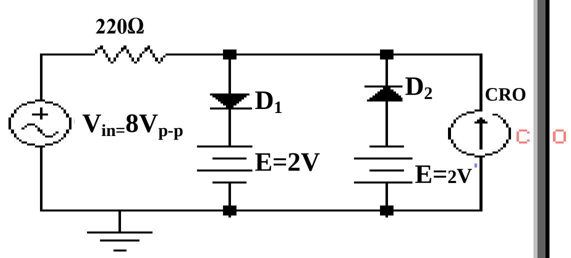


Fig. 3

Discussion:

1. In both cases (A) and (B), Explain how is the output shape of E=0?
2. What is the use of clipping circuits?

Experiment (8)
Clamping Circuits

Aim of the Exp.:

To be able to build up a circuit with AC and DC power supplies, diode and resistor in such a way that clamping the wave.

Apparatus and Components:

1. Two diodes: 1N4007
2. AC power supply, DC power supply
3. CRO: set on DC

Performance of the Exp.:

Part A:

1. Set up the experiment as in Fig. 1.
2. Set input voltage V_{rms}= 7 V.
3. Plot the input and output wave.

Part B:

1. Set up the experiment as in Fig. 2.
2. Plot the output voltage.

Part C:

1. Set up the experiment as in Fig. 3.
2. Set E=2, and plot the output voltage.

Part D:

1. Set up the experiment as in Fig. 4.
2. Set E=2, and plot the output voltage.

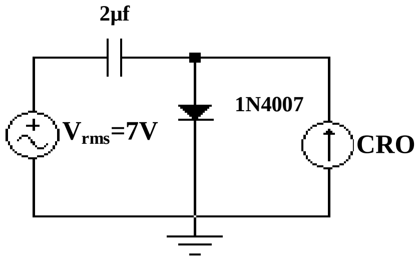


Fig. 1

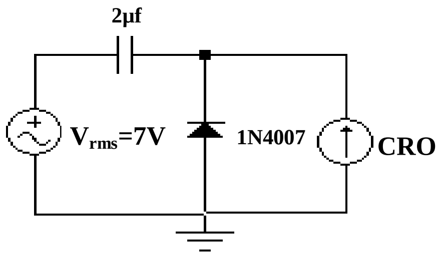


Fig. 2

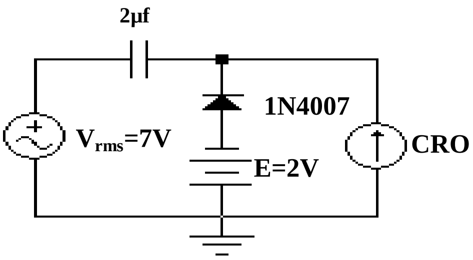


Fig. 4

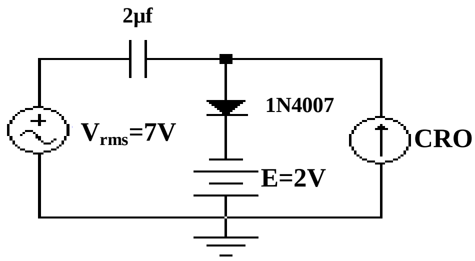


Fig. 3

Discussion:

1. Discuss the circuit operation in Fig. 1 and compare it with Fig. 2?
2. Compare the case in Fig. 1 with that of Fig. 3?
3. What are the uses of clamping circuit?

Experiment (9)
Characteristic curves of Transistor using Point-by-Point Method

Aim of the Exp.:

To construct measurement circuit use power supply and Avometer for drawing the important characteristics of transistor.

Apparatus and Components

1. Transistor type BC177
2. Constant resistant, 220 Ω, 10 KΩ
3. Two Potentiometers, 1 KΩ
4. Micrometer, CRO

Note:

1. The transistor BC177 is of the type PNP and has the following values:
   I_c(max)= 100 mA I_B(max)= 50 mA V_{CBO}= 45 V
   V_{EBO}= 5 V P=300 mW
2. The most widely circuits for the transistor are the Common emitter circuit (CE), so, the characteristic curves of transistor BC177 studied by connect the emitter with the Earth (i.e. reference).

Part 1: Control properties

1. Set P_2 nearly at midpoint of its scale.
2. Do not connect the dash lines of the circuit.
3. Construct the circuit as given in the following figure and be careful on the polarity of the power supply.

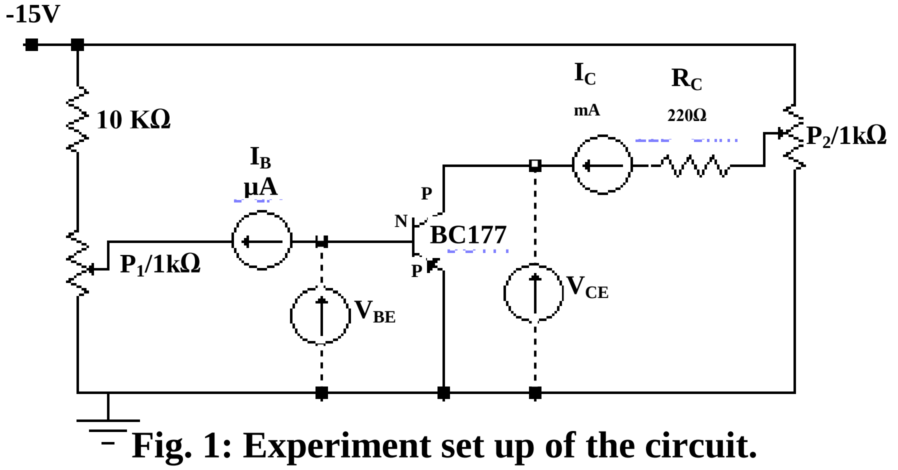


Fig. 1: Experiment set up of the circuit.

4. Set the wiper of P1 to the right-handed by steps to get I_c according to the table(1) and then read the values of I_B.

Table (1): Control properties.

I_C/mA	0.1	0.5	1	2	3	4	5	7	10	15
I_B/μA										

5. Plot a graph according to $I_C=f(I_B)$ and compute the slope as the line(What is meaning the slope?).

<u>Part 2: Input properties</u>

1. Connect Avometer V_{BE}(voltmeter or CRO).
2. For the same values of I_c in Table (1), read I_B, V_{BE}. Recognize that $I_{\mathbf{B}}$ varies slowly. Why? Write the results according to the Table (2).

Table (2): Input properties.

I_C/mA	0.1	0.5	1	2	3	4	5	7	10	15
I_B/μA										
V_{BE}/Volt										

3. Plot a graph for input properties according to $I_B=f(V_{BE})$.

<u>Note</u>: Recognize that the P2 is in the midpoint of scale, so V_{CE} is constant and the line drawn in the input properties graph is one of the infinite lines for the input property whereas they are very close to each other however, very sensitive instrument is required to distinguish among them whenever V_{CE} varied.

<u>Part 3: Output properties</u>

Set values of I_B by P_1, and values of V_{CE} by P_2 then read values of I_c according to Table (3).

I_B/μA	10	20	50	75	100
V_{CE}/Volt	I_C/mA				
1					
2					
3					
4					
5					
6					

Table(3): Output properties.

Calculations(using graphs)

1. Compute input resistance from input properties ($R_{in}=\Delta V_{BE}/\Delta I_B$).
2. Compute forward conversion ratio from control properties ($\beta=\Delta I_c/\Delta I_B$).
3. Compute output resistance from output properties ($R_{out}=(\Delta V_{CE}/\Delta I_c)_{IB=cost.}$).
4. Compute gain in voltage (static) from output properties ($A_v=\Delta V_{CE}/\Delta V_{BE}$).

Discussion: Discuss all the three graphs.

Experiment (10)
Common emitter amplifier

General

The common emitter circuit is the most widely used for voltage and this circuit gives current amplification, so, the highest power amplification.

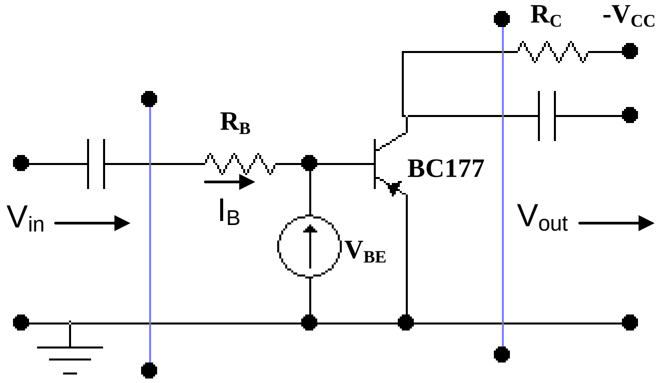


Fig.(1): shows the principle of the common emitter circuit.

The voltage amplification is:	$A_v=\Delta V_{CE}/\Delta V_{BE}$	…(1)
The current amplification is:	$A_i=\Delta I_C/\Delta I_B$	…(2)
The input resistance is:	$R_{in}=\Delta V_{BE}/\Delta I_B$	…(3)
The output resistance is:	$R_{out}=\Delta V_{CE}/\Delta I_C$	…(4)
The power amplification is:	$P_A=A_v.\ A_i$	…(5)

Note: The difference of the variables is taken because the elements have nonlinear properties (see Equations. 1-4).

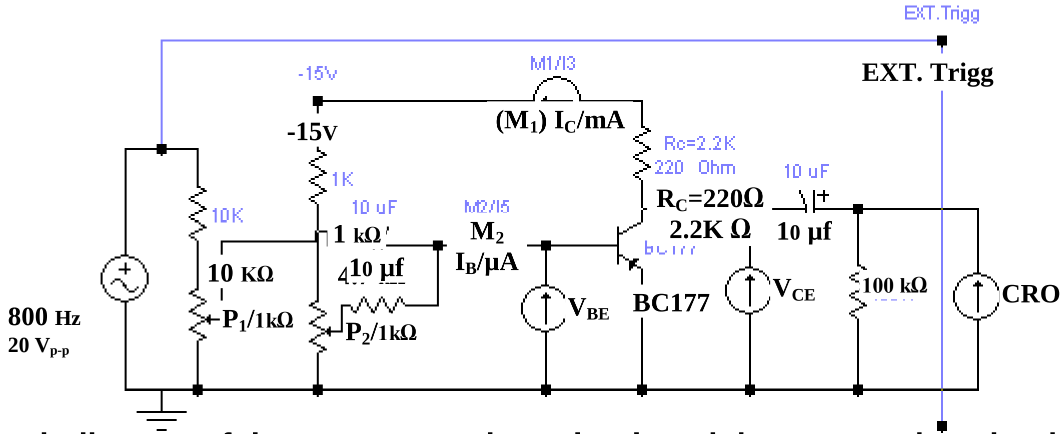
Aim of the Exp.:

1. To construct a common emitter circuit and calculate: amplifications of current and voltage, input and output resistance, and amplification of power.
2. Inserting sine wave shapes with frequency 800 Hz and calculating dynamic amplification of voltage.
3. To study the relation between input resistance and amplification of voltage.
4. To study the noise.

Apparatus and components: (See Fig.(2))

Part 1: Static properties

1. Construct the circuit as in Fig.(2) and remove Oscillator and voltmeter V_{BE} and V_{CE}.



Fig(2): Circuit diagram of the common emitter circuit and the measuring circuit.

Note: You need only two Multimeters, M1 and M2. M1 set to milliamper for measuring I_C while M2 for measuring current and voltage. When remove M2 where used to measure I_B, its place must be short-circuited.

2. Set I_B as in following table and measure I_C, V_{BE}, and V_{EC}.
3. Calculate A_i, A_v, R_{in}, R_{out}, and P_A.

$I_B/\mu A$	I_C/mA	V_{BE}/V	V_{CE}/V
25			
50			

Part 2: Dynamic properties

1. Remove M2, which used to measure V_{CE} and V_{BE}. Connect the oscillator and input sine wave shape with amplitude 50 mV and frequency 800 Hz. Its amplitude controlled by P1.
2. Connect CRO instead of voltmeter V_{BE}, plot input wave, and calculate their amplitude $V_{1p\text{-}p}$.
3. Connect CRO in the output of the circuit and using P2 to varying base current I_B to obtain the maximum output amplitude without noise.
4. Plot the output wave and write value of I_B, their amplitude $V_{2p\text{-}p}$.
5. Use the following equation to calculate dynamic voltage amplification:

$$A_V = V_{2p-p} / V_{1p-p} \quad \dots (6)$$

Part 3: Effect of collector resistance R_c

1. Set R_c=2.2 KΩ.
2. Try to obtain maximum output voltage without noise by varying P2, than write I_B, V_{2p-p} and plot the curve.
3. Use Equ.(6) to calculate $\mathbf{A}_v^{`}$.

Part 4: Studying of Noise

1. Set R_c=2.2 KΩ.
2. Answer the following question: you get noise for the wave shape for values $I_B \leq \ldots \mu A$ and for $I_B \geq \ldots \mu A$ [use P2 to vary I_B]. Why?

Discussion:

1. What is the time relation (phase difference) between V_1 and V_2?
2. Why amplification of voltage increased by increasing R_c?
3. Can use transistor in this circuit to obtain amplifications in D.C. voltage and current? Why?
4. Is there difference between A_v and $A`_v$?
5. What are the reasons of noise in this amplifier?

Experiment (11)
Common base amplifier

General:

In the common base circuit (Fig.(1)) the base of the transistor is the common reference potential for the emitter electrodes as the input and the collector electrode as the output. Compared with other basic circuits, control of the common base circuit requires the largest control current, since the sum of the collector current and the base current takes effect in the emitter. The current amplification is therefore less than one. The input resistance is lowest with this circuit arrangement.

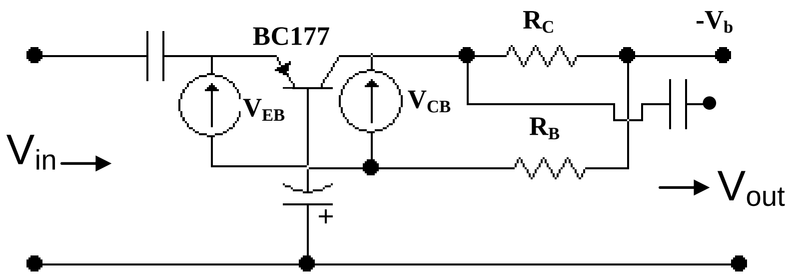


Fig.(1): shows the principle of the common base circuit.

The following Equations are necessary:

The voltage amplification is: $A_v=\Delta V_{CB}/\Delta V_{EB}$ …(1)

The current amplification is: $A_i=\Delta I_C/\Delta I_E$ …(2)

The input resistance is: $R_{in}=0.025\ v/I_E$ …(3)

The power amplification is: $A_P=A_v.\ A_i$ …(4)

Note: Equ.(3)is empirical equation and it more accurate than ($R_{in}=\Delta V_{EB}/\Delta I_E$) Because ($V_{EB}$) is very small.

Aim of the Exp.:

1. Construct a common base circuit for measuring the characteristic values of transistor BC177.
2. Insert sine wave shape with frequency 800 Hz and finding amplification of voltage and study the noise.
3. Study the frequency response curve for amplifier.

Apparatus and Components:(see Fig.(2)

Part 1: Static properties

1. Construct the circuit as in Fig.(2) and remove the dotted parts.
2. Set I_E=20 mA by P2 and read I_c by ammeter M2.
3. Connect short circuit instead of M2 and use it to measure V_{EB} and V_{CB}. Then, return M2 for measuring I_c.
4. Set IE= 30 mA. Repeat the work…read I_c then V_{EB} and V_{CB}. Arrange the results in a table.

   Note that V_{EB} and V_{CB} measured relative to the base not to the earth.
5. Calculate A_v, A_v, A_p and R_{in}. Arrange the results in a table.

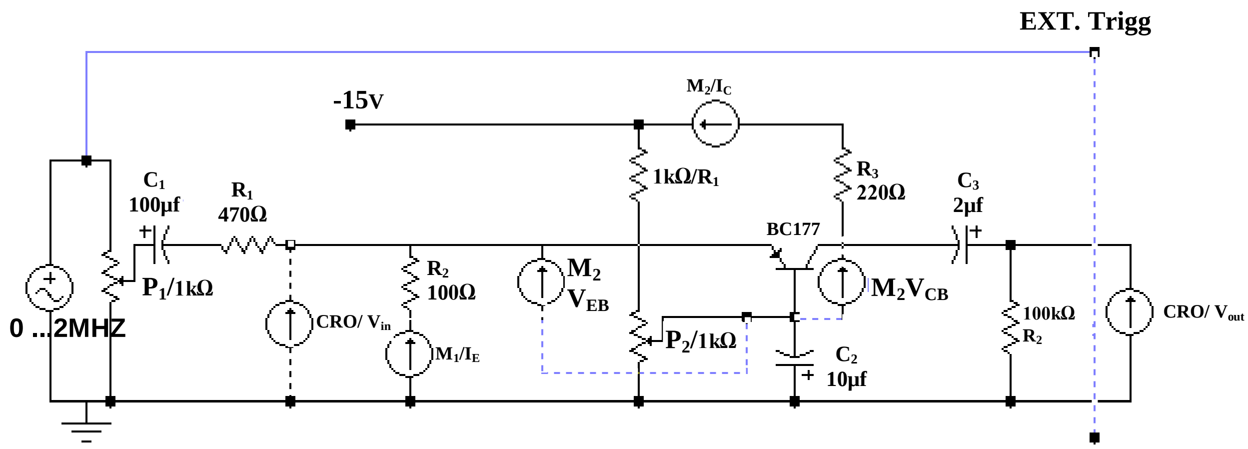


Fig. (2): Circuit diagram of the common emitter circuit and the measuring circuit.

Part 2: Dynamic properties and study the Noise

1. Prepare CRO for measuring and return M2 to measure I_c.
2. Set I_E=25 mA by P2 (i.e. between 20-30 mA).
3. Set the voltage of input signal 50 mV by P1, and then draw the output wave.

V_{in}=50 mV V_{out} = ( )V_{p-p} $A_v^- = \dfrac{V_{out}\, p-p}{V_{in}\, p-p}$ …(5)

Noise(Clipping Distortion)

Introduction: This type of noise occur when the transistor go out from their operating range as a result of several reasons like nearly large amplitude wave input, low or large emitter current which lead to going the transistor to cut-off region or saturation region.

1. Decrease I_E by P2 in steps to obtain the noise of the wave, write I_E, the increase I_E by P2 to obtain noise again and write I_E, so:

   Whenever $I_{E} <$ …… mA there be noise in the output wave.

   Whenever $I_{E} >$ …… mA there be noise in the output wave.

Part 3: Frequency Response

Introduction: There are special response curves for each transistor which depend on the several parameters like configuration of the circuit and the type of the transistor (NPN or PNP). As known, the carriers of Bipolar Junction Transistors (BJT) are electrons and holes. If type of the transistor is NPN, the signal propagates from emitter to base dependent on the mobility of electrons. There are some limits for the permission frequencies depend on the thickness of the base and mobility of the electrons (which depend on the type of the semiconductor). In general, the mobility of the electrons larger than holes because the holes are more weighted from the electrons. Therefore, the response of the hole is slower than electron for signals. In the amplifier circuit, the response also depend on the magnitude of the capacitors and resistors connected with the terminals of the transistor where they work as in the High pass or Low pass RC circuits. Therefore, the frequencies for the amplifier must not be very low or very high, which lower the gain (in current, voltage, and power). In this experiment, these limits must identify.

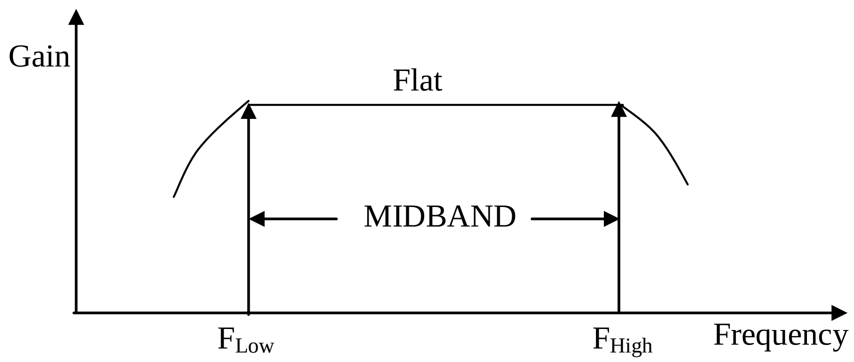


Fig.(3): Typical Amplifier Gain vs Frequency plot.

1. Prepare the amplifier to obtain output pulse without noise for an input sine wave has amplitude 50 mV.
2. Set the following frequencies and calculate $\mathbf{A}_v^{/}$ from Eq.(5) and find F_{Low} and F_{High}: (0.1, 0.2, 0.3, 0.7, 1, 3, 5, 10, 15, 20, 30, 50, 100, 200, 300, 400, 600, 800) KHz, (1.2, 1.4, 1.6, 1.8, 2) MHZ.

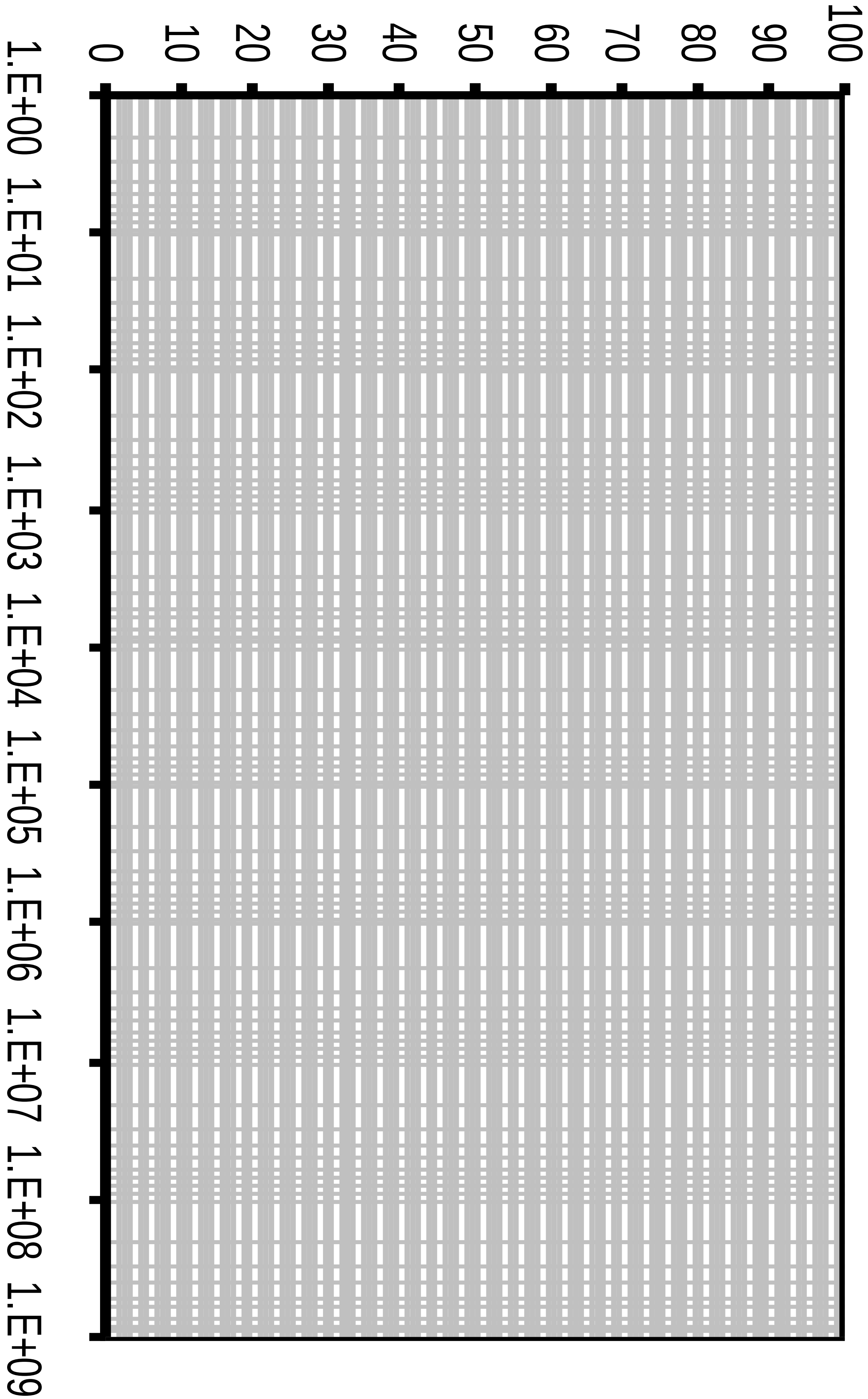
Discussion:

1. Insert these and previous results into the following table and the compare between them and discuss it.

Quantity	Common Emitter Circuit	Common Base circuit
A_i		
A_v		
A_p		
A_v^-		
Φ		
R_{in}		

2. Input resistance in the Common Base circuit is low. Why?
3. What is the benefit of C_3 in the Fig.(2)?
4. What represent R_4 in the Fig.(2)?
5. Discuss the frequency response curve and are you prefer NPN or PNP transistor for amplifying signals with large frequency?

Note: Use Semilog paper graph for drawing the frequency response curve.


0
10
20
30
40
50
60
70
80
90
100
1.E+00
1.E+01
1.E+02
1.E+03
1.E+04
1.E+05
1.E+06
1.E+07
1.E+08
1.E+09

Experiment (12)
Common Collector Amplifier

General

The name emitter follower, impedance transformer or, better, impedance converter, also knows the common collector circuit. The name impedance converter is attributable to the fact that this circuit has a high input resistance and a low output resistance. This property is due to a high current amplification, whereas the value of the voltage amplifications approximately 1.

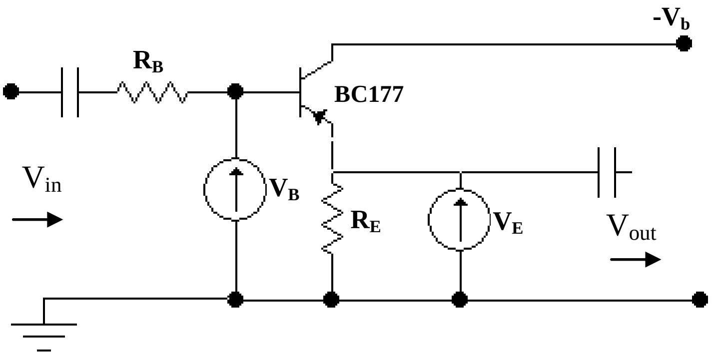


Fig.(1): shows the principle of the common collector circuit.

The input resistance R_{in} gives by: $R_{in}=R_B+R_{BE}+R_E\times A_i$ ... (1)

The output resistance gives by: $$R_{out}=\frac{(R_{in}/A_i).R_E}{(R_{in}/A_i)+R_E}$$ ...(2)

The voltage amplification: $A'_v = V_{out}/V_{in}$ ... (3)

The static current amplification: $A_i = \Delta I_E / \Delta I_B$ ... (4)

The resistance between base and emitter: $R_{BE} = \Delta V_{BE} / \Delta I_B$ ... (5)

The base voltage: $V_B=V_E+V_{BE}$ ... (6)

Aim of the Exp.;

To construct a common collector circuit from a given circuit diagram. To measure the values required for the characteristics.

Apparatus and components

1. resistors: 4.7 KΩ, 220 Ω
2. potentiometer, 1 KΩ, two pieces
3. transistor, BC177

4. capacitor, 10 μF, two pieces, 0.1 μF
5. CRO
6. DC power supply, AC power supply

Part 1: Static Properties (without AC power supply)

1. Connect the circuit as in Fig.(2) without connecting signal source.

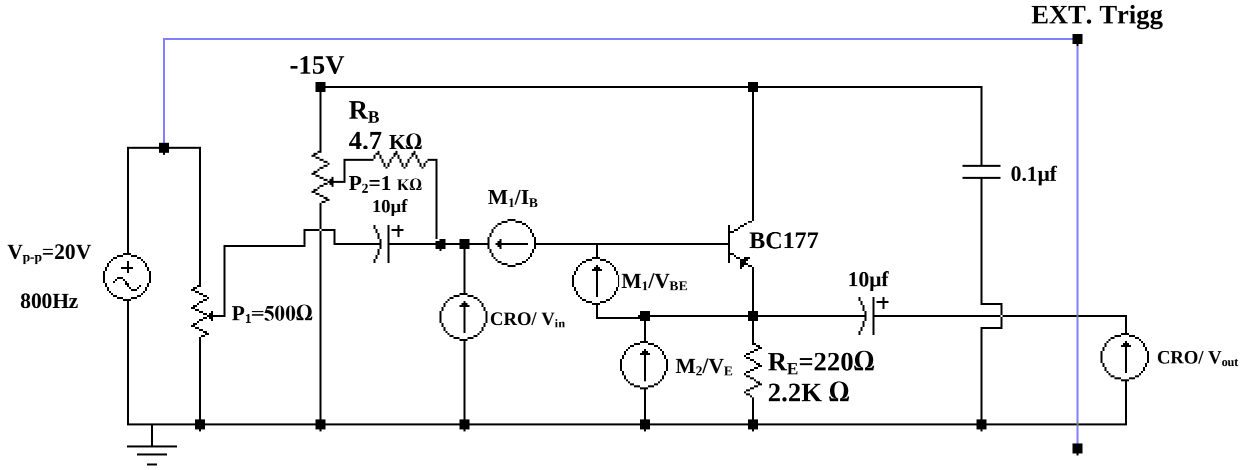


Fig.(2): Circuit diagram of the common collector circuit and measurement arrangements.

2. By P2 set I_B about 30 μA and measure V_E then calculate I_E from the relation $I_E=V_E/R_E$.
3. Measure V_{BE} and calculate V_B from Eq.(6).
4. Set I_B=50 μA and repeat steps 2, 3, and rearrange the results as in the following table.
5. Using Eqs.(1,2,4,5) for calculating R_{in}, R_{out}, and A_i.

I_B	I_E	V_E	V_{BE}	R_{in}	R_{out}

Part 2: Dynamic Properties(with signal source)

1. Remove the measurements instrument.
2. Set V_{in}= 4 V_{p-p}.
3. Connect CRO on the output terminals then varying P2 to obtain the maximum output amplitude without noise then write V_{out}= … V_{p-p}.
4. Use Eq.(3) for calculating $\mathbf{A}'_v$.

5. Set R_E=2.2 KΩ and repeat steps 3, 4, and calculate $\mathbf{A}_v^{'}$. Rearrange the results in another table.

Discuss

1. Compare the input resistance of the Common collector circuit with the common emitter and common base circuits?
2. Compare the output resistance of the Common collector circuit with the common emitter and common base circuits?
3. To which values of V converges when R_E increases?
4. What is the time relation of the V_{in} and V_{out}?
5. Are the V_B increases or decreases by increasing I_E?

Experiment (13)
Two-stage amplification using RC connection

General

There are several way for connecting amplifier stages with each other like direct connection, RC connection, transformer connection which every connection have its prefer. The application of RC connection helps to reducing the cost and the volume of the amplifier with some dissipation of the amplifier gain. This way of connection useful especially in audio amplifiers which have low-level gain and low noise. The responds of the audio amplifiers using RC connection for the frequencies are more than amplifiers using transformer connection. Figure one shows circuit diagram for two-stage amplifier using RC connection that contains two PNP transistors connecting in common emitter. The value of C_2 must be nearly large, 2-10 μF, for low input resistances and low load resistance.

Aim of the Exp.

To construct two stage amplifier circuit and calculating the amplification of the output voltage.

Performance of the experiment

1. Construct the circuit diagram as in Fig. 1 and set input voltage as V_1=20 mV_{p-p} by P1.
2.
3. Vary P2 to get a wave without noise for the first amplifier stage V_{2p-p} then vary P3 to obtain another wave without noise V_{3p-p}. Calculate the gains for the two stages amplifiers in respect to V_{1p-p}.

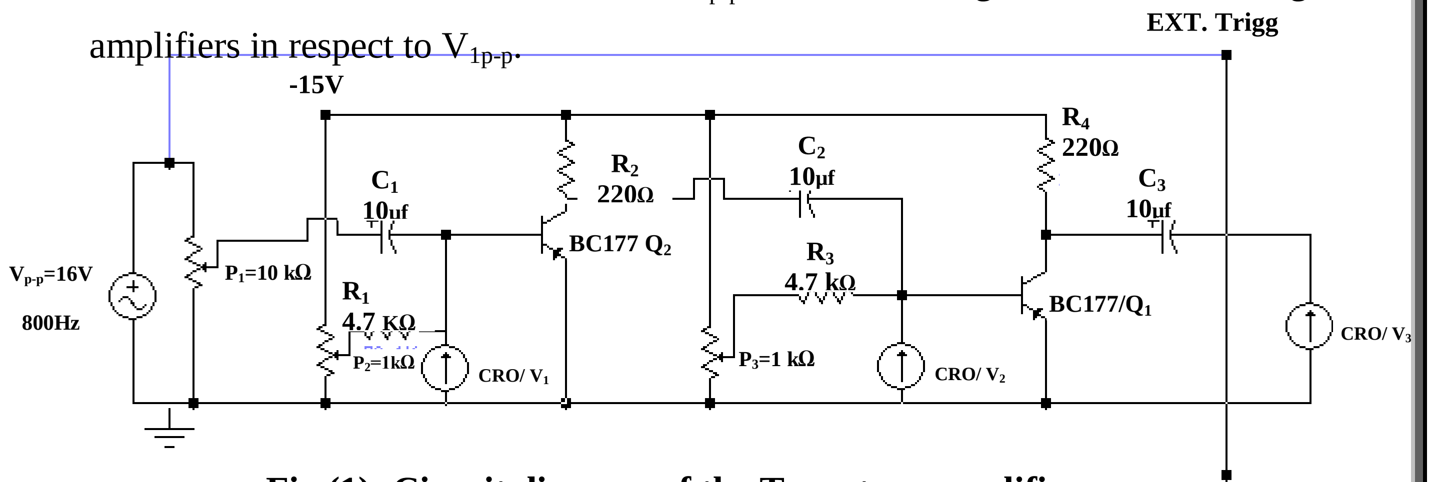


Fig.(1): Circuit diagram of the Two-stage amplifier.

Discuss:

1. Why when the stage of amplifier increased to two stages, the gain not doubled?
2. Why the amplitude of the input wave $V_{1p\text{-}p}$ must be small?

Experiment (14)
Differential Amplifier

General

The Differential Amplifier stage in Fig.(1) consists of two common emitter amplifier stages which have a common emitter resistor which acts as current feedback. If the electrical characteristics of the transistors are identical, the differential voltage amplification given by the ratio of the output voltage V2 to the difference of the two input voltage V11 and V12. Identical signals, i.e. when the difference between the two input voltages equals zero, virtually cancelled out. The degree of in-phase suppression stated by means of a factor F denoting the ratio of amplification of a differential signal to the unsuppressed amplification when the difference is zero.

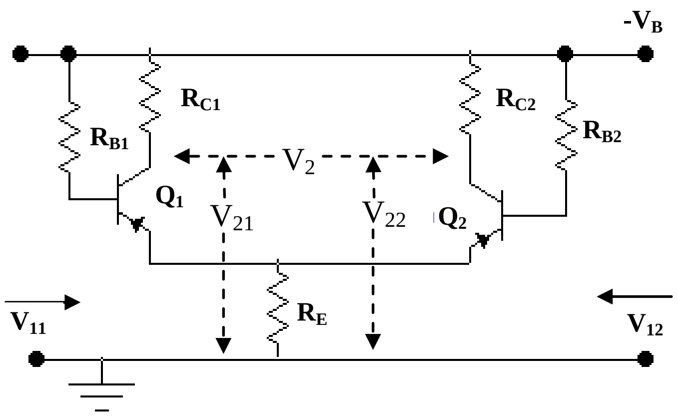


Fig. (1): The basic circuit diagram of a differential amplifier stage.

Accordingly the amplification of the differential stage is:

$$A_v = \frac{V_2}{V_{11} - V_{12}} \qquad \ldots(1)$$

The in-phase amplification A_{IP} given by:

$$A_{IP} = \frac{V_2}{V_1} \qquad (V1=V11-V12);\ (V2=V21-V22) \qquad \ldots(2)$$

Moreover, the in-phase suppression by:

$$F = \left(\frac{A_{IP}}{A_v}\right) X100\% \qquad \ldots(3)$$

Aim of the Exp.:

To construct a differential amplifier input stage to a given circuit diagram. To measure important characteristics values. The calculated amplification and in-phase suppression.

Performance of the Exp.:

Part I: In-phase case

1. Construct the circuit as in Fig.(2). Prepare channels ch1 and ch2 and note the polarity of the source.
2. Prepare Q1 and Q2 by P1 and P2 to obtain the following values of collector voltage: V_{c1}=5 V V_{c2}=5 V V_{RE}≅2 V.

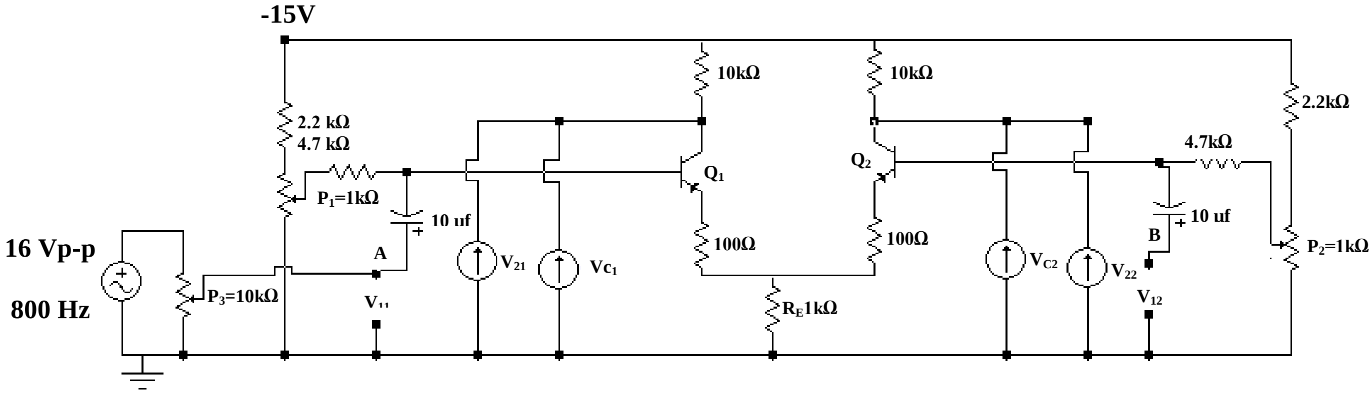


Fig.(2): Circuit diagram of the differential amplifier and measuring arrangements.

3. Now connect the CRO between point A and the ground and set input voltage: V_{11p-p}=40 mV then return to their original place.
4. Connect points A with B (i.e. connect input of Q1 with the input of Q2 then repeat the measurements of step 2).
5. If there is no difference between Vc1 and Vc2, see CRO. Is there are phase difference between the output voltages? Now write the values:

   V_{21p-p}=….V, V_{22p-p}=….V, Φ_{11}=….(phase difference between output voltages)

   Calculate $V_{2p-p}=V_{21p-p}-V_{22p-p}$=……V, A(in phase)=$V_{2p-p}/V_{1p-p}$

Part II: Difference phase case

6. Remove the connection of point A with B.

7. Connect point B with the ground and recognize the existence of the phase difference between output voltages. Write the following values:

   V_{21p-p}=….V, V_{22p-p}=….V, Φ_{12}=…. V_{2p-p}(Diff.)=V_{21p-p}-V_{22p-p}=……V,

   A_v(diff. phase)=V_{2p-p}(Diff.)/V_{1p-p}

8. Calculate the percentage ratio for coefficient F.

Important Note: In step 7, reversed the sign of the output when there happen some phase difference. So, recognize this when you write V21p-p and V22p-p and in the calculation of difference between them.

Discuss:

1. What is the benefit of the difference amplifier? Where it used?
2. What is the purpose of connecting R_E in the emitter circuit of the Q1 and Q2? Is this connection necessary?
3. How you can to reduce the coefficient of F?
4. What is the purpose of step 6?

Experiment (15)
RC phase-shift oscillator

General

The RC oscillator Fig.(1) consists of an amplifier stage with feedback circuit. The feedback circuit consists of a multi-element circuit with RC elements lying between the output and input of the amplifier.

In order to fulfill the positive feedback condition (output and input voltage The transistor oscillation phase) the 180^o phase shift that occurs, must be cancelled out by the RC networks.

If there are three phase networks, the minimum number, the individual RC networks have to shift the phase by 60^o. Four RC networks require a phase shift of 45^o per element. This phase shift achieved when the reactance of the capacitor is exactly equal to the value of the resistor, which corresponds at the same time to the cut-off frequency of the RC network.

The power gain must exceed one to conserve the oscillation process. Whenever, it lowers than one, the oscillation decrease exponentially with time to stop. The oscillator circuits need power gain more than one because the output power divide between the load and feedback circuit.

The frequency of the circuit determined by RC circuit, LC circuit, and crystal. They put in the base or collector circuit.

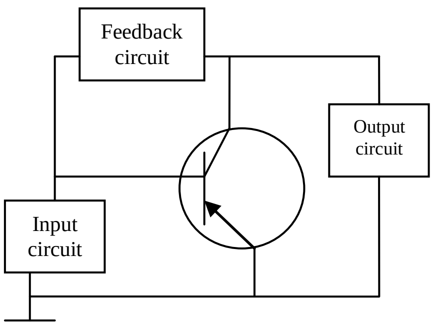


Fig.(1): A simple circuit diagram shows that the output power divided between the load and feedback circuit.

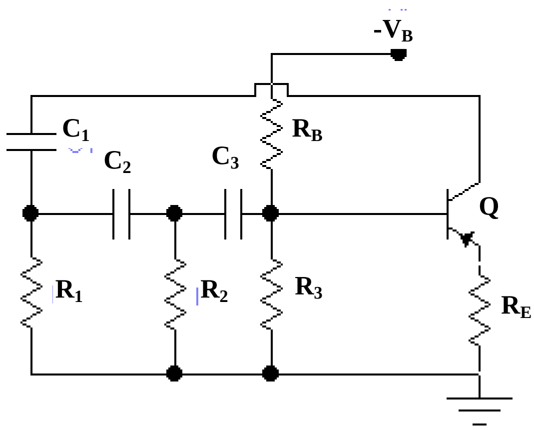


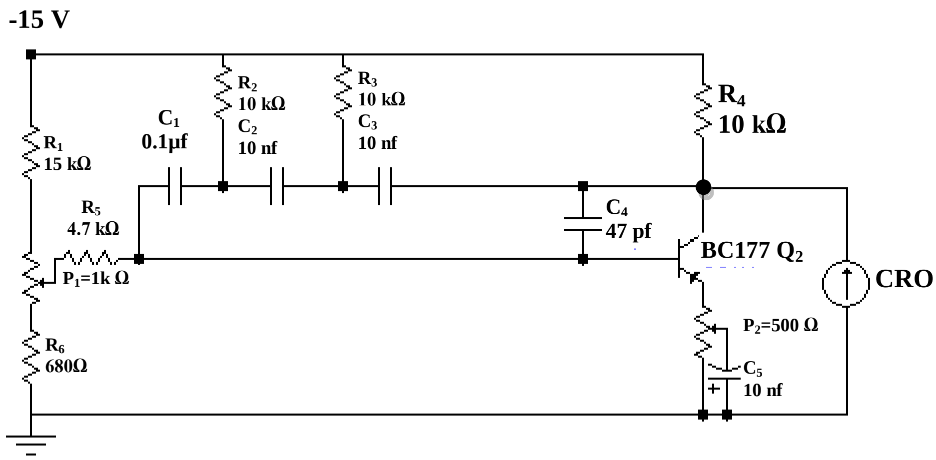
Fig.(2): RC phase-shift oscillator.

Aim of the Exp.:

To construct an RC phase-shift oscillator to given data and to measure the oscillation that is generated with the CRO.

Performance of the Exp.:

1. Prepare CRO and construct the circuit as in Fig.(3).



Set P2 in the middle (i.e. 5) and vary P1 to get 12 $V_{p\text{-}p}$. Then adjust P2 to obtain a sine wave without noise.

2. Calculate the frequency.
3. Repeat the processes for the following cases:

C_1	C_5	T	F=1/T
0.1µf	10µf		
0.1µf	2µf		
10nf	10µf		
10nf	2µf		

4. Vary the values of R_2, R_3, and R_4 to 4.7 KΩ then set C_1= 10 μF and C_5=10μF. calculate the frequency and describe the results.

C_1	C_5	T	F=1/T
10nf	10μf		

5. Remove C_4 what happened? Discuss it?
6. Remove C_5 what happened? Discuss it?

Experiment (16)
Colpitts LC Oscillator

General

The Colpitts oscillator works with a capacitive three-point connection (fig. 1). The resonance voltage, which appears across the oscillatory circuit, split to two component voltages by capacitors C_1 and C_2.

The voltage across C_2 fed to the emitter as positive feedback voltage. As regards alternating voltage, the base lies at zero potential via capacitor C_B, as a result of which the transistor works in the common base configuration in which no phase shift occurs between the input and output voltages.

The advantage of this circuit arrangement lies in the fact that the base is at zero potential and thus acts as an electrical separator between the output and input. Capacitive reactive effects are thereby avoided, resulting in a raising of the upper limiting frequency of the amplifier.

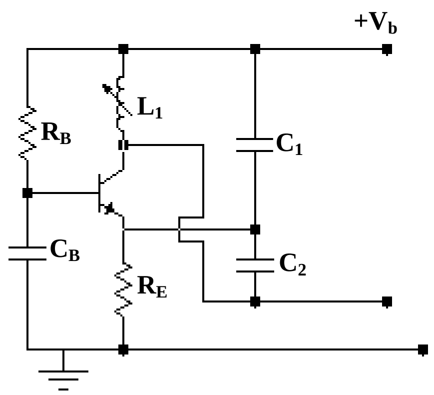


Fig.(1): Colpitts Oscillator

Aim of the Exp.:

To construct a Colpitts LC oscillator to given data, and to measure the oscillation generated with the oscilloscope.

Performance of the Exp.:

1. Construct the circuit as in Fig. (2).

2.

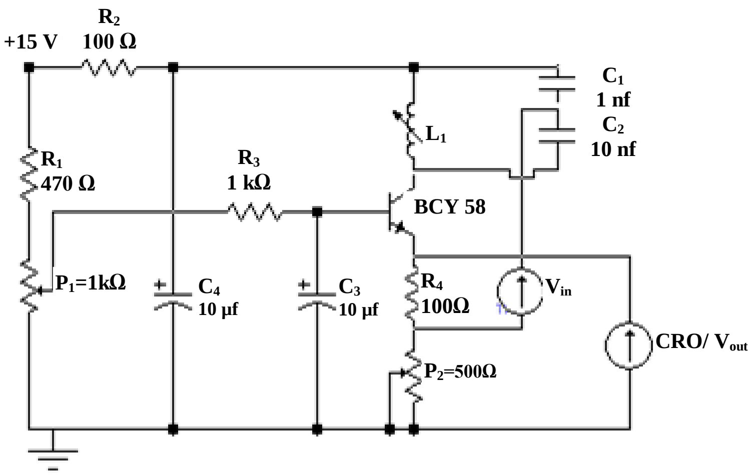


Fig.(2): Arrangement for testing the Colpitts Oscillator.

3. Rotate P_2 to the end of right side (position zero). Adjust P2 to obtain maximum output amplitude and calculate the amplitude and frequency of the signal:

   V_o=…….. V_{p-p} F=……….Hz

4. Calculate the alternating voltage across the emitter resistor R_E: V_i=……..V_{p-p} And the gain coefficient: $A_v^- = V_o / V_i$
5. Rotate P_L slowly toward left and right and release the CRO. What happened?.
6. Repeat calculations V_o, V_i and A'_v. What happened to A'_v? Why?
7. Rotate P_2 toward the end of left side, then measure the alternating voltage from the collector. Vary P_1 to obtain maximum amplitude. Remove C_3. What happened?
8. Measure the alternating voltage from emitter, remove C3, and note the changes.

Experiment (17)
HARTLY LC OSCILLATOR

General

In the Hartley circuit two coils, L_1 and L_2 are connected together in series (fig 1). Consequently, the junction forms a tapping of the total winding thus created (principle of the autotrans former). As regards A.C. voltage, the tapping lies at zero potential.

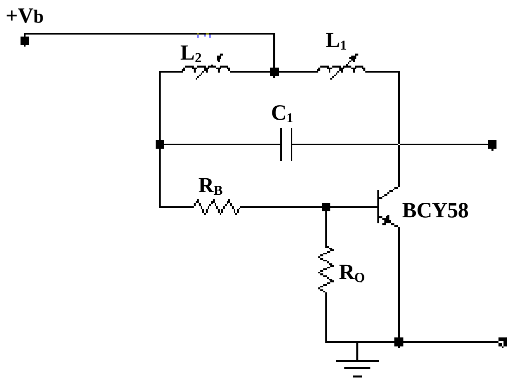


Fig (1): Hartly oscillator (inductive three-point connection)

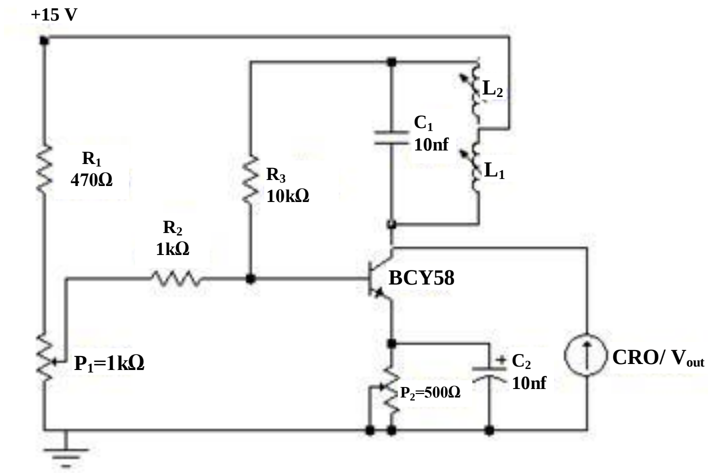
Two alternating voltages which are 180^0 out of phase with reference to the center tap are produced in the component windings $L_1$and L_2.the voltage across coil is the output voltage of the amplifier .the voltage induced in coil L_2 is fed to the amplifier as positive feedback voltage via the potential divider $R_{B,}R_{Q.}$ the frequency of oscillation is determined by the total inductance,$L_1$and L_2. And capacitor $C_{1.}$ By virtue of the way in which the coils are connected, the Hartley oscillator is also called an inductive therr-point connection.

Aim of the Exp.:

To construct a Hartley LC oscillator to given data, and to measure the oscillation generated with the oscilloscope.

Performance of the Exp.:

1. Construct the circuit as in Fig.(2).



Fig(2): Arrangement for testing the hartly LC oscillator.

2. Rotate P_2 to the end of right side (position zero). Adjust P_1 to obtain maximum amplitude(20 $V_{p.p}$) without noise and calculate the amplitude and frequency of the signal: V_o=…….. V_{p-p} F=……….Hz

3. Calculate the alternating voltage across coil L_2 V=……..V_{p-p}
4. Calculate alternating voltage at the base of T_1 V=……..V_{p-p}

□□ = □□□□

5. What happens when the core of either coil L_1 or coil L_2 is screwed out?

Discuss:

1. Compare between the circuits and output frequency of this oscillator and previous one?
2. Explain reason of changing frequency in step 4 of procedure of the exp.

Experiment (18)
Pulse Generator(IC 555)

General

The 555 timer is one of the most popular and versatile integrated circuits ever produced. It includes 23 transistors, 2 diodes and 16 resistors on a silicon chip installed in an 8 pin mini dual-in-line package (DIP). The 556 is a 24 pin (DIP) that combines two 555 's on a single chip. In addition, ultra-low power versions of the 555 are available. The 555 has two principle operating modes.

Mono stable mode: in this mode the 555 functions as a "one-shot". Applications include timers, missing pulse detection, bounce free switches, touch logic locks, tone generation switches …etc.

A stable mode: the 555 can operate as an oscillator. Uses include LED and LAMP flashers, pulse generations, security alarms …etc.

555 SPECIFICATIONS

Supply voltage (v_{cc}) 4.5 to 15V

Supply current (V_{cc}=+5V)[2] 3 to 6 mA

Supply current (V_{cc}=+15V)[2] 10 to 15mA

Output current 200mA (maximum)

Power dissipation 600 mw

Operating temperature 0 to 70^0 C

1- Values shown apply to NE 555 (8 PIN MINI-DIP)
2- Output current =0

Aim of the Exp.:

To construct an IC555 pulse generator circuit, and to measure the output frequency by the oscilloscope.

Performance of the Exp.:

1. Construct the circuit as in Fig.(2).
2. Change values of R and C according to following table.

3. Draw frequency with resistance curve for different C.

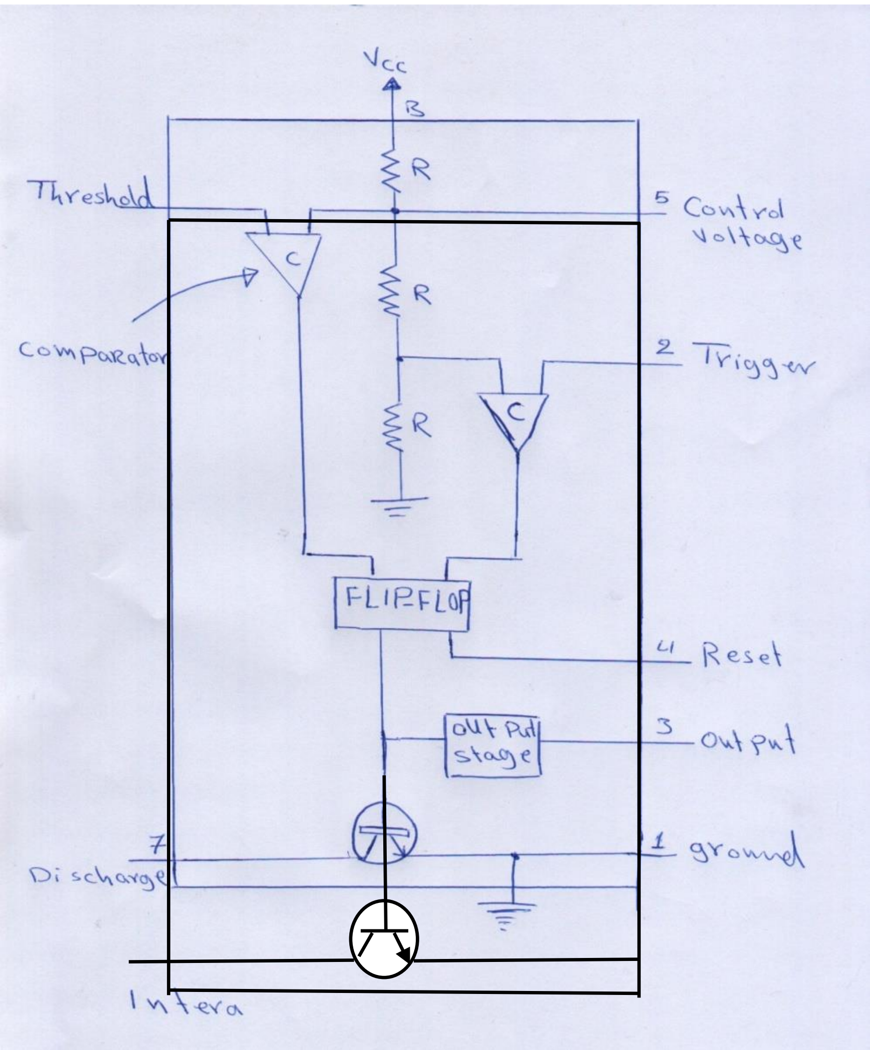


Fig.(1): Internal block diagram.

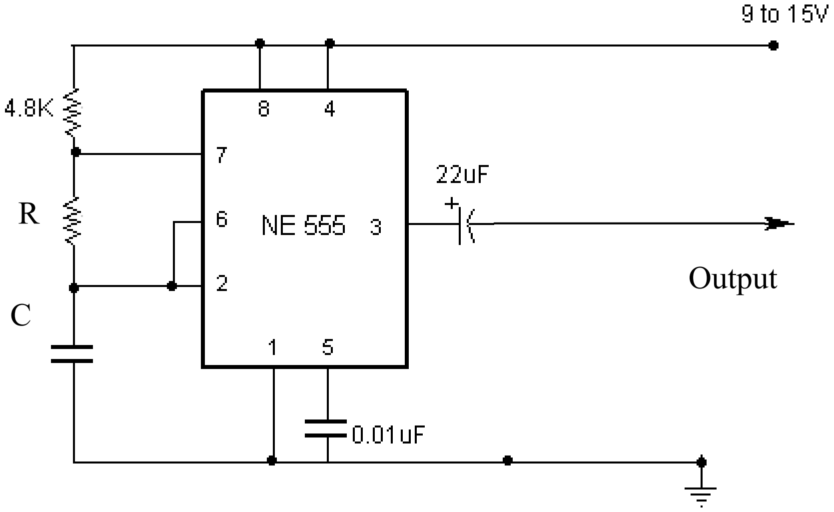


Fig.(2): Arrangement for IC 555 oscillator circuit.