Basic Electronics Lab Manual

School of Physical Sciences
National Institute of Science Education and Research
Bhubaneswar

IDENTIFICATION OF CIRCUIT COMPONENTS

Breadboards:

In order to temporarily construct a circuit without damaging the components used to build it, we must have some sort of a platform that will both hold the components in place and provide the needed electrical connections. In the early days of electronics, most experimenters were amateur radio operators. They constructed their radio circuits on wooden breadboards. Although more sophisticated techniques and devices have been developed to make the assembly and testing of electronic circuits easier, the concept of the breadboard still remains in assembling components on a temporary platform.

(a)

(b)

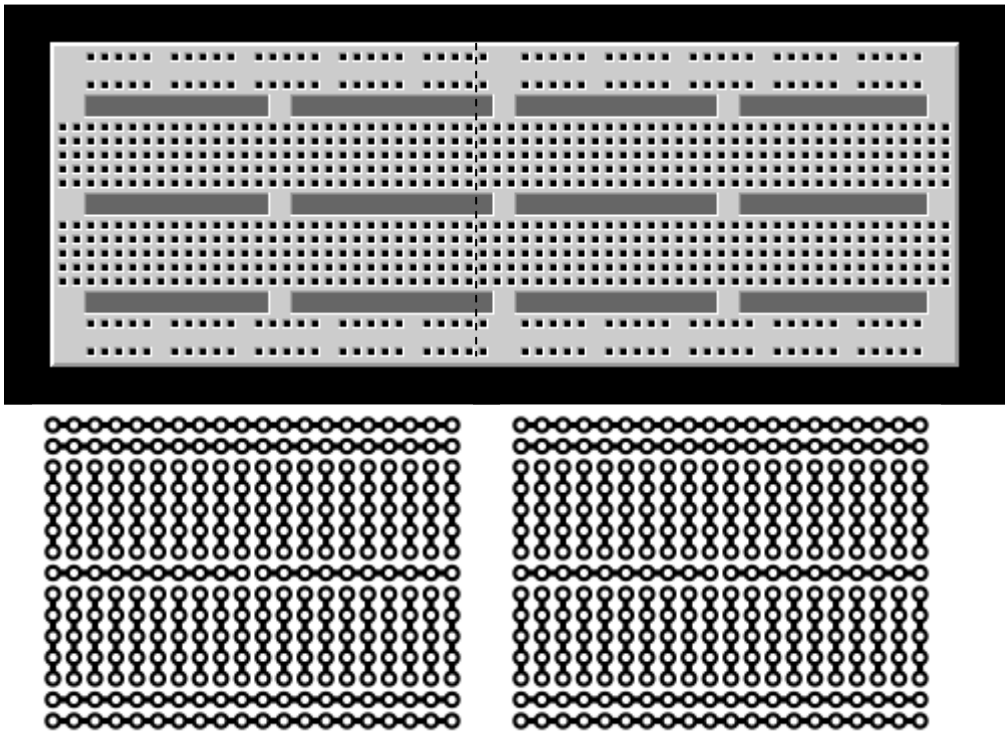


Fig. 1: (a) A typical Breadboard and (b) its connection details

A real breadboard is shown in Fig. 1(a) and the connection details on its rear side are shown in Fig. 1(b). The five holes in each individual column on either side of the central groove are electrically connected to each other, but remain insulated from all other sets of holes. In addition to the main columns of holes, however, you'll note four sets or groups of holes along the top and bottom. Each of these consists of five separate sets of five holes each, for a total of 25 holes. These groups of 25 holes are all connected together on either side of the dotted line indicated on Fig.1(a) and needs an external connection if one wishes the entire row to be connected. This makes them ideal for distributing power to multiple ICs or other circuits.

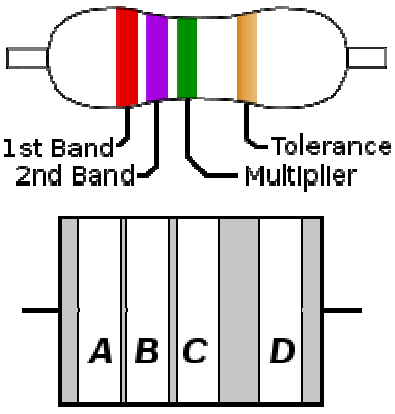
These breadboard sockets are sturdy and rugged, and can take quite a bit of handling. However, there are a few rules you need to observe, in order to extend the useful life of the electrical contacts and to avoid damage to components. These rules are:

- Always make sure power is disconnected when constructing or modifying your experimental circuit. It is possible to damage components or incur an electrical shock if you leave power connected when making changes.
- Never use larger wire as jumpers. #24 wire (used for normal telephone wiring) is an excellent choice for this application. Observe the same limitation with respect to the size of component leads.
- Whenever possible, use ¼ watt resistors in your circuits. ½ watt resistors may be used when necessary; resistors of higher power ratings should never be inserted directly into a breadboard socket.
- Never force component leads into contact holes on the breadboard socket. Doing so can damage the contact and make it useless.
- Do not insert stranded wire or soldered wire into the breadboard socket. If you must have stranded wire (as with an inductor or transformer lead), solder (or use a wire nut to connect) the stranded wire to a short length of solid hookup wire, and insert only the solid wire into the breadboard.

If you follow these basic rules, your breadboard will last indefinitely, and your experimental components will last a long time.

Resistors

Most axial resistors use a pattern of colored stripes to indicate resistance. A 4 band identification is the most commonly used color coding scheme on all resistors. It consists of four colored bands that are painted around the body of the resistor. Resistor values are always coded in ohms (Ω). The color codes are given in the following table in Fig. 1.



Band Color	Digit	Multiplier	Tolerance
Black	0	1	---
Brown	1	10	±1%
Red	2	100	±2%
Orange	3	1,000	±3%
Yellow	4	10,000	±4%
Green	5	100,000	---
Blue	6	1,000,000	---
Violet	7	10,000,000	---
Gray	8	100,000,000	---
White	9	---	---
Gold	---	0.1	±5%
Silver	---	0.01	±10%
None	---	---	±20%

Fig. 1: Color codes of Resistors

- band **A** is first significant figure of component value
- band **B** is the second significant figure
- band **C** is the decimal multiplier
- band **D** if present, indicates tolerance of value in percent (no color means 20%)

For example, a resistor with bands of *yellow, violet, red, and gold* will have first digit 4 (yellow in table below), second digit 7 (violet), followed by 2 (red) zeros: 4,700 ohms. Gold signifies that the tolerance is ±5%, so the real resistance could lie anywhere between 4,465 and 4,935 ohms.

Tight tolerance resistors may have three bands for significant figures rather than two, and/or an additional band indicating temperature coefficient, in units of ppm/K. For large power resistors and potentiometers, the value is usually written out implicitly as "10 kΩ", for instance.

Capacitors:

You will mostly use electrolytic and ceramic capacitors for your experiments.

Electrolytic capacitors

An **electrolytic capacitor** is a type of capacitor that uses an electrolyte, an ionic conducting liquid, as one of its plates, to achieve a larger capacitance per unit volume than other types. They are used in relatively high-current and low-frequency electrical circuits. However, the voltage applied to these capacitors must be polarized; one specified terminal must always have positive potential with respect to the other. These are of two types, axial and radial capacitors as shown in adjacent figure. The arrowed stripe indicates the polarity, with the arrows pointing towards the negative pin.

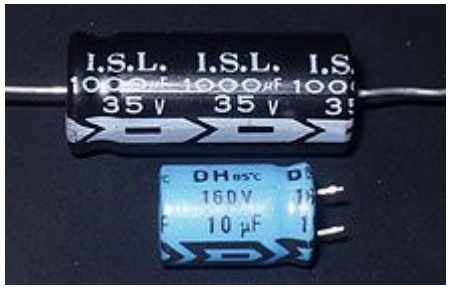


Fig. 2:Axial and Radial Electrolytic capacitors

Warning: connecting electrolytic capacitors in reverse polarity can easily damage or destroy the capacitor. Most large electrolytic capacitors have the voltage, capacitance, temperature ratings, and company name written on them without having any special color coding schemes.

Axial electrolytic capacitors have connections on both ends. These are most frequently used in devices where there is no space for vertically mounted capacitors.

Radial electrolytic capacitors are like axial electrolytic ones, except both pins come out the same end. Usually that end (the "bottom end") is mounted flat against the PCB and the capacitor rises perpendicular to the PCB it is mounted on. This type of capacitor probably accounts for at least 70% of capacitors in consumer electronics.

Ceramic capacitors are generally non-polarized and almost as common as radial electrolytic capacitors. Generally, they use an alphanumeric marking system. The number part is the same as for resistors, except that the value represented is in pF. They may also be written out directly, for instance, 2n2 = 2.2 nF.

SMT

Fig. 3: Ceramic capacitors

Diodes:

A standard specification sheet usually has a brief description of the diode. Included in this description is the type of diode, the major area of application, and any special features. Of particular interest is the specific application for which the diode is suited. The manufacturer also provides a drawing of the diode which gives dimension, weight, and, if appropriate, any identification marks. In addition to the above data, the following information is also provided: a static operating table (giving spot values of parameters under fixed conditions), sometimes a characteristic curve (showing how parameters vary over the full operating range), and diode ratings (which are the limiting values of operating conditions outside which could cause diode damage). Manufacturers specify these various diode operating parameters and characteristics with "letter symbols" in accordance with fixed definitions. The following is a list, by letter symbol, of the major electrical characteristics for the rectifier and signal diodes.

RECTIFIER DIODES

DC BLOCKING VOLTAGE [V_R]—the maximum reverse dc voltage that will not cause breakdown.

AVERAGE FORWARD VOLTAGE DROP [$V_{F(AV)}$]—the average forward voltage drop across the rectifier given at a specified forward current and temperature.

AVERAGE RECTIFIER FORWARD CURRENT [$I_{F(AV)}$]—the average rectified forward current at a specified temperature, usually at 60 Hz with a resistive load.

AVERAGE REVERSE CURRENT [$I_{R(AV)}$]—the average reverse current at a specified temperature, usually at 60 Hz.

PEAK SURGE CURRENT [I_{SURGE}]—the peak current specified for a given number of cycles or portion of a cycle.

SIGNAL DIODES

PEAK REVERSE VOLTAGE [PRV]—the maximum reverse voltage that can be applied before reaching the breakdown point. (PRV also applies to the rectifier diode.)

REVERSE CURRENT [I_R]—the small value of direct current that flows when a semiconductor diode has reverse bias.

MAXIMUM FORWARD VOLTAGE DROP AT INDICATED FORWARD CURRENT [$V_F@I_F$]— the maximum forward voltage drop across the diode at the indicated forward current.

REVERSE RECOVERY TIME [t_{rr}]—the maximum time taken for the forward-bias diode to recover its reverse bias.

The ratings of a diode (as stated earlier) are the limiting values of operating conditions, which if exceeded could cause damage to a diode by either voltage breakdown or overheating.

The PN junction diodes are generally rated for: MAXIMUM AVERAGE FORWARD CURRENT, PEAK RECURRENT FORWARD CURRENT, MAXIMUM SURGE CURRENT, and PEAK REVERSE VOLTAGE

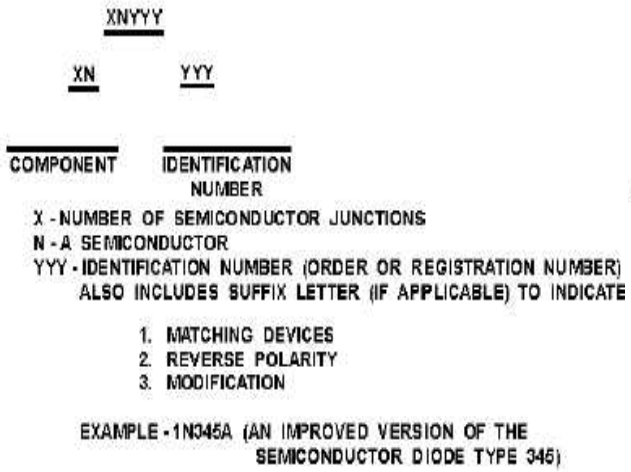
Maximum average forward current is usually given at a special temperature, usually 25º C, (77º F) and refers to the maximum amount of average current that can be permitted to flow in the forward direction. If this rating is exceeded, structure breakdown can occur.

Peak recurrent forward current is the maximum peak current that can be permitted to flow in the forward direction in the form of recurring pulses.

Maximum surge current is the maximum current permitted to flow in the forward direction in the form of nonrecurring pulses. Current should not equal this value for more than a few milliseconds.

Peak reverse voltage (PRV) is one of the most important ratings. PRV indicates the maximum reverse-bias voltage that may be applied to a diode without causing junction breakdown. All of the above ratings are subject to change with temperature variations. If, for example, the operating temperature is above that stated for the ratings, the ratings must be decreased.

There are many types of diodes varying in size from the size of a pinhead (used in subminiature circuitry) to large 250-ampere diodes (used in high-power circuits). Because there are so many different types of diodes, some system of identification is needed to distinguish one diode from another. This is accomplished with the semiconductor identification system shown in Fig. 4. This system is not only used for diodes but transistors and many other special semiconductor devices as well. As illustrated in this figure, the system uses numbers and letters to identify different types of semiconductor devices. The first number in the system indicates the number of junctions in the semiconductor device and is a number, one less than the number of active elements. Thus 1 designates a diode; 2 designates a transistor (which may be considered as made up of two diodes); and 3 designates a tetrode (a four-element transistor). The letter "N" following the first number indicates a semiconductor. The 2- or 3-digit number following the letter "N" is a serialized identification number. If needed, this number may contain a suffix letter after the last digit. For example, the suffix letter "M" may be used to describe matching pairs of separate semiconductor devices or the letter "R" may be used to indicate reverse polarity. Other letters are used to indicate modified versions of the device which can be substituted for the basic numbered unit. For example, a semiconductor diode designated as type 1N345A signifies a two-element diode (1) of semiconductor material (N) that is an improved version (A) of type 345.



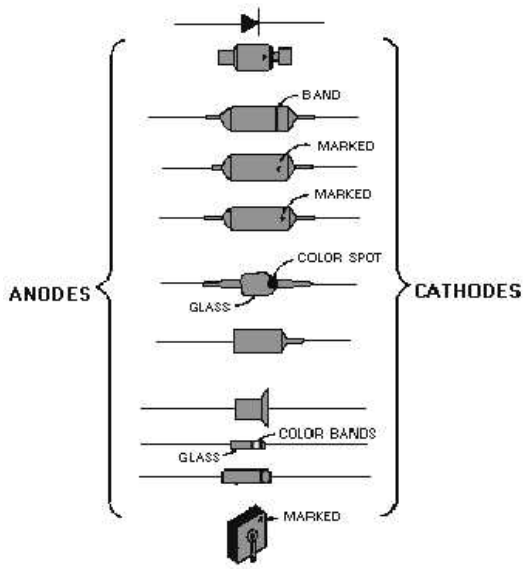


Fig. 4: Identification of Diode **Fig. 5: Identification of Cathode**

When working with different types of diodes, it is also necessary to distinguish one end of the diode from the other (anode from cathode). For this reason, manufacturers generally code the cathode end of the diode with a "k," "+," "cath," a color dot or band, or by an unusual shape (raised edge or taper) as shown in Fig. 5. In some cases, standard color code bands are placed on the cathode end of the diode. This serves two purposes: (1) it identifies

the cathode end of the diode, and (2) it also serves to identify the diode by number.

Transistors:
Transistors are identified by a Joint Army-Navy (JAN) designation printed directly on the case of the transistor. If in doubt about a transistor's markings, always replace a transistor with one having identical markings, or consult an equipment or transistor manual to ensure that an identical replacement or substitute is used.

Example:

2	N	130	A
NUMBER OF JUNCTIONS (TRANSISTOR)	SEMICONDUCTOR	IDENTIFICATION NUMBER	FIRST MODIFICATION

There are three main series of transistor codes used:

- **Codes beginning with B (or A), for example BC108, BC478**
  The first letter B is for silicon, A is for germanium (rarely used now). The second letter indicates the type; for example C means low power audio frequency; D means high power audio frequency; F means low power high frequency. The rest of the code identifies the particular transistor. There is no obvious logic to the numbering system. Sometimes a letter is added to the end (eg BC108C) to identify a special version of the main type, for example a higher current gain or a different case style. If a project specifies a higher gain version (BC108C) it must be used, but if the general code is given (BC108) any transistor with that code is suitable.
- **Codes beginning with TIP, for example TIP31A**
  TIP refers to the manufacturer: Texas Instruments Power transistor. The letter at the end identifies versions with different voltage ratings.
- **Codes beginning with 2N, for example 2N3053**
  The initial '2N' identifies the part as a transistor and the rest of the code identifies the particular transistor. There is no obvious logic to the numbering system.

TESTING A TRANSISTOR to determine if it is good or bad can be done with an ohmmeter or transistor tester. PRECAUTIONS should be taken when working with transistors since they are susceptible to damage by electrical overloads, heat, humidity, and radiation. TRANSISTOR LEAD IDENTIFICATION plays an important part in transistor maintenance because before a transistor can be tested or replaced, its leads must be identified. Since there is NO standard method of identifying transistor leads, check some typical lead identification schemes or a transistor manual before attempting to replace a transistor. Identification of leads for some common case styles is shown in Fig. 6.

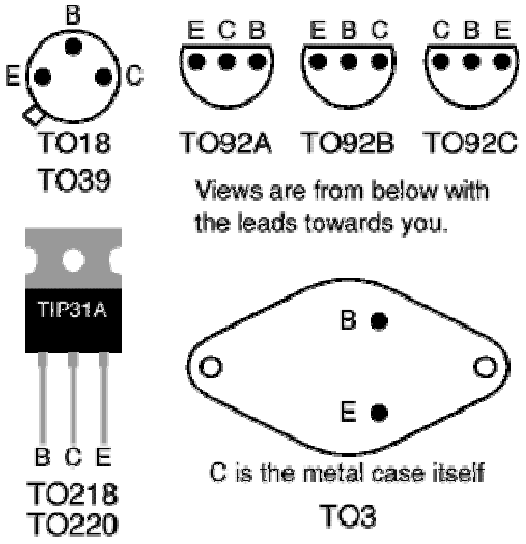


Fig. 6

Testing a transistor

Transistors are basically made up of two *Diodes* connected together back-to-back (Fig. 7). We can use this analogy to determine whether a transistor is of the type PNP or NPN by testing its Resistance between the three different leads, Emitter, Base and Collector.

Testing with a multimeter

Use a multimeter or a simple tester (battery, resistor and LED) to check each pair of leads for conduction. Set a digital multimeter to diode test and an analogue multimeter to a low resistance range.

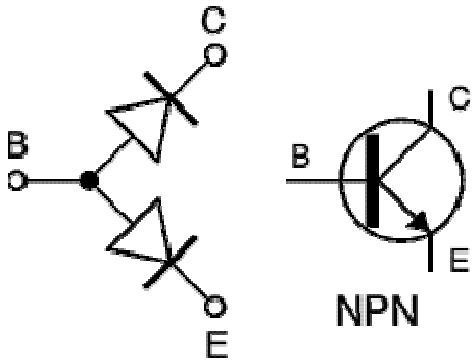


Fig. 7: Testing an NPN transistor

Test each pair of leads both ways (six tests in total):

- The **base-emitter (BE)** junction should behave like a diode and conduct one way only.
- The **base-collector (BC)** junction should behave like a diode and conduct one way only.
- The **collector-emitter (CE)** should not conduct either way.

The diagram shows how the junctions behave in an NPN transistor. The diodes are reversed in a PNP transistor but the same test procedure can be used.

Transistor Resistance Values for the PNP and NPN transistor types

Between Transistor Terminals		PNP	NPN
Collector	Emitter	R_{HIGH}	R_{HIGH}
Collector	Base	R_{LOW}	R_{HIGH}
Emitter	Collector	R_{HIGH}	R_{HIGH}
Emitter	Base	R_{LOW}	R_{HIGH}
Base	Collector	R_{HIGH}	R_{LOW}
Base	Emitter	R_{HIGH}	R_{LOW}

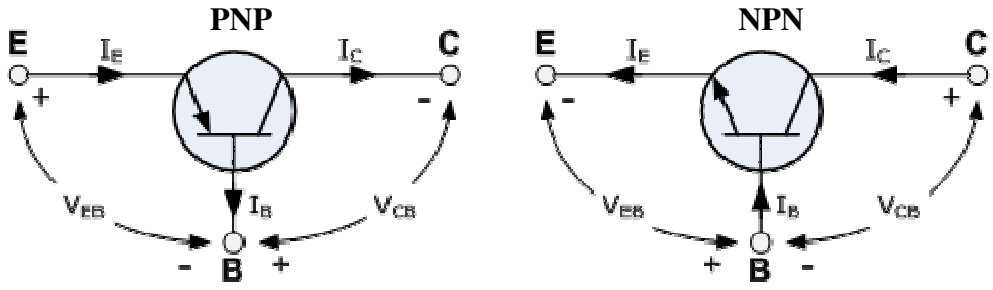
Bipolar Junction Transistor Static Characteristics

Objective:

(i) To study the input and output characteristics of a PNP transistor in Common Base mode and determine transistor parameters.
(ii) To study the input and output characteristics of an NPN transistor in Common Emitter mode and determine transistor parameters.

Overview:

A **Bipolar Junction Transistor**, or **BJT** is a three terminal device having two PN-junctions connected together in series. Each terminal is given a name to identify it and these are known as the Emitter (E), Base (B) and Collector (C). There are two basic types of bipolar transistor construction, NPN and PNP, which basically describes the physical arrangement of the P-type and N-type semiconductor materials from which they are made. Bipolar Transistors are "CURRENT" Amplifying or current regulating devices that control the amount of current flowing through them in proportion to the amount of biasing current applied to their base terminal. The principle of operation of the two transistor types NPN and PNP, is exactly the same the only difference being in the biasing (base current) and the polarity of the power supply for each type.



The symbols for both the NPN and PNP bipolar transistor are shown above along with the direction of conventional current flow. The direction of the arrow in the symbol shows current flow between the base and emitter terminal, pointing from the positive P-type region to the negative N-type region, exactly the same as for the standard diode symbol. For normal operation, the emitter-base junction is forward-biased and the collector-base junction is reverse-biased.

Transistor Configurations

There are three possible configurations possible when a transistor is connected in a circuit: (a) Common base, (b) Common emitter (c) Common collector. We will be focusing on the first two configurations in this experiment. The behaviour of a transistor can be represented by d.c. current-voltage (I-V) curves, called the static characteristic curves of the device. The three important characteristics of a transistor are: (i) Input characteristics, (ii) Output characteristics and (iii) Transfer Characteristics. These characteristics give information about various transistor parameters, e.g. input and out dynamic resistance, current amplification

factors, etc.

Common Base Transistor Characteristics

In common base configuration, the base is made common to both input and output as shown in its circuit diagram.

(1) ***Input Characteristics:*** The input characteristics is obtained by plotting a curve between I_E and V_{EB} keeping voltage V_{CB} constant. This is very similar to that of a forward-biased diode and the slope of the plot at a given operating point gives information about its input dynamic resistance.

Input Dynamic Resistance (r_i) This is defined as the ratio of change in base emitter voltage (ΔV_{EB}) to the resulting change in emitter current (ΔI_E) at constant collector-emitter voltage (V_{CB}). This is dynamic as its value varies with the operating current in the transistor.

$$r_i = \left.\frac{\Delta V_{EB}}{\Delta I_E}\right|_{V_{CB}}$$

(2) ***Output Characteristics:*** The output characteristic curves are plotted between I_C and V_{CB}, keeping I_E constant. The output characteristics are controlled by the input characteristics. Since I_C changes with I_E, there will be different output characteristics corresponding to different values of I_E. These curves are almost horizontal. This shows that the output dynamic resistance, defined below, is very high.

Output Dynamic Resistance (r_o): This is defined as the ratio of change in collector-base voltage (ΔV_{CB}) to the change in collector current (ΔI_C) at a constant base current I_E.

$$r_o = \left.\frac{\Delta V_{CB}}{\Delta I_C}\right|_{I_E}$$

(3) ***Transfer Characteristics****:* The transfer characteristics are plotted between the input and output currents (I_E versus I_C).

Current amplification factor (α)

This is defined as the ratio of the change in collector current to the change in emitter current at a constant collector-base voltage (V_{CB}) when the transistor is in active state.

$$\alpha_{ac} = \left.\frac{\Delta I_C}{\Delta I_E}\right|_{V_{CB}}$$

This is also known as small signal current gain and its value is very large. The ratio of I_C and I_E is called α_{dc} of the transistor. Hence,

$$\alpha_{dc} = \left.\frac{I_C}{I_E}\right|_{V_{CB}}$$

Since I_C increases with I_E almost linearly, the values of both α_{dc} and α_{ac} are nearly equal.

Common Emitter Transistor Characteristics

In a common emitter configuration, emitter is common to both input and output as shown in its circuit diagram.

(1) ***Input Characteristics:*** The variation of the base current I_B with the base-emitter voltage V_{BE} keeping the collector-emitter voltage V_{CE} fixed, gives the input characteristic in CE mode.

Input Dynamic Resistance (r_i): This is defined as the ratio of change in base emitter voltage (ΔV_{BE}) to the resulting change in base current (ΔI_B) at constant collector-emitter voltage (V_{CE}). This is dynamic and it can be seen from the input characteristic, its value varies with the operating current in the transistor:

$$r_i = \left.\frac{\Delta V_{BE}}{\Delta I_B}\right|_{V_{CE}}$$

The value of r_i can be anything from a few hundreds to a few thousand ohms.

(2) ***Output Characteristics:*** The variation of the collector current I_C with the collector-emitter voltage V_{CE} is called the output characteristic. The plot of I_C versus V_{CE} for different fixed values of I_B gives one output characteristic. Since the collector current changes with the base current, there will be different output characteristics corresponding to different values of I_B.

Output Dynamic Resistance (r_o): This is defined as the ratio of change in collector-emitter voltage (ΔV_{CE}) to the change in collector current (ΔI_C) at a constant base current I_B.

$$r_o = \left.\frac{\Delta V_{CE}}{\Delta I_C}\right|_{I_B}$$

The high magnitude of the output resistance (of the order of 100 kW) is due to the reverse-biased state of this diode.

(3) ***Transfer Characteristics****:* The transfer characteristics are plotted between the input and output currents (I_B versus I_C). Both I_B and I_C increase proportionately.

Current amplification factor (β)

This is defined as the ratio of the change in collector current to the change in base current at a constant collector-emitter voltage (V_{CE}) when the transistor is in active state.

$$\beta_{ac} = \left.\frac{\Delta I_C}{\Delta I_B}\right|_{V_{CE}}$$

This is also known as small signal current gain and its value is very large. The ratio of I_C and I_B we get what is called β_{dc} of the transistor. Hence,

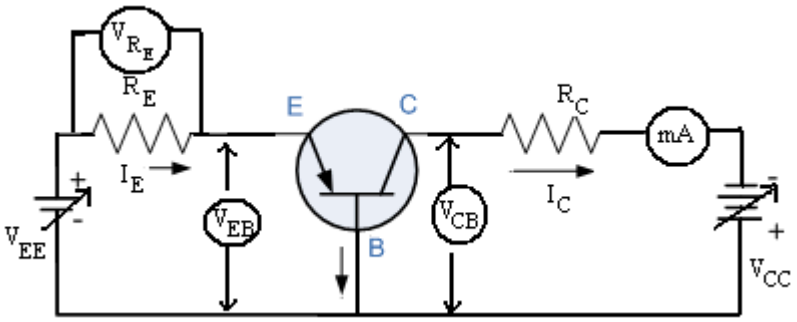
$$\beta_{dc} = \left.\frac{I_C}{I_B}\right|_{V_{CE}}$$

Since I_C increases with I_B almost linearly, the values of both β_{dc} and β_{ac} are nearly equal.

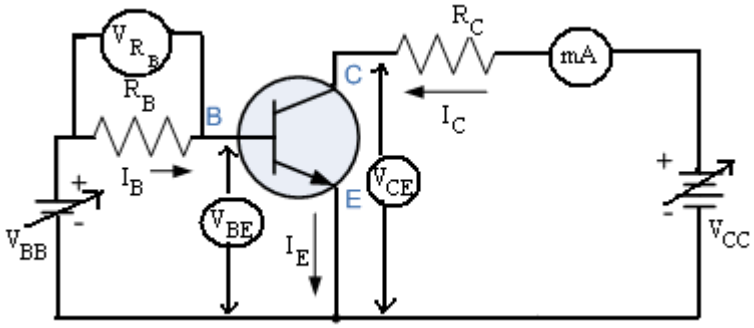
Circuit components/Equipments:

(i) Transistors (2 Nos: 1 PNP (CK 100 or equivalent) and 1 NPN (BC 107 or equivalent)), (ii) Resistors (4 Nos.) (iii) Multimeters (3 Nos.), (iv) D.C. power supply, (v) Connecting wires and (vi) Breadboard.

Circuit Diagrams:



PNP transistor in CB configuration



NPN transistor in CE configuration

Procedure:

1. Note down the type number of both the transistors.
2. Identify different terminals (E, B and C) and the type (PNP/NPN) of the transistors. For any specific information refer the datasheet of the transistors.

(I) PNP Common Base (CB) characteristics

1. Configure CB circuit using the PNP transistor as per the circuit diagram. Use $R_E = R_C =$ 150 Ω.
2. For input characteristics, first fix the voltage V_{CB} by adjusting V_{CC} to the minimum possible position. Now vary the voltage V_{EB} slowly (say, in steps of 0.05V) by varying V_{EE}. Measure V_{EB} using a multimeter. If V_{CB} varies during measurement bring it back to the initial set value To determine I_E, measure V_{RE} across the resistor R_E and use the relation $I_E = V_{RE}/R_E$.
3. Repeat the above step for another value of V_{CB} say, 2V.
4. Take out the multimeter measuring V_{EB} and connect in series with the output circuit to measure I_C. For output characteristics, first fix $I_E = 0$, i.e. $V_{RE} = 0$. By adjusting V_{CC}, vary the collector voltage V_{CB} in steps of say 1V and measure V_{CB} and the corresponding I_C using multimeters. After acquiring sufficient readings, bring back V_{CB} to 0 and reduce it further to get negative values. Vary V_{CB} in negative direction and measure both V_{CB} and I_C, till you get 0 current.
5. Repeat the above step for at least 5 different values of I_E by adjusting V_{EE}. You may need to adjust V_{EE} continuously during measurement in order to maintain a constant I_E.
6. Plot the input and output characteristics by using the readings taken above and determine the input and output dynamic resistance.
7. To plot transfer characteristics, select a suitable voltage V_{CB} well within the active region of the output characteristics, which you have tabulated already. Plot a graph between I_C and the corresponding I_E at the chosen voltage V_{CB}. Determine α_{ac} from the slope of this graph.

(II) NPN Common Emitter (CE) characteristics

1. Now configure CE circuit using the NPN transistor as per the circuit diagram. Use $R_B =$ 100kΩ and $R_C = 1$ kΩ.
2. For input characteristics, first fix the voltage V_{CE} by adjusting V_{CC} to the minimum possible position. Now vary the voltage V_{BE} slowly (say, in steps of 0.05V) by varying V_{BB}. Measure V_{BE} using a multimeter. If V_{CE} varies during measurement bring it back to the set value To determine I_B, measure V_{RB} across the resistor R_B and use the relation $I_B = V_{RB}/R_B$.
3. Repeat the above step for another value of V_{CE} say, 2V.

4. For output characteristics, first fix $I_B = 0$, i.e. $V_{RB} = 0$. By adjusting V_{CC}, vary the collector voltage V_{CE} in steps of say 1V and measure V_{CE} and the corresponding I_C using multimeters. If needed vary V_{CE} in negative direction as described for CB configuration and measure both V_{CE} and I_C, till you get 0 current.
5. Repeat the above step for at least 5 different values of I_B by adjusting V_{BB}. You may need to adjust V_{BB} continuously during measurement in order to maintain a constant I_B.
6. Plot the input and output characteristics by using the readings taken above and determine the input and output dynamic resistance.
7. Plot the transfer characteristics between I_C and I_B as described for CB configuration for a suitable voltage of V_{CE} on the output characteristics. Determine β_{ac} from the slope of this graph.

Observations:

CB configuration:

Transistor code: _____, Transistor type: _____ (PNP/NPN)
R_E = _____, R_C = _____.

Table (1): Input Characteristics

Sl. No.	**V_{CB} = ___V**			**V_{CB} = ___V**		
	V_{EB} (V)	**V_{RE} (V)**	**I_E (mA)**	**V_{EB} (V)**	**V_{RE} (V)**	**I_E (mA)**
1						
2						
..						
..						
10						

Table (2): Output Characteristics

Sl. No.	**$I_{E1} = 0$**		**I_{E2} = __**		**I_{E3} = __**		**I_{E4} = __**		**I_{E5} = __**	
	V_{CB} (V)	**I_C (mA)**	**V_{CB} (V)**	**I_C (mA)**	**V_{CB} (V)**	**I_C (mA)**	**V_{CB} (V)**	**I_C (mA)**	**V_{CB} (V)**	**I_C (mA)**
1										
2										
..										
..										
10										

Table (3): Transfer Characteristics V_{CB} = _____ V

Sl. No.	I_E (mA)	I_C (mA)
1		
2		
3		
4		
5		

CE configuration: Transistor code: _____, Transistor type: _____ (PNP/NPN)
R_B = _____, R_C = _____.

Table (5): Input Characteristics

Sl. No.	V_{CE} = ___V			V_{CE} = ___V		
	V_{BE} (V)	V_{RB} (V)	I_B (µA)	V_{BE} (V)	V_{RB} (V)	I_B (µA)
1						
2						
..						
..						
10						

Table (4): Output Characteristics

Sl. No.	I_{B1} = 0		I_{B2} = __		I_{B3} = __		I_{B4} = __		I_{B5} = __	
	V_{CE} (V)	I_C (mA)	V_{CE} (V)	I_C (mA)	V_{CE} (V)	I_C (mA)	V_{CE} (V)	I_C (mA)	V_{CE} (V)	I_C (mA)
1										
2										
..										
..										
10										

Table (6): Transfer Characteristics V_{CE} = _____ V

Sl. No.	I_B (µA)	I_C (mA)
1		
2		
3		
4		
5		

Graphs:

Plot the input, output and transfer characteristics for each configuration.

CB configuration:

(1) Input characteristics: Plot V_{EB} ~ I_E, for different V_{CB} and determine the input dynamic resistance in each case at suitable operating points.
(2) Output characteristics: Plot V_{CB} ~ I_C, for different I_E and determine the output dynamic resistance in each case at suitable operating points in the active region.
(3) Transfer characteristics: Plot I_E ~ I_C, for a fixed V_{CB} and determine α_{ac}.

CE configuration:

(1) Input characteristics: Plot V_{BE} ~ I_B, for different V_{CE} and determine the input dynamic resistance in each case at suitable operating points.
(2) Output characteristics: Plot V_{CE} ~ I_C, for different I_B and determine the output dynamic resistance in each case at suitable operating points in the active region.
(3) Transfer characteristics: Plot I_B ~ I_C, for a fixed V_{CE} and determine β_{ac}.

Results/Discussions:

Precautions:

Study of Common Emitter Transistor Amplifier circuit

Objectives:

1. To design a common emitter transistor (NPN) amplifier circuit.
2. To obtain the frequency response curve of the amplifier and to determine the mid-frequency gain, A_{mid}, lower and higher cutoff frequency of the amplifier circuit.

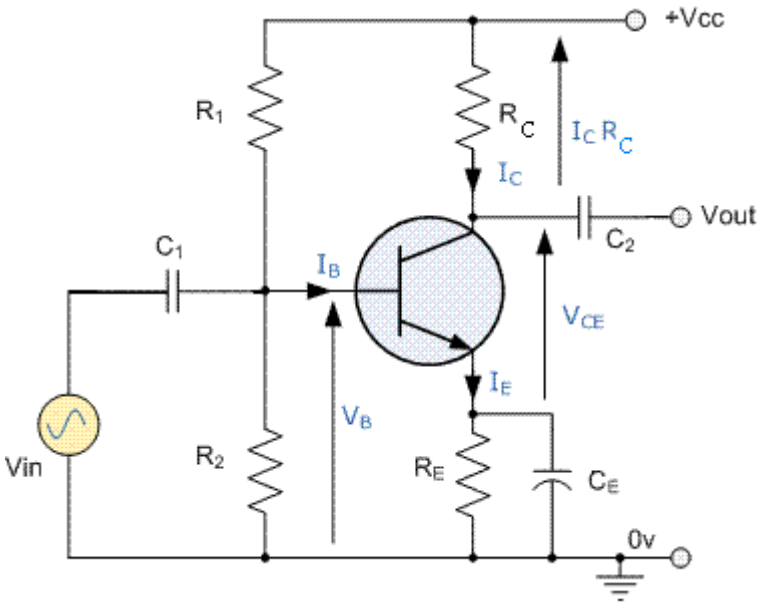
Overview:

The most common circuit configuration for an NPN transistor is that of the *Common Emitter Amplifier* and that a family of curves known commonly as the *Output Characteristics Curves*, relates the Collector current (I_C), to the output or Collector voltage (V_{CE}), for different values of Base current (I_B). All types of transistor amplifiers operate using AC signal inputs which alternate between a positive value and a negative value. Presetting the amplifier circuit to operate between these two maximum or peak values is achieved using a process known as *Biasing*. Biasing is very important in amplifier design as it establishes the correct operating point of the transistor amplifier ready to receive signals, thereby reducing any distortion to the output signal.

The single stage common emitter amplifier circuit shown below uses what is commonly called "Voltage Divider Biasing". The Base voltage (V_B) can be easily calculated using the simple voltage divider formula below:

$$V_B = \frac{V_{CC} R_2}{R_1 + R_2}$$

Thus the base voltage is fixed by biasing and independent of base current provided the current in the divider circuit is large compared to the base current. Thus assuming $I_B \approx 0$, one can do the approximate analysis of the voltage divider network without using the transistor



gain, β, in the calculation. Note that the approximate approach can be applied with a high degree of accuracy when the following condition is satisfied:

$$\beta R_E \geq 10 R_2$$

Load line and Q-point

A static or DC load line can be drawn onto the output characteristics curves of the transistor to show all the possible operating points of the transistor from fully "ON" ($I_C = V_{CC}/(R_C + R_E)$) to fully "OFF" ($I_C = 0$). The quiescent operating point or **Q-point** is a point on this load line which represents the values of I_C and V_{CE} that exist in the circuit when no input signal is applied. Knowing V_B, I_C and V_{CE} can be calculated to locate the operating point of the circuit as follows:

$$V_E = V_B - V_{BE}$$

So, the emitter current,

$$I_E \approx I_C = \frac{V_E}{R_E}$$

$$\text{and } V_{CE} = V_{CC} - I_C(R_C + R_E)$$

It can be noted here that the sequence of calculation does not need the knowledge of β and I_B is not calculated. So the Q-point is stable against any replacement of the transistor.

Since the aim of any small signal amplifier is to generate an amplified input signal at the output with minimum distortion possible, the best position for this Q-point is as close to the centre position of the load line as reasonably possible, thereby producing a Class A type amplifier operation, i.e. $V_{CE} = 1/2V_{CC}$.

Coupling and Bypass Capacitors

In CE amplifier circuits, capacitors C_1 and C_2 are used as Coupling Capacitors to separate the AC signals from the DC biasing voltage. The capacitors will only pass AC signals and block any DC component. Thus they allow coupling of the AC signal into an amplifier stage without disturbing its Q point. The output AC signal is then superimposed on the biasing of the following stages. Also a bypass capacitor, C_E is included in the Emitter leg circuit. This capacitor is an open circuit component for DC bias, meaning that the biasing currents and voltages are not affected by the addition of the capacitor maintaining a good Q-point stability. However, this bypass capacitor acts as a short circuit path across the emitter resistor at high frequency signals increasing the voltage gain to its maximum. Generally, the value of the bypass capacitor, C_E is chosen to provide a reactance of at most, 1/10th the value of R_E at the lowest operating signal frequency.

Amplifier Operation

Once the Q-point is fixed through DC bias, an AC signal is applied at the input using coupling capacitor C_1. During positive half cycle of the signal V_{BE} increases leading to increased I_B. Therefore I_C increases by β times leading to decrease in the output voltage, V_{CE}. Thus the CE amplifier produces an amplified output with a phase reversal. The voltage Gain of the common emitter amplifier is equal to the ratio of the change in the output voltage to the change in the input voltage. Thus,

$$A_V = \frac{V_{out}}{V_{in}} = \frac{\Delta V_{CE}}{\Delta V_{BE}}$$

The input (Z_i) and output (Z_o) impedances of the circuit can be computed for the case when

the emitter resistor R_E is completely bypassed by the capacitor, C_E:

$$\mathbf{Z_i = R_1 \left\| R_2 \right\| \beta r_e \text{ and } Z_o = R_C \left\| r_o \right.}$$

where r_e (26mV/I_E) and r_o are the emitter diode resistance and output dynamic resistance (can be determined from output characteristics of transistor). Usually $r_o \geq 10\ R_C$, thus the gain can be approximated as

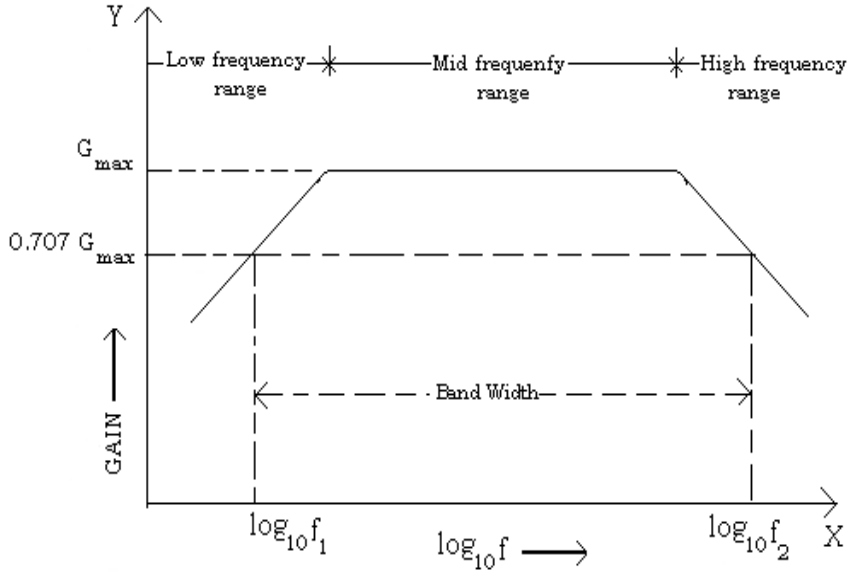
$$A_V = \frac{V_{out}}{V_{in}} = -\frac{\beta I_B \left(R_C \| r_o\right)}{I_B \beta r_e} \cong -\frac{R_C}{r_e}$$

The negative sign accounts for the phase reversal at the output. In the circuit diagram provided below, the emitter resistor is split into two in order to reduce the gain to avoid distortion. So the expression for gain is modified as

$$A_V \cong -\frac{R_C}{(R_{E1} + r_e)}$$

Frequency Response Curve

The performance of an amplifier is characterized by its frequency response curve that shows output amplitude (or, more often, voltage gain) plotted versus frequency (often in log scale). Typical plot of the voltage gain of an amplifier versus frequency is shown in the figure below. The frequency response of an amplifier can be divided into three frequency ranges.



The frequency response begins with the lower frequency range designated between 0 Hz and lower cutoff frequency. At lower cutoff frequency, f_L, the gain is equal to 0.707 A_{mid}. A_{mid} is a constant mid-band gain obtained from the mid-frequency range. The third, the higher frequency range covers frequency between upper cutoff frequency and above. Similarly, at higher cutoff frequency, f_H, the gain is equal to 0.707 A_{mid}. Beyond this the gain decreases with frequency increases and dies off eventually.

The Lower Frequency Range

Since the impedance of coupling capacitors increases as frequency decreases, the voltage gain of a BJT amplifier decreases as frequency decreases. At very low frequencies, the capacitive reactance of the coupling capacitors may become large enough to drop some of the input voltage or output voltage. Also, the emitter-bypass capacitor may become large enough

so that it no longer shorts the emitter resistor to ground.

The Higher Frequency Range

The capacitive reactance of a capacitor decreases as frequency increases. This can lead to problems for amplifiers used for high-frequency amplification. The ultimate high cutoff frequency of an amplifier is determined by the physical capacitances associated with every component and of the physical wiring. Transistors have internal capacitances that shunt signal paths thus reducing the gain. The high cutoff frequency is related to a shunt time constant formed by resistances and capacitances associated with a node.

Design:

Before designing the circuit, one needs to know the circuit requirement or specifications. The circuit is normally biased for V_{CE} at the mid-point of load line with a specified collector current. Also, one needs to know the value of supply voltage V_{CC} and the range of β for the transistor being used (available in the datasheet of the transistor).

Here the following specifications are used to design the amplifier:

V_{CC} = 12V and I_C = 1 mA

Start by making V_E= 0.1 V_{CC}. Then $R_E = V_E/I_E$ (Use $I_E \approx I_C$).

Since V_{CE} = 0.5 V_{CC}, Voltage across R_C = 0.4V_{CC}, i.e. $R_C = 4.R_E$

In order that the approximation analysis can be applied, $\boldsymbol{R_2 \leq 0.1\beta R_E}$. Here β is the minimum rated value in the specified range provided by the datasheet (in this case β =50).

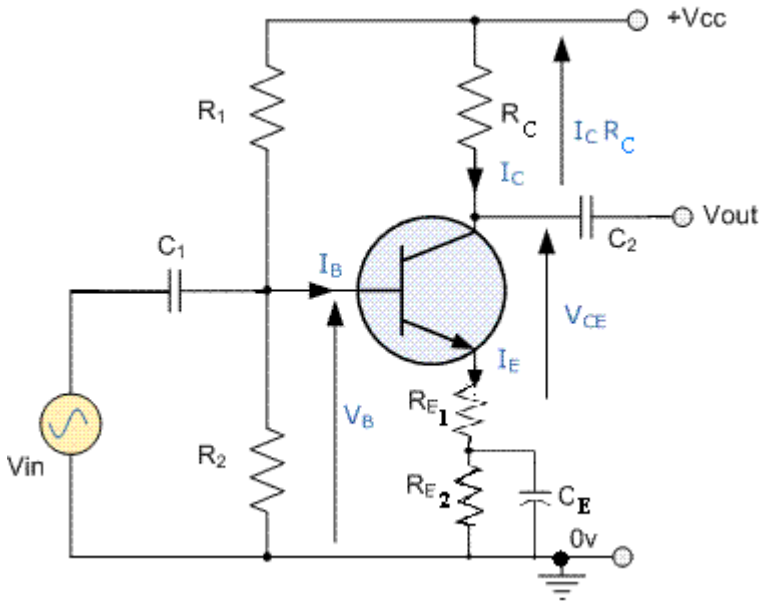
Finally, $\boldsymbol{R_1 = \frac{V_1}{V_2} R_2}$, V_1 (= V_{CC}-V_2)and V_2 (= V_E+V_{BE}) are voltages across R_1 and R_2, respectively.

Based on these guidelines the components are estimated and the nearest commercially available values are used.

Components/ Equipments:

1. Transistor: CL100 (or equivalent general purpose npn)
2. Resistors: R_1= 26 (27) KΩ, R_2= 5 (4.7+0.22) KΩ, R_C = 4 (3.9) KΩ, R_E= 1kΩ (R_{E1}=470 Ω, R_{E2}=560 Ω)
3. Capacitors: C_1= C_2= 1 μF (2 nos.), C_E= 100μF
4. Power Supply (V_{CC} = 12V)
5. Oscilloscope
6. Function Generator (~ 100-200 mV pp, sinusoidal for input signal)
7. Breadboard
8. Connecting wires

Circuit Diagram:



Procedure:

1. Measure and record all the values of resistance and capacitance and β of the transistor using a multimeter. Configure the circuit as per the diagram.
2. Apply supply voltage to the circuit. Measure and record all the dc parameters listed in Table 1 in absence of the ac signal.
3. Next, set the function generator in 20Hz "Frequency" range. Also, set the "Attenuation" button at 40dB. Connect the output to the oscilloscope and adjust the "Amplitude" knob till you get a sinusoidal input signal, Vi ≈ 100-200 mV peak-to-peak value. DO NOT CHANGE THIS SETTING THROUGHOUT THE EXPERIMENT.
4. Now apply this input signal to the circuit you have made keeping the connection to oscilloscope in tact. Feed the output of the circuit to the other channel of oscilloscope. Take care to make all the ground pins common.
5. With input signal amplitude always constant, increase signal frequency slowly. Observe, measure and record the output voltage, V_o. Scan the entire frequency in the range 20 Hz – 2 MHz. You may have to measure V_i and take the ratio V_o/V_i each time in case input fluctuation is too large to hold constant.
6. Calculate the voltage gain for each frequency. Observe the inverted output.
7. Plot the frequency response curve, i.e. voltage gain in dB versus frequency on a semi-log graph-sheet.
8. Estimate the mid-frequency gain and also the lower and higher cut off frequencies and hence the bandwidth.

Observations:

β = _____

R_1= ____, R_2= ____, R_C= ____, R_E= ____; C_1= ___, C_2= ___, C_E= ___

Table 1: D.C. analysis of the circuit

$V_{CC} = 12V$

Parameter	Computed value	Observed value
V_B (V)		
V_E (V)		
$I_C \approx I_E$ (mA)		
V_{CE} (V)		

Q-point is at (__V, __mA)

Table 2: Frequency response

$V_i(pp)$ = ___ mV

Sl. No.	Frequency, f (kHz)	V_0(pp) (Volt)	Gain,$A_V = \frac{V_o(pp)}{V_i(pp)}$	Gain (dB)
1				
2				
..				
..				

Calculations: r_e = _____ , Z_i = _____, Z_o = _____
Theoretical value of A_v in mid-frequency range = _____

Graphs: Plot the frequency response curve (semi-log plot) and determine the cut-off frequencies, bandwidth and mid- frequency gain.

Discussions:

Precautions:

1. Vary the input signal frequency slowly.
2. Connect electrolytic capacitors carefully.

Reference: Electronic devices and circuit theory, Robert L. Boylestad & Louis Nashelsky (10^{th} Edition)

Two Stage RC Coupled Transistor Amplifier

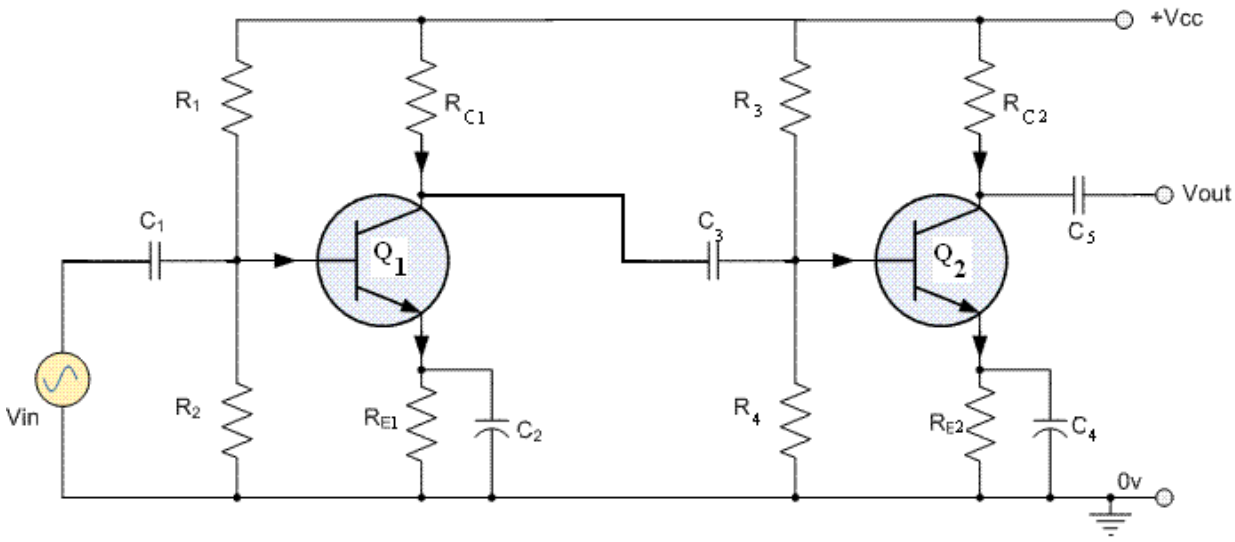
Objective:

1. To design a two stage RC coupled common emitter transistor (NPN) amplifier circuit and to study its frequency response curve.

Overview:

A single stage of amplification is often not enough for a particular application. The overall gain can be increased by using more than one stage, so when two amplifiers are connected in such a way that the output signal of the first serves as the input signal to the second, the amplifiers are said to be connected in *cascade.* The most common arrangement is the common-emitter configuration.

Resistance-capacitance (RC) coupling is most widely used to connect the output of first stage to the input (base) of the second stage and so on. It is the most popular type of coupling because it is cheap and provides a constant amplification over a wide range of frequencies. These R-C coupled amplifier circuits are commonly used as voltage amplifiers in the audio systems.



The circuit diagram above shows the 2-stages of an R-C coupled amplifier in CE configuration using NPN transistors. Capacitors C_1 and C_3 couple the input signal to transistors Q_1 and Q_2, respectively. C_5 is used for coupling the signal from Q_2 to its load. R_1, R_2, R_{E1} and R_3, R_4, R_{E2} are used for biasing and stabilization of stage 1 and 2 of the amplifier. C_2 and C_4 provide low reactance paths to the signal through the emitter.

Overall gain:

The total gain of a 2-stage amplifier is equal to the product of individual gain of each stage. (You may refer to the handout for single stage amplifier to calculate individual gain of the stages.) Once the second stage is added, its input impedance acts as an additional load on the first stage thereby reducing the gain as compared to its no load gain. Thus the overall gain characteristics is affected due to this loading effect.
The loading of the second stage i.e. input impedance of second stage, $\mathbf{Z_{i2} = R_3 \parallel R_4 \parallel \beta r_{e2}}$

Thus loaded gain of the first stage, $A_{V1} = -\dfrac{R_{C1} \| Z_{i2}}{r_{e1}}$

and the unloaded gain of second stage, $A_{V2} = -\dfrac{R_{C2}}{r_{e2}}$

In the circuit diagram provided below, the emitter resistor is split into two in order to reduce the gain to avoid distortion. So the expression for gain each stage is modified as

$$A_{V1} = -\frac{R_{C1} \| Z_{i2}}{(R_{E1} + r_{e1})}$$

$$A_{V2} = -\frac{R_{C2}}{R_{E2} + r_{e2}}$$

The overall gain of the 2 stage amplifier is $A_V = A_{V1} \times A_{V2}$.

Frequency Response Curve

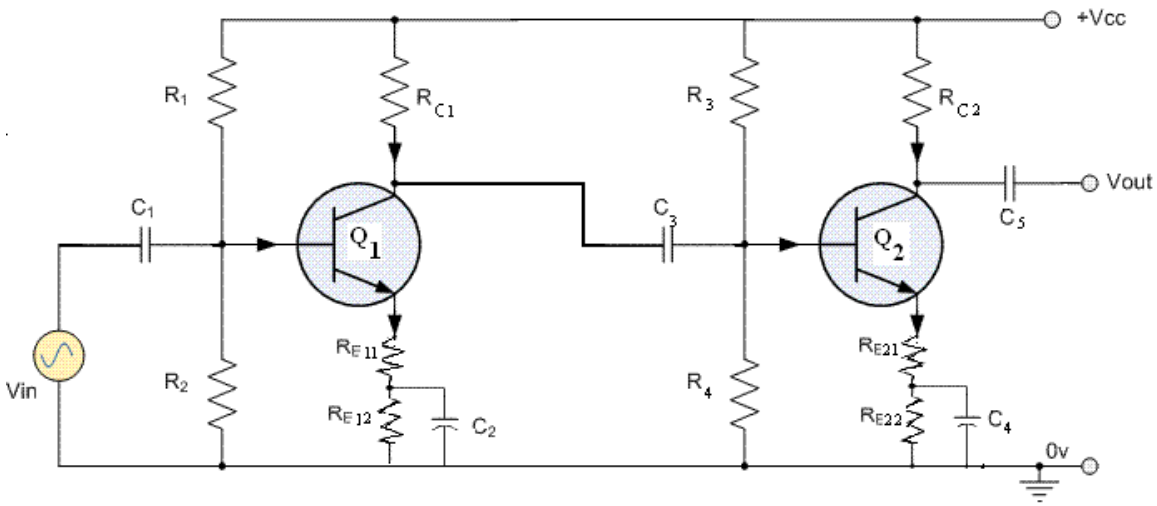
The performance of an amplifier is characterized by its frequency response curve that shows voltage gain (often expressed in dB units) plotted versus frequency. The frequency response begins with the lower frequency region designated between 0 Hz and lower cutoff frequency. At lower cutoff frequency, f_L, the gain is equal to 0.707 A_{mid}. A_{mid} is a constant mid-band gain obtained from the mid-band frequency region. The third, the upper frequency region covers frequency between upper cutoff frequency and above. Similarly, at upper cutoff frequency, f_H, the gain is equal to 0.707 A_{mid}. Beyond the upper cutoff frequency, the gain decreases as the frequency increases and dies off eventually. (More details are given in the hand out for single stage amplifier.)

Design: The design details are already given in the single stage amplifier hand out.

Components/ Equipments:

1. Transistor: CL100 (or equivalent general purpose npn, 2 nos)
2. Resistors: R_1, R_3= 26 (27) KΩ, R_2, R_4= 5 (4.7+0.22) KΩ, R_{C1}, R_{C2}= 4 (3.9) KΩ, R_{E1}, R_{E2}= 1kΩ (R_{E11}, R_{E21}=470 Ω; R_{E12}, R_{E22}=560 Ω) (2 nos. of each resistance value)
3. Capacitors: C_1= C_3= C_5= 1 μF (3 nos.), C_2= C_4= 100μF (2 nos.)
4. Power Supply (V_{CC} = 12V)
5. Oscilloscope
6. Function Generator (~ 100-200 mV pp, sinusoidal for input signal)
7. Breadboard
8. Connecting wires

Circuit Diagram:



Procedure:

1. Measure and record all the values of resistance and capacitance and β of the transistor using a multimeter. Configure the circuit as per the diagram. Make a provision so that the two stages can either be separated or connected as and when required.
2. Apply supply voltage to the circuit. Measure and record all the dc parameters of each individual stage separately as listed in Table 1 in absence of the ac signal.
3. Next, set the function generator at **20kHz by putting the frequency knob in 20KHz range and adjusting the variable knob.** Also, set the "Attenuation" button at 40dB. Connect the output to the oscilloscope and adjust the "Amplitude" knob till you get a sinusoidal input signal, Vi ≈ 100-200 mV peak-to-peak value. DO NOT CHANGE THIS SETTING THROUGHOUT THE EXPERIMENT.
4. To fill up first row of Table 2 the two stages should be separated. Now apply the input signal to the circuit at the first stage keeping the connection to oscilloscope intact. Feed the output of the first stage to the other channel of oscilloscope. Take care to make all the ground pins common. Measure the output and calculate unloaded gain of stage 1.
5. Similarly by applying input at the second stage measure the output and calculate the unloaded gain of second stage.
6. For the second row of Table 2, connect the two stages. Apply the input signal at the first stage Measure the output of first stage and calculate its loaded gain.
7. Now, with the input signal at the first stage, measure the output at the second stage and calculate the total gain of the two stage amplifier and complete Table-2.
8. To study the frequency response of the two stage amplifier, vary the input signal frequency in the range 20 Hz – 2 MHz, keeping the input signal amplitude always constant. Observe measure and record the output voltage, V_o at the second stage. (You may have to measure V_i and take the ratio V_o/V_i each time in case input fluctuation is too large to hold constant.) Calculate voltage gain for each frequency.
9. Plot the frequency response curve, i.e. voltage gain in dB versus frequency on a semi-log graph-sheet.
10. Estimate the mid-frequency gain and also the lower and higher cut off frequencies and hence the bandwidth.

Observations:

$\beta 1$ = _____ , $\beta 2$ = _____

Stage 1: R_1= ____, R_2= ____, R_C= ____, R_E= ____+___; C_1= ___, C_2= ___, C_E= ___
Stage 2: R_1= ____, R_2= ____, R_C= ____, R_E= ____+___; C_1= ___, C_2= ___, C_E= ___

Table 1: D.C. analysis of the circuit

V_{CC} = 12V

Parameter	Stage 1 (Q1)		Stage 2 (Q2)	
	Computed value	Observed value	Computed value	Observed value
V_B (V)				
V_E (V)				
$I_C \approx I_E$ (mA)				
V_{CE} (V)				
$r_e(\Omega)$				
Q-point				

Table 2: Mid frequency voltage Gain ($f \approx 20$ kHz)

Vi = _____

Parameter	Stage 1 (Q1)		Stage 2 (Q2)	
	Computed	Measured	Computed	Measured
Unloaded Voltage Gain (V_o/ V_i)				
Loaded Voltage Gain				

Total mid frequency gain = Loaded Voltage Gain (Q1) × Unloaded Voltage Gain (Q2)

Total gain (computed) = _____

Total gain (measured) = _____

Table 3: Frequency Response

V_i(pp) = _____

Sl. No.	Frequency, f (kHz)	V_0(pp) (Volt)	Gain,$A_V = \frac{V_o(pp)}{V_i(pp)}$	Gain (dB)
1				

2				
..				
..				

Calculations: Stage 1: Z_{i1} = _____, Z_{o1} = _____

Stage 2: Z_{i2} = _____, Z_{o2} = _____

Graphs: Plot the frequency response curve and determine the cut-off frequencies, bandwidth and mid-band gain.

Discussions:

Precautions:

3. Vary the input signal frequency slowly.
4. Connect electrolytic capacitors carefully.

Reference: Electronic devices and circuit theory, Robert L. Boylestad & Louis Nashelsky (10^{th} Edition)

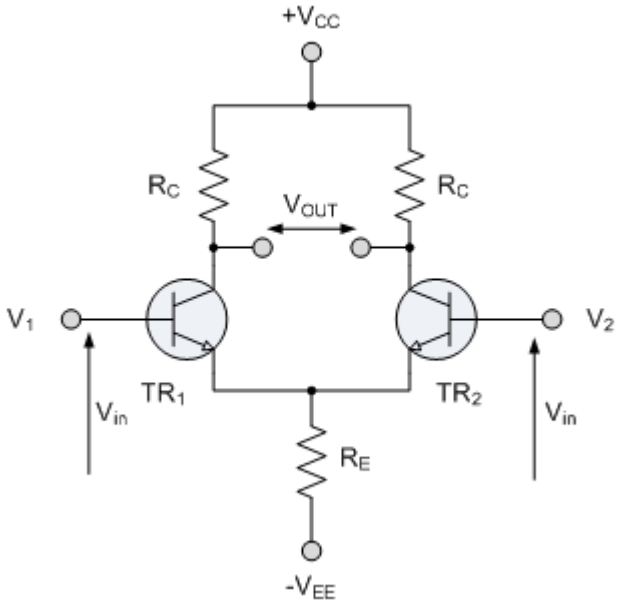
Operational Amplifiers
(Supplementary note)

Ideal Operational Amplifier

As well as resistors and capacitors, **Operational Amplifiers**, or **Op-amps** as they are more commonly called, are one of the basic building blocks of Analogue Electronic Circuits. It is a linear device that has all the properties required for nearly ideal DC amplification and is used extensively in signal conditioning, filtering or to perform mathematical operations such as add, subtract, integration and differentiation. An ideal operational amplifier is basically a 3-terminal device that consists of two high impedance inputs, one an **Inverting input** marked with a negative sign, ("-") and the other a **Non-inverting input** marked with a positive plus sign ("+").

The amplified output signal of an Operational Amplifier is the difference between the two signals being applied to the two inputs. In other words the output signal is a *differential* signal between the two inputs and the input stage of an Operational Amplifier is in fact a differential amplifier as shown below.

Differential Amplifier

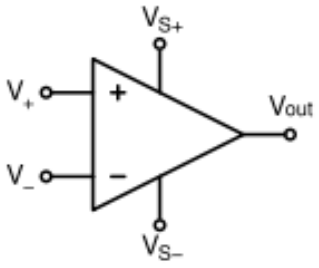


The circuit shows a generalized form of a differential amplifier with two inputs marked V_1 and V_2. The two identical transistors TR_1 and TR_2 are both biased at the same operating point with their emitters connected together and returned to the common rail, $-V_{EE}$ by way of resistor R_E. The circuit operates from a dual supply $+V_{CC}$ and $-V_{EE}$ which ensures a constant supply. As the two base inputs are out of phase with each other, the output voltage, V_{OUT}, is the difference between the two input signals. So, as the forward bias of transistor TR_1 is increased, the forward bias of transistor TR_2 is reduced and vice versa. Then if the two transistors are perfectly matched, the current flowing through the common emitter resistor, R_E will remain constant.

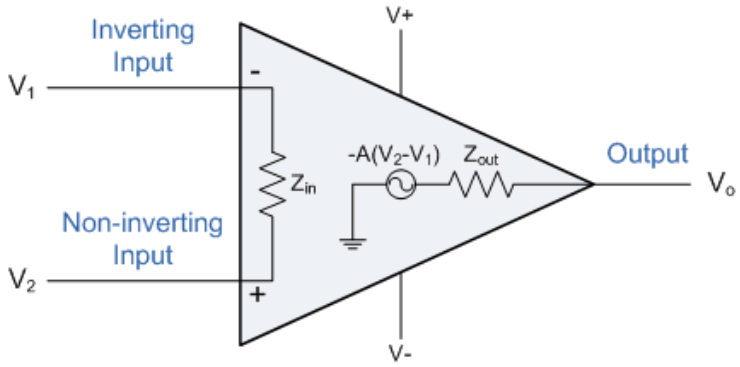
Ideal Operational Amplifiers have an output of low impedance that is referenced to a common ground terminal and it should ignore any common mode signals. That means, if identical signals are applied to both the inverting and non-inverting inputs there should be no change at the output. However, in real amplifiers there is always some variation and the ratio of the change to the output voltage with regards to the change in the common mode input voltage is called the **Common Mode Rejection Ratio** or **CMRR**.

Operational Amplifiers have a very high open loop DC gain, commonly known as the

Open Loop Differential Gain, and is given the symbol (A_o). By applying some form of **Negative Feedback** we can produce an operational amplifier circuit with a very precise gain characteristic that is dependent only on the feedback used. An operational amplifier only responds to the difference between the voltages at its two input terminals, known commonly as the "*Differential Input Voltage*" and not to their common potential. Then if the same voltage potential is applied to both terminals the resultant output will be zero.



Symbol of OpAmp



Equivalent Circuit for Ideal OpAmp

Idealized Characteristics

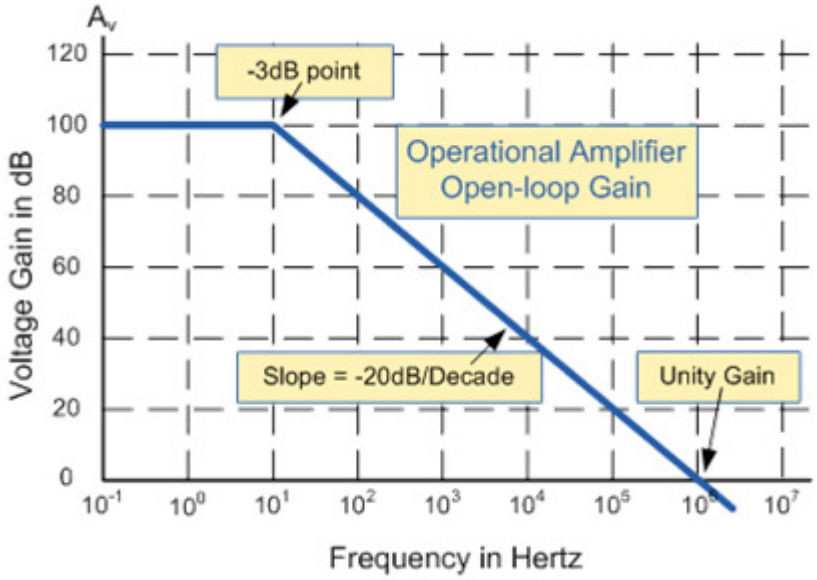
PARAMETER	IDEALIZED CHARACTERISTIC
Voltage Gain, (A)	**Infinite** - The main function of an operational amplifier is to amplify the input signal and the more open loop gain it has the better, so for an ideal amplifier the gain will be infinite.
Input impedance, (Z_{in})	**Infinite** - Input impedance is assumed to be infinite to prevent any current flowing from the source supply into the amplifiers input circuitry.
Output impedance, (Z_{out})	**Zero** - The output impedance of the ideal operational amplifier is assumed to be zero so that it can supply as much current as necessary to the load.
Bandwidth, (BW)	**Infinite** - An ideal operational amplifier has an infinite Frequency Response and can amplify any frequency signal so it is assumed to have an infinite bandwidth.
Offset Voltage, (V_{io})	**Zero** - The amplifiers output will be zero when the voltage difference between the inverting and non-inverting inputs is zero.

It is important to remember two properties known as the **golden rules**, as they help understand the working of the amplifier with regards to analysis and design of operational amplifier circuits.

1. **No current flows into either input terminal** (the current rule)
2. **The differential input offset voltage is zero** (the voltage rule).

However, real **Operational Amplifiers** (e.g. 741) do not have infinite gain or bandwidth but have a typical "Open Loop Gain" which is defined as the amplifiers output amplification without any external feedback signals connected to it and for a typical operational amplifier is about 100dB at DC (zero Hz). This output gain decreases linearly with frequency down to "Unity Gain" or 1, at about 1MHz and this is shown in the following open loop gain response curve. From this frequency response curve we can see that the product of the gain against frequency is constant at any point along the curve. Also that the unity gain (0dB) frequency also determines the gain of the amplifier at any point along the curve. This constant is generally known as the **Gain Bandwidth Product** or **GBP**.
Therefore, GBP = Gain × Bandwidth or A × BW.

Open-loop Frequency Response Curve



For example, from the graph above the gain of the amplifier at 100kHz = 20dB or 10, then the
GBP = 100,000Hz x 10 = 1,000,000.
Similarly, a gain at 1kHz = 60dB or 1000, therefore the
GBP = 1,000 x 1,000 = 1,000,000. **The same!**.

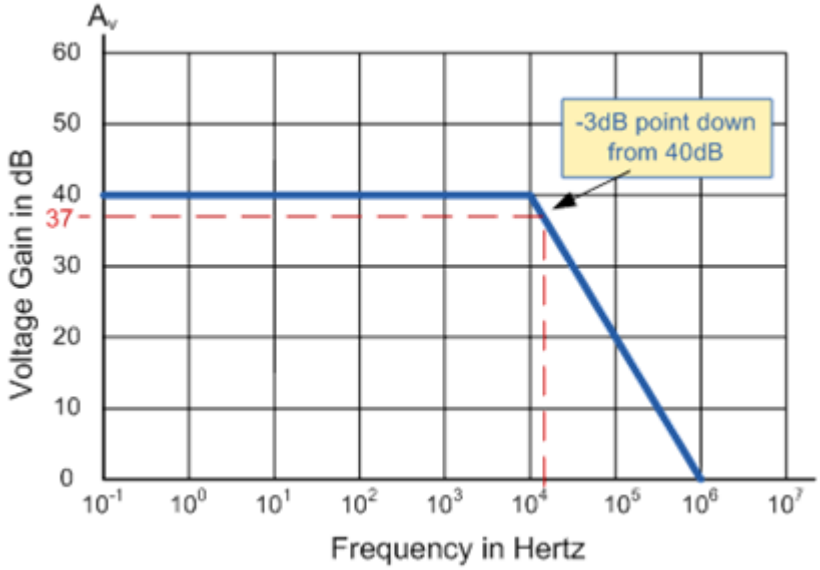
The **Voltage Gain** (A) of the amplifier can be found using the following formula:

$$\textit{Gain}, A = \frac{V_o}{V_i}$$

and in **Decibels** or (dB) is given as:

$$\mathbf{20}\log A = \mathbf{20}\log\frac{V_o}{V_i}$$

Bandwidth of Operational Amplifier



The operational amplifiers bandwidth is the frequency range over which the voltage gain of the amplifier is above **70.7%** or **-3dB** (where 0dB is the maximum) of its maximum output value as shown below.

Here we have used the 40dB line as an example. The -3dB or 70.7% of V_{max} down point from the frequency response curve is given as 37dB. Taking a line across until it intersects with the main GBP curve gives us a frequency point just above the 10kHz line at about 12 to 15kHz. We can now calculate this more accurately as we already know the GBP of the amplifier, in this particular case 1MHz.

Example No1.
Using the formula 20 log (A), we can calculate the bandwidth of the amplifier as:
37 = 20 log A therefore, A = anti-log (37 ÷ 20) = 70.8
GBP ÷ A = Bandwidth, therefore, 1,000,000 ÷ 70.8 = 14.124Hz, or 14kHz
Then the bandwidth of the amplifier at a gain of 40dB is given as **14kHz** as predicted from the graph.

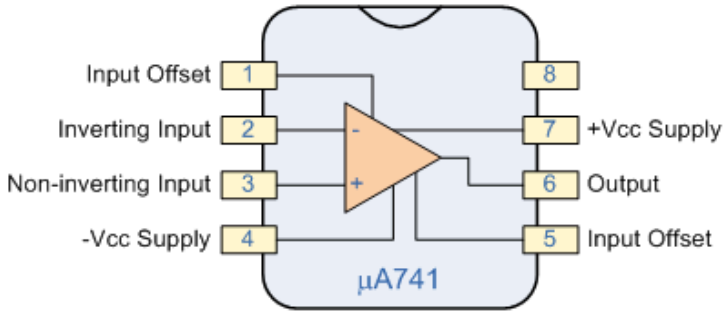
Example No2.
If the gain of the operational amplifier was reduced by half to say **20dB** in the above frequency response curve, the -3dB point would now be at 17dB. This would then give us an overall gain of 7.08, therefore **A = 7.08**. If we use the same formula as above this new gain would give us a bandwidth of **141.2kHz**, ten times more than at 40dB.

It can therefore be seen that by reducing the overall open loop gain of an operational amplifier its bandwidth is increased and vice versa. The -3dB point is also known as the "half power point", as the output power of the amplifier is at half its maximum value at this point.

Op-amp types

Operational amplifiers can be connected using external resistors or capacitors in a number of different ways to form basic "Building Block" circuits such as, Inverting, Non-

Inverting, Voltage Follower, Summing, Differential, Integrator and Differentiator type amplifiers. There are a very large number of operational amplifier IC's available to suit every possible application.
The most commonly available and used of all operational amplifiers is the industry standard **741** type IC.



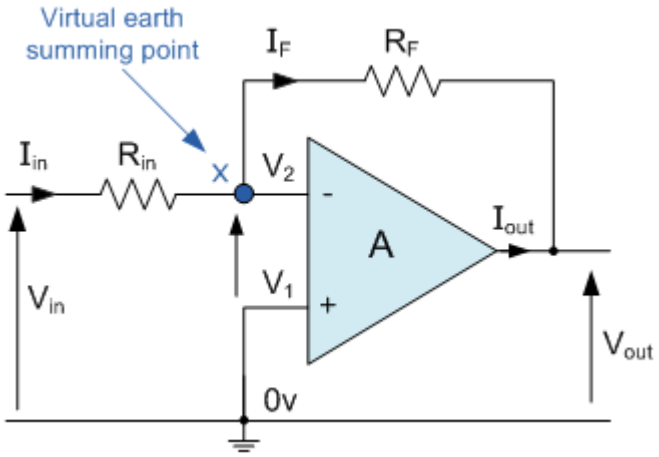
Inverting Amplifier

The open loop gain of an ideal Operational Amplifier can be very high, up to about 1,000,000 (120dB) or more. However, this very high gain is of no real use to us as it makes the amplifier both unstable and hard to control as the smallest of input signals, just a few micro-volts, would be enough to cause the output to saturate and swing towards one or the other of the voltage supply rails losing control. As the open loop DC gain of an operational amplifier is extremely high we can afford to lose some of this gain by connecting a suitable resistor across the amplifier from the output terminal back to the inverting input terminal to both reduce and control the overall gain of the amplifier. This then produces an effect known commonly as **Negative Feedback**, and thus produces a very stable Operational Amplifier system.

Negative Feedback is the process of "feeding back" some of the output signal back to the input, but to make the feedback negative we must feed it back to the "Negative input" terminal using an external **Feedback Resistor** called R_f. This feedback connection between the output and the inverting input terminal produces a closed loop circuit to the amplifier resulting in the gain of the amplifier now being called its **Closed-loop Gain**.

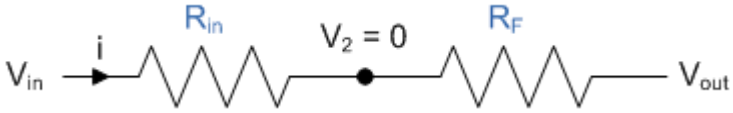
This results in the inverting input terminal having a different signal on it than the actual input voltage as it will be the sum of the input voltage plus the negative feedback voltage giving it the label or term of a *Summing Point*. We must therefore separate the real input signal from the inverting input by using an **Input Resistor**, R_{in}. As we are not using the positive non-inverting input this is connected to a common ground or zero voltage terminal as shown below. But the effect of this closed loop feedback circuit results in the voltage at the inverting input equal to that at the non-inverting input producing a *Virtual Earth* summing point because it will be at the same potential as the grounded reference input.

Inverting Amplifier Circuit



In inverting amplifier circuit the operational amplifier is connected with feedback to produce a closed loop operation. There are two very important rules to remember about inverting amplifiers: "no current flows into the input terminal" and that "V_1 equals V_2". This is because the junction of the input and feedback signal (X) is at the same potential as the positive (+) input which is at zero volts or ground then, the junction is a **"Virtual Earth"**. Because of this virtual earth node the input resistance of the amplifier is equal to the value of the input resistor, R_{in}. Then by using these two rules one can find the equation for calculating the gain of an inverting amplifier, using first principles.
Current (i) flows through the resistor network as shown.

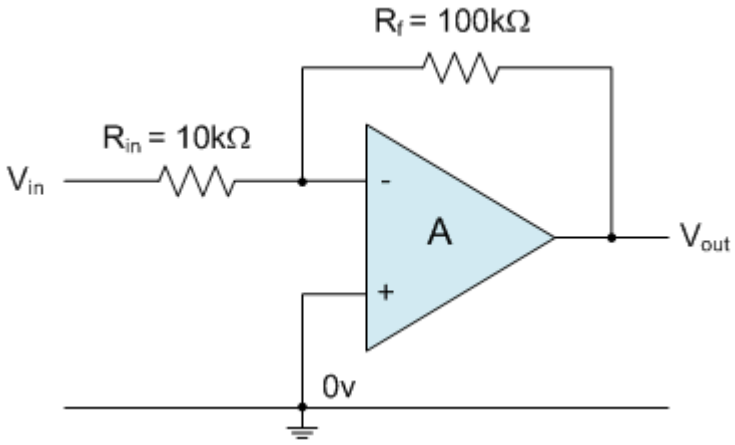
$$i = \frac{V_{in}}{R_{in}} = -\frac{V_o}{R_f}$$



The negative sign in the equation indicates an inversion of the output signal with respect to the input as it is 180° out of phase. This is due to the feedback being negative in value.
Then, the **Closed-Loop Voltage Gain** of an Inverting Amplifier is given as.

$$Gain = \frac{V_o}{V_{in}} = -\frac{R_f}{R_{in}}$$

Example No1
Find the closed loop gain of the given inverting amplifier circuit.



Using the previously found formula for the gain of the circuit

$$Gain = \frac{V_o}{V_{in}} = -\frac{R_f}{R_{in}}$$

$R_{in} = 10k\Omega$ and $R_f = 100k\Omega$.
Gain = $-R_f/R_{in} = 100k/10k = 10$.
Therefore, the closed loop gain of the given inverting amplifier circuit is given **10** or **20dB**.

Example No2.
The gain of the original circuit is to be increased to **40**, find the new values of the resistors required.
Assume that the input resistor is to remain at the same value of 10KΩ, then by re-arranging the closed loop voltage gain formula we can find the new value required for the feedback

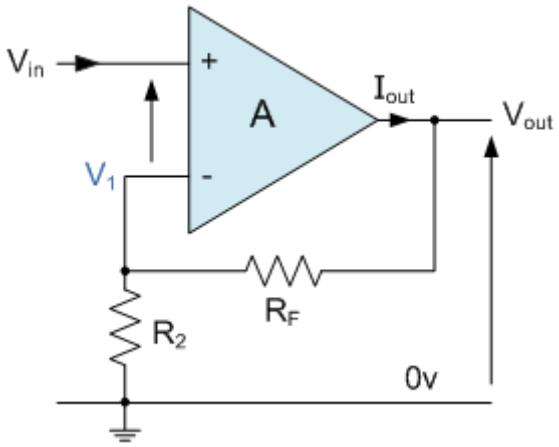
resistor R_f.
Gain = $-R_f/R_{in}$
So, R_f = Gain x R_{in}
Rf = 400,000 or 400KΩ
The new values of resistors required for the circuit to have a gain of **40** would be,
R_{in} = 10KΩ and R_f = 400KΩ.
The formula could also be rearranged to give a new value of R_{in}, keeping the same value of R_f.

Unity Gain Inverter
One final point to note about **Inverting Amplifiers**, if the two resistors are of equal value, R_{in} = R_f then the gain of the amplifier will be **-1** producing a complementary form of the input voltage at its output as $V_{out} = -V_{in}$. This type of inverting amplifier configuration is generally called a **Unity Gain Inverter** of simply an *Inverting Buffer*.

Non-inverting Amplifier

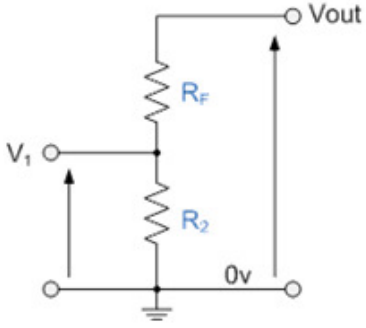
The second basic configuration of an operational amplifier circuit is that of a **Non-inverting Amplifier**. In this configuration, the input voltage signal, (V_{in}) is applied directly to the Non-inverting (+) input terminal which means that the output gain of the amplifier becomes "Positive" in value in contrast to the "Inverting Amplifier" circuit whose output gain is negative in value. Feedback control of the non-inverting amplifier is achieved by applying a small part of the output voltage signal back to the inverting (-) input terminal via a R_f - R_2 voltage divider network, again producing negative feedback. This produces a Non-inverting Amplifier circuit with very good stability, a very high input impedance, R_{in} approaching infinity (as no current flows into the positive input terminal) and a low output impedance, r_{out} as shown below.

Non-inverting Amplifier Circuit



Since no current flows into the input of the amplifier, $V_1 = V_{in}$. In other words the junction is a **"Virtual Earth"** summing point. Because of this virtual earth node, the resistors R_f and R_2 form a simple voltage divider network across the amplifier and the voltage gain of the circuit is determined by the ratios of R_2 and R_f as shown below.

Equivalent Voltage Divider Network
Then using the formula to calculate the output voltage of a potential divider network, we can calculate the output Voltage



Gain of the **Non-inverting Amplifier** as:

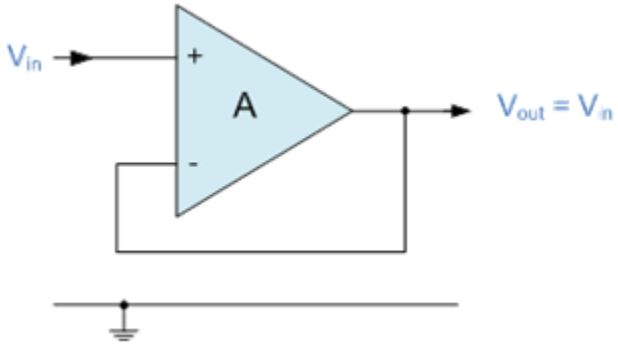
$$V_0 = V_{in}(1 + \frac{R_f}{R_2})$$

$$\boldsymbol{Gain = \frac{V_o}{V_{in}} = 1 + \frac{R_f}{R_2}}$$

We can see that the overall gain of a Non-Inverting Amplifier is greater but never less than 1, is positive and is determined by the ratio of the values of R_f and R_2. If the feedback resistor R_f is zero the gain will be equal to 1, and if resistor R_2 is zero the gain will approach infinity, but in practice it will be limited to the operational amplifiers open-loop differential gain, (A_o).

Voltage Follower (Unity Gain Buffer)

If we made the feedback resistor, $R_f = 0$ then the circuit will have a fixed gain of "1" and would be classed as a **Voltage Follower**. As the input signal is connected directly to the non-inverting input of the amplifier the output signal is not inverted resulting in the output voltage being equal to the input voltage, $V_{out} = V_{in}$. This then makes the Voltage Follower circuit ideal as a *Unity Gain Buffer* circuit because of its isolation properties as impedance or circuit isolation is more important than amplification. The input impedance of the voltage follower circuit is very high, typically above 1MΩ.



In this circuit, R_{in} has increased to infinity and R_f reduced to zero, the feedback is 100% and V_{out} is exactly equal to V_{in} giving it a fixed gain of 1 or unity. As the input voltage V_{in} is applied to the non-inverting input the gain of the amplifier is given as:

$$\boldsymbol{V_o = A(V_{in} - V_o)}$$

$$\boldsymbol{V_{in} = V_+ \quad V_o = V_-}$$

$$\boldsymbol{Gain = \frac{V_o}{V_{in}} = 1}$$

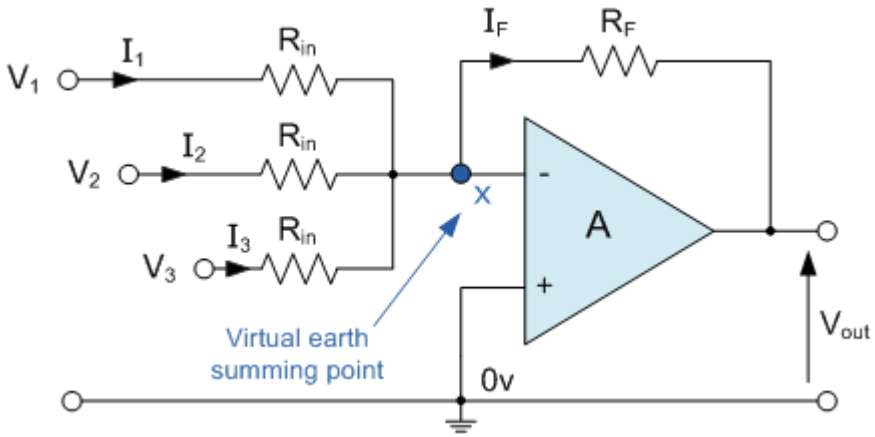
The voltage follower or unity gain buffer is a special and very useful type of **Non-inverting amplifier** circuit that is commonly used in electronics to isolate circuits from each other especially in High-order state variable or Sallen-Key type active filters to separate one filter stage from the other. Typical digital buffer IC's available are the 74LS125 Quad 3-state buffer or the more common 74LS244 Octal buffer.

One final thought, the output voltage gain of the voltage follower circuit with closed loop gain is **Unity**, the voltage gain of an ideal operational amplifier with open loop gain (no feedback) is infinite. Then by carefully selecting the feedback components we can control the amount of gain produced by an Operational Amplifier anywhere from 1 to infinity.

Summing Amplifier

The **Summing Amplifier** is a very flexible circuit based upon the standard *Inverting Operational Amplifier* configuration. We saw previously that the inverting amplifier has a single input signal applied to the inverting input terminal. If we add another input resistor equal in value to the original input resistor, R_{in} we end up with another operational amplifier circuit called a **Summing Amplifier**, "*Summing Inverter*" or even a "*Voltage Adder*" circuit as shown below

Summing Amplifier Circuit



The output voltage, (V_{out}) now becomes proportional to the sum of the input voltages, V_1, V_2, V_3 etc. Then we can modify the original equation for the inverting amplifier to take account of these new inputs thus:

$$I_F = I_1 + I_2 + I_3 = -\left[\frac{V_1}{R_{in}} + \frac{V_2}{R_{in}} + \frac{V_3}{R_{in}}\right]$$

$$then, \quad V_{out} = -\frac{R_F}{R_{in}}(V_1 + V_2 + V_3)$$

The **Summing Amplifier** is a very flexible circuit indeed, enabling us to effectively "Add" or "Sum" together several individual input signals. If the input resistors are all equal a unity gain inverting adder can be made. However, if the input resistors are of different values a "scaling summing amplifier" is produced which gives a weighted sum of the input signals.

Example No1

Find the output voltage of the following *Summing Amplifier* circuit.

The gain of the circuit

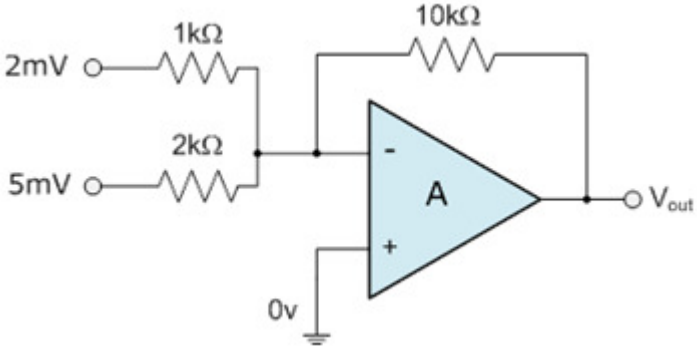
$$Gain = \frac{V_o}{V_{in}} = -\frac{R_f}{R_{in}}$$

substituting the values of the resistors,

$$A_1 = \frac{10k\Omega}{1k\Omega} = -10, \quad A2 = \frac{10k\Omega}{2k\Omega} = -5$$

The output voltage is the sum of the two amplified input signals:

$$V_o = (-10(2mV)) + (-5(5mV))$$
$$= -45\,mV$$

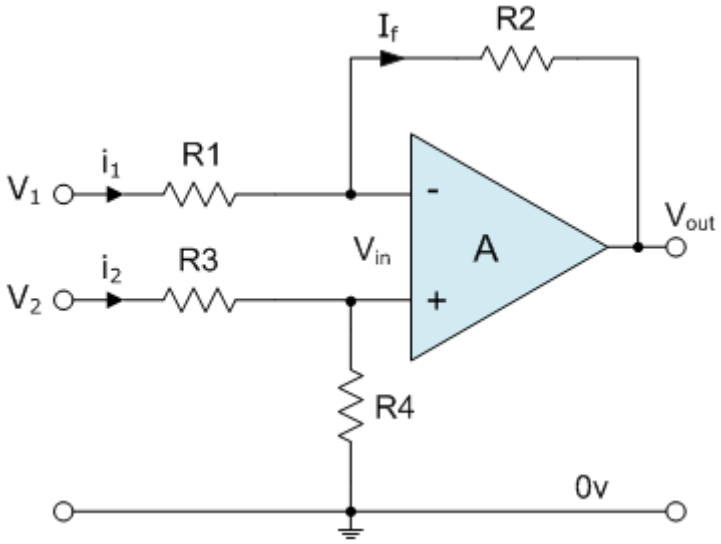


If the input resistances of a summing amplifier are connected to potentiometers the individual input signals can be mixed together by varying amounts. For example, for measuring temperature, you could add a negative offset voltage to make the display read "0" at the freezing point or produce an audio mixer for adding or mixing together individual waveforms (sounds) from different source channels (vocals, instruments, etc) before sending them combined to an audio amplifier.

Differential Amplifier

Up to now we have used only one input to connect to the amplifier, using either the "Inverting" or the "Non-inverting" input terminal to amplify a single input signal with the other input being connected to ground. But we can also connect signals to both of the inputs at the same time producing another common type of operational amplifier circuit called a differential amplifier.
The resultant output voltage will be proportional to the "Difference" between the two input signals, V_1 and V_2. This type of circuit can also be used as a subtractor.

Differential Amplifier Circuit



The transfer function for a differential amplifier circuit is given as:

$$V_o = -\frac{R_2}{R_1}V_1 + (1+\frac{R_2}{R_1})(\frac{R_4}{R_3+R_4})V_2$$

When R1 = R3 and R2 = R4 the transfer function formula can be modified to the following:

$$V_o = \frac{R_2}{R_1}(V_2 - V_1)$$

If all the resistors are all of the same ohmic value the circuit will become a **Unity Gain Differential Amplifier** and the gain of the amplifier will be 1 or Unity.

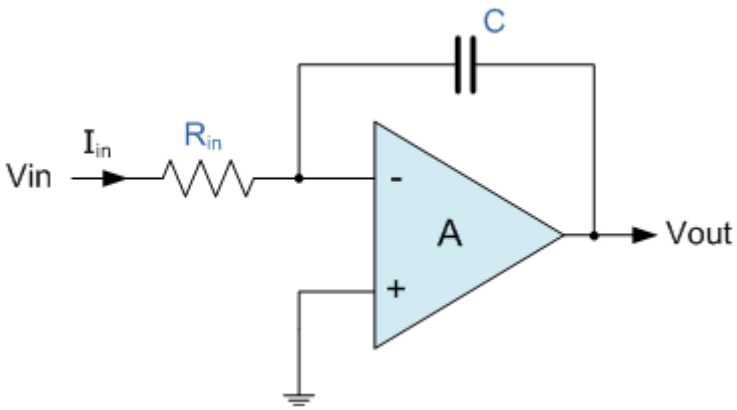
One major limitation of this type of amplifier design is that its input impedances are lower compared to that of other operational amplifier configurations, for example, a non-inverting (single-ended input) amplifier. Each input voltage source has to drive current through an input resistance, which has less overall impedance than that of the op-amps input alone. One way to overcome this problem is to add a *Unity Gain Buffer Amplifier* such as the

voltage follower seen in the previous tutorial to each input resistor. This then gives us a differential amplifier circuit with very high input impedance and is the basis for most "Instrumentation Amplifiers", mainly used to amplify very small differential signals from strain gauges, thermocouples or current sensing resistors in motor control systems.

The Integrator Amplifier

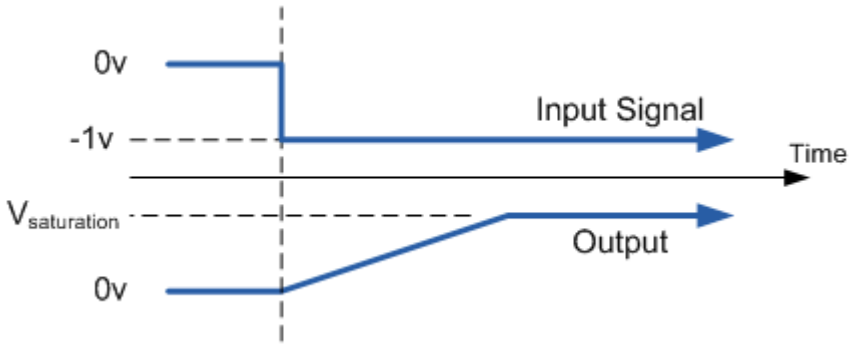
Till now we saw how an operational amplifier can be used as part of a positive or negative feedback amplifier or as an adder or subtractor type circuit using pure resistors in both the input and the feedback loop. But what if we were to change the purely Resistive (R_f) feedback element of an inverting amplifier to that of a reactive element, such as a *Capacitor, C*. We now have a resistor and capacitor combination forming an *RC Network* across the operational amplifier as shown below.

Integrator Amplifier Circuit

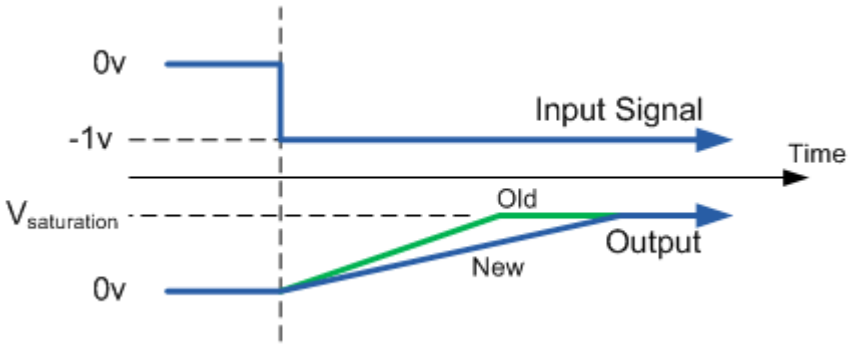


The integrator amplifier performs the mathematical operation of **integration**, that is, we can cause the output to respond to changes in the input voltage over time and the integrator amplifier produces a voltage output which is proportional to that of its input voltage with respect to time. In other words the magnitude of the output signal is determined by the length of time a voltage is present at its input as the current through the feedback loop charges or discharges the capacitor.

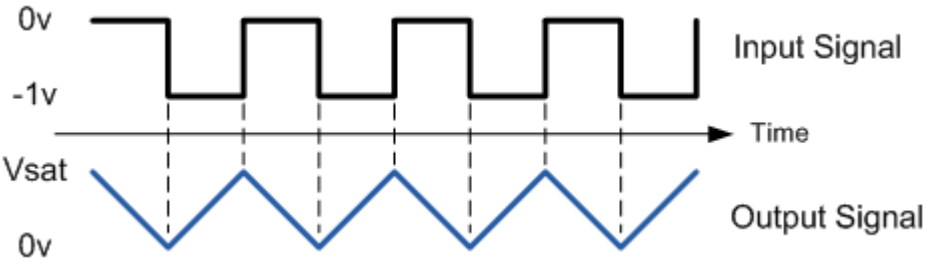
When a voltage, V_{in} is firstly applied to the input of an integrating amplifier, the uncharged capacitor C has very little resistance and acts a bit like a short circuit (voltage follower circuit) giving an overall gain of less than 1, thus resulting in zero output. As the feedback capacitor C begins to charge up, the ratio of Z_f/R_{in} increases producing an output voltage that continues to increase until the capacitor is fully charged. At this point the ratio of feedback capacitor to input resistor (Z_f/R_{in}) is infinite resulting in infinite gain and the output of the amplifier goes into saturation as shown in the diagram. (Saturation is when the output voltage of the amplifier swings heavily to one voltage supply rail or the other with no control in between).



The rate at which the output voltage increases (the rate of change) is determined by the value of the resistor and the capacitor, "RC time constant". By changing this RC time constant value, either by changing the value of the Capacitor, C or the Resistor, R, the time in which it takes the output voltage to reach saturation can also be changed.



If we apply a constantly changing input signal such as a square wave to the input of an **Integrator Amplifier** then the capacitor will charge and discharge in response to changes in the input signal. This results in an output signal with a sawtooth waveform and its frequency is dependent upon the time constant (RC) of the circuit. This type of circuit is also known as a **Ramp Generator** and the transfer function is given below.



Since the node voltage of the integrating op-amp at its inverting input terminal is zero, the current I_{in} flowing through the input resistor is given as:

$$I_{in} = \frac{V_{in}}{R}$$

The current flowing through the feedback capacitor C is given as:

$$I_{in} = C\frac{dV_{out}}{dt}$$

Assuming that the input impedance of the op-amp is infinite (ideal op-amp), no current flows into the op-amp terminal. Therefore, the nodal equation at the inverting input terminal is given as:

$$\frac{V_{in}}{R} = C\frac{dV_{out}}{dt} = 0$$

From which we have an ideal voltage output for the Integrator Amplifier as:

$$V_{out} = -\frac{1}{RC}\int V_{in}dt = -\frac{1}{j\omega RC}V_{in}$$

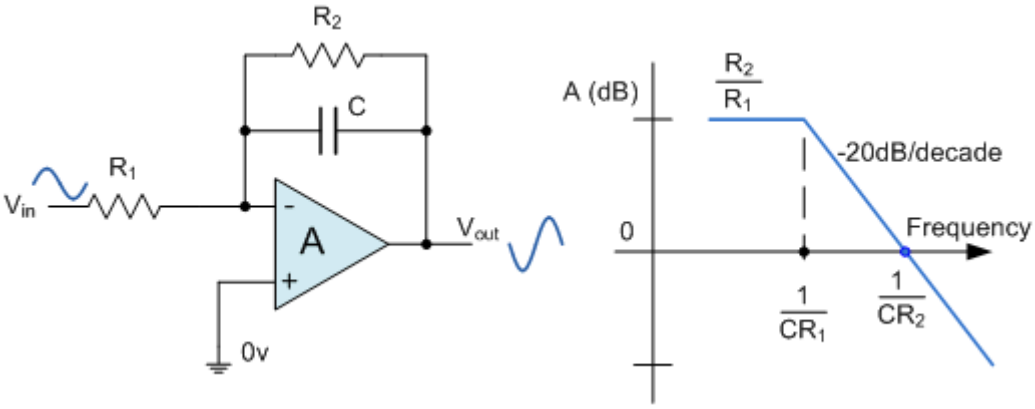
Where $j\omega = 2\pi f$ and the output voltage V_{out} is a constant 1/RC times the integral of the input voltage V_{in} with respect to time. The minus sign (-) indicates a 180^0 phase shift because the input signal is connected directly to the inverting input terminal of the op-amp.

Active Low Pass Filter

If we changed the above square wave input signal to that of a sine wave of varying frequency the **Integrator Amplifier** begins to behave like an active "Low Pass Filter",

passing low frequency signals while attenuating the high frequencies. However, at DC (0Hz) the capacitor acts like an open circuit blocking any feedback voltage resulting in zero negative feedback from the output back to the input of the amplifier. Then the amplifier effectively is connected as a normal open-loop amplifier with very high open-loop gain resulting in the output voltage saturating.

The addition of a large value resistor, R_2 across the capacitor, C gives the circuit the characteristics of an inverting amplifier with finite closed-loop gain of R_2/R_{in} at very low frequencies while acting as an integrator at higher frequencies. This then forms the basis of an *Active Low Pass Filter*.



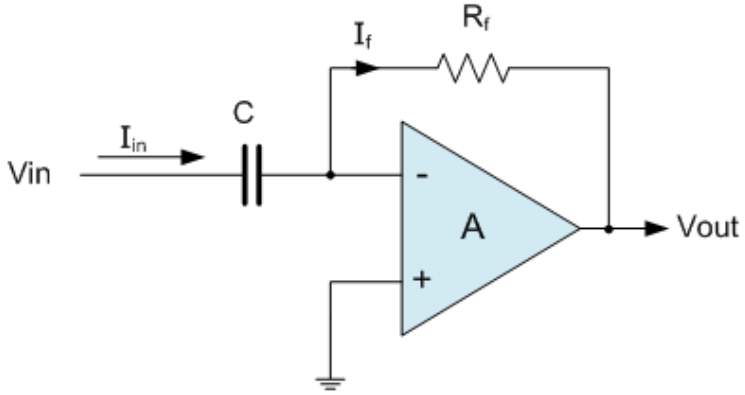
The AC Integrator with DC Gain Control

The Differentiator Amplifier

The basic differentiator amplifier circuit is the exact opposite to that of the *Integrator* operational amplifier circuit. Here, the position of the capacitor and resistor have been reversed and now the Capacitor, C is connected to the input terminal of the inverting amplifier while the Resistor, R_f forms the negative feedback element across the operational amplifier. This circuit performs the mathematical operation of **Differentiation**, i.e. it produces a voltage output which is proportional to **rate-of-change** of the input voltage and the current flowing through the capacitor. In other words the faster or larger the change to the input voltage signal, the greater the input current, the greater will be the output voltage change in response becoming more of a "spike" in shape.

As with the integrator circuit, we have a resistor and capacitor forming an RC Network across the operational amplifier and the reactance (Xc) of the capacitor plays a major role in the performance of a differentiator amplifier.

Differentiator Amplifier Circuit



Since the node voltage of the operational amplifier at its inverting input terminal is zero, the current, i flowing through the capacitor will be given as:

$$i_{IN} = I_F \quad \text{and} \quad I_F = -\frac{V_o}{R_F}$$

The Charge on the Capacitor = Capacitance x Voltage across the Capacitor

$$Q = C \times V_{IN}$$

The rate of change of this charge is

$$\frac{dQ}{dt} = C\frac{dV_{IN}}{dt}$$

but dQ/dt is the capacitor current i

$$i_{IN} = C\frac{dV_{IN}}{dt} = I_F$$

From which we have an ideal voltage output for the Differentiator Amplifier is given as:

$$V_O = -R_F C\frac{dV_{IN}}{dt}$$

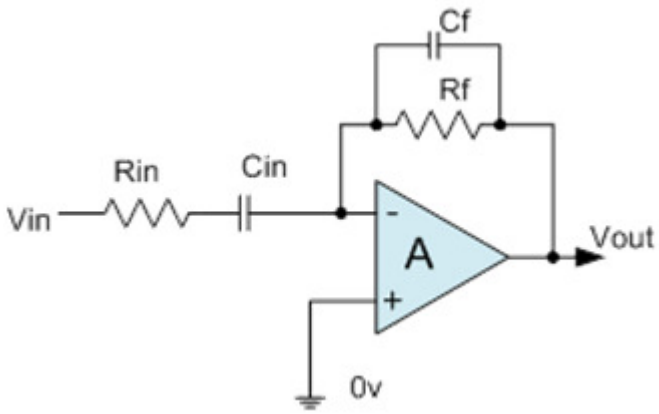
Therefore, the output voltage V_{out} is a constant $-R_f.C$ times the derivative of the input voltage V_{in} with respect to time. The minus sign indicates a 180^0 phase shift because the input signal is connected to the inverting input terminal of the operational amplifier.

Active High Pass Filter

The capacitor blocks any DC content only allowing AC type signals whose frequency is dependent on the rate of change of the input signal, to pass through. At low frequencies the reactance of the capacitor is "High" resulting in a low gain (Rf/Xc) and low output voltage from the op-amp. At higher frequencies the reactance of the capacitor is much lower resulting in a higher gain and higher output voltage from the differentiator amplifier. Thus with sinusoidal wave at the input this circuit will act as an active high pass filter circuit.

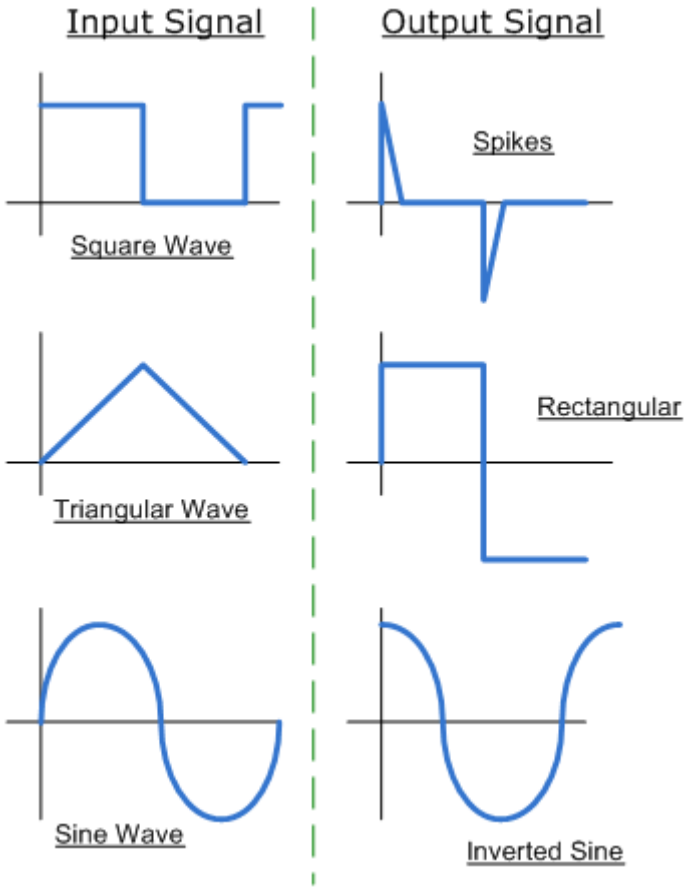
The basic single resistor and single capacitor differentiator circuit is not widely used to reform the mathematical function of differentiation because of the two inherent faults mentioned above: Instability and Noise.

At high frequencies a differentiator circuit becomes unstable and will start to oscillate. To avoid this, the high frequency gain of the circuit needs to be reduced by adding an additional small value capacitor, C_f, across the feedback resistor R_f. Also, the capacitive input makes it very susceptible to random noise signals and any noise or harmonics present in the circuit will be amplified more than the input signal itself. This is because the output is proportional to the slope of the input voltage. So some means of limiting the bandwidth in order to achieve closed-loop stability is required. In order to reduce the overall closed-loop gain of the circuit at high frequencies, an extra Resistor, R_{in} is added to the input as shown below. Thus, the new circuit acts like a Differentiator amplifier at low frequencies and an amplifier with resistive feedback at high frequencies giving much better noise rejection.



Differentiator Waveforms

If we apply a constantly changing signal such as a Square-wave, Triangular or Sine-wave type signal to the input of a differentiator amplifier circuit the resultant output signal will be changed and whose final shape is dependent upon the RC time constant of the Resistor/Capacitor combination.



Study of Basic OPAMP configurations

Objectives:

(I) Study of the inverting amplifier configuration and to find its gain
(II) Study of the non-inverting amplifier configuration and to find its gain

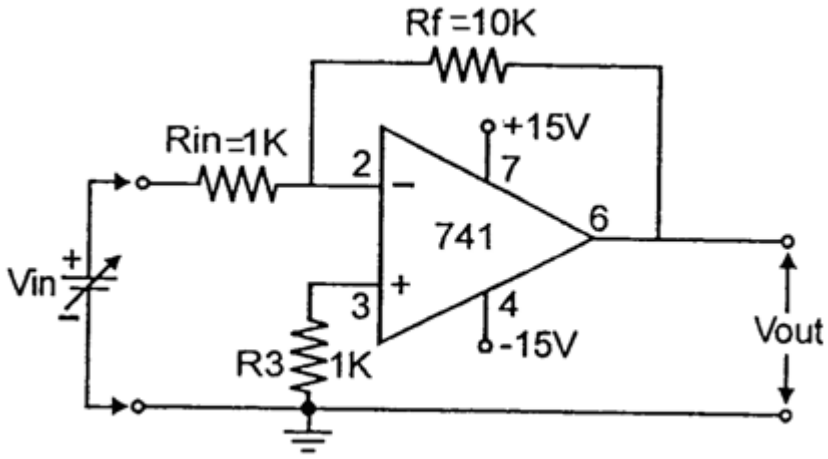
Apparatus:

1. OPAMP IC 741
2. Resistors: 1K (2 nos), 10K, 50K
3. D.C. power supply
4. Digital multimeter
5. Bread board
6. Connecting wires

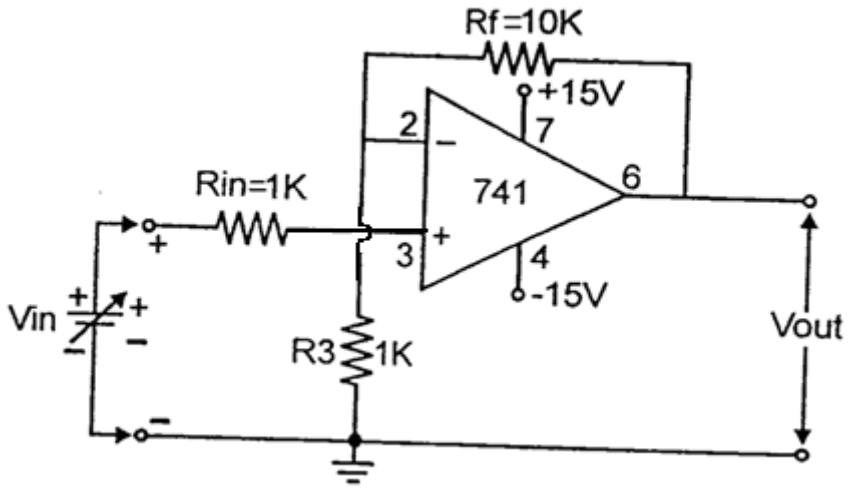
Theory:

Please refer the supplementary note.

Circuit Diagram:



Inverting amplifier



Non-inverting amplifier

Procedure:

(I) Inverting amplifier

1. Configure the circuit as shown in the circuit diagram. Connect the pins 7 and 4 of the IC to the ±15V output terminals of the D.C. power supply. Connect the 0V terminal to ground. Choose R_{in} = 1K and R_f = 10K. Measure the resistance values with multimeter and calculate gain, $-(R_f/R_{in})$. Connect a resistor R_3 (= $R_{in} \| R_f \approx R_{in}$) as shown in the circuit diagram so as to minimize offset due to input bias current.
2. Connect one of the output terminals of the D.C. power supply (0-30V) at the **inverting input (pin no. 2).**
3. Switch on the power supply and apply different voltages in the range 0- 1.5V (why?) in steps of 0.2 V at the inverting terminal. Measure this input using a digital multimeter.
4. Measure the corresponding output voltages with the multimeter and calculate gain V_o/V_{in}. Note the sign of the output voltage.
5. Now, replace R_f by 50K. Measure the resistance value with multimeter and calculate gain, $-(R_f/R_{in})$.
6. Apply different voltages in the range 0- 0.5V in steps of 0.1 V at the inverting terminal. Measure this input using a digital multimeter.
7. Measure the corresponding output voltages with the multimeter and calculate gain V_o/V_{in}.
8. Plot graphs for $V_{in} \sim V_o$ for both the values of R_F.
9. You may also use a function generator to give a sinusoidal input and notice the output waveform using an oscilloscope.

(II) Non-inverting amplifier

1. Configure the circuit as shown in the circuit diagram with R_{in} = 1K and R_f = 10K. using the measured value of resistance calculate gain, $1+ (R_f/R_{in})$.
2. Connect one of the output terminals of the D.C. power supply (0-30V) at the **non-inverting input (pin no. 3).**

3. Repeat steps 3 onwards of procedure (I) with inputs applied at non-inverting terminal.

Observations

Table (I):

Obs. No.	Input (V)	$-\frac{R_f}{R_{in}}$ = ------			$-\frac{R_f}{R_{in}}$ = ------		
		Output (V)	Gain V_o/V_{in}	Average	Output (V)	Gain V_o/V_{in}	Average
	0.2						
	0.4						
	…						

Table (For II):

Obs. No.	Input (V)	$1+\frac{R_f}{R_{in}}$ = ------			$1+\frac{R_f}{R_{in}}$ = ------		
		Output (V)	Gain V_o/V_{in}	Average	Output (V)	Gain V_o/V_{in}	Average
1	0.1						
2	0.2						
..	…						

Graph:

Calculate gain from graph in each case.

Discussions:

Conclusions/precautions:

Applications of OPAMP as Comparator and Schmitt Trigger

Objectives:

(i) Study of OPAMP as comparator

(ii) Study of OPAMP as Schmitt trigger

(i) Comparator

Theory

When the feedback signal (voltage) is applied to the inverting (-) input of the op-amp then the feedback is negative. Negative feedback tends to reduce the difference between the voltages at the inverting and non-inverting terminals and make linear circuits. Without negative feedback the op-amp output is highly sensitive to the input, which can be used to design *switching* or *nonlinear* circuits. The voltage *comparator* is a device which uses no feedback; then saturation is the desired result. In this circuit we want a simple yes-no answer to be signified by either positive saturation or negative saturation of the output.

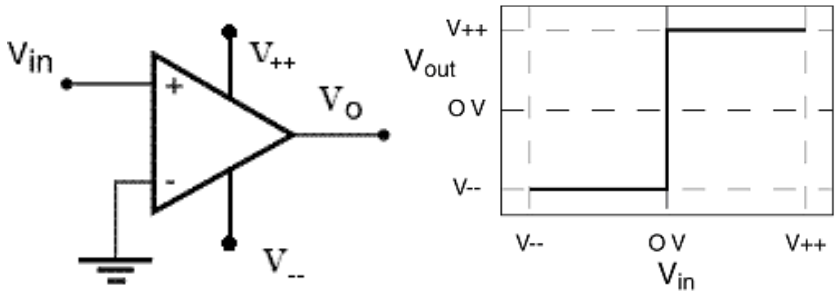
In the circuit diagrams shown below, for Fig.(i), if $V_{in} > 0$, $V_o \approx V_{++}$ and if $V_{in} < 0$, $V_o \approx V_{--}$.The output is no longer linearly related to the input– it's more like a digital signal, high or low depending on how V_{in} compares to ground (0 V). Needless to mention that, if V_{in} is applied at the inverting terminal with respect to a grounded non-inverting terminal, the output will switch to low when $V_{in} > 0$. Figure (ii) shows a small modification, allowing the circuit to switch its output when V_{in} crosses a certain preset voltage level, often called the **threshold voltage**, V_{th}.

Typical applications of this circuit are crossover detectors, analog to digital converters or counting applications where one wants to count pulses that exceed a certain voltage level.

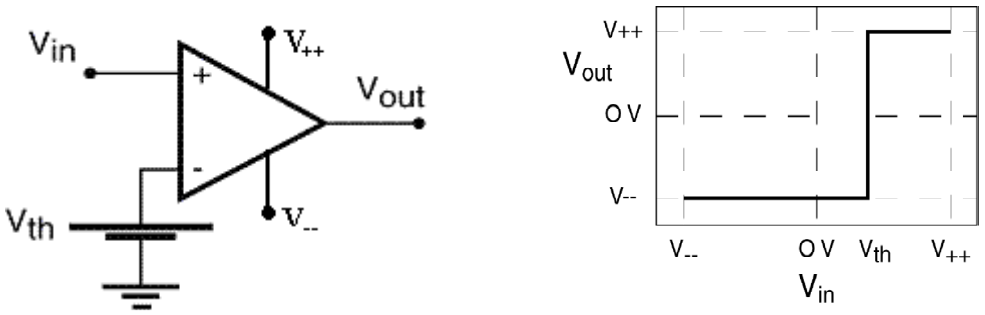
Components/Equipments:

(i) OPAMP (IC-741) chip, (ii) A D.C. power supply, (iii) A digital multimeter (DMM), (iv) A digital storage oscilloscope (DSO), (v) Connecting wires, (vi) Breadboard

Circuit Diagram:



(i)



(ii)

Procedure:

1. Construct the comparator circuit on the breadboard as shown in the circuit diagram. Take care to give proper connections at the desired pins of the IC.
2. Use terminal **C** of the d.c. power supply (denoted by V_+ and V_- knobs) to provide power supply to IC. Connect the 0Vterminal to ground.
3. Connect terminal A of the d.c. power supply (0-30V) at the input. Use terminal B (5V) to provide threshold voltage V_{th} for circuit shown in Fig. (ii).
4. Vary the input from a negative value to a positive value through 0.
5. Using the DMM, measure and tabulate V_{in} and $V_{out.}$ You can also look at the output using a DSO by coupling the output to it in DC mode.
6. Make a plot of V_{out} vs V_{in} for each circuit. Estimate V_{th} from graph for Fig. (ii) and compare with the V_{th} value actually applied. You can repeat the same procedure for different values of threshold.
7. Repeat the entire procedure described above with input at the inverting terminal and the non-inverting terminal being grounded w/o and with the threshold voltage connected to it.

Observations:

For Fig. (i)

Obs.No	**V_i (V)**	**V_o (V)**
1		
..		
..		

For Fig. (ii)

V_{th} = ----- V

Obs.No	**V_i (V)**	**V_o (V)**
1		
..		
..		

Discussions:

Precautions:

(ii) Schmitt trigger

The Schmitt trigger is a variation of the simple comparator which has hysteresis, that is, it has a toggle action. It uses a positive feedback. When the output is high, positive feedback makes the switching level higher than it is when the output is low. A little positive feedback makes a comparator with better noise immunity.

Now, to understand what causes the hysteresis let's analyze the circuit diagram given below, using the same rules as in the previous section for the comparator. The key in understanding this circuit will again be in calculating the voltages that cause its output to switch. If V_+ and V_- are the actual voltages at the non-inverting and inverting terminals of the OPAMP, then the output will be the following, considering that $V_-=0$:

$$\begin{aligned} &\text{if} \quad V_+ > 0, \qquad V_{out} \approx V_{++} \\ \&\ &\text{if} \quad V_+ < 0, \qquad V_{out} \approx V_{--}. \end{aligned}$$

Since V_{out} changes its state whenever V_+ crosses 0V, we need to find what value of V_{in} results in $V_+ = 0$. The two values of V_{in} for which the output switches are called the trip points. V_+ acts as a voltage divider formed by R_1 and R_2 between V_{in} and $V_{out.}$ Thus the trip points of a noninverting Schmitt trigger are:

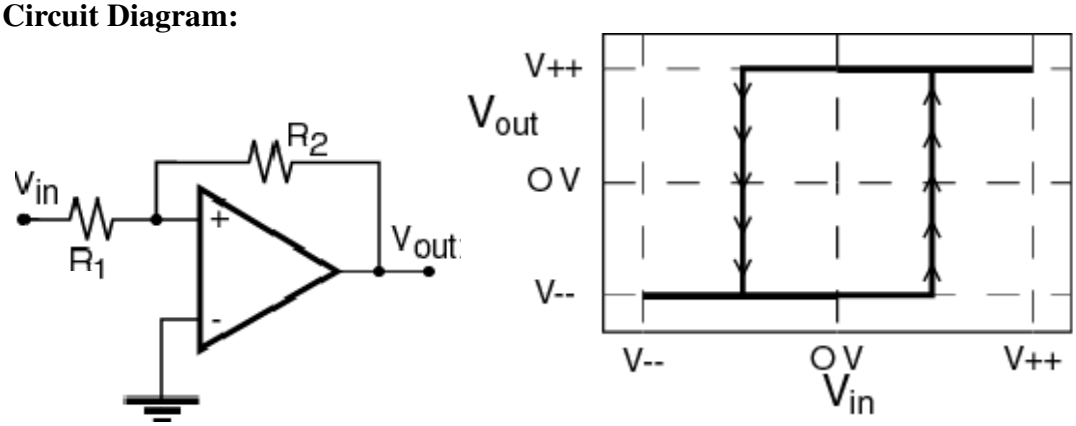
$$\mathbf{V_{in} = -V_{out}\,(R_1/R_2)\ \ (Lower\ trip\ point,\ LTP)}$$
$$\mathbf{= +\,V_{out}\,(R_1/R_2)\ \ (Upper\ trip\ point,\ UTP)}$$

Choosing suitable ratios of R_1 to R_2, enough hysteresis can be created in order to prevent unwanted noise triggers.

Components/Equipments:

(i) OPAMP (IC-741) chip
(ii) A D.C. power supply
(iii) A digital multimeter
(iv) Connecting wires
(v) Breadboard
(vi) Digital storage oscilloscope (DSO)

Circuit Diagram:



Procedure:

1. Construct the schmitt trigger circuit on the breadboard as shown in the circuit diagram.
2. Connect the d.c. power supply at the input. Vary the input from a negative value to a positive value through 0.
3. Using the DMM, measure and tabulate V_{in} and $V_{out.}$
4. Make a plot of V_{out} vs V_{in}. Estimate the trip points from the graph and compare with the computed value, i.e. $\mathbf{V_{in} = \pm V_{out}\, R_1/R_2}$
5. You can also look at the output using a DSO by coupling the output to it in DC mode.

Observations:

Obs.No	**V_i (V)**	**V_o (V)**
1		
..		
..		

Discussions:

Analyze the graph you obtained. Discuss the switching action.

Precautions:

Study of various mathematical operations using OPAMP

Objectives:

(I) To study OPAMP as summing amplifier
(II) To study OPAMP as difference amplifier
(III) Studying multiplication using OPAMP
(IV) Studying division using OPAMP
(V) To find average using OPAMP
(VI) To study OPAMP as a differentiator
(VII) To study OPAMP as an integrator

Apparatus:

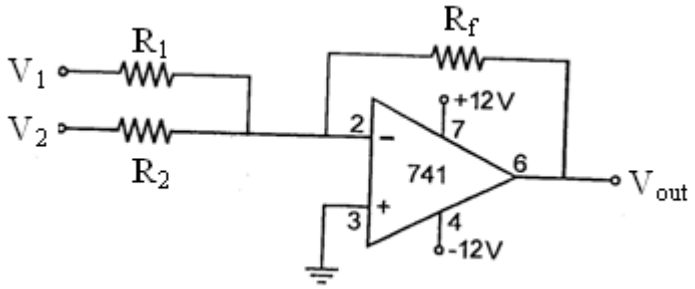
1. OPAMP IC 741
2. D.C. power supply
3. Resistors
4. Digital multimeter
5. Connecting wires
6. Breadboard
7. Function generator
8. Digital storage oscilloscope

Theory:

Please refer to the supplementary note.

(I) To study OPAMP as summing amplifier

Circuit Diagram:



Procedure:

1. Assemble the circuit as shown in circuit diagram choosing R_1, R_2, R_f = 10K each. Use 0- ±15V terminal output to provide supply to the IC.
2. Using 0 – 30V and 5V terminals of the power supply, apply two inputs at the inverting terminal. Measure each input with multimeter.
3. Measure the output with multimeter for at least five input combinations.
4. Compare the output with the sum of the two inputs.

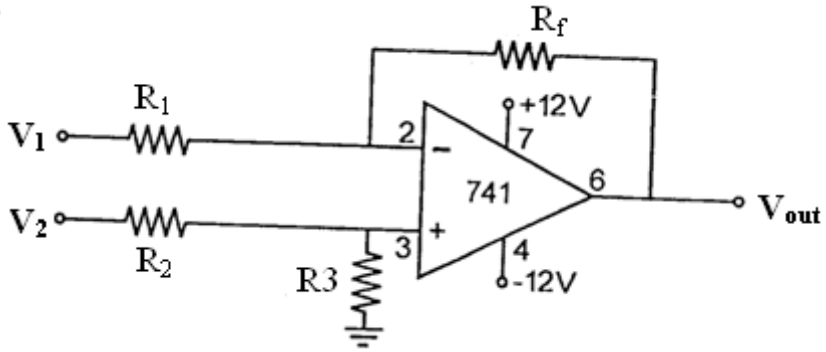
Observations:

Obs.No	V_1 (V)	V_2 (V)	V_{out} (V)	$V_1 + V_2$ (V)
1				
..				
5				

Disscusions:

(II) To study OPAMP as difference amplifier

Circuit Diagram:



Procedure:

1. Assemble the circuit as shown in circuit diagram choosing R_1, R_2, R_3, R_f = 10K each. Use 0- ±15V terminal output to provide supply to the IC.
2. Using 0 – 30V and 5V terminals of the power supply, apply two inputs, one at the inverting and the other at the non-inverting terminal. Measure each input with multimeter.
3. Measure output with multimeter for at least five input combinations.
4. Compare the output with the difference of the two inputs.

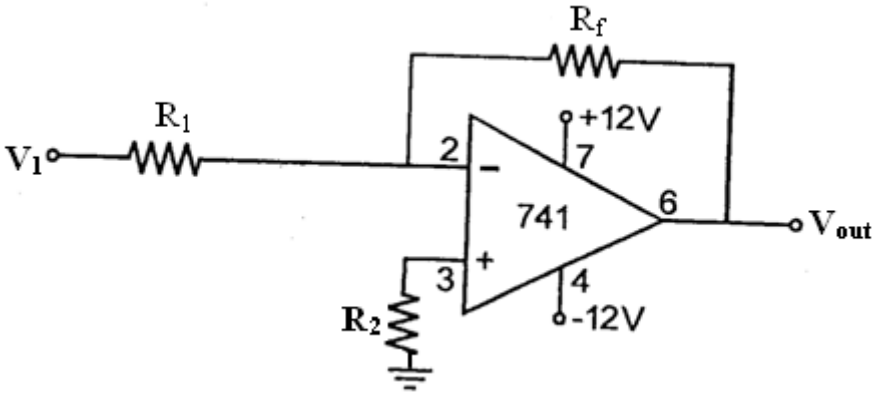
Observations:

Obs.No	V_1 (V)	V_2 (V)	V_{out} (V)	$V_2 - V_1$ (V)
1				
..				
5				

Disscusions:

(III) Studying multiplication using OPAMP

Circuit Diagram:



Procedure:

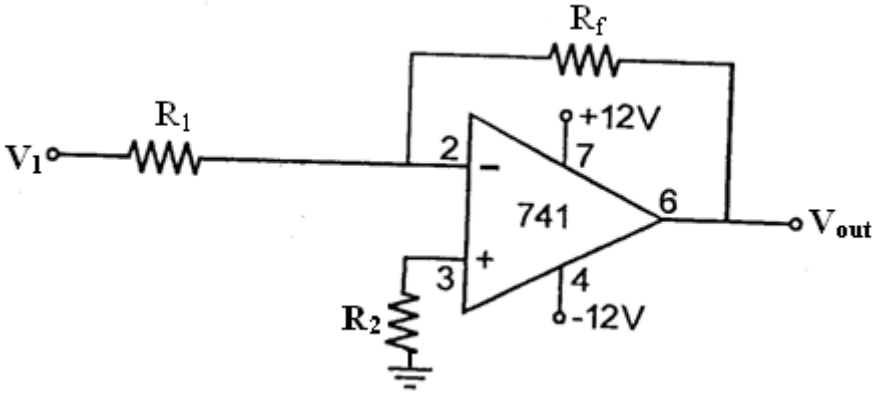
1. Assemble the circuit as shown in circuit diagram choosing R_1, R_2 = 1K each and R_f = 10K. Use 0- ±15V terminal output to provide supply to the IC.
2. Using 0 – 30V terminal of the power supply, apply different input values at the inverting terminal. Measure each input with multimeter.
3. Measure output with multimeter for at least five different input values.
4. Compare the measured output with the calculated one.

Observations:

Obs.No	V_1 (V)	V_{out} (Measured) (V)	V_{out} (Calculated) (V)
1			
..			
..			

Discussions:

(IV) Studying division using OPAMP



Procedure:

1. Assemble the circuit as shown in circuit diagram choosing R_1 = 10K each and R_2, R_f = 1K. Use 0 - ±15V terminal output to provide supply to the IC.
2. Using 0 – 30V supply, apply different input values at the inverting terminal. Measure each input with multimeter.
5. Measure output with multimeter for at least five different input values.
3. Compare the measured output with the calculated one.

Observations:

Obs.No	V_1 (V)	V_{out} (Measured) (V)	V_{out} (Calculated) (V)
1			
..			
..			

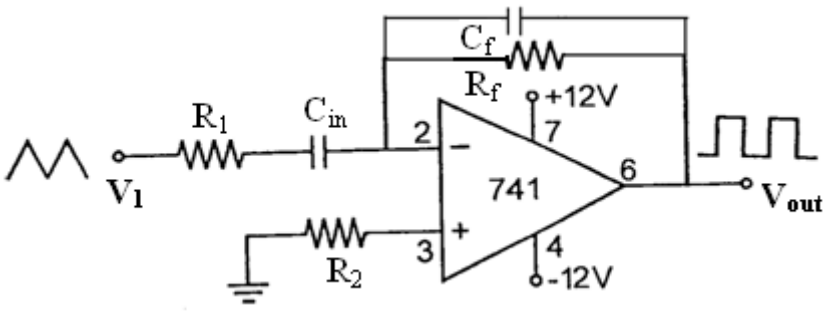
Discussions:

(V) To find average using OPAMP

With the knowledge of a summing and a division amplifier, try to assemble a circuit which can find the average of two given inputs.

(VI) To study OPAMP as a differentiator

Circuit Diagram:



Procedure:

1. Assemble the circuit as shown in circuit diagram choosing R_1, R_2 = 1K each, R_f = 10K, C_{in}= 0.1 μF and C_f = 0.001μF. Use 0- ±15V terminal output to provide supply to the IC.
2. Feed a triangular input signal of required amplitude from the function generator, which is set at 1K frequency.
3. Feed both the input and output signals to an oscilloscope and save. The output should be approximately a square wave.
4. Check the output waveform with sine and square waves as inputs and save.

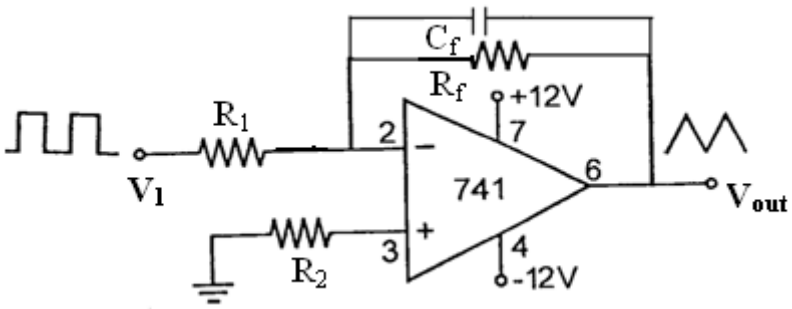
Observations: (Paste the various input and corresponding output waveforms here)

Observation	Waveform		
Input	triangular	sine	square
Output			

Discussions

(VII) To study OPAMP as an integrator

Circuit Diagram



Procedure:

1. Assemble the circuit as shown in circuit diagram choosing R_1, R_2 = 10K each, R_f = 100K, and C_f = 0.1μF. Use 0- ±15V terminal output to provide supply to the IC.
2. Feed a square wave input of required amplitude from the function generator, which is set at 1K frequency.
3. Feed both the input and output signals to an oscilloscope. The output should be a triangular wave.

Observations: (Paste the various input and corresponding output waveforms here)

Observation	Waveform
Input	
Output	

Discussions:

Conclusions:

Precautions:

Phase Shift Oscillator using Opamps

Objectives:

To construct and determine the resonant frequency of
(i) A phase shift oscillator

Overview:

The main principle of oscillator is positive feedback. Block diagram of oscillator is shown in Figure 1.

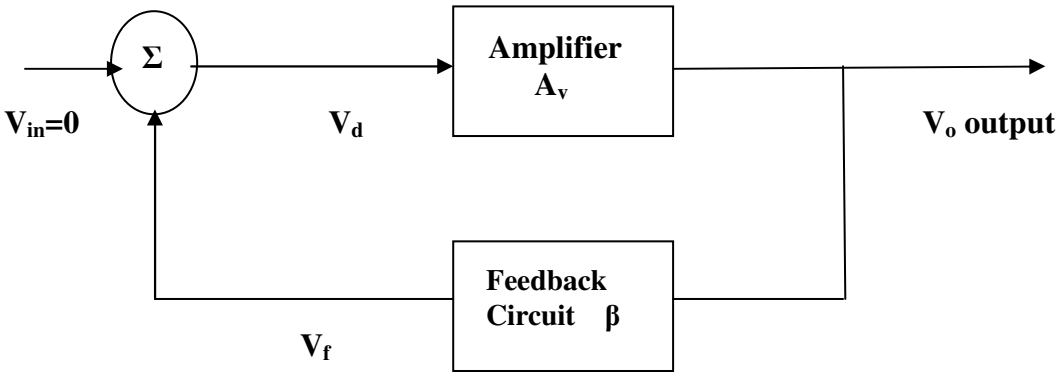


Figure.1

In the block diagram (Fig.1), $V_d = V_f + V_{in}$

$$V_o = A_v V_d \text{ and } V_f = \beta V_o$$

Using these relationships, following equation can be obtained:

$$\frac{V_o}{V_{in}} = \frac{A_V}{1 - A_V \beta}$$

.

When $A_V\beta = 1$, $A_{fb} = \infty = \frac{V_o}{V_{in}}$, This will happen only when $V_{in} = 0$. That is we get a signal at output without any input. The condition $A_V\beta = 1$ is known as Barkhausen condition. This condition expressed in polar form as follows. $A_V\beta = 1 \angle 0° \text{ or } 360°$

Barkhausen condition gives two requirements for oscillation.

1) The magnitude of the loop gain must be equal to 1.
2) The total phase shift of the loop gain must be equal to $0° \text{ or } 360°$.

Phase shift Oscillator: Figure.2 gives the circuit diagram for a phase shift oscillator, which consists of an op-amp as the amplifying stage and three RC cascaded networks as the feedback circuit. The opamp used in this oscillator is in the inverting mode, output is 180° is phase shifted. To feedback the output to input, additional 180° degree phase shift is achieved by RC network.

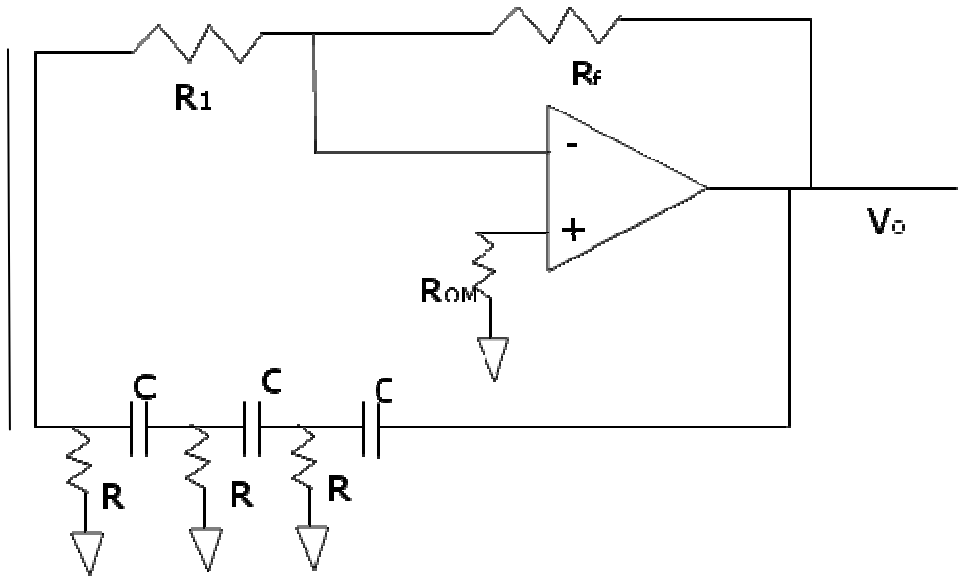


Figure.2

The frequency of oscillation is given by, $f = \frac{1}{2\pi\sqrt{6}RC}$ **and at this frequency gain must be *at least* 29. That is** $\left|\frac{R_F}{R_1}\right| = 29$**.**

Feedback circuit with RC network gives 180 degree phase shift but decreases the output voltage by a factor of 29. That is β=1/29. For the oscillations $A_V\beta = 1$*. Therefore, gain should be at least 29.*

Procedure: Choose R_f = 100 KΩ, R = 2 KΩ, C = 0.1 uF, R=1 KΩ and construct a phase shift oscillator. Determine the oscillating frequency using oscilloscope and compare with calculated oscillation frequency.

Experimentally determine the minimum gain required to sustain oscillations by varying the gain in the circuit. Obtain Lissajous figure (circle) with X-Y mode of the oscilloscope and estimate oscillating frequency. Try to make the circuit for some other oscillating frequency by choosing components appropriately.

References: 1) OPAMPS and linear integrated circuits –Ramakant Gaykwad

2) http://textofvideo.nptel.iitm.ac.in/122106025/lec35.pdf

PASSIVE (RC) FILTER CIRCUIT

OBJECTIVES:

(I) **S**tudy the transfer function and phase shift of a low pass RC filter network.
(II) Study the transfer function and phase shift of a high pass RC filter network.

OVERVIEW:

Filter circuits are used in a wide variety of applications. In the field of telecommunication, band-pass filters are used in the audio frequency range (20 Hz to 20 kHz) for modems and speech processing. High-frequency band-pass filters (several hundred MHz) are used for channel selection in telephone central offices. Data acquisition systems usually require anti-aliasing low-pass filters as well as low-pass noise filters in their preceding signal conditioning stages. System power supplies often use band-rejection filters to suppress the 50-Hz line frequency and high frequency transients.

Frequency-selective or filter circuits pass only those input signals to the output that are in a desired range of frequencies (called pass band). The amplitude of signals outside this range of frequencies (called stop band) is reduced (ideally reduced to zero). The frequency between pass and stop bands is called the cut-off frequency (ω_c). Typically in these circuits, the input and output currents are kept to a small value and as such, the current transfer function is not an important parameter. The main parameter is the voltage transfer function in the frequency domain, Hv(jω) = Vo/Vi. Subscript v of Hv is frequently dropped. As H(jω) is complex number, it has both a magnitude and a phase, filters in general introduce a phase difference between input and output signals.

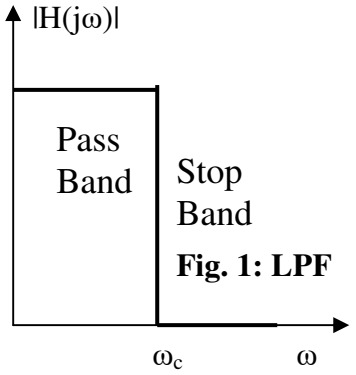


Fig. 1: LPF

LOW AND HIGH-PASS FILTERS

A low pass filter or LPF attenuates or rejects all high frequency signals and passes only low frequency signals below its characteristic frequency called as cut-off frequency, ω_c. An ideal low-pass fillter's transfer function is shown in Fig. 1. A high pass filter or HPF, is the exact opposite of the LPF circuit. It attenuates or rejects all low frequency signals and passes only high frequency signals above ω_c.

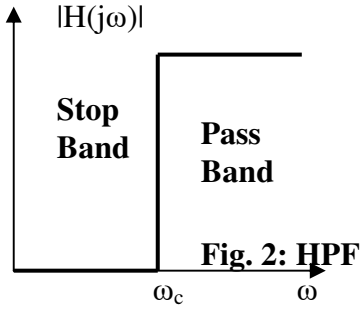


Fig. 2: HPF

In practical filters, pass and stop bands are not clearly defined, |H(jω)| varies continuously from its maximum towards zero. The cut-off frequency is, therefore, defined as the frequency at which |H(jω)| is reduced to $1/\sqrt{2}$ or 0.7 of its maximum value. This corresponds to signal power being reduced by 1/2 as $P \alpha V^2$.

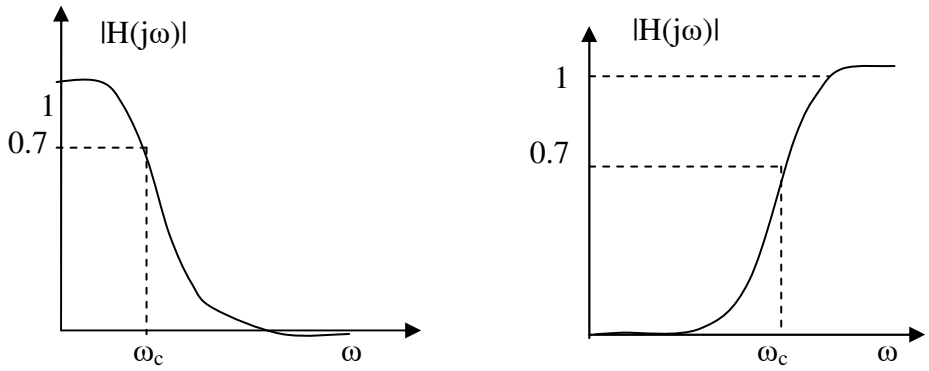


Fig.3: Transfer functions of practical low and high pass filter

RC Filter:

The simplest passive filter circuit can be made by connecting together a single resistor and a single capacitor in series across an input signal, (Vin) with the output signal, (Vout) taken from the junction of these two components. Depending on which way around we connect the resistor and the capacitor with regards to the output signal determines the type of filter construction resulting in either a Low Pass or a High Pass Filter. As there are two passive components within this type of filter design the output signal has amplitude smaller than its corresponding input signal, therefore passive RC filters attenuate the signal and have a gain of less than one, (unity).

Low-pass RC Filter

A series RC circuit as shown also acts as a low-pass filter. For no load resistance (output is open circuit, R → ∞):

$$V_0 = \frac{1/(j\omega C)}{R + (1/j\omega C)} V_i = \frac{1}{1 + j(\omega RC)} V_i$$

$$H(j\omega) = \frac{V_0}{V_i} = \frac{1}{1 + j\omega RC}$$

To find the cut-off frequency (ω_c), we note

$$|H(j\omega)| = \frac{1}{\sqrt{1 + (\omega RC)^2}}$$

When ω→0, |H(jω)| is maximum and →1.
For $\omega = \omega_c$, $|H(j\omega_c)| = 1/\sqrt{2}$. Thus

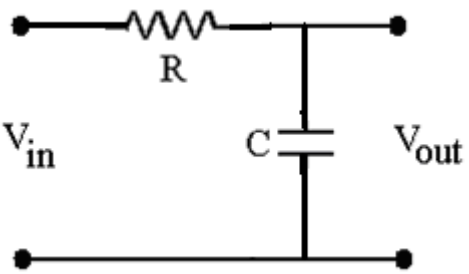


Fig.4: Low pass RC filter circuit

$$|H(j\omega_c)| = \frac{1}{\sqrt{1 + (\omega_c RC)^2}} = \frac{1}{\sqrt{2}}$$

$$\Rightarrow \quad \omega_c = \frac{1}{RC}, \quad H(j\omega) = \frac{1}{1 + \frac{j\omega}{\omega_c}}, |H(j\omega)| = \frac{1}{\sqrt{1 + \left(\frac{\omega}{\omega_c}\right)^2}} \quad \textit{and} \quad \textit{phase}, \phi = -\tan^{-1}\left(\frac{\omega}{\omega_c}\right)$$

Input Impedance:

$$Z_i = R + \frac{1}{j\omega C} \quad and \quad |Z_i| = \sqrt{R^2 + \frac{1}{\omega^2 C^2}}$$

The value of the input impedance depends on the frequency ω. For good voltage coupling, we need to ensure that the input impedance of this filter is much larger than the output impedance of the previous stage. Thus, the minimum value of Z_i is an important number. Z_i is minimum when the impedance of the capacitor is zero (ω→∞), i.e. $Z_i|_{min} = R$.

Output Impedance:

The output impedance can be found by shorting the source and finding the equivalent impedance between output terminals:

$$Z_0 = R \| \frac{1}{j\omega C}$$

where the source resistance is ignored. Again, the value of the output impedance also depends on the frequency ω. For good voltage coupling, we need to ensure that the output impedance of this filter is much smaller than the input impedance of the next stage, the maximum value of Z_0 is an important number. Z_0 is maximum when the impedance of the capacitor is ∞ (ω→0), i.e. $Z_0|_{max} = R$.

Bode Plots and Decibel

The ratio of output to input power in a two-port network is usually expressed in Bell:

$$\textbf{Number of Bels} = \log_{10}\left(\frac{P_0}{P_i}\right) = 2\log_{10}\left(\frac{V_0}{V_i}\right)$$

Bel is a large unit and decibel (dB) is usually used:

$$\textbf{Number of decibels} = 10\log_{10}\left(\frac{P_0}{P_i}\right) = 20\log_{10}\left(\frac{V_0}{V_i}\right)$$

There are several reasons why decibel notation is used:

1) Historically, the analog systems were developed first for audio equipment. Human ear 'hears" the sound in a logarithmic fashion. A sound which appears to be twice as loud actually has 10 times power, etc. Decibel translates the output signal to what ear hears.

2) If several two-port network are placed in a cascade (output of one is attached to the input of the next), it is easy to show that the overall transfer function, H, is equal to the product of all transfer functions:

$$|H(j\omega)| = |H_1(j\omega)| \times |H_2(j\omega)| \times ...$$
$$20\log_{10}|H(j\omega)| = 20\log_{10}|H_1(j\omega)| + 20\log_{10}|H_2(j\omega)| + ...$$
$$|H(j\omega)|_{dB} = |H_1(j\omega)|_{dB} + |H_2(j\omega)|_{dB} + ...$$

making it easier to understand the overall response of the system.

3) Plot of $|H(j\omega)|_{dB}$ versus frequency has special properties that again makes analysis simpler as is seen below.

For example, using dB definition, we see that, there is 3 dB difference between maximum gain and gain at the cut-off frequency:

$$20\log_{10}|H(j\omega_c)| - 20\log_{10}|H(j\omega)|_{max} = 20\log_{10}\frac{|H(j\omega_c)|}{|H(j\omega)|_{max}} = 20\log\left(\frac{1}{\sqrt{2}}\right) = -3dB$$

Bode plots are plots of magnitude in dB and phase of H(jω) versus frequency in a semi-log format. Bode plots of first-order low-pass filters (include one capacitor) display the following typical characteristics:

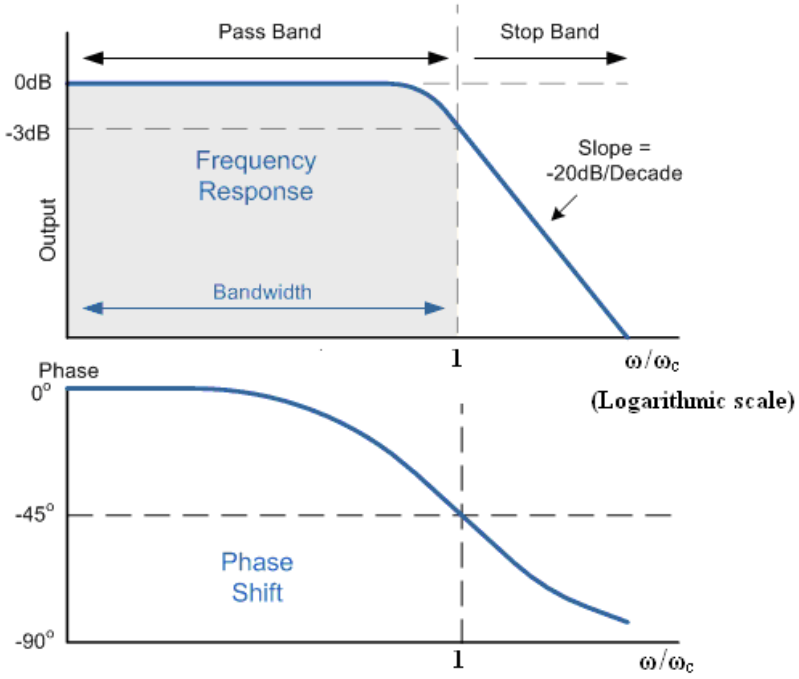


Fig.5: Bode Plots for low-pass RC filter

At high frequencies, $\omega/\omega_c >> 1$, $|H(j\omega)| \approx 1/(\omega/\omega_c)$ and $|H(j\omega)|_{dB} = 20 \log(\omega_c) - 20 \log \omega$, which is a straight line with a slope of -20 dB/decade in the Bode plot. It means that if ω is increased by a factor of 10 (a decade), $|H(j\omega)|_{dB}$ changes by -20 dB.

At low frequencies $\omega/\omega_c << 1$, $|H(j\omega)| \approx 1$, which is also a straight line in the Bode plot. The intersection of these two "asymptotic" values is at $1 = 1/(\omega/\omega_c)$ or $\omega = \omega_c$. Because of this, the cut-off frequency is also called the "corner" frequency.

The behavior of the phase of H(jω) can be found by examining $\phi = -\tan^{-1}(\frac{\omega}{\omega_c})$. At high frequencies, $\omega/\omega_c >> 1$, $\phi \approx -90^0$ and at low frequencies, $\omega/\omega_c << 1$, $\phi \approx 0$. At cut-off frequency, $\phi \approx -45^0$.

High-pass RC Filter

A series RC circuit as shown acts as a high-pass filter. For no load resistance (output open circuit), we have:

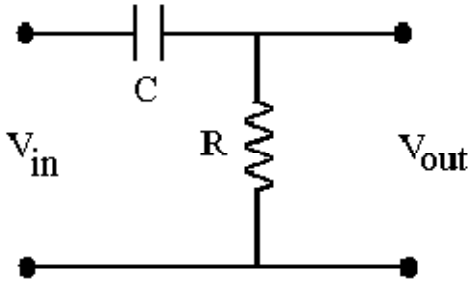


Fig. 6: High pass RC filter circuit

$$V_0 = \frac{R}{R + (1/j\omega C)} V_i = \frac{1}{1 - j(1/\omega RC)} V_i$$

$$H(j\omega) = \frac{V_0}{V_i} = \frac{1}{1 - j(1/\omega RC)}$$

The gain of this filter, |H(jω)| is maximum is maximum when denominator is smallest, i.e., ω⟶ ∞, leading to $|H(j\omega)|_{max} = 1$. Then, the cut-off frequency can be found as

$$|H(j\omega_c)| = \frac{1}{\sqrt{1 + (1/\omega_c RC)^2}} = \frac{1}{\sqrt{2}}$$

$$\Rightarrow \quad \omega_c = \frac{1}{RC}, \quad |H(j\omega)| = \frac{1}{\sqrt{1 + (\frac{\omega_c}{\omega})^2}} \quad and \quad phase, \phi = \tan^{-1}(\frac{\omega_c}{\omega})$$

Input and output impedances of this filter can be found similar to the procedure used for low-pass filters:

$$Input\ impedance: Z_i = R + \frac{1}{j\omega C} \quad and \quad Z_i|_{min} = R$$

$$Output\ Impedance: Z_0 = R \left\| \frac{1}{j\omega C} \right. \quad and \quad Z_0|_{max} = R$$

Bode Plots of first-order high-pass filters display the following typical characteristics:
At low frequencies, $\omega/\omega_c << 1$, $|H(j\omega)| \propto \omega$ (a +20dB/decade line) and $\phi \approx 90^0$.
At high frequencies, $\omega/\omega_c >> 1$, $|H(j\omega)| \approx 1$ (a line with a slope of 0) and $\phi \approx 0^0$.

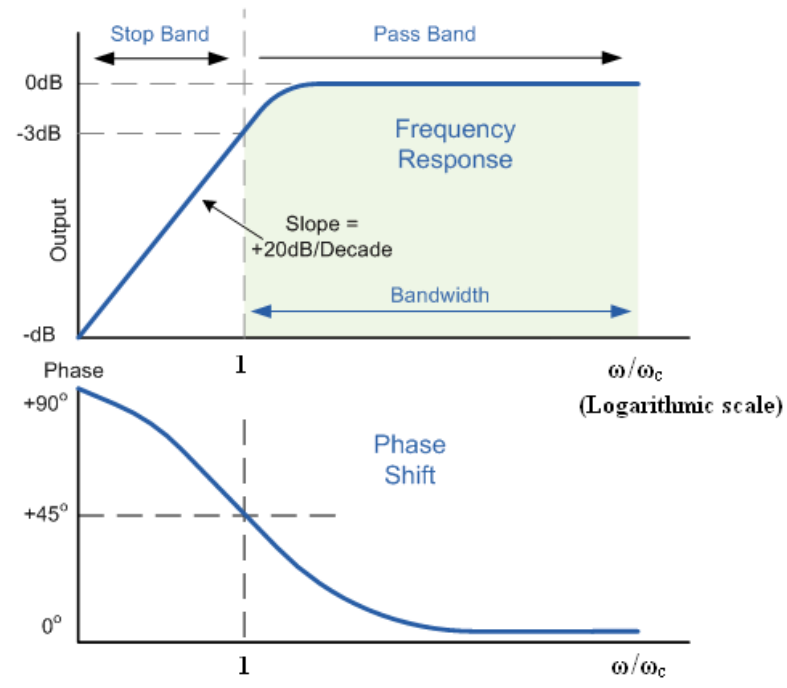
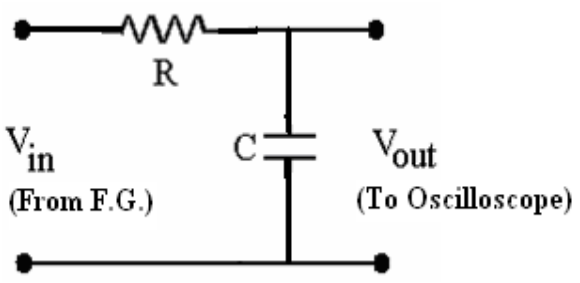


Fig.7: Bode Plots for high-pass RC filter

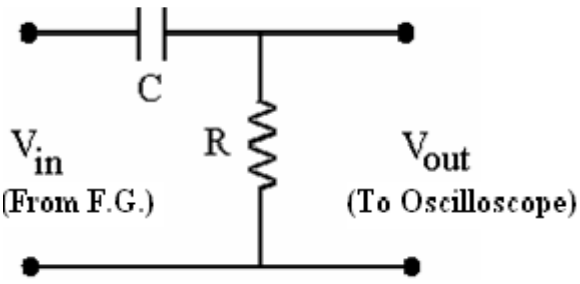
Circuit Components/Instruments:

(i) Resistor (5.6 kΩ), (ii) Capacitor (10 kpF), (iii) Function generator, (iv) Oscilloscope, (v) Connecting wires, (vi) Breadboard

Circuit Diagrams:



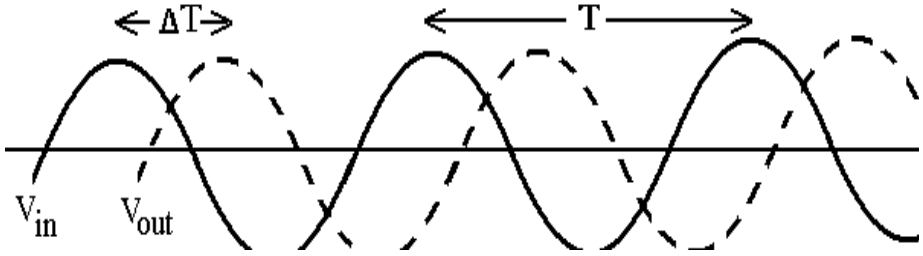
(Low pass RC filter)



(High pass RC filter)

Procedure:

1. Begin lab by familiarizing yourself with the function generator and oscilloscope.
2. Read and also measure the values of R and C.
3. Using the scope set the function generator to produce a 10 V(pp) sine wave. This signal will be used for the input. Do not change the amplitude of this signal during the experiment.
4. Set up the low/high pass RC filter on the breadboard as shown in the circuit diagram. Use the function generator to apply a 10 V(pp) sine wave signal to the input. Use the dual trace oscilloscope to look at both V_{in} and V_{out}. Be sure that the two oscilloscope probes have their grounds connected to the function generator ground.
   For several frequencies between 20 Hz and 20 kHz (the audio frequency range) measure the peak-to-peak amplitude of V_{out} and phase difference ϕ. You could also calculate phase shift by measuring lead/lag time, ΔT, as shown in the diagram below using the expression, $\phi_R(\text{deg}) = \left(\frac{\Delta T}{T}\right) \times 360^0$.
   Check often to see that V_{in} remains roughly at the set value. Take enough data (at least up to 10 times the cut-off frequency, for low pass and down to 1/10 times cut-off frequency, for high pass filter) so as to make your analysis complete. If needed use the STOP button of oscilloscope at a desired frequency to acquire data.



5. From your measurements determine the ratio

$$|H(j\omega)| = \left|\frac{V_o}{V_i}\right| = \frac{V_o(pp)}{V_i(pp)}$$

and compute this ratio by using the formula

$|H(j\omega)| = \dfrac{1}{\sqrt{1+\left(\dfrac{\omega}{\omega_c}\right)^2}}$, for low pass filter and

$|H(j\omega)| = \dfrac{1}{\sqrt{1+\left(\dfrac{\omega_c}{\omega}\right)^2}}$, for high pass filter

Observations:

R = _____, C = _____

(I) For Low Pass Filter: $V_{in}(pp)$ **= _____,** ω_c = 1/RC = _____

(a) Table for |H(jω)|:

Sl. No.	**Frequency, f (kHz)** ($\omega = 2\pi f$)	$\dfrac{\omega}{\omega_c}$	**V_0(pp) (Volt)**	**$\left\|\dfrac{V_o(pp)}{V_i(pp)}\right\|$**	**$\|H(j\omega)\|_{dB}$**	$\|H(j\omega)\| = \dfrac{1}{\sqrt{1+\left(\dfrac{\omega}{\omega_c}\right)^2}}$
1	0.02					
2	..					
..	..					
..	..					

(b) Table for phase angle ϕ:

Sl. No.	**Frequency, f (kHz)** ($\omega = 2\pi f$)	$\dfrac{\omega}{\omega_c}$	**ΔT (ms)**	**T (ms)**	$\phi = \left(\dfrac{\Delta T}{T}\right)\times 360^0$ **(deg)**	$\phi = -\tan^{-1}\left(\dfrac{\omega}{\omega_c}\right)$ **(deg)**
1	0.02					
2	..					
..	..					
..	..					

(II) For High Pass Filter: $V_{in}(pp)$ **= _____,** ω_c **= 1/RC = _____**

(c) Table for |H(jω)|:

Sl. No.	Frequency, f (kHz) ($\omega = 2\pi f$)	$\frac{\omega_c}{\omega}$	V_0(pp) (Volt)	$\lvert H(j\omega)\rvert = \frac{V_o(pp)}{V_i(pp)}$	$\lvert H(j\omega)\rvert_{dB}$	$\lvert H(j\omega)\rvert = \frac{1}{\sqrt{1+\left(\frac{\omega_c}{\omega}\right)^2}}$
1	0.02					
2	..					
..	..					
..	..					

(d) Table for phase angle ϕ:

Sl. No.	Frequency, f (kHz) ($\omega = 2\pi f$)	$\frac{\omega_c}{\omega}$	ΔT (ms)	T (ms)	$\phi = \left(\frac{\Delta T}{T}\right) \times 360^0$ (deg)	$\phi = -\tan^{-1}(\frac{\omega_c}{\omega})$ (deg)
1	0.02					
2	..					
..	..					
..	..					

Graphs: Trace and study bode plots of $|H(j\omega)|_{dB}$ and ϕ versus $f(\times 2\pi)$ in a semi-log format for low/high pass RC filter. Determine the cut-off frequency from graph. Also, estimate the frequency roll-off for each filter.

Discussions:

Precautions:

Active Filter Circuits using Operational amplifiers

Objectives:

To construct and analyze the frequency response of
(ii) A low pass active filter
(iii) A high pass active filter
(iv) A band pass active filter

Overview:

The main disadvantage of passive filters (as you have already seen in one of your previous labs) is the fact that the maximum gain that can be achieved with these filters is 1. In other words, the maximum output voltage is equal to the input voltage. If we make filter circuits using Opamps, then the gain can be greater than 1.
The circuits employed are all based on the inverting Opamps with the addition of a capacitor placed in the correct position for the particular type of filter. These circuits are called active filter circuits because they use Opamps which require a power supply.

Low-Pass filters - the integrator reconsidered

A low pass filter passes only low frequency signals and attenuates signals of high frequencies. We have already considered the time response of the integrator circuit, but its frequency response can also be studied. Figure 1 shows a low pass active filter in inverting configuration.

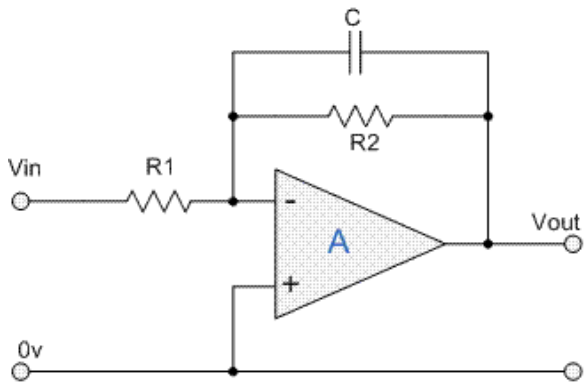


Fig. 1: First Order Low Pass Filter with Op Amp

Gain of the above circuit, $A_V = -\dfrac{R_2 \| X_C}{R_1}$,

where $X_C = \dfrac{1}{2\pi f C}$, is the impedance of the capacitor and f is the frequency of the input signal.

At high frequencies the capacitor acts as a short, so the gain of the amplifier approaches zero. At very low frequencies the capacitor is open and the gain of the circuit is $-(R_2/R_1)$. We can consider the frequency to be high when the large majority of current goes through the capacitor; i.e., when the magnitude of the capacitor impedance is much less than that of R_2. In other words, we have high frequency when $X_C << R_2$. Since R_2 now has little effect on the circuit, it should act as an integrator. Likewise low frequency occurs when $R_2 << X_C$, and the circuit will act as an amplifier with gain $-R_2/R_1$. Thus, the cut-off frequency is given as $f_C = \frac{1}{2\pi R_2 C}$ and the frequency response is as shown below (Fig.2). The frequency response curve of the filter decreases by 20dB/Decade or 6dB/Octave from the determined cut-off frequency point which is always at -3dB below the maximum gain value.

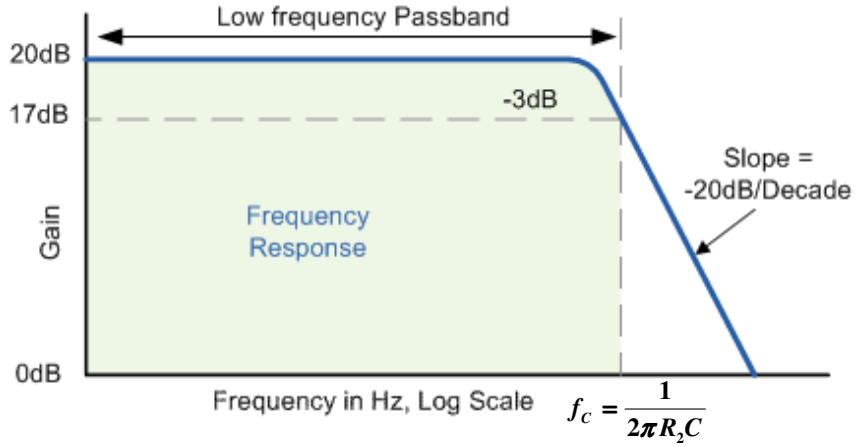


Fig. 2: Frequency response curve of an active low pass filter

High-Pass filters - the differentiator reconsidered

A high pass filter passes only high frequency signals attenuating low frequency signals. The circuit below is similar to a differentiator circuit and acts as a high pass filter.

Gain of the circuit shown, $A_V = -\frac{R_2}{R_1 + X_C}$.

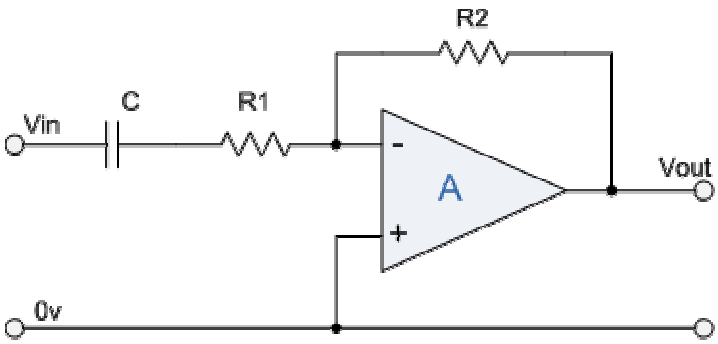


Fig. 3: First Order High Pass Filter with Op Amp

At high frequencies the capacitor acts as a short, so the gain of the amplifier becomes $-R_2/R_1$. At very low frequencies the capacitor is open and the gain of the circuit approaches 0. For this circuit, the cut-off frequency is $f_C = \frac{1}{2\pi R_1 C}$. However, it has to be kept in mind that the maximum pass band frequency response of an active high pass filter is limited to the characteristics or bandwidth of the op-amp being used within the circuit design. In the previous experiments you have seen that the maximum frequency response of an op-amp is limited to the Gain/Bandwidth product or open loop voltage gain of the operational amplifier being used. The frequency response of a high pass filter of gain 40 dB is shown below in Fig. 4. The upper curve is the typical frequency response of the opamp with open-loop. The frequency response curve of the filter increases by 20dB/Decade or 6dB/Octave up to the determined cut-off frequency point.

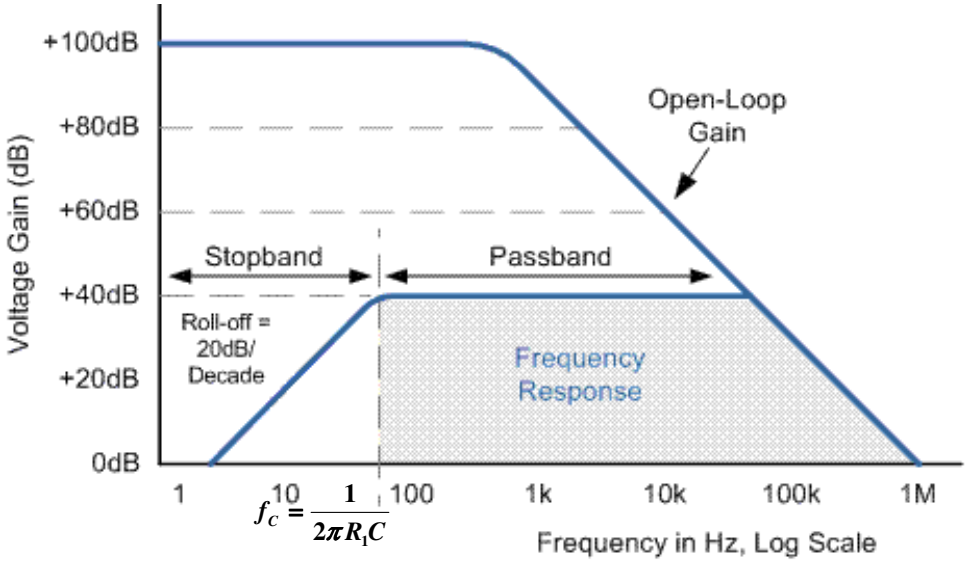


Fig. 4: Frequency response curve of an active high pass filter

Band-Pass Filter

A band-pass filter rejects high and low frequencies, passing signals around some intermediate frequency only. The simplest band-pass filter can be made by combining the first order low pass and high pass filters that we just looked at. The cut-off or corner frequency of the low pass filter (LP) is higher than the cut-off frequency of the high pass filter (HP) and the difference between the frequencies at the -3dB point will determine the "Bandwidth" of the filter while attenuating any signals outside of these points.

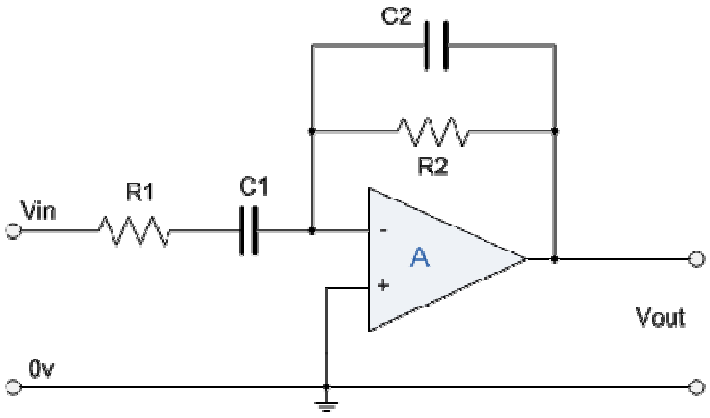


Fig. 5: Band Pass Filter with Op Amp

This circuit will attenuate low frequencies $f_{HP} = \frac{1}{2\pi R_1 C_1}$ and high frequencies $f_{LP} = \frac{1}{2\pi R_2 C_2}$ ($f_{LP} > f_{HP}$), but will pass intermediate frequencies with a gain of $-R_2/R_1$. However, this circuit cannot be used to make a filter with a very narrow band.

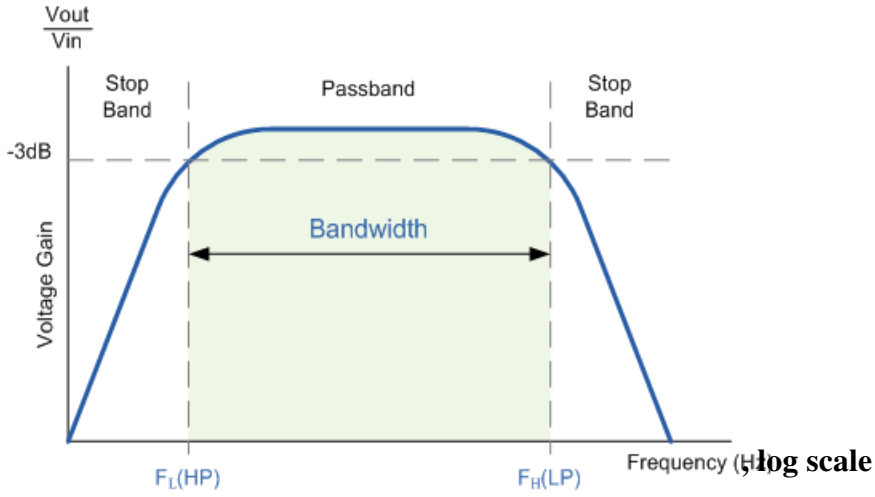


Fig.6: Frequency response curve of an active band pass filter

Circuit Components/Equipments:

(i) OPAMP (IC741), (ii) Resistors, (iii) Capacitors, (iv) Function generator, (v) Oscilloscope, (vi) Breadboard, (vii) Connecting wires.

Circuit Diagrams: Refer to Figs 1, 3 and 5.

Procedure:

6. Read/measure the values of all circuit components to be used. Calculate the cut-off frequencies in each case.

7. Using the scope set the function generator to produce an input voltage of approximately 100 mV(pp) sine wave.
8. Set up the low/high/band pass active filter on the breadboard as shown in the circuit diagrams. Connect the function generator to apply input. Use the dual trace oscilloscope to look at both V_{in} and V_{out}. Be sure that the two oscilloscope probes have their grounds connected to the function generator ground. Match the magnification control both at the probe and the oscilloscope.
9. Set the RANGE of the function generator between 20 Hz to 20 kHz. Measure the $V_{in}(pp)$ and $V_{out}(pp)$. Use **digital filter** or **average** options from oscilloscope to measure voltages whenever needed.
10. From your measurements determine the gain, $\frac{V_o(pp)}{V_{in}(pp)}$ and compare with the calculated value.
11. Plot log f ~ gain (dB).

Observations:

(III) For Low Pass Filter:

R_1=_____, R_2 = _____, C=_____,

Max Gain(calculated) = $-\frac{R_2}{R_1}$ =_____, $f_c = \frac{1}{2\pi R_2 C}$ = _____

Table:

Sl. No.	Frequency, f (kHz)	V_{in}(pp) (Volt)	V_o(pp) (Volt)	Gain,A_V= $\frac{V_o(pp)}{V_i(pp)}$	Gain (dB)
1					
2					
..					
..					

(IV) For High Pass Filter:

R_1 = _____, R_2 = _____, C = _____,

Max Gain(calculated) = $-\frac{R_2}{R_1}$ =_____, $f = \frac{1}{2\pi R_1 C}$ = _____

Table:

Sl. No.	Frequency, f (kHz)	V_{in}(pp) (Volt)	V_0(pp) (Volt)	Gain,A_V= $\frac{V_o(pp)}{V_i(pp)}$	Gain (dB)
1					
2					
..					

..					

(V) For Band Pass Filter:

R_1 = _____, R_2 = _____, C_1 =_____, C_2 = _____

Max Gain(calculated) = $-\frac{R_2}{R_1}$ =_____,

$$f_{HP} = \frac{1}{2\pi R_1 C_1} = \_\_\_\_\_,\ f_{LP} = \frac{1}{2\pi R_2 C_2} = \_\_\_\_\_$$

Table:

Sl. No.	**Frequency, f (kHz)**	**V_{in}(pp) (Volt)**	**V_0(pp) (Volt)**	**Gain,A_V= $\frac{V_o(pp)}{V_i(pp)}$**	**Gain (dB)**
1					
2					
..					
..					

Graphs: Plot the frequency response curve for each filter and determine the cut-off frequencies and frequency roll-off rate (in dB/decade) in each case.

Discussions:

Precautions:
