Basic Electronics

Additional rules for PLC's and Electronics Classrooms:

1. **No cold drinks, tea, coffee, water or food is allowed on workshop floor as it may damage sensitive and expensive equipment used.**
2. **Nobody is allowed to eat or drink on workshop floor.**
3. **Absences for too many days (+ 2days) will result in the learner to be rescheduled, because of the amount of work lost.**
4. **Any learner that cannot be at work or late for work must inform the Responsible Training Officer. (017 – 614 5304)**
5. **No equipment may be used or switched on by a learner if not authorized by Training Officer.**
6. **All electronic tools be given back after course in good order.(Will be checked and inspected)**
7. **Mobile phones must be switched off or be "silent mode". Play or use of phones during class times will not be allowed.**
8. **If a learner leaves the classroom, he must notify the Training Officer or Class captain, of where he is going to. This is especially necessary in an event of an emergency.**
9. **Learners are not allowed to leave the Training Centre without the permission of the Responsible Training Officer.**
10. **Attendance register will be signed every morning at arrival, and again at the end of the day before leaving.**
11. **All learners will keep to working hours and break times, everyone will be in the workshop and ready to work by 07h00 in the morning, late coming will not be tolerated, and all of us will only leave at 16h00.**
12. **Workshop and class rooms will be kept clean at all times.**
13. **Additional "Smoke breaks" will be determined by Responsible Training Officer, and if "smoke breaks" are misused it will be stopped.**
14. **Horseplay will not be tolerated.**
15. **Correct PPE will be worn at all times in workshop (Safety glasses with practical)**
16. **No references (Notes or Electronic devices) are allowed when doing assessments**

Safety Requirements

1. All learners should attend class with the minimum PPE requirements (long sleeve overall jackets, overall pants and approved boots).

2. Please request and sign for safety goggles from your training officer.

3. Before switching on the supply voltage ensure that your training officer has seen and approved your circuit and that you are wearing your safety goggles.

4. No equipment may be used or switched on by a learner if not demonstrated and authorized by the Training Officer.

5. Risk assessments and toolbox talks must be done and signed for with your training officer before any practical assessment.

6. Do not miss-use equipment, tools and components in the class rooms.

If any of the above is not obeyed by the learner, the Responsible Training Officer can take disciplinary action against the Learner.

I ……………………………………..Control number…………fully understand the content of this document and agree to obey by it.

Signed: ___________________________ Date:__________________

Training and development department• insert BU name
Copyright

Approval Page

Manual compiler

Name
Designation
Area

Technical accuracy
Approval

Name
Designation
Area

Training technological
Compliance approval

Name
Designation
Area

Manual approval

Name
Designation
Area

Compilation date: February 2012

Revisions

Book Filename – Basic Electronics

Revision No	Date	Description	Author
01	October 2007	Initial release	
02	May 2008	Updated index, addition of information.	
03	May 2010	Changed format, updated index, addition of tables and figures and method of handling notes	
04	February 2012	Addition of information, addition of PPE and method of handling notes.	
05	February 2014	Removal of Unit Standards and update on Additional Rules	

Table of Contents

List of Illustration

MODULE 1

MODULE 2

MODULE 3

MODULE 4

MODULE 5

MODULE 6

MODULE 7

MODULE 8

MODULE 9

MODULE 10

MODULE 11

MODULE 12

MODULE 13

MODULE 15

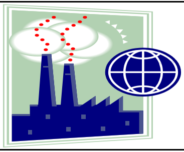
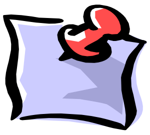
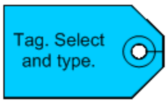
How to use the Learner Guide

To make it easier for you a number of different ICONS have been used throughout the learning material. Refer to the table below for the ICONS and their meanings.

Table 1 ICON Meanings

	Group Discussion. Keep notes of what the group does in the exercise
	Reading or research activity
	When you see this icon, you are required to complete workbook activities
	When you see the icon you are required to complete the activity in real life scenario or workplace
	Glossary of terms
	Tools and Templates that can be copied and re-used throughout the learning programme

Table 2 ICON Descriptions

ICON	Description
	Index
	Unit Standard
	CBT Training
	Individual activity
	Group activity
	Notes
	Safety rules / regulations or tips
	Warning / Dangerous substances
	Self assessment
	Self assessment information

Learner Support

To support you with your learning process, the following resources are available

1. *Your Learner Guide*
2. *Facilitator*

You can contact your learning practitioner via phone or email.

Name of Learning practitioner	Email Address	Contact Number

Learning Method

1. You will be assessed, when you are confident that you may achieve the outcomes as listed, to determine your competence as measured against the assessment criteria.

2. This assessment will be in line with accepted best practices regarding assessment.

    2.1 A practical test will be set at the end of the module and must be completed without using references.

3. When successfully completing the outcomes of this Unit Standard you will be awarded 10 credits that will be registered on the National Record of Learning database.

Activities related to this Learner Guide

You are required to complete the following activities.

- **Activities in the learner guide**

These activities relates to the knowledge component of this learning programme.

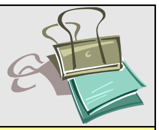
- Construct basic electronic circuit.
- Demonstrate an understanding of basic electronic theory and components.

•Workplace Activities

These activities related to the actual application of skills in the workplace.
Below is a summary of all workplace activities.

Table 3 Workplace Activities

	**Workplace Activities** These activities must be completed in the workplace. Number your documents clearly.

				
Module 1				
			Solder Components (HM-3)	
Module 2				
			Identify and Use Resistors (BE-1)	
Module 3				
			Identify and Use Capacitors (BE-2)	
Module 4				
			Identify and Use Inductors (BE-3)	
Module 5				
			Identify Diodes (BE-4)	
Module 6				
			Construct Rectification Circuits (BE-5)	
Module 7				
			Identify and Use Zener Diodes (BE-6)	
Module 8				
			Construct a Votlage Doubler (BE-7)	
Module 9				
			Identify Transistor Action (BE-8)	
Module 10				
			Identify Transistor Configurations (BE-9)	
Module 11				
			Test Regulated Power Suppliers (BE-10)	
Module 12				
			Test Thyristors (BE-11)	
Module 13				
			Test Thyristor Phase Control (BE-12)	
Module 14				
			Faults and Circuits 2 (FF-2)	
Module 15				
			Use an Oscilloscope (TI-2)	

Module 1
HM - 3 Solder components

Purpose of module

The objective of this module is to enable the learner to solder components.

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Solder and unsolder components on a printed circuit board.

Procedures Relating To This Module

- The wires must protrude at least 1mm above the copper track.
- The solder fillet must cover the land evenly and taper up to blend into the wire in a concave shape.
- The solder fillet must have a height of at least 0,75mm above the copper track.
- The copper strip must not lift from the material because of overheating.

1. Introduction

Soldering is usually used to obtain permanent connections between metals and wire and to ensure that metallic and electrical continuity is established. The alloy most commonly used for general purpose soldering contains 60% tin and 40% lead.

2. Points to observe when soldering

1. Choose the correct soldering iron for the job;

   - A large soldering iron is used for soldering large areas,
   - A low wattage (i.e. 15-50 watts) soldering iron is used for soldering electronic components.

2. Overheating of the soldering iron causes oxidation of the copper and, as a result, poor heat transfer.

3. Unless it is thermostatically controlled the soldering iron must not be left switched on for long periods of time.

4. If the soldering iron is too cold, poor contact between the material and solder will result.

5. For proper heat transfer, the soldering iron must be clean and tinned. A soldering iron is tinned when the copper is covered with a thin layer of solder.

6. Use the correct type of solder. Resin is the only cleaning agent which may be used when soldering electrical apparatus and cables. Acid causes corrosion of the components and this result in poor connections and damage to the appliance.

7. Solder is usually obtainable as stick solder.

   Core solder of different gauges is also available. This type of solder has a hollow core which is filled with either resin or acid which serves as a cleaning flux.

8. Do not place the hot soldering iron on top of material which may burn and cause a fire.

9. Pick the soldering iron up by the handle. (Severe burns will result if you touch it with your bare hands).

Self test

a) What type of soldering iron is used for electronic components?

b) What is caused by overheating a soldering iron?

c) What happens if the soldering iron is too cold while soldering?

d) What is the meaning of "tinned"?

e) What cleaning agent is used for electrical apparatus and cables?

3. Soldering Components

1. Tinning

   Before objects are soldered together, they must be tinned. This is done by cleaning the contact surfaces (wires) with a knife, emery paper, steel wool, fine wire brush or fine sandpaper. Once cleaned, heat must be applied to the contact surfaces with a soldering iron. Then apply solder which should flow over the contact surfaces in a thin, evenly spread layer. Avoid having an excessive film of solder.

2. Application of heat

   Apply the heated tip of the soldering iron to the connection to be soldered so that maximum heat is transferred. The connection must not be overheated. (See Fig. 1).

3. Application of solder

   - Apply solder to the connection when the temperature of the joint is high enough to readily melt the solder (Fig. 1).

     Solder must not be melted by the soldering iron tip and allowed to flow over the connection.

   - Hold the soldering iron in a position to allow the solder to be applied directly to the heated wire and terminal (Fig. 1).

   - Keep the tip of the soldering iron in contact with the connection until the solder has completely melted and the flux has boiled out.

   - Remove the solder stick from the connection before the soldering iron is removed. If you remove the iron first the solder stick will stick to the terminal.

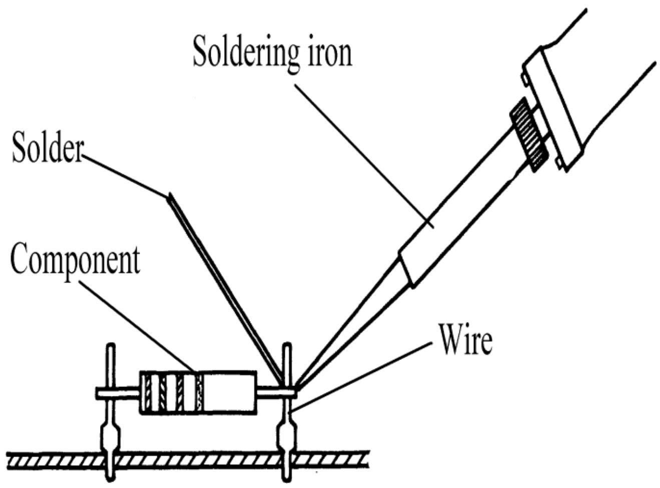


Figure 1

4. The soldered connection

- Apply sufficient solder to form a fillet between the terminal and each side of the wire. (See Fig. 2 and 3).

The contour of the wire must be visible after soldering. Excessive solder which completely overs the wire and terminal is not acceptable.

The boundary of the solder at its junction with the part must be concave, not convex. (See Fig. 2 and 3).

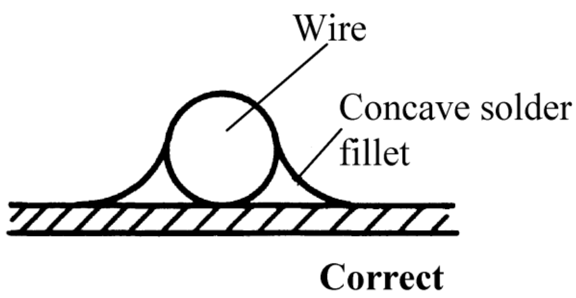


Figure 2

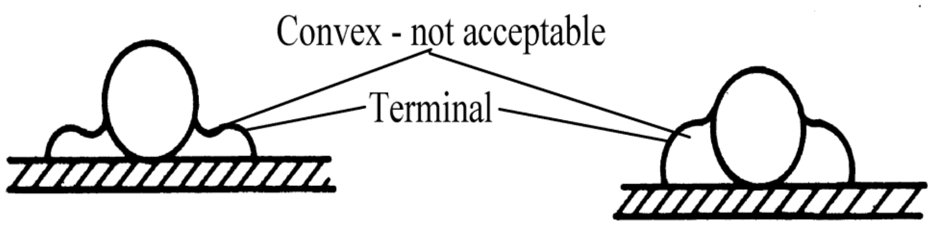


Figure 3

- Use a heat sink as shown in Fig. 4 when recommended. Heat sinks should always be used on component terminal wires between the joint being soldered and the body of the component. They prevent damage to the component due to overheating (i.e. when soldering semi-conductors).

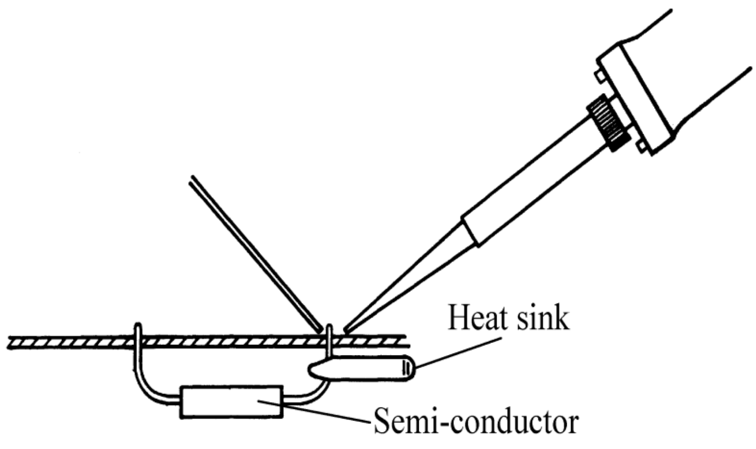


Figure 4

- Heat all the parts that must be soldered together to a temperature above that of the melting point of solder, i.e. the point at which the solder readily melts when applied to the heated surfaces.

If the temperature of the part, e.g. the connection tag, is below the melting point of the solder, the solder will not "wet" it and a "dry joint" will be produced. "Wet" means that the solder is completely fluid and blends with the wire. The land in turn means the surface of which the wire is soldered. (Fig. 5)

The temperature must not be too high because this will result in the wire becoming hard and brittle or in the insulation being burnt.

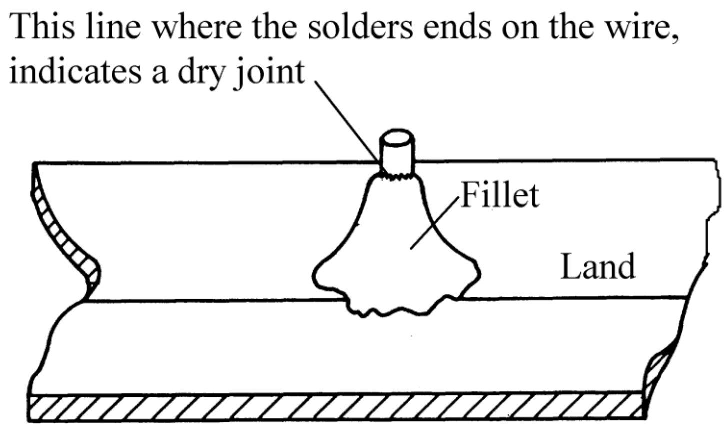


Figure 5

The completed joint should have a bright metallic appearance and should be clean and smooth. There should be no projections or sharp points of solder.

Some of the desirable and undesirable features of soldering are illustrated in Fig. 6 and the display board.

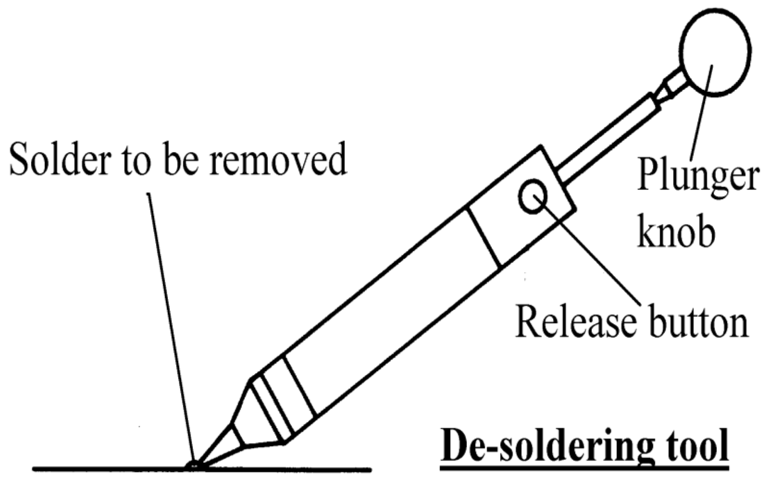


Figure 6

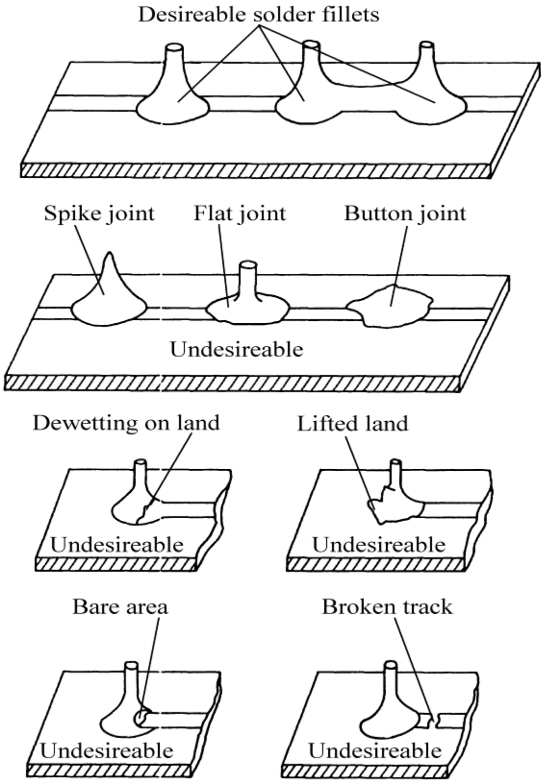


Figure 6

Method of removing solder
(See Fig. 7)

- Heat the solder that must be removed until it melts.
- Push the plunger knob of the suction de-soldering tool in, and apply the tip to the molten solder.
- Push the release button on the side of the suction de-soldering tool; this will cause the solder to be sucked up by the de-soldering tool. If all the solder is not removed, repeat the above until it is.
- Remove the component lead by pulling it away from the soldered surface with a pair of longnose pliers.
- Repeat until all the components are removed.

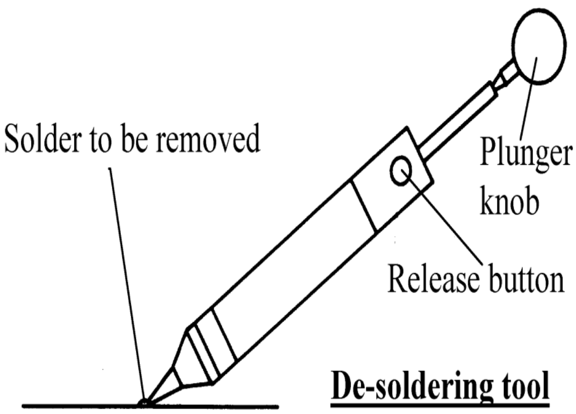


Figure 7

	Practice

Ask your facilitator for a circuit board and the necessary components.

Practice soldering the components to the circuit board according to the procedure described in the notes.

Return the components and circuit board to your facilitator and carry on with the rest of the module.

Ask your facilitator to check your work and if it is correct, then go on to the next section.

Module 2
BE - 1 Identify and use resistors

Purpose of module

This objective will enable the learner to identify and use resistors

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Identify types of resistors and their wattage rating.
2. Name two types of variable resistors, draw their circuit symbols and state what they can control.
3. Decode resistors from their coloured band.
4. Determine the value of resistor with a test instrument.
5. Determine the value of resistors in parallel by calculation and measurement with a test instrument.
6. Determine the value of resistors in series by calculation and measurement with a test instrument.
7. Determine by measurement and calculation, voltage and currents across resistors of different values connected in series.

Procedures Relating To This Module

- Where applicable, your answers must be in writing.
- All the readings measured with the test instrument must be within + 10% of the true value.
- The resistors and their wattage rating must be correctly identified.
- All answers must be correct and in accordance with the module notes.

1. Introduction

There is a wide variety of resistors used in the electrical and electronic field, some of them have a fixed value while the resistance of others can be varied.

The most commonly used types are carbon resistors (Figure 1) and wire wound resistors (Figure 2 and Figure 3). Carbon resistors are generally used in low current applications and wire wound resistors are used to control large currents.

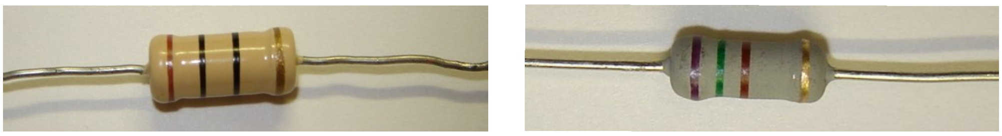


Figure 1

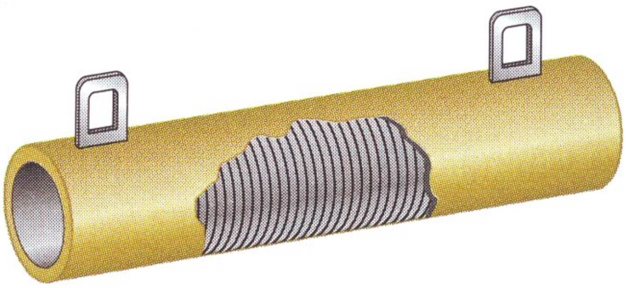


Figure 2

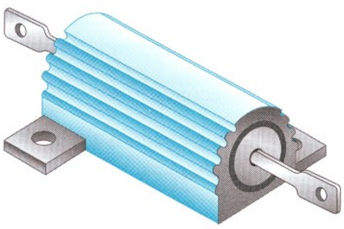


Figure 3

2. *Fixed resistors*

1. Carbon resistors (Fig. 4)

This type of resistor is made by depositing a thin film of carbon on a ceramic rod. The thickness of the carbon film determines the required resistance value. Higher resistance values are obtained by cutting a spiral groove in the carbon film. The electrical path consists of a thin carbon ribbon wound on a ceramic former. The resistor is then coated with a ceramic or enamel coating and the resistance value is printed on the resistor body or marked by a special code.

A brass cap is fitted to each end of the rod, thus making electrical contact with the carbon film. These caps are fitted with short lengths of tinned copper wire which enable the resistor to be connected in a circuit.

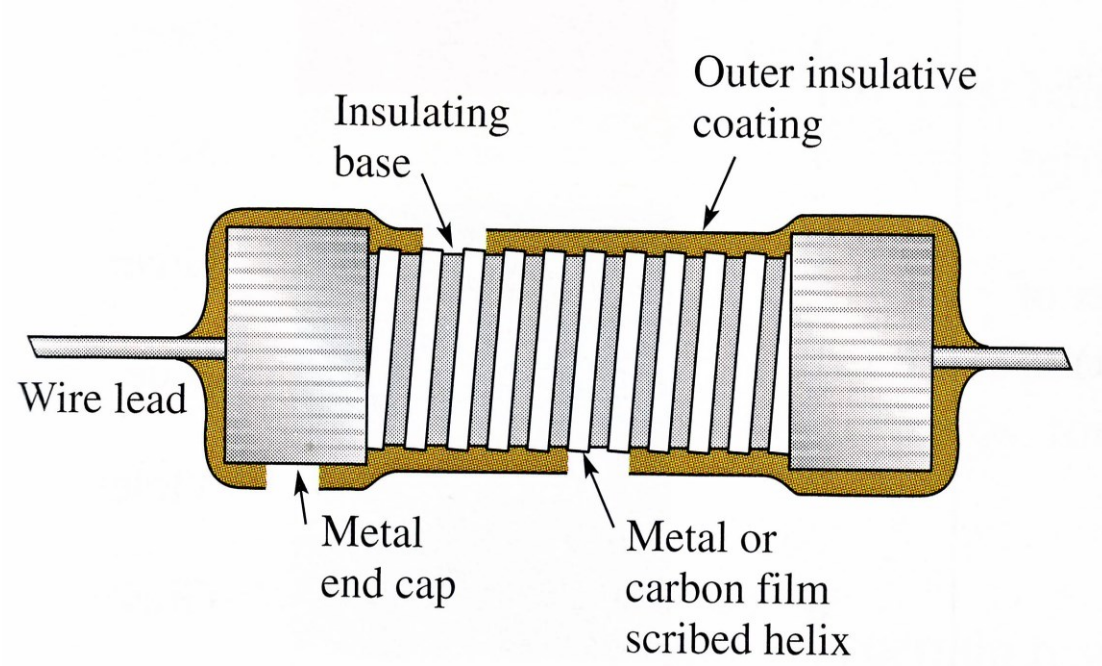


Figure 4 – Film resistor showing spiral technique

2. Wire-wound resistors (see Fig. 5)

This type of resistor consists of a thin wire of resistive metal alloy, wound uniformly on a ceramic former. The resistance value is determined by the thickness of the wire, its length and the specific resistance of the alloy.

The wire-sound former is enclosed in a further layer of ceramic and then baked and glazed.

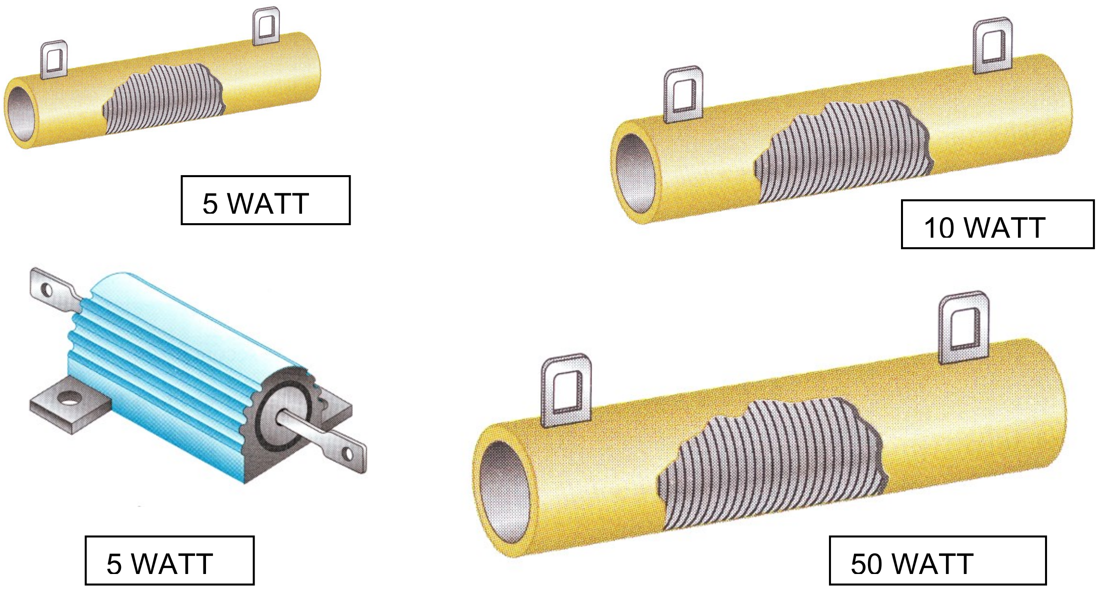


Figure 5

3. Resistor power ratings

Resistors are rated in watts, as well as in ohms of resistance. When the voltage across a resistor forces current through it, heat is generated. To ensure that a resistor is not damaged by excessive heat, the wattage rating of the resistor must always be kept in mind.

Resistors of the same resistance value e.g. 100 ohm, are available in different wattage values. Carbon resistors are commonly made in wattages of ¼, ½, 1 and 2 watts. The higher the wattage rating, the larger the size of the carbon resistor. See Fig. 6 and display board for comparative size of carbon resistors of different wattage ratings.

When resistors of wattage ratings higher than 2 watts are needed, wire-wound resistors are used (Fig. 7 on the next page).

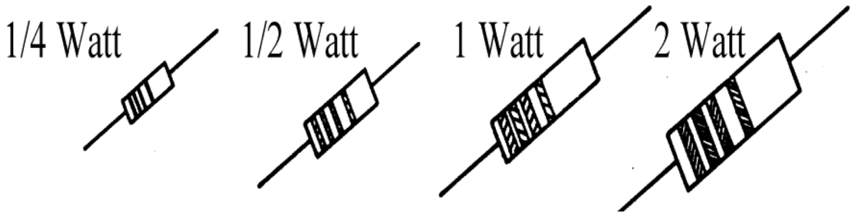


Figure 6 – Resistor power rating

Wire-wound resistors are made in ranges between 5 and 200 watts with special types being used for power in excess of 200 watts.

Remember a 200 ohm ½ watt and a 200 0hm 5 watt resistor will have exactly the same influence on a circuit. It is merely the safety factor of the resistor that differs (½ watt as against 5 watt).

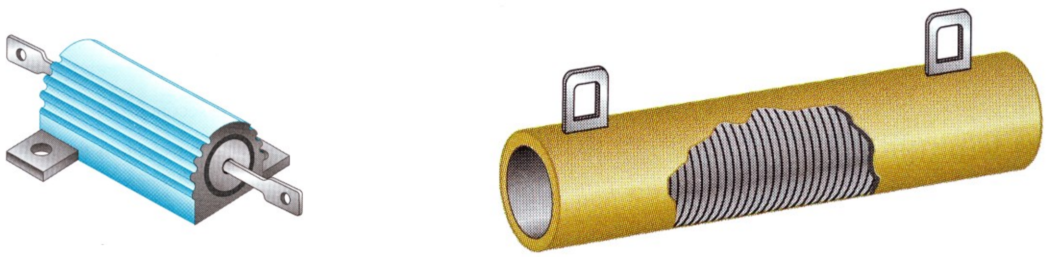


Figure 7

3. *Variable resistors (potentiometers)*

In addition to fixed-value resistors, variable resistors are widely used in electronics. There are mainly two types of variable resistors used in practice, the potentiometer and the preset potentiometer or trimpot. Although the two components look physically different, their operation is the same. Fig. 8 shows what the components look like as well as their circuit symbols. The internal parts and operation is shown in Fig. 9.

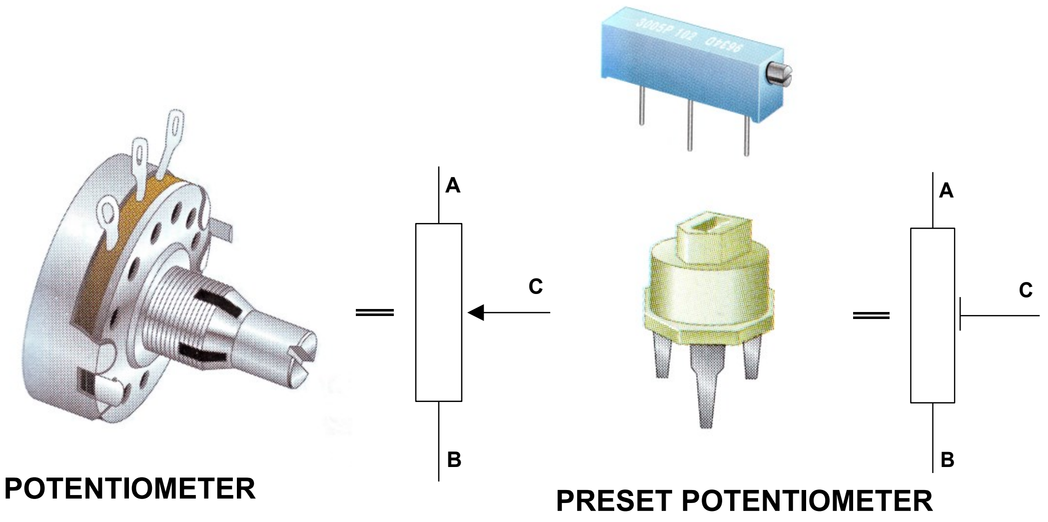


Figure 8

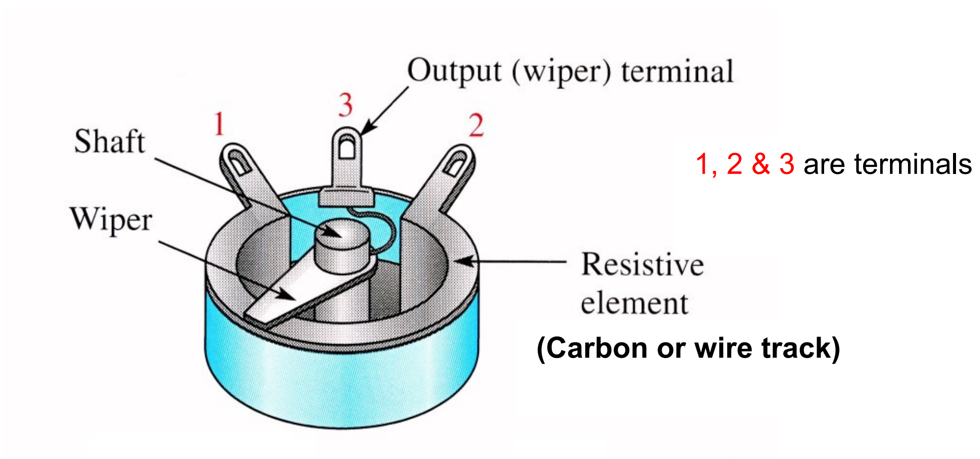


Figure 9

The circuit symbols for both potentiometers (Fig. 10) shows the resistance between points A and B as fixed. Point C is the slider (wiper) arm of the potentiometer. In the case of the potentiometer, the position of the slider arm is changed by means of a shaft that is fixed to the slider arm. In the preset potentiometer, the slider arm is adjusted by means of a small screwdriver.

The slider arm is a metal contact which moves along the surface of the resistance track (carbon for low currents and wire for high currents) selecting different lengths of the resistive surface.

Therefore, the longer the track between points A and C, the higher the resistance between these two points. Similarly the resistance between points B and C varies according to the length of track between points B and C.

A potentiometer is usually connected as a variable voltage divider to control the voltage in a circuit.

A potentiometer may also be used as a rheostat by connecting points A and C, or points B and C together with a piece of wire, which converts it to a two terminal variable resistor. A rheostat is used to control current in a circuit or to a load.

If you understand the section above do the self test on the next page. If not, read and study this section again.

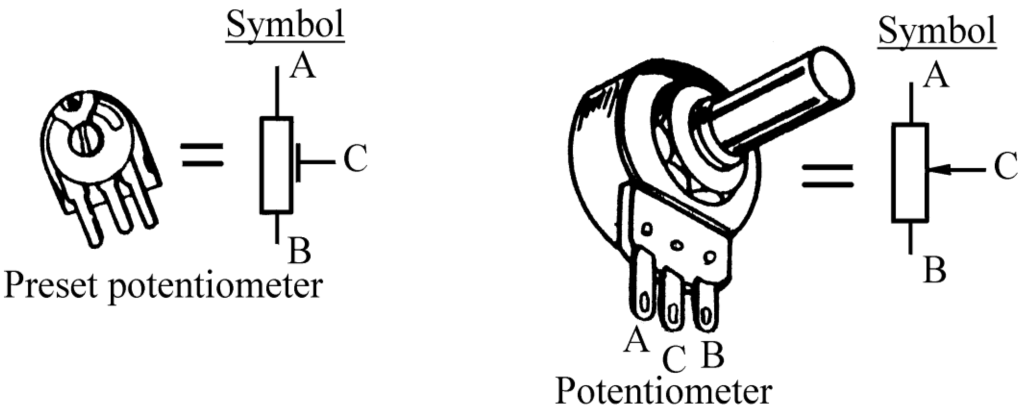


Figure 10

Self test

1. What type of resistor is used for small currents and what type is used for large currents?

2. Explain how carbon and wire resistors are constructed.

3. What would the visual difference be between a ½ watt and a 2 watt resistor?

4. Name two types of variable resistors and draw their circuit symbols.

5. State what each of the two variable resistors mentioned in question 4 controls.

6. What does a rheostat control?

Practice 1

1. Given a variety of resistors, you must practice identifying them and determining their wattage ratings.

If all your answers are correct carry on with the rest of the module, if not, read and study this section again.

4. Colour Code

In circuit diagrams the resistance value in ohms is shown next to the circuit symbol in various ways. For example, a 5600 ohm resistor could be marked on the diagram as 5,6K or as 5K6, although the marking on the actual carbon resistor body is by colour code. Some resistors, however, do have their resistance values printed on them.

It is essential to know this colour code thoroughly, because the misreading of, say 2, 2 kilohm for 22 ohm when building a circuit, could destroy other components.

Four or sometimes five coloured bands are used. Three of the bands are spaced fairly close together and close to one end. This is the end from which the decoding should begin (see Fig. 11).

The first band gives the first numeral of the resistance value, the second band the second numeral, and the third the multiplier (i.e. the number of noughts to add to the first two digits). The fourth band gives the tolerance. When there is no fourth band it means that there is 20% tolerance. In the case of five bands, the first three bands indicate the first three numerals, the fourth band indicates the multiplier (i.e. the number of noughts) and the fifth band indicates the tolerance (see Fig. 12). More details of this coding system are shown in Table 1 on the next page.

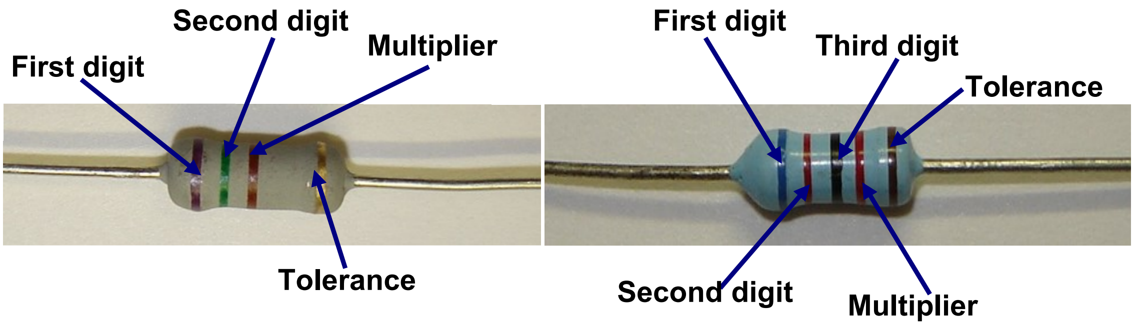


Figure 11

Examples of this coding are shown in Fig. 12.

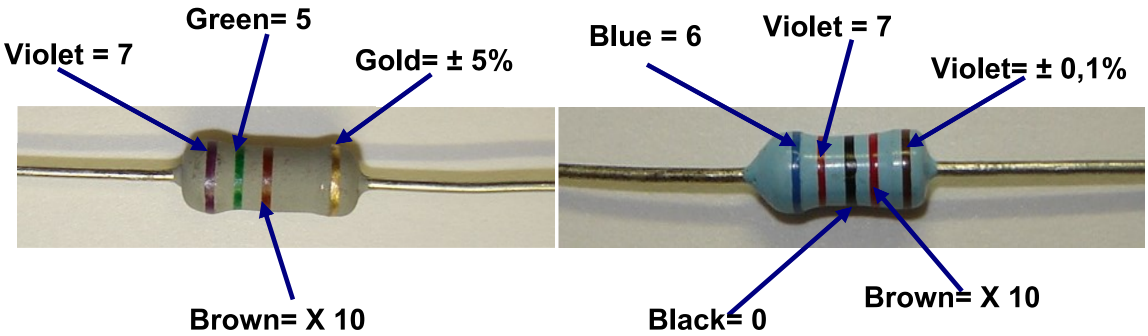
Notice how 27 500 has been abbreviated to 27K5. This method is adopted for resistors and other examples are shown below.

Examples:

4 700=4K7390 000 = 390K
56 000=56K1 800 000 = 1M8
56 500= 56K5

COLOUR	VALUE	MULTIPLIER	TOLERANCE
Black	**0**	**X 1**	**-**
Brown	**1**	**X 10**	**± 1%**
Red	**2**	**X 100**	**± 2%**
Orange	**3**	**X 1 000 = 1K**	**-**
Yellow	**4**	**X 10 000 = 10K**	**-**

Green	5	X 100 000 = 100K	± 0, 5%
Blue	6	X 1 000 000 = 1M	± 0, 25%
Violet	7	-	± 0, 1%
Grey	8	-	-
White	9	-	-
Silver	-	0, 01	± 10%
Gold	-	X 0, 1	± 5%
No colour	-	-	± 20%



75X10 ±5%
=750R ± 5%
Or
750Ω ± 5%

6 7 0 X10 ± 0,1%
=6700Ω ± 0,1%
Or
67K ± 0,1%

Figure 12

5. *Resistor tolerance*

It is difficult to make a resistor to an exact value. Fortunately, in most cases approximate values of resistance will do. Carbon resistors can be obtained with tolerances of $\pm$ 20%, $\pm$ 10%, $\pm$ 5%, $\pm$ 2% and $\pm$ 1%.

If the decoded value of a resistor is 120 ohms $\pm$ 5% tolerance, it would mean that the resistor's value could be between 120 + 5% = 126 ohms and 120 - 5% = 114 ohms and with 10% tolerance between 120 + 10% = 132 ohms and 120 - 10% = 108 ohms.

The following bands are used for indicating tolerances:

Gold band denotes 5% tolerance
Silver band denotes 10% tolerance
No band denotes 20% tolerance

If you understand the section above on colour coding, do the self test on the next page without referring to your notes. If not, study the section again until you understand it.

The method below may assist you in remembering the colour code.

0-Black-Black

1-Brown-Beatles

2-Red-Running

3-Orange-Over

4-Yellow-Your

5-Green-Garden

6-Blue-Brings

7-Violet-Very

8-Grey-Good

9-White-Wheat

Self test

1. Write down the resistance values and tolerances for the following colour bands.

   a) Orange, White, Red, Red

   b) Green, Blue, Green, Brown

   c) Yellow, Violet, Yellow, Gold

   d) Red, Violet, Brown

   e) Grey, Red, Black

   f) Red, Red, Green, Orange, Silver

2. Write down the colour code for the following resistors.

   a) 6K810%

   b) 47K51%

   c) 100 5%

d) 1M20%

e) 1K5%

f) 47Ω2%

g) 10Ω5%

h) 180Ω20%

i) 2K21%

Check your answers with those given on the next page.

	Answers to Self test 2

1. The resistance values and tolerances for the colour bands.

   a) 3K9 with 2% tolerance.

   b) 5M6 with 1% tolerance.

   c) 470K with 5% tolerance.

   d) 270 Ohms with 20% tolerance.

   e) 82 Ohms with 20% tolerance.

   f) 225K with 10% tolerance.

2. The colour code for the resistors.

   a) Blue, grey, red, silver.

   b) Yellow, violet, green, red, brown.

   c) Brown, black, brown, gold.

   d) Brown, black, green.

   e) Brown, black, red, gold.

   f) Yellow, violet, black, red.

   g) Brown, black, black, gold.

   h) Brown, grey, brown.

   i) Red, red, red, brown

6. Resistors between 1 and 10 ohms

For resistors between 1 and 10 ohms the third colour band (multiplier) is gold, which according to the colour chart previously given is 0,1. Therefore if a resistor was marked orange, orange, gold, gold, the first band is the first digit (3), the second band the second digit (also 3) and the third band the multiplier (0,1) the fourth band is the tolerance (5%). Thus 33 x 0, 1 = 3, 3 ohms with 5% tolerance.

Resistances less than 1K are also written as 3, 3 ohm = 3R3, 100 ohms = 100R, 4, 7 ohms = 4R7, 390 ohms = 390R, 56 ohms = 56R, etc.

7. Resistors between 0 and 1 ohm

For resistors between 0 and 1 ohm the third colour band (multiplier) is silver which is 0, 01 on the colour chart.

Looking at a resistor marked blue, grey, silver, gold, the first band would indicate the first digit (6), the second band the second digit (8) and the third band the multiplier (0,01). Therefore 68 x 0, 01 = 0, 68 ohms with 5% tolerance.

	Self test3

1. Write down the resistance and tolerance values for the following colour bands.

    a) Red, Red, Gold, Gold

    b) Yellow, Violet, Silver, Red

    c) Brown, Black, Gold, Gold

    d) Brown, Green, Gold, Gold

2. Write down the colour code for the following resistors.

    a) 5R610%

    b) 3R35%

    c) 1R25%

    d) 0,39Ω 2%

    e) 0,82Ω 1%

	Check your answers with those given on the next page.

	Answers to Self test 3

1. The resistance and tolerance values for the colour bands.

   a) 2R2 with 5% tolerance.

   b) 0,47Ω with 2% tolerance.

   c) 1R with 5% tolerance.

   d) 1R5 with 5% tolerance.

2. The colour code for the resistors.

   a) Green, blue, gold, silver

   b) Orange, orange, gold, gold.

   c) Brown, red, gold, gold.

   d) Orange, white, silver, red.

   e) Grey, red, silver, brown.

Practice

If all your answers to self test 3 are correct, ask your facilitator to supply you with 10 different resistors and a multimeter. Decode the resistor values and complete the table below. Also measure the resistance values with the multimeter and insert the measured values in the table. (If you want to refresh your memory about using the multimeter refer back to module TI-2).

	1	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**
1st Colour										
2nd Colour										
3rd Colour										
4th Colour										
5th Colour (if any)										
Coded Value										
Tolerance										
Measured Value										

If all your answers are not correct do the practice over again.

8. *Parallel and series connections of resistors*

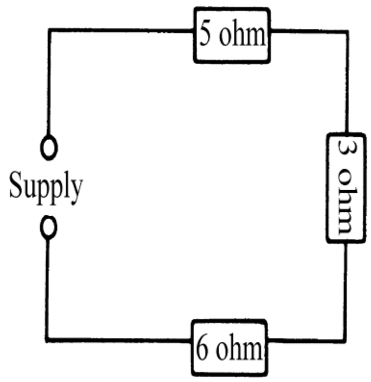
You have already studied parallel and series connections of resistors in module EU-T. Therefore you should be able to recognise and fill in the type of circuit shown in Fig. 13 and 14 and calculate the total resistance for each circuit.

Calculate and show the formulae used for the calculation of resistance in each circuit and check your answers with those shown below**.**

ANSWERS

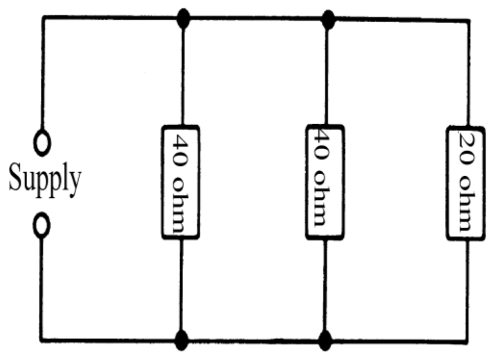
1. 14 Ohms.

2. 10 Ohms.



Resistors connected in ________________

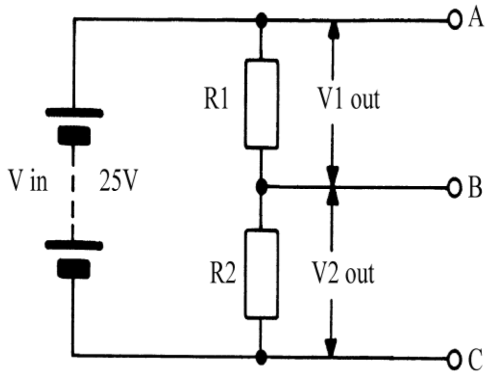
Figure13



Resistors connected in ________________

Figure 14

9. *The voltage divider (see fig. 15)*



Fixed voltage divider

Figure 15

A resistive voltage divider may be used when a certain component requires a lower DC voltage than the supply. The simplest DC voltage divider consists of two resistors R1 and R2 connected in series, across which a DC voltage (V) is applied.

Assume that in Fig. 15 the applied voltage (VIN) is 25 volts DC and that the values of R1 and R2 are 100 and 150 ohms respectively. The total resistance for this circuit is 100 + 150 = 250 ohms respectively. The circuit voltage is 25 volts. Consequently, Ohm's law can be used to calculate current.

Thus

$$I = \frac{VIN}{Rt} = \frac{25}{250} = 0{,}1\ Ampere$$

To calculate the voltage (V1) across R1, you know that R1 = 100 ohm and that the current passing through it is 0, 1 amps. (Remember that the current is the same throughout a series circuit). So:

V1=I x R
=0,1 x 100
V1=**<u>10 volts</u>.**

Similarly

voltage V2 across R2
=0,1 x 150
V2=**<u>15 volts</u>**.

The above calculation can be proved by experiment.

	Experiment 1

To carry out this experiment you need:

A 25 volt DC supply

A voltmeter (0-30 volt DC)

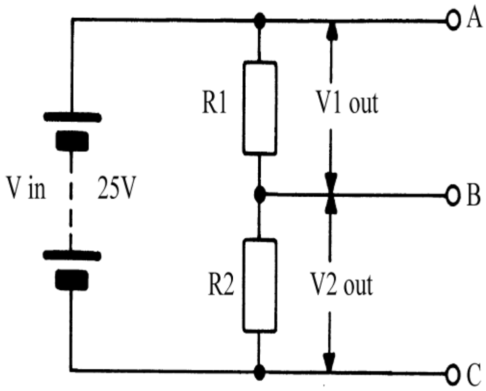
A 100 ohm 2 or 5 watt resistor

A 150 ohm 2 or 5 watt resistor

A circuit board

1. Connecting wires.

   Construct the fixed voltage divider circuit on the circuit board as shown in Fig. 15 and measure the voltages over AB and BC as shown in Fig. 15.



Voltage divider

Figure 15

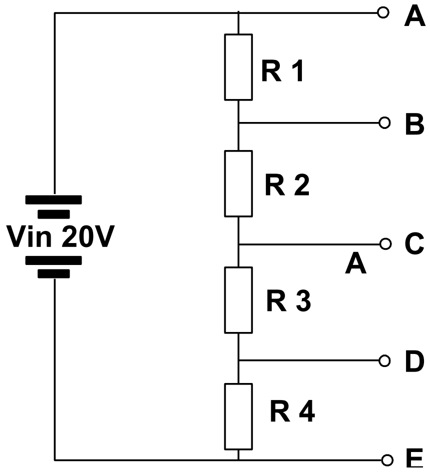


Figure 16

Are your measured voltages the same as those calculated on the previous page ?

- Change the resistors to the values given below and measure the voltages again over AB and BC as shown in Fig. 15. Change the supply voltage to 20V.

1.1 R1=10 K ohm ½ watt,
R2=1 K ohm ½ watt.

1.2 R1=330 ohm ½ watt,
R2= 380 ohm ½ watt.

	In (c) below there are two additional resistors in circuit, showing that, if needed, the voltage can be divided into more than 2 values. Construct the circuit as shown in Fig. 16

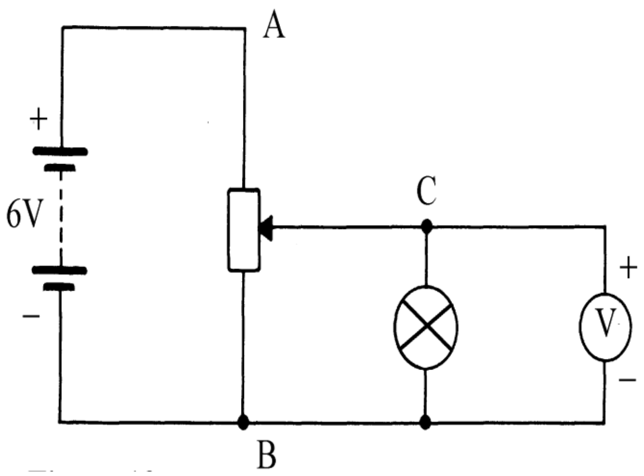
Check your answers against those given below.

1.3 R1 and R2 = 100 ohm 2 or 5 watt,

R3 and R4 = 150 ohm 2 or 5 watt.

Are your readings the same as those given ? If not do the experiments over again. If all your readings are correct continue to Experiment 2.

When it is necessary to develop a voltage that cannot be achieved by a combination of fixed resistors, a potentiometer is used. By manually adjusting the position of the sliding arm any value between zero and the total voltage (supply voltage) may be obtained. See Fig. 17.



Variable potential divider
Figure 17

	Experiment 2

To carry out this experiment you need:

A 6 volt DC supply

A voltmeter (0 -10 volt DC)

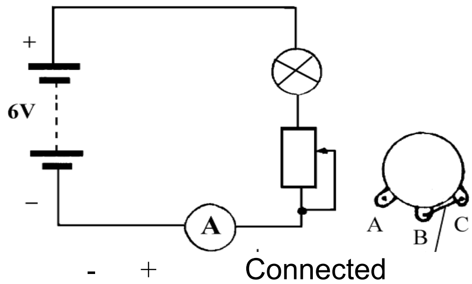
A 100 ohm 1 watt potentiometer

A 6 volt lamp

A circuit board

1. Connecting wires.

- Construct the variable voltage divider circuit as shown in Fig. 18.



Potentiometer connected as a rheostat
Figure 18

<u>Answers to (1.3) above</u>

1. Across R1 = 18,2 volts, across R2 = 1,8 volts
2. Across R1 = 9,0 volts, across R2 = 11,0 volts
3. Across R1 = 4,0 volts, across R2 = 4,0 volts

   Across R3 = 6,0 volts, across R4 = 6,0 volts

- Turn the potentiometer knob in an anti-clockwise direction until it stops.
- Switch on the supply and check the voltage reading on the voltmeter. The reading should be 0 volts. If it is not, change leads A and B around.
- Slowly turn the potentiometer knob in a clockwise direction. Notice that the voltage increases, as well as the intensity of the lamp.

2. Conclusion

A potentiometer is a variable voltage divider with voltage values anywhere between 0 volts and the supply voltage across the potentiometer. Therefore, a potentiometer controls the circuit voltage.

	Experiment 3

To carry out this experiment you need:

A 6 volt DC supply

An ammeter (0-300mA)

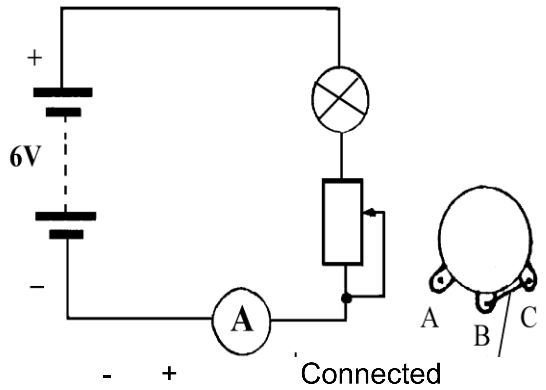
A 500 ohm 1 watt potentiometer

A 6 Volt lamp

A circuit board

1. Connecting wires.

- Construct the circuit as shown in the Figure B and C are connected together, so that the potentiometer may be used as a rheostat.
- Turn the rheostat knob in an anti-clockwise direction until it stops.
- Switch on the supply and check the current reading. The reading should be in the region of 10 milli Ampere; if it is not, change leads A and B around.
- Slowly turn the rheostat knob clockwise. Notice that the current increases, as well as the intensity of the lamp.



Potentiometer connected as a rheostat

Figure 18

If your experiments have been successful, switch off the supply and remove the components from the circuit board.

	Ask for the criterion test when you feel ready.

Module 3
BE - 2 Identify and use capacitors

Purpose of module

This objective will enable the learner to identify and use capacitors

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Identify 6 different types of capacitors from the capacitors supplied.
2. Give the abbreviations and the value in farads of :-

   (a) Microfarad,

   (b) Picofarad,

   (c) Nanofarad.
3. Decode a capacitor using the colour code chart.
4. Calculate the time constant of a RC time delay circuit.
5. Calculate the total capacitance of capacitors connected in series and in parallel.

Procedures Relating To This Module

- Where applicable, all answers must be in writing.
- All answers must be correct and in accordance with the module notes.

1. Introduction

The capacitor is a component that stores an electrical charge. It can be compared to a water tank which stores water. (See Fig. 1).

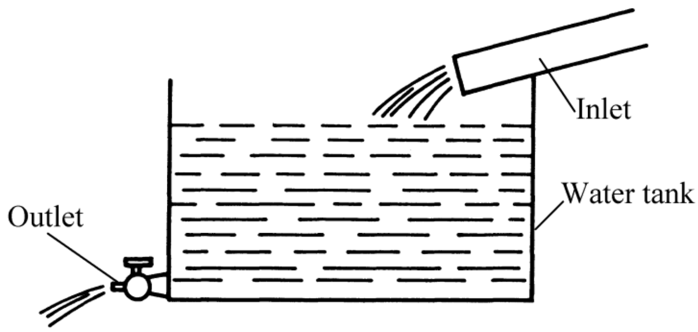


Figure 1

The larger the water tank, the more water it can store.

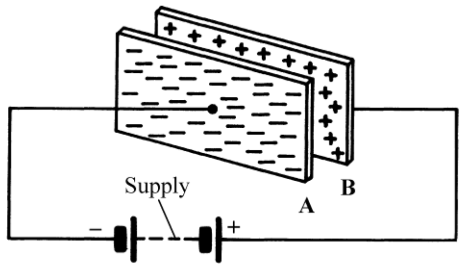


Figure 2

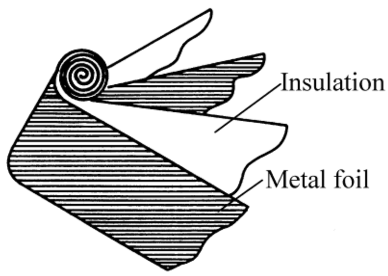


Figure 3

A capacitor consists of two conducting plates which are separated by an insulating or non-conducting material. (See Fig. 2).

When a capacitor is connected to a direct current source it is charged with an electrical charge which accumulates on the two plates. In Fig. 2 electrons (negative charge) flow from the supply source to plate A. Electrons are repelled from plate B and are attracted to the positive terminal of the battery, in this way charging the capacitor.

The larger the capacitance rating of a capacitor the more charge it can accumulate. Because the capacitance of a capacitor depends on the area of the plates, very large plates are required for large capacitances. To limit the size of the capacitor, these plates, which are strips of metal foil, are wound spirally, forming large surface areas as shown in Fig. 3.

Capacitors come in all shapes and sizes and there are fixed and variable types.

2. *Fixed capacitors*

The most common types of fixed capacitors are shown below.

1. Paper

   This type consists of two layers of thin metal foil separated by wax paper (Fig. 4).

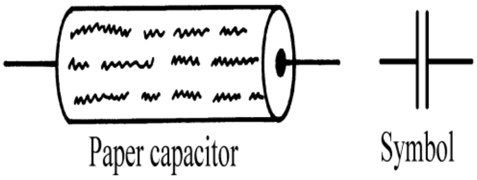


Figure 4

2. Mica

   This type consists of thin metal plates separated by thin sheets of mica (Fig. 5).

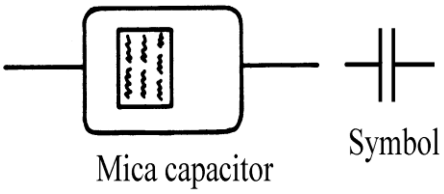
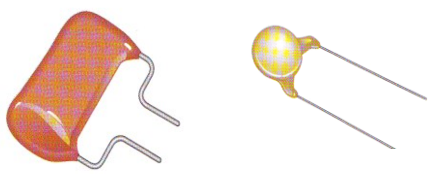


Figure 5

3. Ceramic

   These are small capacitors consisting of metal film deposits on each side of the ceramic. The ceramic is the dielectric (insulation) (see Fig. 6).



Ceramic capacitors

Figure 6

4. Polycarbonate

This is a recent product in the field of plastic insulating materials. A film of polycarbonate can be produced as thin as 2 micrometers ($=2x10^{-6}$m). It is coated with aluminium and wound to form the capacitor elements. (See Fig. 7).

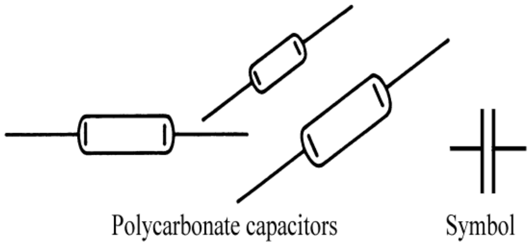


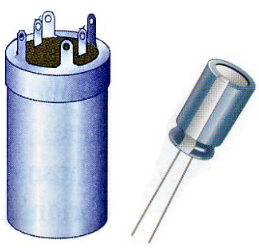
Figure 7

5. Electrolytic

For capacitance values greater than 1μF, the physical size of the capacitor increases with the value of the capacitor. (See Fig. 8).

Unlike other types of capacitors, the electrolytic capacitor is "polarized". When it is connected in the wrong polarity, it will break down and act as a short circuit.

The most important property of a capacitor is its working voltage i.e. if the circuit is a 12 volt system never use with a lower rating than 12 volt. If, for example, you were to use a 3 volt capacitor in a 12V system, the voltage would be too high and it would break down the insulation and damage the capacitor.



Electrolytic capacitors

Figure 8

3. Variable or air capacitor

These capacitors consist of one or more metal plates fixed in position and known as the stator (see Fig. 9).

A second set of plates, called the rotor, can be rotated so that they interleave between each stator plate. The dielectric is air. They are widely used in tuning circuits, e.g. radios.

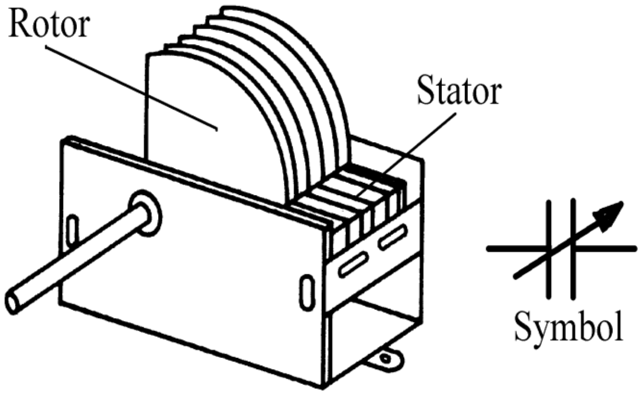


Figure 9

4. Capacitor values

The unit of capacitance is the farad (F) which indicates the amount of electrons stored on the plates of the capacitor. The farad is a very large unit, so the following sub-multiples are used :

The microfarad (abbreviated μF) = 1 millionth of a farad = 10^{-6}F.

The nanofarad (abbreviated nF) = 1 thousand millionth of a farad = 10^{-9}F.

The picofarad (abbreviated pF) = 1 million millionth of a farad = 10^{-12}F.

	Self test 1

Answer the questions below without referring to your notes.

1. Identify six types of capacitors from the capacitors supplied.

   ____

   ____

2. How are capacitors with large plate areas kept physically small?

   ____

   ____

3. Give the abbreviation and the value in farads of the following:

   a) Microfarad = ______________________________

   b) Picofarad = ______________________________

   c) Nanofarad = ______________________________

5. *Capacitor colour code*

Some capacitors e.g. flat film and tantalum types use a colour code to identify the capacitance of the capacitor. See Fig.10.

In Fig.10 the uppermost (top band) indicates the first digit, the second band the second digit and the third band is the multiplier. The fourth band indicates the capacitor tolerance. Notice that the same method is used as for resistors except that the bottom coloured band on the capacitor indicates the maximum voltage allowed. Table 1 on page 39 shows the colours used for the flat film type capacitors.

Examples are shown in Fig11 and 12 for decoding the flat film type capacitors, using the colour code chart.

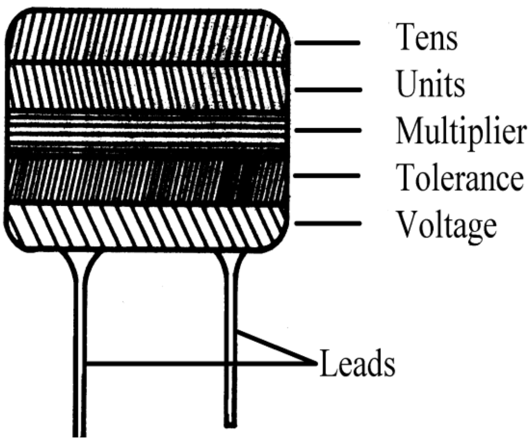


Figure 10

The capacitor can now be decoded from Fig. 11, therefore = 22 x 1 = 22pF at 1% tolerance and a permissible voltage of 400 volts. (The value for these capacitors is always in picofarads. (pF)). Also 68 x 100 = 6 800pF = 6,8nF at 10% tolerance and a permissible voltage of 250 volts. (Fig. 12).

	It is often found that a capacitor has a wide band of one colour equal to the width of two bands. This is in fact two bands which are both of the same colour. If you understand the capacitor colour coding in this section, do the self test on the next page.

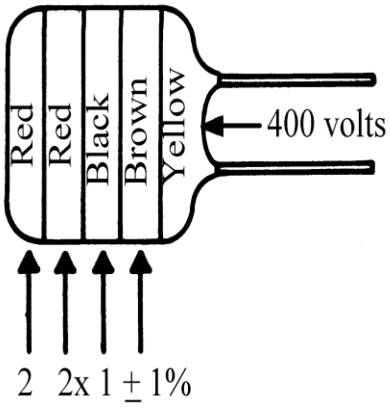


Figure 11

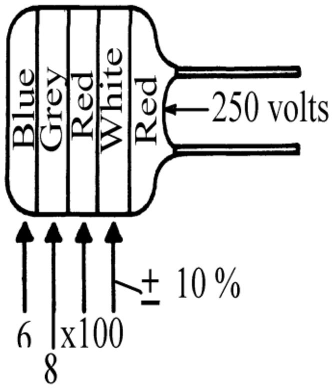


Figure 12

TABLE 1 - CAPACITOR COLOUR CODE

(a)	(b)	(c)	(d)	(e)
Black	**0**	**1**	**20**	**-**
Brown	**1**	**10**	**1**	**100**
Red	**2**	**100 = (10^2)**	**2**	**250**
Orange	**3**	**1000 = (10^3)**	**2,5**	**-**
Yellow	**4**	**10 000 = (10^4)**	**-**	**400**
Green	**5**	**100 000 = (10^5)**	**5**	**-**
Blue	**6**	**1 000 000 = (10^6)**	**-**	**630**
Violet	**7**	**-**	**-**	**-**
Grey	**8**	**-**	**-**	**-**
White	**9**	**-**	**10**	**-**

(a)=Colour

(b)=Tens and units

(c)=Multiplier (pF)

(d)=Tolerance (± %)

(e)=Voltage

Self test

Answer the following questions without referring to your notes.

DO NOT CHEAT because you will have to decode capacitors in the criterion test later.

Decode the following capacitors (all colours are from the top).

a) Red, Violet, Yellow, Black, Yellow = ______________________

b) Green, Blue, Yellow, Red, Red = ______________________

c) Grey, Red, Orange, Green, Brown = ______________________

d) Orange, Orange, Red, White, Blue = ______________________

Give the colours that would be painted on the capacitors for the following values.

e) 12 000 pF ± 5% at 100 volts.

f) 3 900 pF ± 20% at 400 volts.

Be careful with these values as these capacitors are always rated in picofarads and not microfarads, therefore you will have to convert the values. (Module UP-T)

g) 0,018 μF ± 2% at 630 volts

____

h) 0,39 μF +- 1% at 250 volts

____

Check your answers against those given on the next page.

Answers to Self test 2

a) 270 000 pF at ± 20% tolerance and 400 volts.

b) 560 000 pF at ± 20% tolerance and 250 volts.

c) 82 000 pF at ± 20% tolerance and 100 volts.

d) 3 300 pF at ± 20% tolerance and 630 volts.

e) Brown, red, orange, green, brown.

f) Orange, white, red, black, yellow.

g) Brown, grey, orange, red, blue.

h) Orange, white, yellow, brown, red.

In order to develop a better understanding of capacitors, do the experiments on the following pages.

6. *Capacitors in dc circuits*

	Experiment 1

To carry out this experiment you need:

- An electrolytic capacitor of 100 µF 25 volts.
- A voltmeter with 20 volt full scale deflection (F.S.D.)
- A 12 volt DC supply.
- A circuit board or deck on which to mount the components.
- A single pole with a single throw toggle switch.
- Connecting wires.

1. Procedure

- Wire the circuit on the board as shown in the circuit diagram. (Make sure that the capacitor and meter polarity are correct, i.e. as shown in Fig. 13 and that the switch is in the off position).
- Switch on the supply and notice that the meter almost instantly reads the same voltage as the supply.
- Switch off the supply and see if the voltmeter still reads 12 volts, indicating that a capacitor can store an electrical charge. The meter reading will decrease towards zero volts as time passes. This is due partly to the internal leakage of the capacitor and mainly to the meter across the capacitor through which the capacitor discharges.
- To prove that the meter is mainly responsible for the leakage, disconnect the meter, switch the supply on and off again, and leave for one or two minutes. Take the reading across the capacitor momentarily with the voltmeter and notice that the voltage reading will still be high.

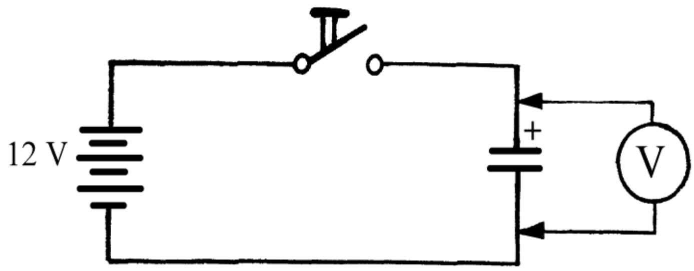


Figure 13

2. Conclusion

The capacitor is a component that can store an electrical charge.

3. Capacitor time constant

All previous experiments in electricity and electronics have been concerned with measuring current and voltage. A new dimension is now considered when experimenting with capacitors, namely time. When referring to time in electronics we immediately think of practical applications like:

(i) A security light that switches on for a certain time after it has been activated and then switches off;

(ii) Warning lights that flash on and off at a specific rate;

(iii) A security door that unlocks for a predetermined time and then locks again.

In all of the abovementioned examples, a capacitor combined with a resistor is used to form an RC time delayed circuit to determine the time or rate in electronic circuits.

We will make use of the RC time delayed circuit in various other modules of this course.

Experiment 2

To carry out this experiment you need:

- An electrolytic capacitor of 1 000 μF 25 volts.
- Two resistors (4K7 and 10K).
- A voltmeter with 20 volt FSD.
- A single pole single throw switch.
- A 12 volt DC supply.
- A circuit board or deck on which to mount the components.

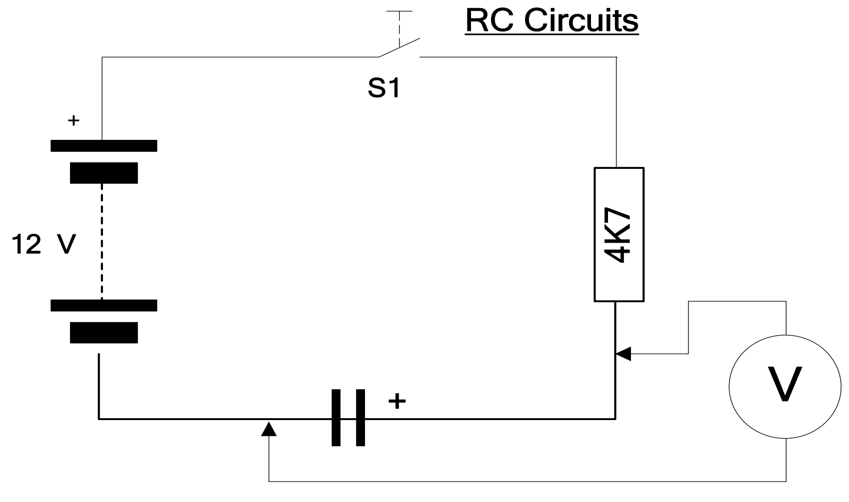


Figure 14

1. Procedure

- Wire the circuit on the board as shown in Fig. 14.

The switch must be in the off position

- Close switch S1 and notice that the meter takes almost 5 seconds to reach 8 volts on the meter and about 25 seconds to reach 12 volts.

If we note down the voltage values measured every half a second from the moment S1 is closed, and these voltage values are plotted as a graph, we would get a curve similar to the one shown in Fig. 15.

This characteristic curve is known as the charge curve of a capacitor.

The time constant of an RC time delayed circuit, like the one in Experiment 2 can be calculated using the formula T = RC.

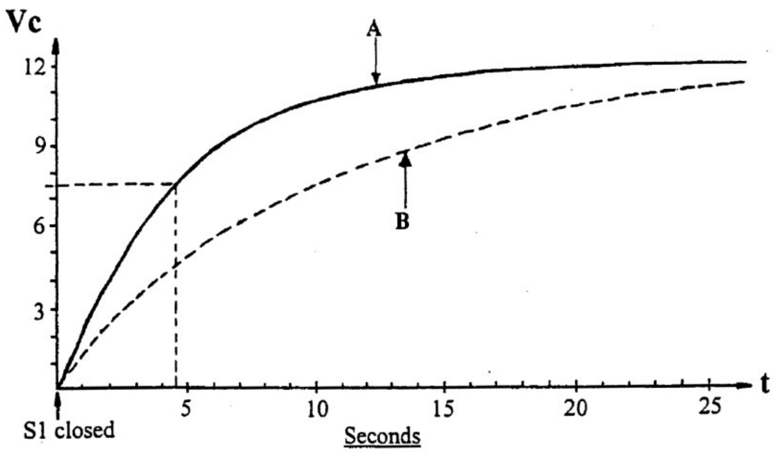


Figure 15

Where:

T = time in seconds

C = capacitance in farads

R = resistance in ohms

The time constant "T" is the time it takes for the capacitor to charge to a voltage which is 63% of the supply voltage.

- Calculate 63% of the supply voltage in Experiment 2

  = ... volts

- Calculate the time constant of the RC time delayed circuit from Experiment 2:-

  T = ... x..........

  = ... x..........

  = ... seconds

- Compare these two answers with the values indicated by the dotted lines on curve A of Fig. 15.

- Close switch S1 and notice that the meter now takes much longer to reach the final voltage (CR is now increased by a factor of just over 2 times) e.g. T = 0,001x10 000 = 10 seconds.

- Switch off the supply and replace the 4K7 resistor with a 10K resistor. The capacitor must be discharged by connecting a 100 Ω resistor across the capacitor terminals for a few seconds.

- Indicate the T = RC point by making an X on curve B of Fig. 15.

  It will be seen from the formula T = R x C that either R or C may be varied to change the time.

2. Conclusion

A "time constant" (RC) is not really a fixed unit of time such as a second, but is a relative unit of time of which the actual value in seconds is determined by the values of R and C in the charging circuit.

If you understand the experiment do the self test on the next page

Self test

1. What does a capacitor "store"?

___

2. Calculate the time constant for the following:

   a) A 220 μF capacitor and a 47K resistor.

   ___

   b) A 100 μF capacitor and a 68K resistor.

   ___

   c) A 1 000 μF capacitor and a 2K2 resistor.

   ___

Check your answers with those given on the next page.

Answers to self test 3

1. An electrical charge.

2. The time constant

   a) 10, 3 seconds.

   b) 6, 8 seconds.

   c) 2, 2 seconds.

7. Capacitors in AC circuits

One of the characteristics of the capacitor is its opposition to changes in voltage e.g. as in alternating current. The slower the changes (the lower the frequency), the bigger the opposition offered by the capacitor and vice versa.

Therefore in direct current circuits the fully charged capacitor has an infinite resistance and no current can flow through it and in AC circuits a certain amount of current will flow depending on the frequency.

This "resistance" to current flow caused by the alternating current is called IMPEDANCE. When only capacitance is in the circuit, then:

IMPEDANCE (Z) = CAPACITIVE REACTANCE (X_c)

Both impedance and capacitive reactance resist current flow and their units of resistance are therefore in ohms (Ω).

There are other components that also have impedance and they will be examined later in this course.

To see the effect of capacitance in an AC circuit carry out the following experiments.

	Experiment 3

To carry out this experiment you need:

- A 12 volt AC supply.
- Two 12 volt filament lamps.
- A 50 µF 63 volt electrolytic capacitor and a 100 µF 63 volt electrolytic capacitor.
- A single pole single throw switch.
- A circuit board or deck on which to mount the components.
- Connecting wires.

1. Procedure

- Wire the circuit on the board as shown in Fig. 16. (Note that the switch is in the off position).

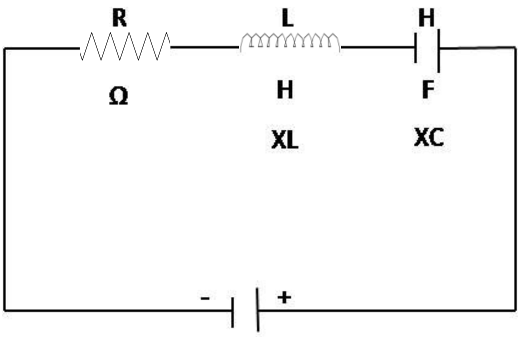


Figure 16

$$Z = \sqrt{R^2 + (XL - XC)^2}$$

- Switch on the supply and compare the light intensity of the lamps 1 and 2. It can be seen that lamp 1 is connected directly over the 12 volt AC supply and lamp 2 is connected in series with the 50 µF capacitor across the supply.

  Is lamp 2 as bright as lamp 1? ___________________________

  Is lamp 2 duller than lamp 1? ___________________________

- Switch off the supply and replace the 50 µF capacitor with a 100 µF 63 volt capacitor.

- Switch on the supply and again compare the light intensity of the two lamps.

  Is lamp 2 brighter than with the 50 μF capacitor ?

  ____

  Is lamp 2 duller than with the 50 μF capacitor ?

  ____

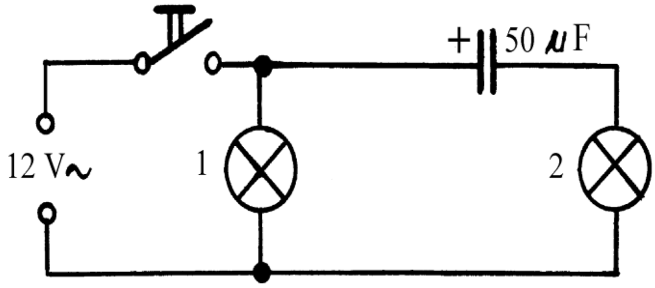


Figure 16

2. Conclusion

At the supply voltage frequency (50 Hz), the smaller capacitor has a higher resistance than the larger capacitor. (Remember that the brighter the lamp glows the higher the current that flows through it and thus the lower the "resistance "of the capacitor).

Therefore capacitance in an AC circuit offers a resistance to voltage changes.

Self test

1. What change does a capacitor oppose in an AC circuit?

2. What is this "resistance" known as? Give its unit and symbol?

3. When the circuit contains capacitance only, what is its unit and symbol?

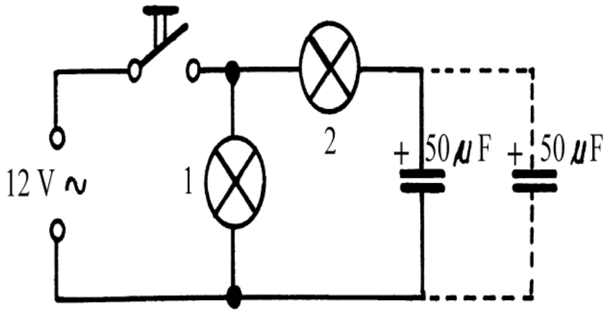
4. Which has a higher "resistance" in an AC circuit, a large or a small value capacitor?

8. *Capacitors connected in series and in parallel*

	Experiment 4

To carry out this experiment you need:

- A 12 volt AC supply.
- Two 12 volt filament lamps.
- Two 50 μF 63 volt electrolytic capacitors.
- A single pole single throw switch.
- A circuit board on which to mount the components.
- Connecting wires.



ure 17 – Capacitors in parallel

1. Procedure

- Wire the circuit on the board as shown in Fig. 17. Exclude the dotted line circuit.
- Switch on the supply and take note of the light intensity of lamp 2.
- Connect another 50 μF capacitor in parallel (as shown by the dotted lines) with the first capacitor already in circuit.

  Is the lamp brighter? ______________________

  Is the lamp duller? ______________________

2. Conclusion

When capacitors are connected in parallel the capacitance increases. (Remember from your previous experiment that the larger capacitance caused the lamp to glow brighter).

The method for calculating the total capacitance for capacitors in parallel is :

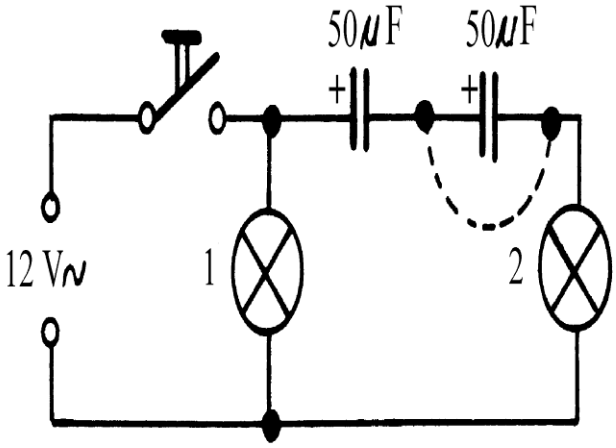
$$Ct = C1 + C2 +$$

Using the same components and equipment as in the previous experiment.

	Experiment 5

1. Procedure

- Wire the circuit on the board as in Fig. 18.



ıre 18 – Capacitors in series

- Switch on the supply and note the light intensity of lamp 2.
- Short out one of the two capacitors by connecting a piece of wire across its terminals as shown by the dotted lines. This has the same effect as removing the capacitor and connecting the circuit through with only one capacitor in circuit.

  Is the lamp brighter? ___________________

  Is the lamp duller? _____________________

2. Conclusion

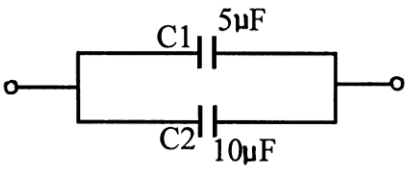
When capacitors are connected in series the capacitance decreases. (The smaller the capacitance; the dimmer the lamp).

The method of calculating the total capacitance for capacitors in series is :

$$\frac{1}{Ct}=\frac{1}{C}1+\frac{1}{C}2+$$

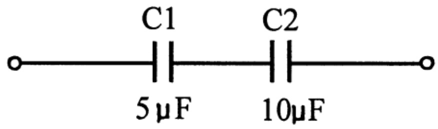
The formula used for capacitors is similar to the formula used for resistors. (Module EU-T). However, for capacitors in parallel the formula is the same as for resistors in series and for capacitors in series the formula is the same as for resistors in parallel.

See the following examples in Fig. 19.



$$C_t = C1 + C2 = 5 + 10 = 15\mu F$$

Parallel



$$\frac{1}{C_t} = \frac{1}{C1} + \frac{1}{C2}$$

$$= \frac{1}{5} + \frac{1}{10}$$

$$\frac{1}{C_t} = \frac{2 + 1}{10} = \frac{3}{10}$$

$$C_t = \frac{10}{3} = 3{,}3\,\mu F$$

Series

Figure 19

Self test

Calculate the total capacitance shown in experiments 4 and 5.

Show the formulae used for both calculations.

Check your answers by comparing them with those on the following page.

	Answers to self test 5

Experiment 4 = 100 μF

Experiment 5 = 25 μF

NOTES

Module 4
BE - 3 Identify and use inductors

Purpose of module

This objective will enable the learner to identify and use inductors

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Identify types of inductors.
2. State what "change" an inductor resists.
3. State whether an inductor offers high or low resistance to current flow in an AC and in a DC circuit.
4. State the name of an inductor resistance to AC current.
5. State the unit of induction.
6. Name two smaller units of induction.
7. Give the symbol for a coil (inductor).
8. Name the two main types of induction coils and draw their circuit symbols.
9. Give the quantity symbol for impedance and it's unit.
10. Give the quantity symbol for inductive reactance and it's unit.
11. Explain in your own words how an inductor affects a DC and an AC circuit.
12. Calculate the total inductance of inductors in series and parallel.
13. State the sequence of events when the DC current through a coil is interrupted.

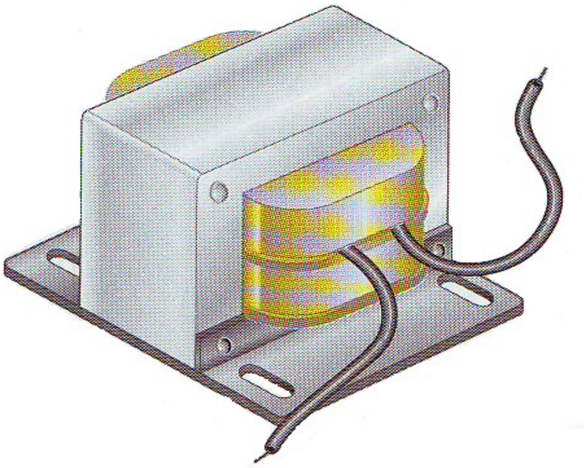
Procedures Relating To This Module

- Where applicable, all your answers must be in writing.
- All your answers must be correct and in accordance with the module notes.

1. Introduction

A coil or inductor is made from turns of conducting wire wound around a former (Figs.1 and 2).

The current flowing through the coil produces a magnetic field around the coil.
This magnetic field stores energy, and is capable of doing work.
A coil is a component or an inductor which resists changes in the current flowing through it. It offers a low resistance to direct current (DC) and a high resistance (or impedance) to alternating current (AC).

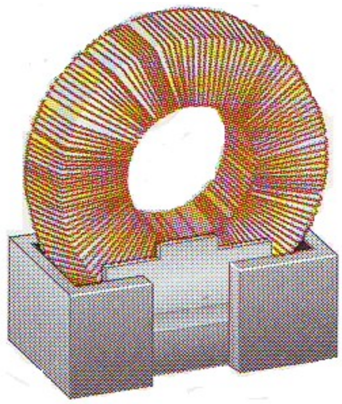


Iron core inductor

Symbol

Figure 1

2. *Types and symbols*



Air core inductor

Symbol

Figure 2

The value of a coil is called "inductance". The unit of inductance is termed the "Henry" (symbol H). Smaller units called millihenrys (mH) and microhenrys (µH) are also used. The inductance of a coil is increased by inserting soft iron through the centre of the coil.

When the current flowing through the coil is switched off, the magnetic field around the coil collapses producing a high voltage and an electric spark between the contacts of the open switch.

A coil is symbolized by the letter L.

There are two main types of induction coils namely:

1. Iron core inductor.

2. Air core inductor.

When only inductance is present in a circuit, then:

IMPEDANCE (Z) = INDUCTIVE REACTANCE (XL)

Inductive reactance (XL) is the amount of opposition offered by an inductor against alternating current and is measured in ohms.

The inductive reactance of coils is not constant. Inductive reactance depends on the inductance (L) and on the frequency (F) of an alternating current.

	Self test 1

Do the test below without referring to your notes.

1. What change does an inductor (coil) resist?

2. What resistance is offered by a coil in a DC circuit?

3. What resistance is offered by a coil in an AC circuit?

4. What is the high resistance to alternating current called?

5. Name the unit in which inductance is measured.

6. Name two smaller units of inductance?

7. What is the letter symbol for a coil?

8. Name the two main types of coils and draw their symbols.

9. How is an inductor made?

10. How can the inductance of a coil be increased?

11. What is the quantity of opposition offered by an inductor called?

12. What is the quantity symbol for impedance and its unit?

13. What is the quantity symbol for inductive reactance and its unit?

____

14. Is the inductive reactance of a coil constant?

____

15. What two things influence inductive reactance?

____

	To develop a better understanding of the effect of inductors in AC and DC circuits, carry out the experiment as explained on the next page.

3. *Effects of inductance on a direct current.*

	Experiment 1

To carry out this experiment you need:

- A variable AC supply 0-30 volts.
- A variable DC supply 0-30 volts.
- An AC Milliammeter 0-30mA
- 4.A DC milliammeter 0-30mA } or a 0 - 30mA
- An AC voltmeter 0-100 volts multimeter
- 6.A DC voltmeter 0-50 volts }
- 7.An Ohmmeter.
- An 5H inductor.
- Resistors as required.
- A single pole double throw toggle switch. (SPDT)
- A single pole single throw toggle switch. (SPST)
- Connecting wires.
- A circuit board or deck on which to mount the components.

1. Procedure

- Measure and record the resistance of the 5H inductor, in Table 1, under resistance and next to inductor as indicated by *.

TABLE 1

	Resistance	DC volts	DC amps	AC volts	AC amps
Resistor (R)	+		25 mA		
Inductor (L)	*				

- Connect the circuit as shown in Fig. 3 on the next page. The resistor R must be selected so that it has a resistance equal to the inductor resistance (or as close as possible to it) and enter this value in the space marked + in table 1.

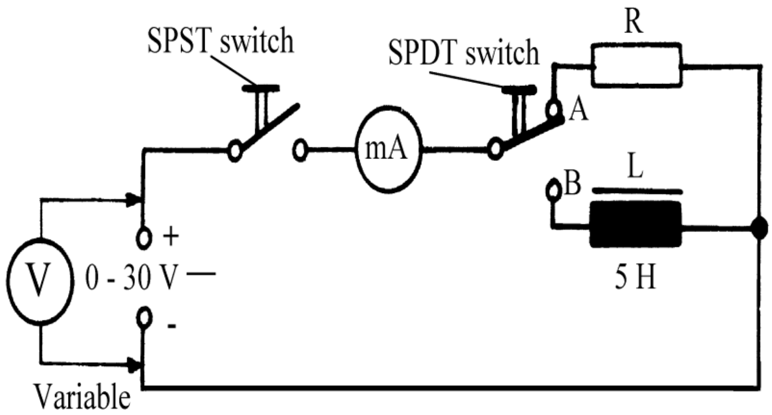


Figure 3

- Throw the SPDT switch to position A. This puts the resistor R in circuit and the inductor L out of circuit.
- Close the SPST switch and adjust the variable DC supply until the ammeter reads 25mA.
- Table 1 records the 25mA under the heading "DC amps" and in the horizontal column marked “Resistor".
- Read and record the voltage under the heading "DC volts" and in the horizontal column marked “Resistor".
- Without varying the supply voltage, throw the SPDT switch over to position B. This puts the inductor L in circuit and the resistor R out of circuit.
- Read and record the voltage amperage in their respective columns for the inductor.

  Is the current constant? ______________________

  Has the current decreased? __________________________

- Switch off the supply.

2. Conclusion

The effect that an inductor (coil) has in a DC circuit is relatively small. It is only the resistance of the copper wire that will influence the circuit to a small degree.

	Experiment 2

1. Procedure

- Disconnect the DC supply and all the meters.
- Replace the DC supply with a 0-100 volt AC supply. The supply must be adjusted to 0 volt.

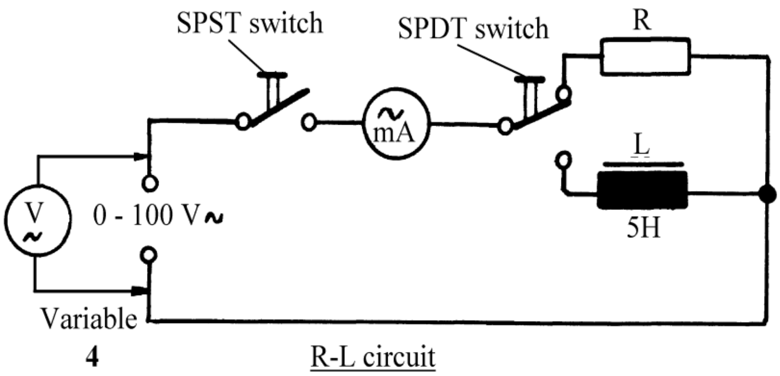


Figure 4

- DC volt and ammeters with AC volt and ammeters.
- Your circuit should now be the same as the circuit shown in Fig. 4
- Throw the SPDT switch to position A.
- Close the SPST switch and adjust the supply voltage until the voltage is equal to the voltage of the previous experiment. This information can be found in Table 1.
- Read and record the voltage and amperage in the columns AC volts and AC amps, Table 1 for the resistor.
- Without varying the supply voltage, throw the SPDT switch over to position B.
- Read and record the voltage and amperage in the columns for the inductor. Then switch off the supply.

  Is the current constant? ________________

  Has the current decreased? _________________

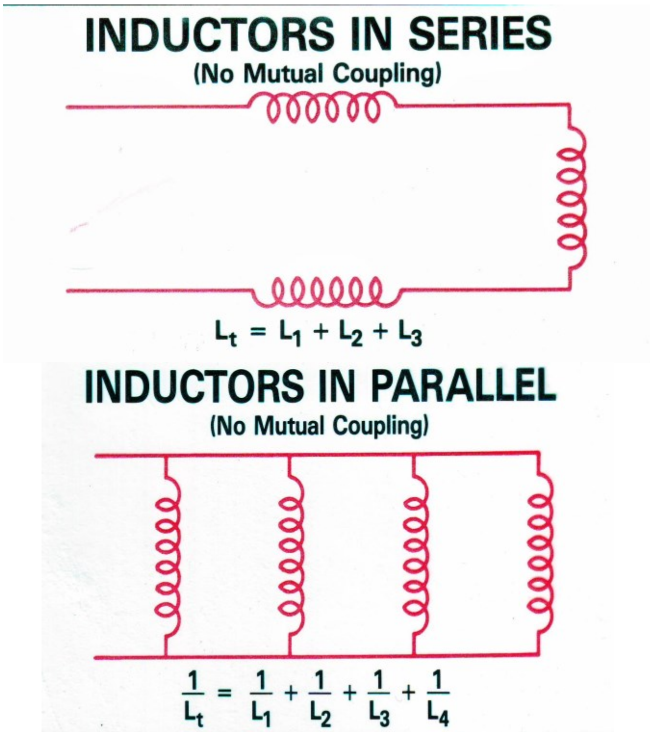
2. Conclusion

An inductor will have a big effect in an AC circuit because:

i) An inductor offers a high inductive resistance (inductive reactance) to an AC circuit.

ii) The resistance of the wire (also known as DC resistance) plus the inductive.

4. *Inductors in series and in parallel*



As in the case of resistors, inductors may be connected in series or in parallel.

1. Inductors in series

Inductors connected in series offer a greater opposition to current flow than a single inductor.

The total inductance (LT) of a series circuit is equal to the sum of the individual inductances

$$LT = L1 + L2 + L3 +$$

i.e.
This can be proved by doing Experiment 3.

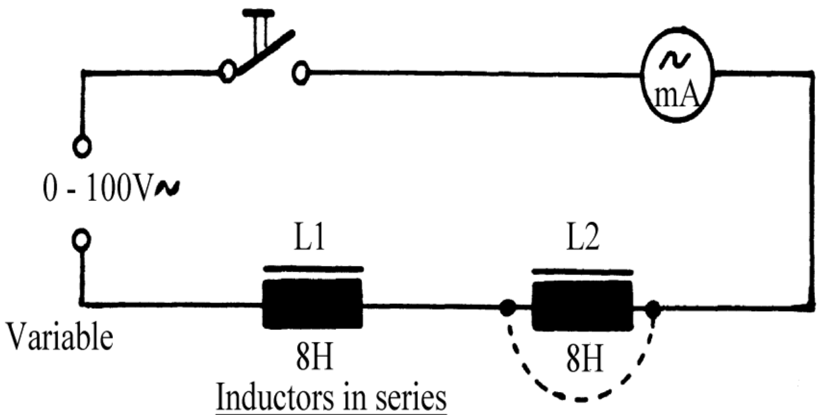


Figure 5

Experiment 3

To carry out this experiment you need:

- A 0-100 volt variable AC supply.
- Two 5H inductors.
- An AC ammeter 0-30mA FSD (or multimeter).
- A SPST toggle switch.
- A circuit board or deck on which to mount the components.
- Connecting wires.

1. Procedure

- Connect the circuit as shown in the circuit diagram (Fig. 5 on the next page) with the variable voltage on 0 volt.
- Switch on the supply.
- Adjust the variable voltage supply until the ammeter reads 10mA. This is the total current flowing through the series circuit.
- Now bridge either one of the two coils with a short length of wire, as shown by dotted lines in Fig. 5. This is the same as removing one of the coils i.e. there is only one coil left in circuit.
- Notice the reading on the ammeter.

  Is the reading constant? ____________________

  Did the current increase? ___________________

  Did the current decrease? __________________

2. Conclusion

The total current doubled when the one inductor was bridged. Therefore the two coils in series offer a higher resistance (impedance) to the flow of current than one inductor. This then proves the formula

$$LT = L1 + L2 + L3 +$$

3. Inductors in parallel

Inductors connected in parallel can also be compared to resistors in parallel. The same type of equation is used for inductors in parallel as for resistors in parallel i.e.

$$\frac{1}{LT} = \frac{1}{L}1 + \frac{1}{L}2 + \frac{1}{L}3 +$$

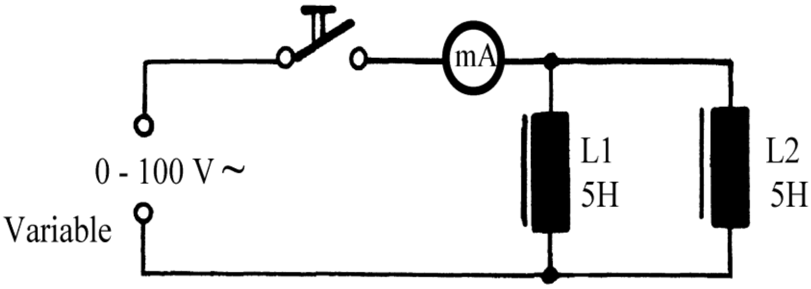
This can be proved by doing Experiment 4.

The same components and equipment are used as in the previous experiment.

Experiment 4

1. Procedure

- Connect the circuit as shown in the circuit diagram (Fig. 6).
- Switch on the supply.
- Adjust the variable voltage supply until the ammeter reads 20mA. This is the total current flowing through the parallel circuit.
- Remove one of the inductors from the circuit.
- Notice the reading on the ammeter.



Inductors in parallel

Figure 6

Is the reading constant? ____________________

Did the current increase? ___________________

Did the current decrease? ___________________

2. Conclusion

The total current was halved when one inductor was removed. Therefore the two coils in parallel offer less resistance (impedance) to the flow of current than one inductor.
The formula is proved in this experiment.

$$\frac{1}{LT} = \frac{1}{L}1 + \frac{1}{L}2 + \frac{1}{L}3 +$$

Self test 2

Answer the questions below without referring to your notes.

1. Explain in your own words how an inductor affects a DC circuit.

2. Explain in your own words how an inductor affects an AC circuit.

3. When inductors are connected in series, is the total inductance more or less than when a single inductor is in the circuit?

4. When inductors are connected in parallel, is the total inductance more or less than when a single inductor is in the circuit?

5. Describe in your own words the sequence of events when the DC current flowing through a coil is switched off.

6. Calculate the total inductance of the following:

Remember that the method used is the same as for resistors, only the components and symbols are different. If you cannot remember the method refer to module EU-T

a) An 8mH and a 4mH in series

b) An 8H; 8H; and 4H in parallel

c) A 12μH; 12μH; 12μH; and 12μH in series

d) A 12H; 12H; 12H and 12H in parallel.

Ask for the criterion test when you are ready.

NOTES

Module 5
BE - 4 Identify diodes

Purpose of module

This objective will enable the learner to identify diodes

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Identify the diode characteristics (peak inverse voltage and maximum current).
2. Distinguish between the cathode and anode of a diode.
3. State to which polarity of the supply the cathode is connected when the diode is forward biased.
4. Draw the circuit symbol for a diode.
5. Give the letter symbols for a cathode and anode of a diode.
6. Describe how diodes are identified.
7. State in which direction a diode is biased when it conducts.
8. State the two voltages which equal the supply voltage (VS) when a diode is forward biased.
9. Name two types of diodes.

Procedures Relating To This Module

- Where applicable all answers must be in writing.
- All answers must be correct and in accordance with the module notes.

1. Introduction

In electronic circuits it is sometimes necessary to allow current to flow in one direction only and to prevent it from flowing in the opposite direction. The diode is a component that can perform this function.

The diode has two terminals or lead out wires. One terminal is distinguished by a single band or spot on the body of the component and is known as the cathode. This side must be connected to the more negative side of the circuit in which it is to be used. The other connection which is usually unmarked, is known as the anode and is connected to the more positive side of the circuit. In this state the diode is <u>forward biased</u> and will conduct current. If, for instance, the condition changes, i.e. the cathode become more positive than the anode, the diode is said to be <u>REVERSE BIASED</u> and will not conduct current in this state.

There is a variety of diodes and they vary in both shape and size. Some types are shown in Fig. 1.

The symbol used for a diode in circuit diagrams is also shown in Fig. 1. The bar represents the cathode and the arrow head represents the anode and points in the direction of conventional current flow, i.e. from a more positive to a less positive or negative voltage.

The symbols for cathode and anode are cathode = k and anode = a.

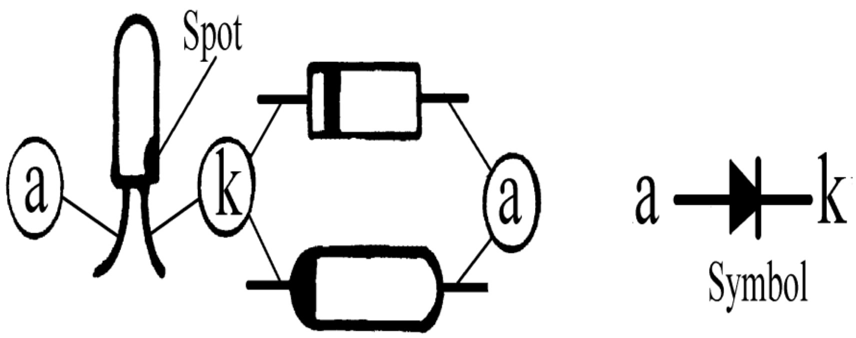


Figure 1

2. *Diode identification (Refer to diode reference chart)*

Most diodes have a number printed on the body for identification. For information on the diode the number may be looked up in a semi-conductor reference book.

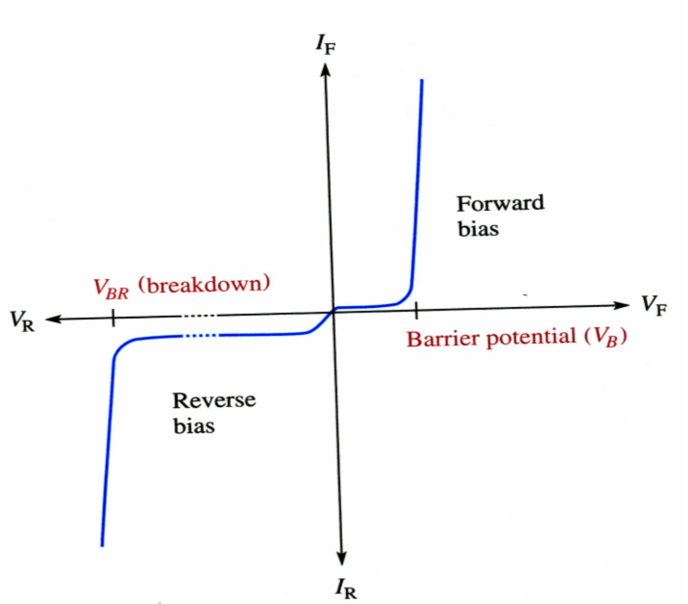


Figure 2

Important information needed is the peak inverse voltage (blocking voltage) and the maximum current that the diode can pass through it. For example, diode number 1N4007 has a peak inverse voltage of 1 000 volts and a maximum current carrying capacity of 1 ampere. This information is shown on the characteristic curve in Fig. 2.

Fig. 3 illustrates the unidirectional (one direction) behaviour of a diode.

The diode acts as a switch

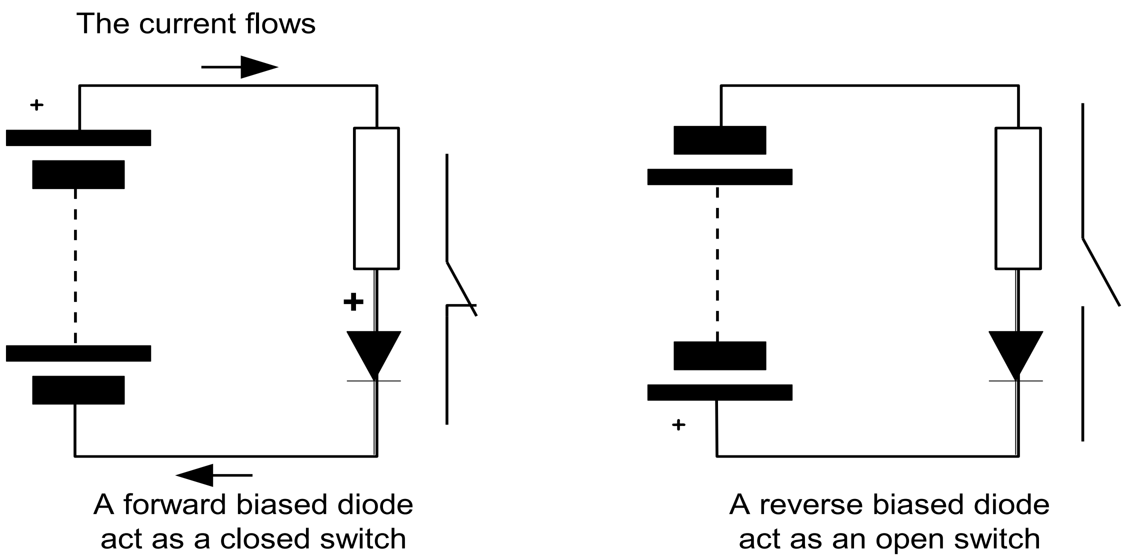


Figure 3

Experiment 1

In Fig. 3(a) the current is flowing and the diode is said to be FORWARD BIASED and in (b), where there is no current flowing, the diode is said to be REVERSE BIASED.

To prove the statements above carry out the experiments that follows.

To carry out this experiment you need:

- A 12 volt DC supply.
- A 12 volt lamp.
- A SPST toggle switch.
- A 1N4007 diode. (Rated at 1 amp with peak inverse voltage of 1000V).
- A circuit board or deck on which to mount the components.
- Connecting wires.
- A multimeter.

1. Procedure

- Construct the circuit as shown in Fig. 4.
- Switch on the supply.

  Is the lamp on? ____________________________

  Is the diode forward biased or reverse biased? ____________________

- Measure the volt drop across the diode.
  (0-2 volt scale).
- Measure the volt drop across the lamp.
  (0-20 volt scale).

  What is the volt drop across the diode? ____________________

  What is the volt drop across the lamp? _____________________

- Switch off the supply.

- Remove the diode from the circuit and reconnect it the other way around as shown in Fig. 5.

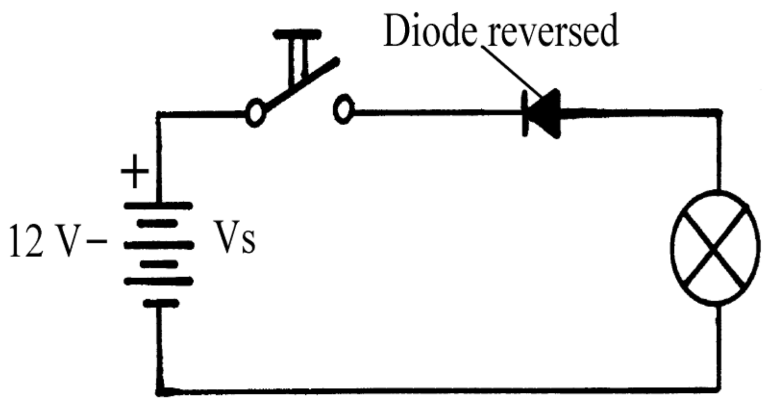


Figure 5

- Switch on the supply.

  Is the lamp on? __________________

  Is the diode forward biased or reverse biased? ___________________

- Measure the volt drop across the diode.
  (0-20 volt scale).

- Measure the volt drop across the lamp.
  (0-2 volt scale).

- What is the volt drop across the diode? ___________________

- What is the volt drop across the lamp? _____________________

- Switch off the supply.

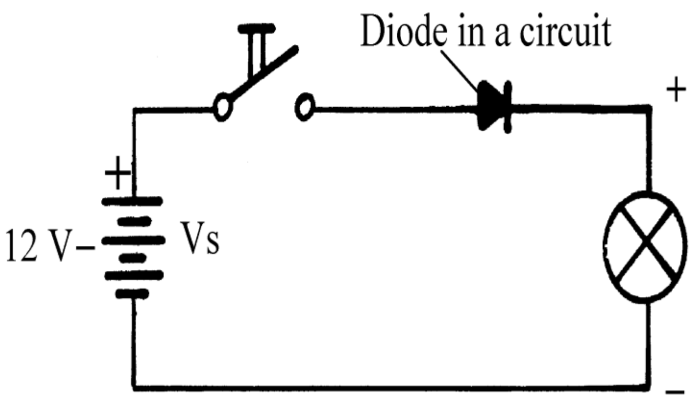


Figure 4

- With the diode connected in the same direction, reverse the supply polarity as shown in Fig. 6.

- Switch on the supply.

  Is the lamp on? ____________________

  Is the diode forward biased or reversed biased ?_______________________

- Switch off the supply.

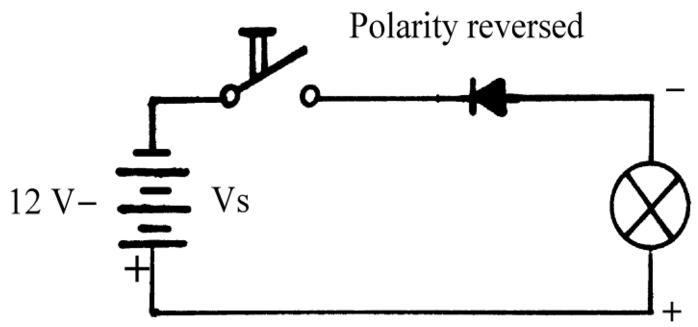


Figure 6

2. Conclusion

From Experiment 1, Fig. 4 it was seen that if the POSITIVE of the supply is connected to the ANODE of the diode, it will conduct and the diode is said to be forward biased. Similarly, if the NEGATIVE of the supply is connected to the CATHODE of the diode, it will also be forward biased as shown in Fig. 5.

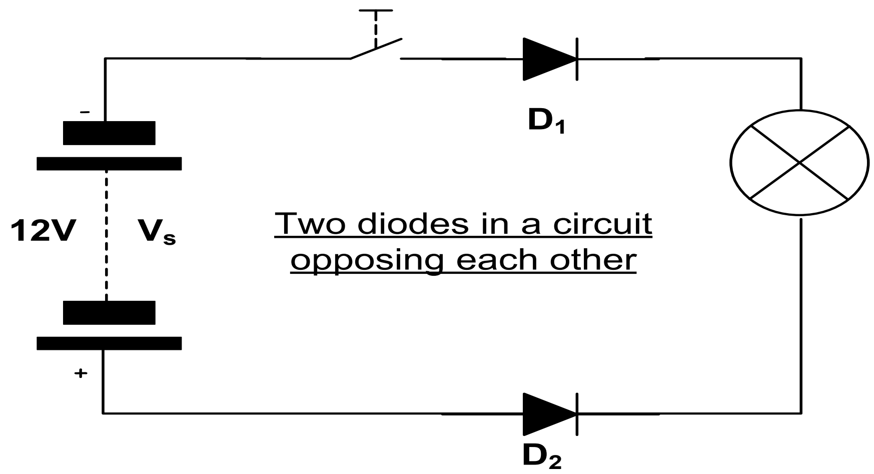


Figure 7

	Experiment 2

The same components and equipment used in the previous experiment are required together with an extra 1N4007 diode.

1. Procedure

- Construct the circuit as shown in Fig. 7.
- Before you switch on the supply, trace out the circuit and delete the incorrect word.

  When switched on the lamp will / will not burn.
- Give reasons for your answer.

  ____

  ____

  ____

  ____
- Switch on the supply.
- Is the lamp on? ______________________
- **Delete the wrong word:**

  Diode D1 is:

forward biased / reverse biased

  Diode D2 is:

forward biased / reverse biased

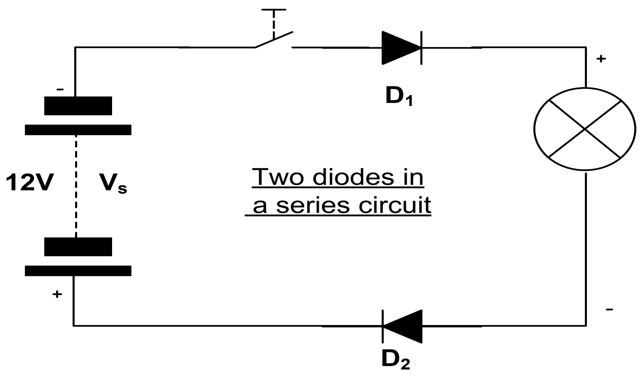


Figure 8

- Switch off the supply.
- Remove diode D2 and reconnect it in circuit as shown in Fig. 8.
- Switch on the supply.

  Is the lamp on ?----------------------

- **Delete the wrong word:**

  Diode D1 is:

forward biased / reverse biased

  Diode D2 is:

forward biased / reverse biased

- Measure the volt drop across D1.

  (0-2 volt scale) --------------------------

  Measure the volt drop across D2.
  (0-2 volt scale) --------------------------

  Measure the volt drop across the lamp.
  (0-20 volt scale) ------------------------

- You have learnt previously (module EU-T) that Kirchoff's voltage law states that the sum of the voltage drops in a closed electric circuit is equal to the supply voltage.

  Is this true for the above experiments?

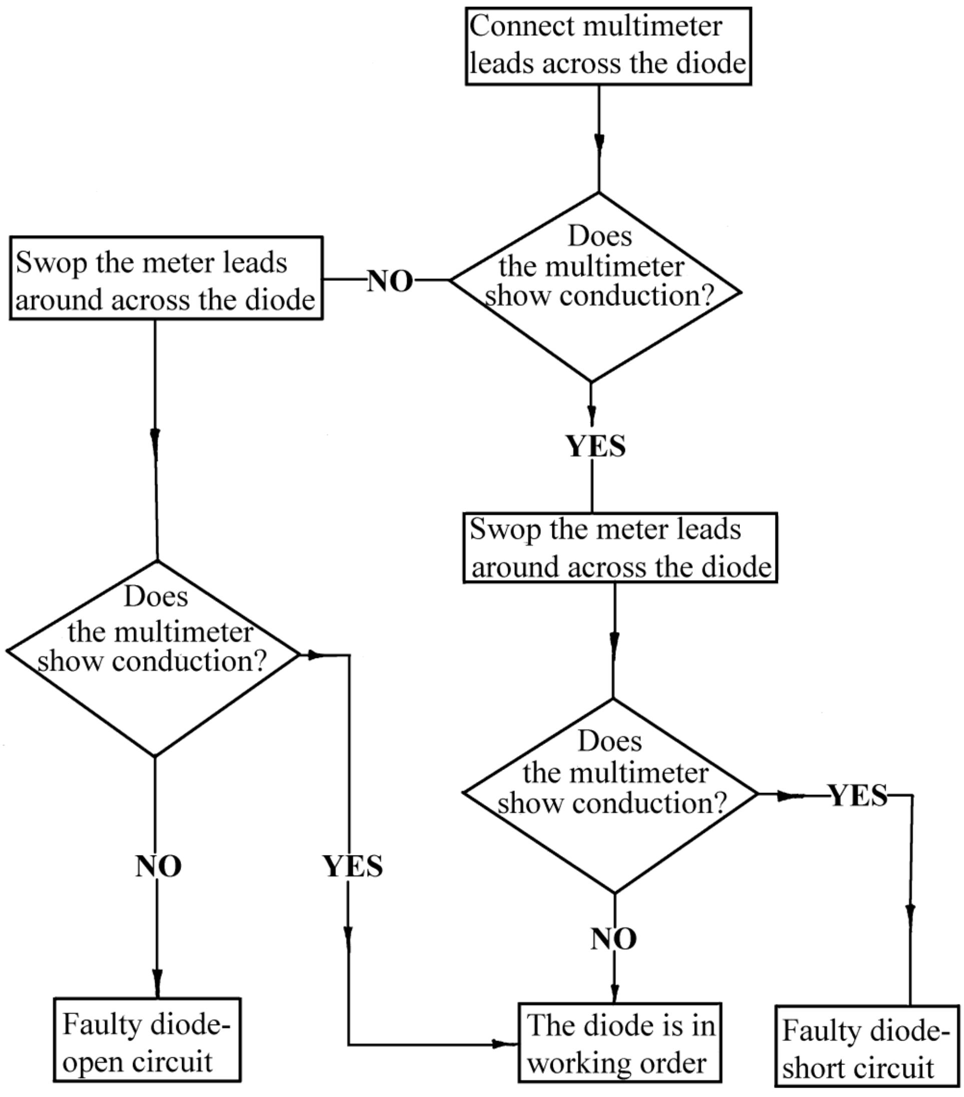
2. Conclusion

A diode will only conduct when it is forward biased and while conducting the forward voltage drop (V_F) across it is always approximately 0,6 volts. This is the case when a silicon diode is used. A germanium diode which is rather uncommon, has a forward voltage drop (V_F) of approximately 0, 2 volts.

3. Testing a diode with a digital multimeter

- Set the multimeter to the continuity or diode scale. (Note that when testing semi-conductors the red wire of the digital multimeter has a positive polarity on the continuity scale).
- Connect the multimeter leads onto the leads of a working diode. If the continuity reading is infinity it means that the multimeter leads are reverse biasing the diode.
- Change the multimeter leads around over the diode. A conduction reading indicates that the multimeter is now forward biasing the diode and that the diode is in working order.

A rule of thumb that may be adopted for testing a diode is that when the leads of the meter is reverse biasing the diode, a reading less than infinity indicates a faulty diode. When the meter leads are forward biasing the diode, a reading that falls outside the range 400 - 600 mV also indicates a faulty diode. Five diodes of unknown condition. Using the flow chart below, test and state the condition of each diode.



Self test

1. How do you distinguish the cathode from the anode on a diode?

2. To what side of the supply is the cathode connected when the diode is forward biased?

3. Draw the circuit symbol for a diode.

4. What are the letter symbols for the anode and the cathode of a diode?

5. How are diodes identified?

6. When a diode is conducting in which direction is it biased?

7. What is the forward (V_F) volt drop across a silicon diode and what is the forward volt drop (V_F) across a germanium diode?

8. Ask your facilitator for at least five diodes of unknown condition. Identify, test and state the condition of each diode.

9. Draw two simple diagrams, each using a diode, a load resistor and a battery. Show on the diagrams how you would connect the battery if you want:

   (a) A forward biased diode, and

   (b) A reverse biased diode.

	When you are ready ask for the criterion test.

NOTES

Module 6
BE- 5 Construct rectification circuits

Purpose of module

This objective will enable the learner to construct rectification circuits

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Draw the circuit diagram for a half wave rectifier, a full wave rectifier with a centre tap transformer and a full wave bridge rectifier.
2. Construct a rectification circuit.
3. Display the waveform on an oscilloscope.
4. Draw half wave and full wave rectified waveforms.
5. Measure and calculate the mean (average) DC voltage of a half wave and a full wave rectifier.
6. Draw a waveform for a smoothed rectified circuit without load.
7. Draw a waveform for a smoothed rectified circuit with a load.
8. State the reason for a smoothed rectified circuit having a higher voltage than an unsmoothed circuit.

Procedures Relating To This Module

- The waveforms must be drawn correctly.
- All answers must be correct and in accordance with the module notes.

1. Introduction

The most important characteristic of a diode is that it will only allow current to flow in one direction. This characteristic makes a diode ideal for rectification, that is, for changing alternating current to direct current.

Alternating current changes direction each cycle i.e. 100 times/second if the mains frequency is 50 Hz. If a diode is connected in series with a load and an AC supply, current will only flow in one direction through the load i.e. only for those cycles that forward bias the diode. This is shown in Fig. 1.

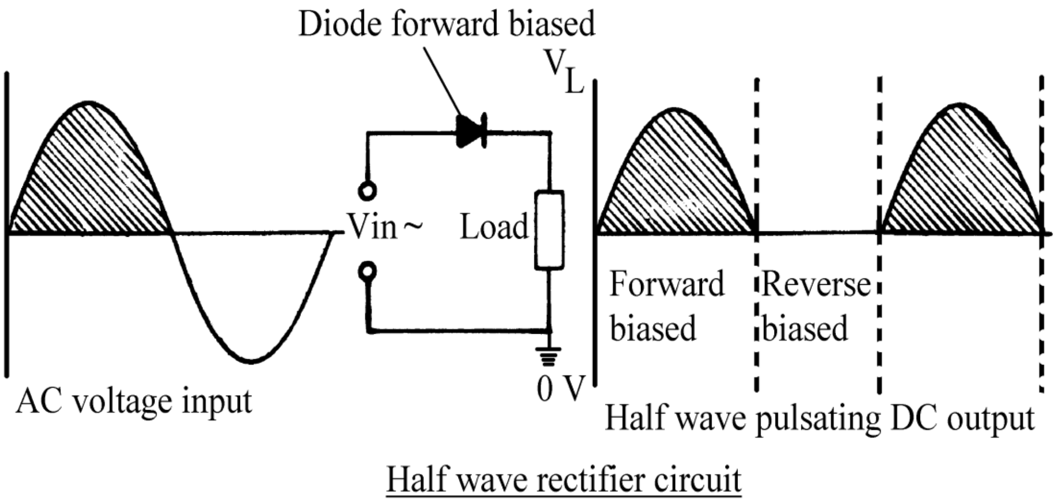


Figure 1

When the diode is connected in the opposite way to that in Fig. 1 the output waveform will be as indicated in Fig. 2.

These voltage shapes can be shown on an oscilloscope by doing the following experiments.

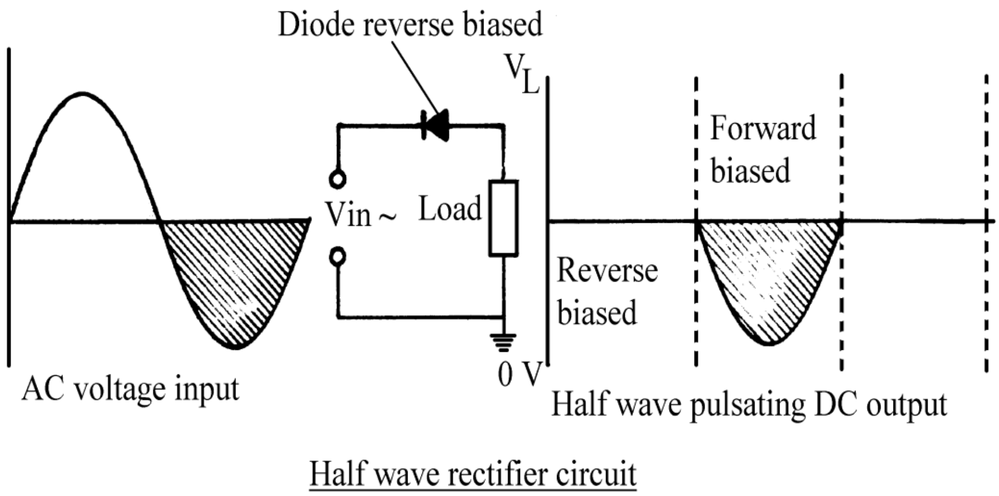


Figure 2

2. *Half-wave rectification*

	Experiment 1

To carry out this experiment you need:

- A 12 volt AC supply.
- A 1N4007 silicon diode.
- AS.P.S.T. toggle switch.
- An oscilloscope.
- A multimeter.
- A 1 kilohm ½ watt 5% tolerance resistor.
- A circuit board on which to mount the components.

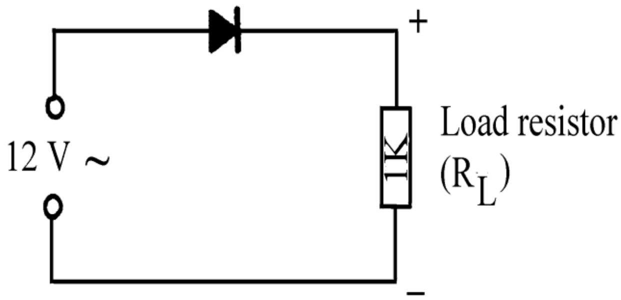


Figure 3

1. Procedure

- Construct the circuit as shown in Fig.3.
- Measure the voltage (VRL) across the load resistor with the oscilloscope; the input coupling must be on DC and the earthclip of the probe at the bottom of the resistor.
- Draw the output voltage (VRL) waveform in detail as measured by the oscilloscope on the axis in Fig. 4 and record the voltage value next to Vp. (peak voltage).

	It can be seen from Fig.4 on the next page that the peak voltage across the load is lower than the peak value of the supply. This is due to the forward volt drop of the diode.

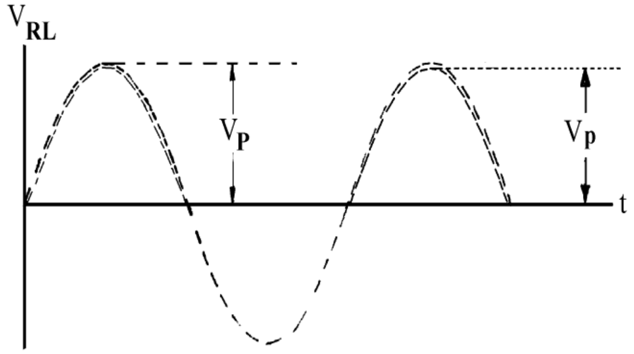


Figure 4

- Measure the voltage (VD) across the diode with the oscilloscope; earth clip must be on the cathode side of the diode.
- Draw the voltage waveform across the diode (VD) in detail on the axis in Fig. 5 and record the voltage value next to Vp. (peak voltage).

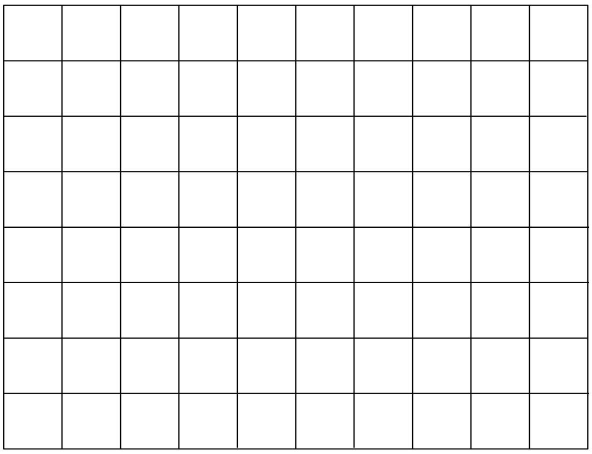


Figure 5

- During a positive ½ cycle, the diode will be ___________________ biased and VFof__________________ volts will appear across the diode. Therefore, the peak value of VRL will be equal to the supply voltage (V's) minus__________________ of VRL will be equal to the supply

  During the negative half cycle, the diode will be _______________ biased and no current will flow.

  The full supply voltage will therefore appear across the diode.

- Measure the mean DC voltage across RL with the multimeter.
  Mean DC Voltage = ___________________

	Meters measure mean (average) voltage values and oscilloscopes measure peak voltage values.

- The mean (average) DC voltage from a half wave rectifier can be calculated from the formula on the next page.

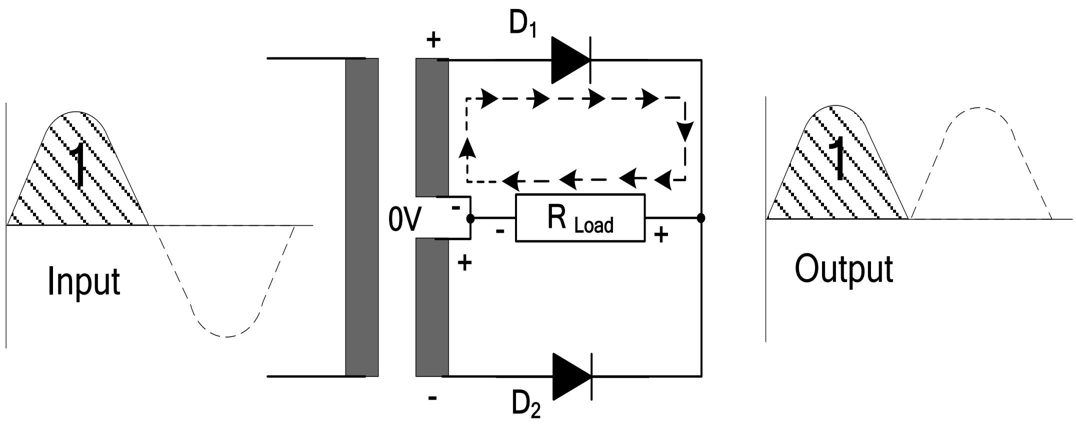
  Half wave formula :

  $$V \text{ mean (avg)} = \frac{vp \times 0{,}637}{2}$$

3. *Full wave rectification with a centre tapped transformer*

In this type of circuit, a diode is placed in series with each half of the transformer secondary, and with the load. Effectively, therefore, there are two half wave rectifiers working on the same load (RL).

During the first half cycle, the transformer's AC voltage on the upper half is positive and the diode D1 is forward biased. The diode conducts and as a result current flows through the load as shown by the dotted lines in Fig. 6.

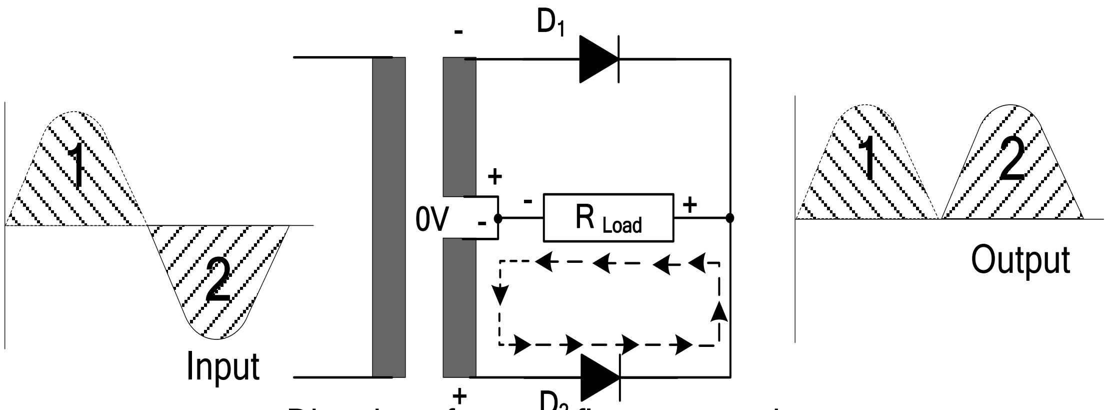


Direction of current flow on positive half

Figure 6

This causes half wave pulsating DC across the load. Whilst the diode D1 conducts and is forward biased, the diode D2 is reverse biased and does not conduct.

During the second half cycle the upper transformer half is negative and the diode D1 cannot conduct. However, the lower transformer half is positive and diode D2 conducts and the current passes through the load. (Fig. 7).



Direction of current flow on negative half

Figure 7

Since both half cycles resulted in current to flow through the load in the same direction, a DC voltage appears across the load, thus the term "direct current" or" DC" is used.

The full wave rectifier has, in fact, changed both halves of the AC input into a full wave pulsating DC output. (Fig. 8).

The voltage wave form can be seen on an oscilloscope by doing Experiment 2 on the next page.

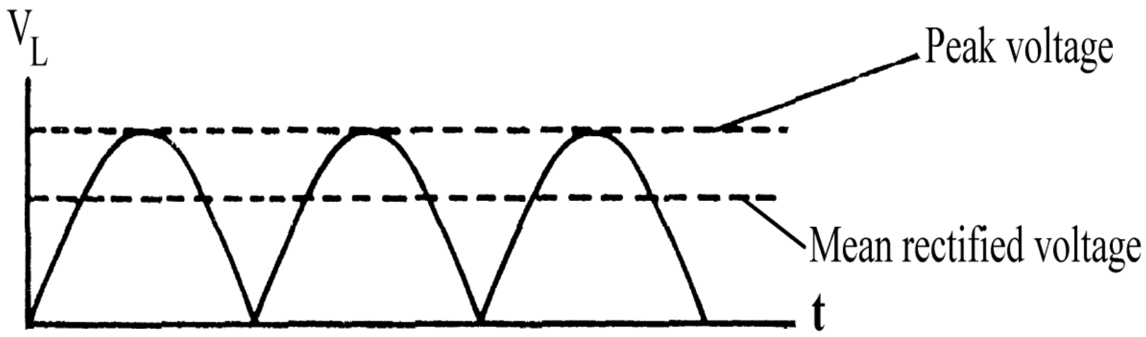
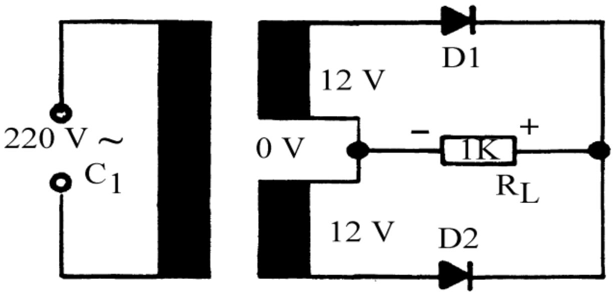


Figure 8

	Experiment 2

To carry out this experiment you need:

- A 220 volt/12-0-12 volt 500mA transformer.
- Two 1N4007 silicon diodes.
- An oscilloscope.
- A multimeter.
- A 1 kilohm ½ watt 5% tolerance resistor.
- A circuit board on which to mount the components.
- Connecting wires.



Rectification circuit

Figure 9

1. Procedure

- Construct the circuit as shown in Fig. 9.
- Measure the voltage (VRL) across the load resistor with the oscilloscope.
- Draw the output voltage (VRL) waveform in detail as measured by the oscilloscope on the axes shown in Fig. 10 on the next page and record the voltage value next to Vp (peak voltage).

- Measure the mean DC voltage across RL with the multimeter.

  MEAN DC VOLTAGE = _____________________________

- The mean DC voltage from a full wave rectifier can be calculated using the formula:

  Mean DC = VL peak x 0,637.

- Calculate the mean DC output voltage for the above circuit and compare this value with the value measured above.

  Mean DC voltage = Vp across

  load x 0,37

  = ... x 0,637

  =....... volts.

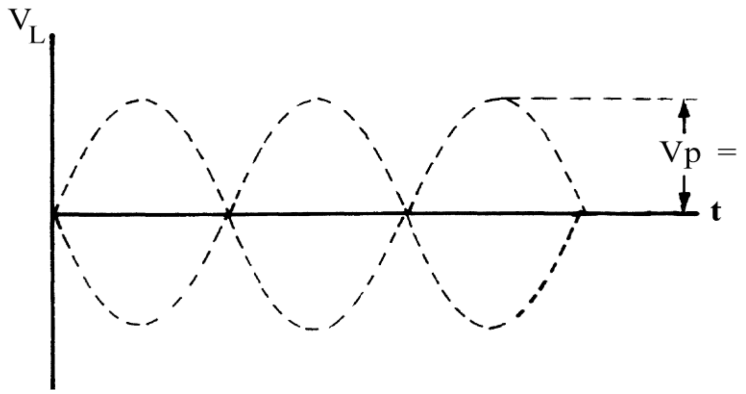


Figure 10

- Draw the output waveform with both D1 and D2 reversed in Fig. 9 on the next page in on the axis in Fig. 11.

- The method shown in Fig. 11 makes use of a centre tapped transformer. Another method of obtaining full wave rectification without a centre tap transformer is to use four diodes connected in a bridge mode. This is explained in the next section.

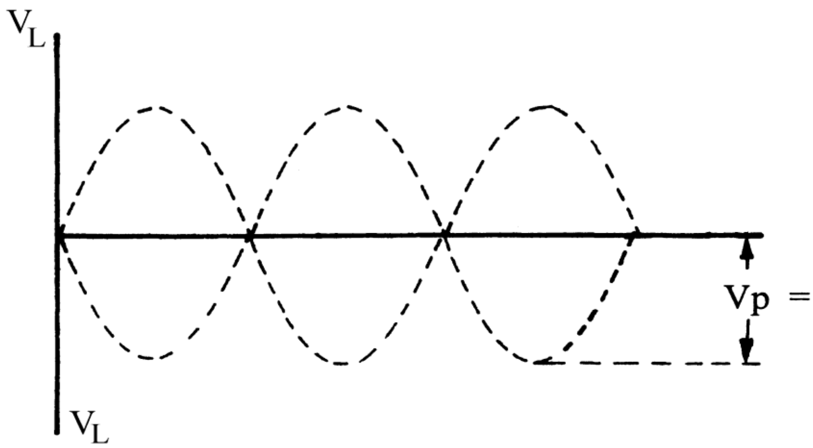


Figure 11

4. *Full wave bridge rectifier*

Four diodes form the four arms of the full wave bridge rectifier circuit.

The AC supply and the DC load are connected as shown in Fig. 12 and the rectifier works as follows:

When point A is positive with respect to point B, diodes D2 and D3 are reverse biased i.e. they block current flow. Diodes D1 and D4 on the other hand are forward biased and current flows through them as shown by the dotted lines in Fig. 12.

When the voltage of the AC cycle is reversed, that is when B is positive with respect to A, diodes D1 and

D4 will be reverse biased while D3 and D2 are forward biased. The current will therefore flow in the direction shown in Fig. 13.

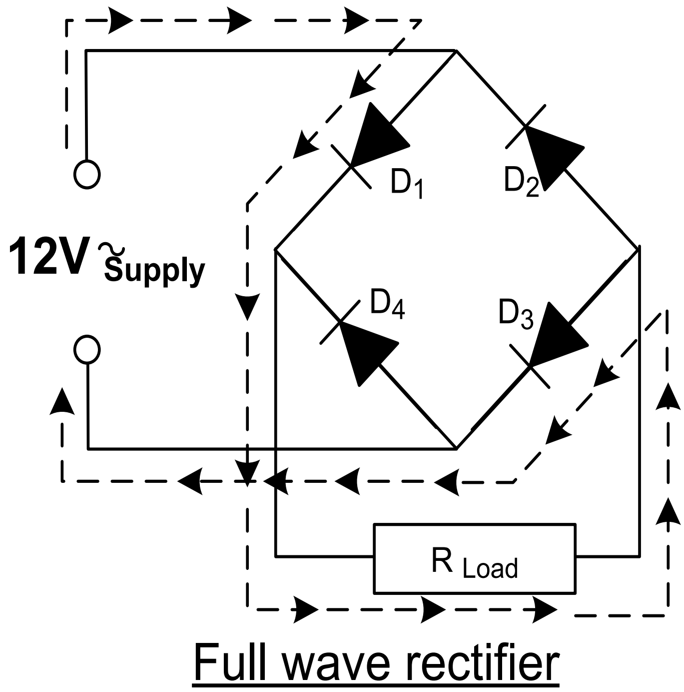


Figure 12

The output waveform is the same as in the previous rectifier and may also be shown on the oscilloscope by doing the following experiment. Although the output waveforms are the same, one small difference does exist. The output voltage is not 0,6V lower than Vp of the AC supply, but 1,2V. The reason for this is the fact that each half cycle passes through two diodes as described below; therefore the voltage drop of 2 x 0,6V across the diode bridge**.**

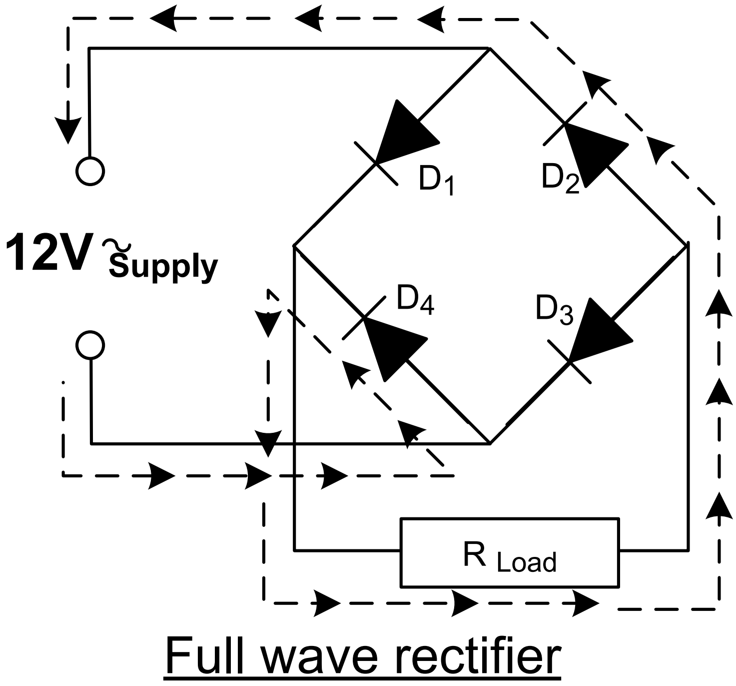


Figure 13

	Experiment 3

To carry out this experiment you need:

- A 12 volt AC supply.
- Four 1N4007 diodes.
- An oscilloscope.
- A multimeter.
- A 1 kilohm ½ watt 5% tolerance resistor.
- A circuit board.
- Connecting wires.

1. Procedure

- Construct the circuit as shown in Fig. 14.
- Measure the voltage (VRL) across the load resistor with the oscilloscope.

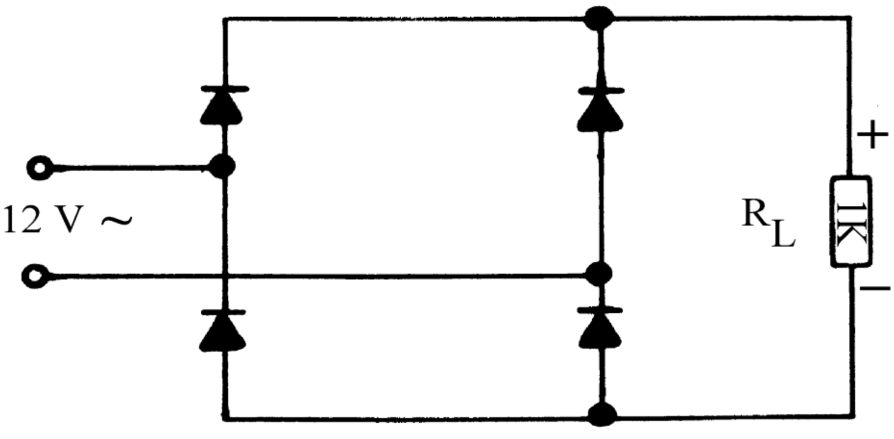


Figure 14

- Draw the output voltage waveform (VRL) in detail as measured by the oscilloscope on the axes shown in Fig. 15 and record the voltage value next to Vp (peak voltage).

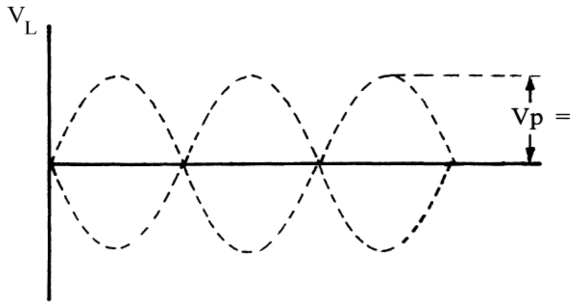


Figure 15

- Measure the mean DC voltage across RL with the multimeter.

  MEAN DC VOLTAGE = ________________________________

  The mean DC voltage from a bridge rectifier can be calculated using the formula:

  Mean DC = VL peak x 0,637

- Calculate the mean DC output voltage for the above circuit and compare this value with the measured value.

  Mean DC voltage = Vp across
  load x 0, 637
  = ... x 0,637
  = …...... volts.

2. Conclusions

The conclusions which can be drawn from these experiments are :

(i) A single diode in a rectification circuit produces half wave pulsating DC due to the fact that the diode is forward biased every second half cycle of the AC supply.

(ii) Two methods of full wave rectification are generally used, viz:

(a) The centre tap transformer with two diodes;

(b) The single secondary winding with bridge rectifier.

(iii) The voltage lost across the half wave rectifier a well as the full wave centre tap arrangement, is only 0,6V for each half cycle compared with 1,2V for each half cycle in the case of the bridge rectifier.

(iv) In full wave rectification, both half cycles of the AC supply are rectified. This produces a higher mean value voltage than half wave rectification.

(v) The main objective of a rectification circuit is to change an____________________ to a ______________________________output.

If this section of the module is clear to you do the self test on the next page. If you are not certain, read the notes and do the experiments again.

Self test 1

Answer the questions below without referring to your notes.

1. Draw the circuit diagram for a half wave rectifier with a load and also draw the load waveform.

2. Draw the circuit diagram for a full wave rectifier with centre tap transformer, two diodes and a load. Also draw the load waveform.

3. Draw the circuit diagram for a full wave bridge rectifier with a load. Also draw the load wave form.

4. Calculate the mean DC voltage across a load when a bridge rectifier is used to rectify 10V AC.

5. What formula is used to calculate the mean voltage value of a half wave rectifier output from the peak voltage?

____

6. Why is the term "Direct current" used?

____

7. What type of DC voltage does a diode produce in the process of rectification?

____

8. State which one of a half wave rectifier or a full wave rectifier gives a higher mean voltage output?

____

	Experiment 4

To carry out this experiment you need:

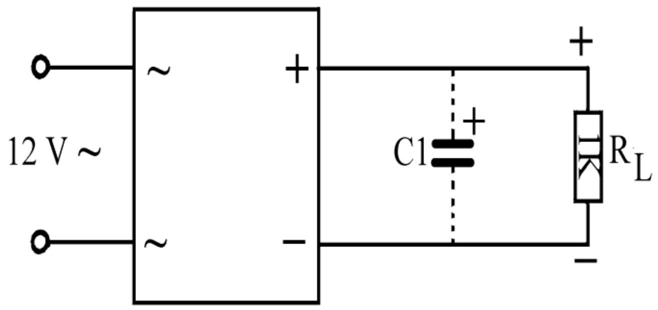
- A 12 volt AC supply.
- A bridge rectifier. (50 volt 1 amp).
- A 1000 μF electrolytic capacitor (16 volts).
- A 1000 ohm ½ watt 5% tolerance resistor.
- A 100 ohm 2 or 5 watt 5% tolerance resistor.
- An oscilloscope.
- A multimeter.
- A circuit board.
- Connecting wires.

1. Procedure

- Construct the circuit as shown in Fig. 16. Do not insert the 1000μF (C1) capacitor at this stage.

	Only a block for the bridge rectifier is shown. This is because the four diodes needed are very often supplied in a single package with four leads. This is known as a bridge rectifier

- Measure the voltage (VRL) across the load resistor with the oscilloscope.
- Draw the output voltage (VRL) waveform in detail as measured by the oscilloscope on the axis in Fig. 17, indicating the peak value. This pulsating DC waveform is also known as an unsmoothed DC waveform.



Bridge rectifier

Figure 16

Vp across load = ... volts.

- Measure the mean DC voltage across RL with the multimeter.

  MEAN DC VOLTAGE = --------volts.

- Connect a 1000 μF capacitor (C1) across RL (as shown by the dotted lines in the circuit diagram, Fig. 16).

- Measure the voltage across the load resistor with the oscilloscope.

- Draw the output voltage waveform on the axis in Fig. 17 and clearly mark it 'A'.

Figure 17

- Note down the voltage value measured with the oscilloscope.

- Measure the DC output voltage with the multimeter.

  DC OUTPUT VOLTAGE = ---- volts.

- Replace the 1000 ohm resistor with a 100 ohm resistor.

- Draw the output waveform on the axes in Fig. 18 and clearly mark it with 'B'.

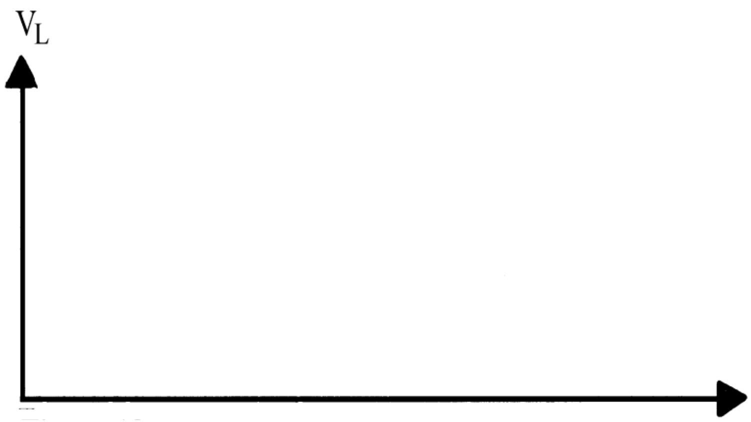


Figure 18

- Measure the DC output voltage with the multimeter.

2. Conclusions

The conclusions which can be drawn are :

i) The smoothed DC voltage has a higher value than the average unsmoothed DC voltage because the capacitor charges to peak value and maintains $V_{out\ at}$ peak value or a value close to it. In the case of unsmoothed DC, the pulsating nature of the waveform causes the voltage to continuously change its value between zero volts and peak volts which results in an average voltage measurement.

ii) If the load resistance is decreased, the ripple waveform increases and the measured voltage decreases because the capacitor is partly discharged through RL during the period between peaks.

iii) The rated DC voltage of the capacitor must exceed the peak value of the applied DC.

If this section on capacitive smoothing is clear to you, do the self test on the next page, if it is not clear, read the notes again and do the experiment over.

Self test 2

Answer the questions below without referring to your notes.

1. Draw an unsmoothed DC waveform with a load.

2. Draw a smoothed DC waveform with a 1000Ω load.

3. Draw a smoothed DC waveform with a 100Ω load.

4. What happens to the mean DC output voltage when a smoothing capacitor is connected across the load?

    ____

    ____

5. What happens to the ripple waveform when the circuit's load resistance is decreased?

    ____

    ____

6. In selecting a smoothing capacitor, the rated DC voltage value of the capacitor must be HIGHER/LOWER than the applied DC MEAN/PEAK voltage.

    DELETE THE INCORRECT WORDS.

Delete the incorrect words.

Ask for the criterion test when you feel ready.

Module 7
BE- 6 Identify and use zener diodes

Purpose of module

This objective will enable the learner to identify and use zener diodes

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Identify zener voltages.
2. State in which direction a zener diode is connected in a circuit to act as a voltage regulator.
3. Draw the circuit symbol for a zener diode.
4. Draw a curve of the forward voltage (VF) against current (IF) for a zener diode.
5. State the reasons why a series resistor (RS) is connected in a zener parallel voltage regulated circuit.
6. Draw a characteristic curve for zener voltage (VZ) against current (IZ) for a zener diode.
7. Draw aIZ/VZ characteristic curve for the operating range of a zener diode and show all the calculations made.
8. Draw a circuit diagram showing how different voltages may be obtained from a common supply by means of zener diodes.
9. State what the effect would be if zener diodes were connected in series.
10. State what the effect would be if zener diodes were connected in parallel.
11. Draw a waveform indicating peak voltage, when the voltage is

Procedures Relating To This Module

- No other references will be allowed.
- Where applicable, all answers must be in writing.
- All answers must be correct and in accordance with the module notes.

1. Introduction

A zener diode is a special type of diode. It is connected into a circuit in the reversed biased direction. When the voltage connected over the diode is lower than the rated zener voltage, the diode will not conduct. However, when the voltage over the diode reaches the value of the ratedzener voltage, the diode suddenly starts to conduct. When the diode conducts, the voltage over the zener remains constant at the rated value, no matter what amount of current flows through it. This constant voltage over the zener diode is maintained as long as the current flow through it is kept within certain limits. These limits will be pointed out in part 2 of this module.

It can be seen, therefore, that zener diodes are used to regulate voltages, i.e. they keep the voltage steady. The breakdown voltage (zener voltage) of a zener diode is set during its manufacture and cannot be changed. Zener diodes are designed to deliver zener voltage from one to several hundred volts.

The circuit symbol for a zener diode is shown in Fig. 1.

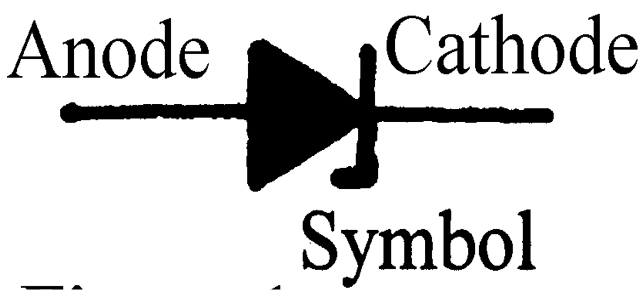


Figure 1

	Experiment 1

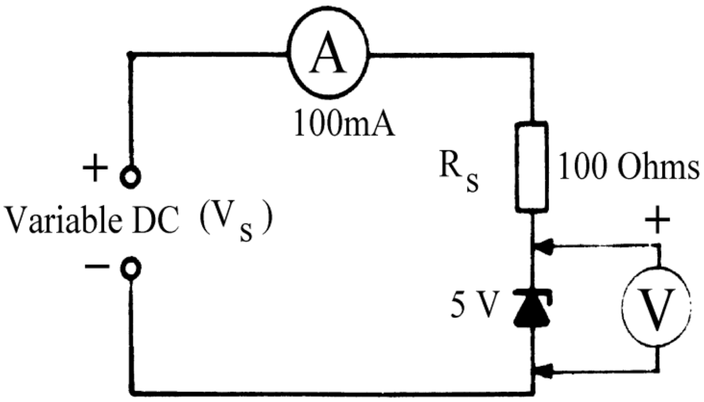
The characteristics and different applications for zener diodes are explained by means of the following experiments.

To do this experiment you need:

- A variable DC voltage supply (O-10 volts).
- An ammeter (O-100mA).
- A voltmeter (O-10V).
- A 100 ohm ½ watt 5% tolerance resistor.
- A 5 volt 1 watt zener diode.
- A circuit board.
- Connection wires.

1. Procedure

- Construct the circuit as shown in Fig. 2 on the next page. (Notice that the zener diode is reverse biased).



Zener diode in reverse biased condition

Figure 2

- Adjust the variable power supply in 1 volt steps from 1 to 10 volts and complete Table 1 by filling in the zener voltage (VZ) and the zener current (IZ) for each step.

TABLE 1. REVERSE BIASED ZENER DIODE

V_S	-1	-2	-3	-4	-5	-6	-7	-8	-9	-10
V_Z										
I_Z										

It will be seen from Table 1 that when a zener diode is reverse biased, no current flows until the rated zener voltage is reached. Beyond this point VZ stays constant and IZ increases as the supply voltage is increased.

- Reverse the zener diode in the above circuit so that the zener diode is forward biased.

- Again adjust the voltage in steps between O and -10V as shown in Table 2 and complete the table by filling in VZ and IZ for each step.

TABLE 2:FORWARD BIASED ZENER DIODE

V_S	+0,2	+0,4	+0,6	+0,8	+1	+2	+4	+6	+8	+10
V_F										
I_F										

From Table 2 it is obvious that when a zener diode is forward biased, it has characteristics similar to a forward biased diode.

- Plot from Table 1 the zener current (IZ) as a function of a zener voltage (VZ) on the reverse characteristic axis of Fig. 3. Also plot from Table 2 forward current (IF) as a function of forward breakover voltage (VF) on the forward characteristic axis of Fig. 3.

It will be seen that although IZ changes, VZ stays relatively constant.

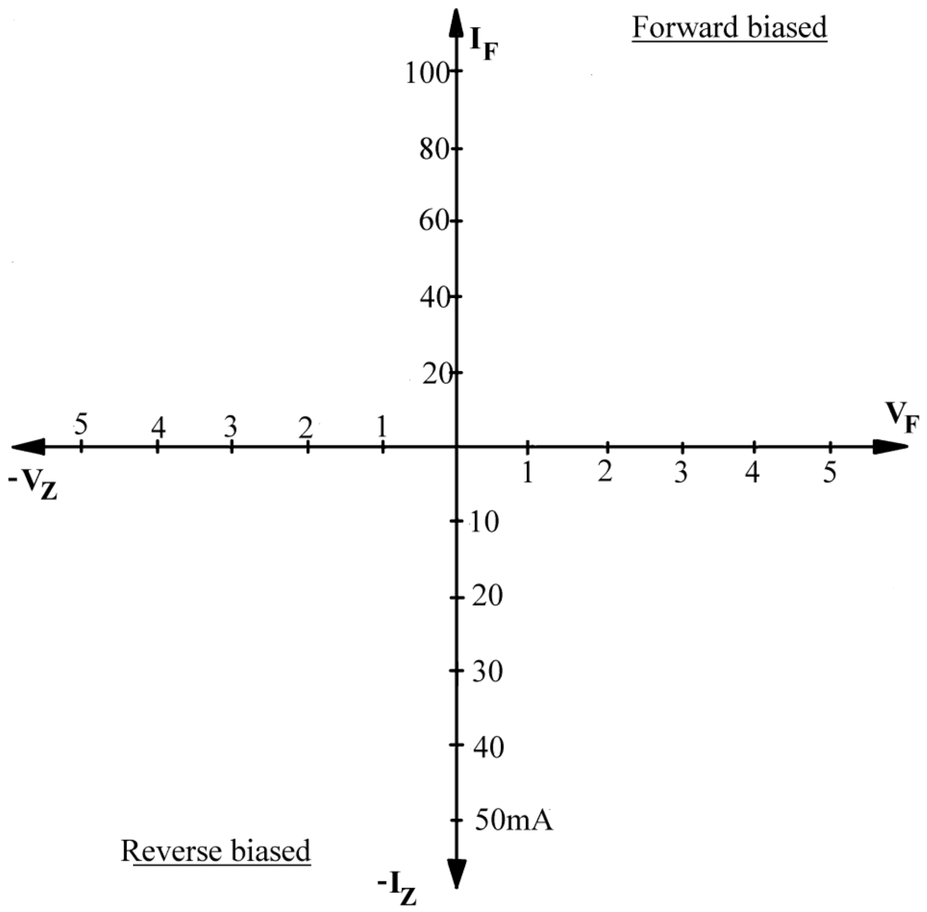


Figure 3

3. Conclusions

i) A zener diode provides a constant voltage across it even from a supply source, of which the voltage may vary appreciably.

ii) Because the zener voltage stays constant over a range of currents, a zener diode is known as a constant voltage device.

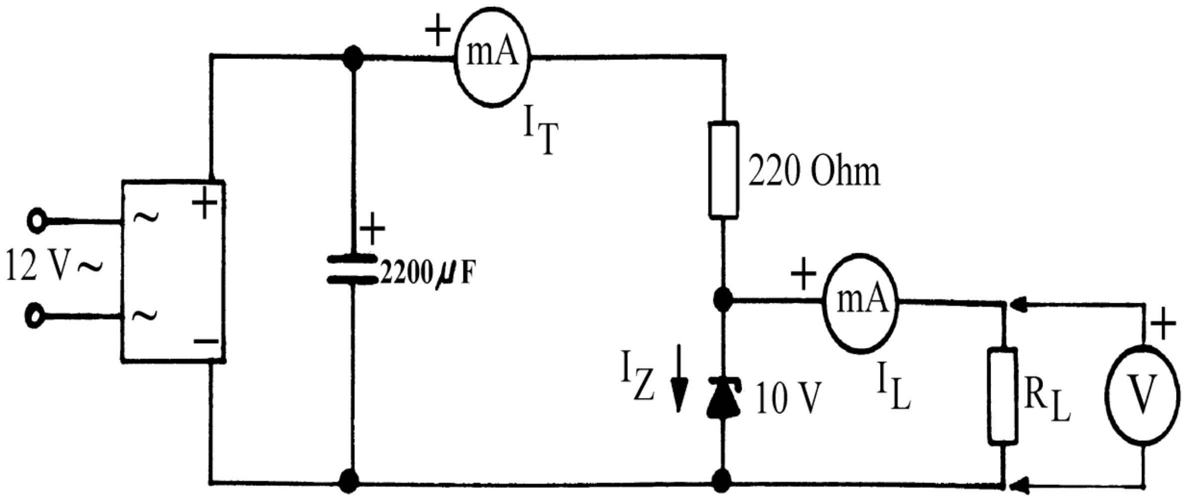
iii) The reasons for connecting the series resistor (RS) are to limit the current through the diode to a safe value and to drop excess voltage.

Experiment 1 shows how the zener voltage remains constant with a varying supply voltage. In the next experiment you will see what happens to the zener voltage when the load current varies.

	Experiment 2

To do this experiment you need:

- A 12 volt AC supply.
- Two ammeters (0-100mA).
- A voltmeter (0-20V).
- A bridge rectifier (50V, 1A).
- A 2 20O F electrolytic capacitor.
- A 10V, 1W, zener diode.
- 10K, 4K7, 2K2, 1K, 470Ω, 330Ω, 220Ω, and 2 x 150Ω ½ watt resistors.
- A circuit board.
- Connecting wires.

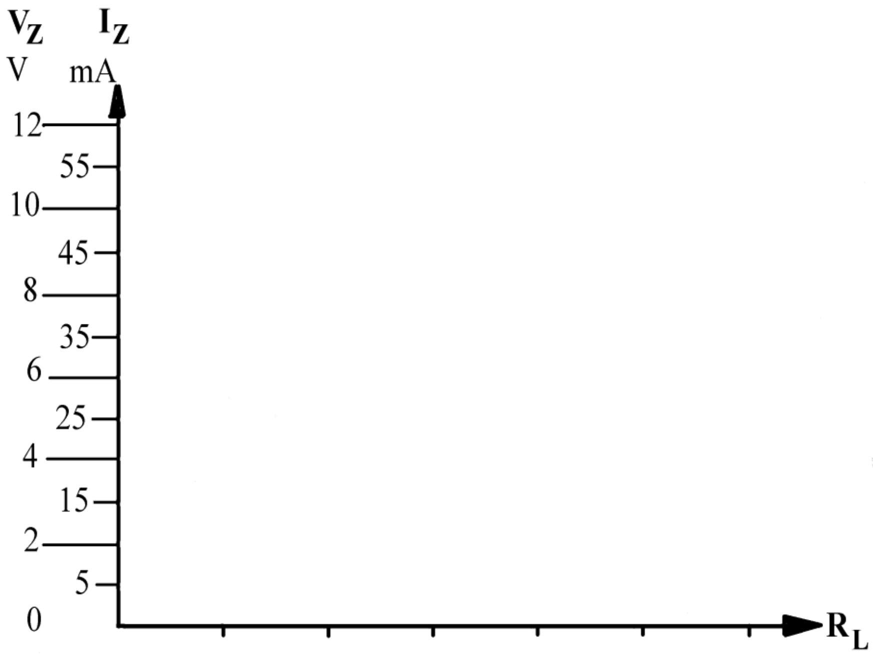


Load on Zener diode

Figure 4

1. Procedure

- Construct the circuit of a zener parallel regulator as shown in Fig. 4.



Curves for different loads

Figure 5

- Insert each value of resistor for RL in the circuit and measure and record all the values asked for in Table 3 below.

TABLE 3: LOAD RESISTOR VALUES

R_L	10K	4K7	2K2	470	330	220	150
V_Z							
I_Z							
I_L							
$I_T = I_Z + I_L$							

- Using the results shown in Table 3, plot the following curves on Fig. 5.

  i) Zener voltage as a function of load resistance. ($V_z = f(R_L)$).

  ii) Zener current as a function of load resistance. ($I_z = f(R_L)$).

2. Conclusions

   i) From the curves above it can be seen that VZ can only be constant while IZ is above a certain minimum value.

   ii) $_{OUT}$ is constant because VZ____________________

   iii) The relationship between IT, IZ and IL is :

   IT = __________ + __________

   If you have completed the above section and if it is clear to you, do the self test on the next page.

   If, however, it is still not clear, read the notes again before attempting the self test.

Self test 1

Answer the questions below without referring to your notes.

1. In which direction is a zener diode biased in a circuit to act as a voltage regulator?

   ____

2. Draw the circuit symbol for a zener diode.

3. State the reasons why a series resistor (RS) is connected in a zener parallel voltage regulator circuit.

   ____

4. Is the zener voltage constant when the supply voltage
   Varies?

   ____

5. Is the zener voltage constant when the load current varies?

   ____

2. *The current capability of a zener diode*

All zener diodes have a certain specified voltage (VZ) and power (PS) rating and these values are used to calculate the maximum and minimum current ratings. These maximum and minimum current values within which the zener diode must operate to keep the voltage constant across the diode, forms the limits that were mentioned earlier in the introduction.

This is shown in the following example where the zener diode is specified as 10V. 1W.

$$IZ \text{ max.rating} = \frac{\text{Power in watts (PS)}}{VZ}$$

$$= \frac{1}{10} = 0{,}1 \text{ amperes}$$

$$= \underline{100mA}$$

The maximum current that should be allowed through the zener diode with a safety factor of 20%, is 100% - 20% = 80% or 0, 8 of the IZ maximum rated value.

Therefore

IZ max permissible = 0,8 x 100mA

= <u>80mA</u>.

Normally IZ min. to keep VZ constant is, taken in practice as follows:

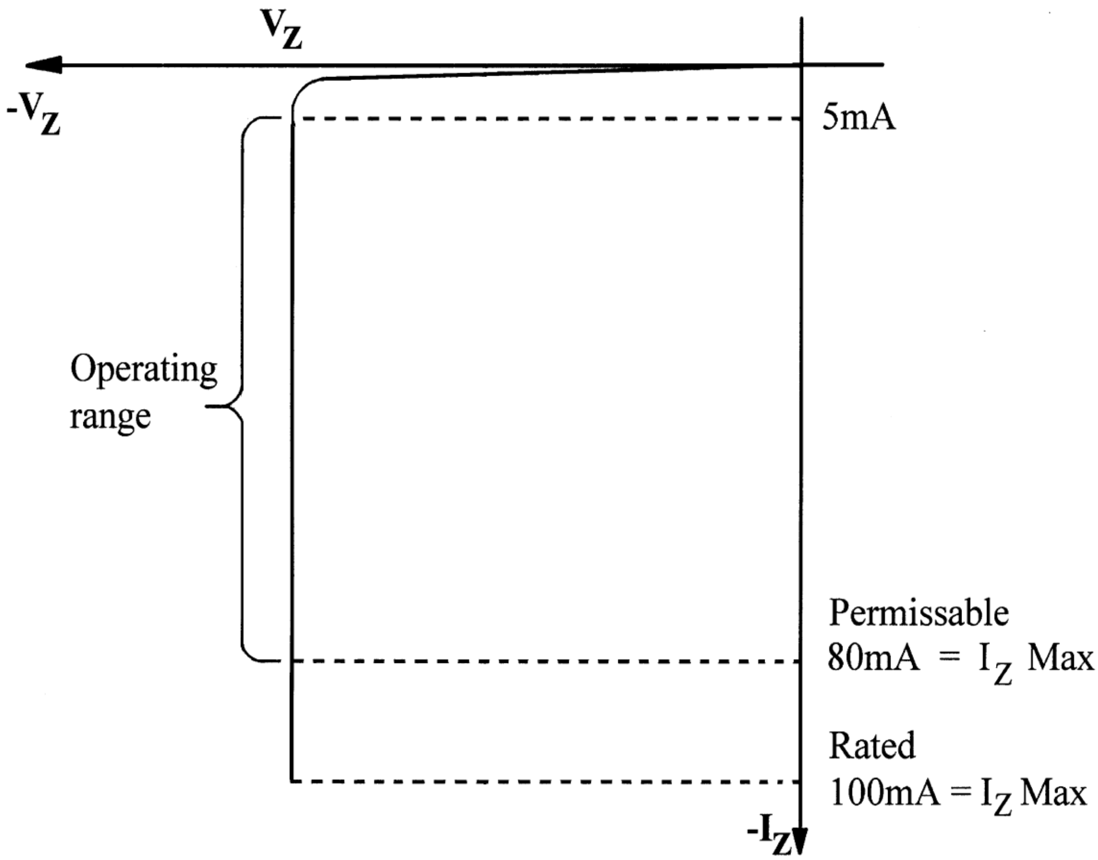
PS up to 1W, then IZ min. = 5mA

PS above 1W, then IZ min. = 10mA.

The above limits can be indicated on the IZ/VZ characteristic curve to show the <u>operating range</u> as follows (Fig. 6) on the next page :

If IZ stays between these limits then VZ stays constant within its tolerance.

It is important to understand this section on the current capabilities of zener diodes, because otherwise they will be misused and destroyed. Always work in the operating range of the zener diode.



Operating range of a Zener diode

Figure 6

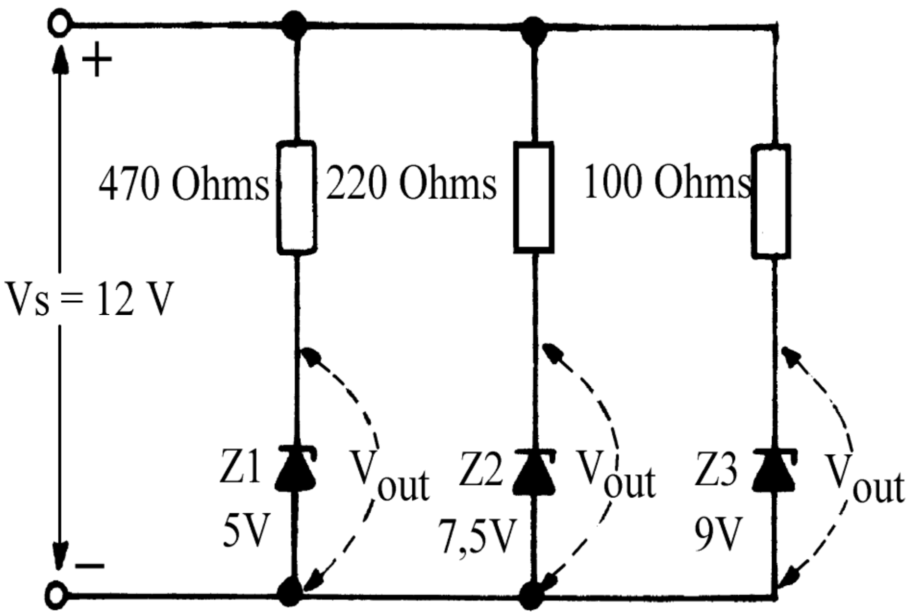
	Experiment 3

To do this experiment you need:

- A 12 volt DC supply.
- A voltmeter (O-10 volt F.S.D.) or a multimeter.
- Three zener diodes with voltage ratings: 5V, 7,5V and 9V.
- 4.Three resistors, 470 ohm, 220 ohm and100 ohm ½ watt.
- A circuit board.
- Connecting wires.

1. Procedure

    - Construct the circuit as shown in Fig. 7 on the next page.



Unknown Zener diodes in parallel

Figure 7

The purpose of the resistors in series with each zener diode is to

____

- Measure the voltage across each zener.

  VZ1 = ____________________

  VZ2 = ____________________

  VZ3 = ____________________

- Calculate the current through each zener diode and indicate the operating points for zener voltage and current on the axis of Fig. 8 on the next page.

- Draw a circuit diagram showing how different voltages may be obtained from one supply, by means of zener diodes.

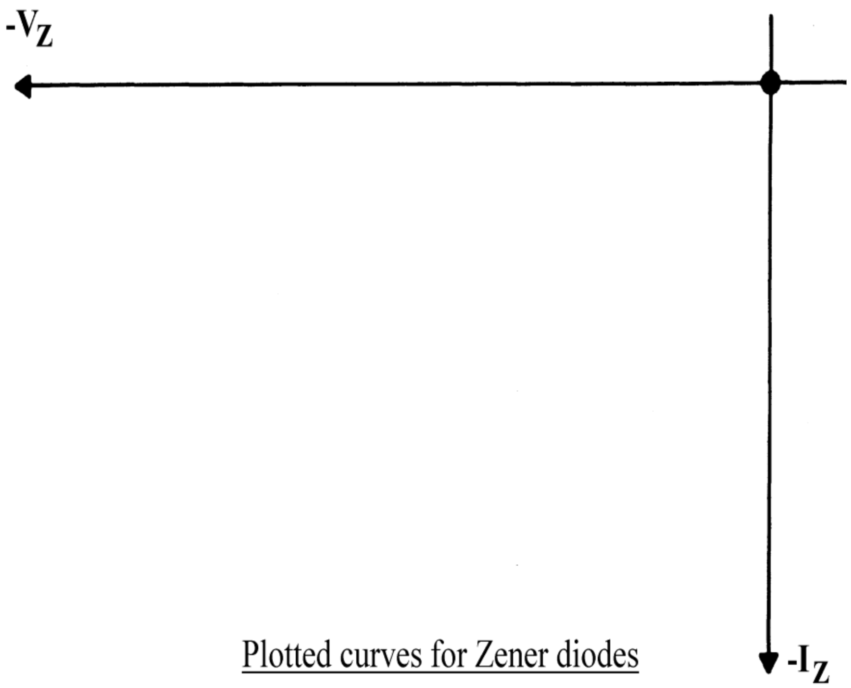
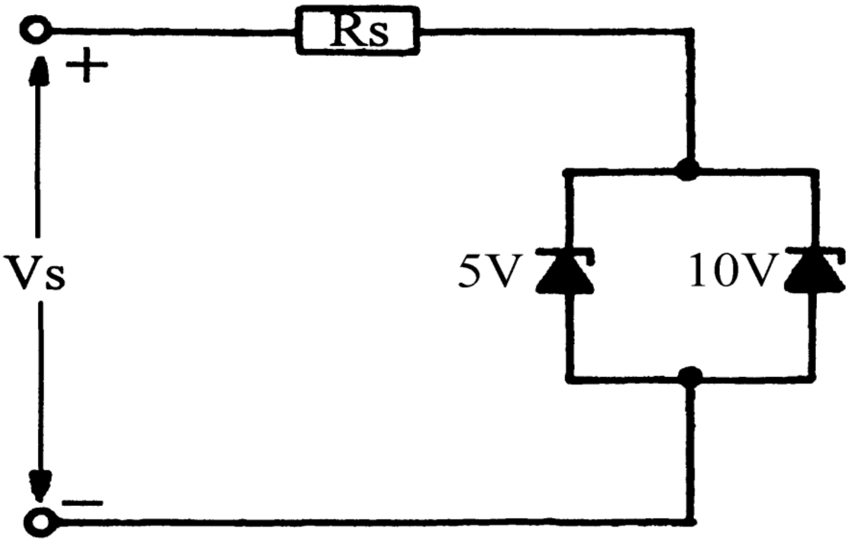


Figure 8

3. *Zeners in parallel*

Imagine two zeners of say 5 volts and 10 volts are connected in parallel as shown in Fig. 9.



Zener diodes with different voltage ratings

Figure 9

It is obvious that the 5V zener will start conducting before the other 10V zener and if you think back to experiment 1, you will remember that when the zener voltage was reached, it remained constant, even when the supply voltage was increased. Therefore, the break through voltage (zener voltage) of the 10V zener will never be reached. The current through the 5V zener will only increase if the voltage is increased and will continue to increase until the zener is destroyed

This can further be explained by imagining two zeners with 12 Volt ½ Watt ratings connected in parallel with the view that this will be the same as using a 12 Volt 1 Watt zener.

It can be reasoned that one 12V zener, <u>due to tolerance differences</u>, will start conducting before the other 12V zener and again if you think back to experiment 1, you will remember that when the zener voltage is reached, it remained constant, even when the supply voltage was increased.

Therefore the break through voltage (zener voltage) of the zener with the slightly higher value will also never be reached. The current through the 12V zener with the slightly lower value will increase when the voltage is increased and will again continue to increase until the zener is again destroyed.

Therefore the break through voltage (zener voltage) of the zener with the slightly higher value will also never be reached. The current through the 12V zener with the slightly lower value will increase when the voltage is increased and will again continue to increase until the zener is again destroyed.

Experiment 4

It will be clear from the explanation above, that when two zeners of the same voltage ratings are connected in parallel, the output voltage will be that of the lower voltage zener and therefore it will do all the work.

To do this experiment you need:

- A 20 volt DC supply.
- A multimeter.
- Two unmarked zener diodes (unknown voltages).
- One 1N4007 diode.
- A 1 kilohm ½ watt resistor.
- A circuit board.
- Connecting wires.

1. Procedure

    - Construct the circuit as shown in Fig. 10 on the next page.
    - Measure the voltage over each diode.

      VZ1 = ______________

      VZ2 = ______________

      VD1 = ______________

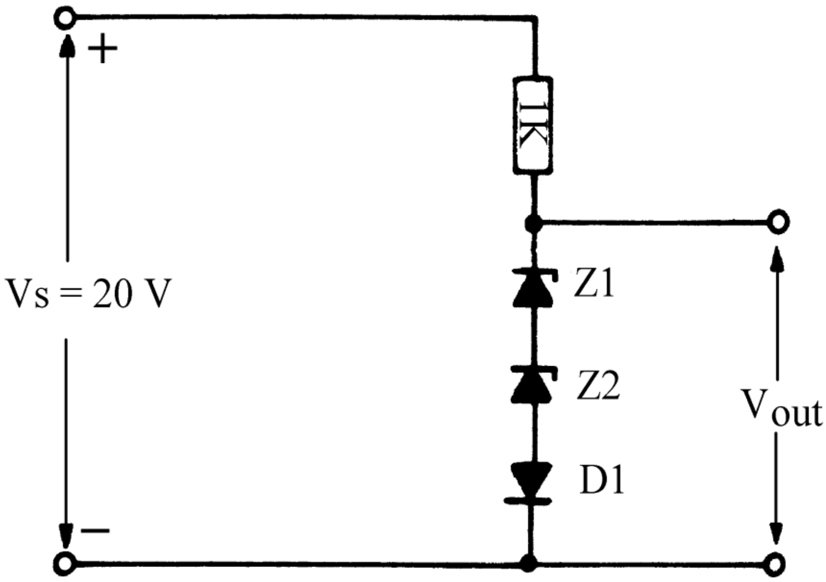
    - Measure the output voltage.

      V_{OUT}= ___________

      It can be seen that:

      V_{OUT} = ___________ + ___________ + ___________

- Different zeners and/or diodes can thus be connected in __________ to obtain a specific voltage.



Unknown Zener diodes in a circuit

Figure 10

Self test

Answer the questions below without referring to your notes.

1. What is the IZ maximum rated and the IZ permissible maximum values for a 20V. 10W. zener diode?

2. To keep the zener voltage (VZ) constant, what minimum current (IZ) must be passed through the diode in question 1 above?

3. Explain what the effect would be if two zener diodes of the same voltage ratings are connected in parallel.

4. What happens to the output voltage when zener diodes are connected in series?

The next experiment, on the next page and also the last in this module, deals with connecting zeners in a circuit, to obtain bi-directional voltage clamping.

Experiment 5

To do this experiment you need:

- A variable AC voltage supply. (Variac 0-15 volts).
- Two 10V. 1W zener diodes.
- A 1 kilohm ½ watt resistor.
- A multimeter.
- A circuit board.
- Connecting wires.
- An oscilloscope.

1. Procedure

- Construct the voltage clamping circuit as shown in Fig. 11.

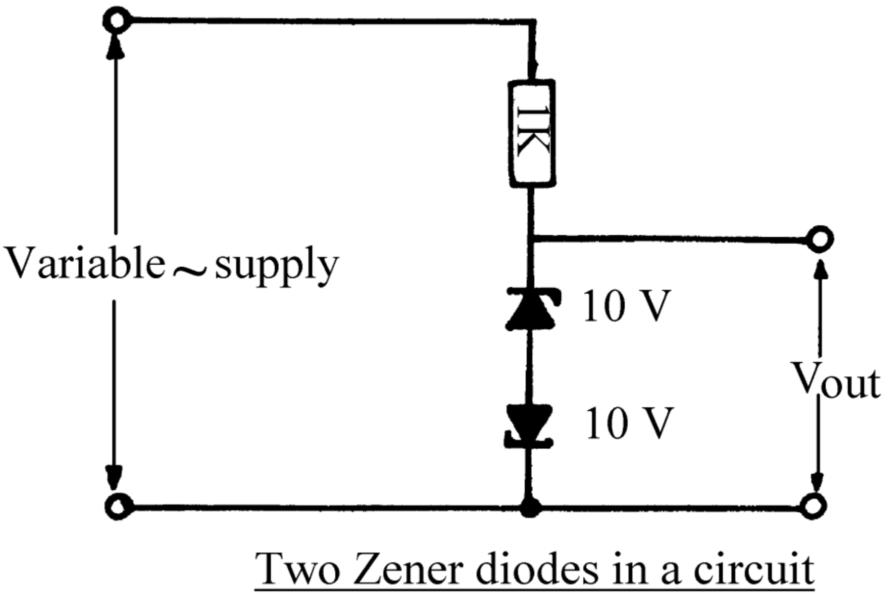
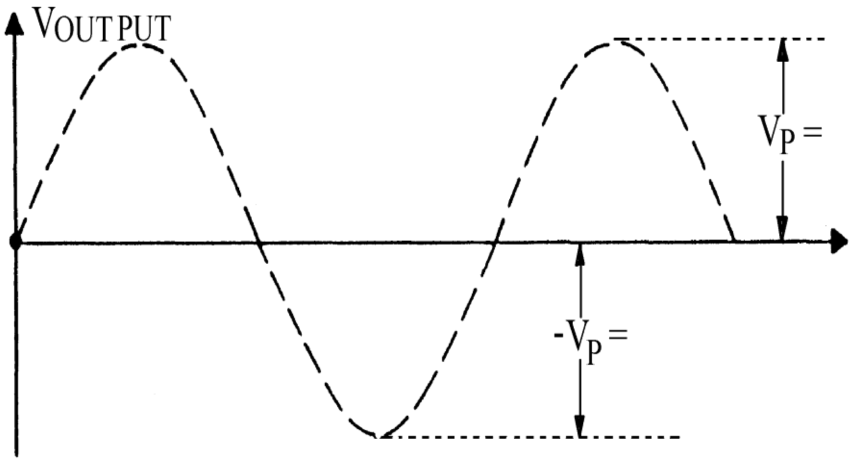


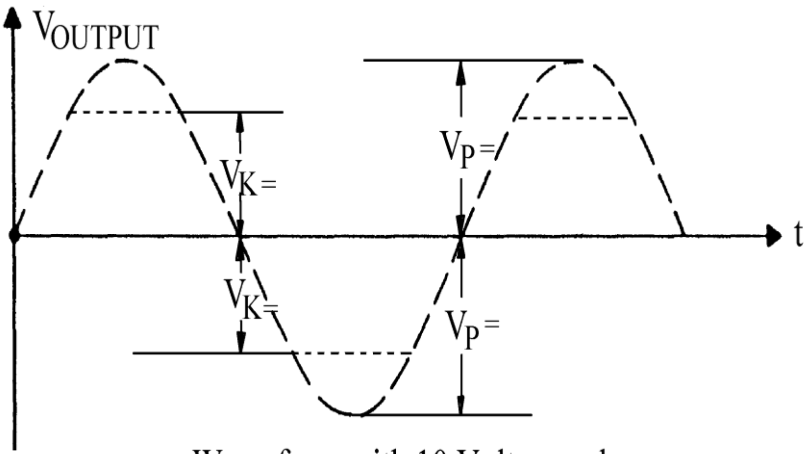
Figure 11

- Adjust the variable AC supply to 5 volts.
- Connect the oscilloscope to the output of the circuit and set the oscilloscope coupling control to DC, measure Vp and note it down on Fig. 12 on the next page.
- Connect the multimeter to the output of the variable AC supply.
- Draw the output waveform on the axes of Fig. 12 on the next page.
- Adjust the variable output to 10V rms measured with the multimeter and draw the output waveform as displayed on the oscilloscope on the axis of Fig. 13 indicating the clamping effect as well as the clamp voltage (VK).



Wave form with 5 volts supply

Figure 12



Wave form with 10 Volts supply

Figure 13

- Calculate the peak voltage for 10 V_{rms} and write this value next to Vp on the axis of Fig. 13.

2. Conclusion

Refer to Fig. 11 on the next page when reading the section below the figure.

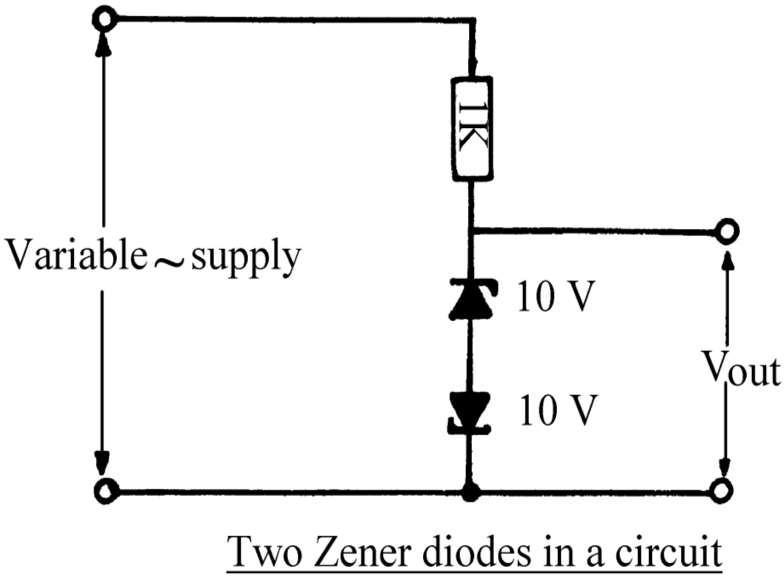


Figure 11

The clamp voltage (VK) is equal to the zener voltage (VZ) of the zener that is reverse biased at that instant of the waveform, plus the forward volt drop (VF) of the zener diode that is forward biased at that instant of the waveform. Each cycle of the output will be clamped at a positive and negative value of:

VK=VZ + VF

=......... volts.

If the experiment is not clear to you do it over again.

	Ask for the criterion test when you feel ready.

Module 8
BE - 7 Construct a voltage doubler

Purpose of module

This objective will enable the learner to construct a voltage doubler

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Construct a voltage doubler.
2. Measure the input and output voltage.
3. State whether a voltage doubler output is AC or DC.
4. State the ratio between the input and output voltage.
5. Explain what would happen to the output voltage of a voltage doubler if the load was increased.

Procedures Relating To This Module

- The voltage doubler must be correctly constructed and the input and output voltages measured correctly.
- Where applicable all answers must be in writing.
- All answers must be correct and in accordance with the module notes.

1. Introduction

An advantage of a voltage doubler is that it doubles the input voltage; therefore the ratio of the input to the output voltage is 1:2. The voltage is doubled without using a transformer. In this way cost and space can be reduced. Therefore a voltage doubler is convenient when a high voltage low-current output is required.

The circuit for one type of voltage doubler is shown in Fig.1.

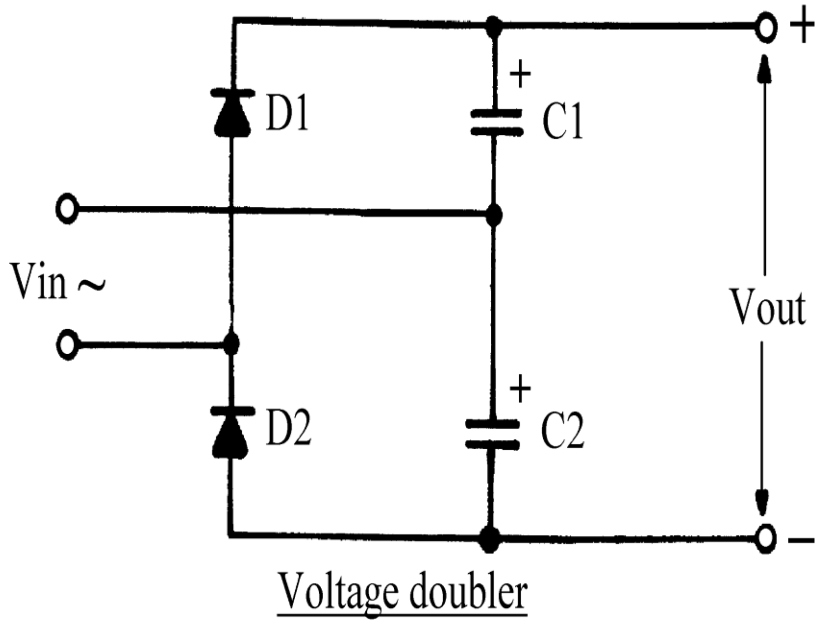


Figure 1

When the top end of the AC supply line is positive, current flows through D1 and C1. The capacitor C1 becomes charged to the peak value of the supply. When the alternating current reverses and the bottom end of the AC supply line becomes positive, current flows through C2 and D2, thus C2 becomes charged to the peak value of the supply.

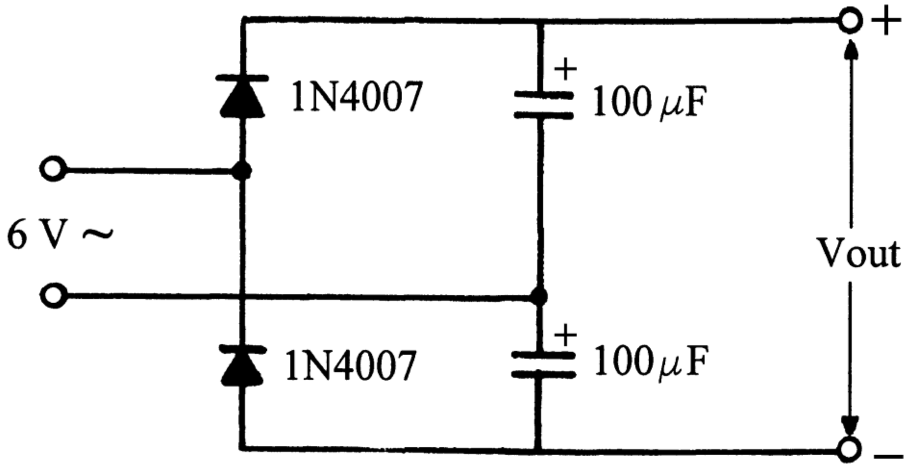
The voltage across capacitor C1 and C2 is in series and therefore is added together, thus the voltage output is twice the peak input voltage.

The explanation above can be proved by carrying out the experiment overleaf.

	Experiment 1

To do this experiment you need:

- A 6 volt AC supply.
- Two 1N4007 diodes.
- Two 100 µF 25V electrolytic capacitors.
- A 100 kilohm and a 470 ohm ½ watt resistor.
- A multimeter.
- A circuit board.
- Connecting wires.



Construction of a voltage doubler

Figure 2

1. Procedure

- Construct the circuit as shown in Fig. 2.
- Measure V_{IN}____________________Volts
- Calculate the peak value (Vp) of the AC supply using the formula:
- Measure V_{OUT}____________________ Volt.
- Determine the ratio of V_{OUT} to V_{IN} peak

$$\frac{V_{OUT}}{V_{IN}\,(peak)} =$$

- The above proves that V_{OUT} = __________x V_{IN} peak, hence the name voltage doubler.
- Load the circuit at the output with a 100 ohm and 470 ohm resistor in turn and note:

  a) V_{OUT} with 100 ohm load resistor

  ____

  b) V_{OUT} with 470 ohm load resistor

  ____

2. Conclusion

- A voltage doubler has a voltage ratio of one to two (1:2).
- The output voltage of a voltage doubler is DC.
- A voltage doubler can only be used where low currents are drawn, e.g. oscilloscope and television tubes to generate the necessary high voltage.

Self test

1. Answer the questions below without referring to your notes.

   a) Is a voltage doubler output AC or DC? ?

   ____

   b) What is the ratio between input and output voltage?

   ____

   c) What would happen to the output voltage of a voltage doubler if the load resistance was decreased?

   ____

2. Calculate the output voltage of a voltage doubler if the input voltage is 100V AC.

   ____

Ask for the criterion test when you feel ready.

NOTES

Module 9
BE - 8 Identify transistor action

Purpose of module

This objective will enable the learner to identify transistor action

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Construct a circuit with a transistor.
2. State what is meant by the term “transistor action".
3. State the relationship between Ie, Ib and Ic.
4. State two kinds of semi conductor material used in a transistor.
5. Name the three transistor leads.
6. Name two types of transistors.
7. Draw the circuit symbols for the two types of transistors.
8. State how a b e junction is biased to get the transistor to conduct.
9. Draw batteries in a circuit for a transistor biasing.
10. Indicate the direction of conventional current flow on a transistor circuit.
11. Indicate on a diagram, which terminal of Vbe and which terminal of Vbc is most positive.
12. State what the voltage is over a

    (a) Conducting and

    (b) Non conducting transistor.

Procedures Relating To This Module

- The circuit must be constructed correctly and the transistor action identified.
- Where applicable, all answers must be in writing.
- All answers must be correct and in accordance with the module notes.

1. Introduction

Scientific interest in semi-conductors led to the development of the transistor. An advantage of the transistor is that it is small and light, thus enabling the manufacture of miniature electronic equipment, e.g. radios, calculators, etc.

The transistor has no moving parts, operates at low voltages and currents, and thus uses little power.

2. Construction and symbols

The transistor is simply a sandwich made from two kinds of semi-conductor material which is known as either:

(i) An n-type material which is negatively charged, or

(ii) A p-type material which is positively charged.

Accordingly, two types of transistors are available: the NPN transistor and the PNP transistor. Three leads are brought out from the sandwich and are named the collector, base and emitter.

They are shown in Fig. 1.

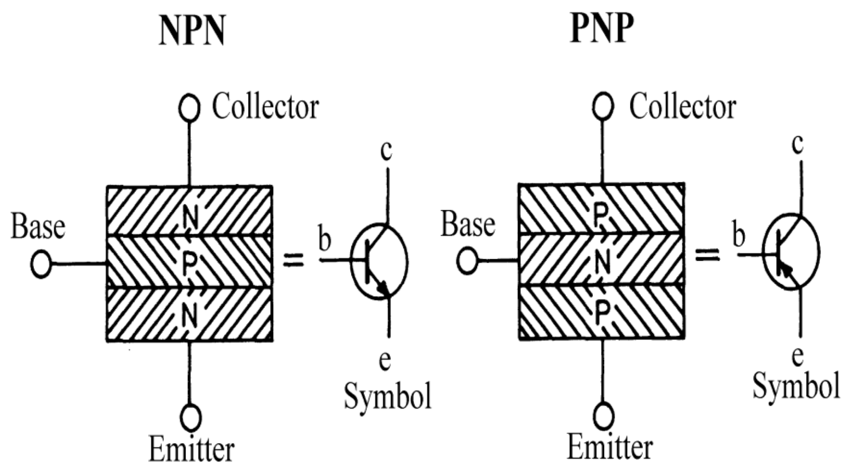


Figure 1

The principles of operation of an NPN and a NPN transistor are similar, except that all currents and potentials, are reversed. This feature enables a design to obtain many useful circuit configurations, in which the two transistors play complementary roles. The circuit symbols shown in Fig. 1 should be carefully studied, because circuits will fail, and transistors possibly destroyed, if the leads are mixed up.

The arrow is always the emitter but points in a different direction for NPN and PNP type transistors. (The arrow indicates the direction of conventional current flow from positive tonegative).

It is useful to know that a transistor may be considered as two diodes connected back to back. An NPN sandwich is therefore two diodes back to back, sharing a common anode in the middle and a PNP sandwich is two diodes back to back sharing a common cathode in the middle. This knowledge can be used advantageously for testing and biasing transistors.

Do the following experiment to determine if the transistors are in working order.

	Experiment 1

To do this experiment you need:

- A digital multimeter.
- An NPN transistor (2N2222).
- A PNP transistor (2N2907).

1. Procedure

- Study Fig. 2. An NPN transistor can only be drawn for testing and biasing purposes as follows:

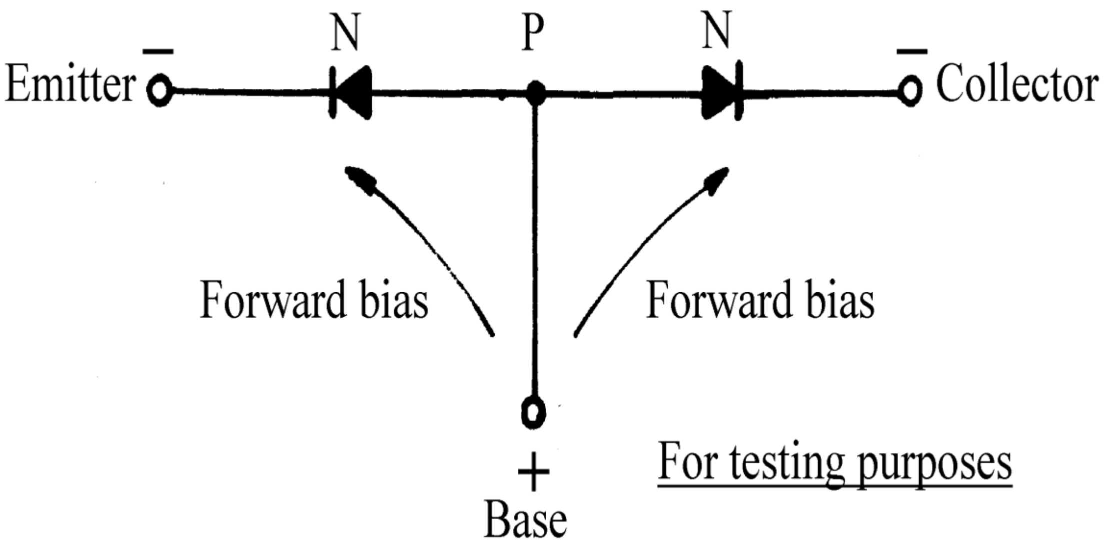


Figure 2

- Set the multimeter to the continuity scale.
(Note that when an analog multimeter is used, the Ohm scale should be selected, in which case the red lead takes on a negative polarity).

- Put the digital multimeter leads on any two of the NPN transistor leads. If the reading is very high, i.e. infinity or showing no conduction, it means that the multimeter leads are reverse biased over the transistor, in which case the multimeter leads should be changed around. If this still gives no conduction reading, the red meter lead should be moved and then a conduction reading of approximately 500 - 600 mV will be obtained on the meter.

A conduction reading, indicates that the multimeter is now forward biasing the two transistor leads. Since you are testing an NPN transistor and the red lead of the multimeter has a positive polarity on the continuity scale, the red lead of the multimeter will be connected to the base of the transistor.

- Remove the black lead of the multimeter from the transistor lead and connect it to the other transistor lead. There should also be a conducting reading across these two leads, indicating two imaginary diodes anode to anode and brought out as the base of the transistor.

- This test indicates that the transistor is healthy. It is, however, not the only and best method of testing. Special test instruments are available for testing transistors but are not necessary for our purposes.

To summarise the above mentioned method:

- Take an NPN transistor which is in a working condition and test it using the flow chart provided on the next page.

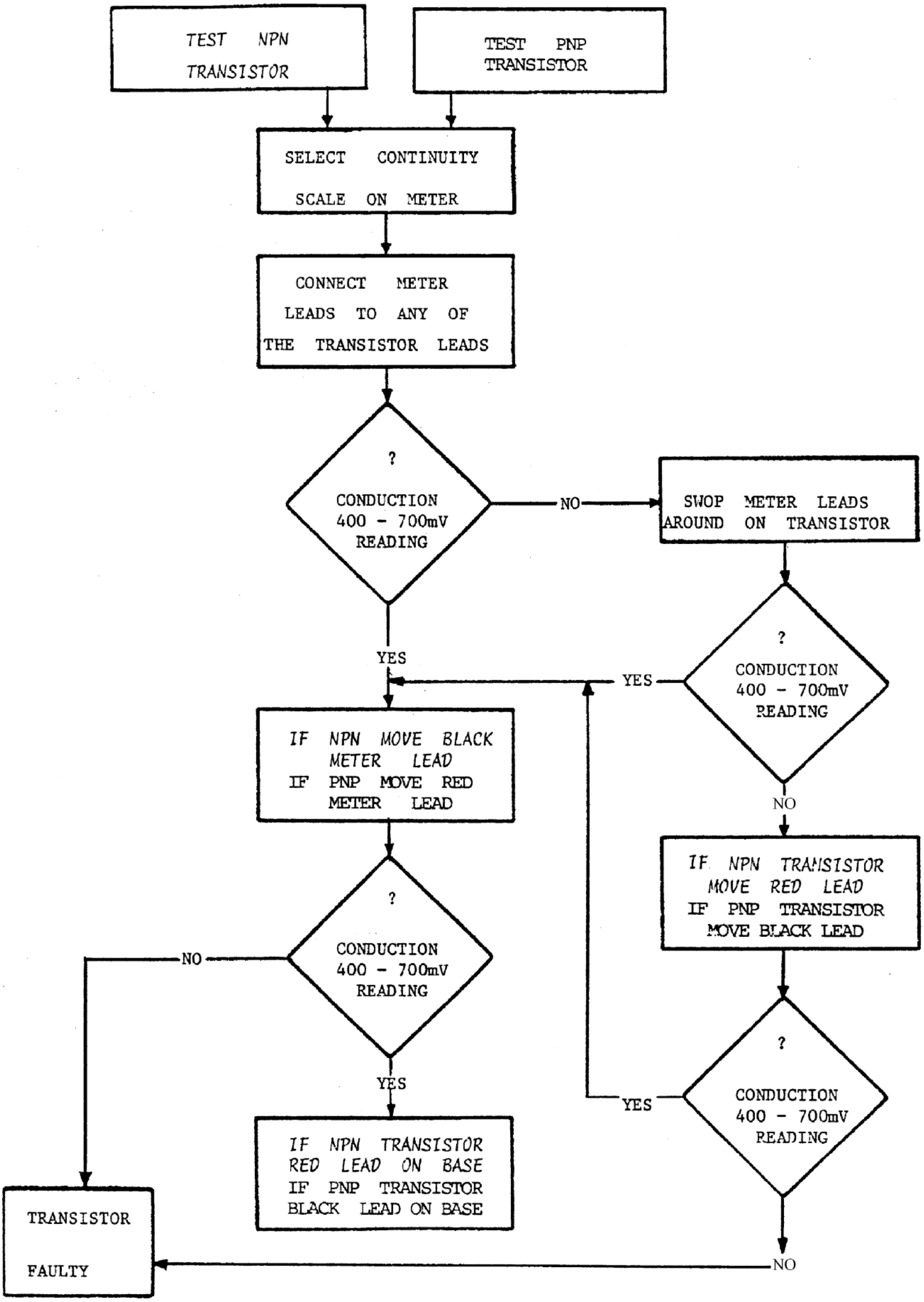
The flow chart is compiled for the testing of both NPN and PNP transistors.

While testing using the guidelines in the flow chart, there are only two readings of significance when the transistor is in a working condition:

- Conduction with a meter reading that falls well within the range 400 - 700mV, and,

- No conduction with the meter showing infinity.

Any reading that falls outside these parameters will be indicative of a faulty transistor.


TEST NPN TRANSISTOR
TEST PNP TRANSISTOR
SELECT CONTINUITY SCALE ON METER
CONNECT METER LEADS TO ANY OF THE TRANSISTOR LEADS
?
CONDUCTION 400 - 700mV READING
NO
SWOP METER LEADS AROUND ON TRANSISTOR
YES
?
CONDUCTION 400 - 700mV READING
YES
IF NPN MOVE BLACK METER LEAD
IF PNP MOVE RED METER LEAD
NO
IF NPN TRANSISTOR MOVE RED LEAD
IF PNP TRANSISTOR MOVE BLACK LEAD
?
CONDUCTION 400 - 700mV READING
NO
YES
?
CONDUCTION 400 - 700mV READING
YES
IF NPN TRANSISTOR RED LEAD ON BASE
IF PNP TRANSISTOR BLACK LEAD ON BASE
TRANSISTOR FAULTY
NO

- Carry out the same test on a PNP transistor by using the flow chart and a multimeter. Fig. 3 shows the analogy for a PNP transistor for testing purposes.

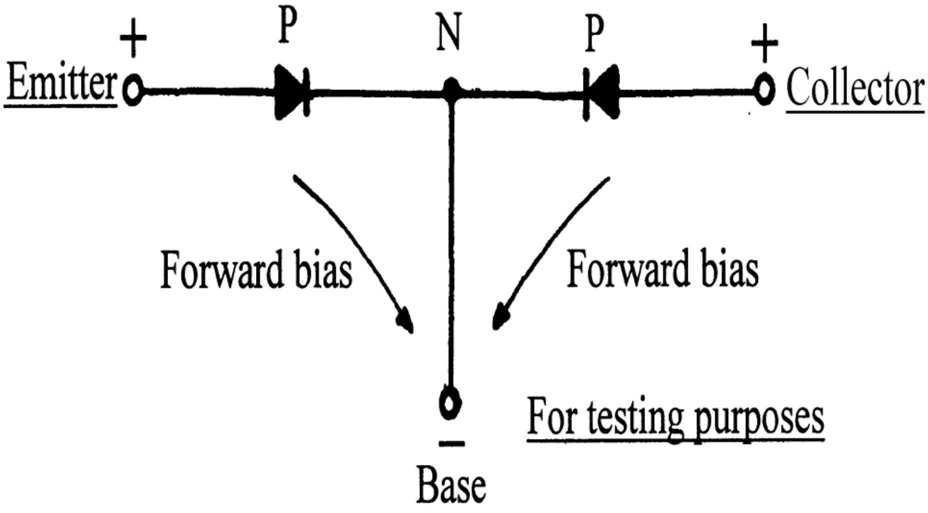


Figure 3

The correct biasing conditions for normal in circuit operating conditions of a transistor are as follows:

a) The b-e junction (between base and emitter) must always be forward biased. Shown in Fig. 4.

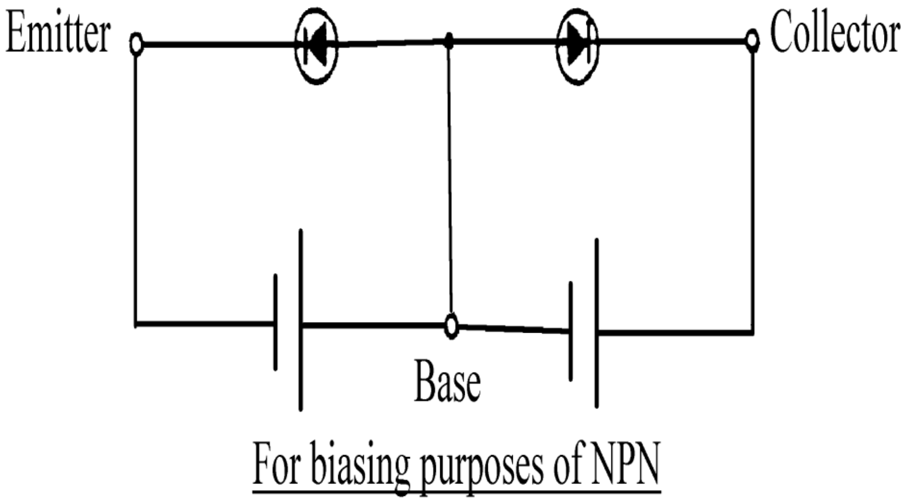


Figure 4

b) The b-c junction (between base and collector) must always be reverse biased. Shown in Fig. 5.

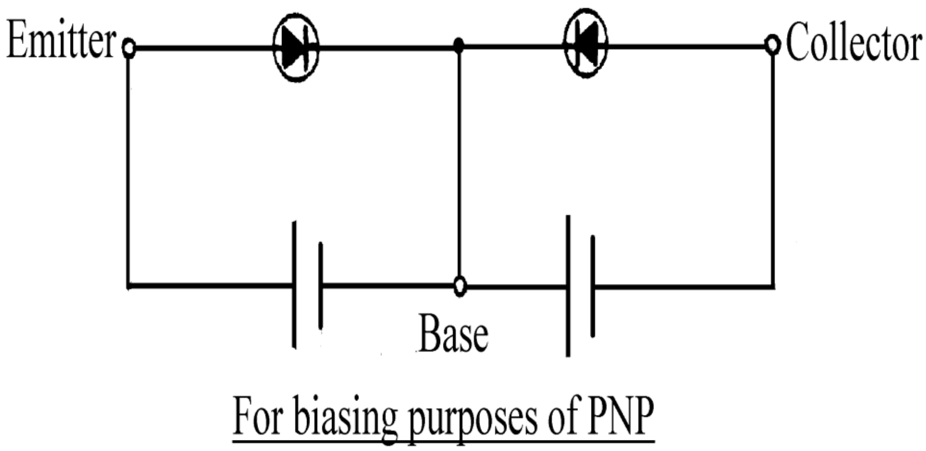


Figure 5

- Use the knowledge gained above and draw the batteries in on the circuits shown in Fig. 6 as illustrated below so as to bias the transistors correctly.

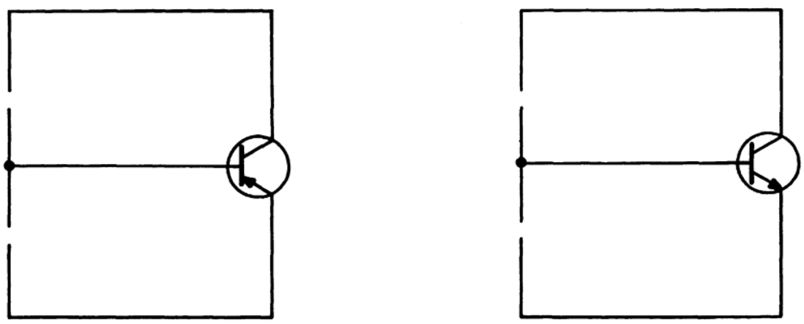
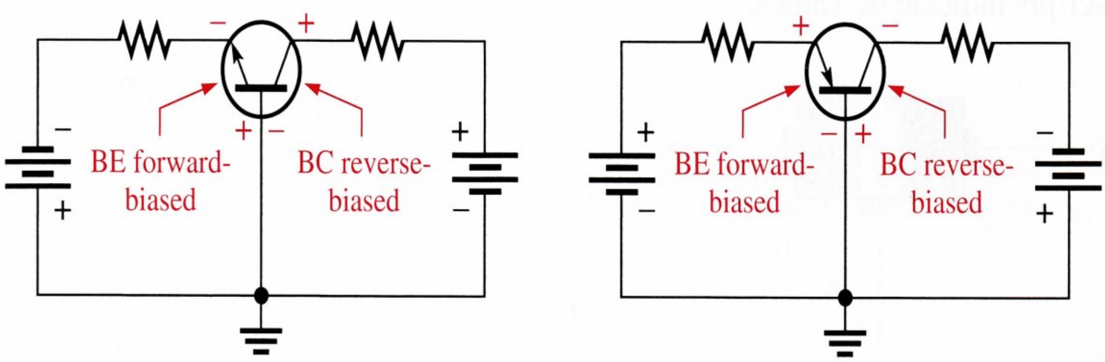


Figure 6



	Self test 1

Answer the questions below without referring to your notes.

1. Name the two kinds of semi-conductor material used in a transistor.

___

2. What are the names of the three transistor leads?

___

3. Name two types of transistors, draw their circuit symbols and label them.

4. Show how an NPN transistor can be drawn for testing and biasing purposes.

5. How is a b-e junction always biased?

___

6. How is a b-c junction always biased?

___

7. Draw in batteries on the circuits in Fig.6 on the next page so as to bias the transistors correctly.

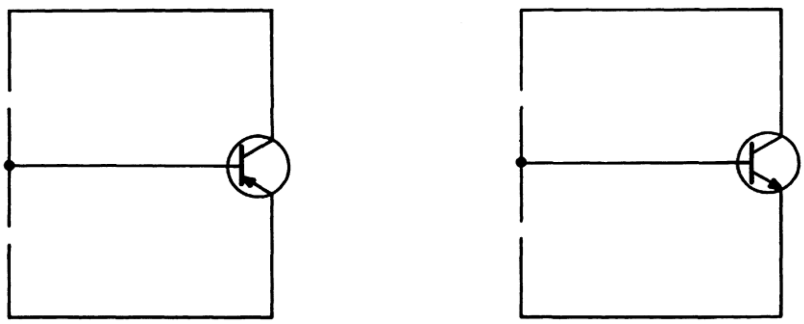


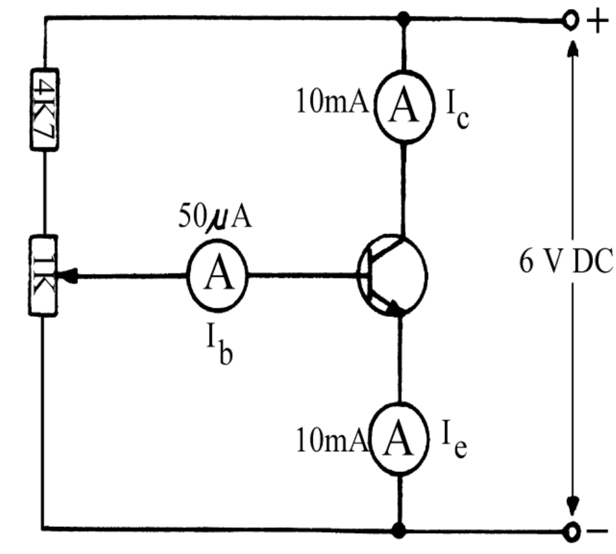
Figure 6

- Check your answers with your notes.
- Find and correct your mistakes if your answers are not all correct.
- If they are all correct, congratulations. Ask your Facilitator for some healthy and faulty transistors to test.
- Take your results to your Facilitator to check.

	Experiment 2

To do this experiment you need:

- A 4 volt DC supply.
- An NPN transistor (2N2222).
- A 4K7 ½ watt resistor.
- A 1K potentiometer.
- Three identical ammeters installed as indicated in the circuit diagram, Fig. 7.
- A circuit board.
- Connecting wires.



Transistor in a circuit

Figure 7

1. Procedure

- Construct the circuit as shown in Fig. 7.
- Adjust the potentiometer (p) until the ammeter for Ie measures 10mA and note down the following readings:

Ib = __________ µA

Ic = __________ mA

- Increase the base current Ib by 10 µA and note down the following readings:

  Ib -------------- µA

  Ic --------------- mA

- From the results above it should be clear that a small change in Ib causes a large change in Ic. Thus a small base current can control a relatively large collector current. This is known as transistor action.

  The relationship between Ie, Ib and Icis :

  Ie = Ib + Ic

- Indicate with arrows on the circuit diagram (Fig. 5) the direction of Ie, Ib, and Ic.
- Measure the following voltages, indicating which one of the two terminals is most positive when 10mA is on the emitter ammeter as in point 2.

  Vbe = --------The-------- is the most positive.

  Vbc = --------The-------- is the most positive.

2. Conclusions

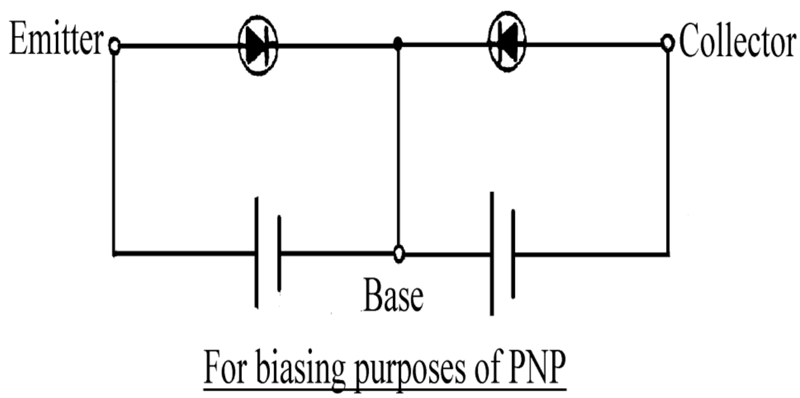


Figure 5

2.1 From the voltages above, it can be seen that the resistor is correctly biased, because:

The b - e junction is ____________________ biased and

The b - c junction is ____________________ biased.

2.2 To forward bias the b - e junction of an NPN transistor and therefore bring it into conduction, the base voltage (Vb) must be at least 0,6V more positive than the voltage on the emitter (Ve) and in the case of the PNP transistor, the emitter voltage (Ve) must be at least 0,6V more positive than the base voltage (Vb).

2.3 Therefore once a transistor is in conduction, the relatively large __________ current is being controlled by a relatively small __________ current, which means that if Ib is increased then Ic__________ and if Ib is decreased, then Ic__________.

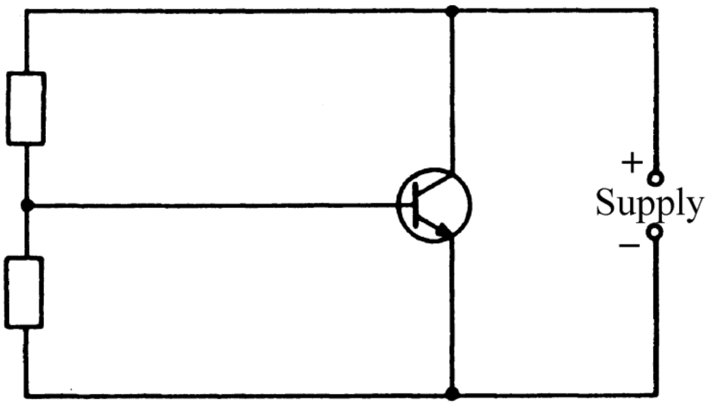
Self test 2

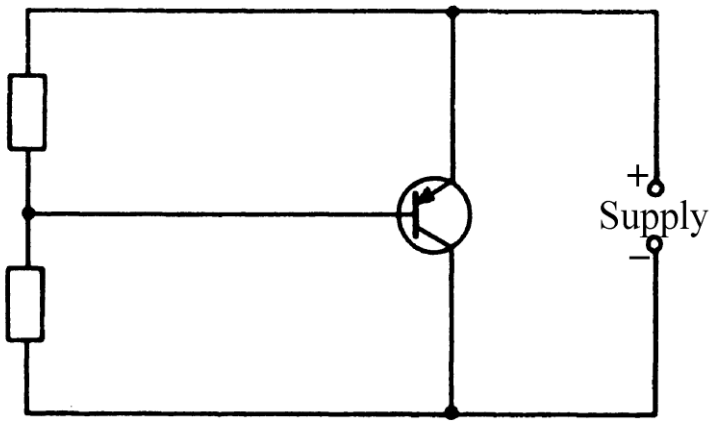
Answer the following questions without referring to your notes.

1. What is transistor action?

2. What is the relationship between Ie, Ib and Ic?

3. Indicate the direction of current flow for Ie, Ib and Ic in both the circuits below. (Conventional flow).





4. In the case of an NPN transistor, which terminal of Vbe and which terminal of Vbc is the most positive?

5. When is a transistor correctly biased?

____

____

- Find and correct your mistakes if your answers are not all correct.
- If they are correct, carry on with the rest of the module.

2.4 The transistor can be used in one of two modes:

(a) In the linear mode

(b) In the switching mode.

When very small electrical signals from transducers like microphones, thermocouples and radio receivers needs to be amplified, the transistor is used in the linear mode and therefore as an amplifier (more about this in modules Be-9 and IE-1). Linear mode means that the transistor operates where Ib and therefore Ic is gradually increased and decreased, preventing Ib being driven to the extremes of maximum mA or zero mA.

The second mode of operation, i.e. the switching mode is where the transistor is an electronic switch. A signal on the base of the transistor, determines whether the transistor is fully on (conducting) or fully off (not conducting).

To understand the explanation above, carry out the following experiment.

	Experiment 3

To do this experiment you need:

- A 12 volt DC supply.
- An NPN transistor (2N5496) or equivalent e.g. TIP 41C.
- A 1000 ohm potentiometer (2 watt).
- A 2K7 resistor (½ watt).
- A 12 volt lamp.
- A SPST toggle switch.
- A circuit board.
- Connecting wires.
- A multimeter.

1. Procedure

- Construct and test the circuit as shown in Fig. 8 on the next page.

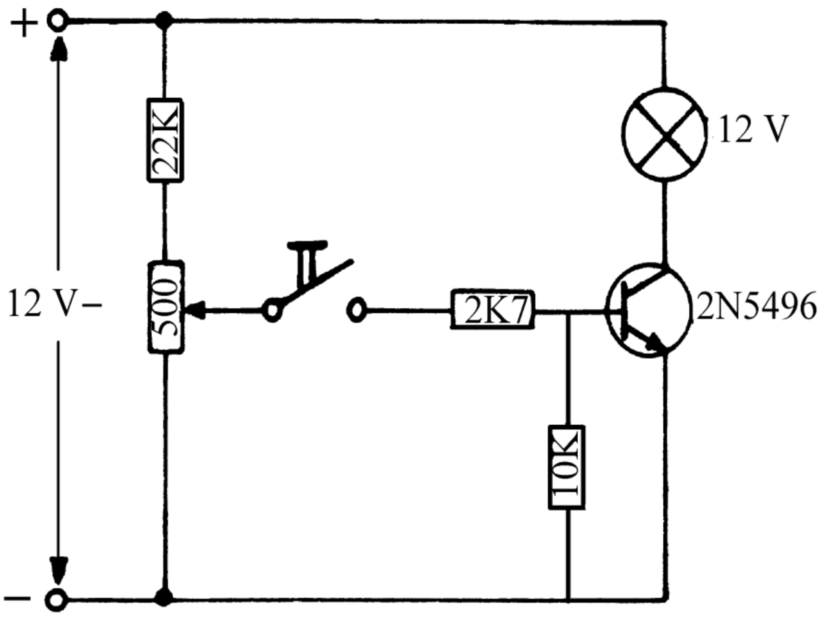


Figure 8

- Switch on the supply with switch S in the off position.
- Measure the voltage over the collector and emitter and record the voltage ________V.

  Is the lamp burning? _________________.

  Is the transistor acting as an on or off switch? _______________.

  Consequently we can say that Ib = ___________ mA and therefore Ic will also be equal to ___________ mA.
- Switch the switch S on.
- Adjust the potentiometer until the lamp is at full brilliance.
- Again measure and record the voltage over the collector and emitter ____________V.
- Measure and record the voltage over the lamp ________________V.

  Is the transistor acting as an on or off switch? _____________________.
- Again connect the voltmeter across the collector and emitter and adjust the potentiometer until the meter reads 6 volts.
- Adjust the potentiometer so that Ice varies from 6V to 10V to 2V.
- Note that the lamp's brilliance gradually changes and therefore the transistor is operating in the linear mode or region.

2. Conclusion

a) When the transistor is off (not conducting) the full supply voltage appears over the transistor i.e. as it would with an open switch.

b) When the transistor is on (conducting) the voltage over the transistor is close to OV, i.e. as it would be with a closed switch and almost the full supply voltage appears over the lamp.

c) When the transistor is operating in the linear mode it is continuously conducting, i.e. Ib is flowing and thus Ic also flows. The signal that influences Ib must be limited to ensure that the transistor is not driven into the extremes as in a) and b), i.e. Vce approximately equals zero volts or Vce is equal to supply voltage.

It is obvious that Experiment 3 would not be used in a practical application, as the switch could have been used to switch the lamp directly. The experiment was done purely to show that a transistor can switch power to a load when a small base current is applied.

The transistor as a switch and the amplifier will be explained in detail in the second part of the course.

	Self test 3

Answer the following questions without referring to your notes:

1. Name the two modes in which a transistor normally operates.

____

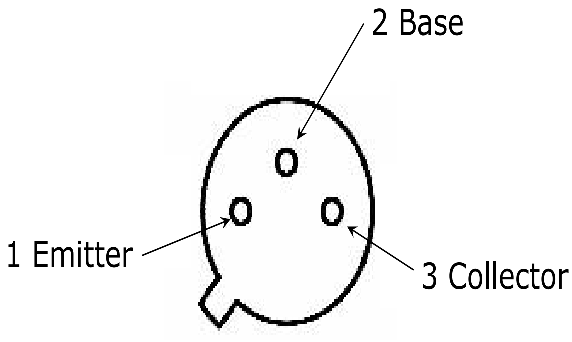
____

2. When a transistor is operating in the linear mode, it is used to amplify weak signals and as such is working as an ------------ whereas when it is used as an electronic switch, it is working in the -------------------- mode.

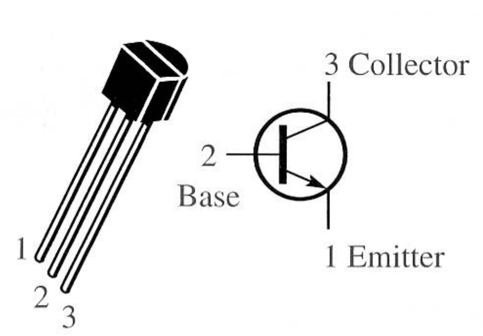
	• Ask your Facilitator to check your answers. • Find and correct your mistakes if your answers are not all correct.

	Ask for the criterion test when you feel ready.

Pin Configuration (Bottom View)
Emitter Closest to the tab.

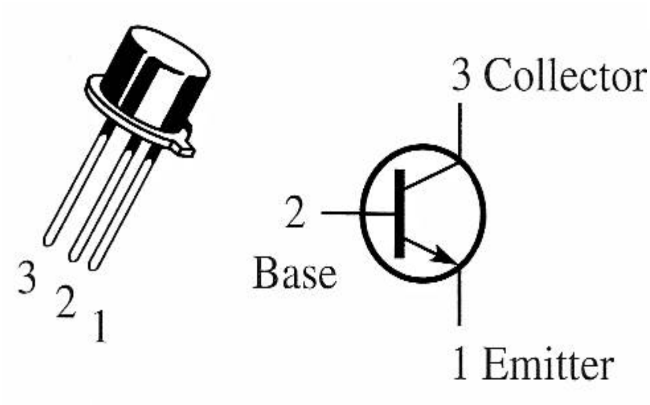


Transistor Pin Layout



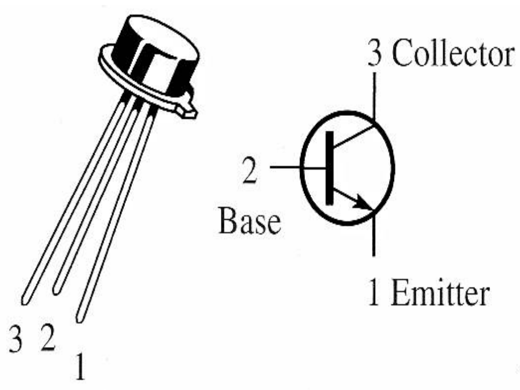
TO-92 or TO-226AA

Transistor Pin Layout



TO-39 or TO-205AD

Transistor Pin Layout



TO-18 or TO-206AA

Module 10
BE - 9 Identify transistor configurations

Purpose of module

This objective will enable the learner to identify transistor configurations in circuit diagrams.

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Identify the following transistor configurations in circuit diagrams:

   (a) Common base,

   (b) Common collector, and

   (c) Common emitter.

2. State the current and voltage gain of a common base, common emitter and common collector transistor circuit.

3. State the input and output impedance for a common base, common emitter and common collector transistor circuit.

4. State the phase relationship between input and output signals for common base, common emitter and common collector transistor circuits.

Procedures Relating To This Module

- Each configuration must be identified correctly.
- Where applicable, all answers must be in writing.
- All answers must be correct and in accordance with the module notes.

1. *Introduction*

The transistor may be connected in a circuit in any of three different arrangements (configurations). These are (1) the common base (2) the common emitter and (3) the common collector. The three different configurations have different characteristics, each of which will be explained by doing experiments.

2. *The common base configuration*

	Experiment 1

To do this experiment you need:

- An NPN transistor. (2N2222 or equivalent).
- A 100 kilohm potentiometer.
- Two DC supplies (1, 5 volt and 6 volt).
- Two ammeters (0-1mA).
- A circuit board.
- Connecting wires.

1. Procedure

    - Construct the circuit shown in Fig. 1.

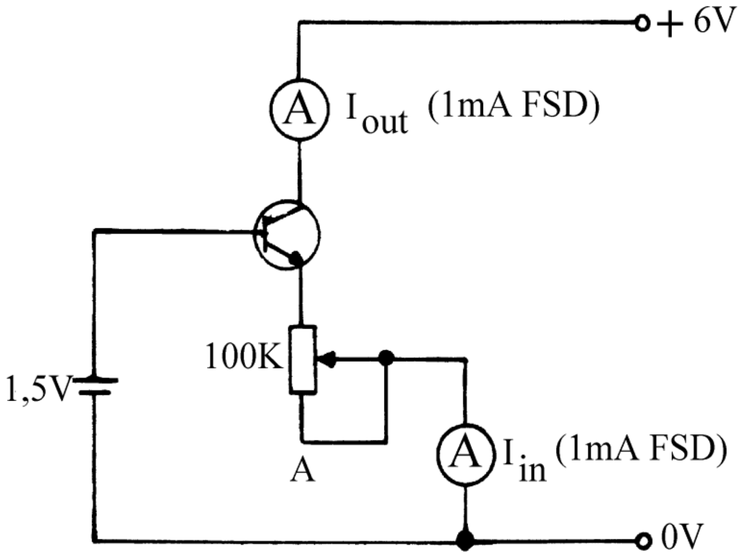


Figure 1

- Ensure that the slider of the potentiometer is in position A (i.e. all resistance in circuit) before the 1, 5 volt supply is connected.
- Slowly turn the potentiometer knob to obtain the Iin (current in) values shown in the table below.
- Read and record the Iout (current out) values which correspond with the Iin values given in Table 1.

TABLE 1

Iin	0,1	0,2	0,3	0,4	0,5	0,6
Iout						

The current gain of any circuit is given by the formula

Current gain $= \frac{Iout}{Iin}$

Calculate this gain for any two corresponding values in the table.

Current gain = ____________= ....times.

The gain will always be close to 1 and in practice it is said to have a current gain of 1.

This configuration is called a Common base configuration because the input signal is on the emitter and the output signal is on the collector.

It can also be seen from Table 1 that if Iin increases, Iout increases.

For further details of the common base configuration do the next experiment.

	Experiment 2

To do this experiment you need:

- Three resistors, 4K7, 1K5 and 150 ohm (all ½ watt).
- A 50K potentiometer.
- Two 50 microfarad 25 volt electrolytic capacitors.
- An NPN transistor. (2N2222 or equivalent).
- A 10 volt DC supply.
- A signal generator.
- An oscilloscope.
- A circuit board.
- Connecting wires.

1. Procedure

- Construct the circuit shown in Fig. 2.
- Connect the oscilloscope to the output of the circuit.
- Apply a 1kHz sine wave signal to the input and adjust the potentiometer P1 and the amplitude of the input signal to obtain the highest possible amplitude of an undistorted waveform on the oscilloscope.
- Use the oscilloscope to measure the input amplitude and the output amplitude and calculate the voltage gain.

Input amplitude = ---------------- volts

Output amplitude = ---------------- volts

Gain = $\frac{Vout}{Vin}$ = ----------------- times

- State whether it is a high voltage gain or a low voltage gain.__________________

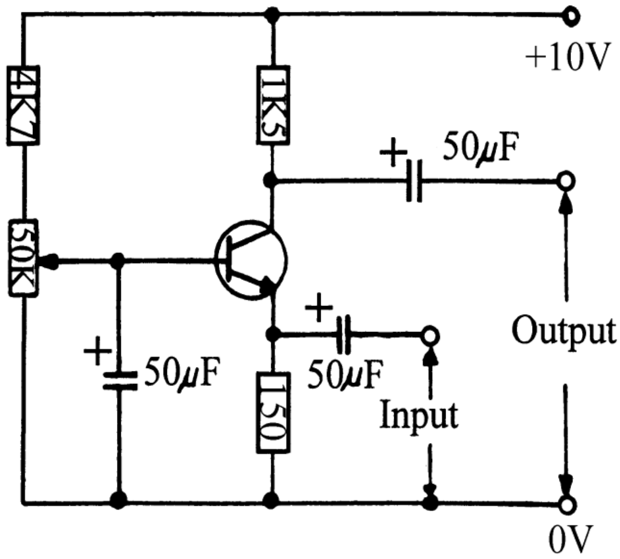


Figure 2

- Compare the input and output signals to establish their phase relationship (Fig. 3).

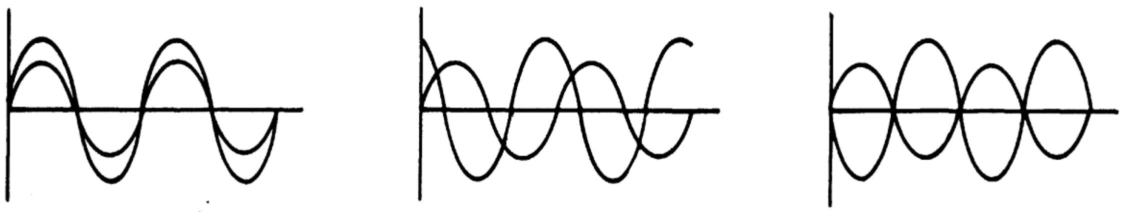


Figure 3

- State your observations _____________________________

The other two important features of the common base configuration are :

a) A low input impedance (Zin)

b) A high output impedance (Zout)

	Self test 1

Answer the questions below without referring to your notes.

1. How can a transistor in circuit be identified as a common base configuration?

   ____

2. What is the current gain of a common base circuit?

   ____

3. Is the voltage gain of a common base circuit high or low?

   ____

4. Are the input and output signals of a common base circuit in or out of phase?

   ____

5. What is the input and output impedance of a common base circuit?

   ____

	Ask your facilitator to check your work and to sign below if it is correct, then go on to the next section. If all your answers are not correct, repeat the experiments.

3. *The common emitter configuration*

	Experiment 3

To do this experiment you need:

- An NPN transistor. (2N2222 or equivalent).
- An 100 kilohm potentiometer.
- Two DC supplies (1, 5 volt and 6 volt).
- Two ammeters. (0-1mA and 0-200mA).
- A circuit board.
- Connecting wires.

1. Procedure

- Construct the circuit shown in Fig. 4.

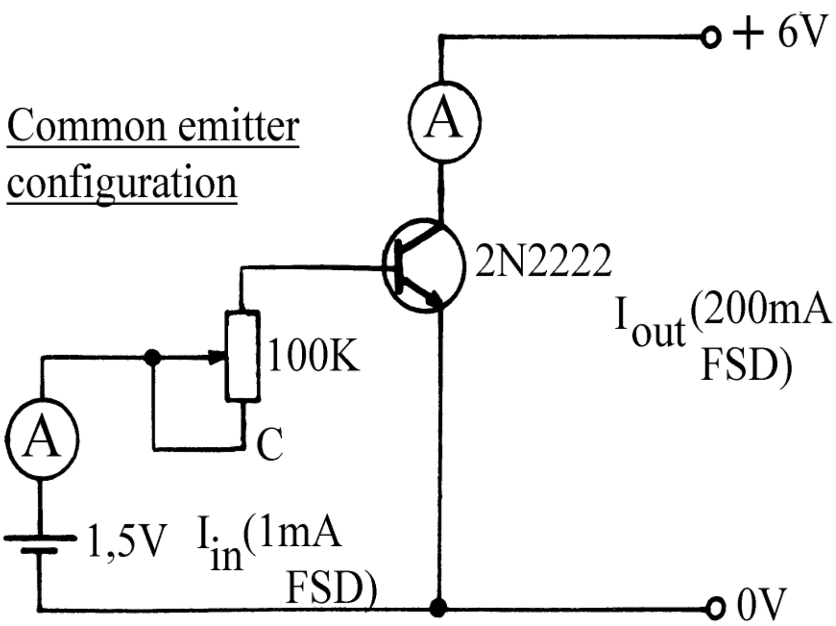


Figure 4

- Ensure that the slider of the potentiometer is in position A before the 1, 5 volt supply is connected.

- Adjust the potentiometer to the Iin values shown in Table 2 below and record the corresponding Iout values.

TABLE 2

Iin	0,05	0,1	0,15	0,2	0,25	mA
Iout						mA

- Calculate the current gain for all the corresponding values shown in the table.

$$Current\ gain = \frac{Iout}{Iin} = ...Times$$

This configuration has a high current gain.

Current gain is listed in transistor data books under the headings of hFE or hfe.

> This configuration is called a Common Emitter configuration because the input signal is on the base and the output signal is on the collector.

For further details of the common emitter configuration do the next experiment.

Experiment 4

To do this experiment you need:

- A 10 volt DC supply.
- An NPN transistor. (2N2222 or equivalent).
- Three resistors 4K7, 1K5 and 150 ohm. (All ½ watt).
- Two 50 microfarad 25 volt electrolytic capacitors.
- A 10 kilohm potentiometer.
- A signal generator.
- An oscilloscope.
- A circuit board.
- Connecting wires.

1. Procedure

    - Construct the circuit shown in Fig. 5 on the next page.

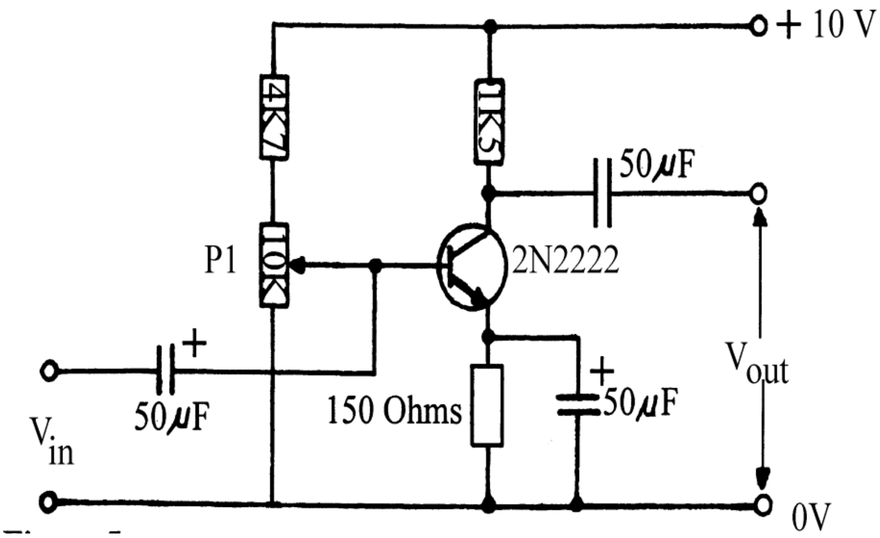


Figure 5

- Connect the oscilloscope to the circuit output.
- Apply a 1kHz sine wave signal to the input and adjust P1 and the amplitude of the input signal to obtain the highest possible amplitude of an undistorted waveform on the oscilloscope.
- Use the oscilloscope to measure the input amplitude and the output amplitude and calculate the voltage gain.

  Input amplitude = ____________ volts

  Output amplitude = ____________ volts

- Calculate the voltage gain

  Gain = $\frac{Vout}{Vin}$ = ____________times.

- State whether it is a high voltage gain or a low voltage gain___________________
- Compare the input and output signals to establish their phase relationship.
- State your observations ___________________

The other two important features of the common emitter configuration are :

a) Medium input impedance (Zin).

b) Medium output impedance (Zout).

Self test 2

Answer the questions below without referring to your notes.

1. How can a transistor in circuit be identified as a common emitter configuration?

   ____

2. Does the common emitter have a high or low current gain?

   ____

3. Does the common emitter have a high or low voltage gain?

   ____

4. Are the input and output signals of a common emitter in or out of phase?

   ____

5. What is the input and output impedance of a common emitter circuit?

   ____

If your answers are not all correct, do the experiments again, if they are all correct, do the next experiment to see what happens when a transistor is connected in a common collector configuration.

4. *The common collector configuration*

	Experiment 5

To do this experiment you need:

- Two DC voltage supplies (1, 5 volt and 4 volts).
- Two ammeters (0-1mA and 0-200mA).
- A 100 kilohm potentiometer.
- An NPN transistor. (2N2222 or equivalent).
- A circuit board.
- Connecting wires.

1. Procedure

- Construct the circuit shown in Fig. 6.
- Ensure that the slider of the potentiometer is in position A before the 1, 5 volt supply is connected.

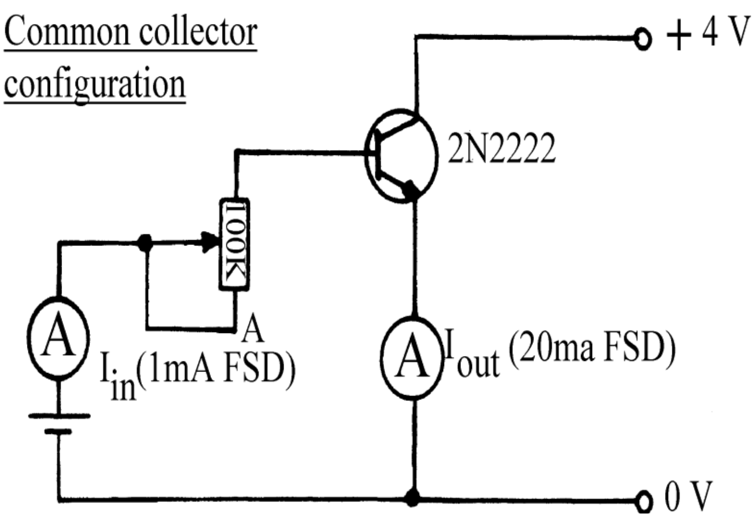


Figure 6

- Adjust the potentiometer to the Iin values shown in table 3 below and record the corresponding values.

TABLE 3

Iin	0,1	0,2	0,3	0,4	0,5	mA
Iout						mA

$$Current\ gain = \frac{Iout}{Iin} = ...Times$$

- Calculate the current gain for all the corresponding values shown in Table 3.This configuration also has a high current gain.

> This configuration is called a Common Collector configuration because the input signal is on the base and the output signal is on the emitter. This configuration is also known as an emitter follower.

For further study of the common collector configuration, do the next experiment.

	Experiment 6

To do this experiment you need:

- A 10 volt DC supply.
- An NPN transistor. (2N2222 or equivalent).
- Two resistors of 4K7, ½ watt each.
- A 50 microfarad 25 volt electrolytic capacitor.
- A 10 kilohm potentiometer.
- A signal generator.
- An oscilloscope.
- A circuit board.
- Connecting wires.

1. Procedure
    - Construct the circuit shown in Fig. 7.

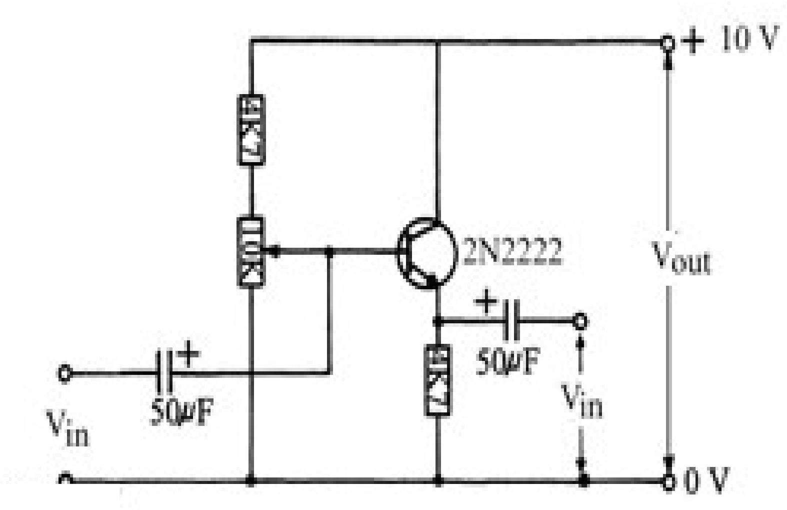


Figure 7

- Connect the oscilloscope to the output of the circuit.

- Apply a 1kHz sine wave signal to the input and adjust P1 and the amplitude of the input signal to obtain the highest possible amplitude of an undistorted waveform on the oscilloscope.

- Use the oscilloscope to compare the input and output voltage amplitude.

  Input amplitude = ________ volts

  Output amplitude = ________ volts

- Calculate the voltage gain.

  Gain $= \frac{Vout}{Vin}$ = ________times.

- State if the voltage gain is high or low _________________

  In practice it is said to have a voltage gain of 1.

- Compare the input and output signals to establish their phase relationship.

- State your observation ________________________

  The other important features of the common collector configuration are:

  a) High input impedance (Zin).

  b) Low output impedance (Zout).

	Self test 3

Answer the following questions without referring to your notes:

1. How can a transistor in circuit be identified as a common collector configuration?

   ____

2. Does the common collector have a high or low voltage gain?

   ____

3. Does the common collector have a high or low current gain?

   ____

4. Are the input and output signals in or out of phase?

   ____

5. What is the input and output impedance of a common collector circuit?

   ____

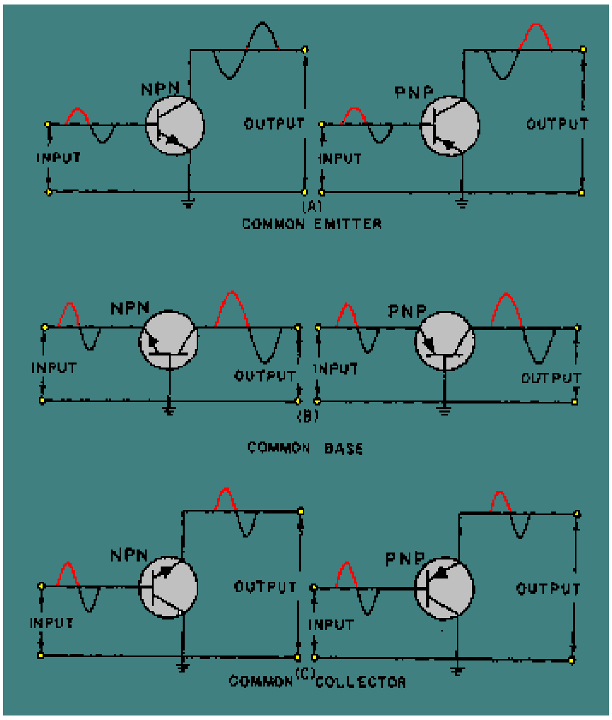
6. Summarize the features of the different configurations by completing Table 4 below without using your notes.

	(a)	(b)	(c)
Voltage gain			
Current gain			
Voltage phase (relationship input and output)			
Zin			
Zout			
Input terminal			
Output terminal			

	(a) = Common base, (b) = Common emitter, (c) = Common collector
	Ask for the criterion test when you feel ready.

5. *Transistor Configurations*

A transistor may be connected in any one of three basic configurations (fig. 2-16) : common emitter (CE), common base (CB), and common collector (CC). The term common is used to denote the element that is common to both input and output circuits. Because the common element is often grounded, these configurations are frequently referred to as grounded emitter, grounded base, and grounded collector.



Transistor configurations

Each configuration, as you will see later, has particular characteristics that make it suitable for specific applications. An easy way to identify a specific transistor configuration is to follow three simple steps :

- Identify the element (emitter, base or collector) to which the input signal is applied.
- Identify the element (emitter, base or collector) from which the output signal is taken.
- The remaining element is the common element, and gives the configuration its name.

Therefore, by applying these three simple steps to the circuit in figure 2-12, we can conclude that this circuit is more than just a basic transistor amplifier. It is a common-emitter amplifier.

NOTES

Module 11
BE - 10 Test regulated power suppliers

Purpose of module

This objective will enable the learner to test regulated power suppliers

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Test the output voltages for the different loads supplied.
2. Draw a circuit diagram for a regulated power supply.
3. Name the four main parts into which a regulated power supply may be divided.
4. State the output voltage of a regulated power supply if the zener voltage is known.
5. State the transistor configuration used in a regulated power supply and its important properties.
6. Draw a comparison between unregulated and regulated power supply output voltage ripple.

Procedures Relating To This Module

- Where applicable, all answers must be in writing.
- All answers must be correct and in accordance with the module notes.

1. *Introduction*

The use of the zener diode as a parallel voltage regulator was described in module BE-6. However, it does have limitations when it is loaded. The simple zener diode regulator is improved with the addition of a transistor series regulator.

The operation of a regulated power supply will be better understood by carrying out the following experiment.

	Experiment 1

To do this experiment you need:

- A 1 amp 400 volt bridge rectifier.
- A 1000 microfarad 25 volt electrolytic capacitor.
- A 10 volt 1 watt zener diode.
- An NPN transistor (2N5496) or equivalent.
- A 270 ohm ½ watt series resistor.
- Two ammeters (0-50mA and 0-500mA).
- A voltmeter (0-15 volts).
- 330 ohm ½ watt, 220 ohm ½ watt, 150 ohm 1 watt, 100 ohm 1 watt, 47 ohm 5 watt and 33ohm 5 watt load resistors.
- A 12 volt AC supply.
- A circuit board.
- Connecting wires.
- An oscilloscope.

1. Procedure

- Use a 12V transformer and not a Variac.
- Test the circuit shown in Fig. 1 on the next page.

	Note that it is divided into 4 main parts, namely: a) The rectification and smoothing circuit b) The parallel regulator c) The series regulator d) The load.

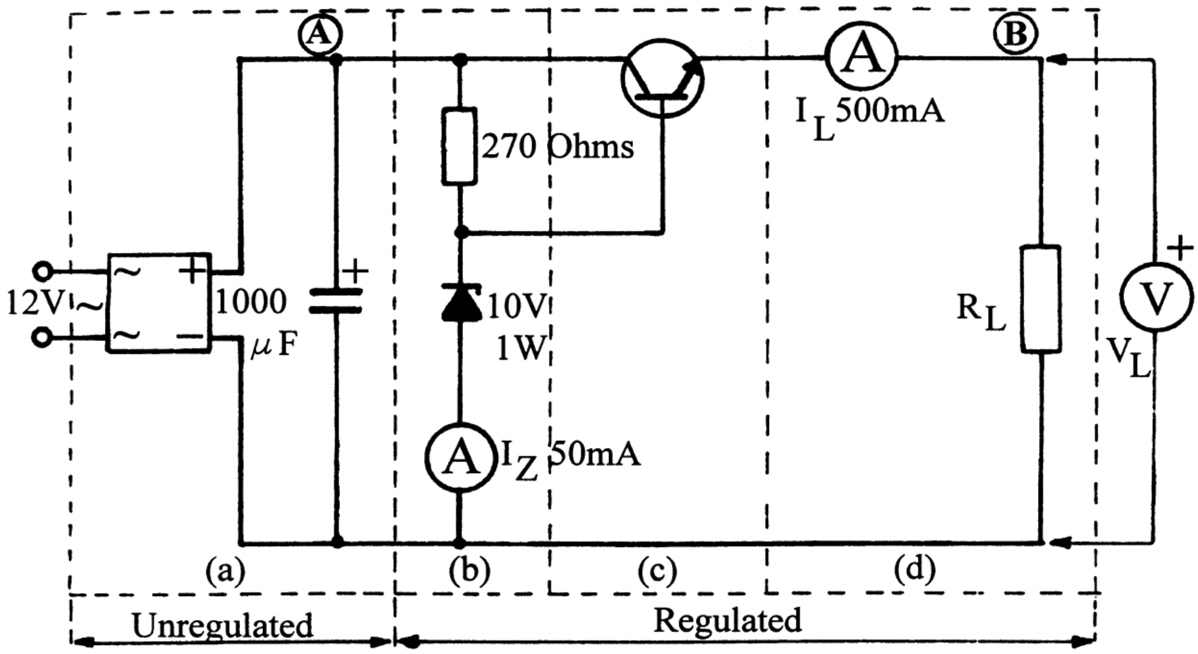


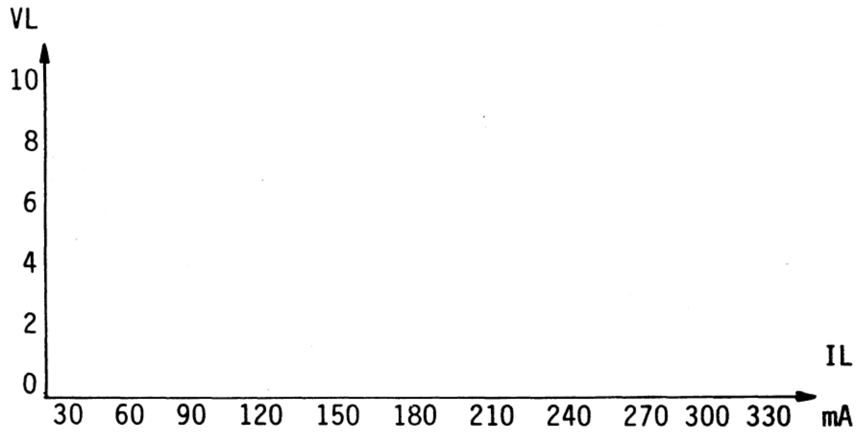
Figure 1 – Regular Power supp.y

- Complete the load values in Table 1 below.

TABLE 1

RL	330	220	150	100	47	33	ohm
VL							volts
IL							mA
IZ							mA
VZ							volts

- Plot VL as a function of IL= F (IL).



It is obvious from the curve above that as long as VZ stays constant Vout will stay constant.

It is important to note that Vout of the rated voltage of the zener diode will determine the output voltage of the circuit.

It will be noted that the output voltage is not 10 volts but 9, 4 volts. This is due to a voltage drop of 0, 6 volts over the transistor emitter/base junction. See Fig. 2 below.

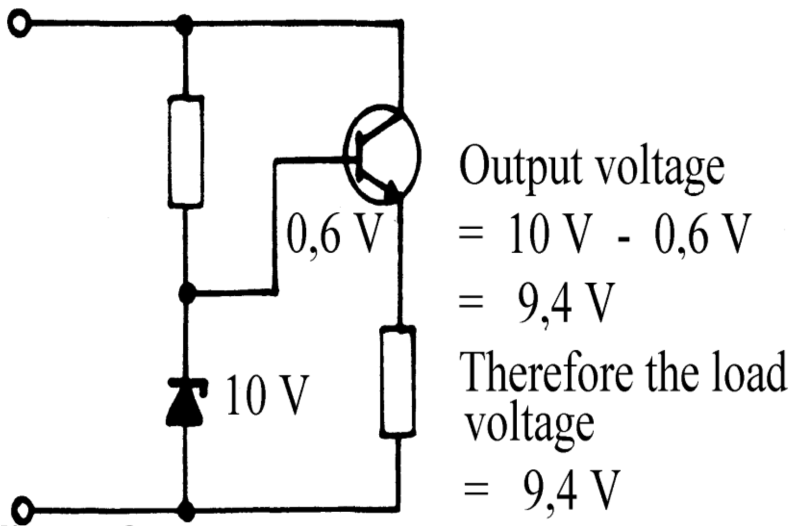


Figure 2

- Remove the 33Ω load resistor and replace it with the 150Ω resistor.
- Connect the oscilloscope across the smoothing capacitor at point A and then across the load resistor at point B and measure both voltages with the oscilloscope. Compare the amount of ripple present between these two points.
- Draw these waveforms and note down their respective voltages.

The input is on the transistor base and the output on the transistor emitter. Therefore, the transistor is said to be connected in a common collector configuration with the important properties of:

(i) A high current gain (hFE),

(ii) A low output impedance (Z), and

(iii) A voltage gain of 1.

2. Conclusion

This regulated DC power supply makes use of a transistor series regulator as an addition to the zener parallel regulator dealt with in BE-6 and is a big improvement on the performance of the zener regulator. This power supply can deliver much higher load currents at a fixed regulated voltage than the zener regulator circuit. As shown in the experiment, the output voltage is also ripple free.

	Self test

Answer the questions below without referring to your notes.

1. Draw a circuit diagram for a regulated power supply on a separate sheet of paper and indicate and name the four main parts.

   ____

2. Draw the waveforms as they would appear on an oscilloscope if the probe is first connected across the smoothing capacitor and then across the load. Use the same sheet of paper as used for

   Question 1.

   ____

3 When will Vout stay constant?

   ____

4. Why is the output of the regulated power supply equal to 9, 4 volts?

   ____

5. What transistor configuration was used for the regulated power supply?

   ____

6. What are the important properties of a common collector connection for this circuit?

   ____

	Ask for the criterion test when you feel ready.

Module 12
BE - 11 Test thyristors

Purpose of module

This objective will enable the learner to test thyristors

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Connect the circuit of a thyristor full wave bridge to supply an AC load.
2. Test to see when the silicon controlled rectifier (SCR) is on or off by controlling it with the gate switch.
3. Explain what voltage appears over the SCR when it is both conducting and not conducting.
4. Draw the circuit symbol for an SCR.
5. Draw an approximate equivalent circuit for an SCR using two transistors.
6. State whether an SCR is uni directional or bi directional.
7. State the conditions for an SCR to start conducting.
8. Explain how an SCR may be turned off.
9. Explain the effect when the gate signal is removed from a conducting SCR when:
   (a) The supply is DC and
   (b) When the supply is pulsating DC.
10. State the output to a load when a single SCR is connected in series with the load in an AC circuit.
11. Draw a simple circuit to switch AC to a load using two SCR's.
12. Draw a circuit where a DC load fed from an AC supply has to be switched by using a diode bridge and a SCR.
13. Use the same circuit in point 12 and connect the load in circuit so that the load is switched to AC.

Procedures Relating To This Module

- The thyristor full wave bridge circuit must supply an AC load.
- The test must indicate whether the SCR is on or off.
- Where applicable all answers must be in writing.
- All answers must be correct and in accordance with the module notes.

1. Introduction

The thyristor described is a silicon controlled rectifier (SCR). Other types of thyristors are available, but the most useful thyristor is the SCR. It is similar to a diode which is controlled by means of an extra terminal connected to it. The SCR is really a solid state switch which, depending on the type, can operate at potentials up to several thousand volts and can handle currents up to hundreds of amps.

The device involves a four-layer construction (pnpn) and is illustrated in Fig. 1, where its symbol is also shown.

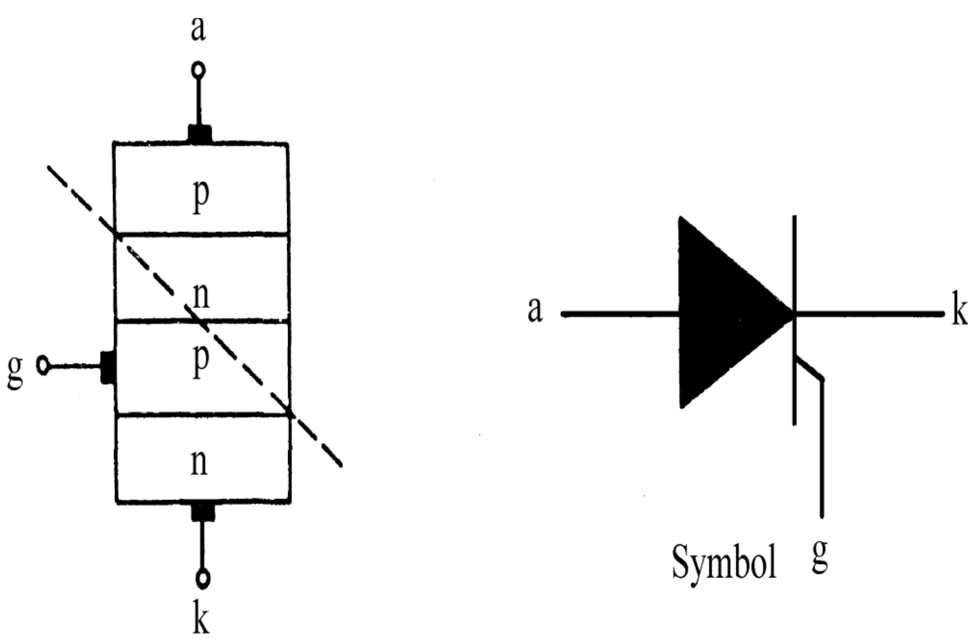


Figure 1

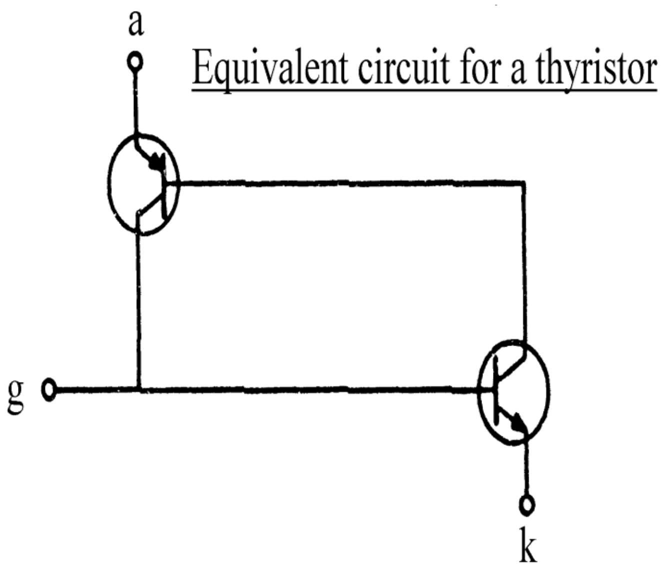


Figure 2

If the four layers were cut along the dotted line the two parts would resemble pnp and NPN transistors. Fig. 2 shows an approximate equivalent circuit based on this idea.

The SCR will block current flow up to the limit imposed by the voltage ratings of the device if the gate is left unconnected. This blocking applies to current which is attempting to pass either way between the anode and cathode. However, a short duration current pulse into the gate will cause the device to turn on and current will start to flow from anode to cathode, i.e. when conventional current flow is assumed.

As long as there is sufficient current between anode and cathode, the thyristor will stay on, even after the gate current pulse has been removed. If this current is switched off momentarily at some other point, or drops to a sufficiently low level, the conduction path within the thyristor will block or switch off until a pulse is provided on the gate that will turn the thyristor on once again.

> Once the gate current pulse has activated the conduction path, the gate loses control and this makes the thyristor gate quite different to the usual base control in a transistor.

To study the behaviour of the two transistor equivalent circuit of an SCR, do the following experiments.

	Experiment 1

To do this experiment you need:

- A 10 volt DC supply.
- A pnp and a NPN transistor.
- A light emitting diode (LED).
- A 1N4007 diode.
- Two SPST Switches.
- A 5K potentiometer.
- ½ watt resistors. One 10 ohm, one 390 ohm, two 1K, one 4K7 and one 10K.
- A circuit board
- Connecting wires.

1. Procedure

- Test the circuit shown in Fig. 3 on the next page.

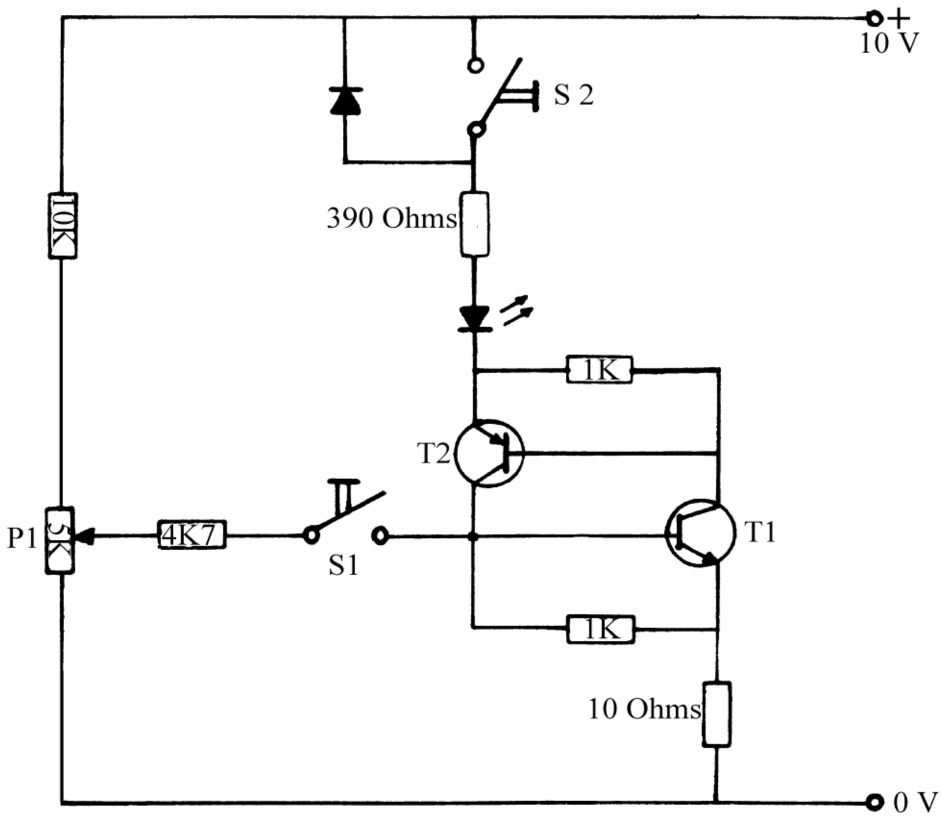


Figure 3

- Switch on the supply with the P1 slider adjusted to OV.
- Close S1 and S2 and turn up P1 until the LED lights up.
- Open S1. Does the LED turn off? ______________

2. Conclusion

2.1 P1 provided base current (Ib) for T1 via S1.

2.2 T1 when on, provides (Ib) for T2.

2.3 When T2 is on, it latches T1 on, because it in turn provides Ib for T1.

2.4 The opening of S1 now, will not affect the circuit. Therefore once the circuit is triggered, S1 is ineffective.

With S1 open, open S2. Does the LED turn off?
____

Complete the statements below where necessary:

- Once the latching action is complete, the controlling terminal ______________ cannot be used to turn the circuit ______________
- The only way to turn it off is to ______________
- Current flow through the transistor pair is uni-directional/bi-directional and the behaviour of the circuit when on, is similar to that of a ______________

- Fill in the p and n sections of the constructions in Fig. 4.

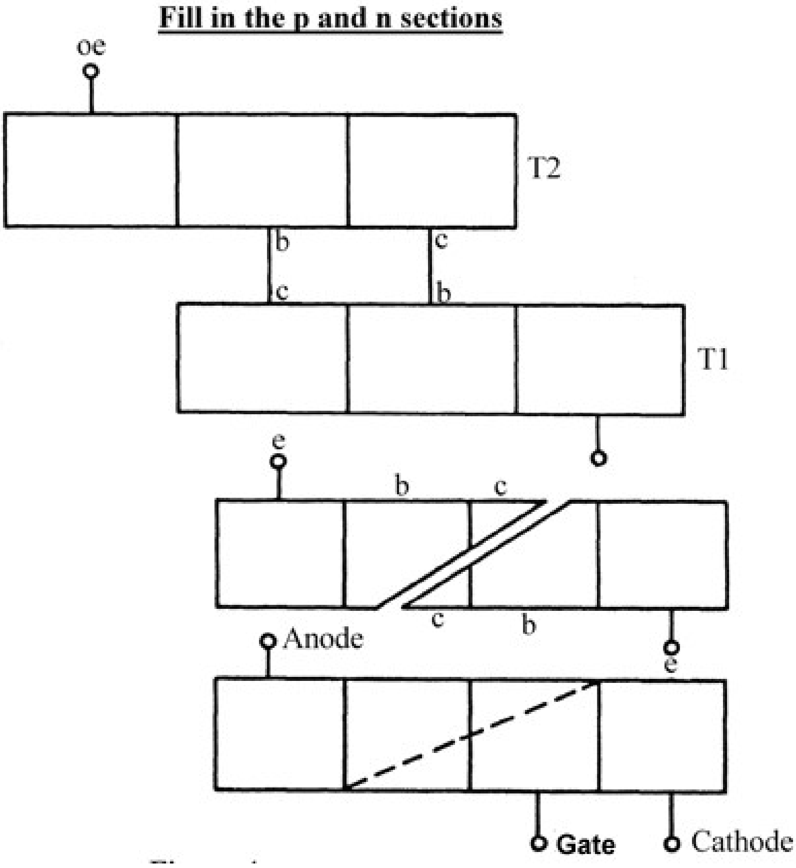


Figure 4

	Experiment 2

To do this experiment you need:

- A 10 volt DC supply.
- A 1 amp 100 volt SCR.
- Two ammeters, 10mA (FSD) and 30mA (FSD)
- A multimeter.
- Two potentiometers, 5K and 10K.
- Three ½ watt resistors: 100 ohm, 470 ohm and 1K.
- Two SPST switches.
- A circuit board.
- Connecting wires.

1. Procedure

    - Test the circuit shown in Fig. 5, where the SCR replaces two transistors.

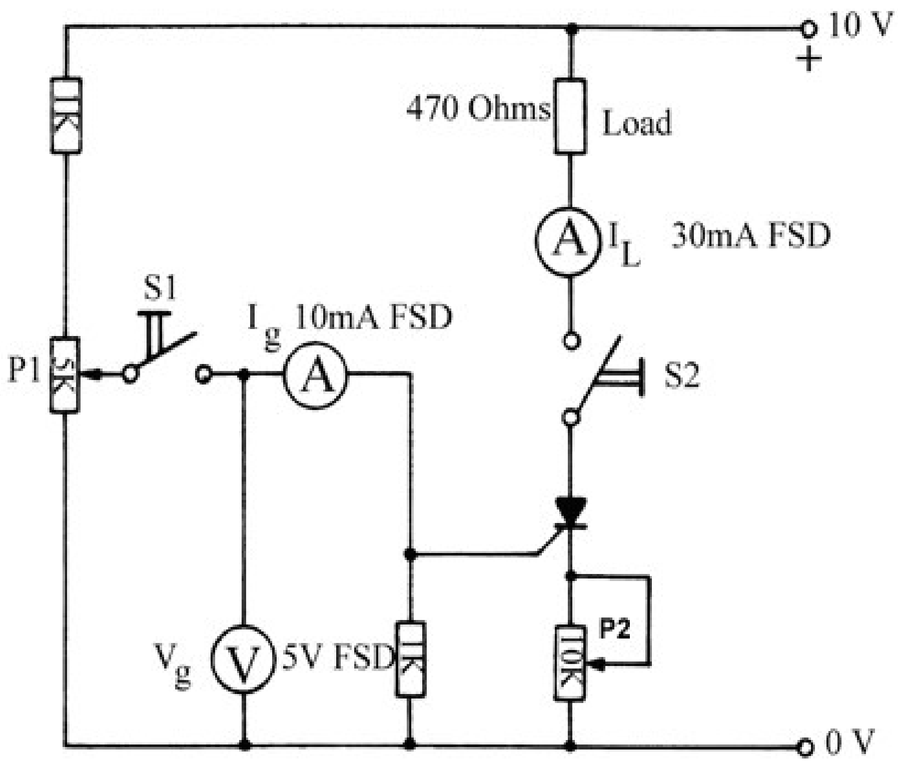


Figure 5

- Switch on the supply with S2 closed, S1 open, P1 slider to OV and P2 slider to OV.
- Make the following measurements:
  a) Load voltage (VL) = __________ volts.
  b) Voltage over SCR (VA-K) = __________ volts.
  c) Loadcurrent (IL) = __________mA.

How can a voltmeter be connected to establish when the SCR turns on? _______________

- With the voltmeter connected as described in your answer above, switch on S1 and turn P2 up until the SCR turns on.

	Note the following values at the SCR firing point. Ig = ------------mA, Vg = -------------- volts

From the measured values above, it will be seen that the gate is activated by the current and not the voltage.

Note the following values after the SCR has turned on:

VL = ____________ volts

VA-K = ____________ volts

IL = ____________ mA.

- Calculate the load current of the circuit and then explain the value of IL.

- Open S1 and note whether the SCR turns off.

- Does the SCR turn off when S2 is momentarily opened?

- Re-trigger the SCR. Open S1 and ensure that the SCRis still conducting. Slowly start increasing the resistance of P1. At a certain minimum value of anode - cathode current or IL, the SCR will suddenly cease to conduct. This minimum amount of current that is needed to keep the SCR in conduction is known as the holding current (Ih).

  Ih = ------------------ mA

  What happens when the load is such, that the load current is below the holding current of the SCR and receives a trigger pulse?

2. Conclusions

2.1 To turn on a SCR the anode must be _________ with respect to the _________while the gate receives a pulse which is _________with respect to the cathode.

2.2 When an SCR is on, the _________cannot be used to turn it off.

2.3 When the SCR is conducting, it behaves like a _________because the current flow through it is _________

2.4 To turn an SCR off, the _________must be reduced to a value below the __________________

	Experiment 3

To do this experiment you need:

- A 12 volt AC supply.
- A 1, 5 volt DC supply.
- A 1 amp 100 volt SCR.
- A 12 volt filament lamp.
- A SPST switch.
- A 10 ohm ½ watt resistor.
- An oscilloscope.
- A circuit board.
- Connecting wires.

1. Procedure

- Test the circuit shown in Fig. 6 below, to study various applications of SCR's.

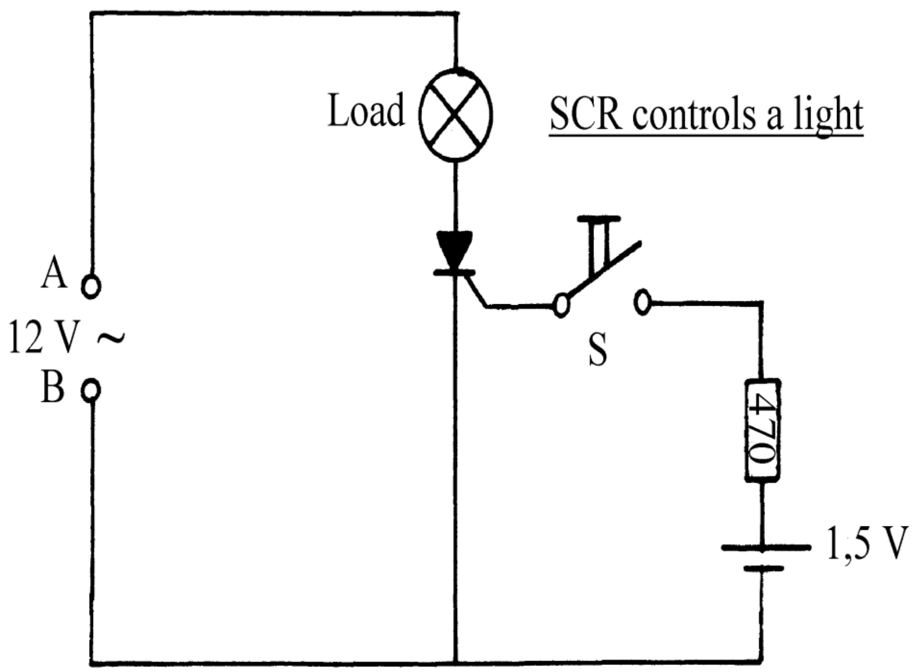


Figure 6

- With switch S open, measure the following wave- forms with the oscilloscope and draw the wave- forms on the axis of Fig. 7.

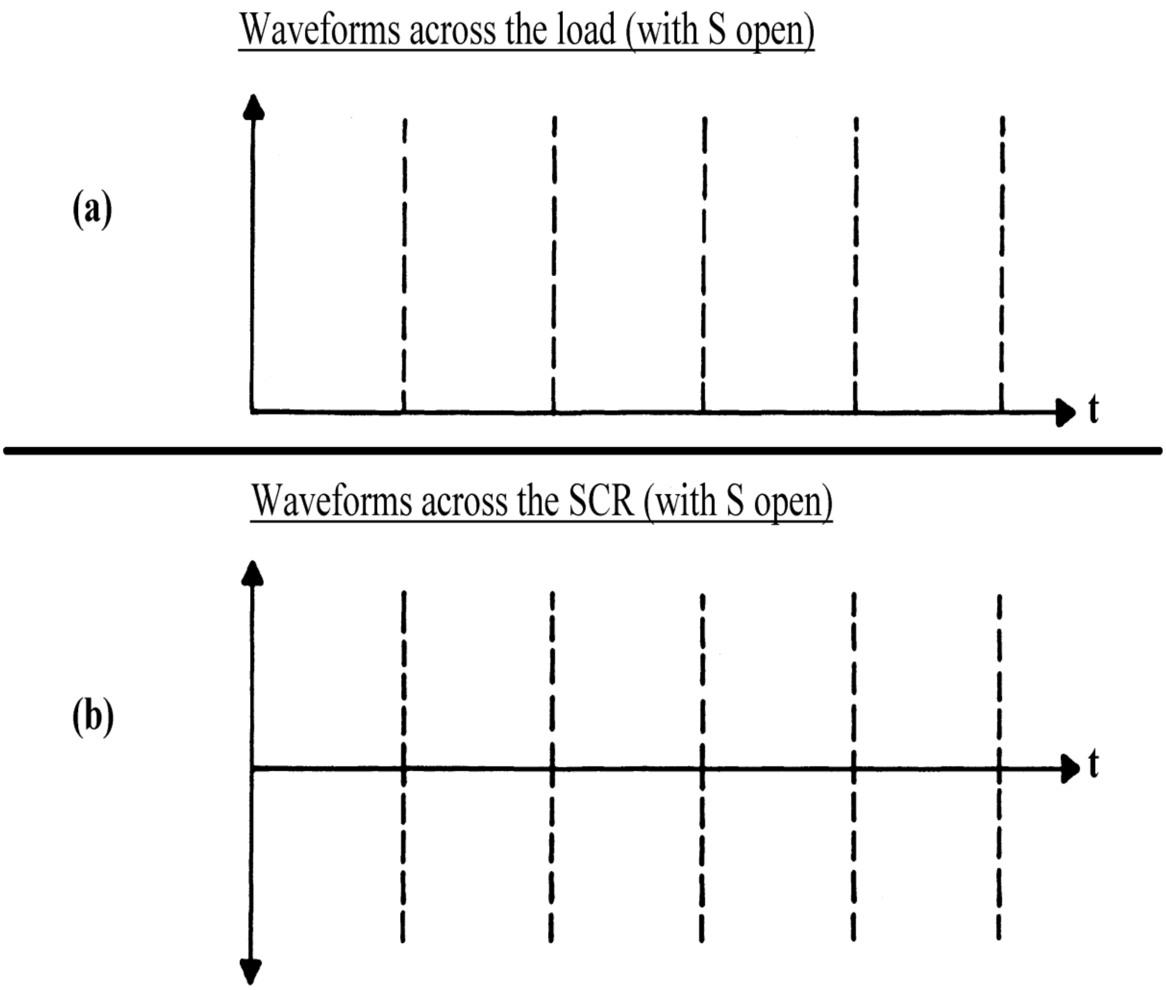


Figure 7

During the period, when point B of the AC supply is more positive than point A and the SCR is receiving gate power, it does not turn on because

____

- Open switch S and state what happens.

  ____

  ____

It was seen in point 4 above, that the SCR stopped conducting when switch S was opened. This in contrast to the behaviour of the SCR in Experiment 2 in that it remained on even when the gate current was removed by opening S1. The reason for the difference in behaviour of the SCR's in these two experiments can be explained as follows:

Due to the fact that a DC power supply was used in Experiment 2, the anode-cathode Current (i.e. load current) took on a fixed value of approximately 20 mA. After switch S1 was opened the SCR continued to conduct because the load current exceeded the holding current (Ih) value of the SCR.

In Experiment 3, an AC power supply was used which means that the nature of the current flow through the SCR changed direction continuously. This changing current becomes zero ampere at the end of each half cycle when the mains supply passes through zero. If the switch S is opened during the time the SCR is conducting, the SCR will not turn off immediately, but turns off with the completion of the half cycle, and remains off.

In experiment 3, the SCR limited the power to half of that which is available. This may be overcome by connecting the SCR in a full wave bridge circuit, which is studied in Experiment.

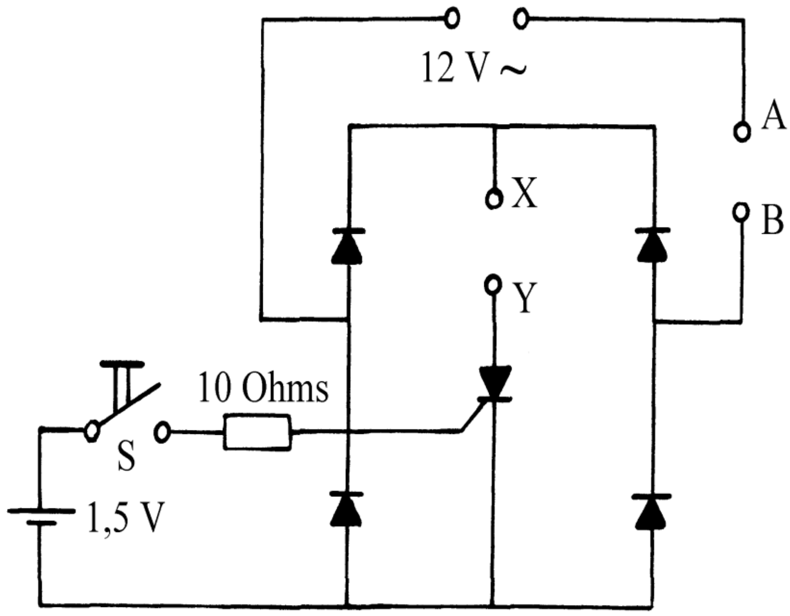
Experiment 4

To do this experiment you need:

- A 12 volt AC supply.
- A 1, 5 volt DC supply.
- A 12 volt filament lamp.
- A 1 amp 100 volt SCR.
- Four 1N4007 diodes.
- A SPST switch.
- A 10 ohm ½ watt resistor.
- An oscilloscope.
- A circuit board.
- Connecting wires.

1. Procedure

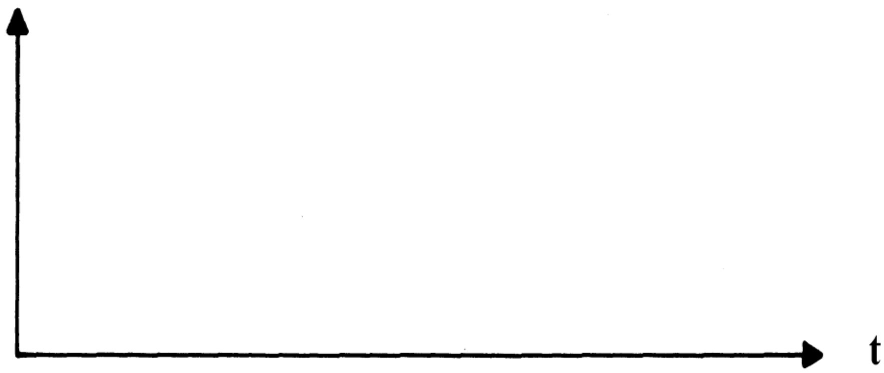
- Test the circuit shown in Fig. 9.



SCR and bridge used as a switch

Figure 9

- Before switching on the supply, connect the lamp between points X and Y and link points A and B.
- Switch on the supply and close switch S.
- With the oscilloscope measure and draw the wave- form across the load on the axis of Fig. 10 illustrated below.



Wave form across the load with A and B bridged

Figure 10

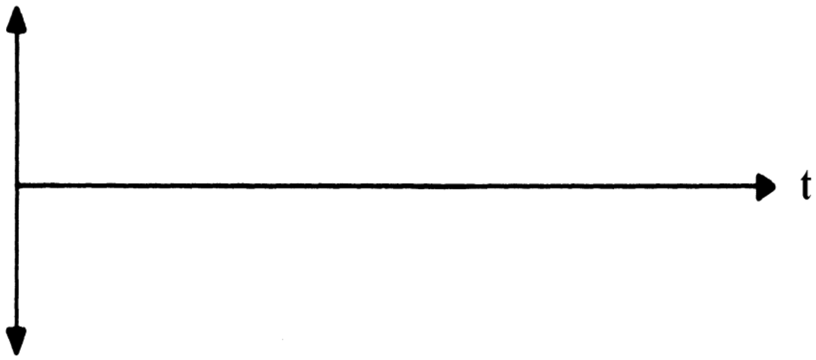
Is this waveform AC or DC? ____

The SCR switches both half cycles to the load, due to the fullwave rectifying action of the ____

The SCR will ______________________________ when S is opened, for the same reason as explained in points 5 and 6 of Experiment 3.

This circuit may be used where a heavy DC load, fed by AC has to be switched.

- Switch off the AC supply and connect the load between points A and B, while X and Y are linked.
- Switch on the supply and close switch S.
- With the oscilloscope, again measure and draw the waveform across the load on the axis of Fig. 11.



Wave form across the load with X and Y bridged

Figure 11

Is the waveform AC or DC? ______________________________

From the two waveforms, it can be seen that the output waveform depends on whether the load is connected in the AC or DC side of the bridge rectifier circuit.
It will also be seen on the AC waveform, that there is "dead time" between the half cycles. This is because the voltage drop across two diodes and the SCR must be overcome, before the circuit goes into conduction.

Why is the diode bridge necessary in this case?

____

- Show the direction of current flow through the load for each half cycle in different colours, on the circuit diagram of this experiment.

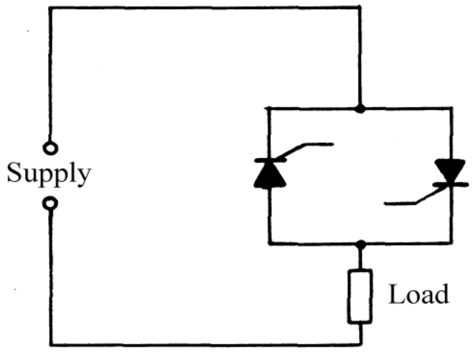
2. Conclusions

2.1 When a single thyristor is connected in series with a resistive load in an AC circuit, the load waveform is:

____

2.2 2In the circuit of 1 above, the thyristor turns off if the gate signal is removed because the current

____



Two SCR'S used as a switch

Figure 12

2.3 A way of switching AC to a load, is to use one SCR and a diode bridge as described in the second circuit of experiment 4. An alternate way is to use two thyristors as shown in Fig. 12.

	Take your results to your Facilitator to be checked.

	Ask for the criterion test when you feel ready.

Module 13
BE - 12 Test thyristor phase control

Purpose of module

This objective will enable the learner to test thyristor phase control

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

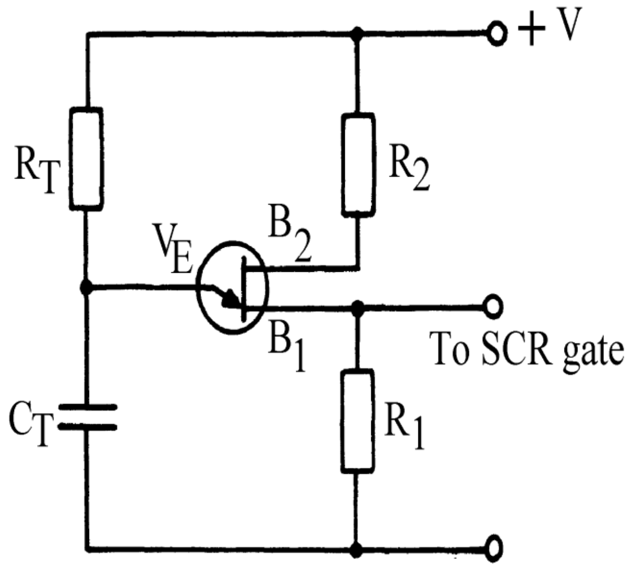
1. Tune a potentiometer in a thyristor phase control circuit approximately half way between the two ends of its control range.
2. Test and measure the waveform using an oscilloscope over the load.
3. Draw the waveform indicating α and β.
4. Draw the circuit symbol for a unijunction transistor.
5. State another name for a unijunction transistor.
6. Give the abbreviation for a unijunction transistor.
7. State which of the two bases of a unijunction transistor generates pulses to trigger an SCR.
8. State which terminal of a unijunction transistor generates a sawtooth waveform.
9. Explain how the period of the sawtooth waveform of a unijunction transistor may be changed.
10. Name three applications where phase control techniques may be used.
11. Draw fullwave DC waveform with α at any stipulated angle and indicate α and β on the waveform.
12. State in how many milli seconds (ms) a half cycle is completed when the frequency is given.
13. State whether the supply to the trigger pulse unit of a phase controlled thyristor circuit is smoothed or unsmoothed.

Procedures Relating To This Module

- The potentiometer must be tuned correctly.
- The waveform must be tested, measured and drawn as explained in the module notes.
- Where applicable all answers must be in writing.
- All answers must be correct and in accordance with the module notes.

1. Introduction

Many techniques are used for triggering an SCR such as DC gate current in the previous experiments. Only short duration pulses were needed to switch on the SCR. Because the unijunction transistor (UJT) can generate the required pulses, it is frequently used as the active component in a trigger source.



UJT connected in a relaxation oscilator

Figure 1

2. *The unijunction transistor (ujt)*

The circuit symbol for the UJT is shown in Fig. 2.

	This transistor has two bases, and is also referred to as a double base diode.

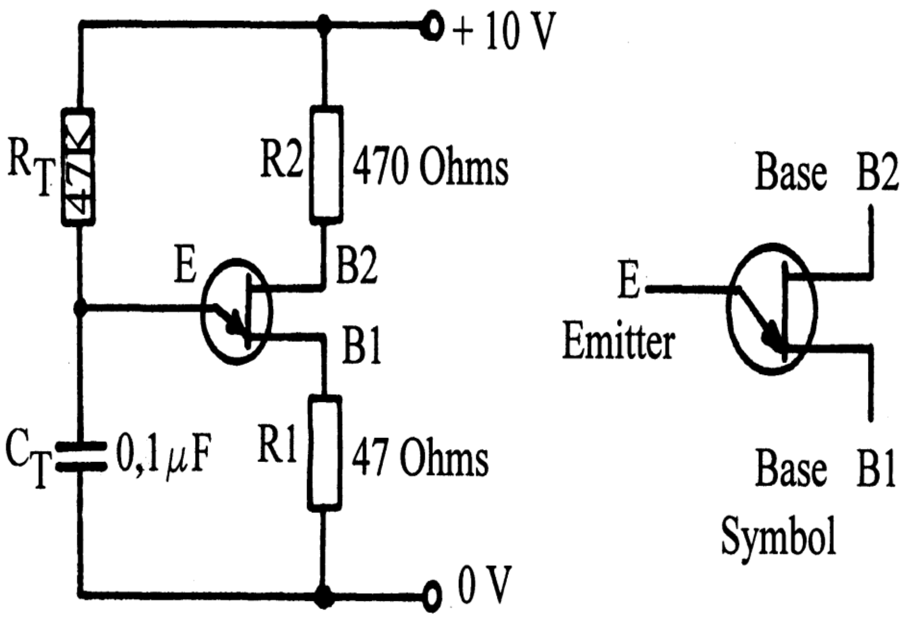


Figure 2

3. *The ujt connected as a relaxation oscillator*

The UJT connected as a relaxation oscillator, as shown in Fig. 1, generates a voltage waveform at B1. This may be applied as a triggering pulse to an SCR gate, to turn on the SCR.

The circuit operates as follows:

When the supply is switched on, the capacitor (CT) starts charging through the resistor (RT) and consequently the voltage across the capacitor rises over a period of time. Since the emitter of the uni-junction transistor is connected to the capacitor, its voltage (VE) will also rise. When the voltage across the capacitor reaches the threshold voltage of the UJT, it will bring the UJT into conduction. The charged capacitor "detects" a low resistance path via the emitter and base to the zero volt line and discharged rapidly, generating the short duration pulse mentioned in the introduction.

During the discharge, the voltage across the capacitor will decrease to approximately 2V. The UJT turns off, allowing the capacitor to charge up again to repeat the cycle.

The waveforms in Fig. 3 **on the next page** illustrates the **sawtooth** voltage VE, generated by the charging and discharging of capacitor CT and the output pulse VB1 developed across R1. VB1 is the pulse which will be applied to the gate of an SCR to trigger the SCR.

The period and thus the frequency of the relaxation oscillator depends on the time constant CT X RT.

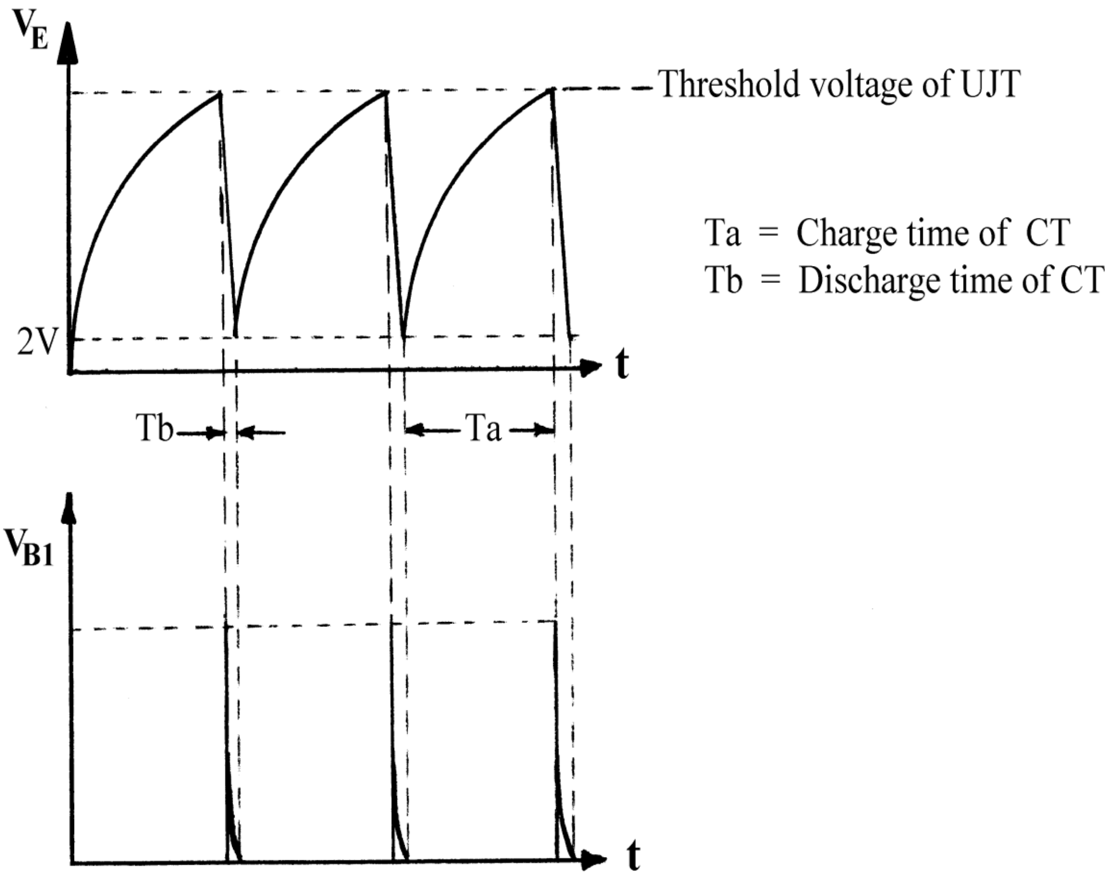


Figure 3

The operation and waveforms may be seen by doing the following experiment.

	Experiment 1

To carry out this experiment you need:

- A 10 volt DC supply.
- A unijunction transistor.
- Four ½ watt resistors: One 47K, one 470 ohm, one 10K and one 47 ohm.
- One 0, 1 microfarad capacitor.
- An oscilloscope.
- A circuit board.
- Connecting wires.

1. Procedure

- Test the circuit shown in Fig. 2 below.

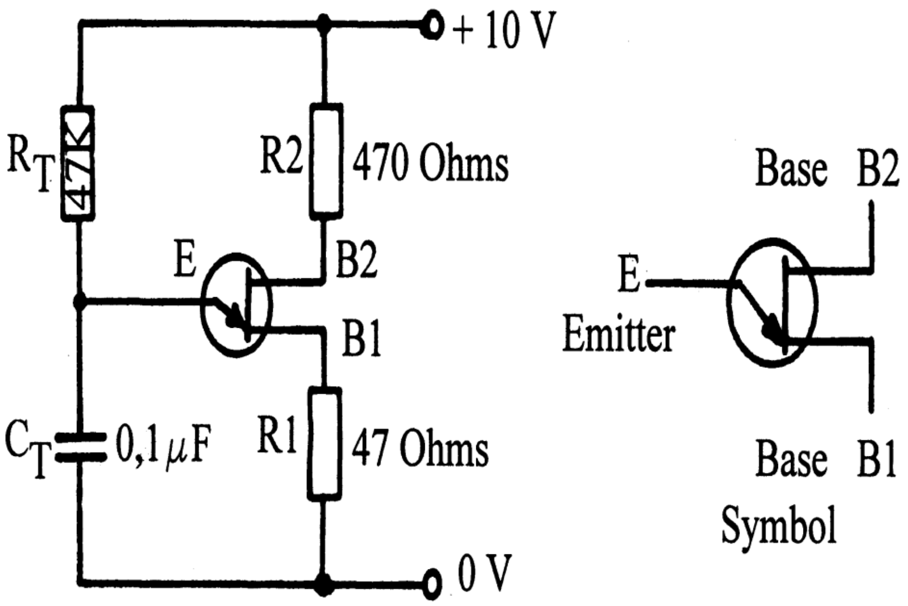
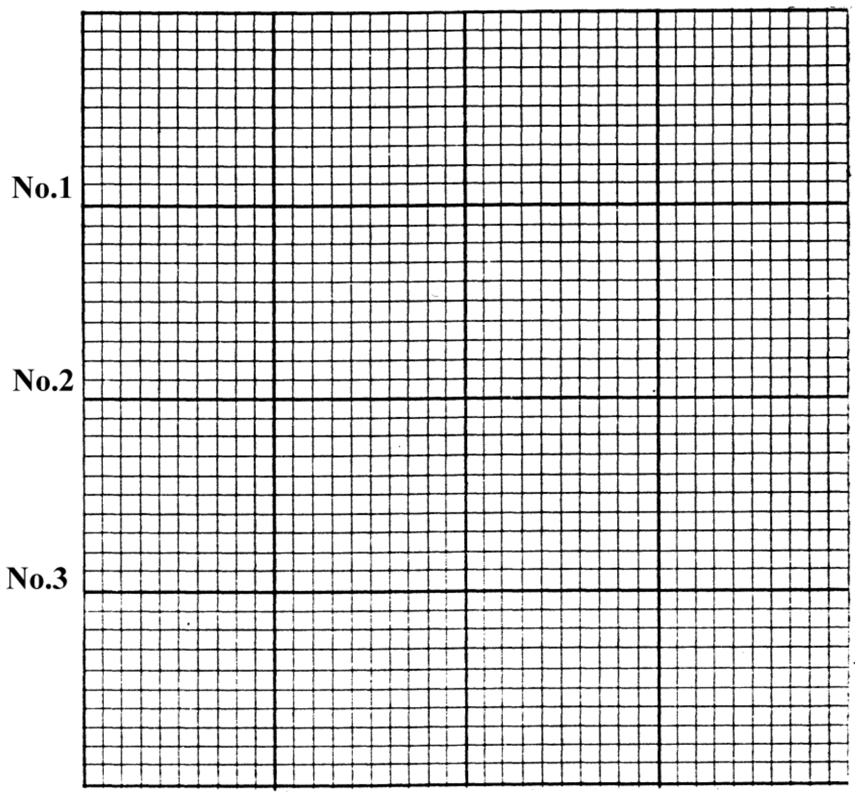


Figure 2

- Switch on the supply.
- Connect the oscilloscope to point E and draw the waveform on axis No. 1 on the graph paper below paying special attention to period and amplitude.



- Connect the oscilloscope to point B1 and draw the waveform on axis No. 2 of the graph paper.
- Connect the oscilloscope to point B2 and draw the waveform on axis No. 3 of the graph paper on the previous page.
- Change resistor RT to 10K. What change is there in the output waveform at B1?

____

____

2. Conclusion

From experiment 1 it can be seen that the wave at B1 can be changed by varying the value of RT. The voltage at B1 can be used to trigger pulses to an SCR gate to turn it on.

Self test 1

Answer the questions below without referring to your notes.

1. Draw the circuit symbol for a unijunction transistor.

2. What is a UJT also known as?

____

____

3. Which base generates trigger pulses to trigger a SCR?

____

4. Which terminal generates a sawtooth waveform?

____

5. How may the period of the sawtooth waveform be changed?

____

____

6. Draw the circuit diagram of a relaxation oscillator that uses a UJT as its active component. Clearly indicate the components that will change the period between the trigger pulses.

If all your answers are not correct repeat this section.

4. *Phase controlled triggering*

The main use of the SCR today is in the field of **power control**. The main advantage is its high efficiency resulting from its low power dissipation. Examples of applications of thyristor phase control are:

- Light dimming;
- Control of power supplied to heating elements; and
- Variation of electric motor speed.

The basic technique of power control of an AC supply is known as **phase control** which is explained in experiment 2.

In the circuit diagram and load waveform, shown in Fig. 4 and 5, the SCR is triggered at the delay angle **alpha** (α). This is the time taken, from the moment that the supply passes through zero until the pulse is generated. In the region between 90° and 180°, the SCR conducts and passes current through the load, this region is the conduction angle **beta** (β). At the angle 180°, the SCR switches off because the current drops below the holding current (Fig. 5).

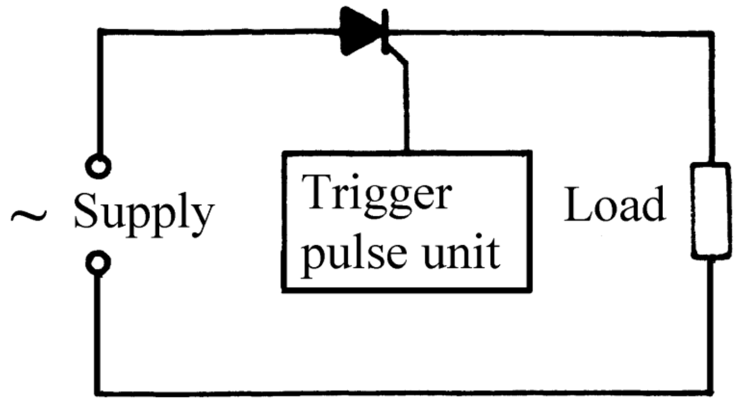


Figure 4

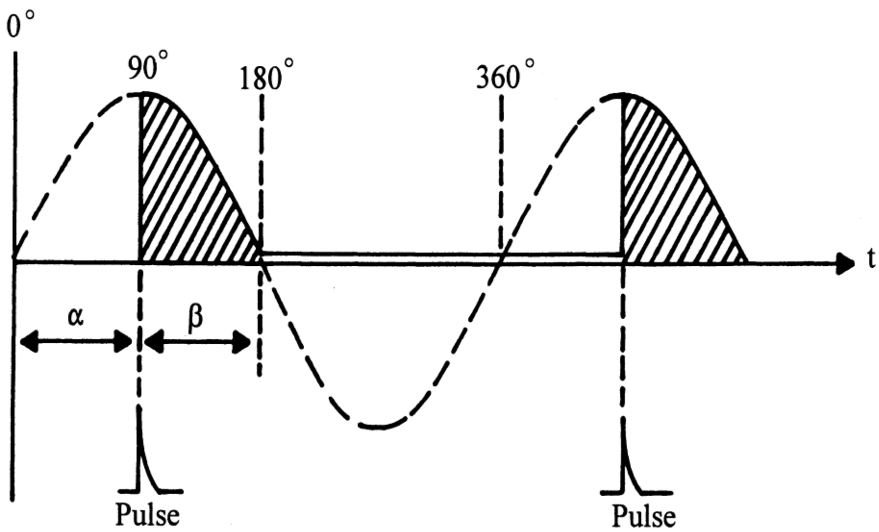


Figure 5

Phase controlled triggering is best explained by carrying out the following experiment.

	Experiment 2

To carry out this experiment you need:

- A 220 volt AC supply.
- A 220/0-12, 0-12 500mA transformer.
- A 12 volt lamp that acts as a load for the circuit.
- Two 1N4007 diodes.
- Two 1 amp 100 volt SCR's.
- A 50 volt 1 amp diode bridge rectifier.
- A unijunction transistor.
- A 6, 2 volt zener diode.
- A 0, 1 microfarad capacitor.
- A 100K potentiometer.
- Six ½ watt resistors: two 10 ohm, one 220 ohm, one 470 ohm, one 47 ohm and one 2K2.
- An oscilloscope.
- A circuit board.
- Connecting wires.
- A multimeter.
- A 100 microfarad capacitor.

1. Procedure

- Test the circuit shown in Fig. 6 on the next page and connect the oscilloscope probe across the lamp with the earth clip to the negative side of the half controlled bridge, i.e., the anode side of the SCR's.

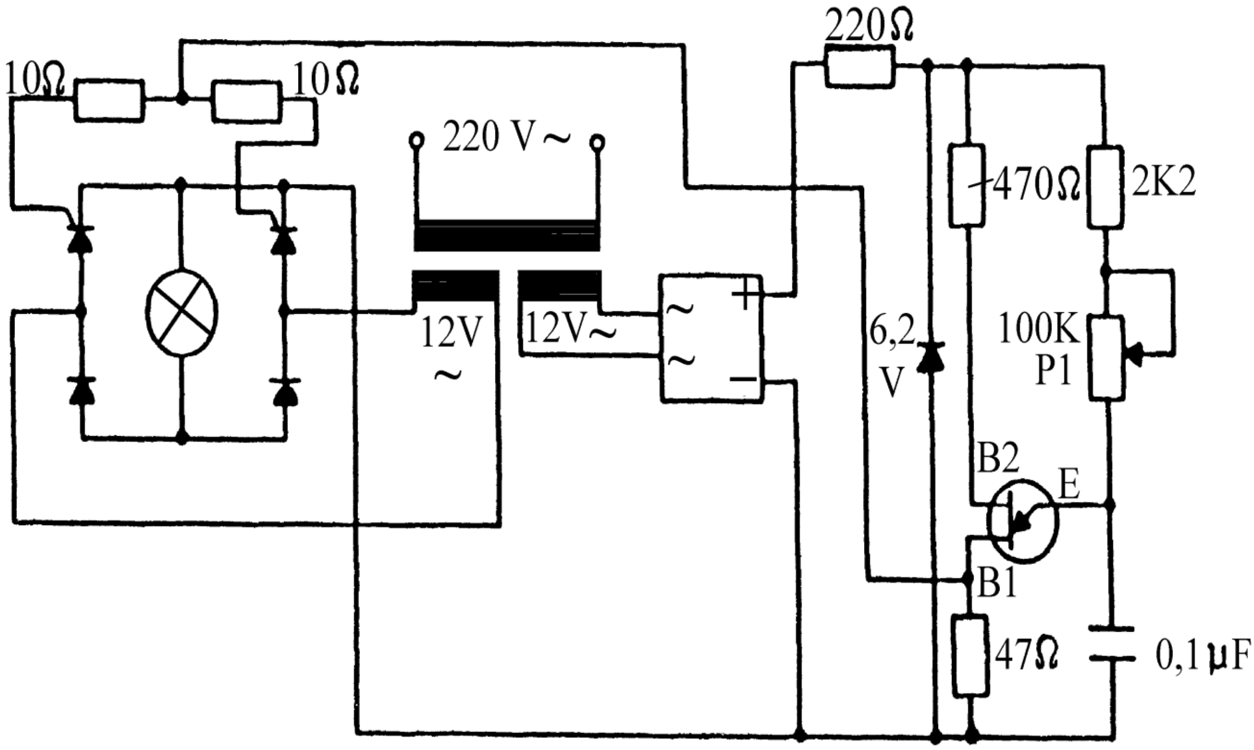
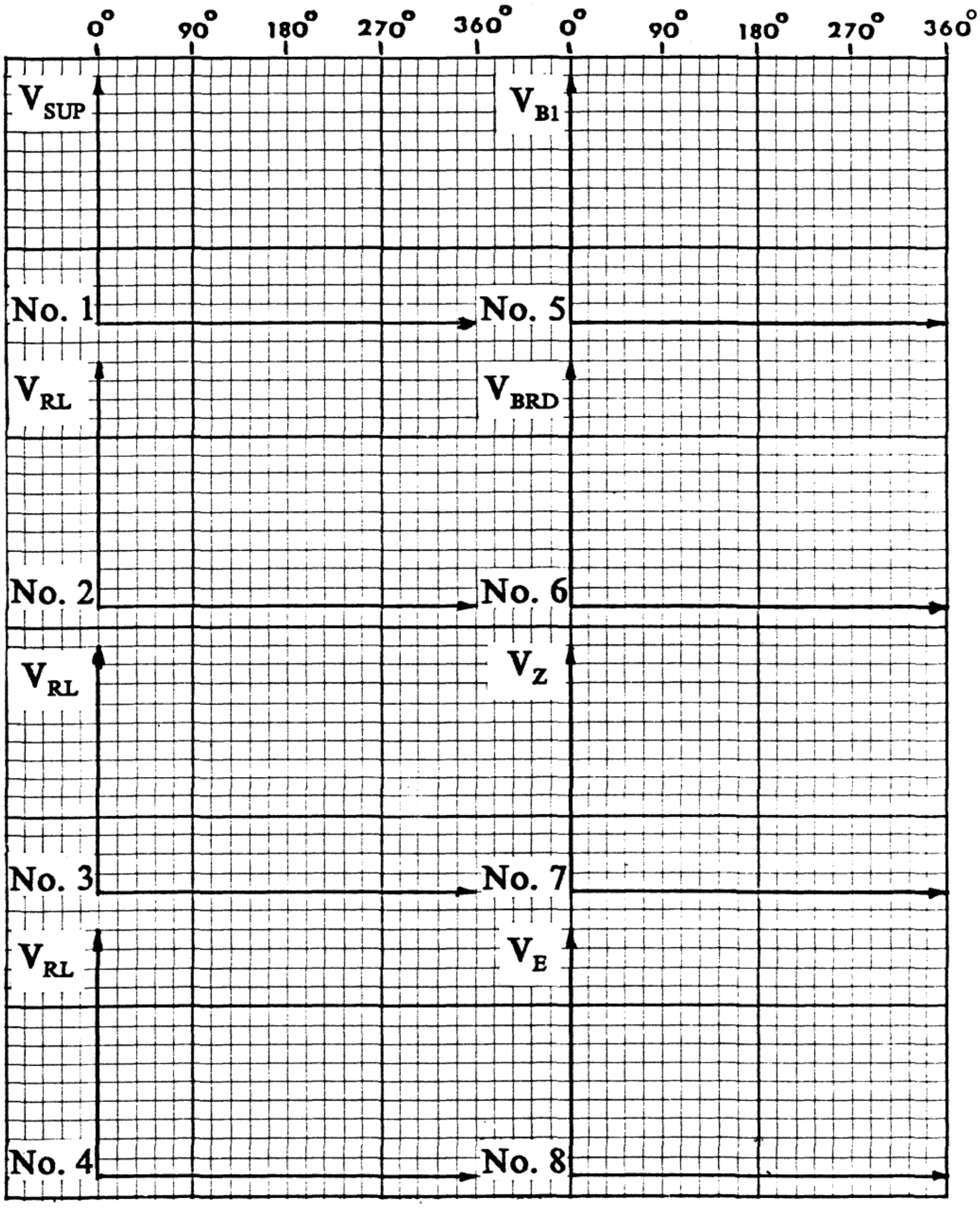


Figure 6

- Adjust the potentiometer P1 and note the effect on the lamp brilliance as well as on the load waveform.

- Adjust P1 to get approximately half brilliance, i.e. alpha (α) set to 90°.

- Measure the AC voltage supply to the half controlled bridge with the oscilloscope and draw the waveform on axes No. 1 on the graph paper on the next page.



- For the waveform in point 4, complete the sentences below. (The waveform on the oscilloscope is divided into time and not degrees, therefore the waveform must be converted into degrees where necessary).

  a) Period: __________ms or __________ degrees.

  b) Period of half cycle: ____________________ or ------ degrees.

  c) Therefore degrees per 1 ms = __________ degrees

  d) Peak voltage: __________ volts.

- Connect the oscilloscope over the lamp and draw the waveform on axes No. 2.
- Measure the mean DC voltage over the lamp with the multimeter.

  Voltage = __________volts.

- Indicate the delay angle α and the conduction angle β on the waveform.
- Adjust P1 for maximum α and draw the waveform, complete with delay and conduction angles in degrees using axis No. 3 of the graph paper.
- Measure the mean DC voltage over the lamp with the multimeter.

  Voltage = __________volts.

- Adjust P1 for minimum α and draw the waveform, complete with conduction and delay angle in degrees using axis No. 4 of the graph paper.
- Measure the mean DC voltage over the lamp with the multimeter.

  Voltage = __________volts.

- Compare the average DC voltages measured for different settings of α.

  α Minimum average voltage = __________volts.

  α 90^o average voltage = __________volts.

  α Maximum average voltage = __________volts.

  Seeing that the load in Experiment 2 is a resistive one, we can assume that the current through the load, has also decreased. From this and the voltage values above it should be clear that the **bigger** the delay angle α is, the **lower** the average DC power (in watt) delivered to the load.

- With α at approximately 90^o, connect the oscilloscope to measure gate trigger pulses and draw the waveform on axes No. 5 of the graph paper.
- Adjust the potentiometer in an anti-clockwise direction to decrease its resistance and explain why the first trigger pulse in a particular half cycle, appears earlier with a reduced value of P1.

  ____

  ____

  ____

- Measure the output waveform of the bridge rectifier and draw this waveform on axes No. 6 of the graph paper.
- What is the phase relationship at 180^o and 360^o, between the output waveform of the half controlled thyristor bridge and the bridge rectifier output voltage ?

  ____

- Measure the voltage waveform across the zener diode and draw this on axes No. 7 of the graph.

- Explain why the full wave rectified waveform is being clipped off. (If you have forgotten the operation of the zener diode, refer to module BE-6).

- Explain what happened to the clipped off portions and verify your answer by using the oscilloscope.

- Connect the oscilloscope probe across the lamp and adjust the potentiometer to ensure that α is 36^o.

Remember 1 ms = 18^o

- Remove the probe and connect it to the emitter of the UJT.

- Draw this waveform on axes No. 8 of the graph paper. This sawtooth waveform can be used in practice as a reference waveform for fault finding the trigger pulse unit, seeing that this waveform is displayed much easier on the oscilloscope screen than the short duration pulse generated on base 1 of the UJT. When it has been established that the sawtooth waveform is present on the emitter terminal, it can be safely assumed that the UJT is in working order and that the trigger pulse will be generated on base 1 of the UJT.

- It was seen a few points earlier that the outputs of the half controlled thyristor bridge and the bridge rectifier are in phase and therefore synchronised. To see the effect on the circuit, when there is no synchronisation between the half controlled bridge and the trigger pulse unit, complete the following :

  a) Connect the oscilloscope to measure lamp voltage waveform and adjust α to approximately 150^o

  b) Temporarily connect a 100 microfarad capacitor across the DC output of the bridge rectifier and describe what happens with the waveform across the lamp and a possible explanation for this behaviour.

  c) What effect does the capacitor have on the voltage across the zenerdiode ?

2. **Conclusion:**

To ensure that **α stays constant** for each individual half cycle for a given setting of P1, the trigger pulse unit must be **SYNCHRONISED / NON-SYNCHRONISED** with the half controlled bridge. To ensure this, the supply to the trigger pulse unit must be **SMOOTHED / UNSMOOTHED**. **(Delete the inapplicable words).**

Self test 2

Answer the questions below without referring to your notes.

1. Name three applications where in practice the technique of phase control may be used.

   a) ______________________________

   b) ______________________________

   c) ______________________________

2. Draw a full wave DC load waveform with α at 60^{o} and indicate α and β on the drawing.

3. In how many ms. is a complete cycle of 360^{o} completed when the frequency is 50Hz? Show the formula.

   ____

4. Why is the DC supply to the trigger pulse unit not smoothed?

   ____

5. Why is it necessary to ensure that the trigger pulse unit is synchronised with the half controlled bridge?

   ____

6. Circle and name the half controlled thyristor bridge as well as the trigger pulse unit on the circuit drawing in Fig. 6.

Ask for the criterion test when you feel ready.

NOTES

Module 14
FF - 2 Faults and circuits 2

Purpose of module

This objective will enable the learner to

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Identify the faulty component or components, if any, in a circuit.
2. Identify the type of components in a circuit.
3. State the labels for each pin of the components in a circuit.
4. Name the mode (configuration) in which the component is connected in a circuit.
5. State the purpose of the component in the particular mode in a diagram.
6. Describe the test conditions to be expected at the pin connections of the components should the circuit be healthy.

Procedures Relating To This Module

- All faulty components must be identified.
- All answers must be in writing.
- All answers must be correct and in accordance with the module notes.

1. *Introduction*

How to go about finding faults in electronic circuits

For successful fault finding in electronic circuits, a systematic and thoughtful approach to the testing of the circuits must be used. However, a systematic procedure alone is not enough: you must know the reason for every step in the procedure and must consider the results you obtain at each step. The deductions you make will be a guide in accurately locating the fault or faults.

In general, the successful testing of electronic circuits depends on the following points.

1. A fundamental knowledge of the circuit to be tested.

2. An understanding of the operation of the components in the circuit.

3. The ability to use the test equipment available.

4. The ability to analyse your results correctly.

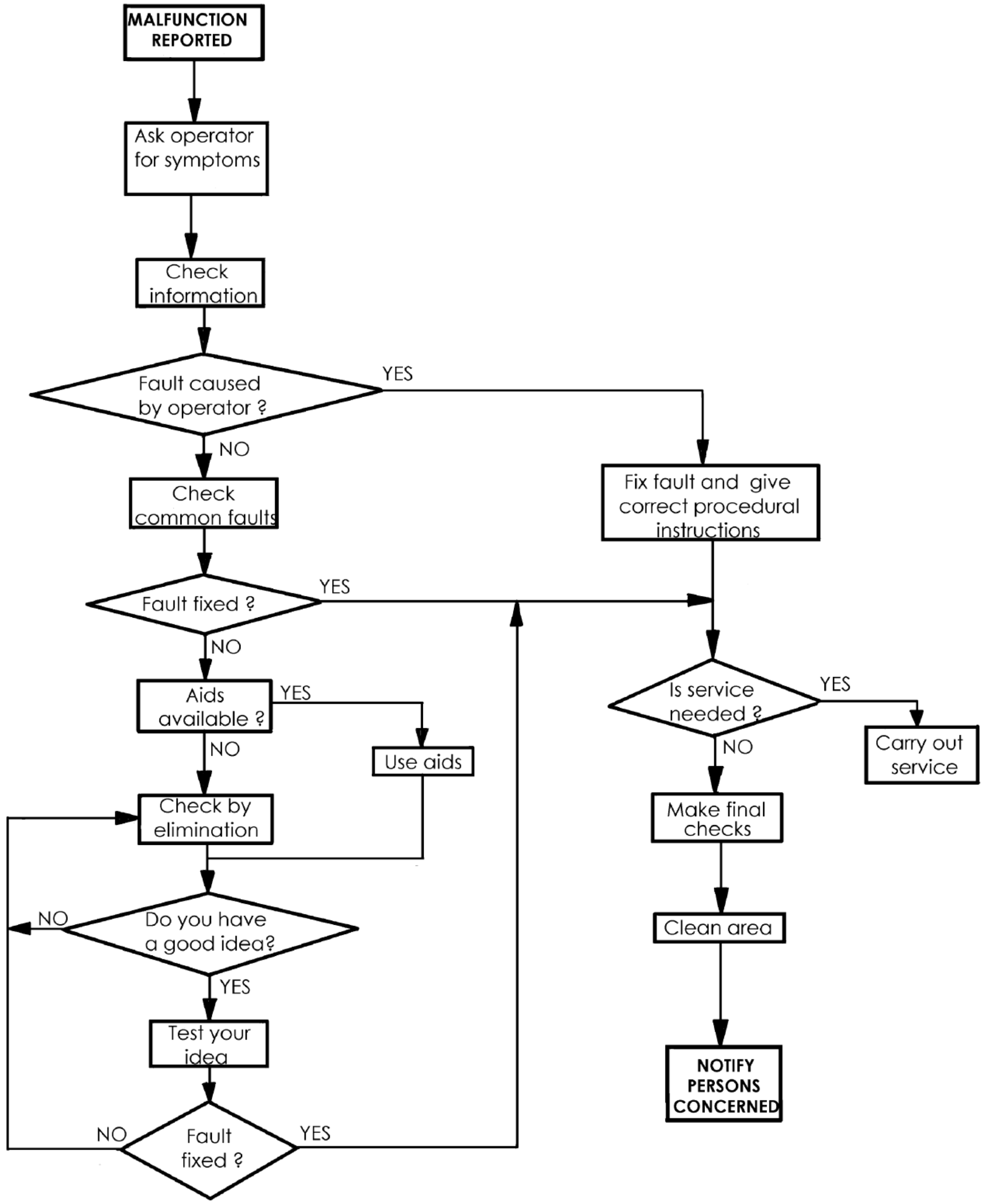
5. Use the BASIC FAULT FINDING FLOW CHART on page 4 of this module.

In dealing with the simple electronic circuits, isolate the defective stage first and then find the defective component within that stage. In more complex electronic circuits it is necessary to first locate the defective section, then the defective stage, and finally the defective component.

If the methods and procedures described above are clear, ask your Facilitator for a faulty circuit to practice on until you are confident that you can trace faults.

	Ask for the criterion test when you feel ready.

Basic fault finding flow chart



NOTES

Module 15
TI - 2 Use an oscilloscope

Purpose of module

The objective of this module is to enable the learner to use an oscilloscope.

Learning outcomes

The learning outcomes identify what you, as a learner, will be able to achieve on completion of this module.

You will be able to

1. Set up an oscilloscope for practical application in accordance with the oscilloscope manual.
2. Measure periods from the cycles on the screen of the oscilloscope to obtain frequency.
3. Measure AC and DC voltages, peak to peak values and calculate RMS voltage values from measured results.

Procedures Relating To This Module

- All the measurements must be within 10% of the true values.

1. Introduction

The cathode ray oscilloscope presents a visual display of time varying voltage and also plots graphs of AC voltages over a wide range of frequencies and amplitudes.

It automatically plots a graph of voltage against time.

The main components of a cathode ray tube (CRT) are shown in Fig. 1.

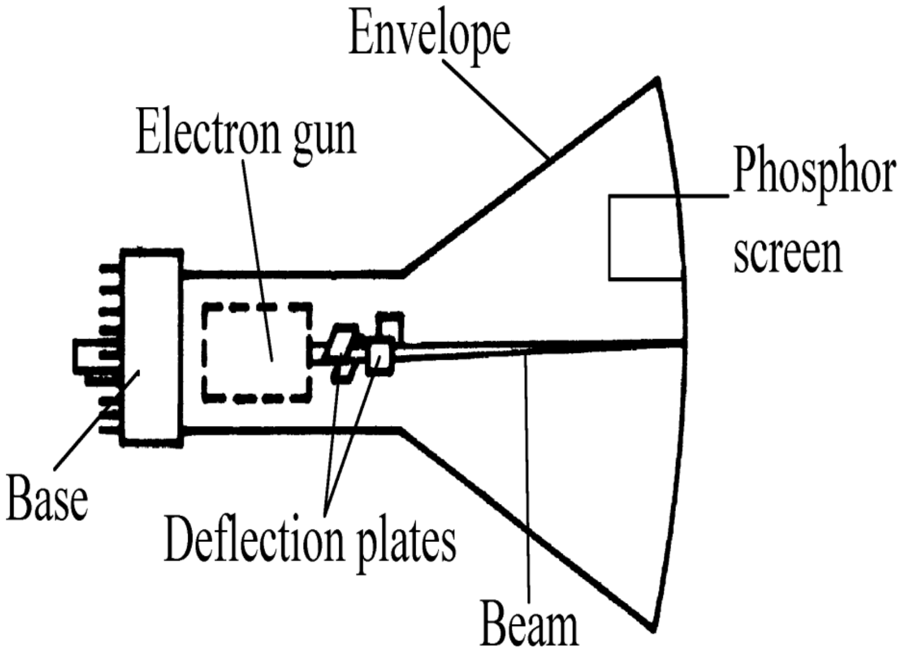


Figure 1

2. Construction and Operation of the Cathode-ray tube (crt)

Cathode-ray tubes are used as the screen in an oscilloscope and have faces which are either round or rectangular.

The envelope is made of glass which has been evacuated. The electron gun assembly is inside the envelope.

The glass face or screen of the CRT is coated internally with phospherous chemicals. These chemicals emit light when they are struck by the fast moving beam of electrons which is emitted from the electron gun.

The elements of the electron gun are connected to pins at the base. The beam is accelerated by a high voltage e.g. as high as 33000 volts.

Deflection plates are used to deflect the beam either upwards, downwards, to the left or to the right. The deflected beam leaves a trace on the screen and this trace is examined and calibrated.

3. Operating controls on the oscilloscope (Fig. 2)

The various controls are described below.

1. Intensity

   This control sets the brightness or intensity of the light trace on the CRT. When the control knob is turned in a clockwise direction the brightness increases. The intensity should not be set too high because this may damage the CRT.

2. Focus

   This control is adjusted in conjunction with the intensity control to obtain the sharpest or clearest trace on the screen. The interaction between these two controls may require readjustment of one, after the other has been adjusted.

3. Horizontal and vertical controls

   These are trace-positioning controls. They are adjusted to centre the trace both vertically and horizontally on the screen. An etched faceplate called the graticule is in front of the CRT screen. (See Fig. 3 on the next page).

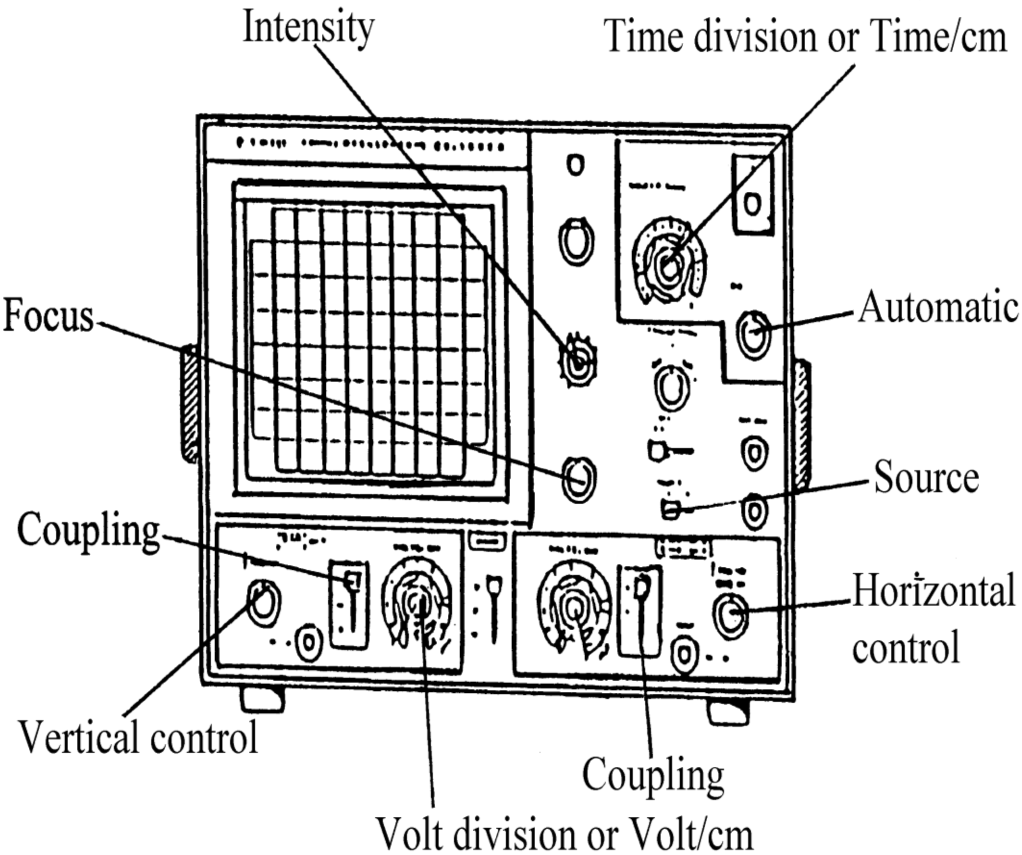


Figure 2

The etchings are in the form of horizontal and vertical lines. Calibration marks are placed on the centre vertical and horizontal lines of the faceplate

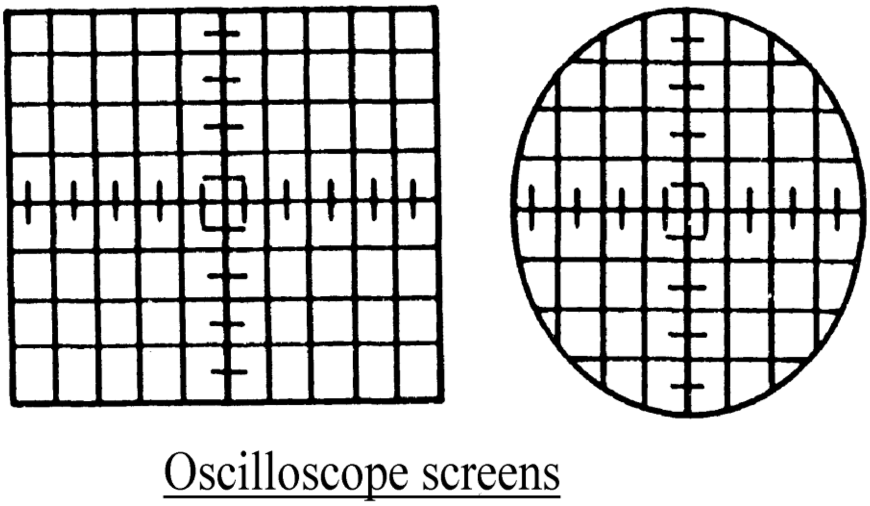


Figure 3

4. Volts / Div. or Volt / Cm.

The control knob labelled Volts/Div. or Volt/cm is a switch control. A dot on the control knob can be turned to correspond to the voltage markings on the panel around the control. The setting of the Volts/Div. control knob, determines the voltage which is equal to every division of vertical signal deflection on the screen. (See Fig. 4).

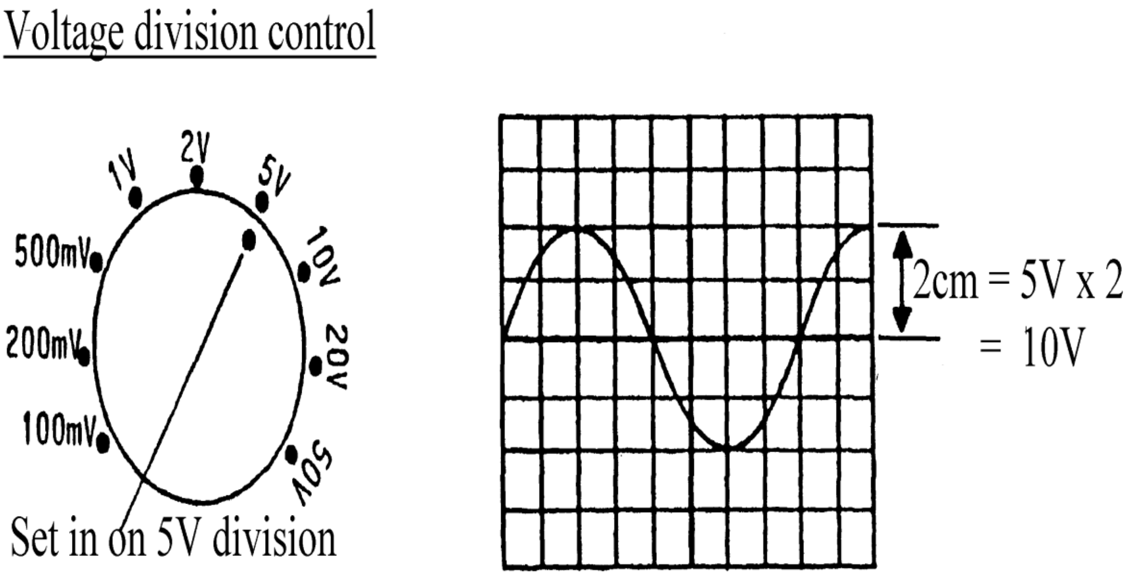


Figure 4

5. Time / Div. or Time / Cm.

This is also a switch control. A dot on the control knob can be turned to correspond with the time markings on the panel around the control knob. The settings of the Time/Div. control knob, determine the time it takes for the trace (beam) to move horizontally across one division of the graticule (Fig. 5 on the next page).

There are usually four triggering controls on the oscilloscope, namely:

- Auto (matic) (Fig. 2)

  In this position the trigger is free running, which means that a trace is always shown on the screen, i.e. a straight horizontal line appears across the screen. The oscilloscope is often used in this mode.

  When the auto knob is not in the auto-mode, triggering depends on some external or internal signal. In the non-automatic mode there will be no trace on the screen in the absence of a triggering signal.

- Slope

  This switch is marked + and –, and its setting determines whether triggering of the sweep, i.e. when the beam sweeps across the screen leaving a trace, is affected by the positive or negative portion of the input signal.

- Coupling (Fig. 2)

  This selects the type of input coupling to the oscilloscope and can be only on AC - DC, or GND. When switched to GND, the trace can be aligned with any of the horizontal graticule lines. When measuring voltages, this line will act zero volt reference.

- Source (Fig. 2)

  This switch is for switching to Ext. (ernal) or Int. (ernal) triggering.

  The controls described may have other names, depending on the make and type of oscilloscope.

  Additional features and controls are found on many oscilloscopes. However, knowledge of their operation is not needed at this stage.

  Because of the wide variety of oscilloscopes available it is advisable to study the manual supplied with your oscilloscope. (Ask your instructor for a copy).

Self test 1

Answer the questions below without referring to your notes.

1. Explain the function of the oscilloscope controls given below.

   a) Intensity

   b) Focus

   c) Horizontal and vertical controls

   d) Volts/cm.

   e) Time/cm.

   f) Auto

   g) Slope

   h) Coupling

   i) Source

	Practice

Ask your instructor for an oscilloscope and set it up in accordance with the instructions given in the manufacturer's manual for practical application.

	When you have set up the oscilloscope and completed the self test, ask your instructor to check your work and if it is correct and then go on to the next section.

4. Measuring periods and frequency

The variation of alternating current with time can have many different shapes. The waveforms shown in Fig. 6 are typical of a few of these shapes.

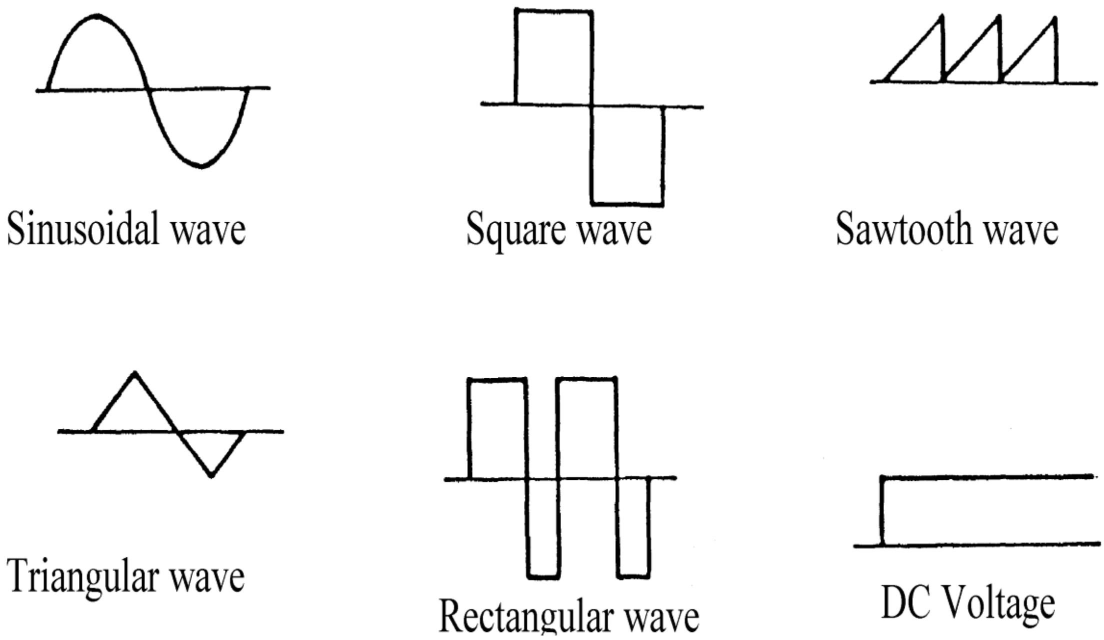


Figure 6

Various methods of measuring the period of a sinusoidal wave are shown in Fig. 7.

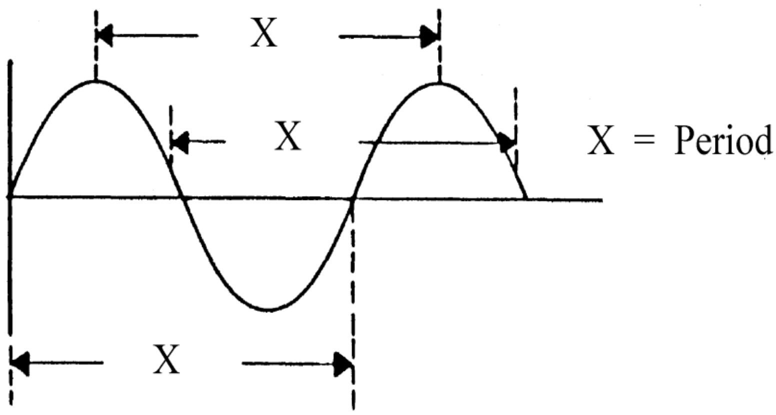


Figure 7

1. Setup and Analysis

- Set up your oscilloscope and obtain a free-running trace using the procedures described in the oscilloscope manual.
- Switch the input coupling control to AC.
- Set the Time/Div. control to 0, 1 ms (100µs).
- Ask your instructor to set an AC signal at a frequency of 2 000 Hz.
- Connect the oscilloscope input probe to the supply and adjust the trigger level to obtain a stable waveform display.
- Set the Volts/Div. switch to display about five divisions of the waveform.
- Centre the waveform vertically by using the vertical position control.
- Turn the horizontal position control until the beginning of the waveform is lined up with the first or second vertical graticule line and so that the waveform is centred within the CRT screen as shown in Fig. 8.

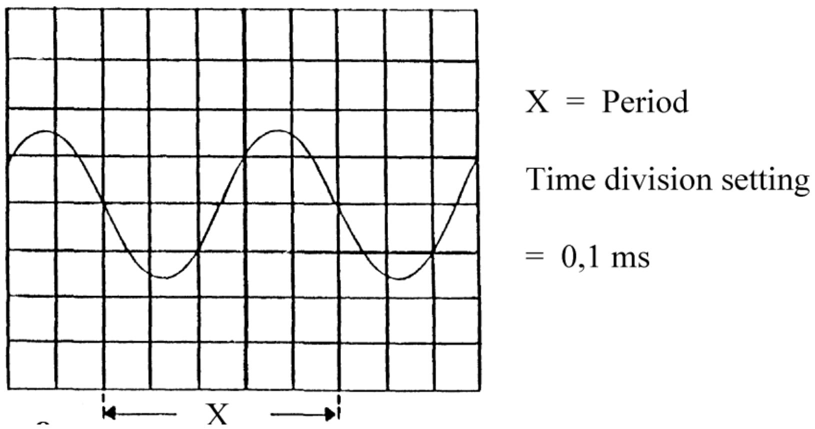


Figure 8

- Obtain the period by counting the number of vertical divisions and sub-divisions necessary for the wave-form to complete one cycle. Then use the formula below to determine the period of the waveform on your screen.

  Period = Number of divisions x Time/Div. setting

  e.g. Period = 5 divisions x 0,1ms

  = 0,5ms.

- Calculate the frequency by using the formula

$$Frequency = \frac{1}{Period}$$

  e.g. If the period is 0,5ms i.e. 0,5 x 10^{-3} seconds then

  = 2 000 Hertz. (Hz)

- Change the frequency of the signal generator and determine the new frequency displayed on the screen using the same method explained in the previous two steps.

  If you feel that you understand this section of the module continue, otherwise carry out more practices provided by your instructor.

5. Measuring AC, DC, Peak-to-Peak, and R.M.S. voltage values

1. Setup

- Set up your oscilloscope and obtain a free-running trace using the procedures described in the oscilloscope manual.
- Switch the input coupling control to GND.
- Set the Time/Div. to 1ms.
- Align this trace with the centre horizontal graticule line by using the vertical position control.

  This line will be the zero-volt reference.
- Set the Volts/Div. control to 5V.
- Ensure that the oscilloscope input probe is set to X1.
- Connect the oscilloscope input probe to 12V DC.
- Switch the input coupling control to DC.
- Your screen should look like that in Fig. 9.

  If the trace jumps below the zero-volt reference line it indicates a negative voltage and it can be moved above the line by changing the probe leads around.

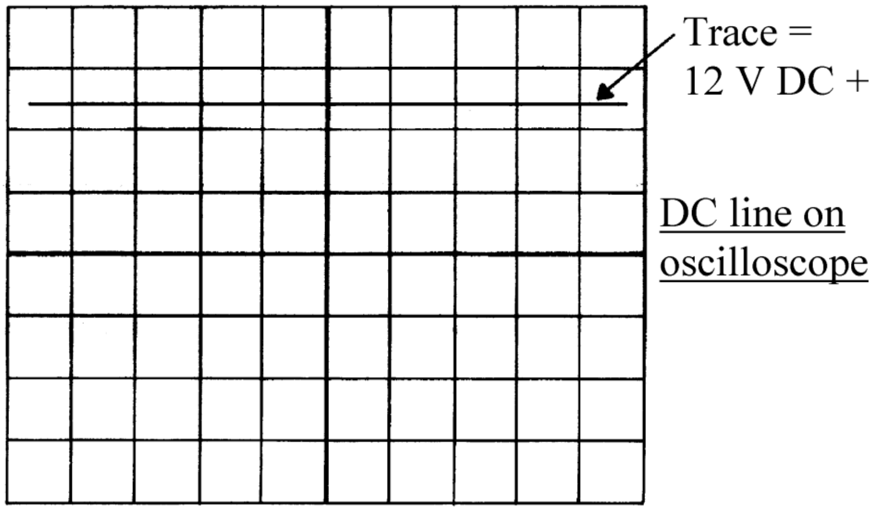


Figure 9

- Set the Volts/Div. control to 10V.

	The trace jumps only halfway towards its previous position. This is because 10 volts for every square are being measured

For accurate voltage measurements, the waveform (in the case of AC) or the deflection of the line (in the case of DC) must be displayed as big as possible on the screen by setting the VOLT/DIV. control as low as possible.

- Repeat steps 2 and 3 above.

- Align the trace with the bottom or second from the bottom horizontal graticule.

- Set the VOLT/DIV. control to 2V and switch the input coupling to DC.

- Count the number of divisions and subdivisions from the zero volt reference line to the trace.

- Determine the DC voltage by multiplying the number of divisions with the VOLG/DIV. setting and with the input probe setting, i.e.:

  i.e. DC voltage = number of divisions x Volt/Div. X probe

  =_______________X_______________X 1

  =_______________Volts

2. Analysis of waveform amplitude

- Again set up your oscilloscope and obtain a free running trace.

- Switch the input coupling control to AC.

- Align this trace with the centre horizontal graticule line by using the vertical position control.

- Set the Volts/Div. control to 2V.

- Connect the oscilloscope input probe 10x to 35 volts AC. Use a voltmeter to set up this voltage. Notice that a probe of 10x is used. Attenuated probes are used to protect the oscilloscope from high input voltages, to prevent the oscilloscope from affecting the system to be tested and to provide more accurate measurements.

- Switch on the 35 volts AC supply.

- Switch the input coupling control to AC.

- Align the negative peaks of the input waveform with one of the horizontal graticule lines, by using the vertical position control. Align the positive peak with the centre vertical graticule line, by using the horizontal position control. See Fig. 10 illustration.

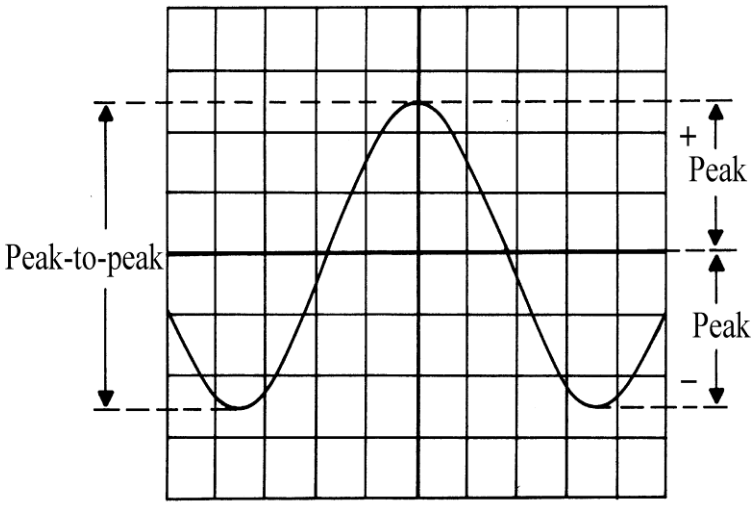


Figure 10

- As before, count the number of horizontal graticule divisions and sub-divisions from a negative peak on the horizontal graticule line, to the most positive peak as shown in Fig. 10. Use the formula below to determine peak-to-peak amplitude.

  Peak-to-peak = number of divisions x Volts/Div. setting x probe attenuation

	The formula considers probe attenuation.

Therefore: Voltage = 5 divisions x 2 Volts/Div X 10 probe attenuation.

Peak-to-peak voltage (Vp-p) = <u>100 volts</u>.

+ Peak and – Peak. If a waveform fluctuates symmetrically about a zero reference, then your + peak or – peak values are one half of the peak-to-peak amplitude.

$$\therefore Voltage = \frac{100}{2} = 50$$

Peak Voltage (Vp) = <u>50 volts</u>

3. Derived measurements

Amplitude measurements give you instantaneous peak values on the oscilloscope screen instead of the actual voltage or current value of a sinusoidal waveform. If the effective (root-mean-square) or average values are required they can be calculated from the amplitude measurements.

The root-mean-square (rms) value is the value of the input at the oscilloscope probe.

In this case 35 volts as set up with the voltmeter. Fig. 11 illustrates the root-mean-square value of a sinusoidal waveform in terms of its percentage of peak values.

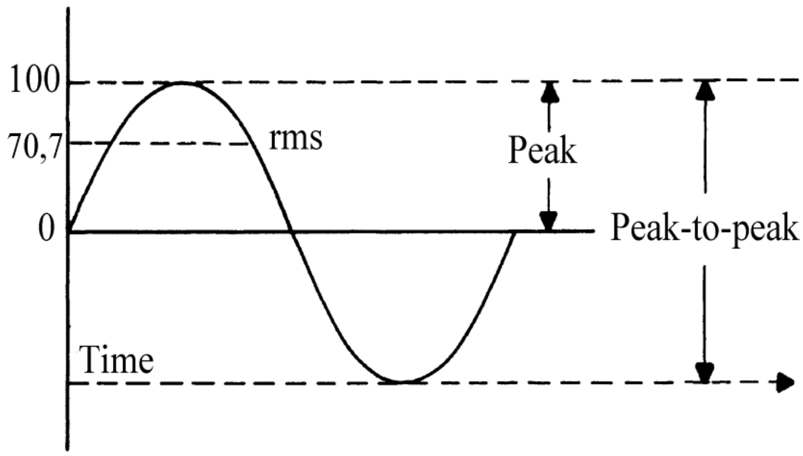


Figure 11

4. Root-mean-square

The most common way of specifying an alternating voltage or current is by its root-mean square (rms) value.

Important: If any AC voltage is shown without any reference to peak or peak-to-peak values, it must be assumed to be an RMS value.

Therefore if we refer to 12V, 24V, 220V, 380V and 550V AC, it in fact means Vrms values.

The rms voltage or current value of a sinusoidal wave is equal to the value of the DC voltage or current that would dissipate the same amount of energy in a purely resistive circuit. Fig. 12 on the next page illustrates graphically that the rms value of a sinusoidal waveform is 70, 7% of the waveform's peak value. Mathematically expressed this is :

RMS Value = 0,707 x Peak Value.

Therefore for the waveform on the screen

RMS = 0,707 x 50 Volts Peak

= <u>35, 35 volts</u>.

Self test 2

1. Given 20 volts peak to peak, calculate the peak voltage and the rms value.

2. Given 15 volts peak, calculate the peak to peak voltage and the rms value.

3. Given 10 volts rms, calculate the peak voltage and the peak to peak voltage.

4. Calculate the time taken to complete one full cycle if the frequency is a 100 Hz frequency. Show the formula used.

Ask for the criterion test when you feel ready.

End of the module

Glossary of terms

CE	Common emitter
CB	Common base
CC	Common collector

End of the guide