RESISTOR COLOUR CODES

RESISTORS WITH FOUR COLOURED BANDS

For traditional resistors there are usually FOUR coloured bands. The first two coloured bands will show the first two digits, the third band provides the multiplier by which the first two digits must be multiplied (i.e number of zeros); together this gives the value of the resistor (the resistance) in Ohms. The fourth coloured band indicates the tolerance of the resistor that is how close the actual resistance may be to the value indicated. A 1k Ohm (1000 Ohm) resistor with a 20% tolerance could have a value anywhere between 800 and 1200 Ohms.

The tolerance band is sometimes spaced further apart from the other three bands, which helps when deciding which way round to read off the value, which is sometimes difficult to establish immediately.

FIRST DIGIT First Colour Band		SECOND DIGIT Second Colour Band		MULTIPLIER Third Colour Band	
BLACK	0	BLACK	0	BLACK	x 1
BROWN	1	BROWN	1	BROWN	x 10
RED	2	RED	2	RED	x 100
ORANGE	3	ORANGE	3	ORANGE	x 1,000
YELLOW	4	YELLOW	4	YELLOW	x 10,000
GREEN	5	GREEN	5	GREEN	x 100,000
BLUE	6	BLUE	6	BLUE	x 1,000,000
VIOLET	7	VIOLET	7	VIOLET	x 10,000,000
GREY	8	GREY	8	GREY	x 100,000,000
WHITE	9	WHITE	9	WHITE	x 1,000,000,000

Tolerance
Fourth Colour Band:

BROWN	RED	GOLD	SILVER	SALMON
1%	2%	5%	10%	20%

Examples:
BROWN BLACK BROWN SILVER = 10 × 10 = 100 Ohms (Usually expressed as 100R) 10% Tolerance
YELLOW VIOLET RED GOLD = 47 × 100 = 4700 Ohms (Usually expressed as 4.7K) 5% Tolerance
ORANGE ORANGE YELLOW SILVER = 33 × 10,000 = 330000 Ohms (Usually expressed as 330K) 10% Tolerance

RESISTORS WITH FIVE COLOURED BANDS

A number of resistors have FIVE coloured bands to indicate their resistance value and tolerance. The first three bands indicate the first three digits, the fourth band provides the multiplier by which the first three digits must be multiplied (i.e. number of zeros) the result gives the value of the resistor. The fifth band indicates the tolerance. Again it is often difficult to tell which way round to read off the value, but the tolerance band is usually spaced a little further apart from the first four bands.

FIRST DIGIT First Colour Band		**SECOND DIGIT Second Colour Band**		**THIRD DIGIT Third Colour band**		**MULTIPLIER Fourth Colour Band**	
BLACK	0	BLACK	0	BLACK	0	BLACK	x 1
BROWN	1	BROWN	1	BROWN	1	BROWN	x 10
RED	2	RED	2	RED	2	RED	x 100
ORANGE	3	ORANGE	3	ORANGE	3	ORANGE	x 1,000
YELLOW	4	YELLOW	4	YELLOW	4	YELLOW	x 10,000
GREEN	5	GREEN	5	GREEN	5	GREEN	x 100,000
BLUE	6	BLUE	6	BLUE	6	BLUE	x 1,000,000
VIOLET	7	VIOLET	7	VIOLET	7		
GREY	8	GREY	8	GREY	8	GOLD	x 0.1
WHITE	9	WHITE	9	WHITE	9	SILVER	x 0.01
Tolerance Fifth Colour Band:							
BROWN 1%							

Examples:

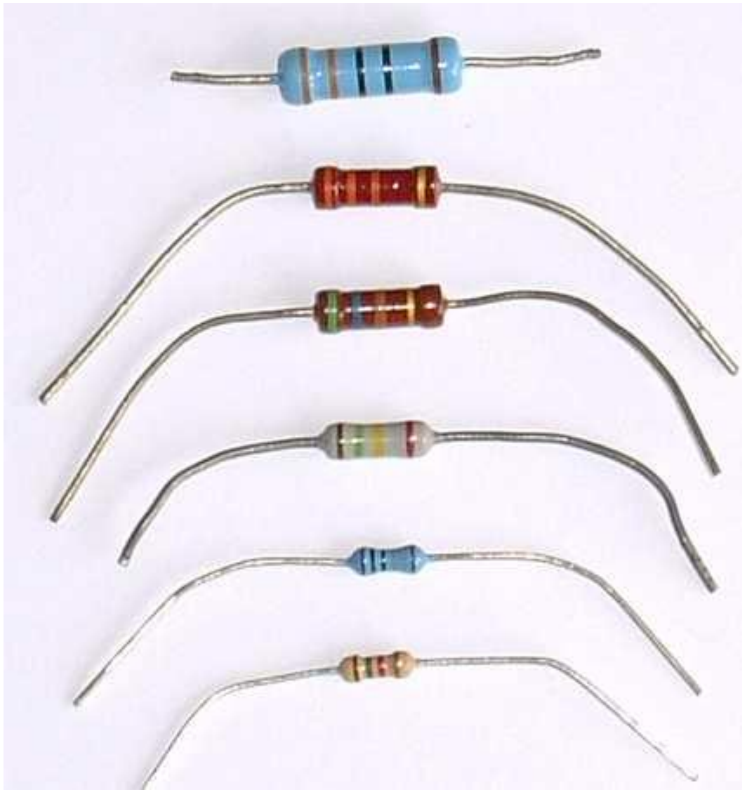
BROWN BLACK BLACK BLACK BROWN = 100 × 1 = 100 Ohms (100R) 1% Tolerance

YELLOW VIOLET BLACK BROWN BROWN = 470 × 10 = 4700 Ohms (4.7K) 1% Tolerance

ORANGE ORANGE BLACK ORANGE BROWN = 330 × 1000 = 330000 Ohms (330K)

1% Tolerance

N.B. Five band resistors are always +/- 1% tolerance

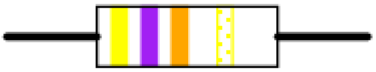


Resistor Color Codes

Color	Digit	Multiplier	Tolerance (%)
Black	0	10^0 (1)	
Brown	1	10^1	1
Red	2	10^2	2
Orange	3	10^3	
Yellow	4	10^4	
Green	5	10^5	0.5
Blue	6	10^6	0.25
Violet	7	10^7	0.1
Grey	8	10^8	
White	9	10^9	
Gold		10^{-1}	5
Silver		10^{-2}	10
(none)			20

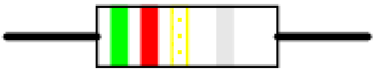
The colors brown, red, green, blue, and violet are used as tolerance codes on 5-band resistors only. All 5-band resistors use a colored tolerance band. The blank (20%) "band" is only used with the "4-band" code (3 colored bands + a blank "band").

Example #1



A resistor colored *Yellow-Violet-Orange-Gold* would be 47 kΩ with a tolerance of +/- 5%.

Example #2



A resistor colored *Green-Red-Gold-Silver* would be 5.2 Ω with a tolerance of +/- 10%.

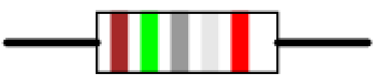
Example #3

A resistor colored *White-Violet-Black* would be 97 Ω with a tolerance of +/- 20%. When you see only three color bands on a resistor, you know that it is actually a 4-band code with a blank (20%) tolerance band.

Example #4

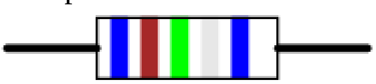
A resistor colored *Orange-Orange-Black-Brown-Violet* would be 3.3 kΩ with a tolerance of +/- 0.1%.

Example #5

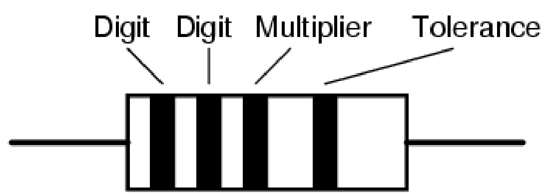


A resistor colored *Brown-Green-Grey-Silver-Red* would be 1.58 Ω with a tolerance of +/- 2%.

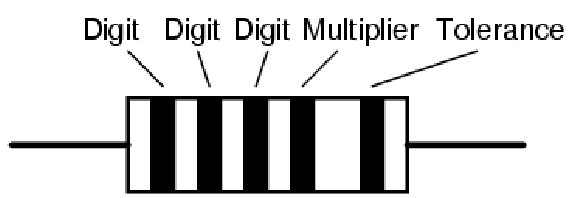
Example #6



A resistor colored *Blue-Brown-Green-Silver-Blue* would be 6.15 Ω with a tolerance of +/- 0.25%.



4-band code



5-band code

<u>SURFACE MOUNT RESISTORS</u> - SMD (Surface Mount Devices) - SMT (Surface Mount Technology)

I have encountered two types of Surface Mount Resistor. The main type seems to be the standard tolerance type which is marked with three identifier digits, the other is a precision tolerance type marked with four identifier digits:

SMD Resistors with <u>Three</u> Digits:

The first two digits are the significant digits, the third is the number of zeros (i.e. power of 10). Example:

270 would indicate a 27 Ohm device: 27 with no zeros (10^0) = 27 Ohms
(To avoid confusion some devices omit the last digit so 27 Ohms would be marked simply as 27 and 33 Ohms would be marked as 33 etc.)

271 would indicate a 270 Ohm device: 27 plus 1 zero (10^1) = 270 Ohms
682 would indicate a 6.8K Ohm device: 68 plus 2 zeros (10^2) = 6,800 Ohms
333 would indicate a 33K Ohm device: 33 plus 3 zeros (10^3) = 33,000 Ohms
274 would indicate a 270K Ohm device. 27 plus 4 zeros (10^4) = 270,000 Ohms
475 would indicate a 4.7M Ohm device. 47 plus 5 zeros (10^5) = 4,700,000 Ohms

A device marked 000 or 0 has essentially no (extremely small) resistance and would be used as a 'link' on an SMD PCB.

SMD resistors having a value lower than 10 ohms:
2R7 = 2.7 ohms
0R27 = 0.27 ohms
0R05 = 0.05 ohms
(In these cases the R indicates the decimal point - familiar on many schematic circuit diagrams.)

SMD Resistors with <u>Four</u> Digits (Precision Tolerance)**:**

In this case the first three digits are the significant digits. The fourth digit indicates the number of zeros (i.e. the power of ten). E.G:

2700 would indicate a 270 Ohm device: 270 plus no zeros (10^0) = 270 Ohms
2703 would indicate a 270K Ohm device. 270 plus 3 zeros (10^3) = 270,000 Ohms
4704 would indicate a 4.7M Ohm device. 470 plus 4 zeros (10^4) = 4,700,000 Ohms

Such a device marked 000 or 0000 has essentially no (extremely small) resistance and would be used as a 'link' on an SMD PCB.

CAPACITOR CONVERSION TABLE

The unit of capacitance is the Farad. The Farad, however, is too large a unit for use with typical electronic circuits, so it is divided into much smaller units, for example the microfarad which is 0.000001 Farads.

LARGE CAPACITORS

Electrolytic

Probably, the most common large capacity capacitor is the *Electrolytic* type. Most *Electrolytic* capacitors are clearly marked with the value of the capacitor in microfarads (μF), the polarity of the leads, and the working voltage. For this reason electrolytic capacitors are often the easiest capacitors to identify and use.

Most electrolytic capacitors will have clearly printed on the body something like: "220μF 50volts". It is very important to remember that most electrolytic capacitors are polarised i.e. the must be connected the correct way round in the circuit - to identify polarity these capacitors will generally have a (usually white) stripe down one side with a -ve sign to indicate that lead is to go only to the negative side of the circuit and the +ve lead will usually be longer than the -ve lead to help identification. Because DC is usually present in a circuit an electrolytic capacitor must be connected the right way round, if it is connected the wrong way round it may explode, so be careful.

Tantalum

Another type of capacitor that is available in large capacities is the *Tantalum Bead* type, they are much smaller than electrolytic capacitors and also usually have lower working voltages. Tantalum capacitors are also polarised and must be connected the right way round in the circuit. Modern tantalum bead capacitors have the value printed on the casing along with the voltage and polarity marking.

Older ones use a colour code in the form of stripes and a spot. The top two stripes give the first two digits - using the colours in the table below, and the spot is the multiplier: Grey Spot = $\times 0.01$: White Spot = $\times$ 0.1 : Black Spot = $\times$ 1 . The third (bottom stripe) is the voltage marking - yellow being 6.3V; black being 10V; green being 16V; blue being 20V; grey being 25V; white being 30V and pink being 35V. The positive lead is the one on the right hand side when the spot is facing you.

SMALL CAPACITORS

Small value capacitors will be unpolarised and therefore can be connected into a circuit either way around. Many circuits specify small capacitors. They are available in a wide range of values, with the various polyester types and ceramic capacitors being popular choices. Some circuits may specify capacitor values in *microfarads* (μF), some in *nanofarads* (nF) while others may use *picofarads* (pF). The different and varied types of component marking used on capacitors can all be rather confusing.

PRINTED VALUES

Some capacitors simply have the value printed on them which sounds easy, but you have to know if the number is in microfarads, nanofarads or picofarads. It seems to be common that if, for example, a capacitor is marked 0.22 this means 0.22 microfarads (μF) and if the printed marking is, for example, 2n2 then this would be a 2.2nF (nanofarad) capacitor.

SIMPLE TWO DIGIT MARKINGS

Often the capacitor will simply be marked with a two digit number printed on the body such as "10" for example. This indicates that it is a 10pF capacitor. However you may find some capacitors marked "10n" and this capacitor will have a value of 10nF (ie 10,000pF), this is sometimes seen on polystyrene types and some resin dipped ceramics.

CODED THREE DIGIT MARKINGS

Many capacitors use a coded marking system, and it seems that the majority of modern capacitors that I have used in recent years fall in to this category, so here is a guide: When we get our bag full of bits through the post, or eventually arrive home from the electronics shop with our little plastic bag full of components, keen to construct a circuit we will often find that many capacitors are marked with a three digit code such as "103" or "104" and some others have a three digit code plus a letter on the end such as "101K" or "102K". This can lead to a bit of 'head scratching' before construction of our exciting project can begin. Once we can familiarize ourselves with these codes or have a chart at hand then progress to the all important construction stage will be much swifter.

The capacitors marked with three digits are similar to resistors in that the first two digits need to be multiplied by the third digit in order to obtain the value in PICOFARADS (pF) as above. The letter is present to indicate the tolerance of the component. So 100 would be 10pF multiplied by zero, i.e. 10 pF. 103 is 10 pF multiplied by 1000 ie 10,000pF or to put is another way 0.01 microfarads. 471K would be a 470pF capacitor with a 10% tolerance.

The code marking, when decoded, will provide the value in Picofarads (pF).

The table below shows you the values in microfarads (μF) and nanofarads (nF).

CODE / Marking	**μF microfarads**	**nF nanofarads**	**pF picofarads**	**F farad**
1RO	0.000001	0.001	1	10^{-12}
100	0.00001	0.01	10	10^{-11}
101	0.0001	0.1	100	10^{-10}
102	0.001	1	1,000	10^{-9}
103	0.01	10	10,000	10^{-8}
104	0.1	100	100,000	10^{-7}
105	1	1,000	1,000,000	10^{-6}
106	10	10,000	10,000,000	10^{-5}
107	100	100000	100,000,000	10^{-4}

CAPACITOR TOLERANCE TABLE	
C	+/- 0.25pF
D	+/- 0.5pF
F	1%
G	2%
J	5%
K	10%
M	20%
Z	+80 -20%

Examples:

103K = 0.01µF i.e 10nF with 10% Tolerance
104K = 0.1µF i.e. 100nF with 10% Tolerance
334J = 0.33µF with 5% Tolerance
101K = 1000pF 10% Tolerance
102J = 0.001uF 5% Tolerance
473J = 47,000 pF i.e. 47nF or 0.047 uF 5% Tolerance

POLYESTER CAPACITORS WITH COLOUR CODES:

It is quite unusual to find capacitors with colour codes as they are no longer manufactured, but they will still be found in older equipment and parts boxes. Sometimes you may run across such polyester caps which will be marked with coloured stripes rather than numbers. Three examples of these polyester capacitors with colour codes can be seen in the photograph below (Right hand side second row). The Fifth Colour Band is The Voltage Rating:

Below is the colour code for some of these capacitors and gives the value in PICOFARADS (pF).

FIRST DIGIT (pF) First Colour		**SECOND DIGIT (pF) Second Colour**		**MULTIPLIER Third Colour**		**TOLERANCE Fourth Colour**	
BLACK	0	**BLACK**	0	**BLACK**	x 1	**BLACK**	20 percent
BROWN	1	**BROWN**	1	**BROWN**	x 10		
RED	2	**RED**	2	**RED**	x 100		
ORANGE	3	**ORANGE**	3	**ORANGE**	x 1000		
YELLOW	4	**YELLOW**	4	**YELLOW**	x 10,000		
GREEN	5	**GREEN**	5	**GREEN**	x 100,000	**GREEN**	5 percent
BLUE	6	**BLUE**	6	**BLUE**	x 1,000,000		
VIOLET	7	**VIOLET**	7	**VIOLET**	x 10,000,000		
GREY	8	**GREY**	8	**GREY**	x 100,000,000		
WHITE	9	**WHITE**	9	**WHITE**	x 1,000,000,000	**WHITE**	10 percent

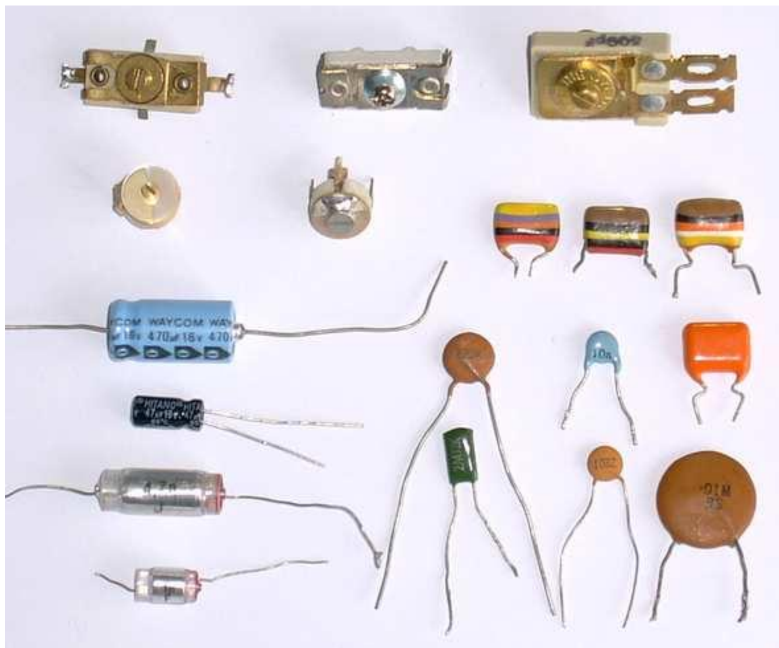
Brown 100 Volts		**Red 250 Volts**		**Yellow 400 Volts**	

The table for polyester capacitors works in pretty much the same way as for resistors. Look at the photo below and reading from the top of the capacitor the colours are:

Yellow = 4 Violet = 7 Orange = Multiply by 1000 Black = 20 % Tolerance Red = 250 Volts. This capacitor therefore has a value of 47,000 pF (i.e. 0.047µF) +/- 20% at 250V.

It must be remembered that unlike resistors there is no space between the coloured bands so if, for example, you have 22,000 pF capacitor of this type there will not be two separate thin red stripes but one thick red stripe.

POLYSTYRENE CAPACITORS:



These are quite rare and often look like silvery plastic cans with a wire at each end, and being made of polystyrene are easily damaged by heat, so care is needed when soldering with the use of a heatsink clip. Polystyrene capacitors generally have the value in pF (e.g. 470p) or nF (e.g 4.7n) and may have a letter to indicate the tolerance as per the table above (e.g. J - i.e. 5%) printed on the body and so are quite easy to identify.

The photo shows some examples of capacitors both variable trimmers, fixed electrolytics, ceramic disc, polyester, tantalum bead and polystyrene types. The polystyrene capacitors are shown on the bottom left hand side with the silvery plastic cans - they are quite rare today and the polystyrene is easily melted so great care needs to be taken when soldering.

More on voltage markings. Although this information is not entirely confirmed, some capacitors may have voltage indicated by a letter, as in the table below: (But this table is unconfirmed information.)		
D = 16 volts	Q = 500 volts	U = 4000 volts
F = 25 volts	R = 1000 volts	W = 5000 volts
H = 50 volts	S = 2000 volts	X = 6000 volts
K = 100 volts	T = 3000 volts	Y = 7500 volts

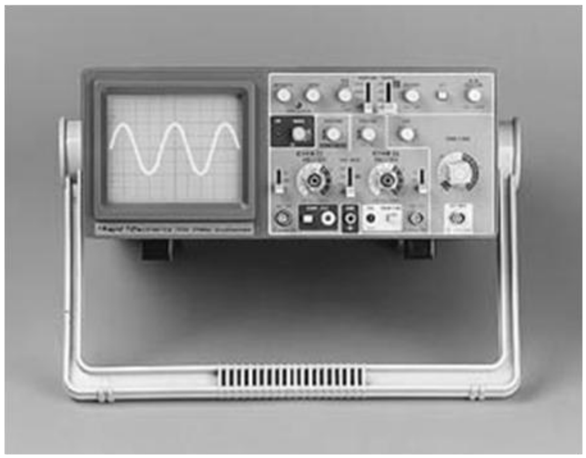
Oscilloscopes (CROs)

An oscilloscope is a test instrument which allows you to look at the 'shape' of electrical signals by displaying a graph of voltage against time on its screen. It is like a voltmeter with the valuable extra function of showing how the voltage varies with time. A graticule with a 1cm grid enables you to take measurements of voltage and time from the screen.

Circuit symbol for an oscilloscope

The graph, usually called the **trace**, is drawn by a beam of electrons striking the phosphor coating of the screen making it emit light, usually green or blue. This is similar to the way a television picture is produced.

Oscilloscopes contain a vacuum tube with a **cathode** (negative electrode) at one end to emit electrons and an **anode** (positive electrode) to accelerate them so they move rapidly down the tube to the screen. This arrangement is called an electron gun. The tube also contains electrodes to deflect the electron beam up/down and left/right.



Cathode Ray Oscilloscope (CRO)

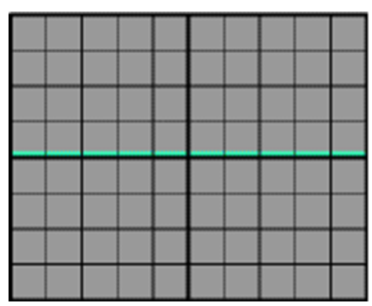
The electrons are called cathode rays because they are emitted by the cathode and this gives the oscilloscope its full name of **cathode ray oscilloscope** or CRO.

A **dual trace** oscilloscope can display two traces on the screen, allowing you to easily compare the input and output of an amplifier for example. It is well worth paying the modest extra cost to have this facility.

- An oscilloscope should be handled gently to protect its fragile (and expensive) vacuum tube.
- Oscilloscopes use high voltages to create the electron beam and these remain for some time after switching off - for your own safety do not attempt to examine the inside of an oscilloscope.

Setting up an oscilloscope

Oscilloscopes are complex instruments with many controls and they require some care to set up and use successfully. It is quite easy to 'lose' the trace off the screen if controls are set wrongly. There is some variation in the arrangement and labeling of the many controls so the following instructions may need to be adapted for your instrument.



This is what you should see after setting up, when there is no input signal connected

1. **Switch on** the oscilloscope to warm up (it takes a minute or two).
2. Do **not** connect the input lead at this stage.
3. Set the **AC/GND/DC** switch (by the Y INPUT) to **DC or AC**.

4. Set the **SWP/X-Y** switch to **SWP** (sweep).
5. Set **Trigger Level** to **AUTO**.
6. Set **Trigger Source** to **INT** (internal, the y input).
7. Set the **Y AMPLIFIER** to **5V/cm** (a moderate value).
8. Set the **TIMEBASE** to **10ms/cm** (a moderate speed).
9. Turn the timebase **VARIABLE** control to **1** or **CAL**.
10. Adjust **Y SHIFT** (up/down) and **X SHIFT** (left/right) to give a **trace across the middle of the screen**, like the picture.
11. Adjust **INTENSITY** (brightness) and **FOCUS** to give a bright, sharp trace.
12. The oscilloscope is now ready to use.

Connecting an oscilloscope

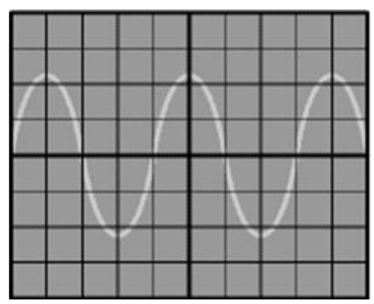
Oscilloscope lead and probes kit

The Y INPUT lead to an oscilloscope should be a **co-axial lead** and the diagram shows its construction. The central wire carries the signal and the screen is connected to earth (0V) to shield the signal from electrical interference (usually called noise).

Most oscilloscopes have a **BNC socket** for the y input and the lead is connected with a **push and twist action**, to disconnect you need to twist and pull. Oscilloscopes used in schools may have red and black 4mm sockets so that ordinary, unscreened, 4mm plug leads can be used if necessary.

Professionals use a specially designed lead and probes kit for best results with high frequency signals and when testing high resistance circuits, but this is not essential for simpler work at audio frequencies (up to 20kHz).

An oscilloscope is connected like a voltmeter but you must be aware that the screen (black) connection of the input lead is connected to mains earth at the oscilloscope. This means it **must** be connected to earth or 0V on the circuit being tested.



The trace of an AC signal with the oscilloscope controls correctly set

Obtaining a clear and stable trace

Once you have connected the oscilloscope to the circuit you wish to test you will need to **adjust the controls** to obtain a clear and stable trace on the screen:

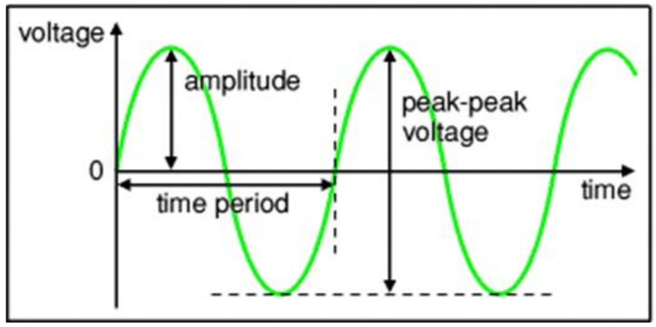
The **Y AMPLIFIER (VOLTS/CM)** control determines the height of the trace. Choose a setting so the trace occupies at least half the screen height, but does not disappear off the screen.

The **TIMEBASE (TIME/CM)** control determines the rate at which the dot sweeps across the screen. Choose a setting so the trace shows at least one cycle of the signal across the screen. Note that a steady DC input signal gives a horizontal line trace for which the timebase setting is not critical.

The **TRIGGER** control is usually best left set to AUTO.

If you are using an oscilloscope for the first time it is best to start with an easy signal such as the output from an AC power pack set to about 4V.

Measuring voltage and time period



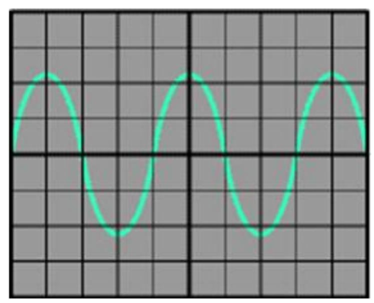
The trace on an oscilloscope screen is a graph of voltage against time. The shape of this graph is determined by the nature of the input signal.

In addition to the properties labelled on the graph, there is frequency which is the number of cycles per second.

The diagram shows a sine wave but these properties apply to any signal with a constant shape.

- **Amplitude** is the maximum voltage reached by the signal. It is measured in **volts**, **V**.
- **Peak voltage** is another name for amplitude.
- **Peak-peak voltage** is twice the peak voltage (amplitude). When reading an oscilloscope trace it is usual to measure peak-peak voltage.
- **Time period** is the time taken for the signal to complete one cycle. It is measured in **seconds (s)**, but time periods tend to be short so **milliseconds (ms)** and **microseconds (μs)** are often used. 1ms = 0.001s and 1μs = 0.000001s.
- **Frequency** is the number of cycles per second. It is measured in **hertz (Hz)**, but frequencies tend to be high so **kilohertz (kHz)** and **megahertz (MHz)** are often used. 1kHz = 1000Hz and 1MHz = 1000000Hz.

$$\text{Frequency} = \frac{1}{\text{Timeperiod}}$$



The trace of an AC signal
Y AMPLIFIER: 2V/cm
TIMEBASE: 5ms/cm

Voltage

Voltage is shown on the vertical y-axis and the scale is determined by the Y AMPLIFIER (VOLTS/CM) control. Usually peak-peak voltage is measured because it can be read correctly even if the position of 0V is not known. The amplitude is half the peak-peak voltage.

If you wish to read the amplitude voltage directly you must check the position of 0V (normally halfway up the screen): move the AC/GND/DC switch to GND (0V) and use Y-SHIFT (up/down) to adjust the position of the trace if necessary, switch back to DC afterwards so you can see the signal again.

Voltage = distance in cm × volts/cm

Example: peak-peak voltage = 4.2cm × 2V/cm = 8.4V

Amplitude (peak voltage) = ½ × peak-peak voltage = 4.2V

Time period

Time is shown on the horizontal x-axis and the scale is determined by the TIMEBASE (TIME/CM) control. The time period (often just called period) is the time for one cycle of the signal. The frequency is the number of cyles per second, frequency = 1/time period

Ensure that the variable timebase control is set to 1 or CAL (calibrated) before attempting to take a time reading.

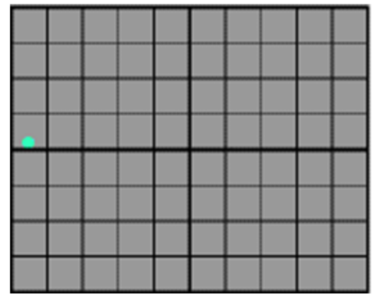
Time = distance in cm × time/cm

Example: Time period = 4.0cm × 5ms/cm = 20ms

Frequency = $^{1}/_{time\ period} = {^{1}/_{20ms}} = 50Hz$

Timebase (time/cm) and trigger controls

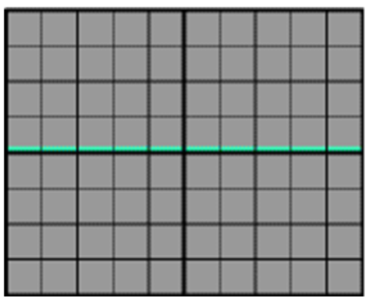
The oscilloscope sweeps the electron beam across the screen from left to right at a steady speed set by the TIMEBASE control. Each setting is labelled with the time the dot takes to move 1cm, effectively it is setting the scale on the x-axis. The timebase control may be labelled TIME/CM.



No input and slow timebase. You can see the dot moving

At slow timebase settings (such as 50ms/cm) you can see a dot moving across the screen but at faster settings (such as 1ms/cm) the dot is moving so fast that it appears to be a line.

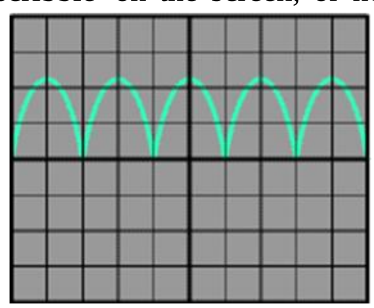
The VARIABLE timebase control can be turned to make a fine adjustment to the speed, but it must be left at the position labelled 1 or CAL (calibrated) if you wish to take time readings from the trace drawn on the screen.



No input and fast timebase. The dot is too fast to see so it appears to be a line

The TRIGGER controls are used to maintain a steady trace on the screen. If they are set wrongly you may see a trace drifting sideways, a confusing 'scribble' on the screen, or no trace at all! The trigger maintains a steady trace by starting the dot sweeping across the screen when the input signal reaches the same point in its cycle each time.

For straightforward use it is best to leave the trigger level set to AUTO, but if you have difficulty obtaining a steady trace try adjusting this control to set the level manually.



Varying DC (always positive)

Y amplifier (volts/cm) control

The oscilloscope moves the trace up and down in proportion to the voltage at the Y INPUT and the setting of the Y AMPLIFIER control. This control sets the voltage represented by each centimetre (cm) on the the screen, effectively it is setting the scale on the y-axis. Positive voltages make the trace move up, negative voltages make it move down.

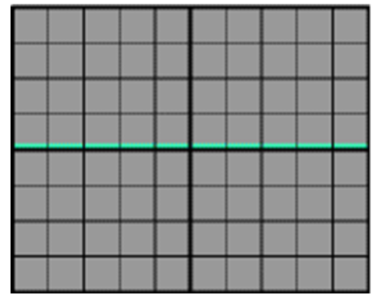
The y amplifier control may be labelled Y-GAIN or VOLTS/CM.

The input voltage moving the dot up and down at the same time as the dot is swept across the screen means that the trace on the screen is a graph of voltage (y-axis) against time (x-axis) for the input signal.

The AC/GND/DC switch

The normal setting for this switch is DC for all signals, including AC.

Switching to GND (ground) connects the y input to 0V and allows you to quickly check the position of 0V on the screen (normally halfway up). There is no need to disconnect the input lead while you do this because it is disconnected internally.



Switching to GND allows you to quickly check the position of 0V (normally halfway up)

Switching to AC inserts a capacitor in series with the input to block out any DC signal present and pass only AC signals. This is used to examine signals showing a small variation around one constant value, such as the ripple on the output of a smooth DC supply. Reducing the VOLTS/CM to see more detail of the ripple would normally take the trace off the screen! The AC setting removes the constant (DC) part of the signal, allowing you to view just the varying (AC) part which can now be examined more closely by reducing the VOLTS/CM. This is shown in the diagrams below:

Function generators

A **function generator** is usually a piece of electronic test equipment or software used to generate different types of electrical waveforms over a wide range of frequencies. Some of the most common waveforms produced by the function generator are the sine, square, triangular and saw tooth shapes. These waveforms can be either repetitive or single-shot (which requires an internal or external trigger source). Integrated circuits used to generate waveforms may also be described as function generator ICs.

Although function generators cover both audio and RF frequencies, they are usually not suitable for applications that need low distortion or stable frequency signals. When those traits are required, other signal generators would be more appropriate.

Some function generators can be phase-locked to an external signal source (which may be a frequency reference) or another function generator.

Function generators are used in the development, test and repair of electronic equipment. For example, they may be used as a signal source to test amplifiers or to introduce an error signal into a control loop.

Simple function generators usually generate triangular waveform whose frequency can be controlled smoothly as well as in steps. This triangular wave is used as the basis for all of its other outputs. The triangular wave is generated by repeatedly charging and discharging a capacitor from a constant current source. This produces a linearly ascending or descending voltage ramp. As the output voltage reaches upper and lower limits, the charging and discharging is reversed using a comparator, producing the linear triangle wave. By varying the current and the size of the capacitor, different frequencies may be obtained. Saw tooth waves can be produced by charging the capacitor slowly, using a current, but using a diode over the current source to discharge quickly - the polarity of the diode changes the polarity of the resulting sawtooth, i.e. slow rise and fast fall, or fast rise and slow fall.

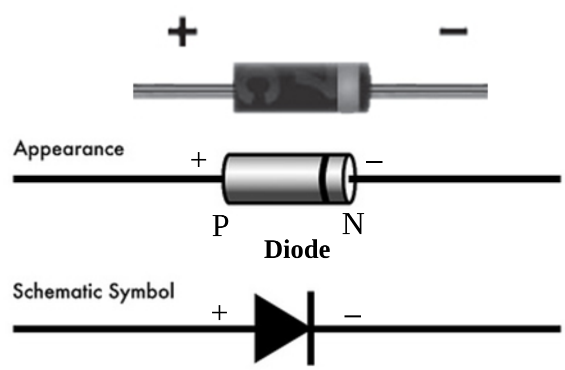
Typical specifications for a general-purpose function generator are:

- Produces sine, square, triangular, sawtooth (ramp), and pulse output. Arbitrary waveform generators can produce waves of any shape.
- It can generate a wide range of frequencies. For example, the Tektronix FG 502 (ca 1974) covers 0.1 Hz to 11 MHz.
- Frequency stability of 0.1 percent per hour for analog generators or 500 ppm for a digital generator.
- Maximum sinewave distortion of about 1% (accuracy of diode shaping network) for analog generators. Arbitrary waveform generators may have distortion less than -55 dB below 50 kHz and less than -40 dB above 50 kHz.
- Some function generators can be phase locked to an external signal source, which may be a frequency reference or another function generator.
- AM or FM modulation may be supported.
- Output amplitude up to 10 V peak-to-peak.
- Amplitude can be modified, usually by a calibrated attenuator with decade steps and continuous adjustment within each decade.
- Some generators provide a DC offset voltage, e.g. adjustable between -5V to +5V.
- An output impedance of 50 Ω.

Exp.No.3.1

Half wave & full wave Rectifiers - study of ripple factor with and without filter

(a) Half wave rectifier



Aim: To construct a half wave rectifier using a diode and to find out its ripple factor with and without filter. Also we aim to find out the rectification efficiency.

Components and accessories required: A step down transformer with input 230 volt and output 6, 9 or 12 volt. A diode (any one of the series from 1N4001 or DR25 series), carbon resistors or a resistance box, capacitor (electrolytic large value of 100 μF or 220 μF), voltmeter and an ammeter (or a multimeter).

Circuit, theory and procedure

Determination of dynamic resistance, r_d:

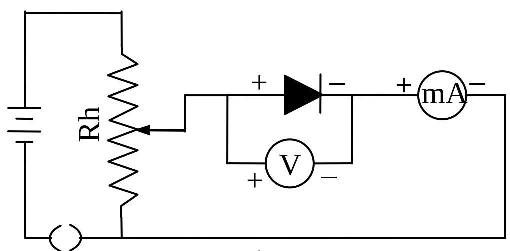


Fig.a

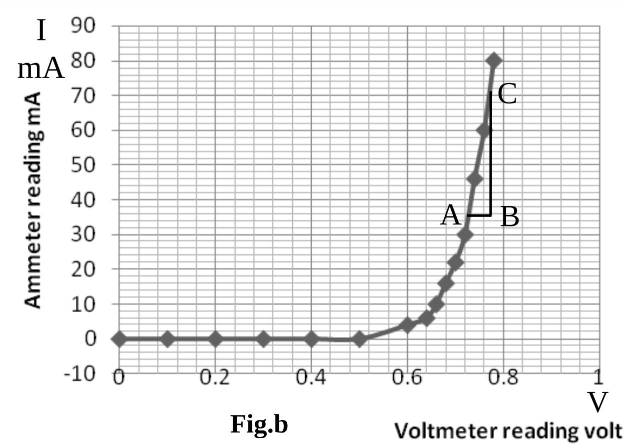


Fig.b

Connections are made as shown in fig.a. The rheostat is adjusted and the voltage is increased in steps and in each case the current in the milliammeter is noted. The graph between I and V, called the characteristics of the diode, is drawn. Calculate the dynamic resistance r_d at the straight line portion of the graph.

$$r_d = \frac{\Delta V}{\Delta I} = \frac{AB}{BC} \times 10^3\ \Omega, \text{ where, } \Delta I \text{ in milliamperes.}$$

Determination of ripple factor

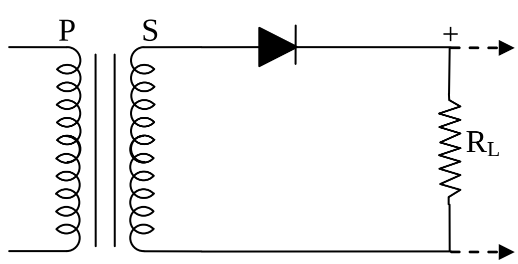


Fig.c: Half wave rectifier without filter

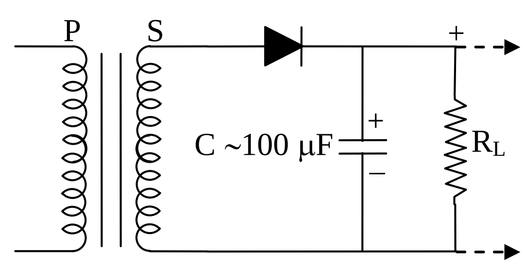


Fig.d: Half wave rectifier with filter

To construct a half wave rectifier without filter connections are made as shown in fig.c. The primary P of the step down transformer is connected to 230 volt a c mains supply. The diode is connected to the secondary of the transformer. R_L is the load resistance across which the output voltage is measured. We can use a resistance box or carbon rod resistors as load. Using a multimeter both a c and d c voltages are measured.

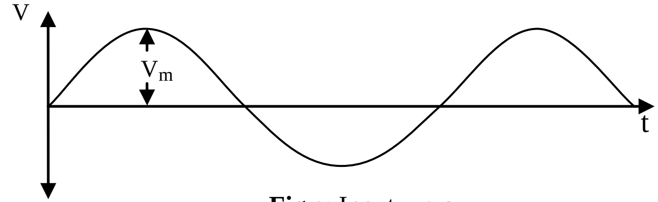


Fig.e: Input = a c

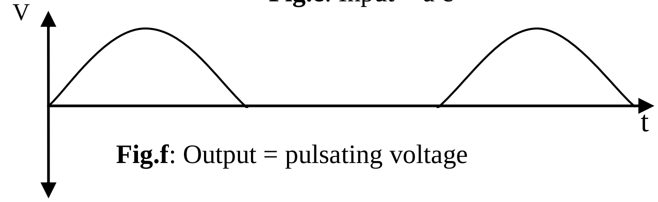


Fig.f: Output = pulsating voltage

Now connect a capacitor of the order of 100 or 220 μF parallel to the load R_L to bypass the ac voltage. The capacitor is called the filter capacitor and the resulting circuit is a half wave rectifier with filter. Again a c and d c voltages are measured.

The ripple factor and rectification efficiency are calculated as follows. Let i_L be the instantaneous current, which is the sum of d c I_{dc} and the instantaneous ac i'_{ac} . That is,

$$i_L = I_{dc} + i'_{ac} \tag{1}$$

The current through load is represented by,

$$i_L = I_m \sin(\omega t) \tag{2}$$

The r m s or effective value of the total current flowing through load is given as,

$$I_{rms} = \sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} i_L^2 d(\omega t)\right\}}$$

Since there is only current in the positive half cycles,

$$I_{rms} = \sqrt{\frac{1}{2\pi}\left\{\int_0^{\pi} I_m^2 \sin^2(\omega t) d(\omega t) + \int_{\pi}^{2\pi} 0.d(\omega t)\right\}} = \frac{I_m}{2} \tag{3}$$

The r ms value of the a c alone through the load is,

$$I'_{rms} = \sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} i'^2 d(\omega t)\right\}} = \sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} (i_L - I_{dc})^2 d(\omega t)\right\}}$$

$$= \sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} i_L^2 d(\omega t) + \int_0^{2\pi} i_{dc}^2 d(\omega t) - \int_0^{2\pi} 2 i_L I_{dc} d(\omega t)\right\}}$$

$$= \sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} i_L^2 d(\omega t) + \int_0^{2\pi} i_{dc}^2 d(\omega t) - \int_0^{2\pi} 2(I_{dc} + i'_{ac}) I_{dc} d(\omega t)\right\}}$$

$$= \sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} i_L^2 d(\omega t) + \int_0^{2\pi} i_{dc}^2 d(\omega t) - \int_0^{2\pi} 2(I_{dc}) I_{dc} d(\omega t)\right\}} = \sqrt{I_{rms}^2 - I_{dc}^2} \tag{4}$$

Since, $$\sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} i'_{ac}\, d(\omega t)\right\}} = 0$$

But I_{dc} is the average value of i_L over the complete cycle and is calculated as,

$$I_{dc} = i_{L\,av} = \frac{1}{2\pi}\left\{\int_0^{\pi} i_L\, d(\omega t)\right\}$$

$$= \frac{1}{2\pi}\left\{\int_0^{\pi} I_m \sin(\omega t)\, d(\omega t)\right\} = \frac{I_m}{\pi} \quad (5)$$

$$\text{Ripple factor} = \frac{\text{r m s value of a c voltage across load}}{\text{d c voltage across load}} = \frac{V'_{rms}}{V_{dc}} = \frac{I'_{rms}}{I_{dc}} \quad (6)$$

$$= \frac{\sqrt{I_{rms}^2 - I_{dc}^2}}{I_{dc}} = \sqrt{\frac{I_{rms}^2}{I_{dc}^2} - 1} = \sqrt{\frac{\left(\frac{I_m}{2}\right)^2}{\left(\frac{I_m}{\pi}\right)^2} - 1} = 1.21 \quad (7)$$

By measuring the r m s value of the a c voltage V'_{rms} and the d c voltage V_{dc} across the load resistance and using eqn.6 we can find out the ripple factor of the half wave rectifier. Repeat the experiment for different loads and in each case the ripple factor is calculated.

Now the filter capacitor C is connected parallel to the load and the a c and d c voltages are measured and the ripple factor with filter is calculated. This value of ripple factor will be much less than the ripple factor without filter. This is because the bypassing of a c through the filter capacitor. In this case also the experiment repeated for different loads.

Rectification efficiency:

The dc power delivered to the load, $P_{dc} = I_{dc}^2 R_L$

Total input a c power, $P_{ac} = I_{r\,m\,s}^2 (r_d + R_L)$

Rectification efficiency, $$\eta = \frac{P_{dc}}{P_{ac}} = \frac{I_{dc}^2 R_L}{I_{r\,m\,s}^2 (r_d + R_L)}$$

Using eqns.3 and 5, $$\eta = \frac{\left(\frac{I_m}{\pi}\right)^2 R_L}{\left(\frac{I_m}{2}\right)^2 (r_d + R_L)} = \left(\frac{4}{\pi^2}\right)\frac{R_L}{(r_d + R_L)} = \frac{0.4053}{\left(1 + \frac{r_d}{R_L}\right)} \quad (8)$$

If $r_d << R_L$, $\eta = 40.53\%$

It means that under best conditions, only 40.53% of a c input power is converted into d c power. The rest remains as a c power in load.

- For half wave rectifier we don't need a centre tapped transformer. Usually we use any two of the three output terminals of a centre taped transformer with input 230 and output 6-0-6, 9-0-9 or 12-0-12 volt.
- Use rheostat of low value of the order of 50 Ω and d c supply of 2V.
- In electronic experiments carbon resistors are preferred.

Observation and tabulation

Determination of dynamic resistance, r_d:

Voltmeter reading V volt	Ammeter reading I mA

Dynamic resistance of the diode, r_d = ……… ohm

Load R_L Ohm	Rectification efficiency η	Without filter			With filter		
		a c volt	d c volt	Ripple factor	a c volt	d c volt	Ripple factor
100							
200							
300							
400							
500							
600							
700							
800							
900							
1000							
2000							
3000							
4000							
5000							
6000							
7000							
8000							
9000							
10000							
∞							

Result

A half wave rectifier is constructed and their ripple factors without filter and with filter for various loads are determined. Also the rectification efficiencies for various loads are calculated.

*Components and Circuit testing

- A diode is good only if there is a low resistance reading in one direction (forward biasing) and high resistance in the other (reverse bias). So by using a multimeter as an ohmmeter you can check a diode.

(b) Full wave rectifier

Aim: To construct a full wave rectifier using two diodes and to find out its ripple factor with and without filter. We also aim to find out the rectification efficiency.

Components and accessories required:

A step down transformer with input 230 volt and output 6, 9 or 12 volt. Two identical diodes (any one of the series from 1N4001or DR25 series), resistances or a resistance box, a capacitor (electrolytic large value of 100 μF or 220 μF), voltmeter and an ammeter (or a multimeter).

Circuit, theory and procedure

Determination of dynamic resistance, r_d:

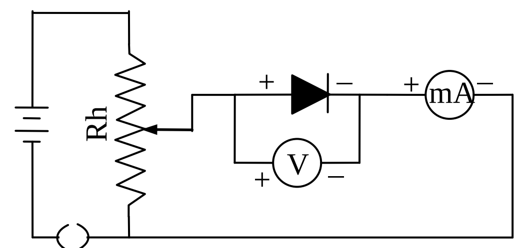


Fig.a

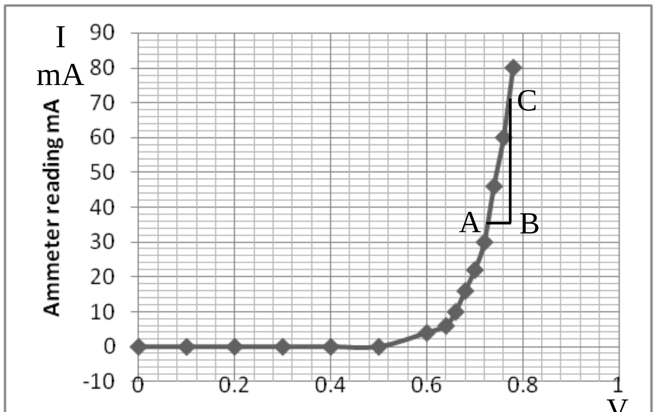


Fig.b

Connections are made as shown in fig.a. The rheostat is adjusted and the voltage is increased in steps and in each case the current in the milliammeter is noted. The graph between I and V, called the characteristics of the diode, is drawn. Calculate the dynamic resistance r_d at the straight line portion of the graph.

$$r_d = \frac{\Delta V}{\Delta I} = \frac{AB}{BC} \times 10^3 \ \Omega$$, where, ΔI in milliamperes.

Determination of ripple factor

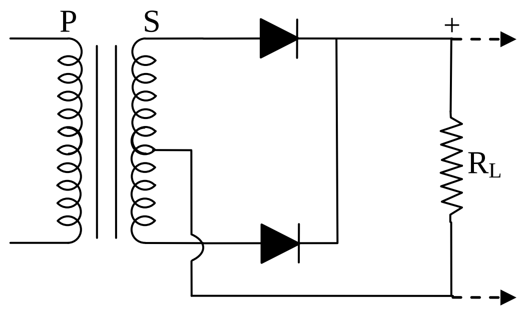


Fig.c: Full wave rectifier without filter

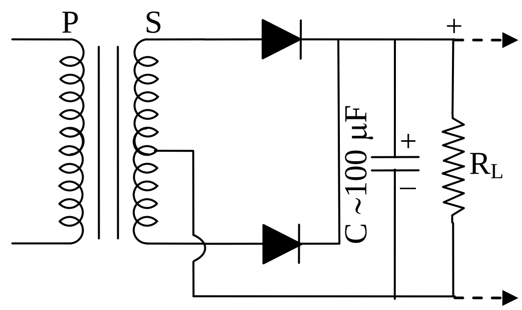


Fig.d: Full wave rectifier with filter

To construct a full wave rectifier without filter connections are made as shown in fig.c. The primary P of the step down transformer is connected to 230 volt a c mains supply. The diode is connected to the secondary of the transformer. R_L is the load resistance across which the output voltage is measured. We can use a resistance box or carbon rod resistors as load. Using a multimeter both a c and d c voltages are measured.

Now connect a capacitor of the order of 100 or 220 μF parallel to the load R_L to bypass the ac voltage as shown in fig.d. The capacitor is called the filter capacitor and the resulting circuit is a full wave rectifier with filter. Again a c and d c voltages are measured.

The ripple factor and rectification efficiency are calculated as follows. Let i_L be the instantaneous current, which is the sum of d c, I_{dc} and the instantaneous a c, i'_{ac}. That is,

$$i_L = I_{dc} + i'_{ac} \qquad (1)$$

The current through the load is represented by,

$$i_L = I_m \sin(\omega t) \qquad (2)$$

The r m s or effective value of the total current flowing through load is given as,

$$I_{rms} = \sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} i_L^2 d(\omega t)\right\}}$$

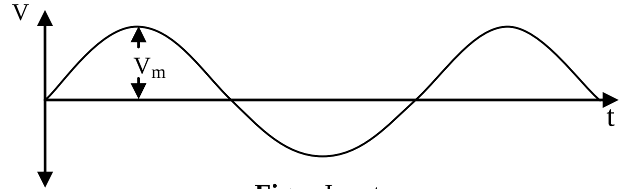


Fig.e: Input = a c

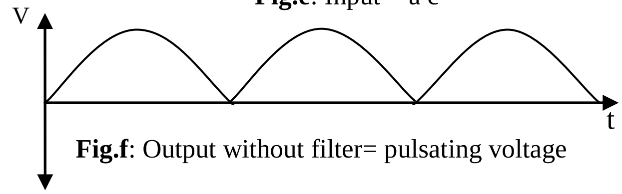


Fig.f: Output without filter= pulsating voltage

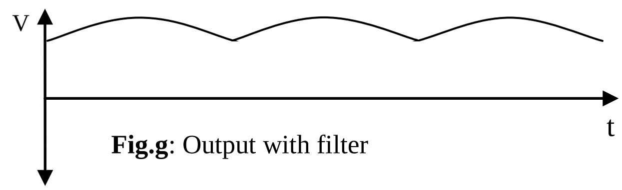


Fig.g: Output with filter

Since there is current in the full cycle,

$$I_{rms} = \sqrt{\frac{1}{2\pi}\left\{\int_0^{\pi} I_m^2 \sin^2(\omega t)\, d(\omega t) + \int_{\pi}^{2\pi} I_m^2 \sin^2(\omega t)\, d(\omega t)\right\}} = \frac{I_m}{\sqrt{2}} \qquad (3)$$

The r ms value of the a c alone through the load is,

$$I'_{rms} = \sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} i'^2 d(\omega t)\right\}} = \sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} (i_L - I_{dc})^2\, d(\omega t)\right\}}$$

$$= \sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} i_L^2 d(\omega t) + \int_0^{2\pi} i_{dc}^2 d(\omega t) - \int_0^{2\pi} 2 i_L I_{dc} d(\omega t)\right\}}$$

$$= \sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} i_L^2 d(\omega t) + \int_0^{2\pi} i_{dc}^2 d(\omega t) - \int_0^{2\pi} 2(I_{dc} + i'_{ac}) I_{dc} d(\omega t)\right\}}$$

$$= \sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} i_L^2 d(\omega t) + \int_0^{2\pi} i_{dc}^2 d(\omega t) - \int_0^{2\pi} 2(I_{dc}) I_{dc} d(\omega t)\right\}} = \sqrt{I_{rms}^2 - I_{dc}^2} \qquad (4)$$

Since, $\sqrt{\frac{1}{2\pi}\left\{\int_0^{2\pi} i'_{ac} d(\omega t)\right\}} = 0$

But I_{dc} is twice the average value of i_L over the complete cycle and is calculated as,

$$I_{dc} = 2 i_{L\,av} = 2 \times \frac{1}{2\pi}\left\{\int_0^{\pi} i_L d(\omega t)\right\}$$

$$= \frac{1}{\pi}\left\{\int_0^{\pi} I_m \sin(\omega t)\, d(\omega t)\right\} = \frac{2 I_m}{\pi} \qquad (5)$$

$$\text{Ripple factor} = \frac{\text{r m s value of a c voltage across load}}{\text{d c voltage across load}} = \frac{V'_{rms}}{V_{dc}} = \frac{I'_{rms}}{I_{dc}} \qquad (6)$$

$$= \frac{\sqrt{I_{rms}^2 - I_{dc}^2}}{I_{dc}} = \sqrt{\frac{I_{rms}^2}{I_{dc}^2} - 1} = \sqrt{\frac{\left(\frac{I_m}{\sqrt{2}}\right)^2}{\left(\frac{2I_m}{\pi}\right)^2} - 1} = 0.4834 \qquad (7)$$

By measuring the r m s value of the a c voltage V'_{rms} and the d c voltage V_{dc} across the load resistance and using eqn.6 we can find out the ripple factor of the half wave rectifier. Repeat the experiment for different loads and in each case the ripple factor is calculated.

Now the filter capacitor C is connected parallel to the load and the a c and d c voltages are measured and the ripple factor with filter is calculated. This value of ripple factor will be much less than the ripple factor without filter. This is because the bypassing of a c through the filter capacitor. In this case also the experiment is repeated for different loads.

Rectification efficiency:

The dc power delivered to the load, $P_{dc} = I_{dc}^2 R_L$

Total input a c power, $P_{ac} = I_{r\,m\,s}^2 \left(r_d + R_L\right)$

Rectification efficiency, $\eta = \dfrac{P_{dc}}{P_{ac}} = \dfrac{I_{dc}^2 R_L}{I_{r\,m\,s}^2 \left(r_d + R_L\right)}$

Using eqns.3 and 5,
$$\eta = \frac{\left(\frac{2I_m}{\pi}\right)^2 R_L}{\left(\frac{I_m}{\sqrt{2}}\right)^2 \left(r_d + R_L\right)} = \frac{0.811}{\left(1 + \frac{r_d}{R_L}\right)} \qquad (8)$$

If $r_d << R_L$, $\eta = 81.1\%$

It means that under best conditions, only 81.1% of a c input power is converted into d c power. The rest remains as a c power in load.

- For full wave rectifier we need a centre tapped transformer.

Observation and tabulation

V volt	I mA

Dynamic resistance of the diode, r_d = ……… ohm

Load R_L ohm	Rectification efficiency η	Without filter			With filter		
		a c volt	d c volt	Ripple factor	a c volt	d c volt	Ripple factor
100							
200							
300							
400							
500							
600							
700							
800							
900							
1000							
2000							
3000							
4000							
5000							
6000							
7000							
8000							
9000							
10000							
∞							

Result

A full wave rectifier is constructed and their ripple factors without filter and with filter for various loads are determined. Also the rectification efficiencies for various loads are calculated.

*Components and Circuit testing

- A diode is good only if there is a low resistance reading in one direction (forward biasing) and high resistance in the other (reverse bias). So by using a multimeter as an ohmmeter you can check a diode.
- Potential drop across a sufficiently forward biased diode is 0.7 V for Si and 0.3 V for Ge.
- Because of impurities the measured value is slightly less than this.

Exp.No.3.2

Bridge Rectifier- study of ripple factor with and without filter

Aim: To construct a full wave bridge rectifier using identical four diodes and find out its rectification efficiency, ripple factor without filter and with filter.

Components and accessories required:

A step down transformer (with input 230 volt and output 6, 9 or 12 volt). Four identical diodes (any one of the series from 1N4001or DR25 series, resistances or a resistance box capacitor (electrolytic large value of 100 μF or 220 μF), voltmeter and an ammeter (or a multimeter).

Circuit, theory and procedure (same as full wave rectifier with two diodes except circuit)

Determination of dynamic resistance, r_d:

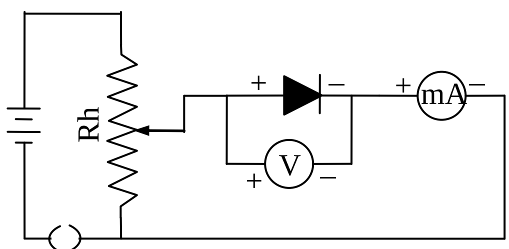


Fig.a

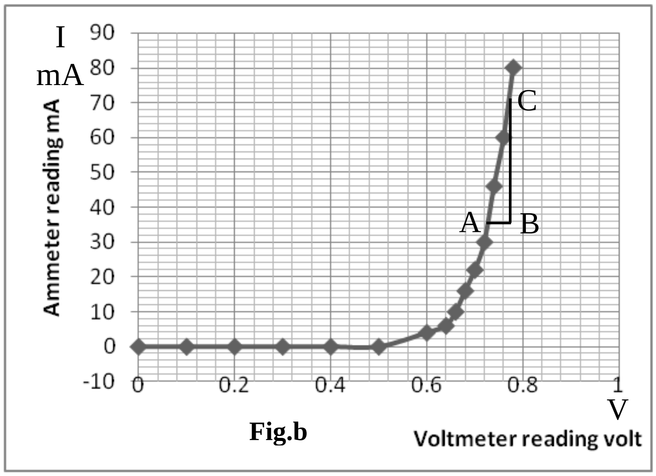


Fig.b

Connections are made as shown in fig.a. The rheostat is adjusted and the voltage is increased in steps and in each case the current in the milliammeter is noted. The graph between I and V, called the characteristics of the diode, is drawn. Calculate the dynamic resistance r_d at the straight line portion of the graph.

$$r_d = \frac{\Delta V}{\Delta I} = \frac{AB}{BC} \times 10^3 \ \Omega$$, where ΔI in milliamperes.

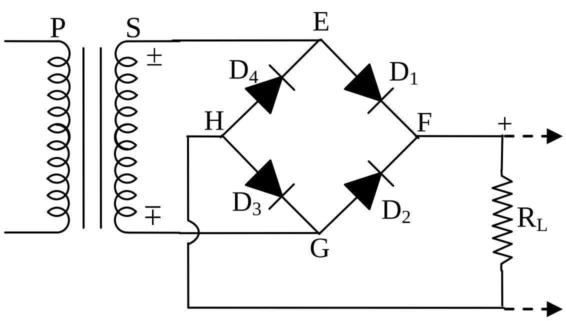


Fig.c: Full wave bridge rectifier without filter

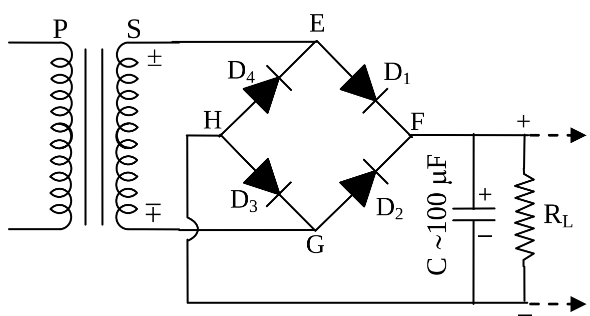


Fig.d: Full wave bridge rectifier with filter

Determination of ripple factor

To construct a full wave bridge rectifier without filter, connections are made as shown in fig.c. The primary P of the step down transformer is connected to 230 volt a c mains supply. The diode is connected to the secondary of the transformer. R_L is the load resistance across which the output voltage is measured. We can use a resistance box or carbon rod resistors as load. Using a multimeter both a c and d c voltages are measured.

Now connect a capacitor of the order of 100 or 220 μF parallel to the load R_L to bypass the ac voltage as shown in fig.d. The capacitor is called the filter capacitor and the resulting circuit is a full wave bridge rectifier with filter. Again a c and d c voltages are measured. The rectification efficiency and ripple factor are calculated using the equations,

$$\text{Rectification efficiency,} \quad \eta = \frac{P_{dc}}{P_{ac}} = \frac{I_{dc}^2 R_L}{I_{r\,m\,s}^2\left(r_d + R_L\right)}$$

$$= \frac{\left(\frac{2I_m}{\pi}\right)^2 R_L}{\left(\frac{I_m}{\sqrt{2}}\right)^2\left(r_d + R_L\right)} = \frac{0.811}{\left(1+\frac{r_d}{R_L}\right)}$$

$$\text{Ripple factor} = \frac{\text{r m s value of a c voltage across load}}{\text{d c voltage across load}} = \frac{V'_{rms}}{V_{dc}} = \frac{I'_{rms}}{I_{dc}}$$

$$= \frac{\sqrt{I_{rms}^2 - I_{dc}^2}}{I_{dc}} = \sqrt{\frac{I_{rms}^2}{I_{dc}^2} - 1} = \sqrt{\frac{\left(\frac{I_m}{\sqrt{2}}\right)^2}{\left(\frac{2I_m}{\pi}\right)^2} - 1} = 0.4834$$

- For full wave Bridge rectifier we don't need a centre tapped transformer. Usually we use any two of the three output terminals of a centre taped transformer with input 230 and output 6-0-6, 9-0-9 or 12-0-12 volt.

Observation and tabulation

V volt	I mA

Dynamic resistance of the diode, r_d = ……… ohm

Load R_L ohm	Rectification efficiency η	Without filter			With filter		
		a c volt	d c volt	Ripple factor	a c volt	d c volt	Ripple factor
100							
200							
300							
400							
500							
600							
700							
800							
900							
1000							
2000							
3000							
4000							
5000							
6000							
7000							
8000							
9000							
10000							
∞							

Result

A full wave bridge rectifier is constructed and their ripple factors without filter and with filter for various loads are determined. Also the rectification efficiencies for various loads are calculated.

- A diode is good only if there is a low resistance reading in one direction (forward biasing) and high resistance in the other (reverse bias). So by using a multimeter as an ohmmeter you can check a diode.
- Potential difference across a sufficiently forward biased diode is 0.7 for Si and 0.3 V for Ge.

Advantage: The output is nearly two times than the ordinary full wave rectifier.

Exp.No.3.3

Clippers and clampers using diodes

Aim: To design and construct clipping and clamping circuits using diodes.

Components and accessories required: Signal generator, diodes (any one of the series from 1N4001or DR25 series, d c source, resistors, C R O, board and other accessories for soldering.

Circuit, theory and procedure of clipping circuits: A clipper is a device which limits, remove or prevents some portion of the wave form (input signal voltage) above or below a certain level. In other words, the circuit which limits positive or negative amplitude, or both is called chipping circuit. By using diode clipper it is possible to clip the wave at any level by providing an additional voltage source. The basic principle used in diode clippers is that, the diode conducts only when it is forward biased.

In *positive clippers* the positive part of the input signal is clipped off. Thus the output consists of only the negative part of the input signal. In *negative clippers* the negative part of the signal is clipped off. We study the clipper circuits of the following types.

1. Series positive clipper
2. Series negative clipper
3. Shunt or parallel positive clipper
4. Shunt or parallel negative clipper
5. Clipping circuits (both series and parallel and positive and negative) with positive and negative bias.
6. Dual combination diode clipper.

By interchanging the diode and resistor of the series clipper we get the parallel clipper.

Series positive clipper: In a series positive clipper, a diode is connected in series with the output, as shown in fig.a. An a c signal of voltage 5 or 10 volt and frequency of the order of 1000 Hz is given to

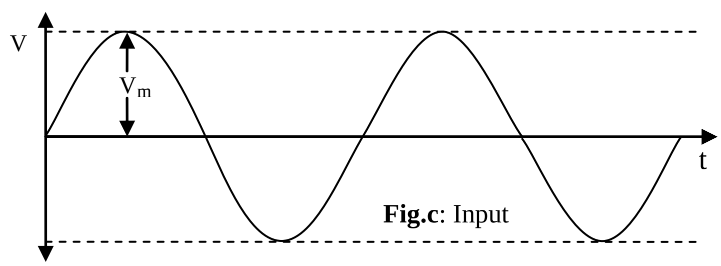


Fig.c: Input

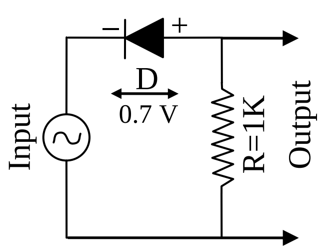


Fig.a: Series positive clipper

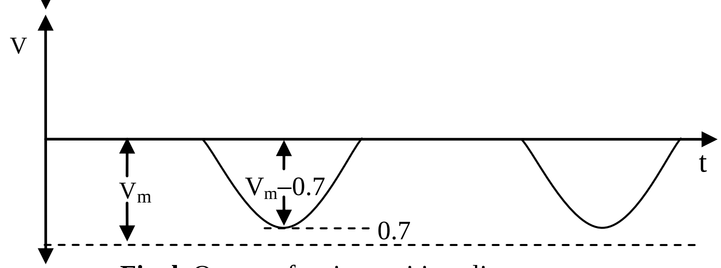


Fig.d: Output of series positive clipper

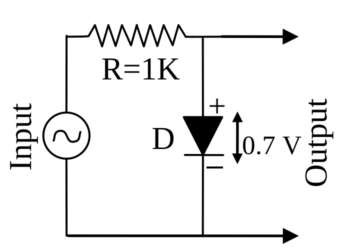


Fig.b: Parallel positive clipper

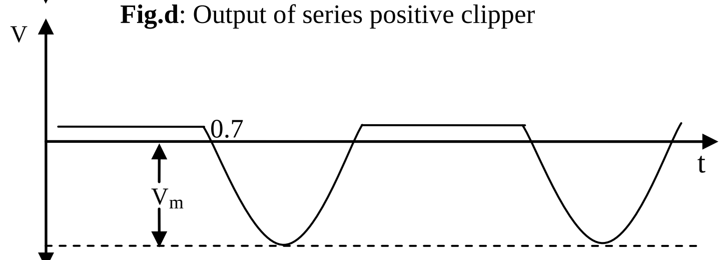


Fig.e: Output of parallel positive clipper.

the input terminals. During the positive half cycle of the input voltage, the diode is reverse biased. Hence the diode acts as an open switch. Therefore all the applied voltage drops across the diode and none across the resistor (since the current through R is zero). As a result of this there is no output voltage during the positive half cycle of the input voltage.

During the negative half cycle of the input voltage the diode is forward biased. Hence the diode acts as a closed switch. Thus, there is a voltage drop across diode during the negative half cycle of the input voltage. All the input voltage is dropped across the resistor as shown in the output wave form. A positive clipper removes or clips off the positive half completely. The output can be studied by a C R O. (*Notice the shortage of the order of 0.7 V in V_m for silicon diode for series clippers.*

Shunt or parallel positive clipper: A parallel clipper circuit uses the same diode theory and circuit operation. A resistor and a diode are connected in series with the input signal and the output signal is developed across the diode. The output is in parallel with the diode. Hence the circuit name *parallel clipper*. The parallel clipper can limit either the positive or negative alternation of the input signal. Figures b and e show the circuit of a shunt positive clipper and its output. In this circuit, the diode acts as a closed switch when the input voltage is positive (i.e. $V_i > 0$ and as an open switch when the input voltage is negative (i.e. $V_i < 0$). The output waveform is almost the same as that of a series positive clipper with a slight difference that there is no shortage of V_m but there is a d c level and a small a c part corresponds to 0.7 V for silicon diodes. The waveform can be studied by a C R O.

Series and parallel negative clippers: By reversing the polarities of the diode for positive clippers we get the negative clippers. In the case of negative clippers the negative part of the a c is clipped off. Figures a and d show the circuit and output of a series negative clipper and figures b and e that for parallel negative clippers.

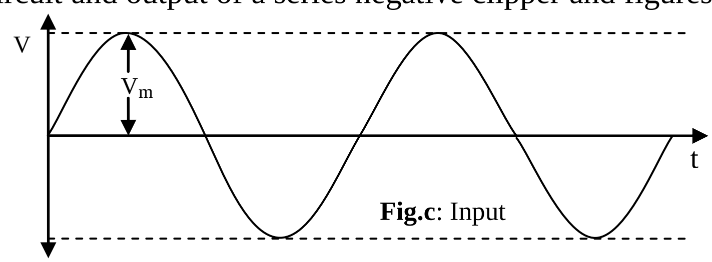


Fig.c: Input

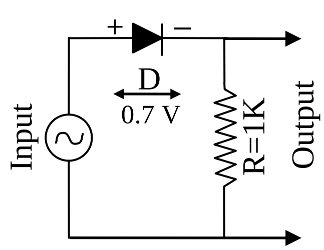


Fig.a: Series negative clipper

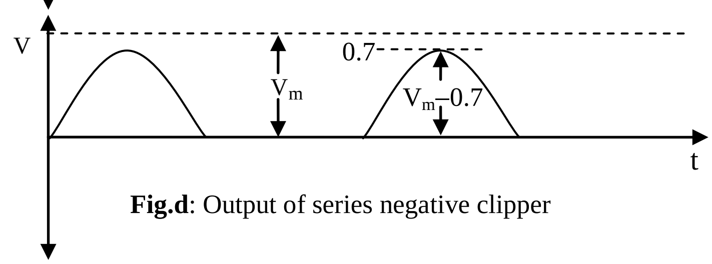


Fig.d: Output of series negative clipper

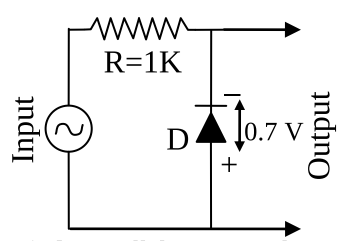


Fig.b: Parallel negative clipper

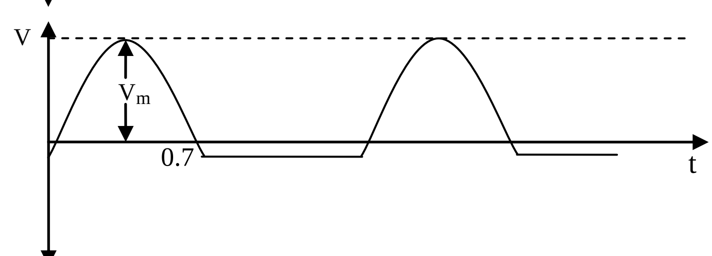


Fig.e: Output of parallel negative clipper.

Biased clippers: Sometimes it is required to remove a desired portion of the positive or negative half cycle of the signal voltage (input signal). For this purpose the biased clipper is used.

Series-positive clipper with bias: Figures below show the circuit of a biased series positive clipper. Do the connections as shown in fig.a and b and the output is studied by a C R O.

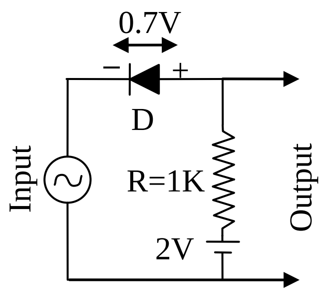


Fig.a: Series positive clipper-positive biased

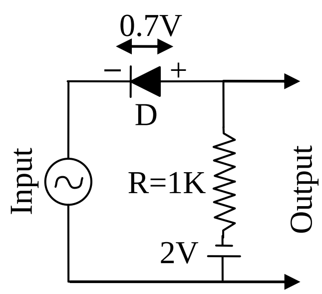


Fig.b: Series positive clipper-negative biased

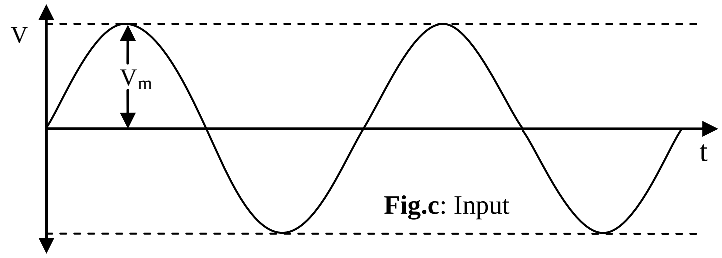


Fig.c: Input

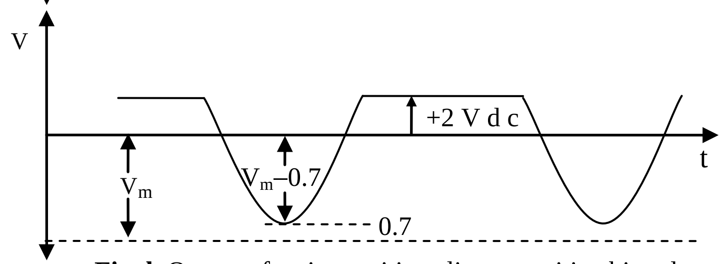


Fig.d: Output of series positive clipper-positive biased

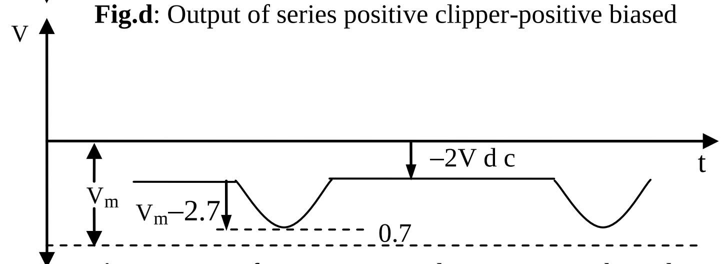


Fig.e: Output of series positive clipper-negative biased.

Series negative clipper with bias: The figures below give the series negative clipper with positive and negative bias. Do the connections as shown in fig.a and b and the output is studied by a C R O.

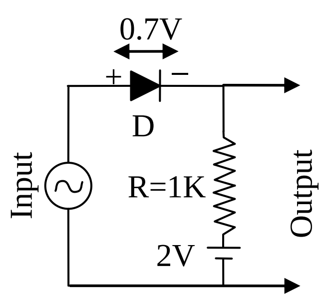


Fig.a: Series negative clipper-positive biased

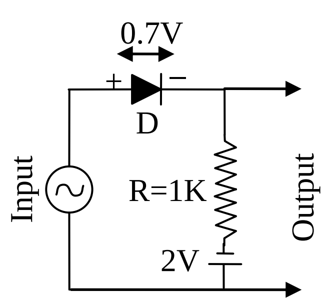


Fig.b: Series negative clipper-negative biased

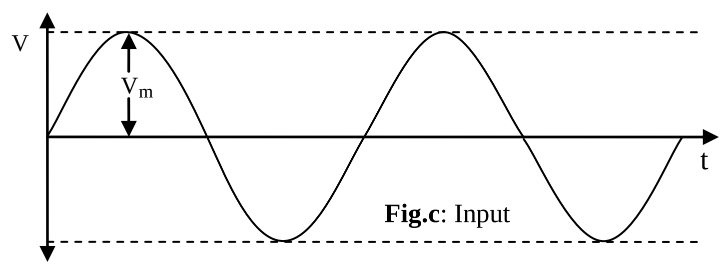


Fig.c: Input

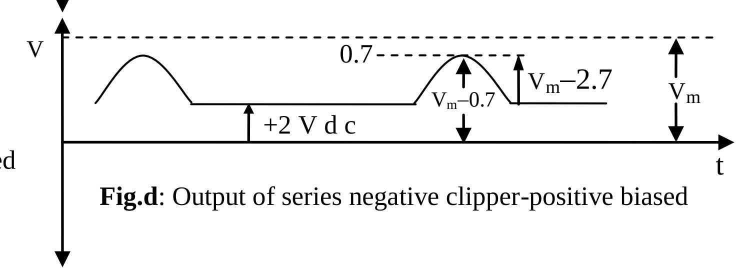


Fig.d: Output of series negative clipper-positive biased

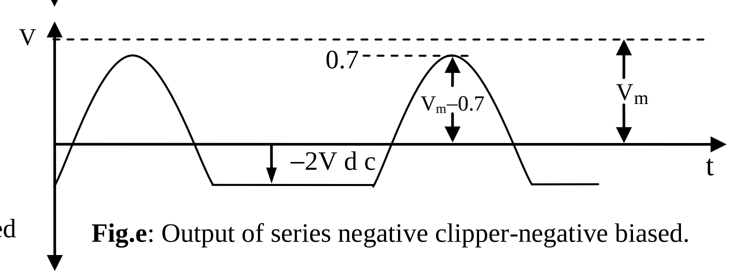


Fig.e: Output of series negative clipper-negative biased.

Parallel positive clipper with bias: Figures below show the circuit of a biased parallel positive clipper. Do the connections as shown in fig.a and b and the output is studied by a C R O.

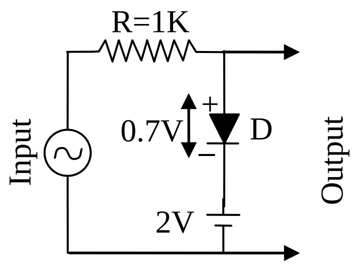


Fig.a: Parallel positive clipper-positive biased

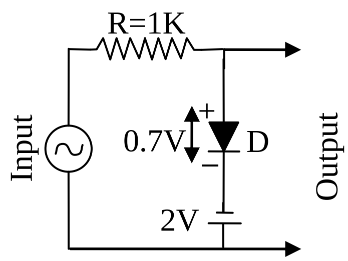


Fig.b: Parallel positive clipper-negative biased

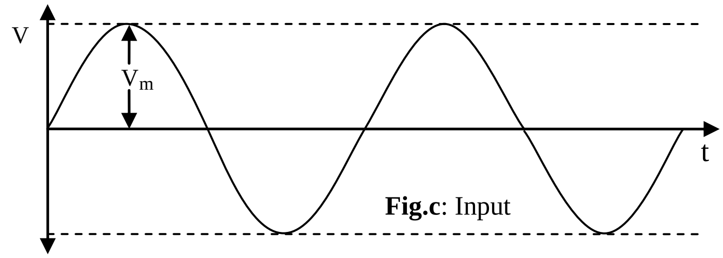


Fig.c: Input

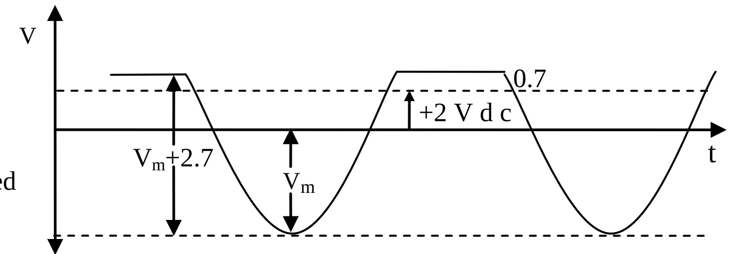


Fig.d: Output of parallel positive clipper-positive biased

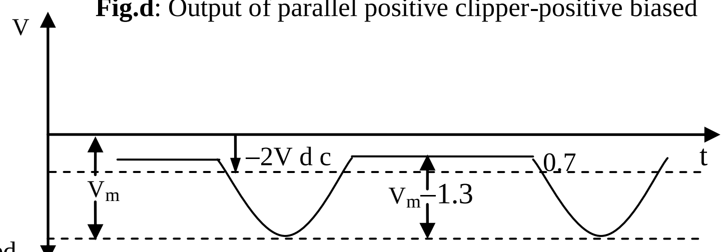


Fig.e: Output of parallel positive clipper-negative biased.

Parallel negative clipper with bias: The figures below give the parallel negative clipper with positive and negative bias. Do the connections as shown in fig.a and b and the output is studied by a C R O.

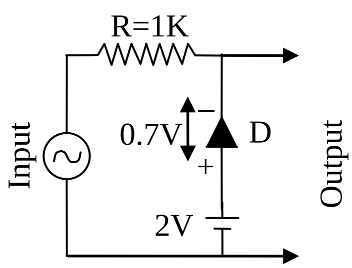


Fig.a: Parallel negative clipper-positive biased

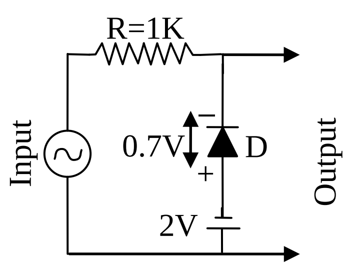


Fig.b: Parallel negative clipper-negative biased

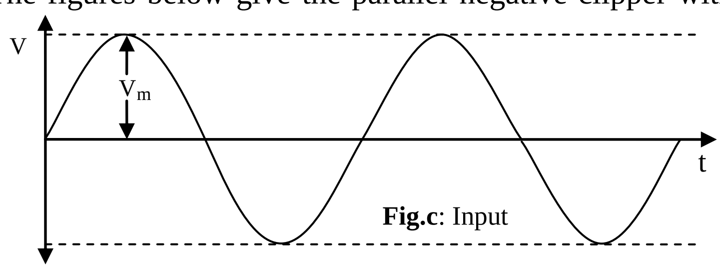


Fig.c: Input

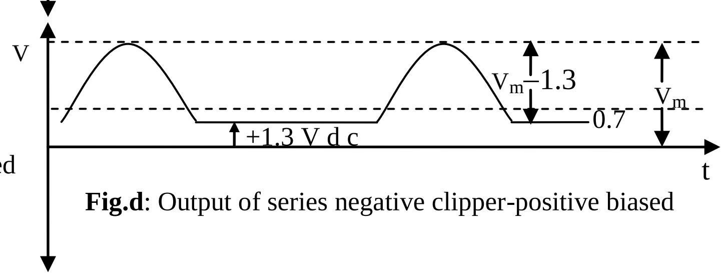


Fig.d: Output of series negative clipper-positive biased

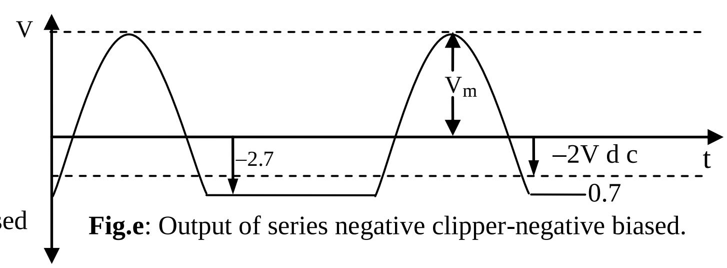


Fig.e: Output of series negative clipper-negative biased.

Dual combination diode clipper: The combination of a biased positive clipper and a biased negative clipper is called combination or dual diode clipper. Such a clipper circuit can clip at both two independent levels depending upon the bias voltages. Fig.a below shows the circuit of a dual (combination) clipper.

Let us suppose a sinusoidal a c voltage is applied at the input terminals of the circuit. Then during the positive half cycle, the diode D_2 is forward biased, while diode D_1 is reverse biased. Therefore the diode D_2 conducts and acts as a short circuit. On the other hand, diode D_1 acts as an open circuit. However, the value of output voltage cannot exceed the bias voltage level plus the diode drop as shown in fig.c.

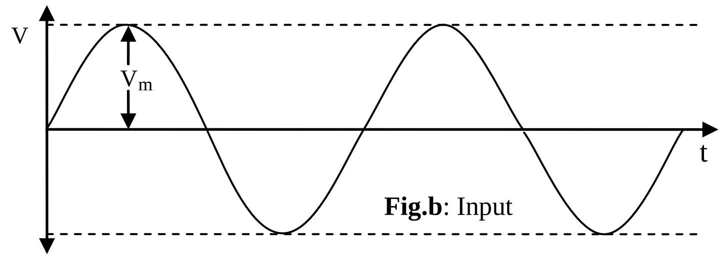


Fig.b: Input

Similarly during the negative input half cycle the diode D_1 acts as a short circuit while the diode D_2 as an open circuit. However the value of output voltage cannot exceed the bias voltage level. It may be noted that the clipping levels of the circuit be varied by changing the values of bias voltages V_{B1} and V_{B2}. If the values of V_{B1} and V_{B2} are equal, the circuit will clip both the positive and negative half cycles at the same voltage level. Such a circuit is known as a symmetrical clipper. Do the connections as shown in fig.a and the output is studied by a C R O.

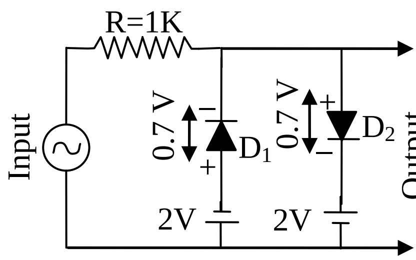


Fig.a: Dual combination diode clipper

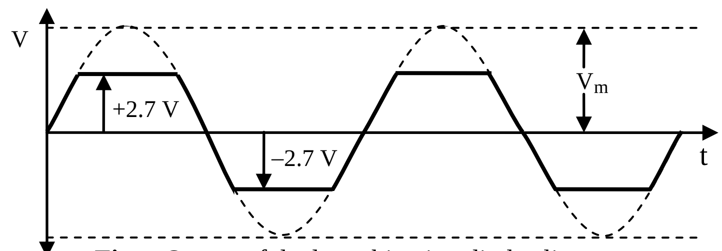


Fig.c: Output of dual combination diode clipper

Circuit, theory and procedure of clampers: A **clamper** is an electronic circuit that fixes either the positive or the negative peak excursions of a signal to a defined value by shifting its DC value. The clamper does not restrict the peak-to-peak excursion of the signal, it moves the whole signal up or down so as to place the peaks at the reference level. Or, simply clamping is a process of introducing a d c level into an a c signal. A **diode clamp** (a simple, common type) consists of a diode, which conducts electric current in only one direction and prevents the signal exceeding the reference value; and a capacitor which provides a d c offset from the stored charge. The capacitor forms a time constant with the resistor load which determines the range of frequencies over which the clamper will be effective.

A clamping circuit (also known as a clamper) will bind the upper or lower extreme of a waveform to a fixed d c voltage level. These circuits are also known as d c voltage restorers. Clampers can be constructed in both positive and negative polarities. When unbiased, clamping circuits will fix the voltage lower limit (or upper limit, in the case of negative clampers) to 0 Volts. These circuits clamp a peak of a waveform to a specific d c level compared with a capacitively coupled signal which swings about its average d c level.

Types: Clamp circuits are categorized by their operation; negative or positive, and biased or unbiased. A positive clamp circuit outputs a purely positive waveform from an input signal; it offsets the input signal so that all of the waveform is greater than 0 V. A negative clamp is the opposite of this. It outputs a purely negative waveform from an input signal. A bias voltage between the diode and ground offsets the output voltage by that amount. For example, an input signal of peak value 5 V (V_{IN} = 5 V) is applied to a positive clamp with a bias of 3 V (V_{BIAS} = 3 V), the peak output voltage will be

$$V_{OUT} = 2V_{IN} + V_{BIAS}$$
$$V_{OUT} = 2 * 5\text{ V} + 3\text{ V}$$
$$V_{OUT} = 13\text{ V}$$

Now we construct the four types of clamping circuits with the minimum requirement of the elements, a diode, a capacitor and a resistor. They are,

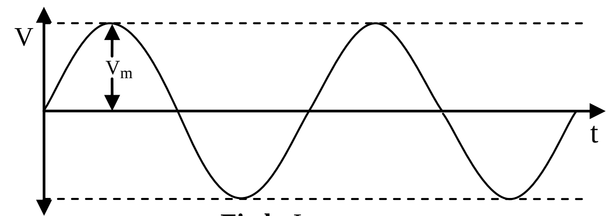


Fig.b: Input

(a) Positive unbiased clamper

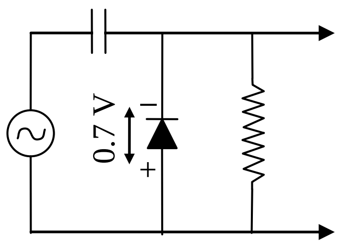


Fig.a: Positive unbiased clamper

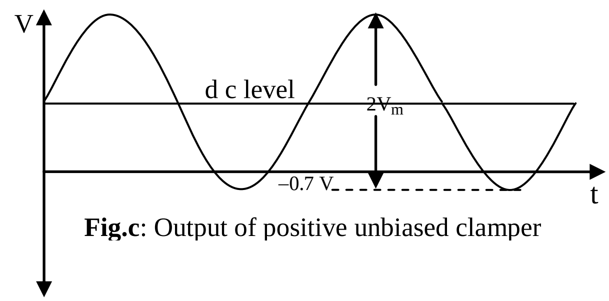


Fig.c: Output of positive unbiased clamper

(b) Negative unbiased clamper

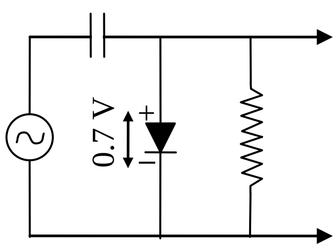


Fig.a: Negative unbiased clamper

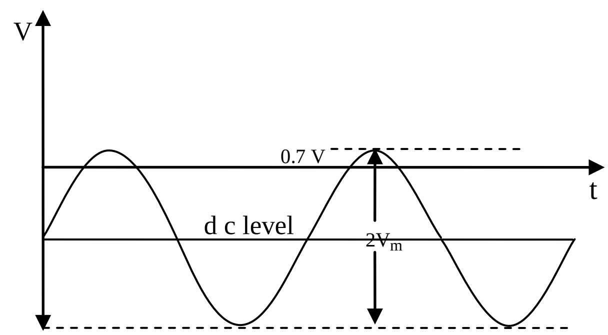


Fig.c: Output of negative unbiased clamper

(c) Positive biased clamper

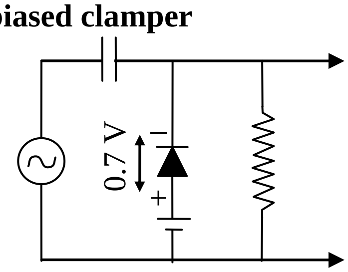


Fig.a: Positive biased clamper

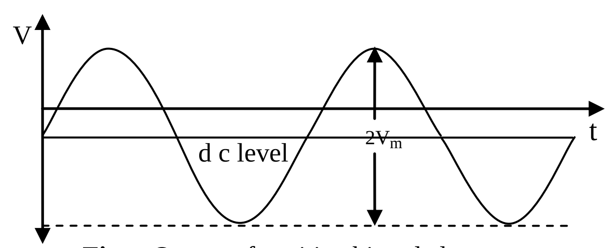


Fig.c: Output of positive biased clamper

(d) Negative biased clamper

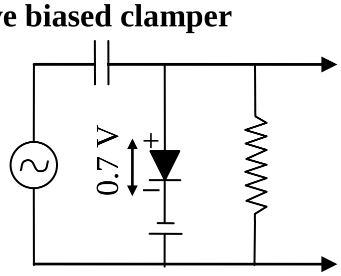


Fig.a: Negative biased clamper

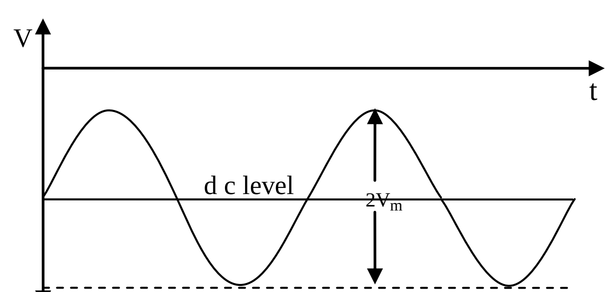


Fig.c: Output of negative biased clamper

See that there is a shifting up of the entire signal corresponding to the d c bias voltage for the positive biased clamper, whereas for negative biased clamper there is a corresponding shifting down. In all the cases of clampers there is no change in the wave shape of the signal and there is no change in the peak to peak voltage of the signal. This shifting can be understood by observing the either the upper side or lower side of the signal.

- The load of the parallel clippers is the load of the measuring instrument.
- It is convenient to choose an a c source of 5 V peak (V_m = 5 V) and frequency ~1000 Hz. For d c biasing purpose use 2 V lead accumulators. Also use a resistor of 1 K.
- Put C R O in D C mode.
- Instead of signal generator we can use a c mains step down transformers or the a c output of power supplies, if available. (This may avoid any large d c shift in the signal generator.
- There is a drop of 0.7 V across the conducting silicon diode. Thus for series clippers the amplitude of a c part is reduced from V_m to V_m – 0.7 V and for parallel clippers the amplitude of a c part remains as V_m.
- For clampers (33μF, 100 μF, 470 μF etc.). Put C R O in D C mode. Resistors 1K, 10 K, 20 K, etc. such that the time constant RC is large enough to ensure that capacitor remains almost fully charged during the time period of the signal. Use input signal of 5V amplitude and frequency 1000 Hz or less.
- When you give the output to C R O, there may be a shifting of the ground level of C R O. To find the new ground level, press the ground button of C R O channel used and find the corresponding line. Then bring the other channel line to coincide with it. Then release the ground button and press the d c button of the concerned channel again.

Result

The different types of clippers and clampers are constructed and the corresponding outputs are studied by a C R O.

Exp.No.3.4

Voltage multiplier using diodes

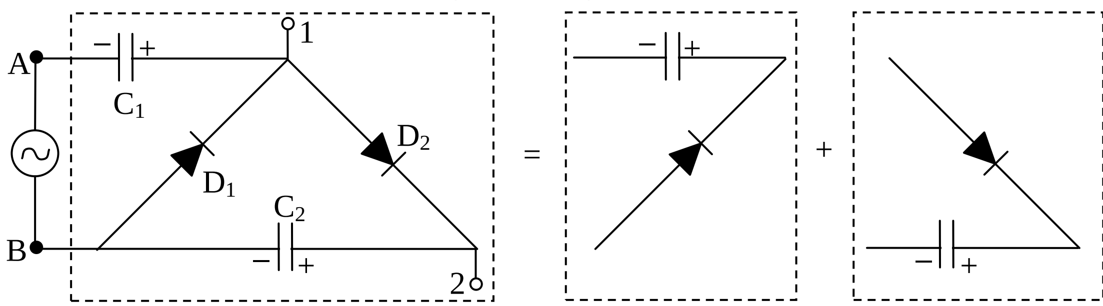
Aim: To construct a voltage multiplier using diodes.

Components and accessories required: Step down transformer or a signal generator, diodes (any one of the series from 1N4001or DR25 series, resistors, capacitors, multimeter, board and other accessories for soldering.

Circuit, theory and procedure: A **voltage multiplier circuit** is a special type of rectifier circuit which produces a d c output voltage which is many times greater than it's a c input voltage. Voltage multiplier circuits are constructed from series combinations of rectifier diodes and capacitors that give a d c output equal to some multiple of the peak voltage value of the a c input voltage. By adding a second diode and capacitor to the output of the simple half-wave rectifier, we can increase its output voltage. The most commonly used type of voltage multiplier circuit is the **Half Wave Series Multiplier** also known as a "cascade voltage doubler".

The high voltage that can be produced by a *voltage multiplier circuit* is, in theory, unlimited. But due to their relatively poor voltage regulation and low current capability most multiplier circuits produce voltages in the range of about 10 kV to 30 kV with a low current of less than ten milli-amperes. Our aim is to construct a voltage quadrupler that can produce a d c voltage four times greater than the a c peak voltage.

To understand the theory of multipliers we consider the doubler and tripler separately. The doubler can be considered as the combination of two half sections. By adding the left half section to the right side or the right half section to the left side of the doubler we get a tripler. By adding full sections of the doubler in series we get a voltage multiplier.



Full section Half section Half section

Fig.a: A doubler is a combination two half sections.
Change of polarity of diode is possible with a change in the polarity of capacitors.

Working of a doubler: During the half cycle at which B is positive diode D_1 is forward biased and D_2 is reverse biased. Hence D_1 conducts and the capacitor C_1 gets charged to the a c peak voltage V_P. During the other half cycle D_1 is reverse biased and D_2 is forward biased. Hence D_2 conducts. For D_2, the charged capacitor C_1 acts as a cell of voltage V_p. Thus the total voltage across it is the sum of the input a c voltage and the voltage V_P of capacitor C_2. Hence the capacitor C_2 gets charged to a voltage $2V_P$. This process repeats in the case of tripler and multiplier. A diode and a capacitor together, thus, increase the d c voltage in steps of V_P.

With respect to terminals A and 1 we get a d c voltage V_P and with respect to terminals B and 2 we get the d c voltage $2V_P$.

A tripler is a combination of a doubler and a half section as shown in fig.b

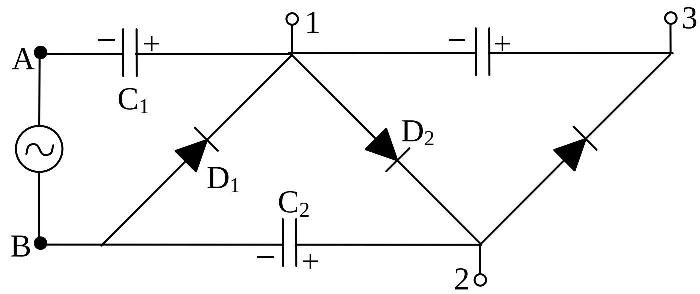


Fig.b: A tripler, which is a combination of a doubler and a half section.

A general multiplier is a combination of large number of doublers as shown in fig.c.

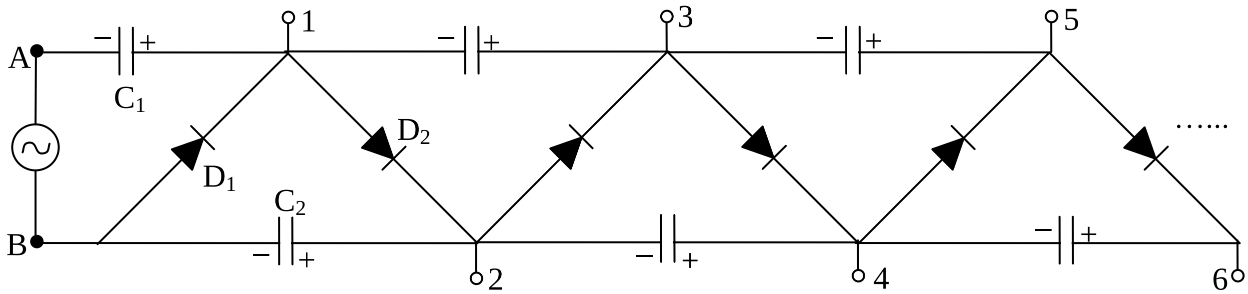


Fig.c: A general multiplier, which is a combination of a number of doublers

If A is the common terminal, then the d c voltages between A and 1, A and 3, A and 5, ……. are V_P, $3V_P$, $5V_P$, ………. and if B is the common terminal, then the d c voltages between B and 2, B and 4, B and 6, ……. are $2V_P$, $4V_P$, $6V_P$, ……….

As a multiplier we construct only a quadrupler as shown in fig.d and measure the voltages at various levels.

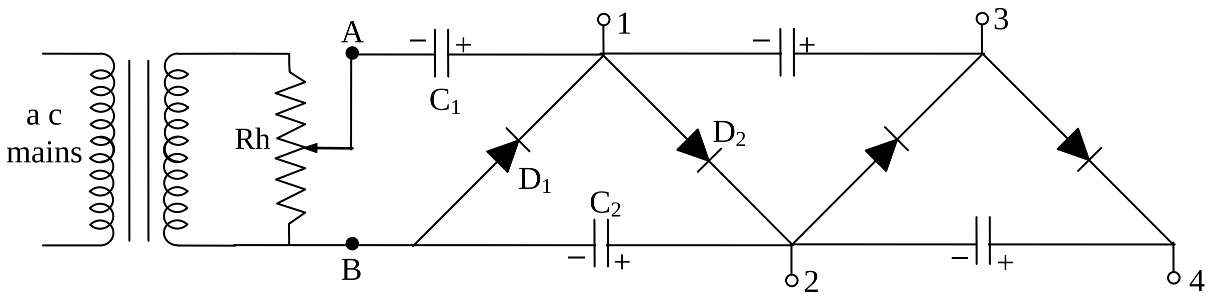


Fig.d: A quadrupler

- The output voltages of multipliers are d c voltages.
- Since at each half sections the d c voltage increases in steps of the a c peak voltage V_p, the capacitors used at various stages must withstand the potential difference at that stages. Expecting a c peak voltage as 10 V, use capacitors of withstanding voltage 63 V. If the input is large the capacitor withstanding voltage must be larger enough accordingly.
- See that the diode polarity and capacitor polarity are opposite at any half section.
- Change of polarity of diode is possible with a corresponding change in the polarity of capacitors.

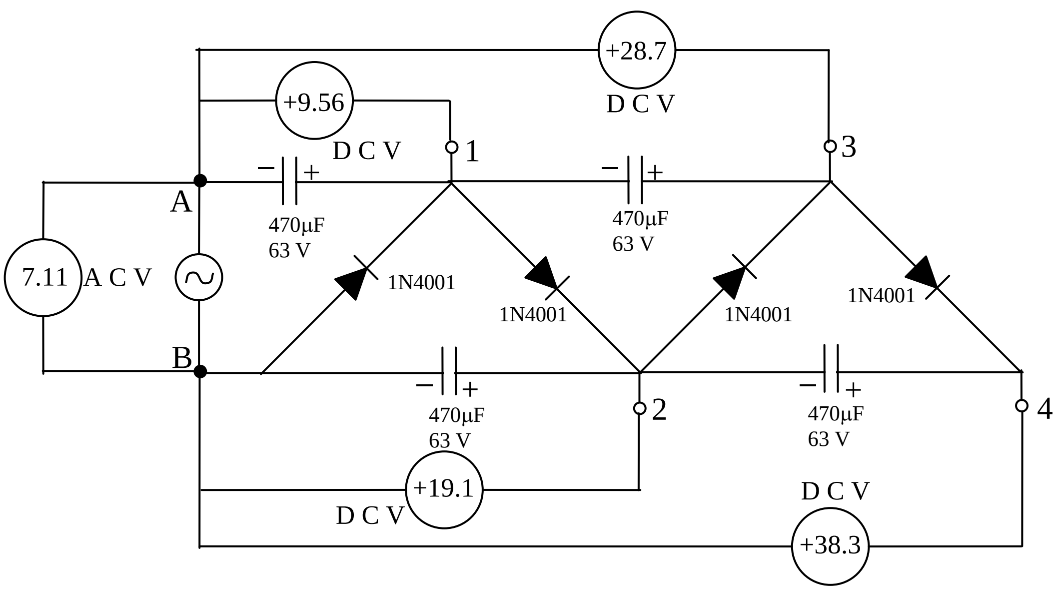
Observation and tabulation

Trial No.	a c voltage between A and B V_{rms} volt	Peak voltage of a c input $V_P = V_{rms} \times \sqrt{2}$ volt	Measured d c voltages			
			Between A and 1 V_P volt	Between B and 2 $2V_P$ volt	Between A and 3 $3V_P$ volt	Between B and 4 $4V_P$ volt
1						
2						
3						
4						
5						

Result

A voltage quadrupler is constructed and the voltages at various levels, doubler, tripler and quadrupler are measured and tabulated.

***An example circuit**

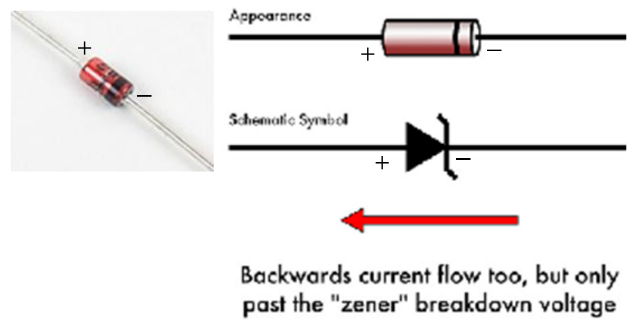


Exp.No.3.5

Construction of a voltage regulator using Zener diode after finding Zener voltage

Aim: To find the Zener voltage of a Zener diode by drawing its reverse bias characteristics and then to construct a voltage regulator using the zener diode.

Components and accessories required: The Zener diode, two ordinary diodes (1N4001), a 12-0-12 step down transformer, voltmeter, milli-ammeter, resistors, electrolytic capacitor (100 or 470 μF), etc.



Normally, zener diode woks under reverse bias condition, i.e. positive side (anode) is connected to the negative terminal and the negative side (cathode) to positive terminal of the battery.

Circuit, theory and procedure

To draw the characteristics and to find the Zener voltage:

The connections are made as shown in the fig.a. Initially the rheostat is adjusted for zero potential difference and zero current. The voltage across the Zener diode is increased in steps from zero and in each case the milli-ammeter reading is noted. A graph is drawn between V and I as shown in the fig.b. The Zener voltage is the break down voltage above which the current increases sharply. From the graph we can find out the zener voltage V_z.

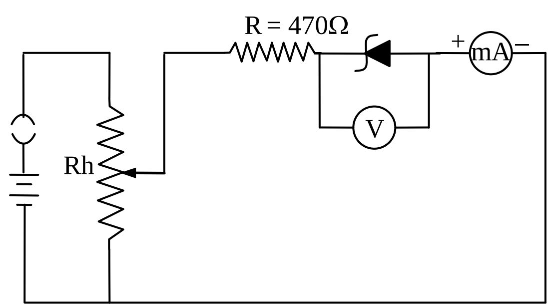


Fig.a

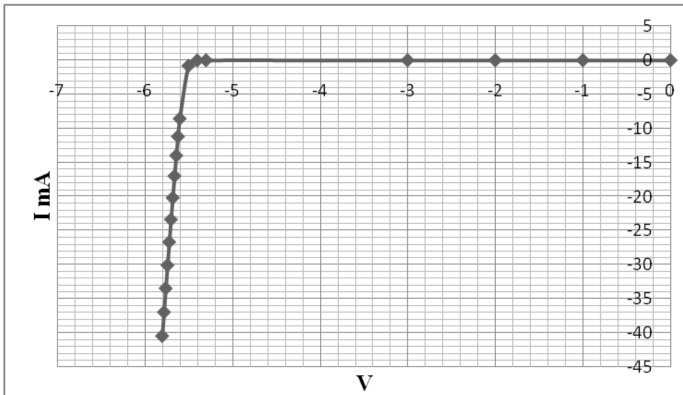


Fig.b: Characteristics of 1N4734A- 5.6 V- 45 mA Zener diode

Construction of voltage regulator

Construction of the voltage regulator consists of three steps.

1. *Unregulated power supply*: This is a full wave rectifier using two diodes described in Exp.No.3.1.
2. *Calculation of R_s*: We now calculate the resistance needed to fix the total current to a required value I, say 20 mA, as follows.

$$V_i = V_z + IR_s$$

$$R_s = \frac{V_i - V_z}{I} \quad (1)$$

3. *Construction of voltage regulator and study of its load regulation*: The voltage regulator for a total current I is constructed as shown in fig.c. The voltage across the load is measured. This is repeated for different loads. Find the voltage V_{NL} for no load (infinite resistance - open circuit voltage at the load). Calculate the percentage of voltage regulation for each load using the equation,

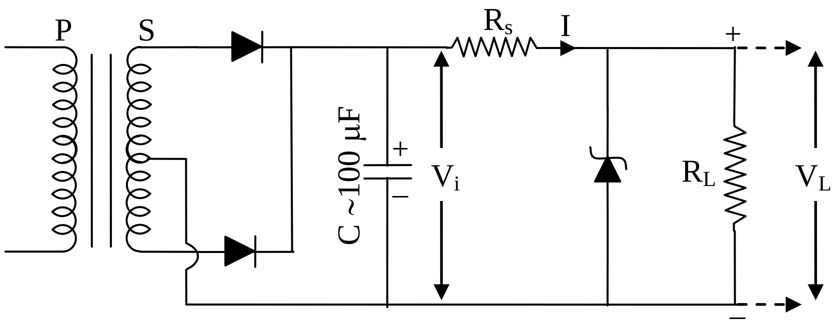


Fig.c: Voltage regulator using Zener diode

$$\text{Percentage of voltage regulation} = \left(\frac{V_{NL} - V_{FL}}{V_{FL}}\right) \times 100\,\% \quad (2)$$

Percentage of voltage regulation is also called percentage of load regulation.

Observation and tabulation

To find the Zener voltage

Trial No.	Voltage across the Zener diode, V volt	Current through the Zener diode, I mA

Zener voltage from graph, V_z = ……… Volt

$$R_s = \frac{V_i - V_z}{I} = \text{……… ohm}$$

To find load regulation

Output voltage with no load, V_{NL} = ……… volt

Load R_L ohm	Output voltage V_{FL} volt	Percentage of voltage regulation $\left(\frac{V_{NL} - V_{FL}}{V_{FL}}\right)\times 100$
100		
200		
300		
400		
500		
600		
700		
800		
900		
1000		
2000		
3000		
4000		
5000		
6000		
7000		
8000		
9000		
10000		

- Use a resistance R to protect the Zener diode from high currents. In order the presence of R not to affect the characteristics, the voltmeter must be connected parallel to the Zener diode. If the current is low reduce the value of R.
- Digital voltmeter is convenient to use for the measurement of small voltage variations.
- No load means zero load current, i.e. infinite R_L.
- Full load means the load current with the given R_L.
- Care must be taken not to increase the current through the diode greater than its limiting value for the given Zener diode. For example, for the Zener diode 1N4734A the current rating is 45 mA.

Result

A voltage regulator using Zener diode is constructed and its percentage of voltage regulation for various loads is determined.

Exp.No. 3.6

Characteristics of a common base transistor

Aim: To study the characteristics of the given transistor in common base configuration.

Components and accessories required: Transistor 2N3055 or B C 107, power supplies, milli-ammeters and voltmeters.

Circuit, theory and procedure: A bipolar transistor is a three terminal device. It can be connected in the circuit in three ways with one of the terminals is common to both input side and the output side. Hence there are (a) common base configuration, in which the base is common, (b) common emitter configuration, in which the emitter is common and (c) common collector configuration, in which the collector is common. For each of these configurations there are two characteristics for the transistor, the *input characteristics* and the *output characteristics*. Input characteristics is the graphical representation of the variation of the input current with the input voltage keeping output voltage constant and the output characteristics is the graphical representation of the variation of output current with the output voltage keeping the input current constant.

In this experiment we study the common base configuration, in which the base is common to both input and output. Connections are made as shown in the figure. Since the transistor is an n p n transistor the emitter side is negative biased and the collector is biased at a large positive potential. (For a p n p transistor reverse the polarities of biasing).

Input characteristics: To draw the input characteristics set the output voltage (V_{CB}) to a fixed value, say 5 volt. Then increase the base-emitter voltage V_{BE} (input voltage) in steps of 0.2 V and in each case the milli-ammeter reading I_E is noted. Draw a graph between V_{BE} and I_E. A sample graph for the transistor

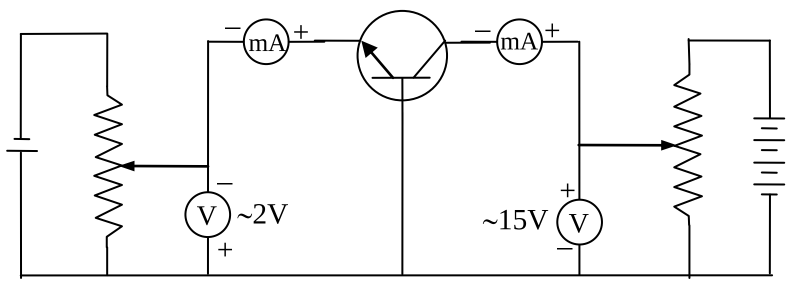


Fig.a

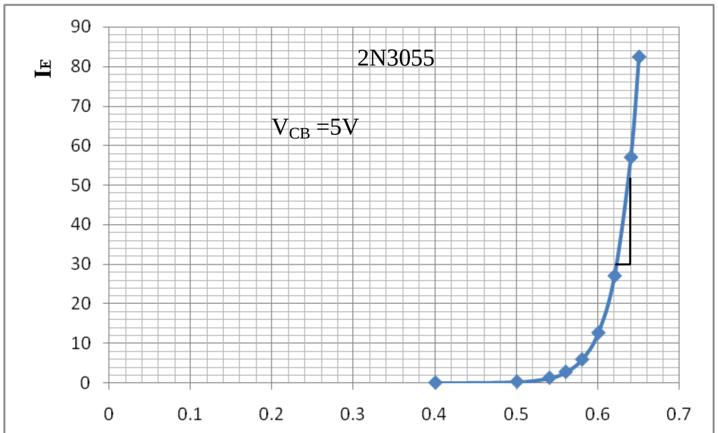


Fig.b

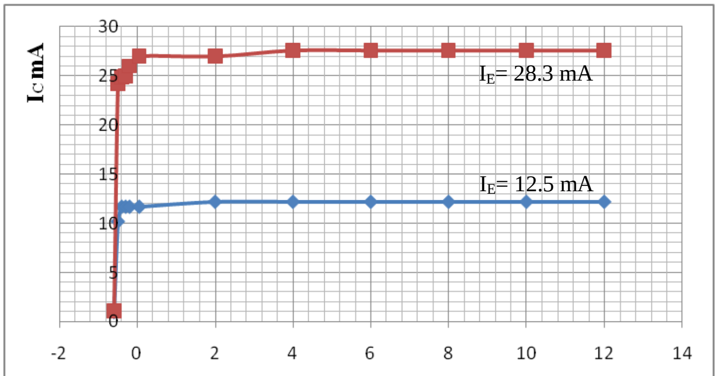


Fig.c

2N3055 for V_{CB} = 5V is given in fig.b. Repeat the entire experiment for other values of V_{CB} and draw all the graphs on the same graph paper. Thus we get the input characteristics of the given transistor for common base configuration.

Output characteristics: To draw the output characteristics set the input current I_E to a fixed value, say 12.5 mA. Then increase the output voltage V_{CB} in steps from zero to 10 or 12 volt and in each case the collector current I_C is noted. Now V_{CB} is reduced to less than zero, then I_E is increased a value a little higher than before and I_C is noted for different negative V_{CB}, till I_C is reduced to zero (V_{CB} = –0.1, –0.2, –0.3, ….). A graph is drawn between V_{CB} and I_C as shown in fig.c. Repeat the entire experiment for different values of I_E and all the graphs are plotted on the same graph paper. A typical family of curves is shown in the fig.c. Thus we get the output characteristics of the given transistor for common base configuration.

The four h-parameters (two forward parameters and two reverse parameters) of the transistor circuit in common base configuration are defined as,

Forward parameters

Input impedance, $$h_{ib} = r_e = \left(\frac{\Delta V_{EB}}{\Delta I_E}\right)_{V_{CB}} \tag{1}$$

Forward current gain, $$h_{fb} = \alpha = \left(\frac{\Delta I_C}{\Delta I_E}\right)_{V_{CB}} \tag{2}$$

Reverse parameters

Reverse voltage gain $$h_{rb} = \left(\frac{\Delta V_{EB}}{\Delta V_{CB}}\right)_{I_E} \tag{3}$$

Output admittance, $$h_{ob} = \left(\frac{\Delta I_C}{\Delta V_{CB}}\right)_{I_E} \tag{4}$$

- Check that the V_{CE} is equal to half of V_{CC} of a properly biased transistor. It is so, the transistor is operating and the circuit is well designed.
- If V_{CE} = 0, check whether the transistor is shorted out, disconnected from V_{CC} or operating in saturation.
- If V_{CE} = V_{CC}, check whether the transistor is open circuited, operating in cut-off or having all resistors in series with V_{CE} shorted.
- The rheostats used should have large value of order of 200 ohm for power transistors. You can also use variable resistance instead of rheostat. The resistance must be large enough to protect the transistor from high currents. The resistance of the order of 1K or 2K.
- If low power transistors are used voltages and currents are much lower.

Observation and tabulation

To draw the input characteristics

V_{EB} volt	V_{CB} = 1 volt	V_{CB} = 4 volt	V_{CB} = 7 volt	V_{CB} = 10 volt
	I_E mA	I_E mA	I_E mA	I_E mA
0.1				
0.2				
0.3				
0.4				
0.50				
0.52				
0.54				
0.56				
0.58				
0.60				
0.62				

To draw the output characteristics

V_{EB} volt	I_E = 10 mA	I_E = 20 mA	I_E = 30 mA	I_E = 40 mA
	I_C mA	I_C mA	I_C mA	I_C mA
–0.7				
–0.6				
–0.5				
–0.4				
–0.3				
–0.2				
–0.1				
1				
2				
3				
4				
5				
6				
7				
8				
9				
10				
11				
12				

From input characteristics

For constant V_{CB}:

$$\Delta V_{BE} = \ldots\ldots ;\ \Delta I_E = \ldots\ldots \ ;\ h_{ib} = r_e = \left(\frac{\Delta V_{BE}}{\Delta I_E}\right)_{V_{CB}} = \ldots\ldots\ \Omega$$

$$\Delta I_C = \ldots\ldots ;\ \Delta I_E = \ldots\ldots \ ;\ h_{fb} = \alpha = \left(\frac{\Delta I_C}{\Delta I_E}\right)_{V_{CB}} = \ldots\ldots$$

For constant I_E:

$$\Delta V_{EB} = \ldots\ldots ;\ \Delta V_{CB} = \ldots\ldots \ ;\ h_{rb} = \left(\frac{\Delta V_{EB}}{\Delta V_{CB}}\right)_{I_E} = \ldots\ldots$$

$$\Delta I_C = \ldots\ldots ;\ \Delta V_{CB} = \ldots\ldots \ ;\ h_{ob} = \left(\frac{\Delta I_C}{\Delta V_{CB}}\right)_{I_E} = \ldots\ldots\ \Omega^{-1}$$

Result

The input and the output characteristics of the transistor for common base configuration are drawn and the h-parameters are determined. They are,

Input impedance, $r_e = h_{ib} = \ldots\ldots\ \Omega$

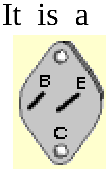
Forward current gain $\alpha = h_{fb} = \ldots\ldots$

Reverse voltage gain $h_{rb} = \ldots\ldots$

Output admittance, $h_{ob} = \ldots\ldots\ \Omega^{-1}$

*2N3055

The **2N3055** is a silicon NPN power transistor intended for general purpose applications. It was introduced in the early 1960s by RCA using a hometaxial power transistor process, transitioned to an epitaxial base in the mid 1970s. Its numbering follows the JEDEC standard. It is a transistor type of enduring popularity.



Metal body itself is collector

2N3055 transistor is mounted on an aluminum heat sink. A mica insulator electrically isolates the transistor case from the heatsink.

Specifications

Packaged in a TO-3 case style, it is a 15 amp, 60 volt, 115 watt power transistor with β (forward current gain) of 20 to 70 at a collector current of 4 A. It has a transition frequency of 2.5 MHz; at this frequency the calculated current gain (beta) drops to 1, indicating the transistor can no longer provide useful amplification in common emitter configuration.

Exp.No.3.7

Characteristics of a common emitter transistor

Aim: To study the characteristics of the given transistor in common emitter configuration.

Components and accessories required: Transistor 2N3055 or B C 107, power supplies, milli-ammeters and voltmeters.

Circuit, theory and procedure: In common emitter configuration emitter is common to both input and output. Transistor is most often used in this configuration. To study the input and output characteristics of this configuration, connections are made as shown in the fig.a.

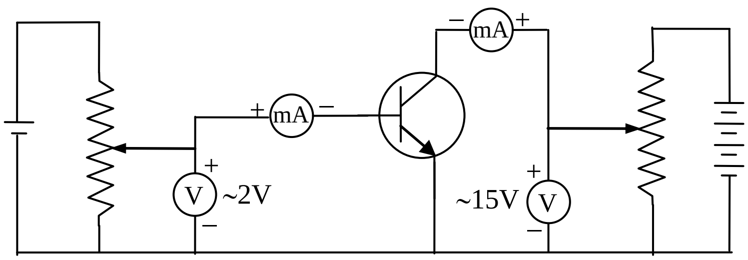


Fig.a

Input characteristics: The input characteristics in this configuration are the graphical representation of the variation of the base current I_B with the base-emitter voltage V_{BE} keeping the collector-emitter voltage V_{CE} constant.

Set the output voltage V_{CE} to particular value, say 5 V. Then increase the input voltage, V_{BE} from zero in steps of 0.02 V to about 0.7 V (for 2N3055 transistor) and in each case the base current I_B is noted. Repeat the experiment for different values of V_{CE}. A family of curves, called the input characteristics, is plotted by taking the input voltage V_{BE} in the X axis and the corresponding input current I_B in the Y axis as shown in fig.b.

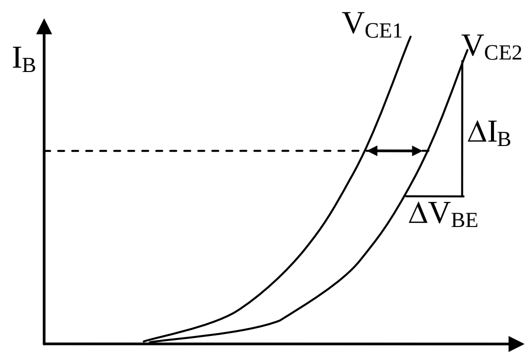


Fig.b

Output characteristics: To draw the output characteristics set the input current I_{EB} to a fixed value, say 1.39 mA. Then increase the output voltage V_{CE} in steps from zero to 10 or 12 volt and in each case the collector current I_C is noted. A graph is drawn between V_{CE} and I_C as shown in fig.c. Repeat the entire experiment for different values of I_B and all the graphs are plotted on the same graph paper. A typical family of curves is shown in the fig.c. Thus we get the output characteristics of the given transistor for common emitter configuration.

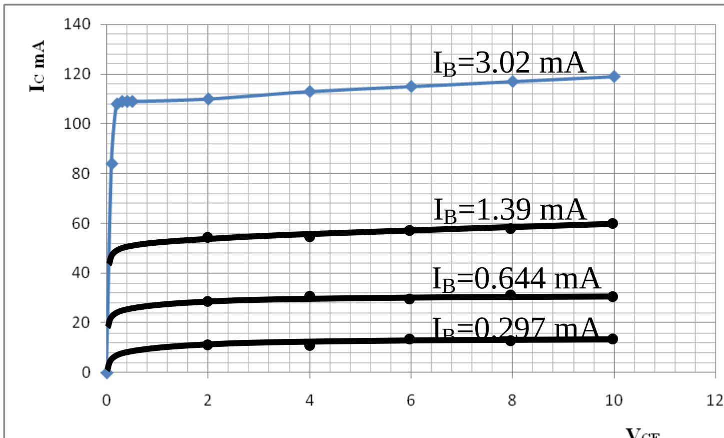


Fig.c: Output characteristics of 2N3055 transistor

The four h-parameters (two forward parameters and two reverse parameters) of the transistor circuit in common emitter configuration are defined as, *Forward parameters*

Input impedance, $h_{ie} = \beta r_e = \left(\frac{\Delta V_{BE}}{\Delta I_B}\right)_{V_{CE}}$ (1)

Forward current gain, $h_{fe} = \beta = \left(\frac{\Delta I_C}{\Delta I_B}\right)_{V_{CE}}$ (2)

Reverse parameters

Reverse voltage gain $h_{re} = \left(\frac{\Delta V_{EB}}{\Delta V_{CE}}\right)_{I_B}$ (3)

Output admittance, $h_{oe} = \left(\frac{\Delta I_C}{\Delta V_{CE}}\right)_{I_B}$ (4)

- Ensure that for forward biasing of the transistor, the base of the n p n transistor is slightly more positive than the emitter and the collector is biased at a large positive potential.
- For 2N3055 transistor the input characteristics for various output voltages almost coincide.

Observation and tabulation

To draw the input characteristics

V_{BE} volt	V_{CE} = ….. volt	V_{CE} = ….. volt	V_{CE} = …… volt	V_{CB} = ….. volt
	I_B mA	I_B mA	I_B mA	I_B mA

To draw the output characteristics

V_{CE} volt	I_B = ….. mA	I_B = …. mA	I_B = ….. mA	I_B = ….. mA
	I_C mA	I_C mA	I_C mA	I_C mA

From input characteristics

For constant V_{CE}:

$$\Delta V_{BE} = \ldots\ldots ;\ \Delta I_B = \ldots\ldots ;\ h_{ie} = \beta r_e = \left(\frac{\Delta V_{BE}}{\Delta I_B}\right)_{V_{CE}} = \ldots\ldots\ \Omega$$

$$\Delta I_C = \ldots\ldots ;\ \Delta I_B = \ldots\ldots ;\ h_{fe} = \beta = \left(\frac{\Delta I_C}{\Delta I_B}\right)_{V_{CE}} = \ldots\ldots$$

For constant I_B:

$$\Delta V_{EB} = \ldots\ldots ;\ \Delta V_{CE} = \ldots\ldots ;\ h_{re} = \left(\frac{\Delta V_{EB}}{\Delta V_{CE}}\right)_{I_B} = \ldots\ldots$$

$$\Delta I_C = \ldots\ldots ;\ \Delta V_{CE} = \ldots\ldots ;\ h_{oe} = \left(\frac{\Delta I_C}{\Delta V_{CE}}\right)_{I_B} = \ldots\ldots\ \Omega^{-1}$$

Result

The input and the output characteristics of the transistor for common emitter configuration are drawn and the h-parameters are determined. They are,

Input impedance, $\beta r_e = h_{ie} = \ldots\ldots\ \Omega$

Forward current gain $\beta = h_{fe} = \ldots\ldots$

Reverse voltage gain $h_{re} = \ldots\ldots$

Output admittance, $h_{oe} = \ldots\ldots\ \Omega^{-1}$

* If low power transistors are used voltages and currents are much lower.
* For pin diagram of 2N3055 see page 45.

Exp.No.3.8
Single transistor voltage regulator

Aim: To construct a Zener controlled single transistor voltage regulator.

Components and accessories required: A power transistor 2N3055 (used for plotting the characteristics-previous experiments), all components and accessories for a full wave rectifier, power supplies, milli-ammeters and voltmeters.

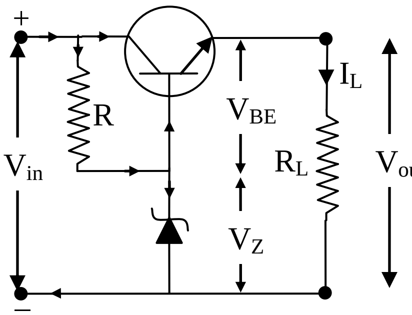


Fig.a: Zener controlled transistor series voltage regulator

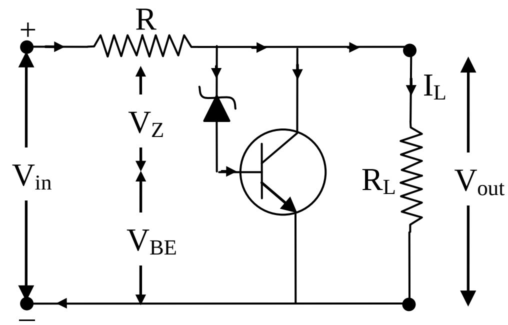


Fig.b: Zener controlled transistor shunt voltage regulator

Circuit, theory and procedure: The efficiency of Zener regulated power supply becomes very low when the load current is high. Under such conditions a Zener controlled transistor is always used for maintaining output voltage constant. Basically there are two types of Zener controlled transistor voltage regulators. They are, **Series Voltage Regulators** and **Shunt Voltage Regulators**.

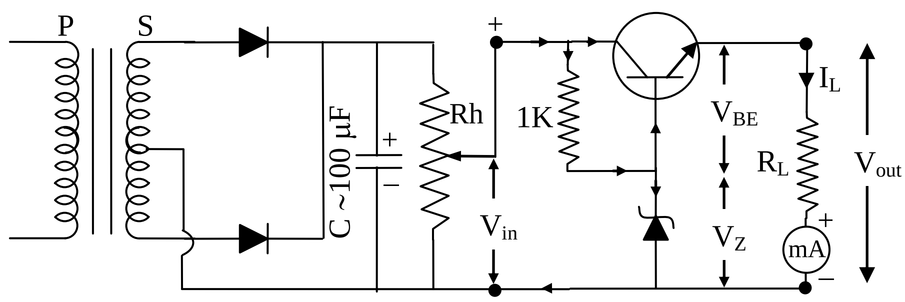


Fig.c: Zener controlled transistor series voltage regulator

In our experiment we construct only the transistor series voltage regulator. A simple series voltage regulator using an NPN transistor and a Zener diode is shown in the fig.a. This circuit is called a series regulator because collector and emitter terminals of the transistor are in series with the load. This circuit is also called an emitter follower voltage regulator because the transistor is connected in emitter follower configuration. Here, the transistor is termed a series-pass transistor because the entire load current passes through it.

To do the experiment the connections are made as shown in the fig.c. The unregulated dc supply, filtered output from the full wave rectifier or bridge rectifier, (the output of bridge rectifier is nearly two times than that of the ordinary full wave rectifier) is fed to the input terminals and regulated output voltage V_{out} is obtained across the load resistor R_L. Zener diode provides the reference voltage and the transistor acts as a variable resistor, whose resistance varies with the operating conditions (base current I_B). The principle of operation of such a regulator is based on the fact that a large proportion of the change in supply (or input) voltage appears across the transistor and, therefore output voltage tends to remain constant. Keeping in mind the polarities of different voltages we have

$$V_{out} = V_z - V_{BE} \approx V_Z, \text{ since } V_{BE} \text{ is small (0.7 V for silicon transistors)}$$

Then from figures a and b,

$$V_{out} = V_L \approx V_Z = V_{in} - V_R$$

where, V_R is the voltage across R.

The base voltage of the transistor remains almost constant being equal to that across the Zener diode, V_z.

Operation: (i) Let the supply (or input) voltage increase which will cause the output voltage V_{out} to increase. An increase in output voltage V_{out} will result in decrease of V_{BE} because V_z is fixed and decrease in V_{BE} will reduce the level of conduction. This will lead to an increase in the collector-emitter resistance of the transistor causing an increase in collector to emitter voltage and as a result the output voltage will be reduced. Thus output voltage will remain constant. Similar explanation can be given for decrease in supply voltage.

(ii) Now let us consider the effect of change in load on the output voltage - say current is increased by decrease in R_L. Under such a situation the output voltage V_{out} tends to fall and, therefore, V_{BE} tends to increase. As a result the conduction level of the transistor will increase leading to decrease in the collector-emitter resistance. The decrease in the collector-emitter resistance of the transistor will cause the slight increase in input current to compensate for the decrease in R_L. Thus the output voltage, being equal to $I_L R_L$, remains almost constant. Similar explanation will hold true for increase in R_L.

The advantage of such a circuit is that the changes in Zener current are reduced by a factor β and thus the effect of Zener effect is greatly reduced and much more stabilized output is obtained.

Output voltage from a series regulator, $V_{out} = (V_z - V_{BE})$, and maximum load current $I_{L(max)}$ can be the maximum emitter current that the transistor is capable of passing. For a 2N3055 transistor load current I_L could be 15 A. When load current I_L is zero, the current drawn from the supply is approximately $(I_z + I_{C(min)})$. The Zener regulator (resistor R and Zener diode form a simple Zener regulator) has to supply only the base current of the transistor. The emitter follower voltage regulator is, therefore, much more efficient than a simple Zener regulator.

Line regulation: It is the capability to maintain a constant output voltage level on the output channel of a power supply despite changes to the input voltage level. *Line regulation is expressed as the percentage of the change in the output voltage relative to the change in the input line voltage*.

To draw the line regulation characteristics, the load current I_L is fixed to a particular value, say 5 mA. Then the input voltage is varied in steps of 1 V from 10 V to 25 V and in each case the output voltage is measured (it is approximately $V_{in} - V_R$). A graph is drawn between V_{in} along X axis and V_{out} along the Y axis. Find out the slope of the graph.

Percentage of line regulation $= 100\times$ Slope of V_{in}-V_{out} graph.

Load regulation is the capability to maintain a constant voltage (or current) level on the output channel of a power supply despite changes in the load (such as a change in resistance value connected across the supply output).

To draw the load regulation characteristics, the input voltage V_{in} is fixed to a particular value, say 15 V. The output voltage V_{out} is measured and load current I_L is either measured or calculated for various load resistances. A graph is drawn between I_L along the X axis and V_{out} along the Y axis.

The output voltage with zero load (I_L zero and R_L infinity) V_{NL} also is measured and the voltage regulation is calculated as, Percentage of load regulation $= \left(\frac{V_{NL} - V_{FL}}{V_{FL}}\right) \times 100\ \%$

Observation and tabulation

Line regulation

V_{in} volt	V_{out} volt	V_{in} volt	V_{out} volt	V_{in} volt	V_{out} volt
10		16		22	
11		17		23	
12		18		24	
13		19		25	
14		20			
15		21			

Percentage of line regulation = 100 × Slope of V_{in}-V_{out} graph = ………. %

Load regulation V no load (R_∞), V_{NL} = ………. volt

R_L ohm	V_{out} volt	Load I_L in mA (measured or calculated)	Percentage of voltage regulation $\left(\frac{V_{NL} - V_{FL}}{V_{FL}}\right) \times 100$
100			
200			
300			
400			
500			
600			
700			
800			
900			
1000			
2000			
3000			
4000			
5000			
6000			
7000			
8000			
9000			
10000			

Result

A Zener controlled single transistor voltage regulator is constructed and the variations of output voltage for different input voltages and output currents are studied.

* For pin diagram of 2N3055 see page 45.

Exp.No.3.9

Realization of Logic gates using transistors

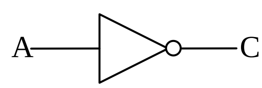
Aim: To construct logic gates using transistors.

Components and accessories required: BC547 transistors, resistors, power supplies, d c voltmeter or LED, etc.

Circuit, theory and procedure: A logic gate is an electronic circuit which makes logic decisions. It has one output and one or more inputs. The output signal appears only for certain combinations of inputs. Logic gates are the building blocks of a digital system. Each logic gate follows a certain logical relationship between input and output voltages. The three basic logic gates are OR gate, AND gate and NOT gate. Each basic logic gate is indicated by a *symbol* and its function is defined either by a truth table or by a *Boolean expression*.

Truth Table: It is a table that shows all possible input combinations and the corresponding output.

NOT gate: It is a device having only one input and one output. It is so called because its output is NOT equal to the input. The logic symbol and truth table of a NOT gate are shown in fig.a and fig.b. A NOT gate can be realized by making use of a transistor. The electronic circuit of NOT gate is shown in fig.c.



$C = \overline{A}$

Fig.a

A	C
0	1
1	0

Fig.b

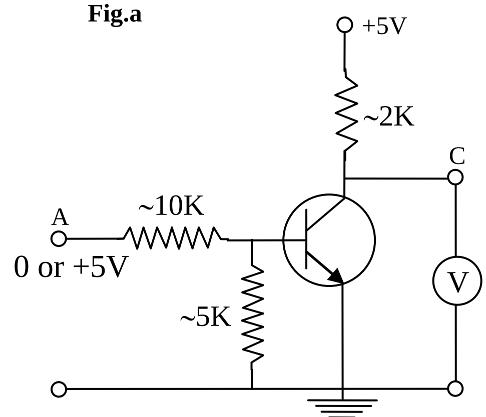


Fig.c: NOT gate

Working

1. When A is earthed (i.e. A = 0), the base-emitter junction will be reverse biased. Hence the transistor will not conduct. Therefore, the potential at C w.r.t earth becomes 5V i.e. the output C equals 1.
2. When A is connected to 5V (i.e. A = 1), the base-emitter junction of the transistor becomes forward biased. Hence, the transistor conducts. Therefore, the potential at C w.r.t earth becomes zero (the collector voltage with respect to the emitter may be near zero), i.e. the output C equals 0.

Connections are made as shown in fig.c and the observations are recorded as the truth table as shown in fig.b.

AND Gate: In Boolean algebra, the multiplication symbol (.) is referred to as AND and the operation may be written as,

$$A \text{ AND } B = A.B = C$$

AND gate is a device which combines A with B to give C in accordance with this expression. The logic symbol of AND gate with its truth table are shown in fig.a and fig.b. The AND gate can be realized by the electronic circuit shown in fig.c.

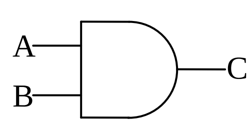


Fig.a

A	B	C
0	0	0
0	1	0
1	0	0
1	1	1

Fig.b

Working

1. When A and B are connected to earth (i.e. A = 0 and B = 0) transistors Q_1 and Q_2 do not conduct. Therefore C will be at zero potential w.r.t earth, i.e. output becomes 0.
2. When A is earthed and B is connected to positive terminal of the power supply 5V (i.e. A = 0 and B = 1), transistors Q_1 and Q_2 do not conduct. In this case also the potential at C w.r.t earth becomes zero. Hence, output becomes 0. (Due to minority carriers there may be a very small current and the voltmeter reads a small voltage. Hence 0 corresponds to the minimum voltage).
3. When A is connected to the positive of the power supply 5V and B is earthed, transistors Q_1 and Q_2 do not conduct. In this case also the potential at C w.r.t earth becomes zero. Hence, output becomes 0.
4. When both A and B are connected to positive terminal of power supply 5V, the transistors Q_1 and Q_2 conduct. Hence the potential at C w.r.t earth becomes maximum, ~4.2V for BC547 transistors. Hence output becomes1.

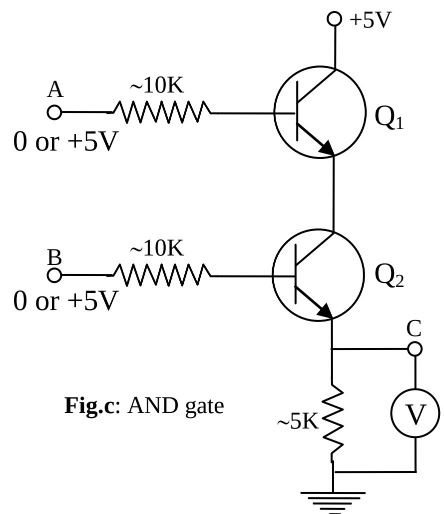


Fig.c: AND gate

The output of an AND gate assumes 1 *only if all the inputs assume* 1. Connections are made as shown in fig.c and the observations are recorded as the truth table as shown in fig.b.

OR Gate: In Boolean algebra, addition symbol (+) is referred to as OR and the operation may be written as,

A OR B = A+ B = C

That is, C is true when either A is true or B is true or both A and B are true.

OR gate is a device that combines A with B to give C as the result in accordance with the above logical expression. The OR gate has two or more inputs and one output. The logic symbol of OR gate with its truth table is shown in fig.a and fig.b. The fig.c represents a two input OR gate with two transistors Q_1 and Q_2.

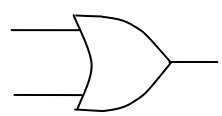


Fig. a

A	B	C
0	0	0
1	0	1
0	1	1
1	1	1

Fig. b

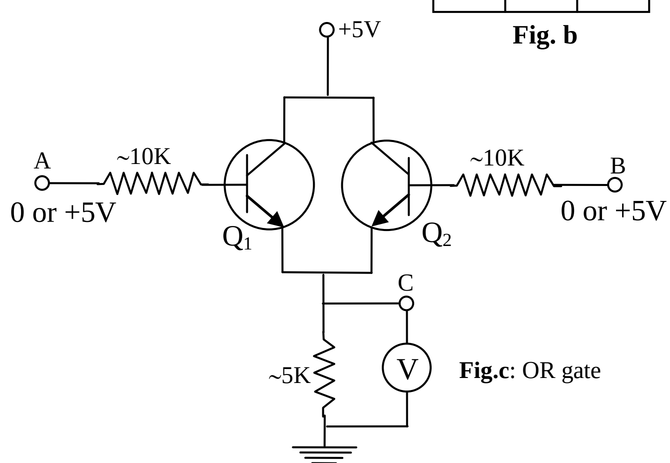


Fig.c: OR gate

Working

1. When both A and B are connected to earth (i.e. A = 0 and B = 0) the transistors do not conduct and therefore no voltage develops across R. Then the voltage at C is zero (minimum) w.r.t earth. Hence output becomes 0.
2. When A is connected to earth and B is connected to positive terminal of battery 5V (i.e. A = 0 and B = 1) transistor Q_1 does not conduct while Q_2 conducts. Now a voltage drop of 4.30V for (BC547 transistor) takes place across R with C at 4.30V (maximum) w.r.t earth. Hence output becomes 1.

3. When A is connected to positive terminal of battery 5V and B is earthed (i.e. A = 1 and B = 0), the transistor Q_1 conducts and Q_2 does not conduct. Now a voltage drop of 4.30V takes place across R, with C at 4.3V (maximum) w.r.t earth. Hence output becomes 1.
4. When A and B are connected to positive of battery 5V (i.e. A = 1 and B = 1), both the diodes conduct. Now a voltage drop of 4.33V takes place across R., with C at 4.33V (maximum) w.r.t earth. Hence output becomes 1.

The output of an OR gate assumes 1 *if one or more inputs assume* 1. In other words, it is *any* or *all* gate because an output occurs when any or all the inputs are present. An OR gate is called inclusive OR gate because it includes the case when both inputs are true.

Connections are made as shown in fig.c and the observations are recorded as the truth table as shown in fig.b.

Universal gates

NOR gate and NAND gate are known as universal gates because they can be used to realize the basic logic functions: OR, AND and NOT .i.e. the fundamental gates can be formed either by the combination of NOR gates alone or by the combination of NAND gates alone.

NAND Gate: If we connect the output of an AND gate to the input of a NOT gate, the gate so obtained is called NAND gate. The function of a NAND gate is just the reverse of the AND gate. NAND *gate gives an output of* 1, *if both the inputs are not* 1. In other words, it gives an output 1 if either A or B or both are 0. Logic Symbol and truth table of a NAND gate are shown in fig.a and fig.b. For the NAND logic gate, the transistors are in series as in AND gate, but the output is above them.

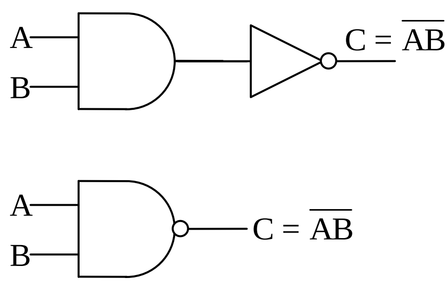


Fig.a

A	B	C
0	0	1
1	0	1
0	1	1
1	1	0

Fig. b

When A = 0 and B = 0, A = 0 and B = 1 and A = 1 and B = 0, the transistors do not conduct and we get the maximum voltage between C and ground, i.e. C = 1 for these three cases. Only when A = 1 and B = 1, both the transistors conduct and the voltage between C and ground is minimum, i.e. C = 0.

Connections are made as shown in fig.c and the observations are recorded as the truth table as shown in fig.b.

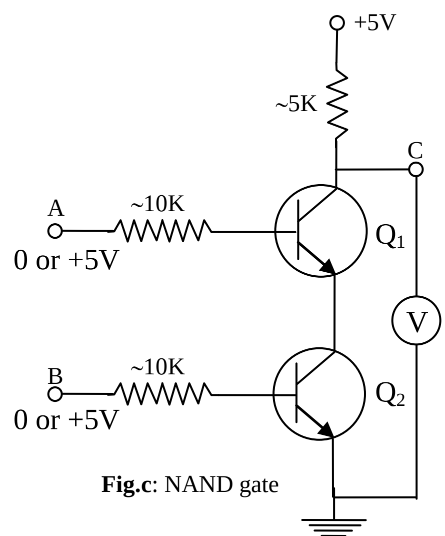


Fig.c: NAND gate

NOR Gate: If we connect the output of an OR gate to the input of a NOT gate, the gate so obtained is called NOR gate. The function of a NOR gate is just the reverse of the OR gate. NOR *gate gives an output of* 1 *only when all inputs are* 0. The logic symbol and truth table for a two input NOR gate is shown in fig.a and fig.b below. For the NOR logic gate, the transistors are in parallel as in OR gate and with the output above them so that if either or both of the inputs are high, the output is driven low.

When A = 0 and B = 0, the transistors do not conduct. Hence the output voltage is maximum. When A = 0 and B = 1, A = 1and B = 0 and A = 1 and B =1, one or both of the transistors conduct and hence the output voltage is minimum.

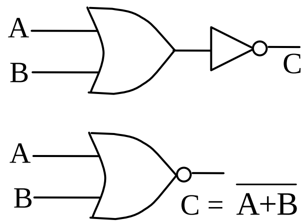


Fig.a

A	B	C
0	0	1
1	0	0
0	1	0
1	1	0

Fig.b

Connections are made as shown in fig.c and the observations are recorded as the truth table as shown in fig.b.

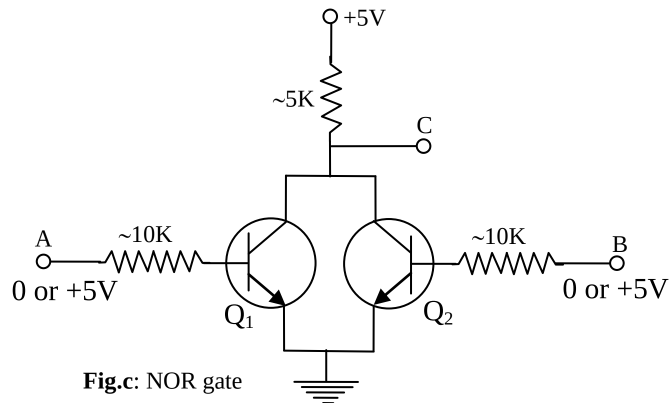


Fig.c: NOR gate

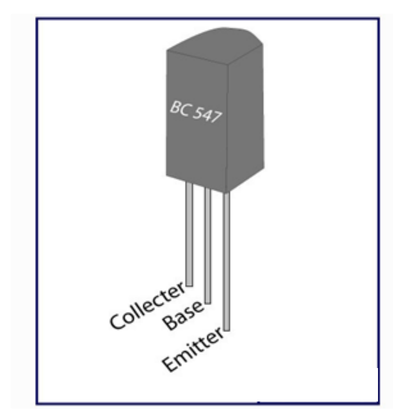
- Ensure that large resistors are connected to the base of the transistor so as to control the base current not to exceed its limiting value.
- Instead of voltmeter we can use LED to observe the maximum and minimum.
- In the logic gates '0' corresponds to minimum voltage by the current due to the minority carriers and '1' corresponds to the maximum voltage by the current due to the majority carriers.
- See that in the AND gate the two transistors are connected in series and in the OR gate they are connected in parallel.
- AND gate can be easily converted into a NAND gate and OR gate into a NOR gate.

Observation and tabulation

The observations are recorded in the truth table.

Result

The three basic logic gates (NOT, AND and OR), NAND and NOR gates using transistors are constructed and the observations are recorded as truth tables.



Exp.No.3.10 & 3.11

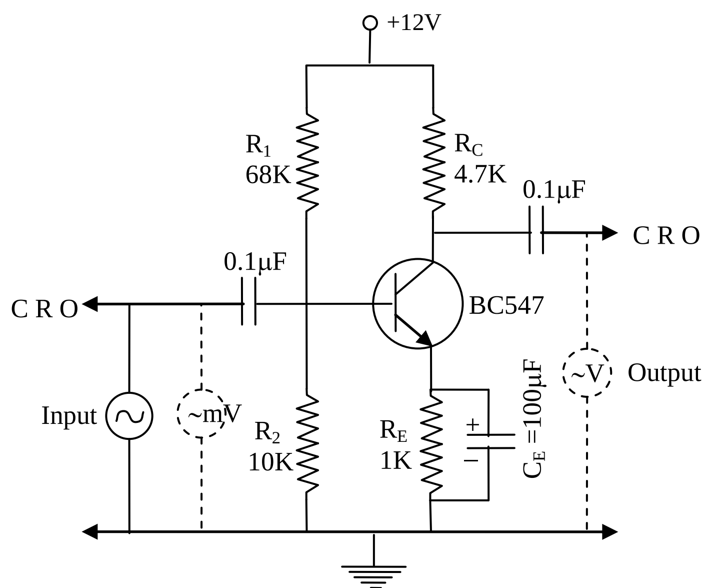
Common emitter amplifier-Gain & frequency response

Aim: To construct a common emitter amplifier and study its gain and frequency response.

Components and accessories required: BC547 (or, BC107) transistor, resistors, capacitors (1 electrolytic 100 μF and 2 polyester type 0.1μF), power supply, C R O or d c voltmeter, signal generator, etc.

Circuit, theory and procedure

An amplifier is a device used to amplify an a c signal. The figure shows an electronic circuit with a BC547 transistor in common emitter configuration. The signal to be amplified is given to the input side. We get the amplified output in the output side. Input voltage and the output voltage are measured either by a C R O or a c voltmeter.



Gain of the amplifier: The output of the signal generator is set to 10 mV amplitude sine wave of frequency 5KHz. It is applied to the input side of the amplifier. The input and output a c voltages are measured either by C R O or by a c voltmeter.

$$\text{Gain of the amplifier,} \quad A_v = \frac{\text{Output voltage}}{\text{Input voltage}} = \frac{V_o}{V_i}$$

Then the input is changed to 20 mV, 30 mV, ……. and in each case the output is measured and the gain is calculated.

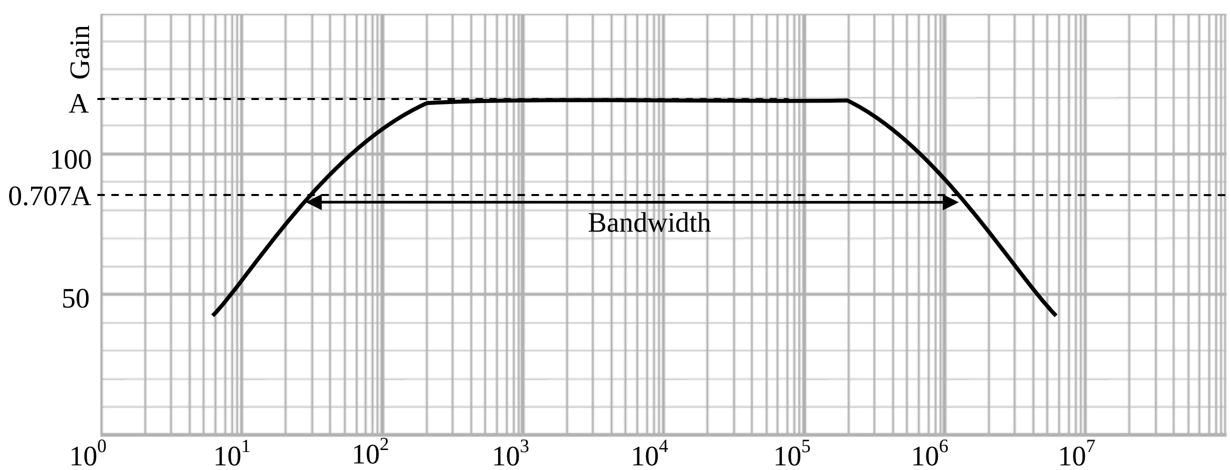


Fig.b: Semi-logarithmic graph

Frequency response: To study the frequency response of the amplifier, set the signal generator for a signal of 20 mV sine wave. The frequency is increased in steps 10Hz, 20Hz, 30Hz, ……..upto 100Hz, then increase in steps as 200Hz, 300Hz, …….1KHz, then, 2KHz, 3KHz, ……,10KHz, 20KHz, …, 100KHZ, 200KHz, …. Then, 1MHz, 2MHz, …….. In each case the output is measured and the gain is calculated. Draw a semi-logarithmic plot on a semi-logarithmic graph, taking frequency in the X axis and the linear value of gain in the Y axis. (Instead of plotting on the semi-logarithmic graph, we can also plot the frequency response curve on ordinary graph paper in which log f is taken in the X axis and gain in the Y axis).

- Resistance R_E is of the order (470Ω~1KΩ)
- C R O measurement is preferred, since the wave form can be observed.

Observation and tabulation

To determine the gain of the amplifier

Observations with C R O

Frequency of the signal = ………. Hz

No. of divisions of peak to peak of input signal	mV/division	Input Voltage V_i volt	No. of divisions of peak to peak of output signal	Volt/division	Output voltage V_o volt	Gain $A_V = \frac{V_o}{V_i}$

***Observation with a c voltmeter**

Frequency of the signal = ………. Hz

Input voltage r m s mV	Input voltage V_i r m s volt	Output voltage V_o r m s volt	Gain $A_v = V_o/V_i$

To draw the frequency response curve of the amplifier

Observations with C R O

Peak to peak voltage of the input signal = ……mV = ….. V

Frequency Hz	Measurement with C R O				*Measurement with a c voltmeter		
	No. of divisions of peak to peak of output signal	Volt per division	Output Voltage V_o volt	Gain $A_V = \frac{V_o}{V_i}$	Input r m s mV	Output r m s volt	Gain $A_V = \frac{V_o}{V_i}$
10							
20							
30							
40							
50							
60							
70							
80							
90							
100							
200							
300							
400							
500							
600							
700							
800							
900							
1000							
2000							
….							
10000							
20000							
….							
100000							
200000							
300000							
400000							
500000							
600000							
700000							
800000							
900000							
1000000							
2000000							
3000000							
4000000							
5000000							

Result: The common emitter amplifier is constructed and its gain is determined. Also the frequency response curve is plotted.

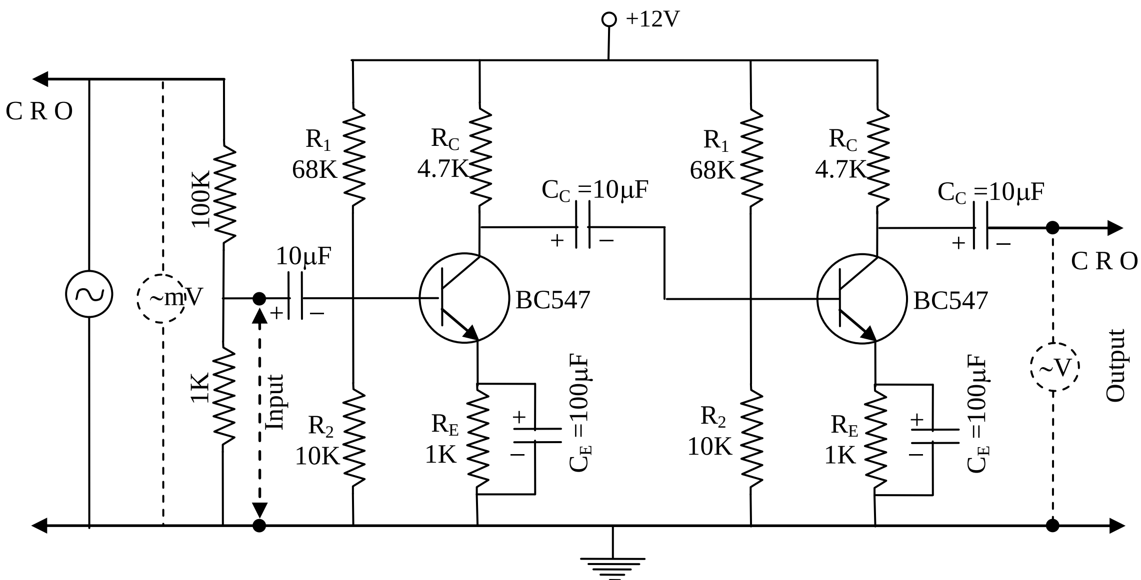
Exp.No.3.12

Two stage R C coupled amplifier-Gain

Aim: To construct a two stage R C coupled amplifier and study its gain and frequency response.

Components and accessories required: BC547 (or, BC107) transistors, resistors, capacitors, power supply, C R O or d c voltmeter, signal generator, etc.

Circuit, theory and procedure



A two stage R C coupled amplifier is the combination of two common emitter amplifiers coupled together. C_C is the coupling capacitor. The gain of the amplifier is approximately the product of the gains of the two stages. It is large of the order of 10,000. In order to avoid the clipping we have to apply very small input signal of the order of 70 μV. To produce such small input signal we use a potential divider arrangement connected to the input side of the amplifier as shown in the figure.

Connections are made as shown in the circuit above. The signal generator is set to produce the desired signal V_s and it is applied to the input side. The signal voltage V_s and the output can be measured either by C R O or a c voltmeter.

$$\text{Gain of the amplifier,} \quad A_v = \frac{\text{Output voltage}}{\text{Input voltage}} = \frac{V_o}{V_i}$$

where, $V_i = \frac{V_s}{101}$. Repeat the experiment for different values of V_S keeping the frequency remains constant and the average gain is determined.

*To study the frequency response of the amplifier, keep the signal voltage constant. The frequency is increased in steps 10Hz, 20Hz, 30Hz, ……..upto 100Hz, then increase in steps as 200Hz, 300Hz, …….1KHz, then, 2KHz, 3KHz, ……,10KHz, 20KHz, …, 100KHZ, 200KHz, …. Then, 1MHz, 2MHz, …….. In each case the output is measured and the gain is calculated. Draw a semi-logarithmic plot on a semi-logarithmic graph, taking frequency in the X axis and the linear value of gain in the Y axis.

- The input signal measured across the 1K may be different from the calculated value. This may be due to the effect of the impedance of the amplifier circuit. Because it is very difficult to measure the input voltage we are forced to calculate the voltage across the 1K as the input voltage. The observed gain with calculated input is less than the actual gain. (For example with 20 mV peak input signal the calculated signal voltage is 141.4 μV and the measured value is 114 μV and the output of the amplifier is 1.09 V. The actual gain is 9561. But with the calculated input the gain is only 7709).

Observation and tabulation

To determine the gain of the amplifier

Observations with C R O

Frequency of the signal = ………. Hz

No. of divisions of peak to peak of signal V_S	mV per division	Signal voltage V_S	Input Voltage $V_i = \frac{V_s}{101}$ volt	No. of divisions of peak to peak of output signal	Volt per division	Output voltage V_o volt	Gain $A_V = \frac{V_o}{V_i}$

***Observation with a c voltmeter**

Frequency of the signal = ………. Hz

Signal voltage r m s V_S mV	Input voltage r m s $V_i = \frac{V_s}{101}$ mV	Input voltage V_i r m s volt	Output voltage V_o r m s volt	Gain $A_v = V_o/V_i$

***To draw the frequency response curve of the amplifier**

Observations with C R O

Peak to peak voltage of the signal V_s = ……mV = ….. V

Peak to peak input voltage, $V_i = \frac{V_s}{101}$ = ……… mV = ……. V

Frequency Hz	Measurement with C R O				*Measurement with a c voltmeter		
	No. of divisions of peak to peak of output signal	Volt per division	Output Voltage V_o volt	Gain $A_V = \frac{V_o}{V_i}$	Input r m s mV	Output r m s volt	Gain $A_V = \frac{V_o}{V_i}$
10							
20							
30							
40							
50							
60							
70							
80							
90							
100							
200							
300							
400							
500							
600							
700							
800							
900							
1000							
2000							
….							
10000							
20000							
….							
100000							
200000							
300000							
400000							
500000							
600000							
700000							
800000							
900000							
1000000							
2000000							
3000000							
4000000							
5000000							

Result: The two stage R C coupled amplifier is constructed and its gain is determined. Also the frequency response curve is drawn.