Step in Electronics practicals

Real world circuits applications

2

IBYIMANIKORA Ibrahim

From *Theories* to *Practicals*

Dedicace

God Almight

Parents

Jyuyisenga Annick Rebecca

ISBN-10: 1517491371
ISBN-13: 9781517491376

Scan this QR code for contact details

Tel: (+250) 788 477 999
E-mail: ibrazzis@hotmail.com
KIGALI-RWANDA

Book with the same Author

Electronics Practicals "*real world circuits applications*";

ISBN: 9781523374861

Preface

The field of electronics is very diverse, and career opportunities are available in many areas. Because electronics is currently found in so many different applications and new technology is being developed at a fast rate, its feature appears limitless. There is hardly an area of our lives that is not enhanced to some degree by electronics technology. The study of electronics is one of the basic steps in gaining an understanding of all modern technology and science also; our everyday life depends a lot on the electronic. So this book of *step in electronics practicals* provides a comprehensive and clear coverage of electronic practical concepts, practical applications and troubleshooting. It is designed to cover a wide range of topics that make up the field of electronics in well-organized and highly informative manner. Special emphasis is placed on practical applications; thus, this book in not theoretical one, but an enlightening practicals of the usefulness of rapidly growing field of electronics. In this book many topics have been strengthened and some topics, innovative and features have been added related to the real world. These practical topics are coordinated with the text showing working principles and their practical design and will make any candidate to be more effective in practical applications. Additionally, real world circuits design make the book more visually interesting and easier to use. The circuits provided have been simulated using reliable and accurate method and tested using real components. The presentation is tutorial in nature in order to enhance the value of the book to the reader and foster a clear understanding of practical project topics.

This book of *Step in Electronics Practicals* will provide the required practical skills that is useful for ENGINEERING STUDENTS and PRACTICING PROFESSIONALS. Over hundred projects have been discussed in this book that give a clear coverage of real world applications and openness to develop any electronic project that responds to everyday person needs.

Features

- Math level is limited to basic algebra
- Full-color format
- A chapter opener includes a list of chapter objectives and reference
- Numerous schematic circuits with their related circuit connections as they appear in real world
- Circuit connection on PCB and its related circuit artwork to give idea of layout for final projects

Contents

Chapter four

Electromagnetic Devices and Applications

Chapter five

Sensors and Transducers

Chapter six

Transistor and Applications

Chapter seven

Operational Amplifier Circuits

Chapter eight

The 555 timer and Application

Apandix

Circuit Designation and Circuit Workplace

Chapter one

Objectives

After completing this chapter, you should

- Be able to identify different circuit notation and circuit reference designators
- Be able to develop printed circuit board circuit layout
- Be able to produce a net presentable circuit project by using printed circuit board
- Have an understanding and the use of electronic testing instruments

Further reading
Study aids for this chapter are available at
IEEE (1975), "22. *Class Designation Letters*", IEEE Std 315-1975: Graphic Symbols for Electrical and Electronics Diagrams (Including Reference Designation Letters) (Reaffirmed 1993), IEEE and ANSI, New York, NY

The flow of electrical current from supply, through electrical or electronic devices and back to the supply form electrical circuit. Electrical circuit is recorded and shared by using diagram called schematic. A schematic diagram is a convenient and informative method for documenting electronic circuitry. For any given electronic circuit that uses standard symbols and designations anyone, no matter what language he speaks, can understand and explain the basic electronic theory from the schematic. A schematic diagram is intended as a drawing that shows, by means of graphic symbols, the electrical connections, components, and functions of a specific circuit arrangement.

The basic building blocks of schematic diagram use a set of standardized symbols to represent different component types. The letters next to the electrical symbols are labels that help identify the component, and these labels are called reference designators.

A circuit is constructed on a board that interconnects the components and the supply. While constructing a circuit the choice of the board or circuit workplace depends on the purpose the circuit is intended to and desired fastening of the components and wires. This chapter presents the most common reference designators that used to identify the electronic components and the boards to construct the circuit.

1. 1. Circuit Notation and Reference Designators

When developing a circuit diagram, it is necessary to identify the individual components. This is particularly important when using a parts list as the components on the circuit diagram can be cross related to the parts list or Bill of Materials. It is also essential to identify components as they are often marked on the printed circuit board and in this way the circuit and the physical component can be identified for activities such as repair, etc..

In order to identify components, what is termed a circuit reference designator is used. This circuit reference designator normally consists of one or two letters followed by a number. The letters indicate the type of component, and the number, defines which particular component of that type it is. An example may be R13, or C45, etc..

To standardize the way in which components are identified, the IEEE introduced a standard IEEE 200-1975 as the "Standard Reference Designations for Electrical and Electronics Parts and Equipments." This was later withdrawn and later the ASME (American Society of Mechanical Engineers) initiated the new standard ASME Y14.44-2008.

Some of the more commonly used circuit reference designators are given in table 1-1.

Most commonly used Circuit Reference Designators	
Reference Designator	**Component Type**
ATT	Attenuator
BR	Bridge rectifier
BT	battery
C	Capacitor
D	Diode
F	Fuse
IC	Integrated circuit - an alternative widely used non-standard abbreviation
J	Connector jack (normally but not always refers to female contact)
L	Inductor
LS	Loudspeaker
P	Plug
PS	Power supply
Q	Transistor
R	Resistor
S	Switch
SW	Switch - an alternative widely used non-standard abbreviation
T	Transformer
TP	Test point
TR	Transistor - an alternative widely used non-standard abbreviation
U	Integrated circuit
VR	Variable resistor

X	Transducer
XTAL	Crystal - an alternative widely used non-standard abbreviation
Z	Zener diode
ZD	Zener diode - an alternative widely used non-standard abbreviation

Table 1-1: Circuit notation and reference designators

On the other hand, there are various symbols used to represent different electronic components and devices in circuit diagrams from wires to batteries and passive components to semiconductors, logic circuits and highly complicated integrated circuits. By using a common set of circuit symbols in circuit diagrams, it is possible for electronic engineers around the globe to communicate circuit information concisely and without ambiguity. Although there are a number of different standards in use for the different circuit symbols around the globe, the differences are normally small, and because most systems are well known there is normally little room for ambiguity.

1. 2. Circuit design

The process of circuit design can cover systems ranging from complex electronic systems all the way down to the individual transistors within an integrated circuit. For simple circuits, the design process can often be done by one person without needing a planned or structured design process, but for more complex designs, teams of designers following a systematic approach with intelligently guided computer simulation are becoming increasingly common. In integrated circuit design automation, the term "circuit design" often refers to the step of the design cycle which outputs the schematics of the integrated circuit. Typically this is the step between logic design and physical design.

Official circuit design usually involves the following stages:

- Writing the requirement specification after liaising with the customer
- Writing a technical proposal to meet the requirements of the customer specification
- Synthesizing on paper a schematic circuit diagram, an electrical or electronic abstract circuit that will meet the specifications
- Calculating the component values to meet the operating specifications under specified conditions
- Performing simulations to verify the correctness of the design
- Building a breadboard or other prototype version of the design and testing against specification
- Making any alterations to the circuit to achieve compliance
- Choosing a method of construction as well as all the parts and materials to be used
- Presenting components and layout information to draught persons, and layout and mechanical engineers, for prototype production
- Testing or type-testing a number of prototypes to ensure compliance with customer requirements

- Signing and approving the final manufacturing drawings
- Post-design services (obsolescence of components etc.)

1. 3. Circuit Workplace

When you are designing a circuit you need to test it and verify its proper working. The circuit can be built either on solderless board for easy replacement of devices and for temporally construction or be soldered for long time testing or for final project. This is achieved by using breadboard or printed circuit board.

1. 3. 1. Breadboard

Breadboards are often used to test new circuit designs because it is faster and easier to experiment on a breadboard than it is to solder (fuse in to place) circuit components. In the next tutorials, you will gain hands-on experience building and testing electronic circuits by creating a breadboard circuit with basic electronic components, and a battery or an AC supply. Even if breadboard is not suitable for large projects, in the next chapters we will give some of the basic electronic projects tested and implemented on breadboard that will help prepare you for more advanced electronic projects development.

Breadboard Description

A breadboard (protoboard) is a construction base for prototyping of electronics. The term is commonly used to refer to solderless breadboard (plugboard). The green lines on the image, figure 1-1, on the right show how the sockets are connected. You can see that the vertical columns of holes labelled with a "+" are connected to each other, as are the columns of holes labeled with a "-". The columns labeled with a "+" are called the power bus, and you will connect one of them to a positive input voltage, such as the positive terminal of a 9V or 12V battery. One of the columns labeled with a "-" (the ground bus) will be attached to the negative terminal of the battery. Note that in each row (numbered 1 through 29) sockets "a" to "e" are connected to each other. And "f" to "j" are also connected to each other. These groups of connected sockets form a node.

Figure 1-1 shows some of the parts of a breadboard, including a map of the connections between breadboard sockets.

Instructions for Working with Breadboards

1. Use 22-gauge (0.33mm^2) solid wire.
    a. If the wire is bigger, it could permanently deform the spring contacts in the breadboard sockets and make those sockets unreliable in the future.
    b. Test probes from multimeters are definitely too big for the holes in breadboards.
2. Some breadboards have bus lines that run all the way from one end of the board to the other. Bus lines are the columns of sockets beneath the + and − signs located at the edges of the breadboard. If you are not sure how your specific breadboard is wired internally, use your multimeter to verify which groups of holes are connected.

The breadboard may also come with a map of its connections. This map, if present, can usually be found in the breadboard's instructions.

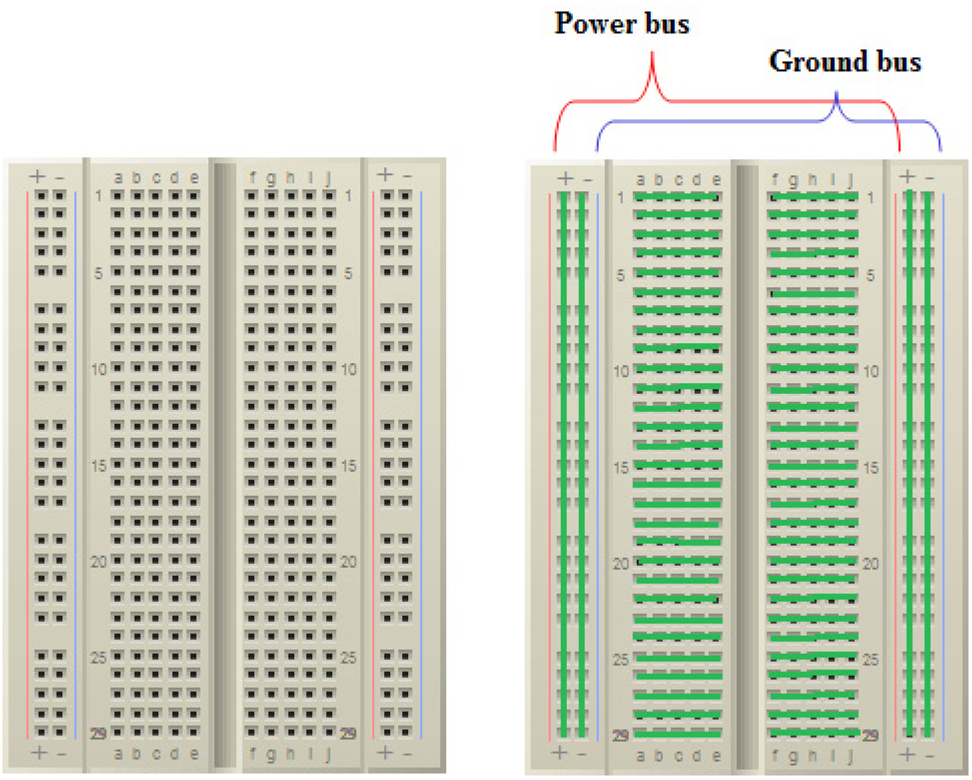


Figure 1-1: Breadboard description

3. Breadboards are not meant for high-current connections.
4. Breadboards are not meant for high-voltage circuitry.
5. Be careful not to push insulation down into the spring contact in the breadboard sockets, as this can lead to a bad connection. Only put the bare, un-insulated part of the wire, into the breadboard socket.
6. Stripping too much insulation or leaving long component leads may create accidental connections in the air above the breadboard, if two wires accidentally touch.
7. Some parts, like diodes, have a direction. Adding a part in the wrong direction might damage it and make the circuit not work.
8. It is good practice to build up a circuit one stage at a time and to check the connections using continuity tester or ohmmeter (a multimeter set to measure resistance) before applying power.
9. Light-emitting diodes (LEDs) always require a resistor in series to protect them from burning out.

Breadboards have their limitations, though. First and foremost, they are intended for temporary construction only. If you pick up a breadboard, turn it upside-down, and shake it, any components plugged into it are sure to loosen, and may fall out of their respective holes. Also, breadboards are limited to fairly low-current (less than 1 amp) circuits.

These spring clips have a small contact area, and thus cannot support high current without excessive heating.

The drawback of the circuit on breadboard is that the circuit can be easily unwillingly modified and lose its proper working principle where need to be soldered on printed circuit board (PCB). For greater permanence, one might wish to choose soldering or wire-wrapping. These techniques involve fastening the components and wires to some structure providing a secure mechanical location (such as a phenolic or fiberglass board with holes drilled in it, much like a breadboard without the intrinsic spring-clip connections), and then attaching wires to the secured component leads.

1. 3. 2. Printed Circuit Boards (PCB)

Printed circuit board, sometimes abbreviated PCB, is thin plate on which chips and other electronic components are placed and which mechanically supports and electrically connects electronic components using conductive tracks, pads and other features etched from copper sheets laminated onto a non-conductive substrate. The circuits are formed by a thin layer of conducting material deposited, or "printed," on the surface of an insulating board known as the substrate. Individual electronic components are placed on the surface of the substrate and soldered to interconnecting the circuits.

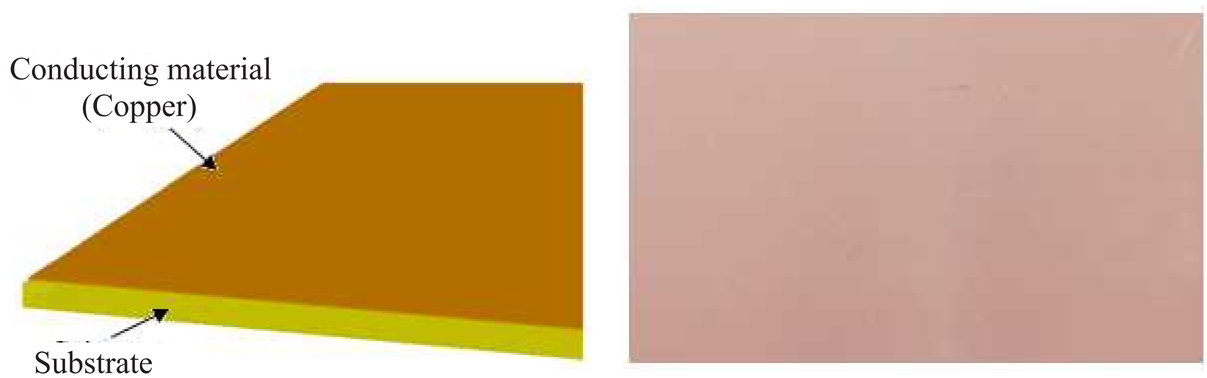


Figure 1-2: Single side plain PCB Circuit board design

Types of Printed Circuit Boards

There are three major types of printed circuit board construction: Single-sided (one copper layer), double-sided (two copper layers), and multi-layered.

- **Single Sided Board**

Single-sided PCB has one copper layer. This is the least complex of the Printed Circuit Boards, since there is only a single layer of substrate. All electrical parts and components are fixed on one side of the substrate and copper traces are on the other side. When the number of components becomes too much for a single-sided board, a double-sided board may be used.

- **Double Sided Board**

Double-sided PCB has two copper layers. This is the most common type of board, where parts and components are attached to both sides of the substrate. In such cases, double-sided PCBs that have connecting traces on both sides are used.

Double-sided Printed Circuit Boards usually use through-hole construction for assembling of components.

- **Multi Layered Board.**

Multi layered PCB consists of several layers of substrate separated by insulation. Most common multilayer boards are: 4 layers, 6 layers, 8 layers, and 10 layers. However, the total number of layers that can be manufactured can exceed over 42 layers. These types of boards are used in extremely complex electronic circuits.

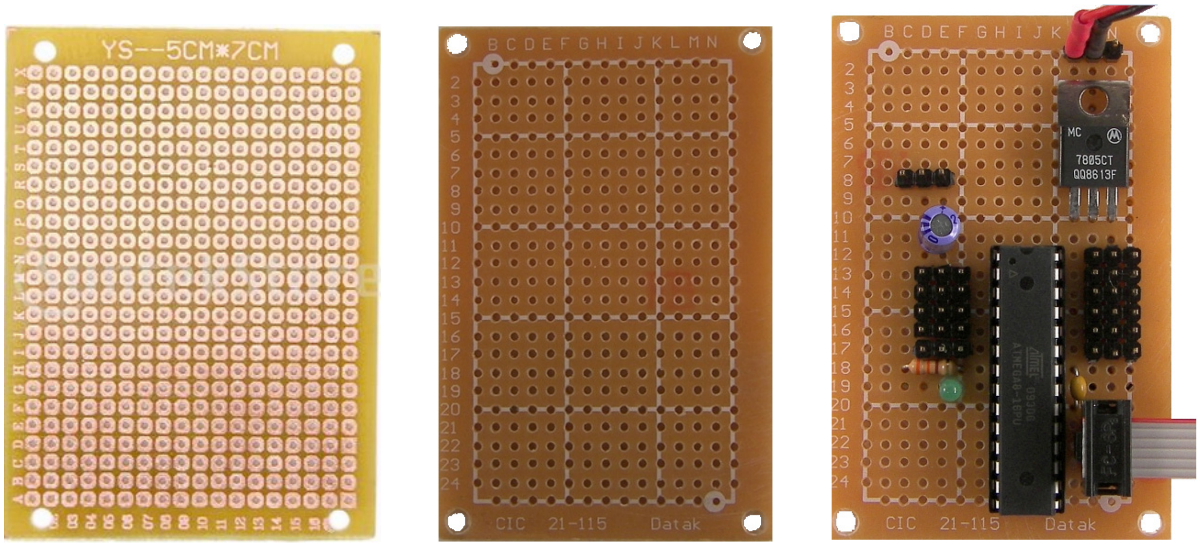


Figure 1-3: Example of a typical printed circuit board PCB; (a) Solder side, (b) Component side, (c) Component soldered on PCB

Creating Printed Circuit Boards

After having the circuit schematics, you do some computer aided simulations when the circuit is working great, you can build it on solderless board (breadboard) for some error correction. The drawback of the circuit on breadboard is that the circuit can be easily unwillingly modified and lose its proper working principle. Whether your circuit is a project for school/college or is a final piece of electronic in a professional product for company, implementing the circuit on a printed circuit board (PCB) will give it a much better professional look besides giving an idea of how the finished product will look like. The PCB boards are made from glass reinforced plastic with copper tracks in the place of wires. Components are fixed in position by drilling holes through the board, locating the components and then soldering them in place. The copper tracks link the components together forming a circuit.

This article will describe the different methods by which you can create a printed circuit board (PCB) for an electrical/electronic circuit using different methods suitable for small to large circuitry.

Procedures

1. Choose a method to use for creating the PCB.

Your choice will usually be based on the availability of materials needed by the method, the technical difficulty level of the method or the quality of PCB you desire to obtain. Here is a brief summary of the different methods and their main features that will help you to choose:

Figure 1-4: Some methods used to create circuit PCB for a given schematic circuit

- **Acid etching method**

This method requires extreme safety measure, the availability of many materials such as the etchant and it is somewhat slow. The quality of PCB obtained varies according to the materials you use but generally, it is a good method for simple to intermediate levels of complexity circuits. Circuits involving more close wiring and tiny wires usually use other methods.

Acid etching method specific steps

a. Choose your etching acid.

Ferric chloride is a common choice for an etchant. However, you can use Ammonium per sulfate crystals or other chemical solutions. No matter what choice for the chemical etchant, it will always be a dangerous material, so besides following the general safety precautions mentioned in this article, you should also read and follow any additional safety instructions that come with the etchant.

b. Draw the PCB layout.

For acid etching, you need to draw the circuit using an etchant resistant material. Special markers can be found easily for this specific purpose if you intend to do the drawing by hand (not appropriate for medium to large circuits). Laser printers' ink is the most commonly used material however. The steps to use laser printers for drawing the circuit layout are as follows:

i. Print the PCB layout on a glossy paper. You should ensure the circuit is mirrored before doing that (most PCB layout programs have this as an option when printing). This only works using a laser printer.

ii. Put the glossy side, with the printing on it, facing the copper.

iii. Iron the paper using an ordinary clothes iron. The amount of time this will take depends on the type of paper and ink used.

iv. Immerse the board and paper in hot water for a few minutes (up to 10 minutes). Remove the paper. If certain areas seem particularly difficult to peel off, you can try soaking a bit more. If everything went well, you will have a copper board with your PCB pads and signal lines traced out in black toner.

Figure 1-5: Different acids used for creating PCB

Figure 1-6: Laser printer for printing circuit layout drawing

Figure 1-7: Preparing the acid etchant to submerge PCB

c. Prepare the acid etchant. Depending on the acid etch that you choose, there might be additional instructions. For example, some crystallized acids require being dissolved in hot water, but other etchants are ready to use.

d. Submerge the board in the acid.

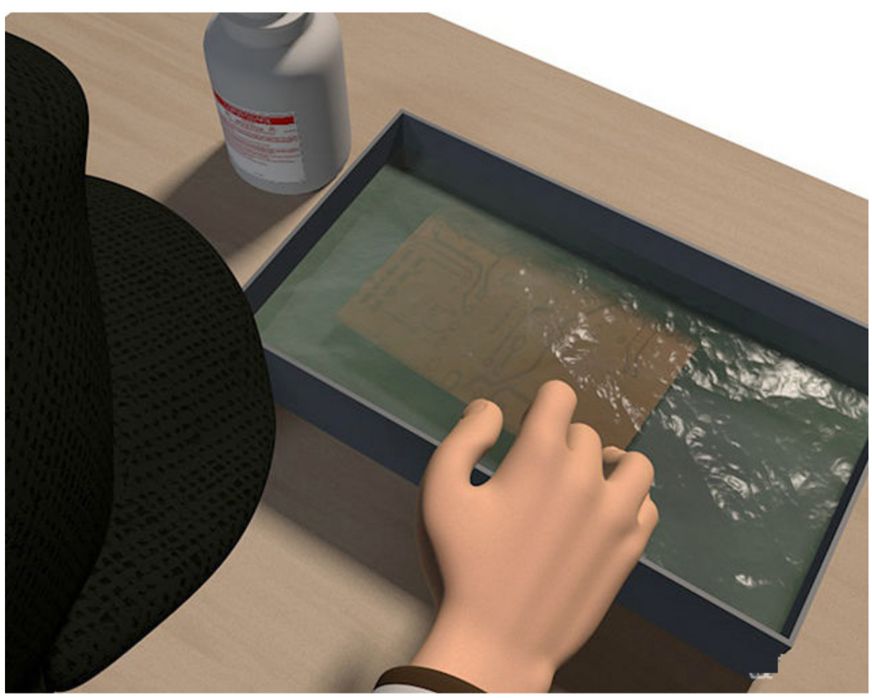


Figure 1-8: Submerge the PCB into the acid

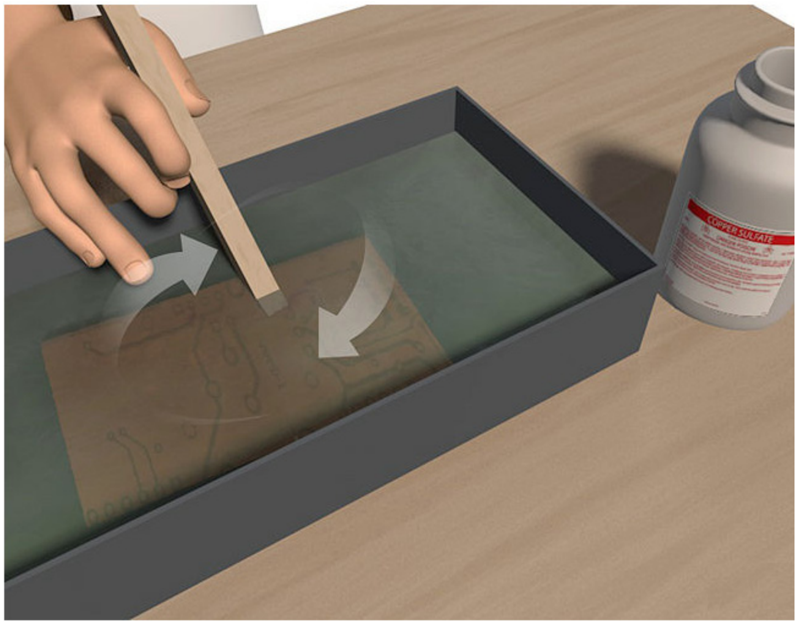


Figure 1-9: Mix the acid periodically once you have submerged the PCB

e. Make sure to stir every 3-5 minutes

f. Take the board out and wash it when all unnecessary copper is etched away from the board.

Figure 1-10: Washing the PCB after submerging in the acid

g. **Remove the insulating drawing material used.**

There are special solvents available for almost all types of insulating drawing material used in drawing PCB layouts. However, if you don't have access to any of these materials, you can always use a sand paper (a fine one).

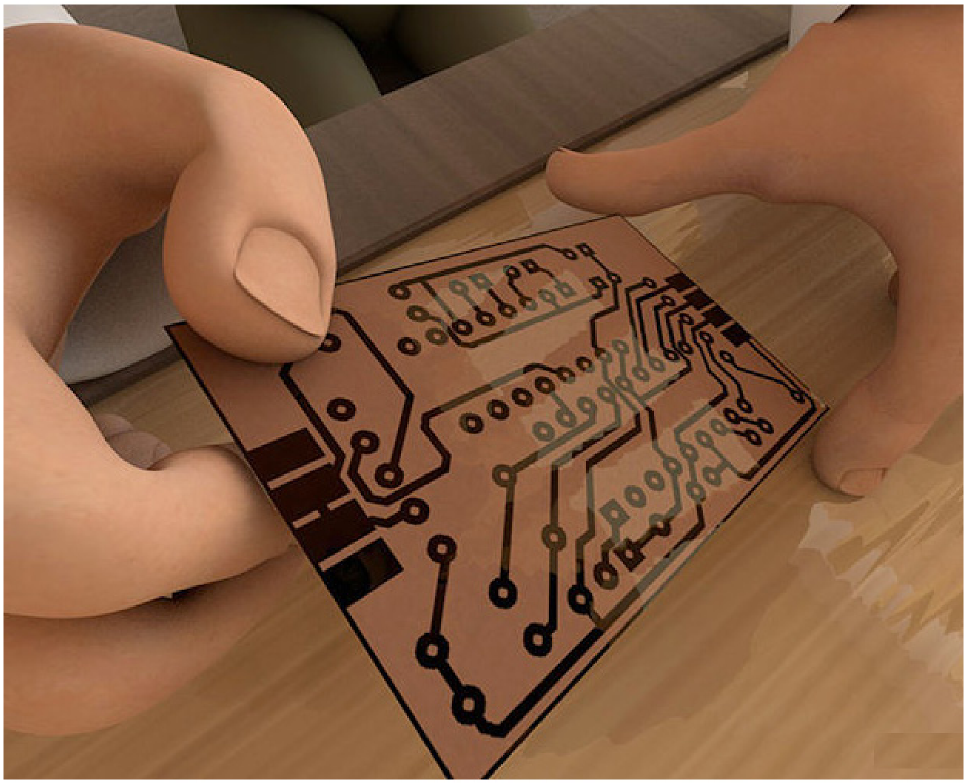


Figure 1-11: Removing the insulation of the drawing material

- **Ultra Violet etching method**

This method requires more expensive materials that might not be available everywhere. However, the steps are simple; it requires less safety measures and can produce finer and more complicated circuit layouts.

Ultra-Violet etching method specific steps

1. Draw the PCB layout on the special copper coated board.
2. Cover the board with a transparent sheet (optional)
3. Put the board in the UV etching machine/chamber
4. Turn on the UV machine for the specified amount of time depending on the specification of the board and machine.

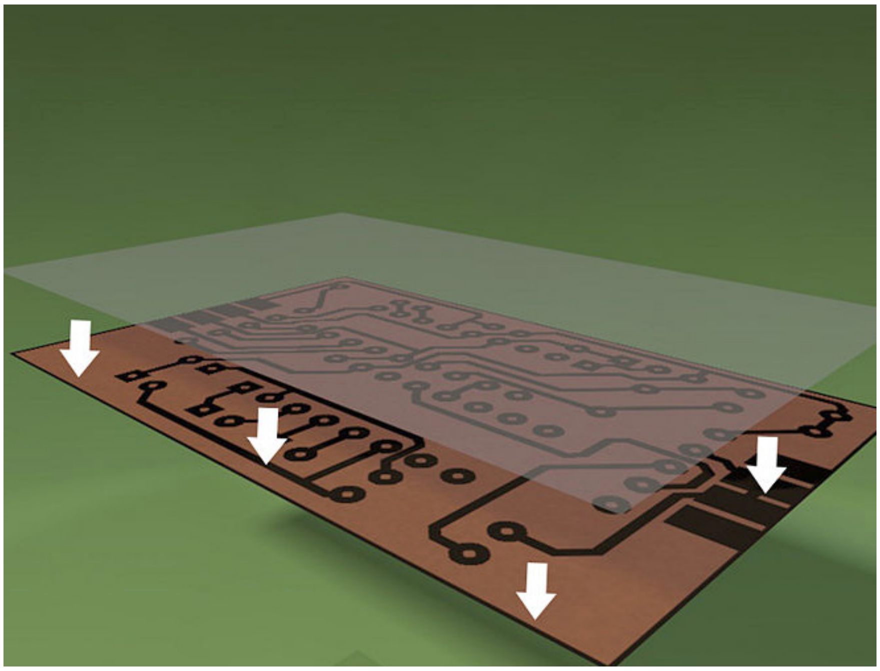


Figure 1-12: Drawing PCB layout on copper coated board

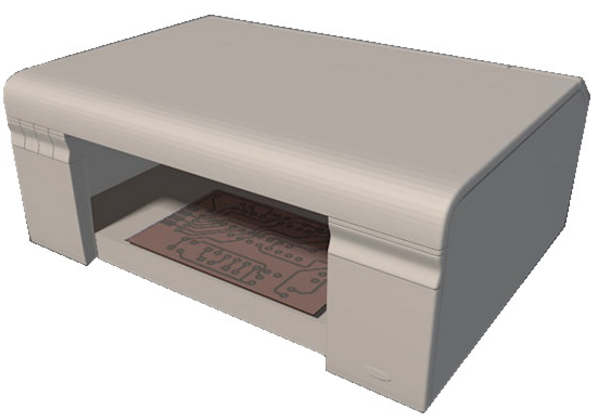


Figure 1-13: Putting the board in the ultra-violet etching machine

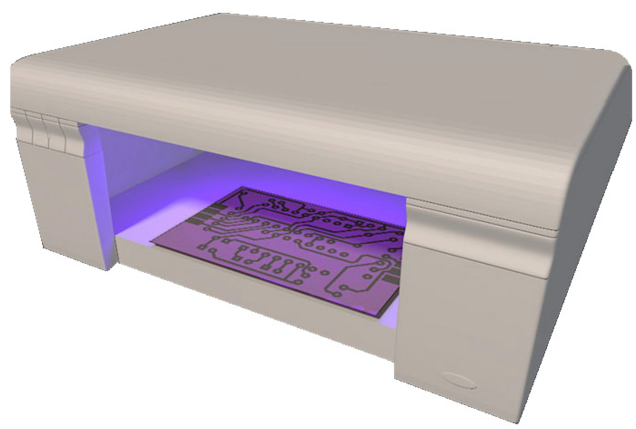


Figure 1-14: Turing on the ultraviolet machine and leave it a while with the board in

- **Mechanical etching/routing method**

This method requires special machines that will mechanically etch away unnecessary copper from the board or route empty separators between wires. It can be expensive if you intend to buy one of those machines and usually leasing them requires the availability of a workshop nearby. However, this method is good if you need to create many copies of the circuit and also can produce fine PCBs.

- **Laser etching method**

This is usually used by large production companies, but can be found on some universities and colleges. The concept is similar to mechanical etching but LASER beams are used to etch the board. It is usually hard to access such machines, but if your local company or university is one of the lucky ones having such machine, you can use their facilities if they allow it.

2. Create the PCB Layout of the circuit.

Once a circuit has been designed and any necessary alterations made, the completed circuit is ready for export to "real PCB". This is usually done by converting your circuit's schematic diagram into a PCB layout using PCB layout software. There are many open source software packages for PCB layout creation and design. Crocodile Technology (Yenka Electronics) is an example of software that allows the design and conversion of circuits to PCB, in an integrated environment.

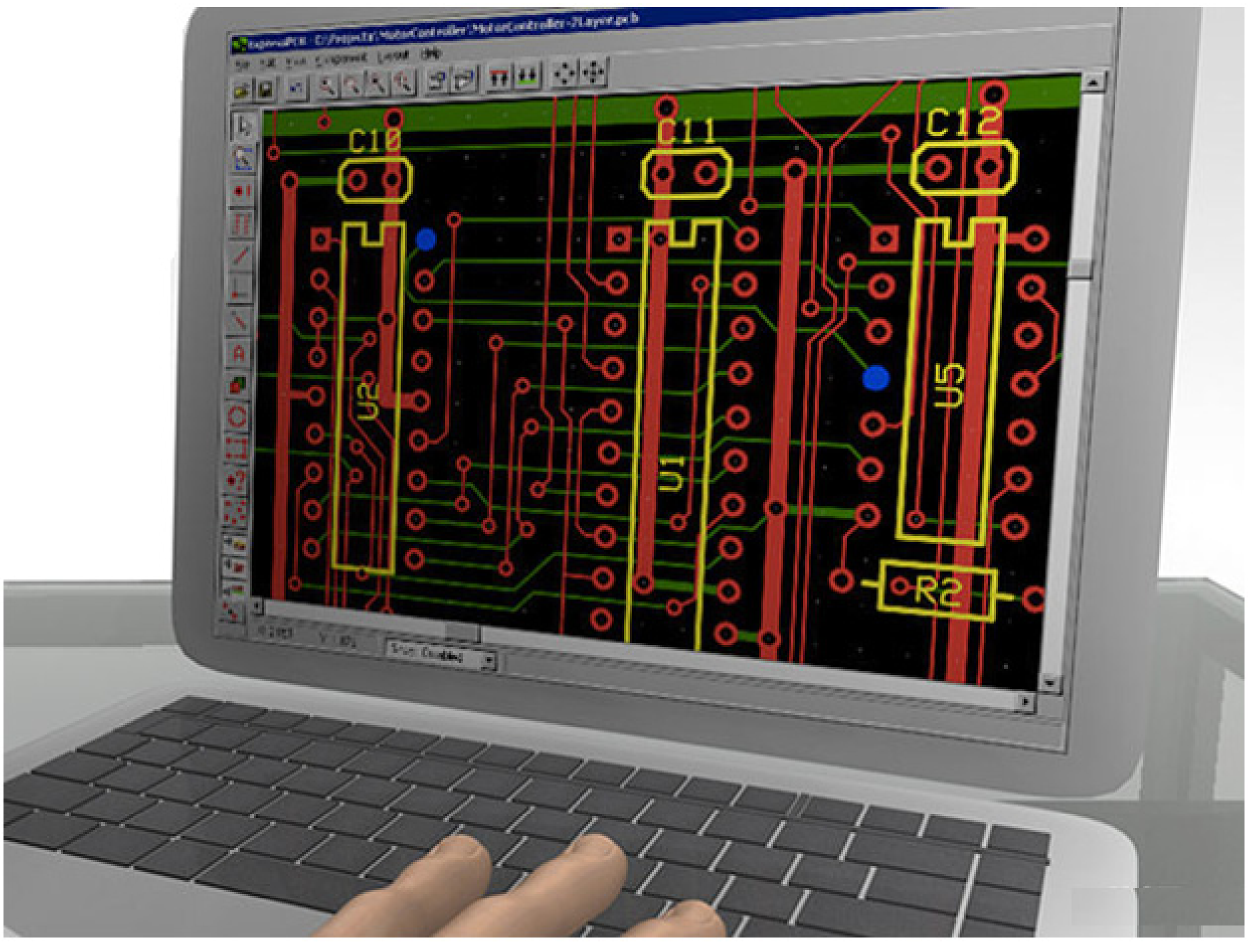


Figure 1-15: Creating circuit PCB layout using software

Advantages of using circuit and PCB software:

- Circuits can be simulated on screen without the need to build a circuit on a breadboard with real components.
- Errors can be corrected and alterations made to the circuit by simply replacing components from menus.
- No soldering is needed; simply move components with a mouse.
- Using simulation software is safer than soldering with hot soldering irons and avoids dangerous fumes being produced whilst soldering.

- Once a PCB has been designed it can be printed onto transparencies repeatedly, all to the same high quality.
- PCB layout software is more accurate than drawing a circuit by hand.

3. Make sure you have collected all the materials needed by the method of your choice.

Figure 1-16: Gathered materials for creating PCB

4. Draw the circuit layout on the copper coated board.

This is only applicable in the first two methods. More details can be found on the detail section of your method of choice.

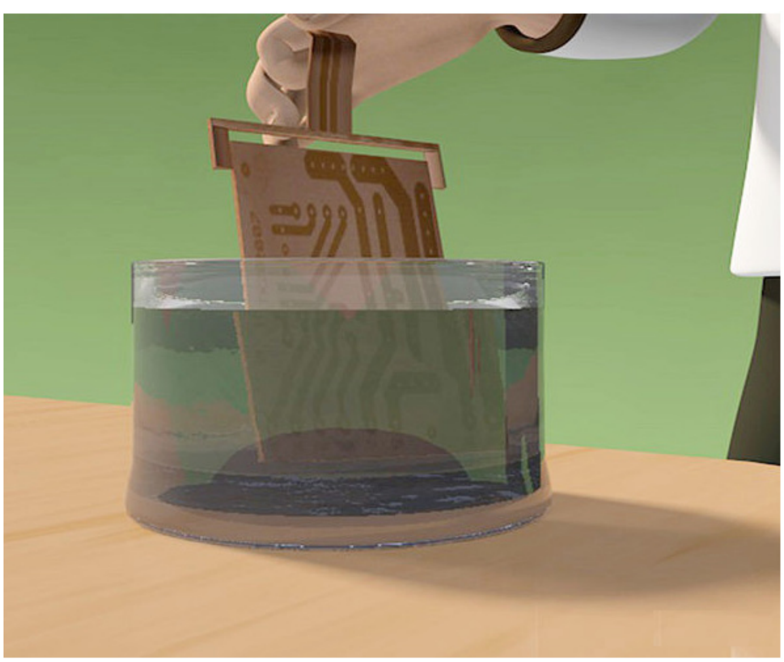


Figure 1-17: Submerging PCB into the acid to remove unwanted copper

5. Etch the board

Look for the details sections for how to etch the board**.** This process removes any unnecessary copper from the board leaving only wiring of the final circuit.

6. Drill mount points.

Drilling machines used for that are usually custom machines designed specifically for this purpose. However, with some adjustments, a usual drilling machine will do the job at home.

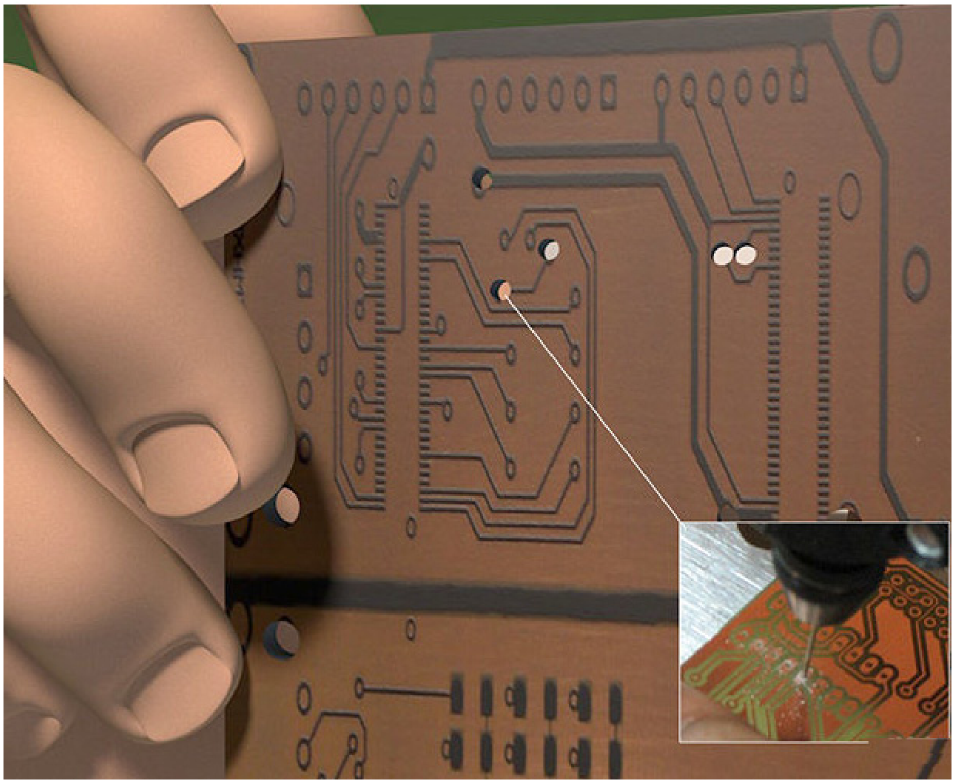


Figure 1-18: Drilling holes on PCB for component fixation

7. Mount and solder the electronic components on board.

Circuit Properties of the PCB

Each trace consists of a flat, narrow part of the copper foil that remains after etching. The resistance, determined by width and thickness, of the traces must be sufficiently low for the current the conductor will carry. Power and ground traces may need to be wider than signal traces. In a multi-layer board one entire layer may be mostly solid copper to act as a ground plane for shielding and power return. For microwave circuits, transmission lines can be laid out in the form of stripline and microstrip with carefully controlled dimensions to assure consistent impedance. In radio-frequency and fast switching circuits the inductance and capacitance of the printed circuit board conductors become significant circuit elements, usually undesired; but they can be used as a deliberate part of the circuit design, obviating the need for additional discrete components.

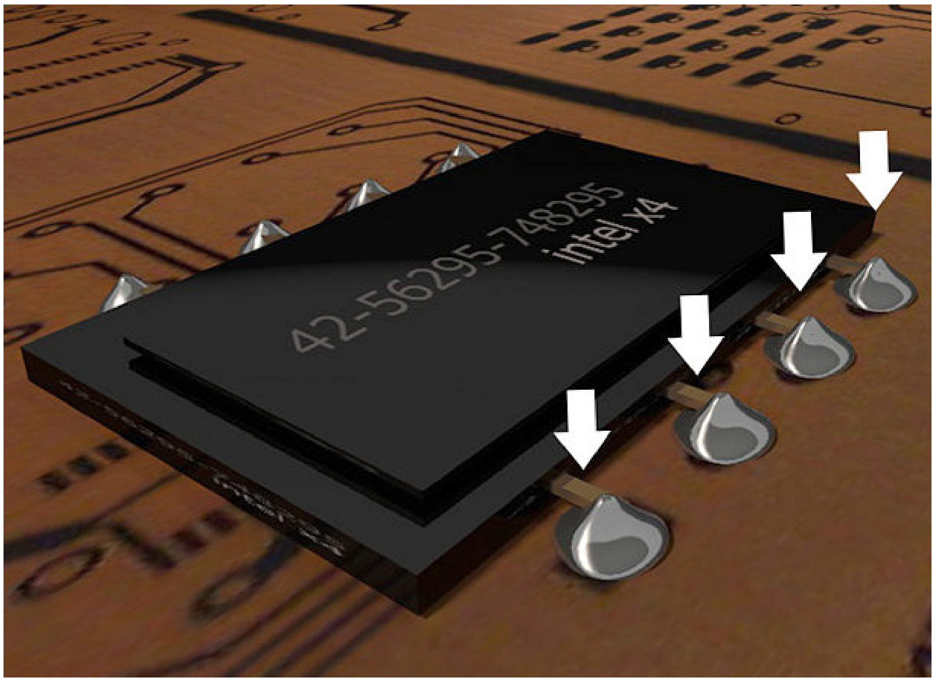


Figure 1-19: Mount and solder the electronic components on PCB

The Advantages of Printed Circuit Boards

There are many good reasons for using printed circuit boards instead of other interconnection wiring methods and component mounting techniques, some of which are as follows:

- The circuit board fabrication cost (PCB cost) is lower with mass quantity production
- Electronic circuit characteristics will be maintained without introducing parasite capacitance with a proper circuit board design.
- Electronic circuit characteristics will be maintained without introducing parasite capacitance with a proper circuit board design.
- PCB's offer uniformity of electrical characteristics from assembly to assembly.
- The location of electronic parts is fixed and so it simplifies components identification and maintenance of equipment.
- Inspection time is reduced because printed circuitry eliminates the probability of error.
- Chances of miswiring or short-circuited wiring are minimized.

1. 4. Testing Instruments

Whether designing new circuit or troubleshooting existing circuit or devices, you need electronic testing instruments to ensure proper operation. Generally, electronic test instrument is used to create signals and capture responses from electronic circuit or devices under test. In this way, the proper operation of the circuit under test can be proven or faults in the device can be marked out. Use of electronic test equipment is essential to any serious work on electronics systems. There are many types of electronic test instruments, in this section we are going to discuss on commonly used.

1. 4. 1. Multimeter

A multimeter is an electronic measuring instrument that combines several measurement functions in one unit and is also known as a multitester or VOM (Volt-Ohm meter). A typical multimeter would include basic features such as the ability to measure voltage, current, and resistance. Multimeter can be analog or digital. Digital multimeters are now far more common than analog ones.

Digital multimeters sometimes abbreviated (DMM or DVOM) display the measured value in numerals, and may also display a bar of a length proportional to the quantity being measured. They can be used to troubleshoot electrical problems in a wide array of industrial and household devices such as electronic equipment, motor controls, domestic appliances, power supplies, and wiring systems.

In order to measure the specific quantity, you need to set the knob to corresponding range or higher than what you are expecting to measure. If you are hesitating about the range then start with the lowest possible range. If the reading in the multimeter shows “1” then it means the value is overflow so you need to move knob to next higher value.

Continuity

Continuity is a physical property which represents the materials ability to conduct current through it with lowest resistance.

Experiment 1-1: Testing the continuity of electrical material

Continuity test will be helpful in many aspects such as verifying soldering joints and to debug the circuit for broken wires and pins. In fact most continuity testing is frequently used in circuit troubleshooting.

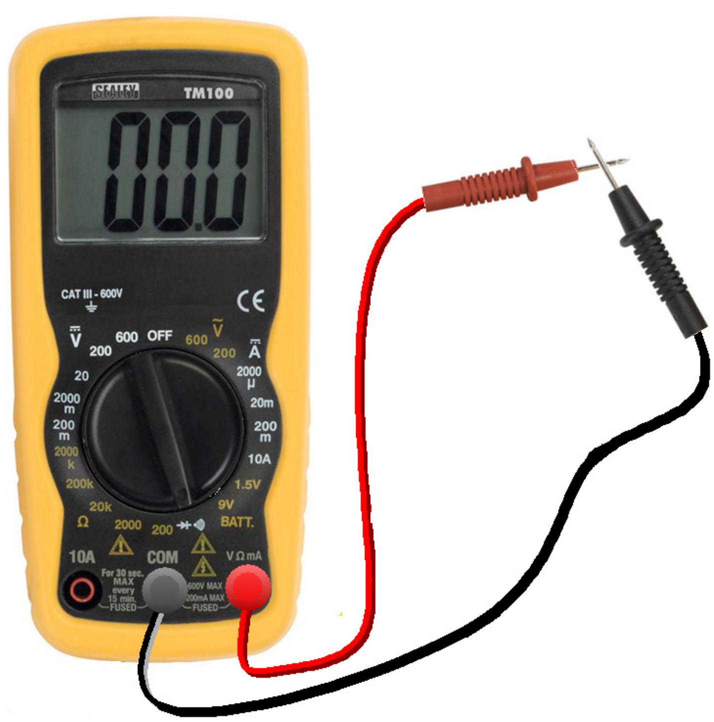


Figure 1-20: Testing continuity

Procedure:

1. Connect the Black probe to COM port (Common).
2. Connect the Red probe to V(mA) port (for measuring small current and voltage)
3. Set the position of knob to Continuity test mode “ ”.
4. Place the Black probe and red probe in two ends of the material that you wish to test the continuity.
5. Now if you hear beep sound then, the material you tested is a conductor and if it doesn’t makes beep sound then the material is not a conductor. Remember to use multimeter with fresh battery to produce sound beep.

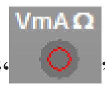
Ohmmeter

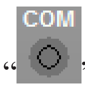
Ohmmeter is an electronic instrument used to measure electrical resistance. The property that allows material to oppose to the flow of current is called resistance while the specific device for such task is called resistor.

The ohmmeter quantifies how much a resistor restricts the follow of current in a given circuit.

Experiment 1-2: To measure the Resistance of a component

Procedure

1. Connect the red probe to VmAΩ jack “

” and black probe to commune jack “

”.
2. Set the knob to Resistance mode (start from the lowest range, that is from 200 ohms).
3. Now place the probes in two opposite ends of the material.
4. Now note down the Resistance value displayed in the LCD screen, if it becomes 1 then it infers that the value you set is overflowed so now set the knob to next higher resistance range.
5. Repeat the above step until you get the stable value.
6. Note that when you are measuring resistance the polarity does not matter and make sure you are using fresh battery for multimeter.

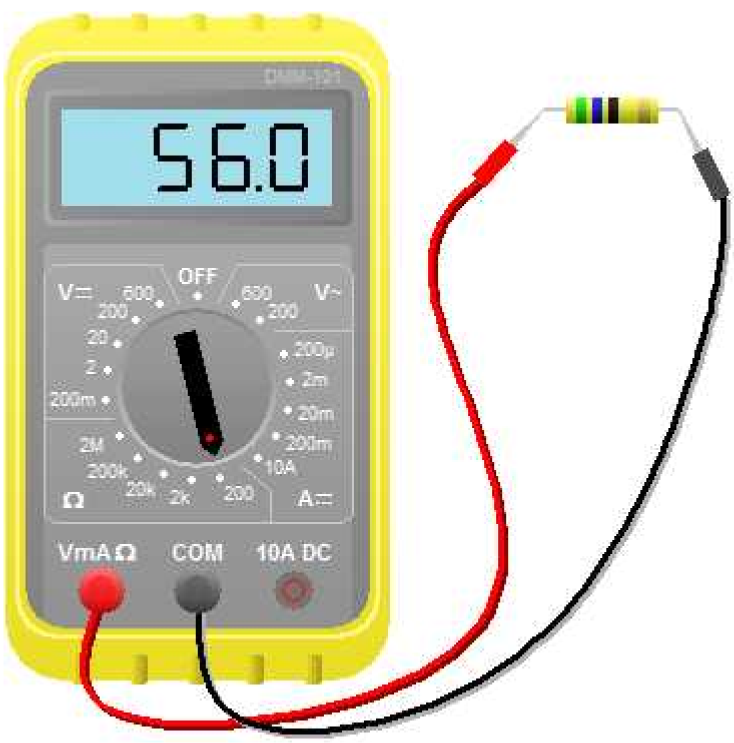


Figure 1-21: Example of multimter set in ohm’s range to measure resistance

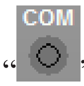
Voltmeter

Voltage is always measure in parallel within the circuit or direct to the terminals of the battery. In their design, voltmeter must have high internal resistance. Voltmeter with low internal resistance loads the circuit and affects the reading since the voltmeter internal resistance is considered to be in parallel with the circuit to be measured.

Experiment 1-3: To measure Voltage

Procedure

1. Connect the red probe to VmAΩ jack "

" and black probe to commune jack "

"
2. Set the knob to volts mode and start from the highest voltage scale with respect what you are expecting to measure.

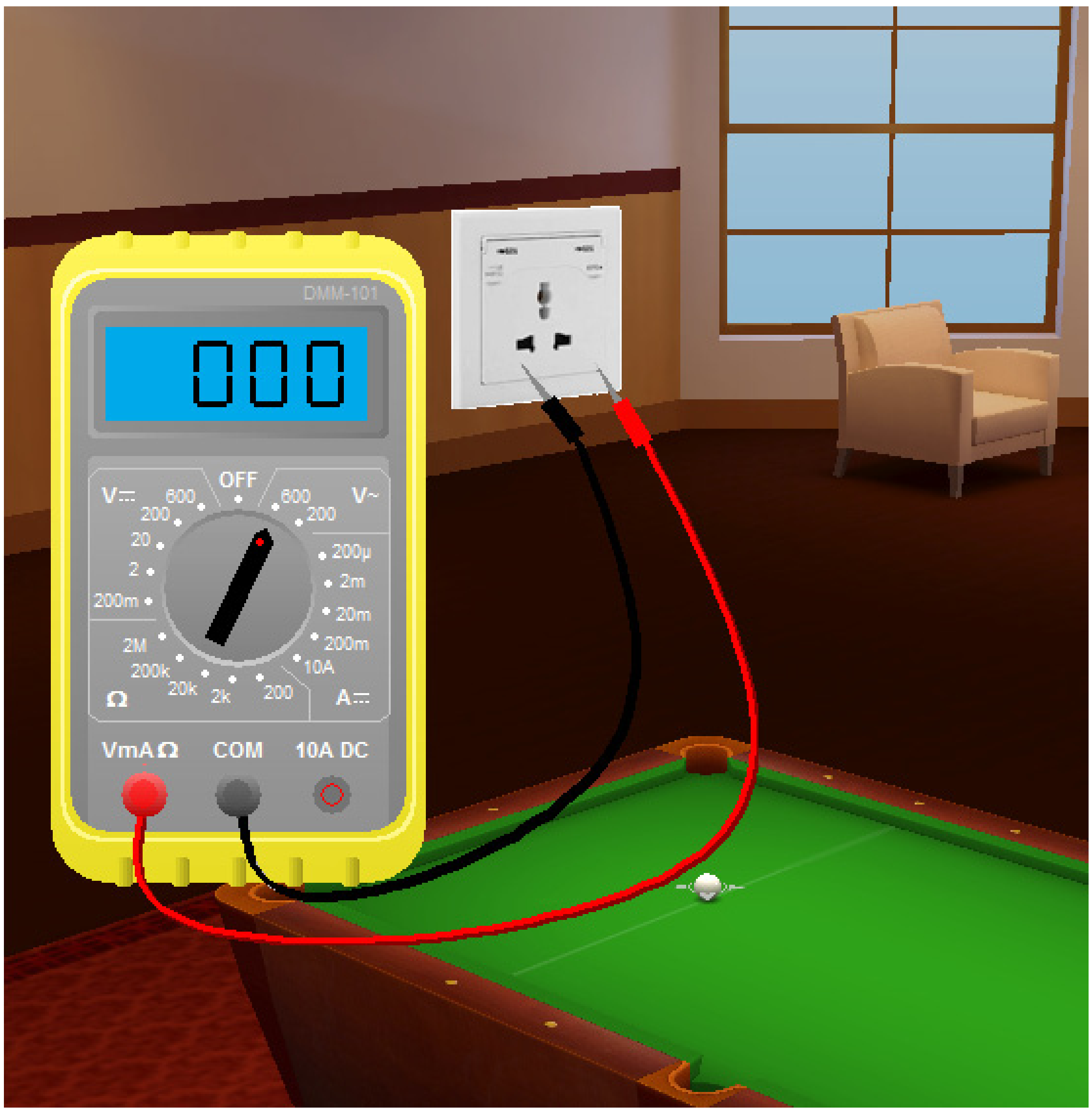


Figure 1-22: Multimeter set in volt mode to measure voltage from wall outlet socket

3. Set in DC volt settings "

" if you are measuring dc voltage.
4. Place the Black probe to negative end of the battery.
5. Place the Red probe to positive end of the battery.

6. Set in AC voltage mode “V~” if you are measuring ac voltage.
7. You can see the voltage value in the LCD screen if it becomes 1 then the voltage of the battery or ac source is above the range you set in step 2.
8. Repeat the above steps until you get the stable value.

Note that even if you are using a fresh battery the actual voltage you get may be deviated from the labeled voltage, this is due to various factors like battery internal resistance etc..

Ammeter

Ammeter is an electronic testing instrument used to measure the current. The ammeter has a very low internal resistance and needs to be connected in series within the circuit to be measured.

Experiment 1-4: To measure current

Procedure

1. Connect the red probe to VmAΩ jack “VmAΩ” for currents lower than 10A and to 10A DC “10A DC” for currents in range of 10A and black probe to commune jack “COM”.

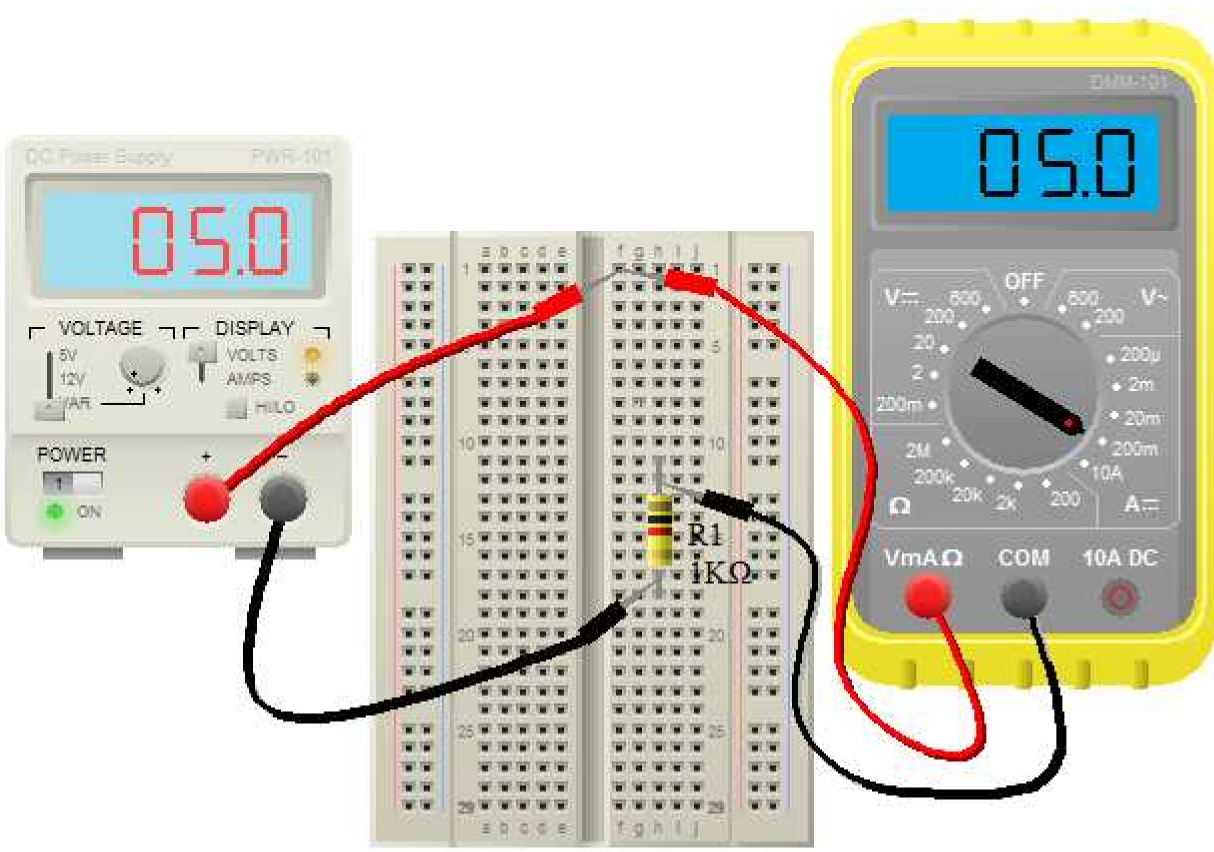


Figure 1-23: Multimeter set in amps range to meaure current across the resistor

2. Set the multimeter in amps function. Set in DC mode if you are measuring DC current, or in AC mode if you are measuring AC current.

3. Set the knob to current scale higher than the value what you are expecting to measure.
4. Place the Black probe to negative end of the battery.
5. Place the Red probe to positive end of the battery.
6. You can see the value of current in the LCD screen if it becomes 1 then the current delivered by the battery or AC source is above the range you set in step 3.
7. Repeat the above steps until you get the stable value.

1. 4. 2. Function Generator

A function generator is an electronic test instrument used to generate different types of electrical waveforms over a wide range of frequencies. The function generator provides pulse or square wave waveforms as well as sine waves, sawtooth waves and triangular waves. Function generators are used in the development, test and repair of electronic equipment. For example, they may be used as a signal source to test amplifiers, rectifiers or to introduce an error signal into a control loop. More advanced function generators are called arbitrary waveform generators.

Figure 1-24: Typical commercial function generator

A general-purpose function generator may have the following specifications:

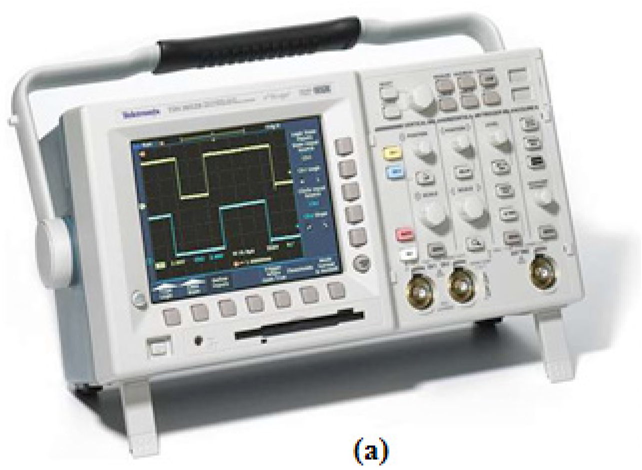
- Produces sine, square, triangular, sawtooth (ramp), and pulse output. Arbitrary waveform generators can produce waves of any shape.
- It can generate a wide range of frequencies. For example, the Tektronix FG 502 covers 0.1 Hz to 11 MHz.
- Frequency stability of 0.1 percent per hour for analog generators or 500 ppm for a digital generator.
- Maximum sine wave distortion of about 1% (accuracy of diode shaping network) for analog generators. Arbitrary waveform generators may have distortion less than -55 dB below 50 kHz and less than -40 dB above 50 kHz.
- Some function generators can be phase locked to an external signal source, which may be a frequency reference or another function generator.
- AM or FM modulation may be supported.

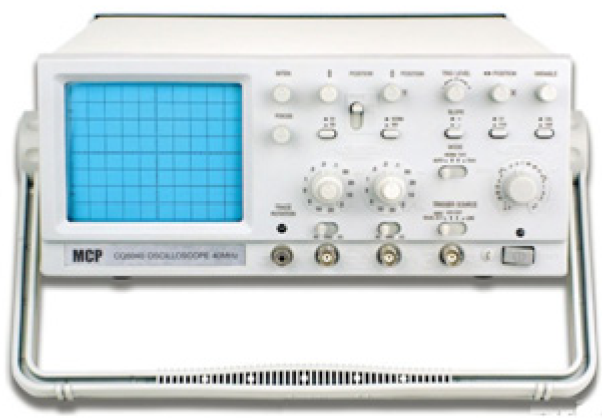
- Output amplitude up to 10 V peak-to-peak.
- Amplitude can be modified, usually by a calibrated attenuator with decade steps and continuous adjustment within each decade.
- Some generators provide a DC offset voltage, e.g. adjustable between -5V to +5V.
- An output impedance of 50 Ω.

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

1. 4. 3. The Oscilloscope

The oscilloscope, or (scope for short), is a widely used and versatile test instrument for observing and measuring waveforms.
The oscilloscope is basically a graph-displaying device that traces a graph of measured electrical signal on its screen. In most applications the graph shows how signal change over time. The vertical axis of the display screen represents voltage and the horizontal axis represents time. You can measure amplitude, period, and frequency of a signal using oscilloscope. Also, you can determine the pulse width, duty cycle, rise time and fall time of a pulse waveform. Most oscilloscopes can display at least two signals on the screen at one time, enabling you to observe their time relationship. A typical oscilloscope is shown in the figure 1-25.


(a)



(b)

Figure 1-25: Typical oscilloscope; (a) Digital oscilloscope, (b) analog oscilloscope

Two basic types of oscilloscopes, analog and digital, can be used to view digital waveforms. The digital oscilloscope is more widely used than the analog oscilloscope. However, either type can be used in many applications; each has characteristics that make it more suitable for certain situations. An analog oscilloscope displays waveform as they occur in "real time". Digital oscilloscopes are useful for measuring transient pulses that may occur randomly or only once. Also, because information about the measured waveform can be stored in a digital scope, it may be viewed at some later time, printed out, or thoroughly analyzed by a computer or other means.

Basic operation of Analog Oscilloscope: To measure a voltage, a probe must be connected from the scope to point in a circuit at which the voltage is present. Generally, a x10 probe is used that reduces (attenuates) the signal amplitude by ten. The signal goes through the probe into the vertical circuits where it is either further attenuated (reduced) or amplified depending on the actual amplitude and on where you set the vertical control of the scope. The vertical circuits then drive the vertical deflection plates of the CRT (cathode ray tube). Also, the signal goes to the trigger circuits that trigger the horizontal circuits to initiate repetitive horizontal sweeps of the electron beam across the screen using a sawtooth waveform. There are many sweeps per second so that the beam appears to form a solid line across the screen in the shape of the waveform.

Basic operation of Digital Oscilloscope: Some parts of a digital scope are similar to the analog scope. However, the digital scope is more complex than analog scope and typically has an LCD (Liquid Crystal Display) screen rather than CRT. Rather than displaying a waveform as it occurs, the digital scope first acquires the measured analog waveform and converts it to a digital processed. The data then goes to the reconstruction and display circuits for display in its original analog form.

Oscilloscope Controls

A front panel view of a typical 4-channel oscilloscope is shown in figure 1-26. Instruments vary depending on model and manufacturer, but most have certain common features. For example, the two vertical sections contain a Position control, a channel menu button, and volts/division control. The horizontal section contains a Sec/Div control. Some of the main controls are now discussed. Refer to the user manual for complete details of your particular scope.

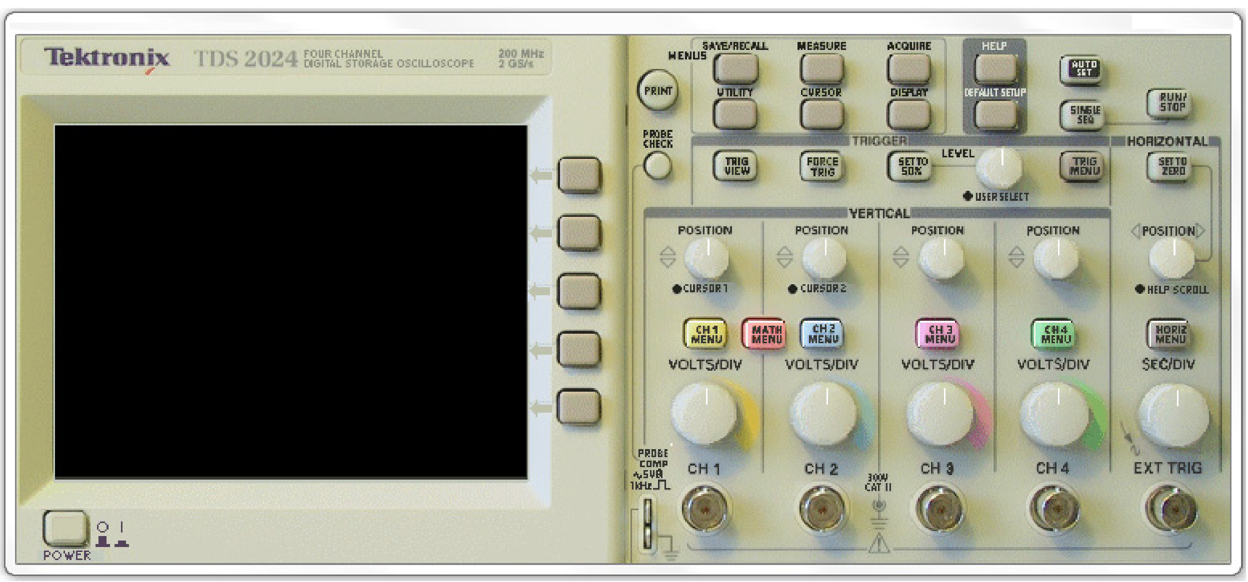


Figure 1-26: Oscilloscope front panel (type: Tektronix TDS 2024 200 MHz)

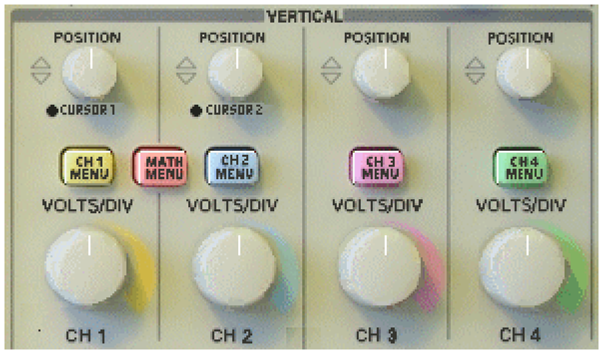
Vertical Controls

In vertical section of the scope there are identical controls for each of the four channels (CH1, CH2, CH3, and CH4). The position control let you move a displayed waveform up or down vertically on the screen. The Menu button "

" provides for the selection for several items that appear on the screen, such as the coupling modes (ac, dc, or ground) and coarse or fine adjustment for the Volts/Div.

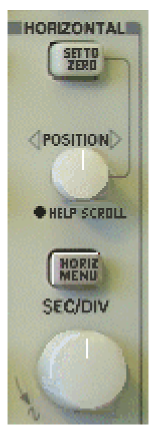
The Volts/Div control adjusts the number of volts represented by each vertical division on the screen. The Volts/Div setting for each channel is displayed on the bottom of the screen.

The Math menu button "" provides a selection of operations that can be performed on the input waveforms, such as subtraction and addition of signals.



Horizontal Controls

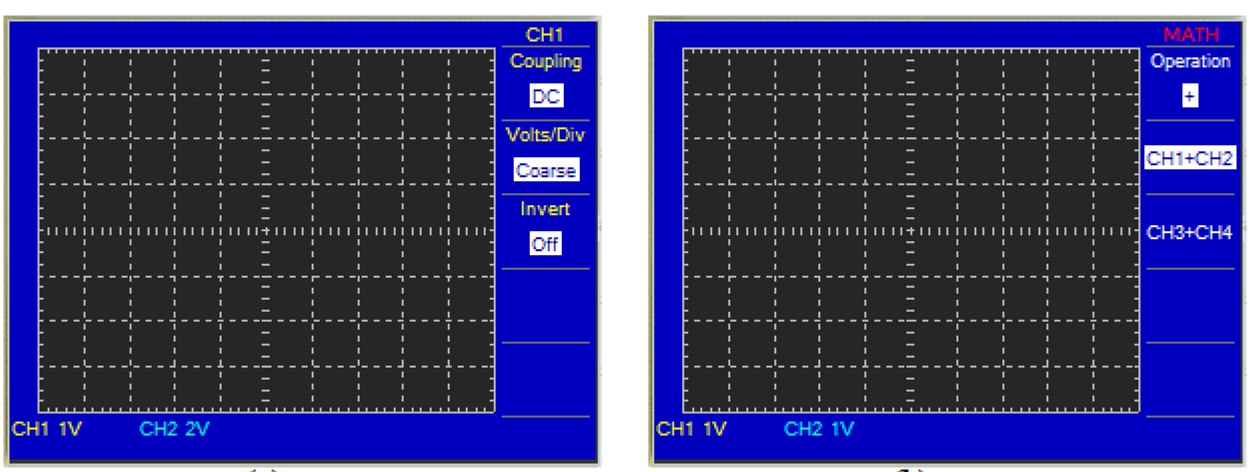
In the Horizontal section, the controls apply to both channels. The position control lets you move the displayed waveform left to right horizontally on the screen.



The menu button provides for the selection of the several items that appear on the screen such as the main time base, expended view of a portion of a waveform, and other parameters.

The Sec/Div control adjusts the time represented by each horizontal division or main time base.

The Sec/Div is displayed at the bottom of the screen.



(a) (b)

Figure 1-27: Scope screen showing examples of menu selection; (a) Example channel menu selection, (b) Example of math menu selection

Trigger Controls

In the trigger section, the Level control determines the point on the triggering waveform where triggering occurs to initiate the sweep to display input waveforms. The Menu button "" provides for the selection of several items that appear on the screen including edge or slope triggering, trigger source, trigger mode, and other parameters. There is also an input for an external trigger signal.

Triggering stabilizes a waveform on the screen and properly triggers on a pulse that occurs only on time or randomly. Also it allows to observe time delays between two waveforms. Figure 1-29 compares a triggered to an untriggered signal.

The untriggered signal tends to drift across the screen producing what appears to be multiple waveforms.

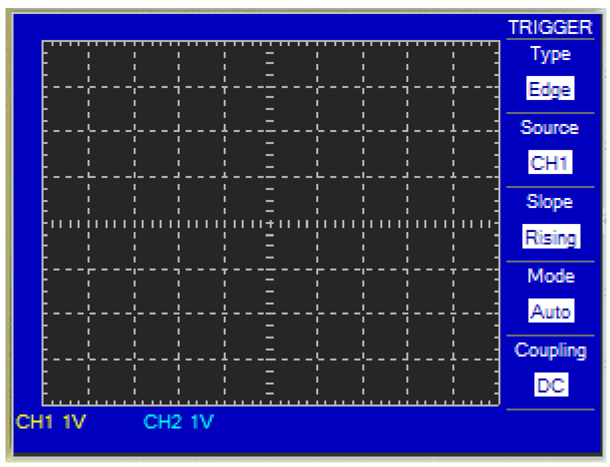
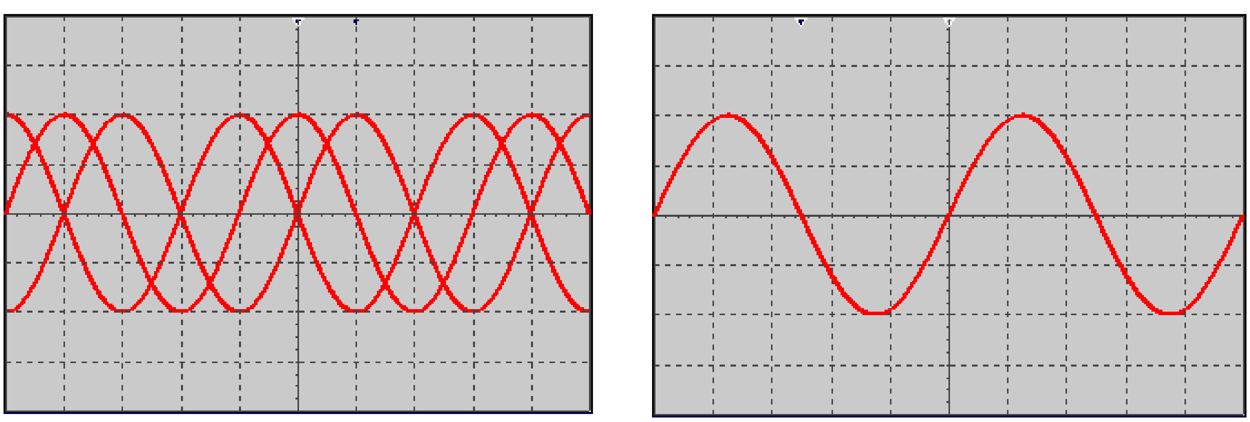


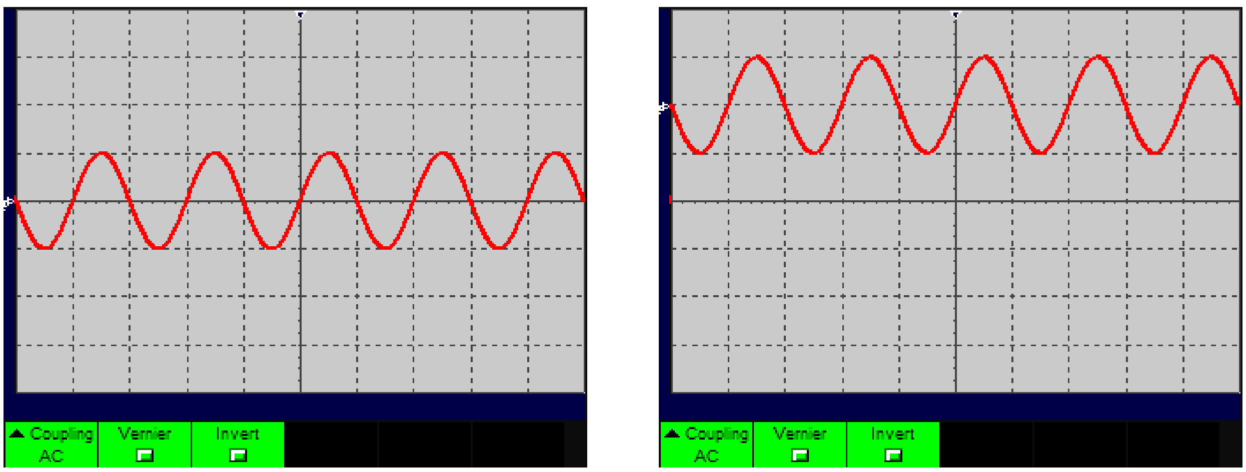
Figure 1-28: Example of trigger menu



(a) (b)

Figure 1-29: Comparison of an untriggered (a) and triggered (b) waveform on an oscilloscope

Coupling a Signal into the Scope: Coupling is the method used to connect a signal voltage to be measured into the oscilloscope. The DC and AC coupling modes are selected from the vertical menu. DC coupling allows a waveform including its dc components to be displayed. AC coupling blocks the dc components of a signal so that you see the waveform centered to 0V. the ground mode allows to connect the channel input to ground to see where the 0V reference is on the screen. Figure 1-30 illustrates the result of DC and AC coupling using a sinusoidal waveform that has a dc component.



(a) (b)

Figure 1-30: Displaying the same waveform having a dc component.
(a) AC coupled waveform; (b) DC coupled waveform

Oscilloscope Probes

Since all instruments tend to affect the circuit being measured due to loading, most oscilloscope probes proved an attenuation network to minimize loading effects. Probes that attenuate the measured signal by a factor of 10 are called x10 (times ten) probes.

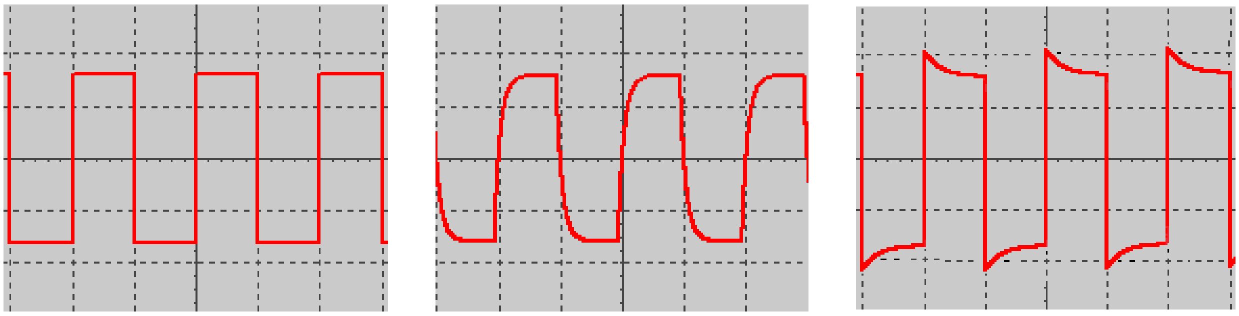
Figure 1-31: The oscilloscope probe

Probes with no attenuation are called x1 (times one) probes. The oscilloscope adjusts its calibration for the attenuation of the type of probe being used. For most measurements, the x10 probe should be used. However, if you are measuring very small signal, x1 may be the best choice.

The probe has an adjustment that allows you to compensate for the input capacitance of the oscilloscope. Most oscilloscopes have a probe compensation output that provides a calibrated square wave for probes compensation. Before making measurement, you should make sure that the probe is properly compensated to eliminate any distortion introduced. Typically, there is a screw or other means of adjustment compensation on probe.

The figure 1-31 shows oscilloscope waveforms for three probe conditions: properly compensated, undercompensated, and overcompensated. If the waveform appears either over- or under-compensated, adjust the probe until the properly compensated square wave is achieved.

Now you are ready to connect a probe to your oscilloscope. It is important to use a probe designed to work with your oscilloscope. A probe is more than a cable with a clip-on tip. It is a high-quality connector, carefully designed not to pick up stray radio and power line noise.



(a) **(b)** **(c)**

Figure 1-32: Probe compensation conditions. (a) Properly compensated, (b) Undercompensated, (c) Overcompensated

Probes are designed not to influence the behavior of the circuit you are testing. However, no measurement device can act as a perfectly invisible observer. The unintentional interaction of the probe and oscilloscope with the circuit being tested is called circuit loading. To minimize circuit loading, you will probably use a 10X attenuator (passive) probe.

1. 5. Testing Leads

Apart from the oscilloscope probes, there are other types of probes or leads used while testing circuit or devices. These are various leaded cables for attaching to multimeters, power supplies, oscilloscopes, function generators, etc. Testing and diagnosis of components and systems in electronic systems presents quite a few special challenges. Sometimes it can be very difficult to make contact with the exact point that you need. Most digital multi-meters (DMMs) are supplied with a set of test leads that are quite useful for a wide variety of testing situations. This type of probe will do quite well for checking the voltage of a power supply or checking a back-up battery. However, most of these probes are going to be too large to be used with confidence on a high-density board. They can easily slip and damage other components or cause a short. The commonly used test leads are described below.

Alligator Clip

An alligator clip (also called *crocodile clip* or *spring clip*) is a simple mechanical device for creating a temporary electrical connection, and is named for its resemblance to alligator's or crocodile's jaws. Alligator clips are toothed clips on the ends of electric wires. They are hinged near the back, making them look like alligator jaws.

Alligator clips are good for test connections to posts or bare wires. Alligator clip is also used to hold work pieces, holding a wire terminated with an alligator clip for electrical test.

Note the plastic boot surrounding the alligator clip, to make it less likely to short to other connections.

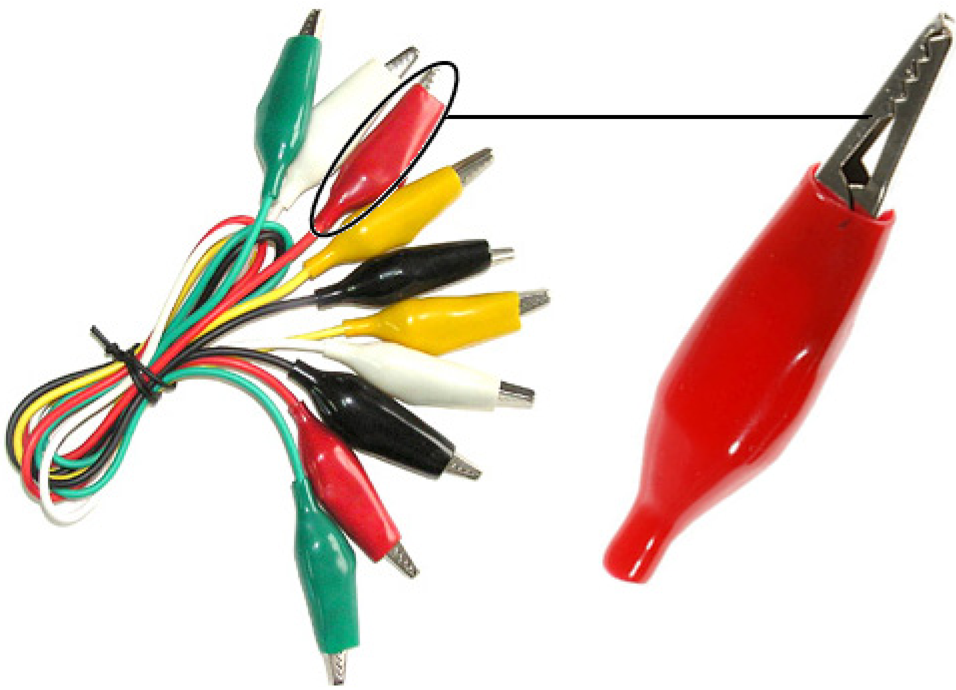


Figure 1-33: Alligator clip

IC Hook Test Leads

When alligator clips and even scope probes are too large, IC hooks test clips are the ideal solution for probing dozens of leads on an IC chip.

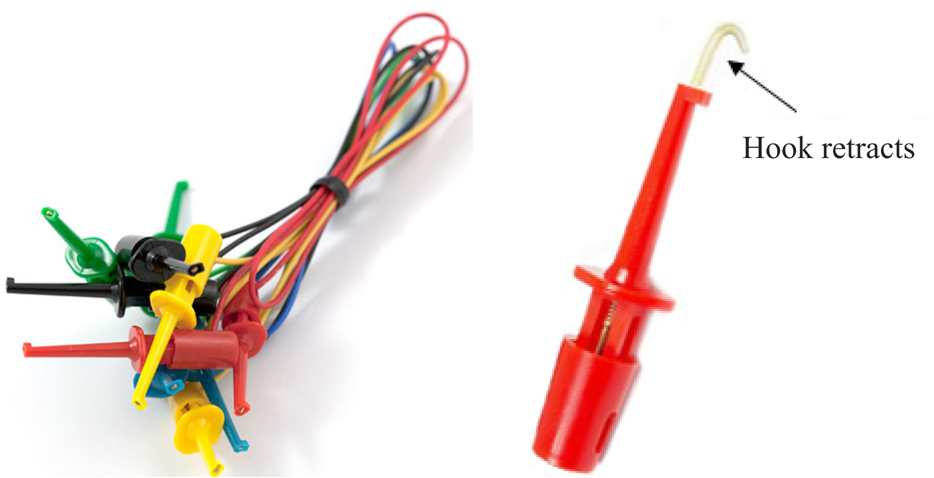


Figure 1-34: IC hook test leads

They also work very well for finding that one chip lead among many. When there are so many you can easily lose count part way down the side of the chip. With an IC test clip, you can reliably connect your probes to as many chip leads that you need to access without damaging the chip. As long as you do not need to attach to more than a few leads at a time, these clips may be the perfect answer to your probing needs. You need to press to extend hooks and depress to retract.

Banana Plug

A banana plug is the male half of a pair of mating connectors, known as a banana connector. A banana connector, made up of the banana plug and banana jack (commonly banana plug for the male, banana socket or banana jack for the female), is a single-wire (one conductor) electrical connector used for joining wires to equipment. A banana connector is commonly used to end patch cords for test equipment. Banana connectors can also be used in high fidelity sound systems, as the plugs on the cables that connect the amplifier to the speakers.

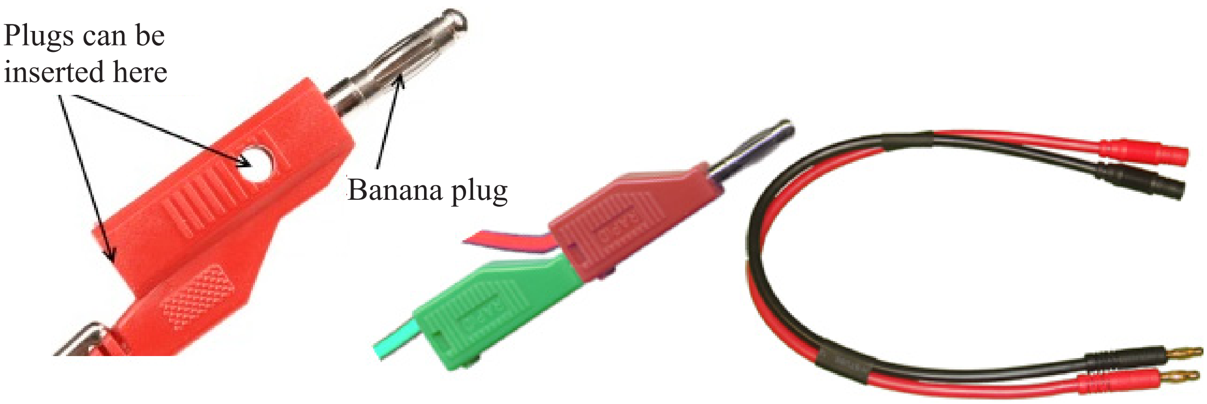


Figure 1-35: Banana clips

Note that one probe can combine two connectors such as banana plug and alligator clip, alligator clip and IC hook. There are also some probes and leads have a BNC (Bayonet Neill–Concelman or Baby N Connector) connector.

BNC is commonly used plug and socket for audio, video, oscilloscope and networking applications and provides a tight connection. Using a mount somewhat similar to the way a bayonet (knife) is mounted onto the end of a rifle, BNCs are used to connect a variety of different coaxial cable types. After the plug is inserted, it is turned, causing pins in the socket to be pinched into a locking groove on the plug.

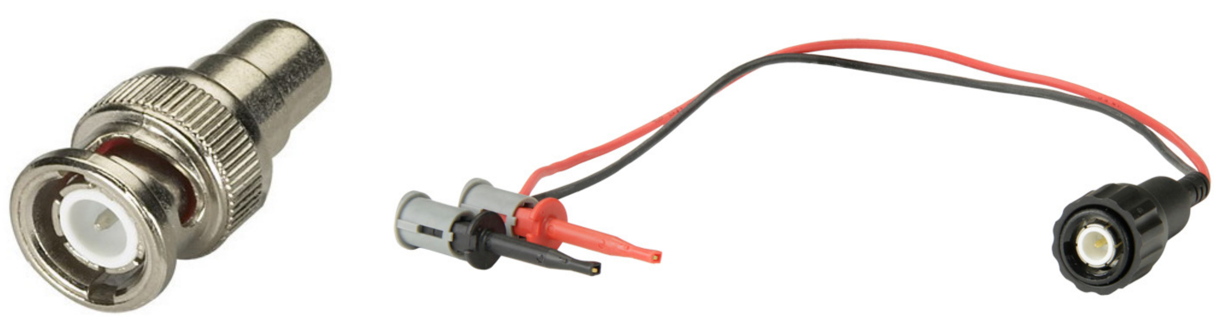


Figure 1-36: BNC connector with IC hook

Chapter two

DC Circuits and AC Circuits

Objectives

After completing this chapter, you should be able to

- Identify different types of DC voltage source
- Identify series and parallel circuit and translate a physical arrangement into a schematic diagram
- Describe some voltage-divider applications
- Describe the role of resistor, capacitor and inductor in a circuit
- Identify the behavior of capacitor and inductor in DC and AC circuits

Further reading
Study aids for this chapter are available at
Scherz, Paul (2006). *Practical Electronics for Inventors* (2nd ed.). McGraw Hill Professional.

Before any use of electricity or any of electronic machines or devices can be made, it must form part of electrical circuit. Even complex machines may be modeled by simple element that, when assembled into a circuit in the right way, can be analysed and so predict the machine's behavior. Accordingly, circuits are the foundation of any study of electrical or electronic engineering. We begin by defining simple circuit elements, then we shall incorporate them into circuits for analysis with the help of a number of laws and theorems. There are not many laws to remember; Ohm's law and Kirchhoff's laws are almost only ones, but from these a number of theorems have been deduced to assist in circuit analysis. In this chapter we shall primarily be concerned with direct current and voltage (DC for short), but the principles developed will serve for analyzing circuits behavior with alternating current and voltage (AC for short).

The five circuit elements considered initially are voltage and current sources and three passive elements: resistance, inductance and capacitance. Though simple these can be combined to form a powerful equivalent circuit. Passive components are presented to give you an idea of how simple resistor, capacitor, and inductor combination can be applied. The capacitor is a device that can store electrical charge, thereby creating an electric field which in turn stores energy. The measure of the charge-storing ability is called *capacitance*. The electrical component designed to have the property of inductance is called an *inductor* or *coil*.

In this chapter we will also discuss the behavior of each electronic passive component in DC circuits and also in AC circuits.

2. 1. DC current

If the current flows in only one direction it is called direct current (DC).

DC Voltage source

A voltage source provides electrical energy or electromotive force (emf), more commonly known as voltage. Voltage is produced by means of chemical energy, light energy, and magnetic energy combined with mechanical motion.

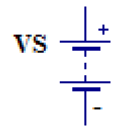


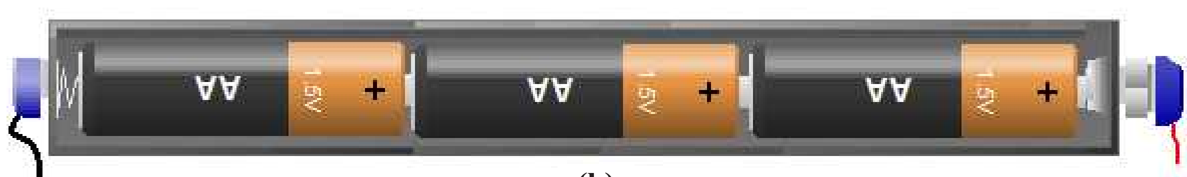
Figure 2-1: Symbol of DC voltage source

2. 1. 1. Types of DC Voltage Sources

1. **Batteries**

A battery is a type of DC source that converts chemical energy into electrical energy. A battery consists of one or more electromechanical cells that are electrically connected together internally. The materials used in battery cell determine the voltage that is produced. Although the voltage of a battery cell is fixed by its chemistry, the capacity is variable and depends on the quantity of materials in the cell. Also the way the cells are connected and the type of cells determine the voltage and capacity of the battery. If the positive is connected to the negative of the next and so on, the battery voltage is the sum of the individual cell voltage. This is called *series connection*.

(a)

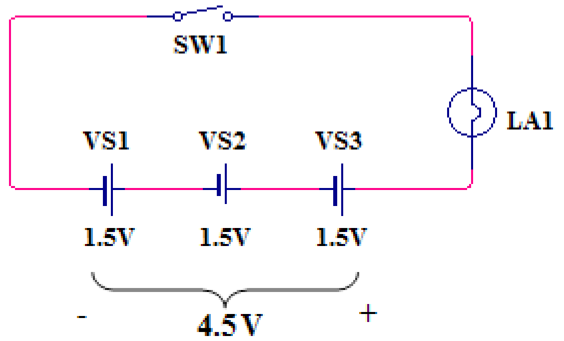


(b)

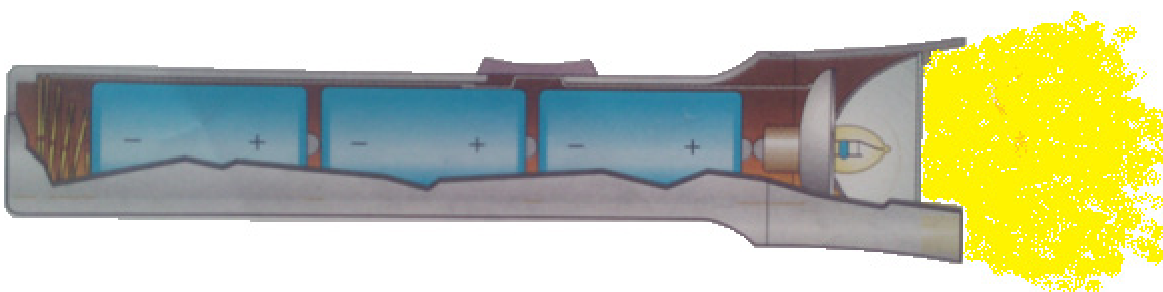
Figure 2-2: Series connected batteries; (a) Schematic circuit, (b) Real world circuit connection

When batteries are placed in a flashlight (torch), they are connected in *series-aiding* arrangement to provide a larger voltage as illustrated in flashlight example figure 2-3. In this example, three 1.5V batteries are placed in series to produce a total voltage $V_{s(total)}$.

$$V_{s(total)} = V_{S1}+V_{S2}+V_{S3}$$
$$= 1.5V+1.5V+1.5V$$
$$= 4.5V$$



(a)



(b)

Figure 2-3: Batteries connection of the torch in series-aiding; (a) Torch schematic circuit, (b) Torch cutaway view

Series voltage sources (batteries in this case) are added because their polarities are in the same direction, or *series-aiding* and are subtracted when their polarities are in opposite direction, or *series-opposing*. For example, if one of the batteries in the flashlight is turned around, as indicated in the schematic circuit figure 8-4, its voltage subtracts because it has a negative value and reduce voltage.

$$V_{s(total)}=V_{S1}-V_{S2}+V_{S3}=1.5V-1.5V+1.5V= 1.5V$$

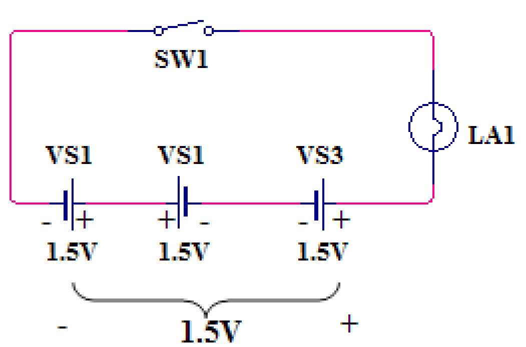
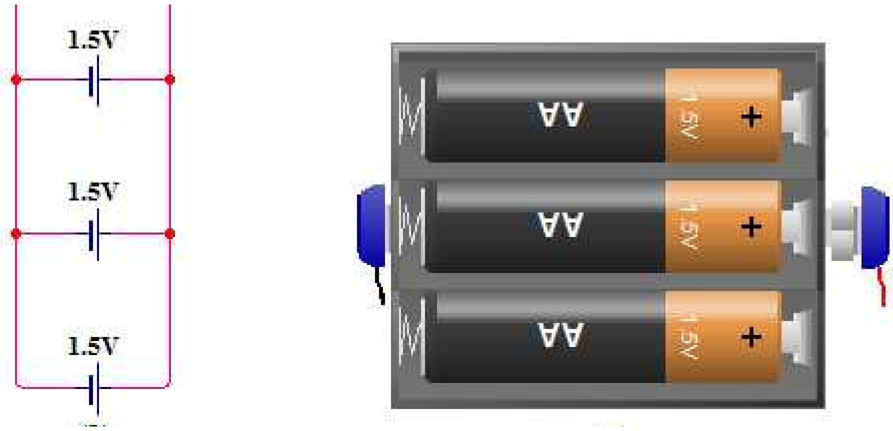


Figure 2-4: Batteries in series-opposing subtract

To increase battery capacity, the positive of several cells are connected together and all the negatives are connected together. This is called *parallel connection*.

Also, by using larger cells, which have a greater quantity of material, the ability to supply current can be increased but the voltage is not affected.



(a) (b)

Figure 2-5: Parallel connected batteries; (a) Schematic circuit, (b) Real world connection

Batteries are divided into two major classes, primary and secondary. Primary batteries are used once and discarded because of their chemical reactions are irreversible. Secondary batteries can be recharged and reused many times because they are characterized by reversible chemical reactions.

There are many types, shapes, and sizes of batteries. Some of the size that you are most familiar with are AAA, AA, C, D and 9V. There is also a less common size called AAAA, which is smaller than the AAA.

Figure 2-6: Example of Batteries

In addition to these common sizes, there are many others physical configurations for various applications from hearing aids to lanterns. Batteries for hearing aids, watches, and other miniature applications are usually in flat round configurations and are often called button batteries or coin batteries. Large multi-cell batteries are used in lanterns and industrial applications and, of course, there are familiar automotive batteries.

Ampere-hour ratings of batteries

Because of their limited source of chemical energy, batteries have a certain capacity that limits the amount of time over which they can produce a given power level. This capacity is measured in ampere-hours (Ah). The *ampere-hour rating* determines the length of time that a battery can deliver a certain amount of average current to a load at the rated voltage.

A rating of one ampere-hour means that a battery can deliver an average of one ampere of current to a load for one hour at the rated voltage output. This same battery can deliver an average of two amperes for one-half hour. The more current the battery is required to deliver; the shorter the life of the battery. In practice, a battery usually is rated for a specified current level and output voltage. For example, a 12V automobile battery may be rated for 70Ah at 3.5A. This means that it can deliver an average of 3.5A for 20h at the rated voltage.

2. **Solar cell**

The operation of solar cells is based on the *photovoltaic effect*, which is the process whereby light energy is converted directly into electrical energy. A basic solar cell consists of two layers of different types of semi-conductive materials joined together to form a junction. When one layer is exposed to light, many electrons acquire enough energy to break away from their parent atoms and cross the junction. This process forms negative ions on one side of the junction and positive ions on the other, and thus a potential difference (voltage) is developed.

3. **Generator**

Electrical generators convert mechanical energy into electrical energy using a principle called *electromechanical induction*. A conductor is rotated through a magnetic field, and a voltage is produced across the conductor.

4. **The electronic power supply**

Electronic power supplies convert the ac voltage from the wall outlet to a constant (dc) voltage that is available across two terminals, as indicated in the figure 2-7.

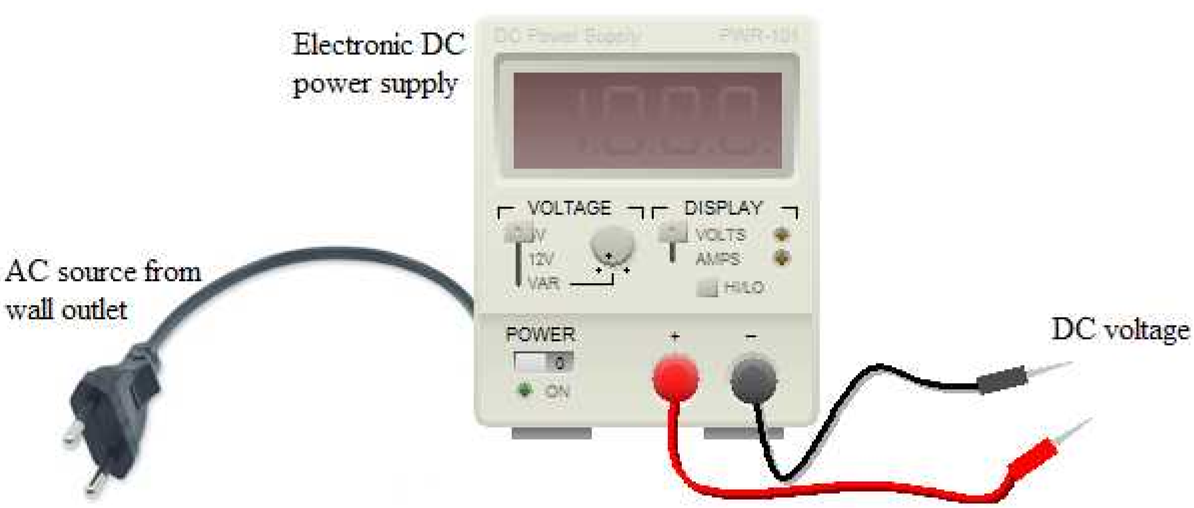


Figure 2-7: Typical commercial electronic DC power supply.

5. **Thermocouple**

The thermocouple is thermoelectric type of voltage source that is commonly used to sense temperature. A thermocouple is formed by a junction of two dissimilar materials, and its operation is based on the *Seebeck effect* that describes the voltage generated at the junction of the metals as a function of temperature.

6. **Piezoelectric sensors**

These sensors act as voltage sources and are based on the *piezoelectric effect* where a voltage is generated when a piezoelectric material is mechanically deformed by an external force. Quartz and ceramic are two types of piezoelectric material. Piezoelectric sensors are used in applications such as pressure sensors, force sensors, accelerometers, microphones, ultrasonic devices, and many others.

2. 2. AC Current

Alternating current (AC) is a type of current which change direction in function of time. The sinusoidal waveform or sine wave is the fundamental type of alternating current (AC) and alternating voltage. It is also referred to as a sinusoidal wave, or, simply, sinusoidal. The electrical service provided by the power companies is in the form of the sinusoidal voltage and current. In addition, other types of repetitive waveform are composites of many individual sine waves called harmonics.

Sine waves, or sinusoids, are produced by two types of sources: rotating electrical machines (AC generators) or electronics oscillator circuits, which are used in instruments commonly known as electronic signal generators or function generators. The figure 2-8 shows the symbol used to present a source of sinusoidal voltage source.

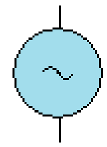


Figure 2-8: Symbol for a sinusoidal voltage source.

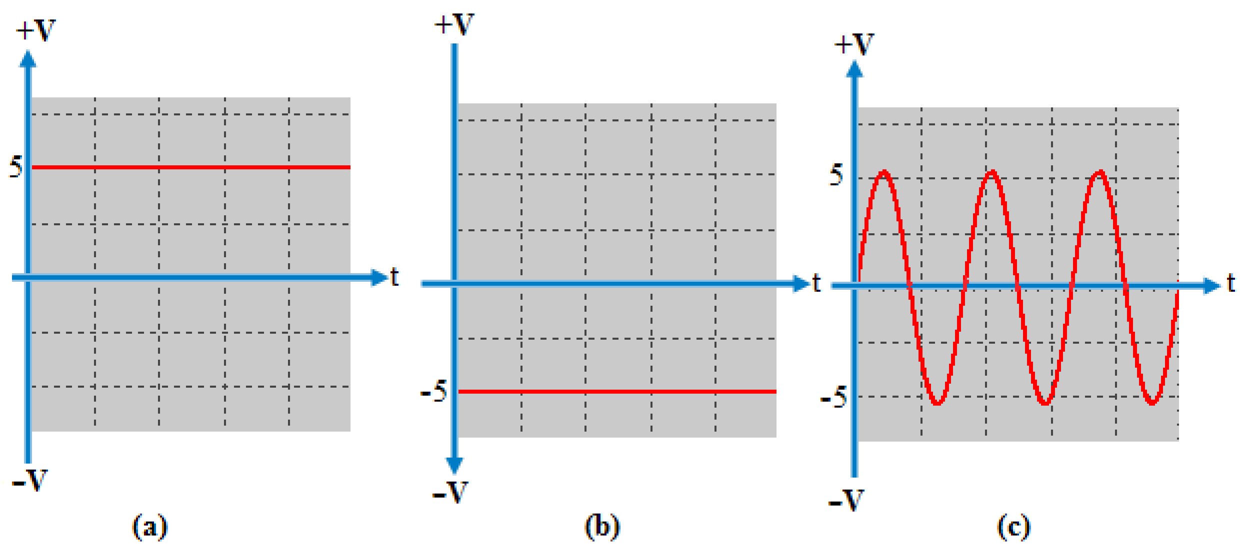


Figure 2-9: DC voltage and AC voltage shapes; (a) Positive DC voltage, (b) Negative DC voltage, (c) AC voltage oscillates both in positive and negative direction in function of time

OHM's Law

Ohm's law describes mathematically how voltage, current, and resistance in a circuit are related. Ohm' law can be written in three equivalent forms; the formula you use depends on the quantity you need to determine.

Ohm's Law states that current is directly proportional to voltage and inversely proportional to resistance.

$$I = V/R \qquad \text{[Equation 2-1]}$$

Where:

I = current in amperes (A)

V = voltage in volts (V)

R = resistance in ohms (Ω)

2. 3. Resistors

Components that are specifically designed to have a certain amount of resistance are called *resistors.* The principle applications of resistors are to limit the current, divider voltage, and, in circuit case, generate heat. Although different types of resistors come in many shapes and sizes, they can be placed in one of two main categories: *fixed* and *variable*.

Fixed resistors

Fixed resistors are available with large selection of resistance values that are set during manufacturing and cannot be changed easily.

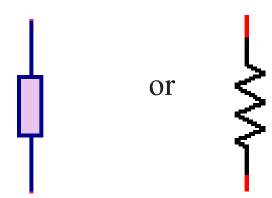


Figure 2-10: Symbols of resistor

2. 3. 1. Resistor in DC Circuits

An electrical circuit may include many resistors. The way these resistors impact the circuit depends on the way they are arranged in the circuit. Resistors may be arranged in series or in parallel with the voltage supply source.

Experiment 2-1: Demonstrate the relationship between voltage, current, and resistance based on Ohm's law

Ohm determined experimentally that if the voltage across a resistor is increased, the current through the resistor will increase; and, likewise, if the voltage is decreased, the current will decrease. For example, if the voltage is doubled, the current will double. Is the voltage is halved; the current will also be halved. This relationship is illustrated in the circuits of figure 2-11 through figure 2-14.

Parts and materials

No	Item	Specification	Quantity
1	Voltmeter and Ammeter	Digital multimeter	2
2	2V-10V Variable DC source	DC power supply	1
3	Breadboard	Prototyping board	1
4	Connecting wire	22-gaude (0.33mm^2) solid wire	20cm
5	1KΩ fixed resistor	1KΩ fixed resistor, ½ watt	2

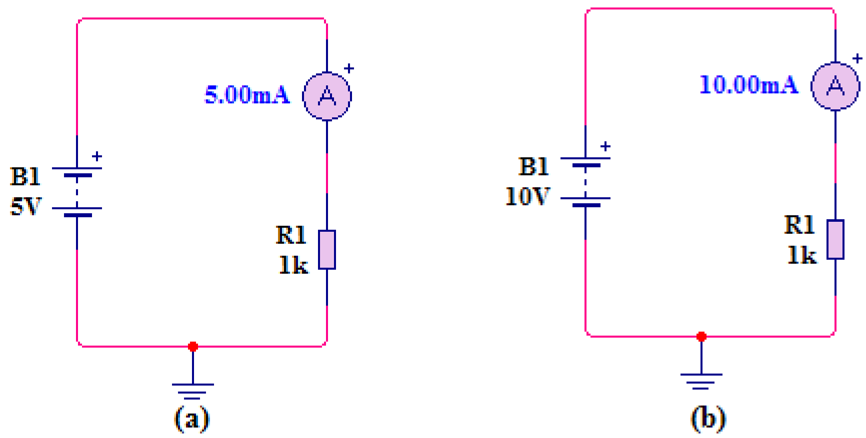


Figure 2-11: Effect on the current of changing of voltage with the resistance at a constant value; (a) Less *V*, less *I*; (b) More *V*, more *I*.

chap 2

Procedure

1. Connect the circuit as illustrated in pictorial view figure 2-12.
2. Set multimeter in amps range, in 200mA scale. Whenever you calibrate in highest or lowest scale with respect to the value you are expecting to measure, the ammeter will display a wrong value.
3. Connect the positive lead of ammeter to the positive terminal of the dc power supply and the negative lead "COM" to one terminal of a resistor R_1. The other terminal of resistor R_1 must be connected to negative terminal of the dc power supply.
4. With power supply set in variable settings, make it to output 5V and record the ammeter reading. This should be 5mA.
5. Allow the voltage to vary from 5V up to 10V with this by observing the ammeter reading. You may find out that as you increase the voltage, the current also increases with 10mA ammeter reading at 10V.

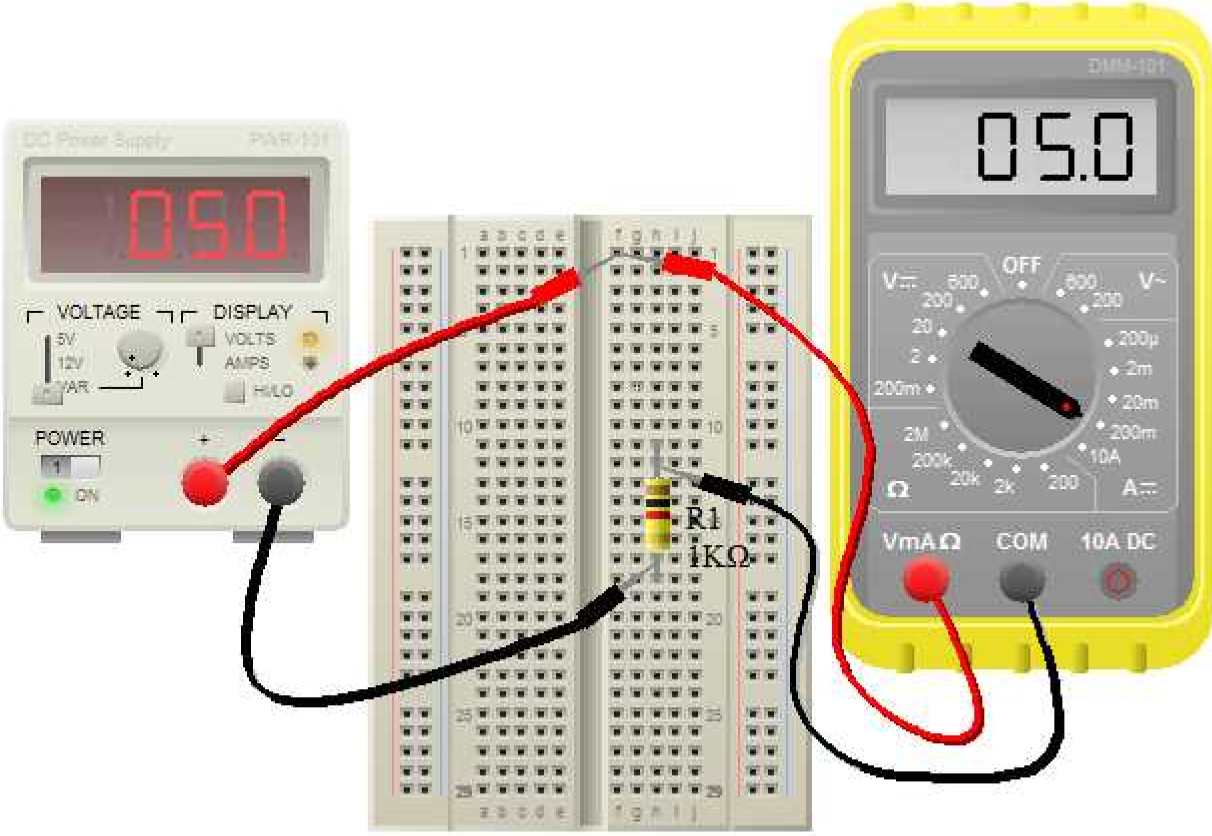


Figure 2-12: Effect on the current by changing voltage with resistance kept at a constant value

Ohm also determined that if the voltage is kept constant, less resistance results in more current, and more resistance results in less current. For example, if the resistance is halved, the current doubles. If the resistance is doubled, the current is halved. This concept is illustrated by the meter indication in the circuits of figure 2-13 where the resistance is increases and the voltage is held constant.

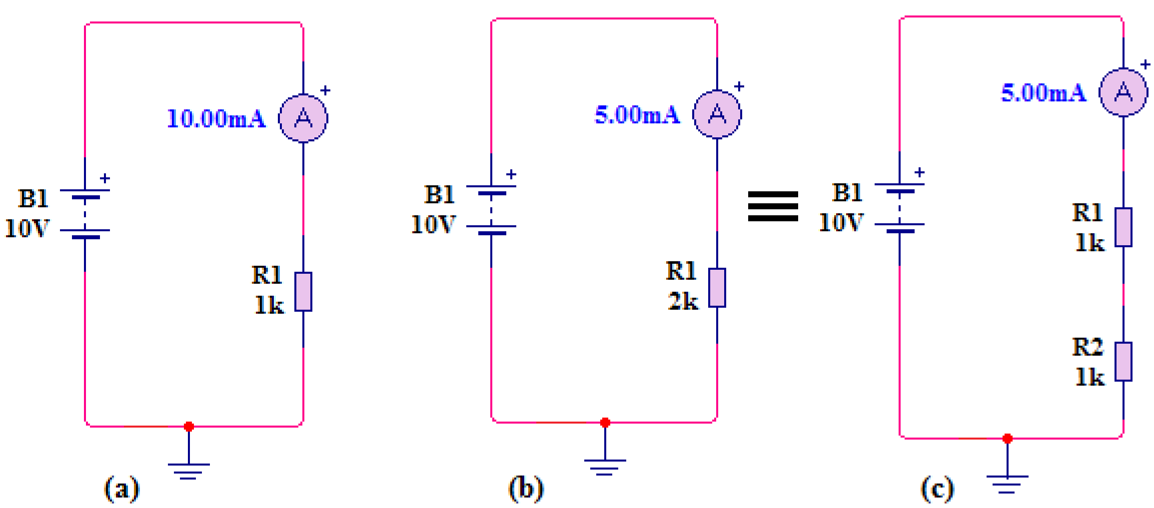


Figure 2-13: Effect on current of changing the resistance with the voltage at a constant value. (a) Less *R*, more *I*; (b) and (c) More *R*, less *I*.

Procedure

1. Connect the circuit as illustrated in pictorial view figure 2-14.
2. Set multimeter in amps range, in 200mA scale. Higher or lower scale with respect to the value you are expecting to measure lead the ammeter to display a wrong value.
3. Connect the positive lead of ammeter to the positive terminal of the dc power supply and the negative lead "COM" to one terminal of a resistor R_1. The other terminal of resistor R_1 must be connected to negative terminal of the dc power supply. Interchanging the lead make the ammeter to read value with a negative sign.
4. Set dc power supply to output 10V and make sure this voltage is held constant all along the measuring process. The ammeter reading should be 10mA.
5. Replace a 1KΩ resistor by 2KΩ resistor and record the ammeter reading again. This should be 5mA. As the 2KΩ resistor sometimes is awkward to get from resistors packet, this value is gotten by making two 1KΩ series-resistors connection; at this time move only the negative terminal of the dc power supply between resistor R_1 (1) and ending terminal of resistor R_2 (2).

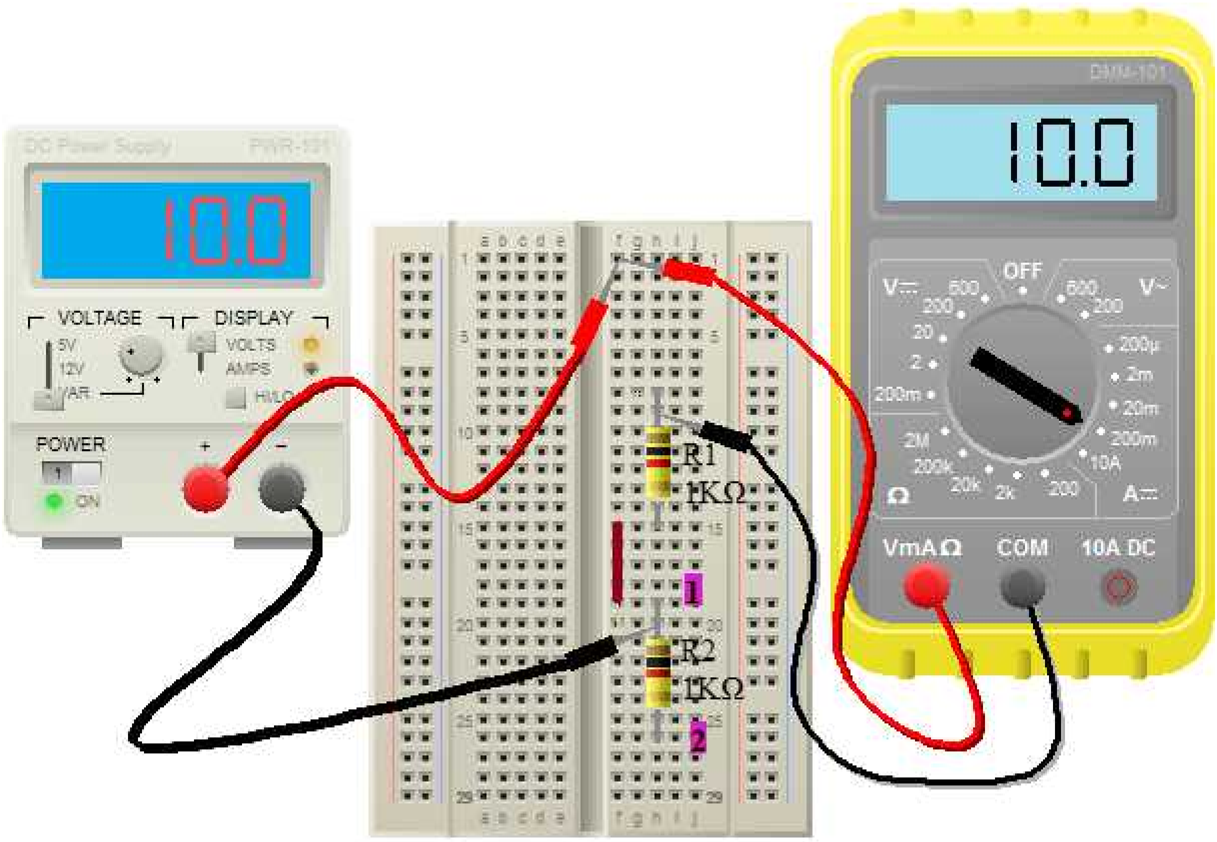


Figure 2-14: Effect on current of changing the resistance with the voltage at a constant value.

Resistors in series

When connected in series, resistors form a "string" in which there is only one path for current. When a voltage source is connected from point A to point B, the only way for current to get from one point to the other in any of the connections is to go through each of the resistors. A series circuit is defined as follows: *A series circuit provides only one path for current between two points so that the current is the same through each resistor.*

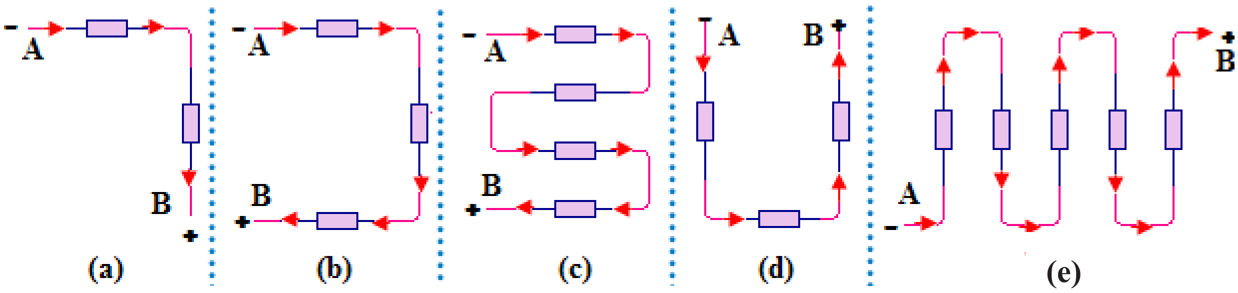


Figure 2-15: Some examples of series connections of resistors. Notice that the current must be the same at all points because the current has on one path, from A to B

If there is only one path between two points, the resistors in between those two points are in series, no matter how they appear in a diagram and pass through the same amount of current regardless the value of each resistor.

Refer to the example of the series- resistors circuit arrangement in figure 2-16.

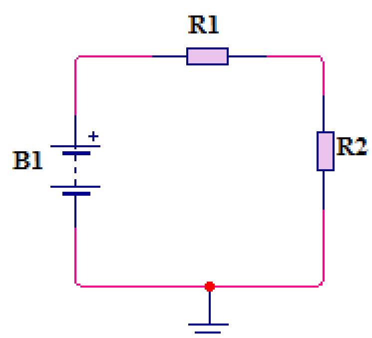


Figure 2-16: Two series-resistors circuit

Figure 2-16 represents an electrical circuit with two resistors in a series arrangement. For current to complete the electrical circuit it must flow from the voltage source (B_1) and pass through both resistor 1 (R_1) and resistor 2 (R_2) and then back to the voltage source (B_1).

Experiment 2-2: Measuring current in series resistors circuit

Current through each resistor in a series circuit is the same as the current through all the other resistors that are in series with it. Let consider three series resistors supplied by 9VDC and measure current at different points of the circuit.

Part list

No	Item	Specification	Quantity
1	Ammeter	Digital multimeter	1
2	9V DC source	9V battery	1
3	Breadboard	390-pin Prototyping board	1

4	Connecting wire	22-gaude ($0.33mm^2$) solid wire	10cm
5	47Ω, 56Ω, 82Ω resistors	47Ω, 56Ω, 82Ω resistors; 1/2watt	3

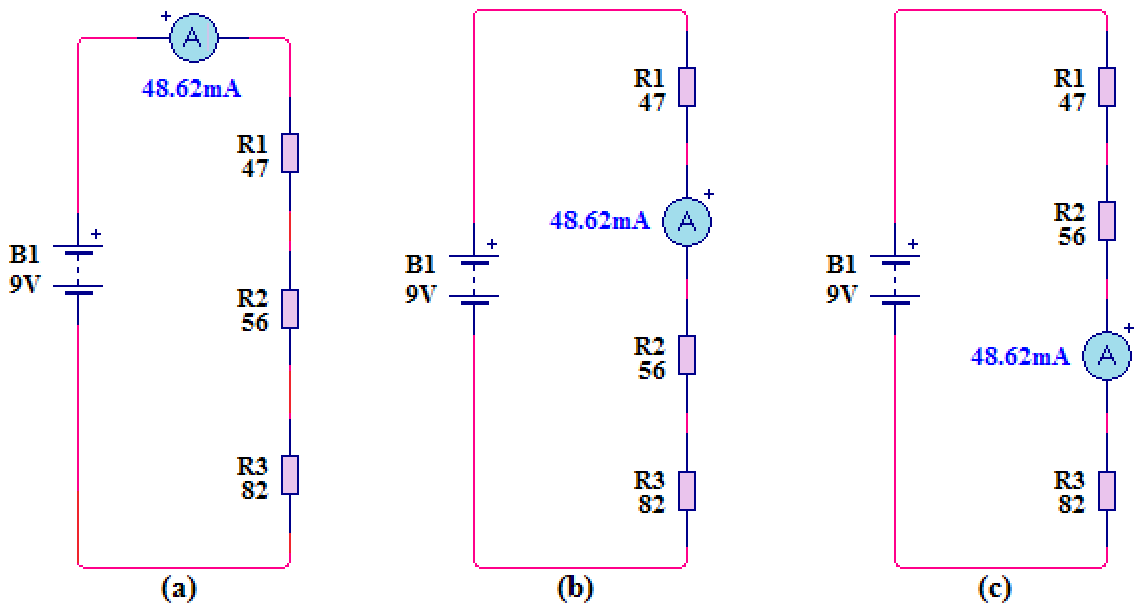


Figure 2-17: Current is the same at all points in a series circuit resistors no matter the values of each resistor.

With Ohm's law the current in the circuit is calculated as follow:

Given $B_1 = 9V$

$R_T = R_1+R_2+R_3$

$= 47\Omega + 56\Omega + 82\Omega$

$= 185\Omega$

$I = V/R$

$= 9V/185\Omega$

$= 0.04864A$

$= 48.64mA$

Procedure

1. Connect the circuit as illustrated in pictorial view figure 2-18. Connect the positive terminal of the battery to the power bus of the breadboard and the negative terminal to the ground bus of the breadboard.
2. Set multimeter in DC amps range, in 200mA scale. Care must be taken for high current measuring as this can destroy the ammeter. It is better to calibrate at highest scale value and go down to the low scale step by step until you get a finite reading.
3. Open the circuit from positive of the supply and the R_1 resistor and insert ammeter as shown in the circuit figure 2-17 part (a). The positive lead of ammeter to the positive of

the battery and the negative lead of ammeter to the terminal of resistor as illustrated in the circuit pictorial connection figure 2-18.

4. The ammeter reading is 48.4mA.
5. Connect the circuit as shown in schematic circuit figure 2-17 part (b).
6. Open the circuit from resistor R_1 and R_2 and insert the ammeter in between as shown in the circuit pictorial figure 2-19.
7. Connect the positive lead of ammeter to the resistor R_1 and the negative lead of ammeter to the terminal of resistor R_2.
8. Again the ammeter reading is 48.4mA.

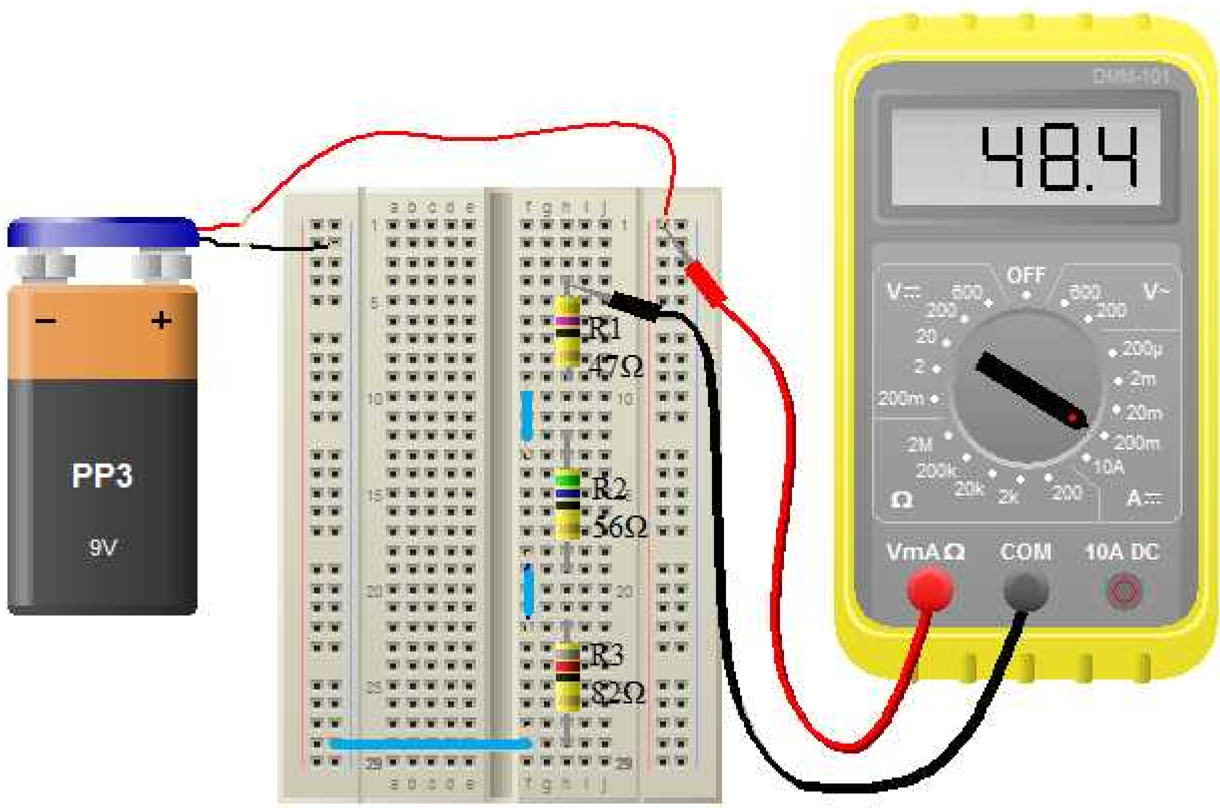


Figure 2-18: Measuring current of the circuit figure 2-17 (a) pictorial connection

9. Connect the circuit as shown in schematic circuit figure 2-17 part (c).
10. Open the circuit from resistor R_2 and R_3 and insert the ammeter in between as shown in the circuit pictorial figure 2-20.
11. Connect the positive lead of ammeter to the resistor R_2 and the negative lead of ammeter to the terminall of resistor R_3.
12. Also the ammeter displays the same reading (48.4mA) as previous connection arrangement.

Note that this reading can deviate to some tolerable values. This deviation is due to imperfection of devices and atmosphere condition. Also once wrong calibrated, the ammeter can read fault or null value.

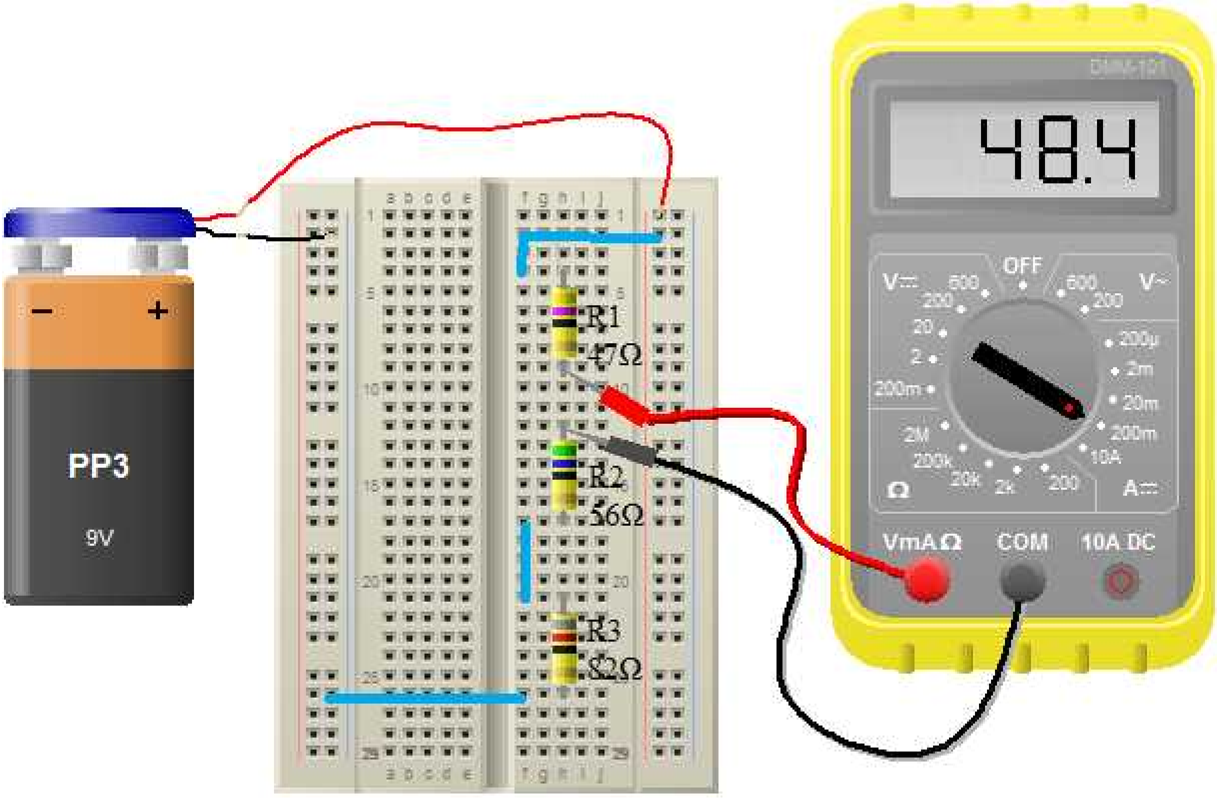


Figure 2-19: Measuring current of the circuit figure 2-17 (b) pictorial connection

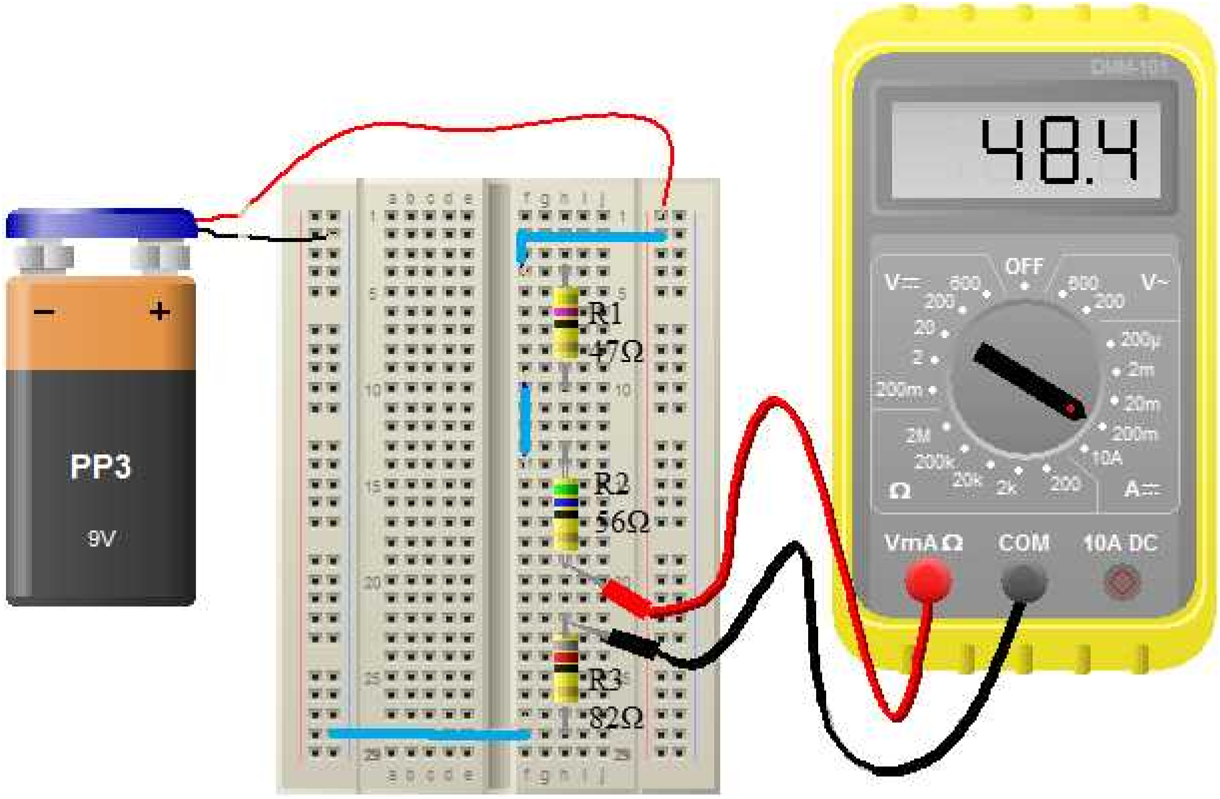


Figure 2-20: Measuring current of the circuit figure 2-17 (c) pictorial connection

Series resistors values add

When resistors are connected in series, the resistor values add because each resistor offers opposition to the current in direct proportion to its resistance. A greater number of resistors connected in series create more opposition to current.

More opposition to current implies a higher value of resistance. Thus for every resistance that is added in series, the total resistance increases.

$$R_T = R_1+R_2+R_3+\ldots+R_n \qquad \text{[Equation 2-2]}$$

Where R_T is the total resistance and R_n is the last resistor in the series string.

For equal resistors in series, the total resistance can be expressed as:

$$R_T = nR \qquad \text{[Equation 2-3]}$$

Where n is the number of equal-value resistors and R is the resistance value.

For any number of individual resistors connected in series, the total resistance is the sum of each of the individual values.

Voltage Dividers

A series circuit acts as a voltage divider. The voltage divider is an important application of series circuits. A circuit consisting of series string resistors connected to a voltage source acts as a voltage divider.

The circuit of figure 2-21 shows a circuit with two resistors in series, although there can be any number.

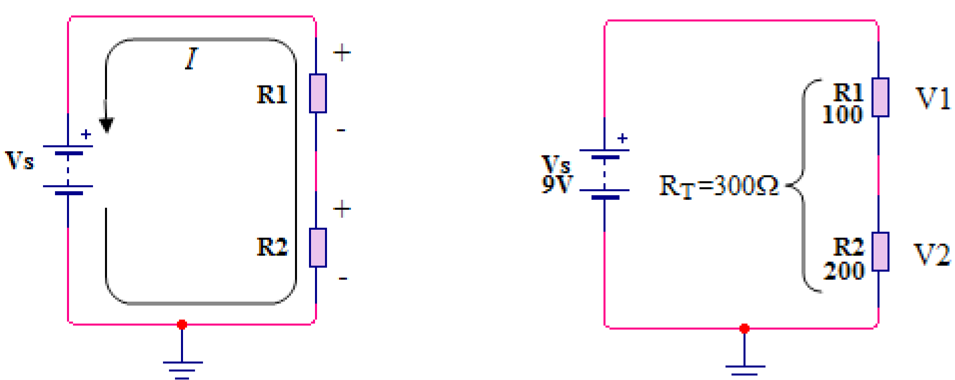


Figure 2-21: Example of a two-resistor voltage divider

As you see, there are two voltage drops: one across R_1 and other across R_2. These voltage drops are labeled V_1 and V_2, respectively, as indicated in the schematic. Since each resistor has the same current through it, the voltage drops are proportional to the resistance values. For example, if the value of R_2 is twice that of R_1, then the value of V_1 is twice that of V_1.

The total voltage drop around a single closed path divides among the series resistors in amounts directly proportional to the resistance values. The smallest resistance has the least voltage, and the largest resistance has the most voltage ($V = IR$).

For example if V_S is 9 V, R_1 is 100 Ω, R_2 is 200 Ω, then V_1 is one-half the total voltage, or 3 V, because R_1 is one-third the total resistance. Likewise, V_2 is two-thirds V_s, or 6 V, because R_2 is two-third the total resistance.

$$V_1 = V_s\,(R_1/R_1+R_2) = V_s\,(R_1/R_T) = 9V(100\Omega\ /\ 300\Omega) = 3V$$

$$V_2 = V_s\,(R_2/R_1+R_2) = V_s\,(R_2/R_T) = 9V(200\Omega\ /\ 300\Omega) = 6V$$

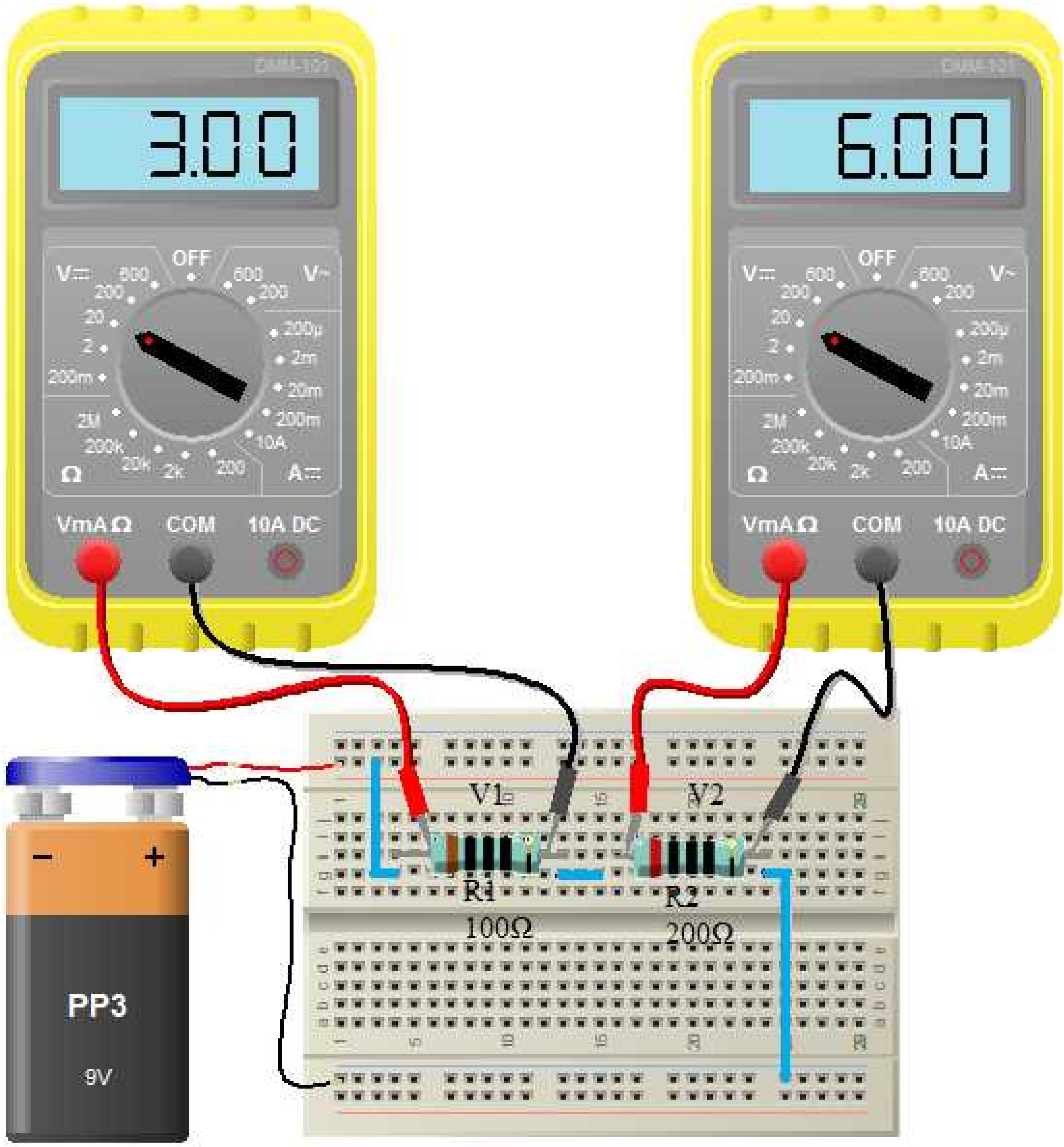


Figure 2-22: Example of a two-resistor voltage divider, real world circuit connection

Loading Effect

In this section you will learn how resistive loads affect the operation of voltage-divider circuits.

The voltage divider in figure 2-23 produces an output voltage (V_{out}) of 5V because the input voltage is 10V and the two resistors are of equal value. This voltage is the unloaded output voltage.

When a load resistor, R_L, is connected from the output to ground as shown in figure 2-24, the output voltage is reduced by an amount that depends on the value of R_L. For our example use 1KΩ as R_L. This effect is called *loading*.

Figure 2-23 and figure 2-24 compose a circuit with unloaded output (a) and loaded output (b)

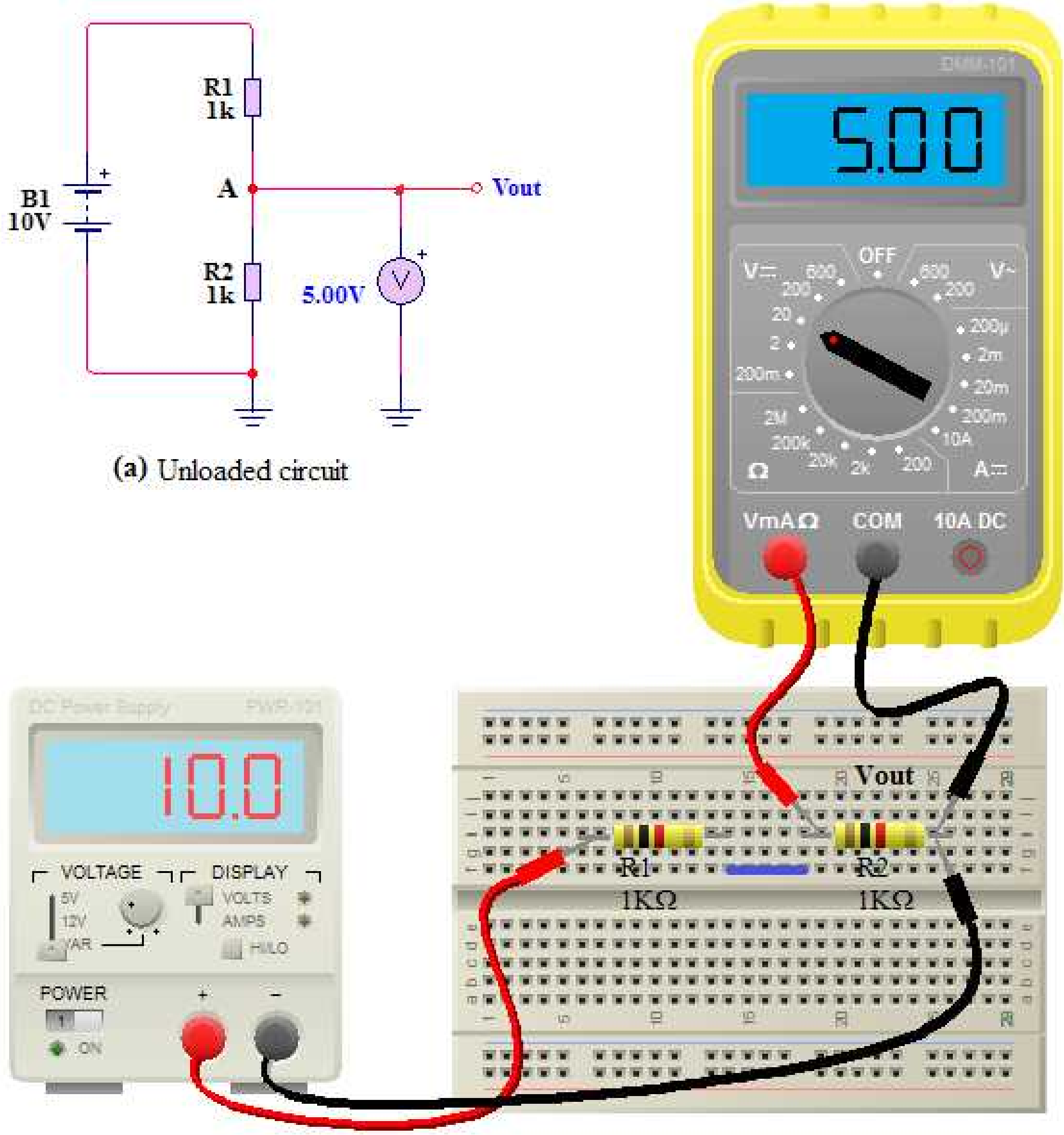


Figure 2-23: A voltage divider with unloaded output

(a) The unloaded output is:

$$V_{out(unloaded)} = (R_2/R_1+R_2)*V_{B1} = (1KΩ/1KΩ+1KΩ)*10V = ½*10V = 5V$$

The load resistor is in parallel with R_2, reducing the resistance from node A to ground and, as a result, also reducing the voltage across the parallel combination. This is one effect of loading a voltage divider.

Another effect of a load is that more current is drawn from the source because the total resistance of the circuit is reduced.

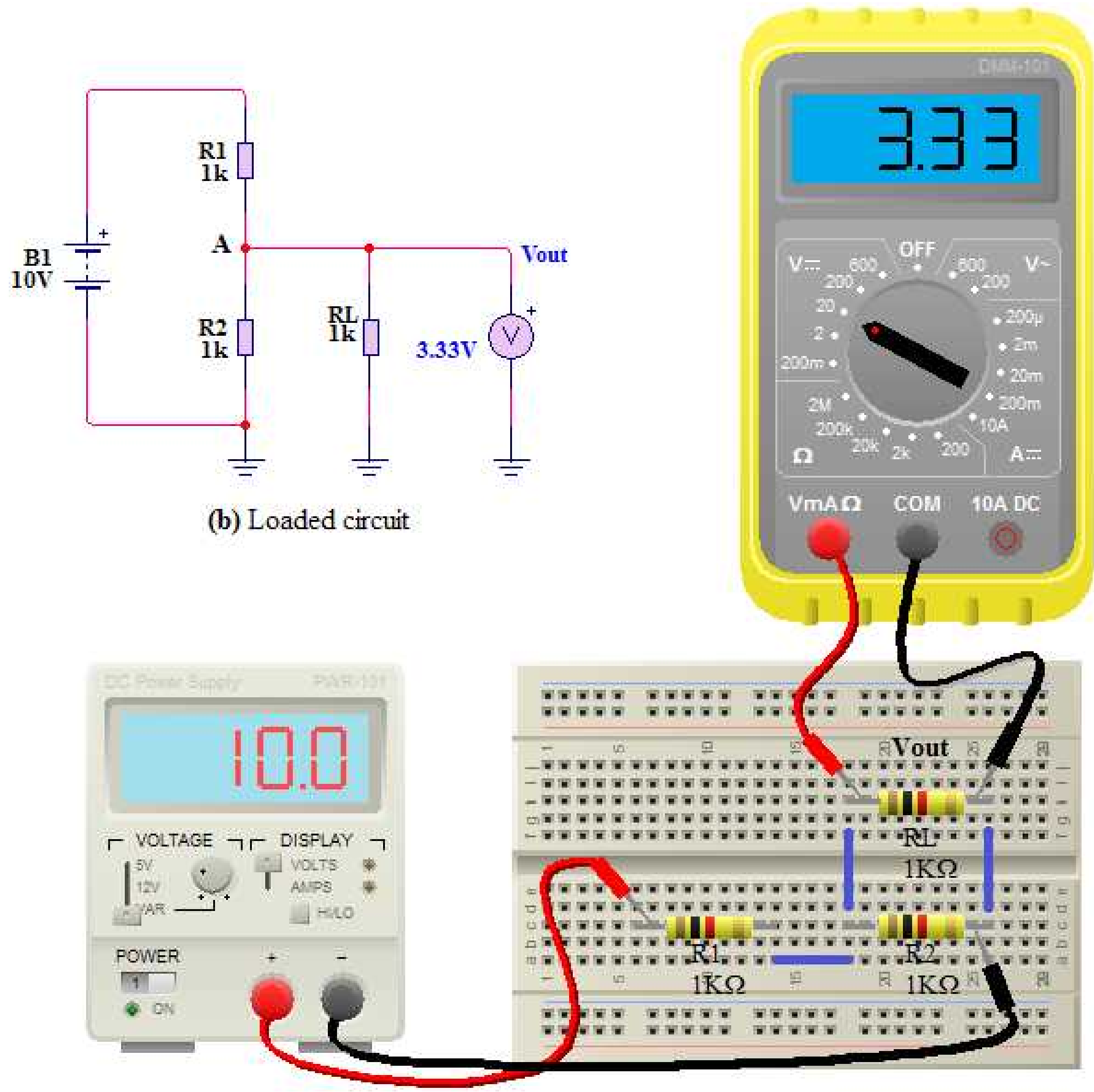


Figure 2-24: A voltage divider with loaded output

(b) The loaded output voltage with 1KΩ load resistor connected, R_L is in parallel with R_2, which gives

$$R_2 \| R_L = R_2 * R_L / R_2 + R_L$$
$$= 1K\Omega * 1K\Omega / 2K\Omega = 0.5K\Omega = 500\Omega$$

$$V_{out(loaded)} = (R_2 \| R_L / R_1 + R_2 \| R_L) * V_{B1}$$
$$= (0.5K\Omega / 1.5K\Omega) * 10V = 3.33V$$

The larger R_L is, compared to R_2, the less the output voltage is reduced from its unloaded value, as illustrated in the figure. When two resistors are connected in parallel and one of the resistors is much greater than the other, the total resistance is close to the value of the smallest resistance.

Bipolar Voltage Dividers

An example of a voltage divider that produces both positive and negative voltages from a single source is shown in the figure 2-25. Notice that neither the positive nor the negative terminal of the source is connected to reference ground or common. Voltage at nodes A and B are positive with respect to ground, and the voltage at nodes C and D are negative with respect to ground.

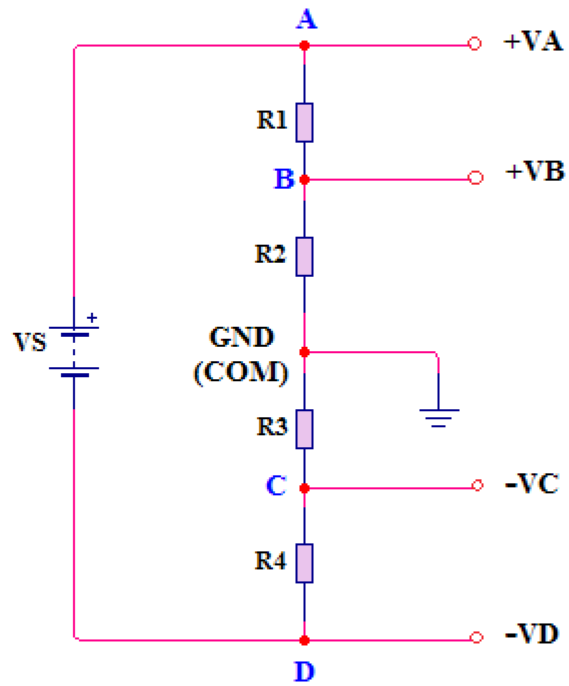


Figure 2-25: A bipolar voltage divide. The positive and negative voltages are with respect to reference ground

Loading Effect of a Voltmeter

As you have learned, voltmeter must be connected in parallel with a resistor in order to measure the voltage across that resistor. Because of its internal resistance, a voltmeter, or any other measuring instrument for that matter, puts a load on the circuit and will affect, to a certain extent, the voltage that is being measured. Until now, we have ignored the loading effect because the internal resistance of a voltmeter is very high, and normally it has negligible effect on the circuit being measured. However, if the internal resistance of the voltmeter is not sufficiently greater than the circuit resistance across which it is connected; the loading effect will cause the measured voltage to be less than its actual value.

When a voltmeter is connected to a circuit as shown in the figure 2-26 part (a), for example, its internal resistance appears in parallel with R_2, as shown in figure 2-26 part (b). The resistance from A to B is altered by the loading effect of the voltmeter's internal resistance, R_M, and is equal to $R_3 \| R_M$, as indicated in figure 2-26 part (c).

chap 2

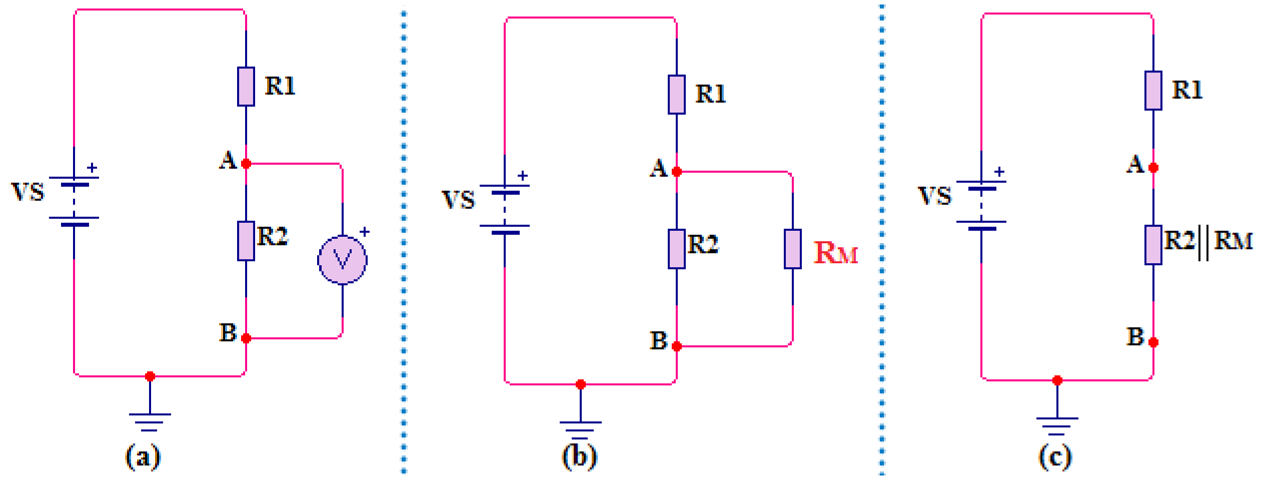


Figure 2-26: The effect of a voltmeter's internal resistance on reading

A good rule of thumb is that *if the meter resistance is at least ten times greater than the resistance across which it is connected, the loading effect can be neglected (measurement error is less than 10%)*.

Example: For the circuits below, figure 2-26, determine how much does the digital multimeter affect the voltage being measured. Assume the meter has an input resistance (R_M) of 10MΩ.

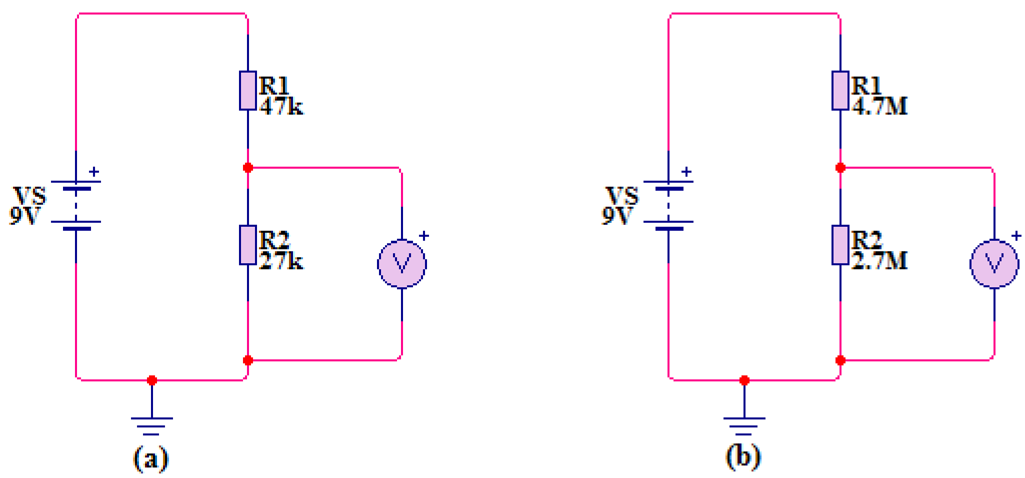


Figure 2-27: Voltmeter loading effect example; (a) Voltmeter with low internal resistance, (b) Voltmeter with high internal resistance

Solution

To show the small differences more clearly, the results are expressed in more than three significant digits in this example.

(a) Refer to the above circuit, figure 2-27 part (a), the unloaded voltage across R_2 in the voltage-divider circuit is

$$V_{R2} = (R_2/R_1+R_2)*V_s = (27K/74K)*9V = 3.2837V$$

The meter's resistance in parallel with R_2 is

$$R_2 \| R_M = (R_2 * R_M)/R_2 + R_M = 27K*10M/10.027M = 26.927K\Omega$$

The voltage actually measured by the meter is

$$V_{R2} = (R_2 \| R_M / R_1 + R_2 \| R_M)*V_s = (26.927K/73.927K)*9V = 3.278V$$

The voltmeter has no measurable loading effect.

(b) Refer to the above circuit figure 2-27 part (b), the unloaded voltage across R_2 in the voltage-divider circuit is

$$V_{R2} = (R_2/R_1+R_2)*V_s = (2.7M/7.4M)*9V = 3.2837V$$

The meter's resistance in parallel with R_2 is

$$R_2 \| R_M = (R_2 * R_M)/R_2 + R_M = 2.7M*10M/12.7M = 2.126M$$

The voltage actually measured by the meter is

$$V_{R2} = (R_2 \| R_M / R_1 + R_2 \| R_M)*Vs = (2.126M/6.826)*9V = 2.803V$$

The loading effect of the voltmeter reduces the voltage by a noticeable amount. As you can see, the higher the resistance across which a voltage is measured, the more the loading effect.

Variable Resistors

Variable resistors are designed so that their resistance values can be changed easily with a manual or an automatic adjustment.

Two basic uses for variable resistors are to divide voltage and to control current. The variable resistor used to divide voltage is called a *potentiometer.* The variable resistor used to control current is called a *rheostat.*

A) Potentiometer

Potentiometer is a variable resistor with three terminals. Notice that the two end terminals are labeled 1 and 2.

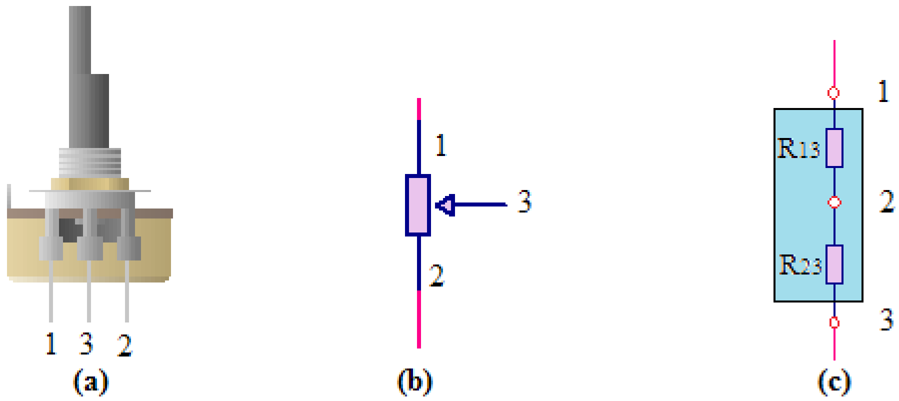


Figure 2-28: Potentiometer; (a) Pictorial, (b) Schematic symbol, (c) Equivalent schematic

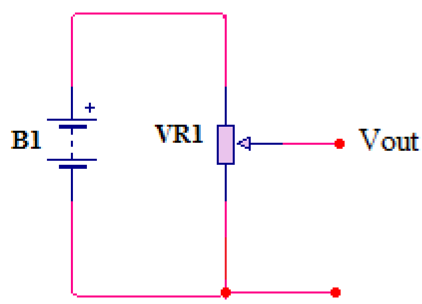
The adjustable terminal or wiper is labeled 3. Terminals 1 and 2 have a fixed resistance between them, which is the total resistance. Terminal 3 is connected to a moving contact (wiper). You can vary the resistance between 3 and 1 or between 3 and 2 by moving the contact.

The potentiometer as an adjustable voltage divider

The potentiometer acts as a voltage divider, which can be illustrated by separating the total resistance into two parts as illustrated in equivalent schematic. The resistance between terminal 1 and terminal 3 (R_{13}) is one part, and the resistance between terminal 3 and terminal 2 (R_{32}) is the other part. Thus, the potentiometer actually is a two-resistor voltage divider that can be manually adjusted.

Experiment 2-3: Potentiometer used as voltage divider

Use potentiometer as a voltage-control device. When a fixed voltage is applied across the end terminal, a variable voltage is obtained at the wiper contact with respect to either end terminal. A series resistor Rs is added in series with potentiometer to avoid short circuit once a wiper is at 0% but this reduced the amount of voltage available at the output since there will be a voltage drop across the series resistor.



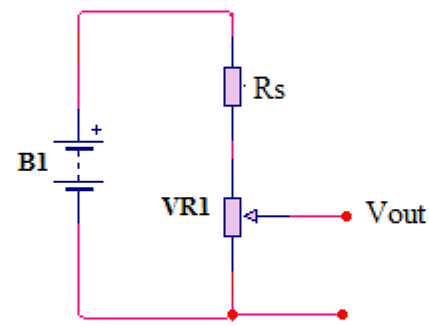


Figure 2-29: Potentiometer as voltage divider; (a) simple potentiometer as voltage divider, (b) A series resistor RS with Potentiometer used as voltage divider

Parts list

No	Item	Specification	Quantity
1	Voltmeter	Digital multimeter	2
2	9V DC source	9V battery	1
3	Breadboard	Prototyping board	1
4	Connecting wire	22-gaude (0.33mm^2) solid wire	10cm
5	Potentiometer	10KΩ Potentiometer	1

Procedure

1. The figure 2-30 part (a) through part (c) shows what happens when a wiper contact (3) is moved.
2. In figure 2-30 part (b) the wiper is exactly centered, making the two resistances equal. If you measure voltage across terminal 3 to 2 as indicated, you have one-half of the total source voltage; that is 4.5V.

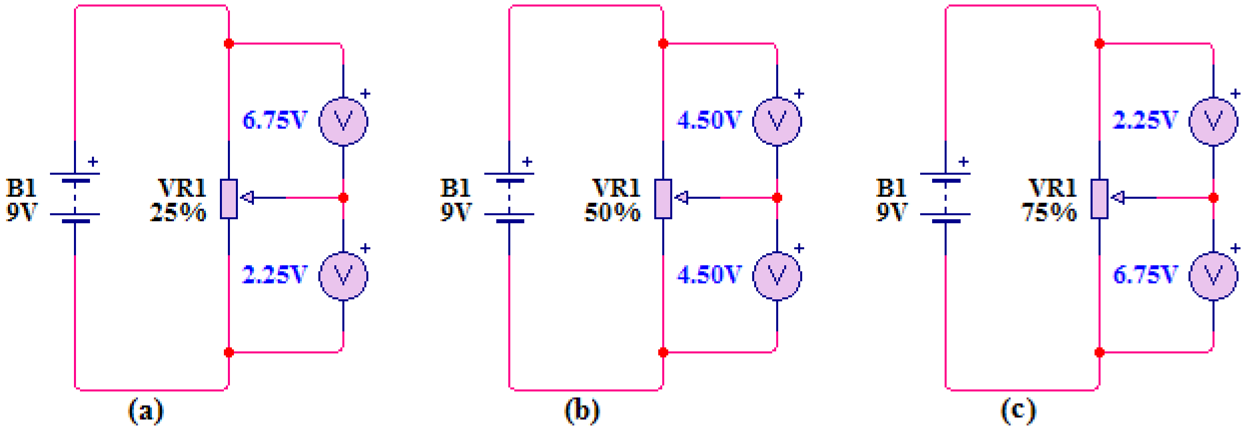


Figure 2-30: Potentiometer used as voltage divider; (a) Quarter turn, (b) Half turn, (c) Three-quarter turn

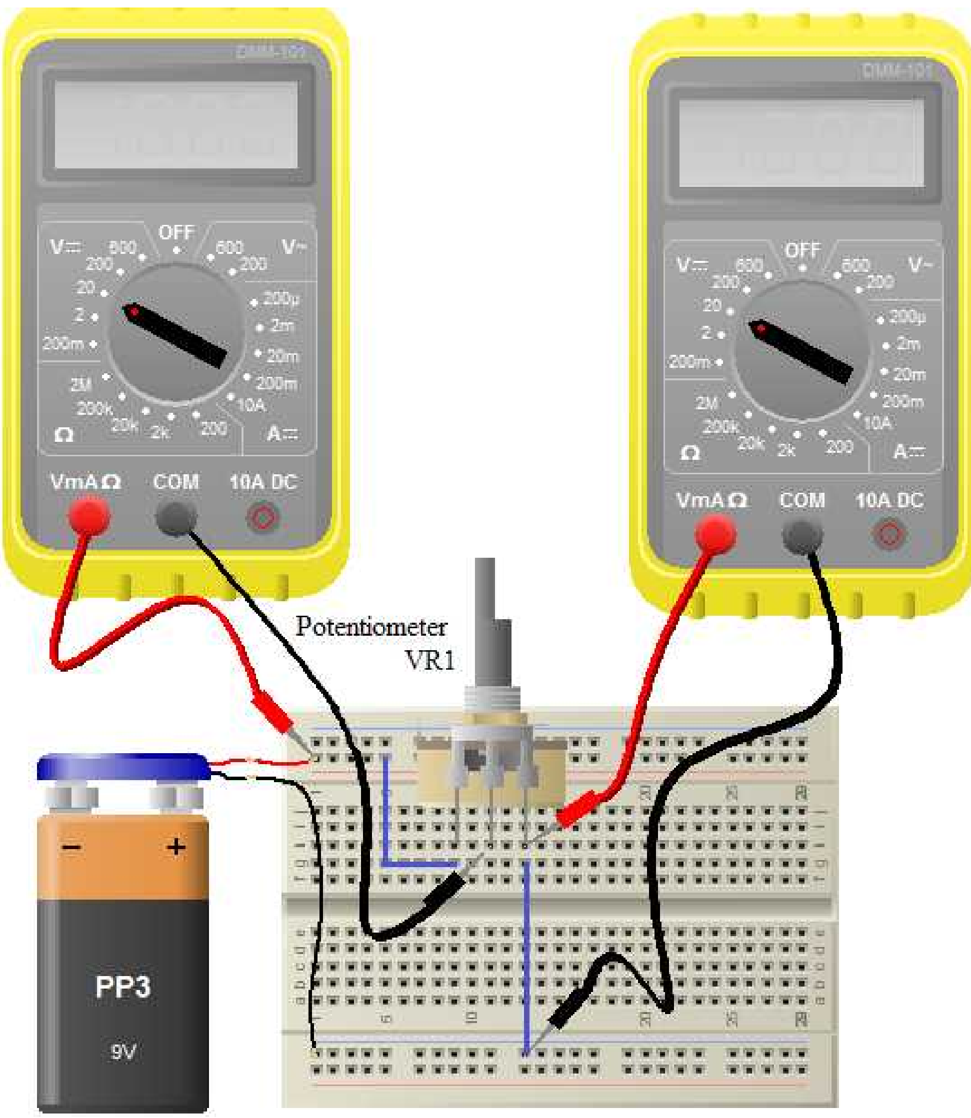


Figure 2-31: Potentiometer used as voltage divider

3. When the wiper is moved up from the center position, in figure 2-30 part (c), the resistance between terminals 3 and 2 increases and the voltage across it increases proportionally. For this example the wiper is at 75%. Thus, the voltage between terminal 3 and 2 is 6.75V with respect to the ground, exactly 75% of 9V the supply voltage.
4. When the wiper is moved down from the center position, as in figure 2-30 part (a), the resistance between terminal 3 and 2 decreases and the voltage decreases proportionally. For this example the wiper is at 25%. Therefore, the voltage between terminal 3 and 2 is 2.25V with respect to ground. This corresponds to 25% of the supply voltage, i.e. 9V.
5. Note that the summation of voltage measured between terminal 3 and 2 and terminal 2 and 1 must be equal to supply voltage or 100%.

Application of potentiometer

- The volume control of a radio receiver is a common application of a potentiometer used as voltage divider. Since the loudness of the sound is dependent on the amount of voltage associated with the radio signal, you can increase or decrease the volume by adjusting the potentiometer, that is, by tuning the knob of the volume control on the set.

The block diagram figure 2-32 shows how a potentiometer can be used for volume control in a typical radio receiver.

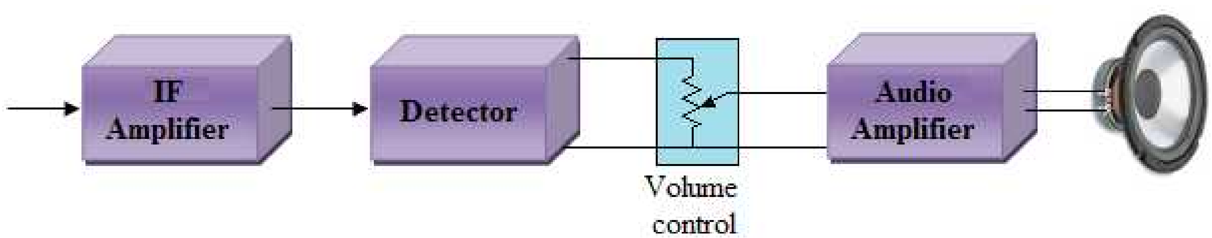


Figure 2-32: A variable voltage divider used for volume control in a radio receiver

- Another application of voltage divider which depicts a potentiometer voltage divider is a level sensor. When potentiometer is used as a positional sensor, the moveable object is connected directly to the rotational shaft or slider of the potentiometer. A DC voltage applied across the two outer fixed terminal of the resistive element of potentiometer serve as reference.
  The output voltage signal is taken from the wiper terminal of the movable contact. Then for example, if you apply a voltage of 10V across the two fixed terminals forming resistive element of the potentiometer the maximum output voltage would be equal to the supply voltage at 10 volts, with the minimum output voltage equal to 0 volts. Then the potentiometer wiper will vary the output signal from 0 to 10 volts, with 5 volts indicating that the wiper or slider is at its half-way or centre position.

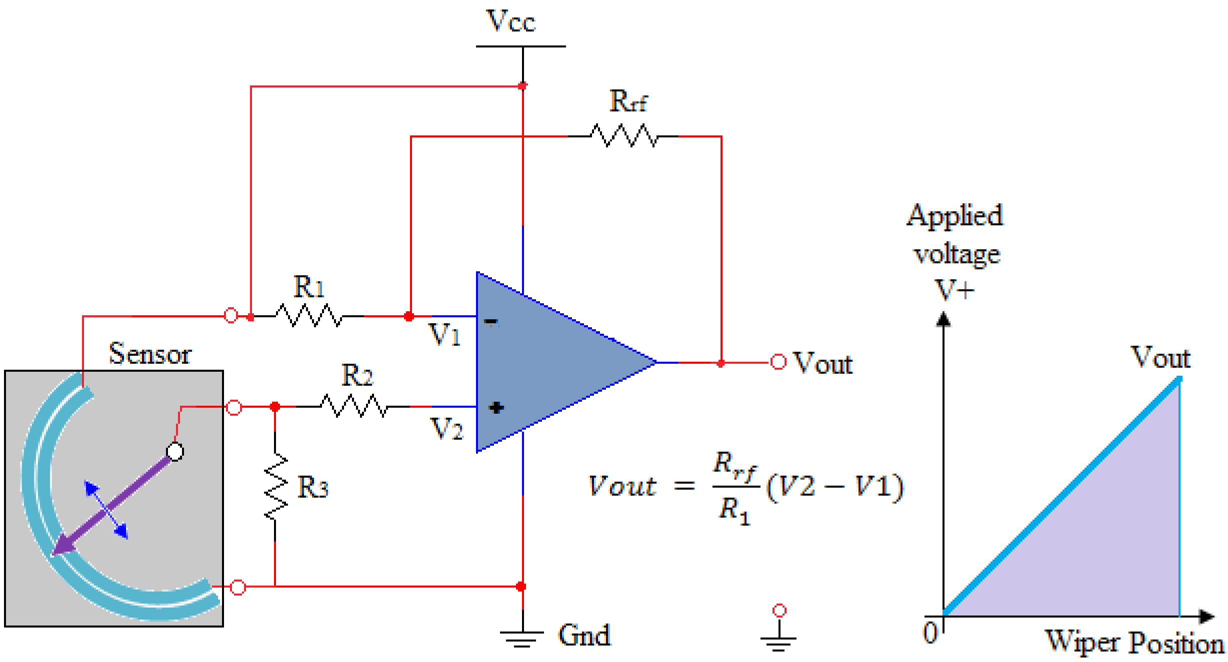


Figure 2-33: A potentiometer voltage divider used as level sensor

- Still another application for voltage dividers in setting the dc operating voltage (bias) in transistor amplifier. The figure 2-34 shows a voltage divider used for this purpose. You will study transistor amplifier and biasing later, so it is important that you understand the basics of voltage divider at this point.

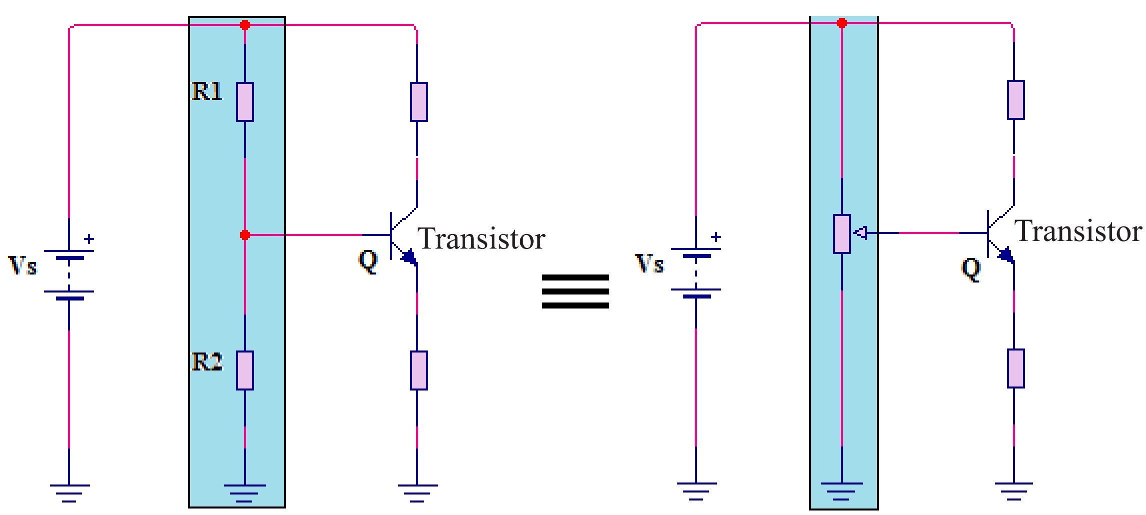


Figure 2-34: The voltage divider used as a bias circuit for a transistor amplifier

These examples are only three out of many possible applications of voltage dividers

Rheostat

Rheostat is a two terminal variable resistor. The rheostat is used as a current-control device; the current can be changed by changing the wiper position. Notice potentiometer can be used as rheostat if the two terminals (include a wiper) are used.

Figure 2-35: Rheostat; (a) Rheostat symbol, (b) Potentiometer connected as Rheostat

The figure 2-36 shows the basic circuit of how a rheostat is configured to control the current.

Example: Expect a load with 100Ω which, in its under normal operation, can support the current ranging between 50mA as minimum current and 60mA as maximum current at which it can work without being destroyed. Once this maximum current is exceeded this load resistor can be burnt. Assume that this load has to be connected on 9V DC supply, with ohm's law; the current through this load resistor is given by:

$$I_{RL} = V_S/R_L$$
$$= 9V/100\Omega$$
$$= 0.09A = 90mA$$

According to the specifications of the load (resistance =100Ω, Minimum current=50mA, Maximum current =60mA) the obtained current (90mA) can cause breakdown to the load resistor.

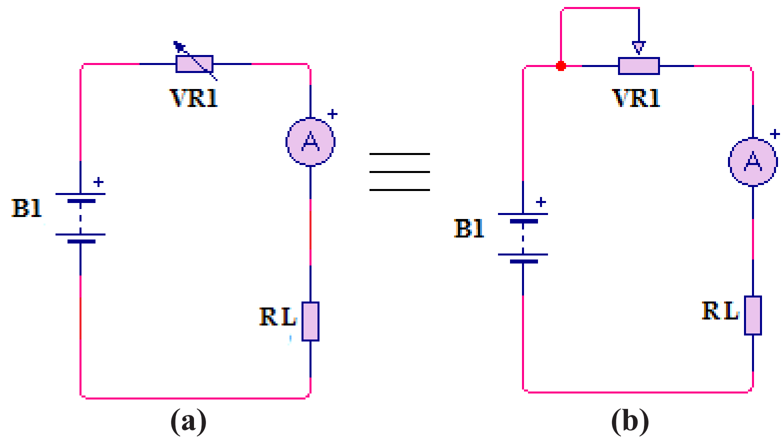
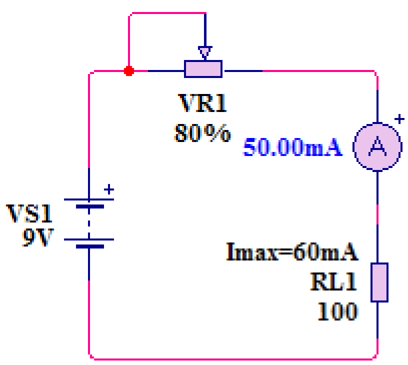


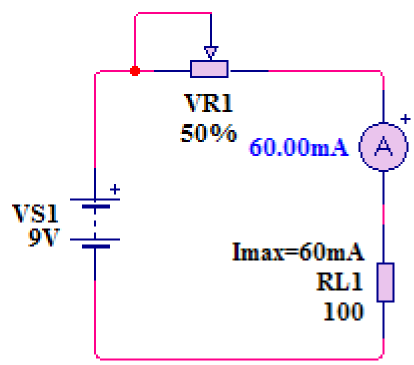
Figure 2-36: Rheostat used for controlling current; (a) Typical rheostat used to control current, (b) Potentiometer configured as rheostat to control current

One way to avoid this malfunction is to insert a series resistor (rheostat) in series with load resistor.

Part list

No	Item	Specification	Quantity
1	Ammeter	Digital multimeter	1
2	9V DC source	9V battery	1
3	Breadboard	Prototyping board	1
4	Connecting wire	22-gaude solid wire	10cm
5	100Ω Load resistor	100Ω,1/2watt, 60mA	1
6	100Ω rheostat	100Ω potentiometer configured as a rheostat	1





Given that the potentiometer's (rheostat) maximum resistance is 100Ω and the load resistance is equal to 100Ω, therefore, the minimum applicable current is given by:

$I_{Low} = V_s/V_{R1}+R_L$

$= 9V/100\Omega+100\Omega = 0.045A = 45mA$ (this current is not enough for the load to operate)

- Minimum required current for load resistor = 50mA, then the rheostat must be set to resistance equal to:

$I_{min} = V_s/V_{R1}+R_L$

$50mA = 9V/V_{R1}+100\Omega$

$50mA\ (V_{R1}+100\Omega)=9V$

$V_{R1} = 9V/50mA-100\Omega$

$V_{R1} = 180\Omega - 100\Omega$

$V_{R1} = 80\Omega$

Thus the rheostat must be set at: 80Ω /100Ω *100=80% of its maximum resistance.

- Maximum required current for load resistor =50mA, then the rheostat must be set to the resistance equal to:

$I_{max} = V_s/V_{R1}+R_L$

$60mA= 9V/V_{R1}+100\Omega$

$60mA\ (V_{R1}+100\Omega) = 9V$

V_{R1} = 9V/60mA-100Ω

V_{R1} = 150Ω -100Ω

V_{R1} = 50Ω

Thus the rheostat must be set at: 50Ω /100Ω *100 = 50% of its maximum resistance.

Parallel Resistors

When two or more resistors are individually connected between the same two points, they are in parallel with each other. A parallel circuit provides more than one path for current. Each parallel path in a circuit is called a *branch.* It is important that you learn to recognize parallel circuits regardless of how they may be drawn. A rule for identifying parallel circuits is as follow:

If there is more than one current path (branch) between two points, and if the voltage between those two points also appears across each of the branches, then there is a parallel circuit between those two points.

The figure 2-37 shows parallel resistors drawn in different ways between two points labeled A and B. Notice that in each case, the current has two paths going from A to B and the voltage across each branch is the same. Although these figures show only two parallel paths there can be any number of resistors in parallel.

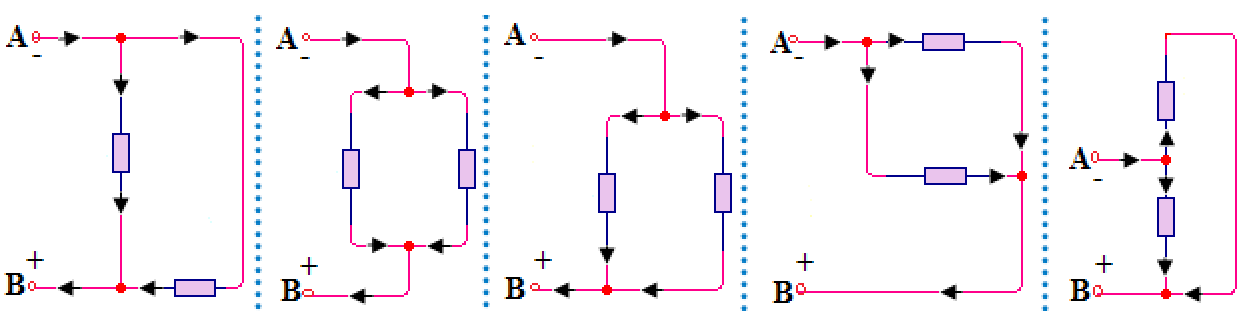


Figure 2-37: Examples of circuits with two parallel paths

Total Parallel Resistance

When resistors are connected in parallel, the total resistance of the circuit decreases. The total resistance of a parallel circuit is always less than the value of the smallest resistor. For example, if a 10 Ω resistor and a 100 Ω resistor are connected in parallel, the total resistance is less than 10 Ω.

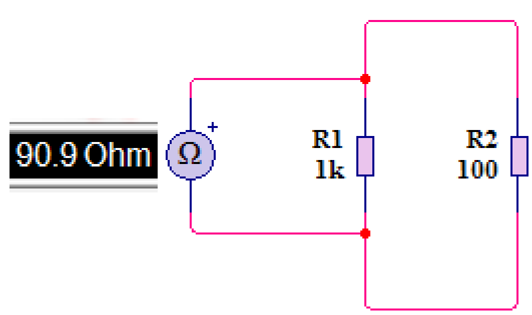


Figure 2-38: Example of two parallel resistors Circuit

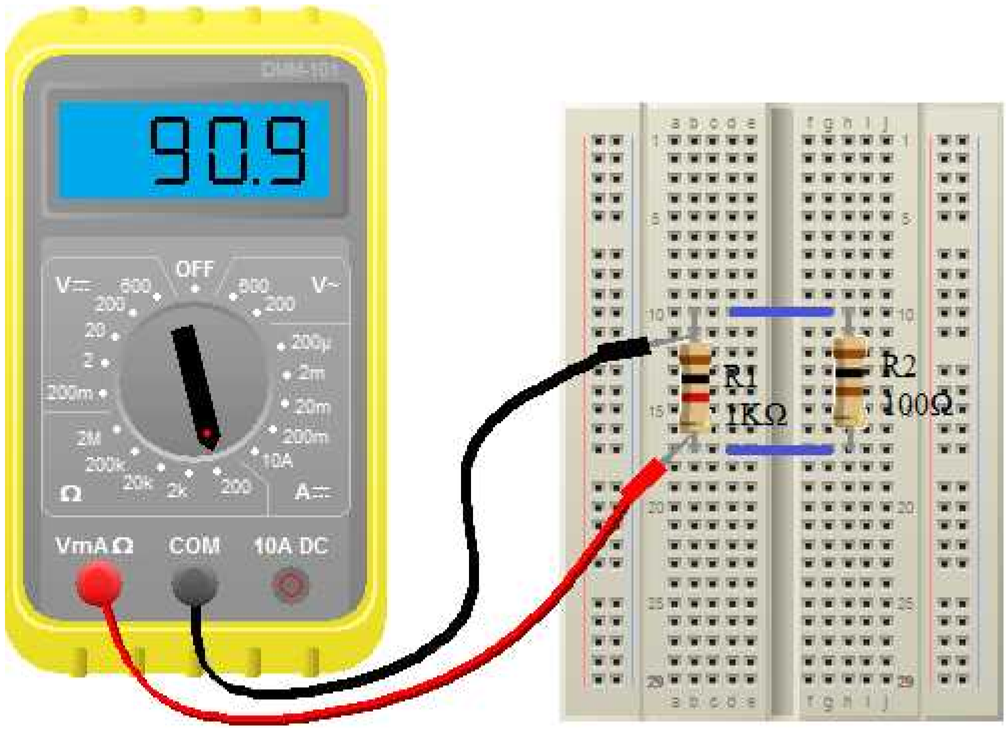


Figure 2-39: Example of two parallel resistors circuit connection

The formula for parallel resistance is as follow:

$$1/R_T = 1/R_1 + 1/R_2 + 1/R_3 + \dots + 1/R_n \qquad \text{[Equation 2-4]}$$

Where R_T is the equivalent of parallel resistance, R_n is the last resistor among the parallel.

For the case of two resistors in parallel, the total resistance of two resistors in parallel is equal to the product of the two resistors divided by the sum of the two resistors.

$$R_T = R_1 * R_2 / R_1 + R_2 \qquad \text{[Equation 2-5]}$$

This equation (Equation 2-5) is sometimes referred to as the “product over the sum” formula.

For the case of equivalent-value resistors in parallel,

$$R_T = R/n \qquad \text{[Equation 2-6]}$$

The equation 2-6 says that when any number of resistors (*n*), all having the same resistance (*R*), are connected in parallel, R_T is equal to the resistance divided by the number of resistors in parallel.

Voltage in a Parallel Circuit

The voltage across any given branch of parallel circuit is equal to the voltage across each of the other branches in parallel.

One advantage of parallel circuit over a series circuit is that when one branch opens, the other branches are not affected.

Applications of a Parallel Circuit

- Automotive: For automobile lighting system, when one headlight on a car goes out, it does not cause the other lights to go out because they are in parallel.
- Residential: Another common use of parallel circuits is in residential electrical systems. All the lights and appliances in home are wired in parallel.

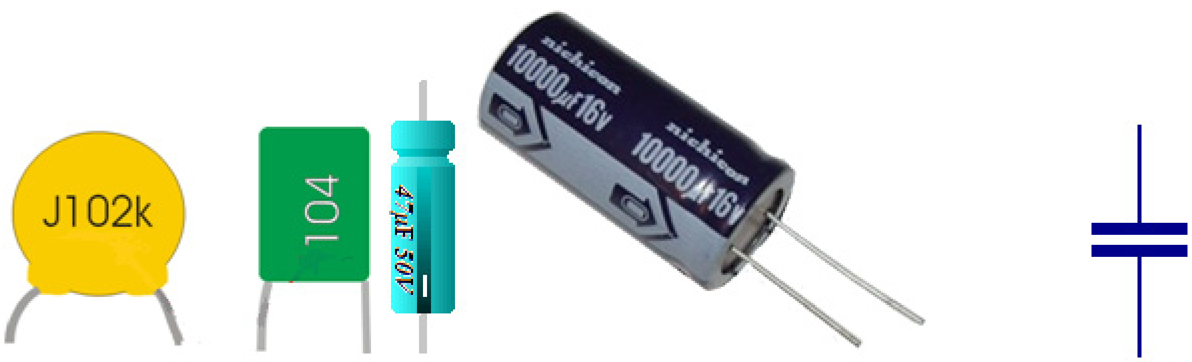
2. 3. 2. Resistor in AC Circuits

When a time-varying AC voltage such as a sinusoidal voltage is applied to a resistive circuit, the circuit laws and power formulas still apply. Ohm's law Kirchhoff's laws and power laws formula apply to AC in the same way that they apply to DC circuits.

2. 4. Capacitor

A capacitor is a passive electrical component that stores electrical charges and has property of capacitance. A capacitor is made of two parallel conductive plates separated by an insulating material called the *dielectric*. Connecting leads are attached to the parallel plates.

Typical capacitors are shown in figure 2-40 in part (a) and the schematic symbol is shown in figure 2-40 part (b).



(a) (b)

Figure 2-40: Capacitor; (a) Example of typical capacitors, (b) Capacitor schematic circuit symbol

Capacitance

The amount of charge that a capacitor can store per unit of voltage across its plates is its capacitance, designated *C*. That is, *capacitance* is a measure of capacitor's ability to store charge. The more charge per unit of voltage that a capacitor can store, the greater its capacitance, as expressed by the following formula:

$$C = Q/V \qquad \text{[Equation 2-7]}$$

Where *C* is capacitance, Q is charge, and V is voltage.

By rearranging the terms in equation 2-7, you can obtain two other formulas.

$$Q = CV \qquad \text{[Equation 2-8]}$$

$$V = Q/C \quad \text{[Equation 2-9]}$$

The unit of capacitance: The *farad* (F) is the basic unit of capacitance. Recall that the *coulomb* (C) is the unit of electrical charge.

One farad is the amount of capacitance when one coulomb of charge is stored with one volt across the plates.

Also, from equation 2-9, $Q = CV$, the amount of charge stored is directly related to the voltage as well as the capacitance. Therefore, the amount of energy stored is also dependent on the square of the voltage across the plates of the capacitor. The formula for energy stored by a capacitor is

$$W = \frac{1}{2} CV^2 \quad \text{[Equation 2-10]}$$

Most capacitors that are used in electronics work have capacitance values in microfarads (µF) and picofarads (pF). A microfarad is one-millionth of a farad ($1\mu F = 1x10^{-6}F$); one picofarad is one-trillionth of a farad ($1pF = 1x10^{-12}F$).

Series Capacitors

The total capacitance of a series connection of capacitors is less than the individual capacitance of any other capacitors because the effective plate separation increases. The calculation of total series capacitance is analogous to the calculation of total resistance of parallel resistors.

In a series connection, the total capacitance is equal to the reciprocal of the sum of the reciprocal values of the capacitors.

chap 2

$$1/C_t = 1/C_1 + 1/C_2 + 1/C_n \quad \text{[Equation 2-11]}$$

All series capacitors store the same amount of charges

$$Q_T = Q_1 = Q_2 = Q_3 = Q_n \quad \text{[Equation 2-12]}$$

By Kirchhoff's voltage law, which applies to capacitive circuits as well as to resistive circuits, the sum of the capacitor voltages equals to the source voltages.

$$V_s = V_1 + V_2 + V_3 + \ldots + V_n$$

The voltage across each capacitor in series connection depends on its capacitance value according to the formula $V = Q/C$. You can determine the voltage across any individual capacitor in series with the following formula:

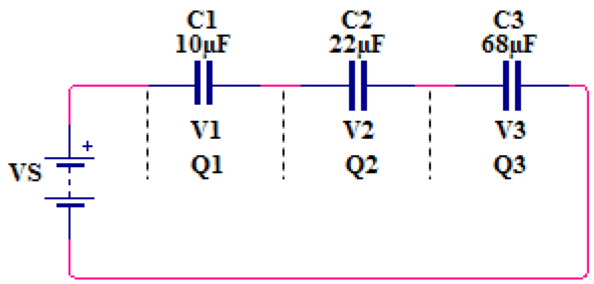
$$V_x = (C_T/C_X)\, V_S \quad \text{[Equation 2-13]}$$

Where C_X is any capacitor in series, such as C_1, C_2 and C_3, and so on, and V_x is the voltage across C_X.

The largest-value capacitor in series connection will have the smallest voltage across it. The smallest-value capacitor will have the largest voltage across it.

Example: C_1=10µF; C_2=22µF and C_3=68µF

The circuit diagram:



Solution:

$1/C_T = 1/10\mu F + 1/22\mu F + 1/68\mu F$

$1/C_T = 0.16\mu F$

C_T=6.2437µF

Parallel Capacitors

Capacitors add when they are connected in parallel. When the capacitors are connected in parallel, the total capacitance is the sum of the individual capacitance because the effective plate area increases. The calculation of total parallel capacitance is analogous to the calculation of total series resistance.

In a parallel connection, the total capacitance is obtained by adding the individual capacitance values of the connected capacitors. The expanded formula is as follow; the subscript *n* can be any number.

$$C_T = C_1 + C_2 + C_n \qquad \text{[Equation 2-14]}$$

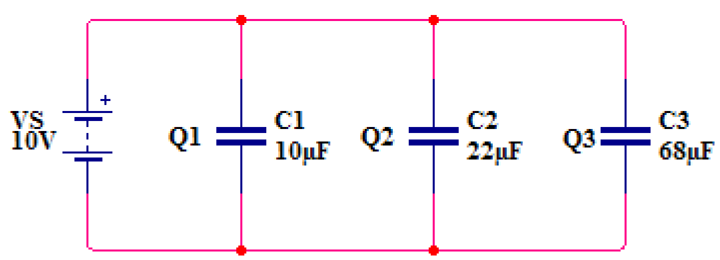
The charge stored by capacitor together equals the total charge that was delivered from the source.

$$Q_T = Q_1 + Q_2 + Q_3 + \ldots + Q_n \qquad \text{[Equation 2-15]}$$

Because the voltages across the parallel capacitor are equal ($V_1 = V_2 = V_3 = V_n$), you can use substitution to get the following relationship:

$$C_T V_s = C_1 Vs + C_2 V_s + C_n V_s \qquad \text{[Equation 2-16]}$$

Example: Given the circuit below, what is the total capacitance of the circuit? The charge Q_T? voltage across each capacitor? and the charge of each capacitor?



Solution:

$C_T = C_1 + C_2 + C_3$

$= 10\mu F + 22\mu F + 68\mu F = 100\mu F$

$V_{C1} = V_{C2} = V_{C3} = V_S = 10V$

$Q_1 = C_1 * V_{C1} = 10V*10\mu F = 100\mu C$

$Q_2 = C_2 * V_{C2} = 10V*22\mu F = 220\mu C$

$Q_3 = C_3 * V_{C3} = 10V*68\mu F = 680\mu C$

$Q_T = C_T * V_S = Q_1 + Q_2 + Q_3 = 1000\mu C$

2. 4. 1. Capacitor in DC Circuits

A capacitor will charge up when it is connected to a dc voltage source. The buildup of charge across the plates occurs in a predictable manner that is dependent on the capacitance and the resistance in a circuit.

Charging a Capacitor

A capacitor will charge when it is connected to a dc voltage source. The capacitor in figure 2-41 part (a) is uncharged; that is, plate A and plate B have equal numbers of free electrons. When the switch is closed, in figure 2-41 part (b), the source moves electrons away from plate A through the circuit to the plate B as the arrows indicate. As plate A loses electrons and plate B gains electrons, plate A becomes positive with respect to the plate B. As this charging process continues, the voltage across the plates builds up rapidly until it is equal to the source voltage, Vs, but opposite in polarity, as shown in figure 2-41 part (c). When the capacitor is fully charged, there is no current.

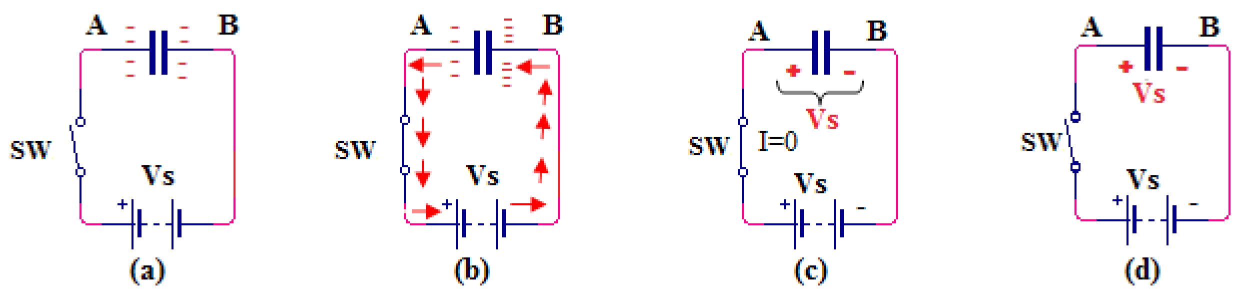


Figure 2-41: Charging a capacitor. (a) Uncharged, (b) Charging, (c) Full charged, (d) Retain charge

A capacitor blocks a constant dc.

When the charged capacitor is disconnected from the source, as shown in figure 2-41 part (d), it remains charged for long period of time, depending on its leakage resistance. The charge on an electrolytic capacitor generally leaks off rapidly than in other types of capacitor.

The portion of total charge that is stored by a capacitor depends on its capacitance value and the voltage applied according to the relationship $Q = CV$

Discharging a Capacitor

When a conductor or a resistive material is connected across a charged capacitor, the capacitor will discharge. In a particular case, a very low resistance path (conductor) is connected across the capacitor with a switch. Before a switch is closed, the capacitor is charged to 4.5V, as indicated in figure 2-42 part (a). When the switch is closed, as shown in figure 2-42 part (b), the excess electrons on plate B moves through the circuit to plate A (indicated by the arrows); as a result of the current through the low resistance of the conductor, the energy stored by the capacitor is dissipated in the resistance of the conductor. The charge is neutralized when the number of free electrons on both plates are again equal. At this time, the voltage across the capacitor is zero, and the capacitor is completely discharged, as shown in figure 2-42 part (c).

Notice that the direction of electron flow during discharge is opposite to that during charging. It is important to understand that, ideally, *there is no current through the dielectric of the capacitor during charging or discharging because the dielectric is an insulating material.* There is current from one plate to the other only through the external circuit.

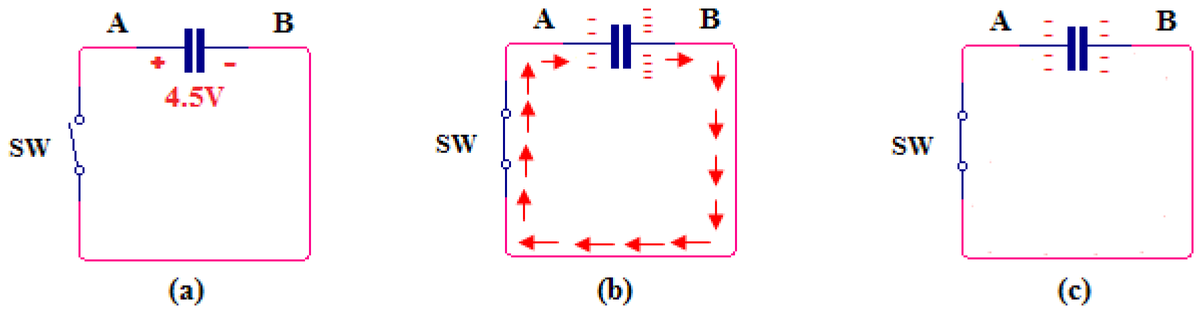


Figure 2-42: Discharging a capacitor; (a) Retains charge, (b) Discharging, (c) Uncharged

The *RC* Time Constant

In practical situation, there cannot be capacitance without some resistance in a circuit. It may simply be small resistance of a wire, or it may be a designed-in resistance. Because of this, the charging and discharging characteristics of a capacitor must always be considered with associated series resistance included. The resistance includes the element of *time* in the charging and discharging of a capacitor.

chap 2

When a capacitor charges or discharges through a resistance, a certain time is required for the capacitor to charge fully or discharge fully. The voltage across a capacitor cannot change instantanuously because a finite time is required to move charge from one point to another. The time constant of a series *RC* circuit determines the rate at which the capacitor charges or discharges.

The RC time constant is a fixed time interval that equal to the product of the resistance and capacitance in a series RC circuit.

The time constant is expressed in the units of seconds when resistance is in ohm and capacitance is in farads. It is symbolized by τ (Greek letter tau), and the formula is

$$\tau = RC \qquad \text{[Equation 2-17]}$$

When the resistance is increased, the charging current is reduced, thus increasing the charging time of the capacitor. When the capacitance is increased, the amount of charge increases; thus, for the same current, more time is required to charge the capacitor.

Experiment 2-4: Charging and discharging a capacitor

This experiment help to understand the charging and discharging processes of capacitor. Large value capacitors are required for this experiment to produce time constants slow enough so that the time of charging and discharging be visible on oscilloscope. Capacitors to be used are of the "electrolytic" type, and they are polarized as well. One terminal of each capacitor should be marked with a definite polarity sign. Usually capacitors of the size specified have a negative (-) marking or series of negative markings pointing toward the negative terminal. Very large capacitors are often polarity-labeled by a positive (+) marking next to one terminal. Failure to pay attention to proper polarity will almost surely result in capacitor failure, even with a source voltage as low as 4.5 volts.

When electrolytic capacitors fail, they typically explode, spewing caustic chemicals and emitting foul odors.

Build the circuit and observe voltage variation across the capacitor by changing the position of switch and notice how it increases and decreases slowly over time. You can "reset" the capacitor back to a voltage of zero by shorting across its terminals with a piece of wire.

Parts list

No	Item	Quantity
1	Oscilloscope	1
2	Resistor 1K	1
3	Breadboard	1
4	SPDT toggle switch	1
5	Electrolytic capacitor 470uF/50V	1
6	Wire 0.33mm^2	10cm
7	3x1.5V cell battery or use 4.5V DC supply	1

Working principle of the circuit

The circuit of figure 2-43 the charging time takes place when switch SW_1 is at position 1 (P_1) while discharging time takes place when the switch SW_1 is at position 2 (P_2). The time constant determines the duration of charging and discharging processes.

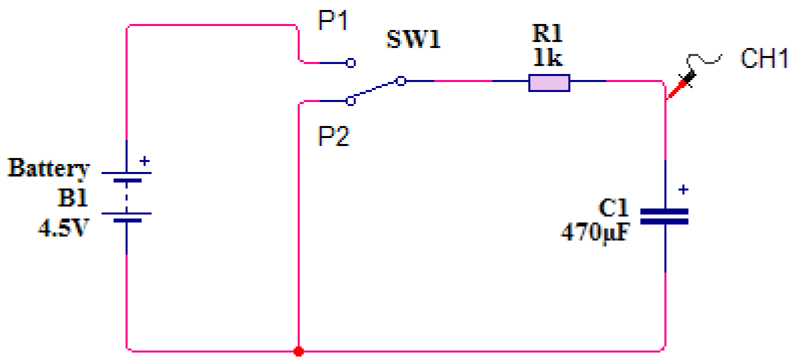


Figure 2-43: Circuit for charging and discharging a capacitor

The "time constant" (τ =RC) of a resistor capacitor circuit is calculated by taking the circuit resistance and multiplying it by the circuit capacitance. For a 1KΩ resistor and a 1000 μF capacitor, the time constant should be 1 second. This is the amount of time it takes for the capacitor voltage to increase approximately 63.2% from its present value to its final value which is the voltage of the battery. For the circuit of figure 2-43 it takes 0.4 second. Normally a one-second time constant doesn't provide much time to take voltmeter readings but for oscilloscope it is possible to change time/division. We can increase this circuit's time constant in two different ways: changing the total circuit resistance, and/or changing the total circuit capacitance.

Procedure

1. Connect the circuit as shown in figure 2-44 by using specified items
2. Check electrolytic capacitor's polarities
3. First discharge capacitor by shorting its terminal
4. Supply the circuit with 4.5V DC supply

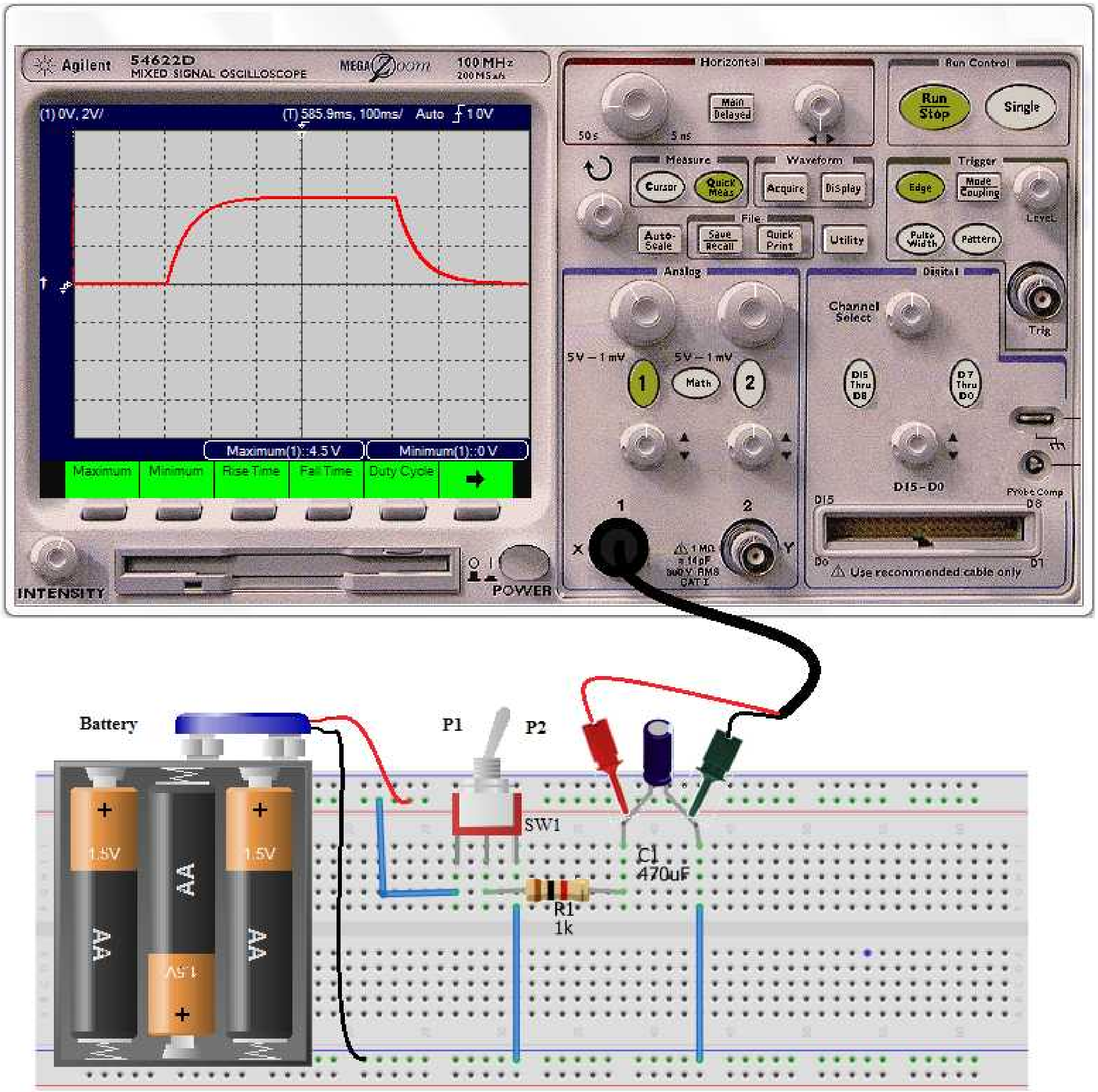


Figure 2-44: Circuit connection for charging and discharging a capacitor

5. Set oscilloscope to 2V/Div and at 100ms/Div, vertical and horizontal sensitivity respectively

6. Connect oscilloscope probes against capacitor's legs (black probe is at negative terminal of the battery)
7. Momentary change the position of switch SW_1 from P_2 to P_1 (charging processes). When the switch is changed from position P_2 to position P_1, the capacitor voltage tries to charge to the same voltage as the battery voltage, but unlike the resistor circuit, the capacitor voltage cannot immediately change to its maximum value, which would be (B_1= 4.5V).
8. As soon as the switch reaches position P_1, the circuit current rises very rapidly, as capacitor C_1 begins to charge. Although the voltage is still low, its rate of change is large and the voltage graph is initially very steep, showing that the voltage is changing in a very short time. As the capacitor charges, the rate of change of voltage slows and charge slows as the charging current falls. The curve describing the charging of the capacitor follows a recognizable mathematical law describing an exponential curve until the current is practically zero and the voltage across the capacitor is at its maximum.
9. Again change the position of switch SW_1 from P_1 to P_2 (discharging processes). If the switch is now changed to position P_2, the supply is disconnected and a short circuit is placed across *C* and *R*. This causes the capacitor to discharge through *R*. Immediately maximum current flows, but this time in the opposite direction to that during charging. Again an exponential curve describes the fall of this negative current back towards zero. The voltage also falls exponentially during this time, until the capacitor is fully discharged.

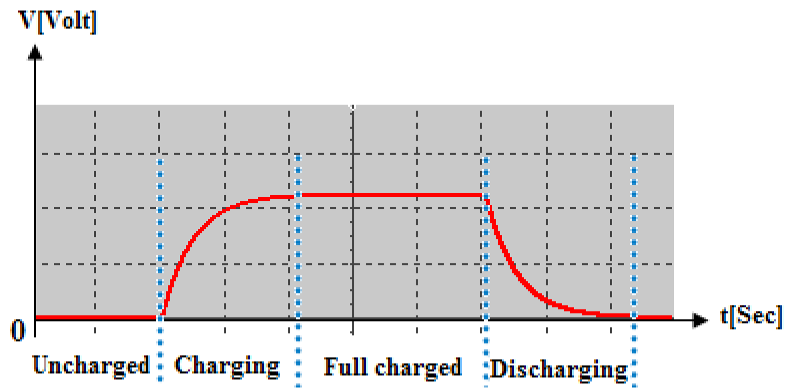


Figure 2-45: Charging and discharging curve of a capacitor

2. 4. 2. Capacitor in AC Circuits

Capacitor blocks constant dc. A capacitor passes ac with an amount of opposition, called capacitance reactance, which depends on the frequency of the ac.

The amount of charge stored by a capacitor determines the voltage across it. Therefore, the rate at which the charge is moved ($Q/t = I$) from one plate to the other determines the rate at which voltage changes. When the current is changing at its maximum rate (at the zero crossings), the voltage is at its maximum value (peak). When the current is changing to its minimum rate (zero at the peak), the voltage is at its minimum value (zero). This phase relationship is illustrated in figure 2-46.

As you can see, the current peaks occur a quarter of a cycle before the voltage peaks. Thus, the current leads the capacitor voltage by 90^o.

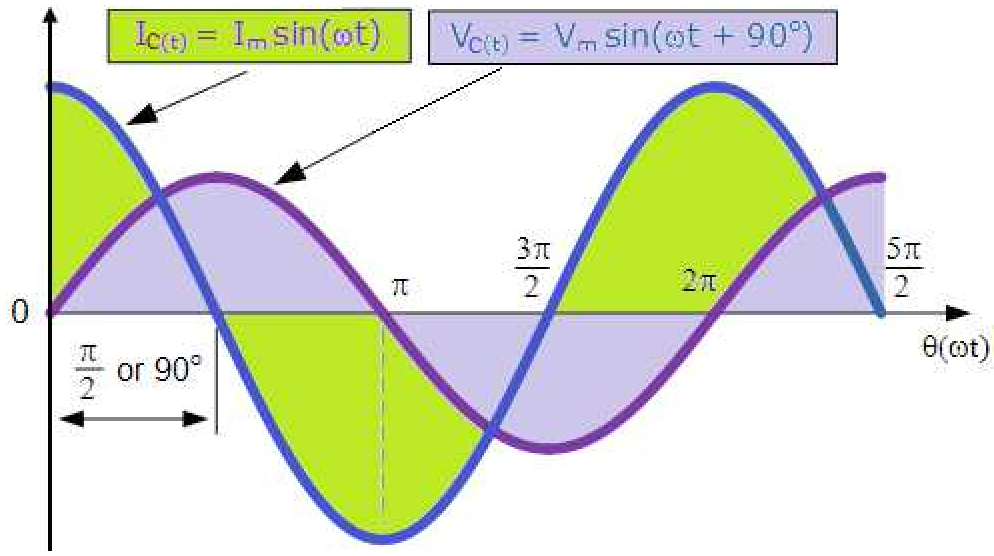


Figure 2-46: Capacitor voltage and current phase relationship. The current is always leading the capacitor voltage by 90°

Capacitance reactance, Xc

For better understanding the capacitive reactance of capacitor let's primarily pass to the experiment where a capacitor is is supplied by a constant voltage while increasing and decreasing the frequency.

Experiment 2-5: Behavior of a capacitor at constant voltage and variable frequency

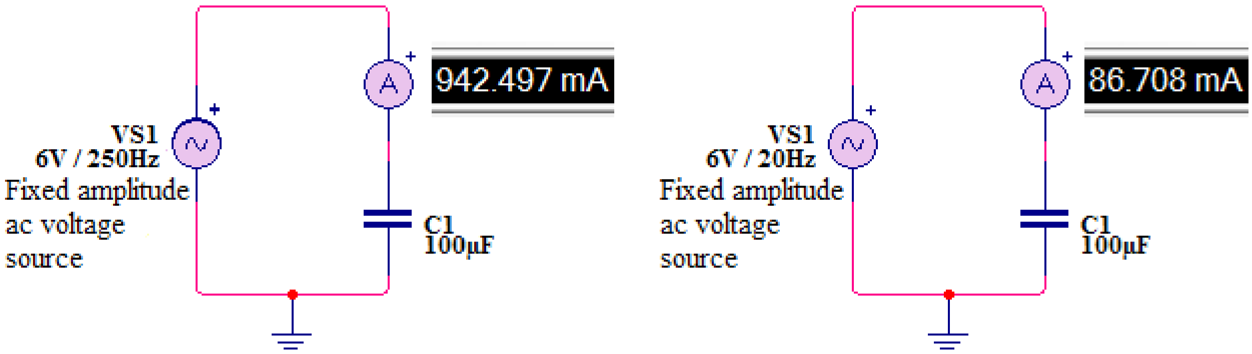
Part list

No	Item	Quantity
1	Function generator	1
2	Breadboard	1
3	Capacitor 100uF/50V	1
4	Ammeter	1
5	Wire 0.33mm^2	10cm

From the circuits of the figure 2-47, a capacitor is shown connected to a sinusoidal voltage source. When the source voltage is held at a constant amplitude value and its frequency is increased, the amplitude of the current increases. Also, when the frequency of the source is decreased, the current amplitude decreases.

Procedure

1. Connect a 220nF (no polarized) capacitor in series with ammeter and signal generator as shown in the circuit connection figure 2-48.
2. Set multimeter in amp's range 1A or 2A scale if provided.
3. Set the function generator to output a 6V rms sine wave with adjustable frequency.



(a) (b)

Figure 2-47: The current in a capacitive circuit varies directly with the frequency of the source voltage; (a) Current increases when the frequency increases, (b) Current decreases when the frequency decreases

4. Change the frequencies by using the knob shown below; by at first turning in clockwise the knob, with this by increasing the frequency and at the second turning the knob in anticlockwise to decrease the frequency and by observing the ammeter reading.

5. As frequency increases, the current increases to around 1A and when the frequency decreases the current decreases to around 0A

When the frequency of the voltage increases, its rate of change also increases. Now, if the rate of which the voltage is changing increases, the amount of charge moving through the circuit in a given period of time must also increase. More change in a given period of time means more current. For example, a tenfold increase in frequency means that the capacitor is charging and discharging 10 times as much in a given time interval. The rate of charge movement has increased 10 times. This means the current has increased by 10 because $I = Q/t$

An increase in the amount of current with a fixed amount of voltage indicates that opposition to the current has decreased.

Therefore, the capacitor offers opposition to current, and that opposition varies inversely with frequency.

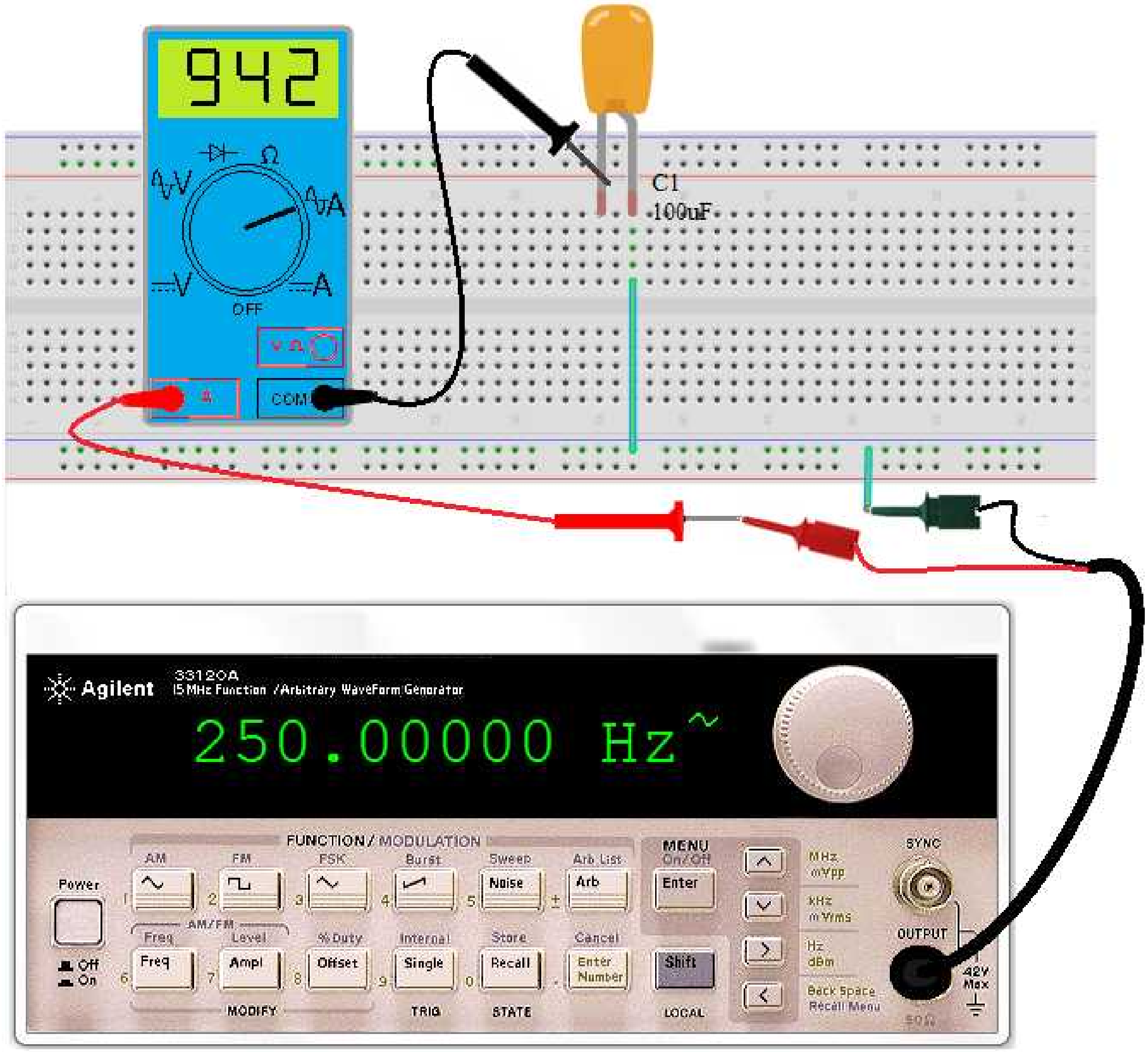


Figure 2-48: Behavior of a ca pacitor at constant voltage and variable frequency circuit connection

The opposition to sinusoidal current in a capacitor is called capacitive reactance

The symbol of capacitive reactance is Xc, and its unit is the Ohms (Ω).

You have just seen that the frequency affects the opposition to current (capacitive reactance) in a capacitor. Now let's see how the capacitance (*C*) itself affects the reactance. The figure 2-49 in part (a) shows that when a sinusoidal voltage with a fixed amplitude and a fixed frequency is equal to 1uF capacitor, there is a certain amount of alternating current. When the capacitance value is increase to 2uF, the current increases, as shown in the figure 2-49 part (b). Thus, when the capacitance is increased, the opposition to current (capacitive reactance) decreases. Therefore, not only is the capacitive reactance inversely proportional to frequency, but it is also inversely proportional to capacitance. This relationship can be stated as follows:

Xc is proportional to 1/*f*C.

It can be proven that the constant of proportionality that relates Xc to 1/*f*C is 1/2π. Therefore, the formula for capacitive reactance (Xc) is

$$X_C = 1/2\pi f C \qquad \text{[Equation 2-18]}$$

Xc is in Ohms where *f* is in hertz and *C* is in farads. The 2π term comes from the fact that, as you learned, a sine wave can be described in terms of rational motion, and one revolution contains 2π radians.

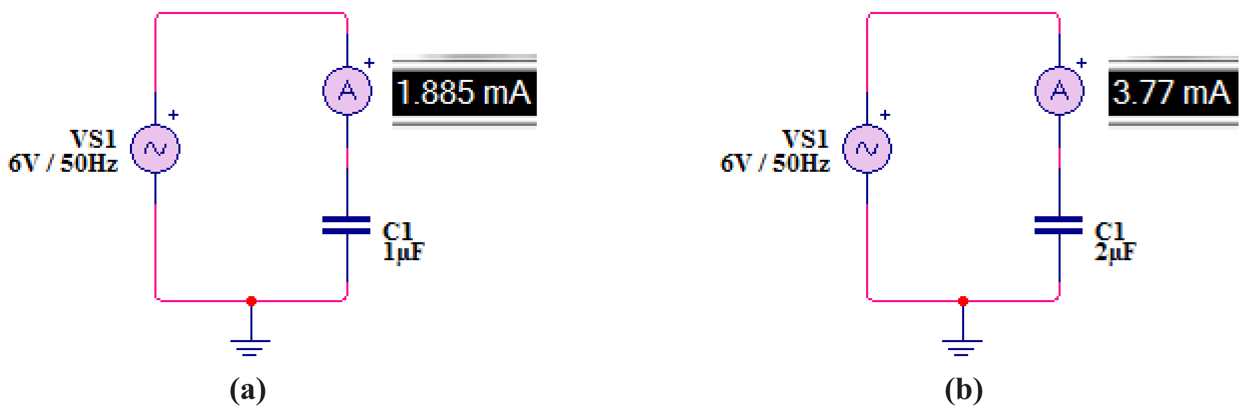


Figure 2-49: For a fixed voltage and fixed frequency, the current varies directly with the capacitance value; (a) Less capacitance, less current; (b) More capacitance, more current

Procedure

1. Connect the circuit as shown in the figure 2-50
2. Set multimeter in amp's range 20mA scale
3. Set function generator to produce 6VAC at 50Hz
4. Connect the neutral of the function generator to ground bus of the breadboard (3) and by using jumper wires, the three leads of the capacitor C_1, C_2 and C_3 are interconnected to ground
5. Connect the multimeter's red lead to the line output of function generator and the black lead of the multimeter to the lead of the capacitor C_1 (2) and then observe the ammeter reading. The ammeter reading is about 1.88mA.
6. Move the black lead of multimeter to the leads of capacitor C_2 parallel to C_3 (2) and again observe the ammeter reading. The ammeter reading is about the twice the previous reading.

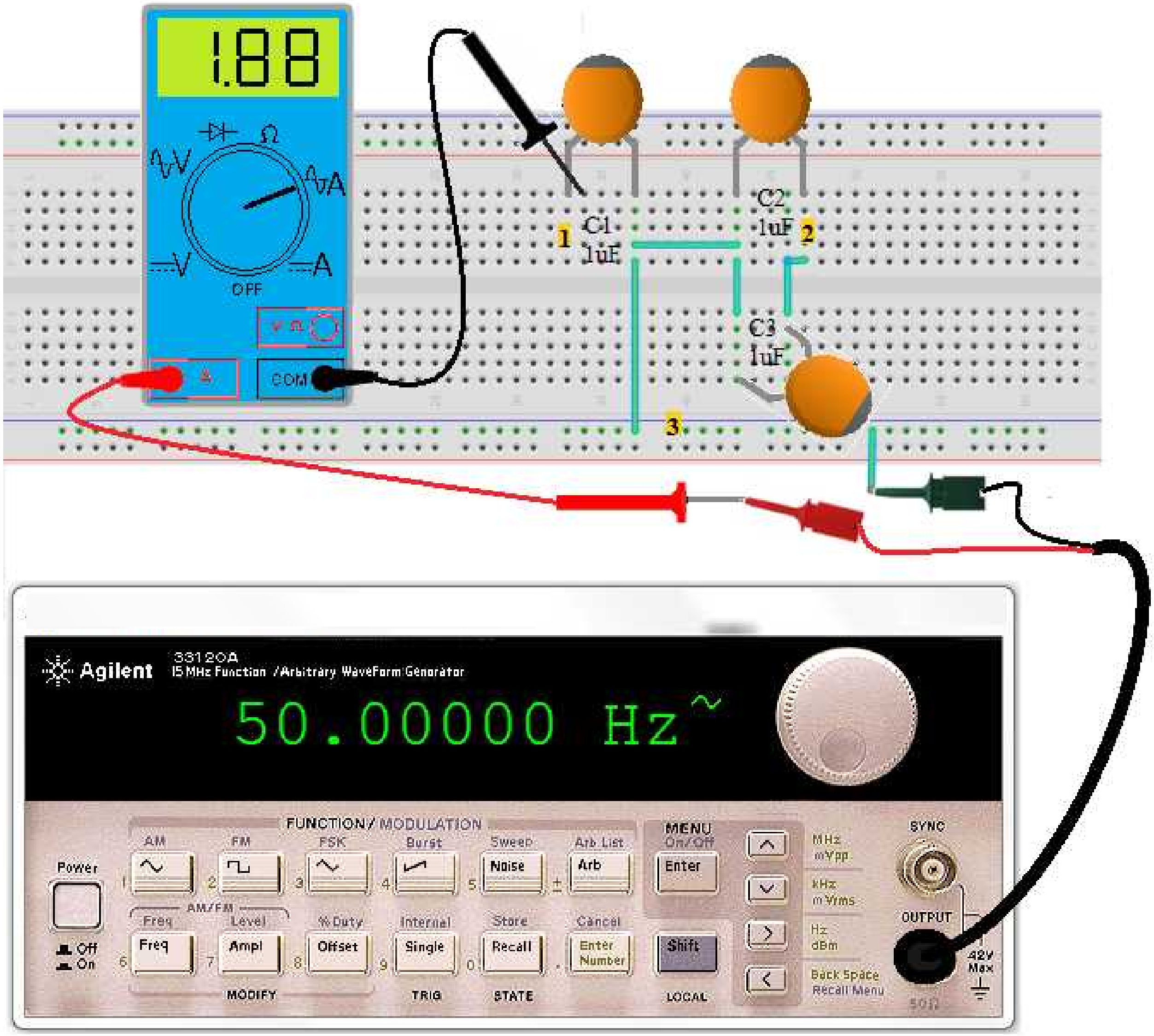


Figure 2-50: For a fixed voltage and fixed frequency, the current varies directly with the capacitance value

Capacitor Applications

Capacitors are widely used in electrical and electronics applications. If you pick up any circuit board, open any power supply, or look inside any piece of electronic equipment, chances are you will find capacitors of any type or another. These components are used for a variety of purpose in both dc and ac applications.

- **Electrical Storage**

One of the applications of a capacitor is as backup voltage source for low power circuits such as certain type of semiconductor memories in computers. This particular application requires a very high capacitance value and negligible leakage.

A storage capacitor is connected between the dc power supply input to the circuit and ground. Whenever the circuit is operating from its normal power supply, the capacitor remains fully charged to the dc power supply voltage. If the normal power source is disrupted, effectively removing the power supply from the circuit, the storage capacitor temporarily becomes the power source for the circuit. The length of time that a capacitor can provide sufficient power to a circuit depends on the capacitance and the amount of current drawn by the circuit.

- **Power Supply Filtering**

A basic dc power supply consists of a circuit known as a *rectifier* followed by a filter. Both half-wave and full-wave rectifier voltage are dc because even though they are changing, they do not alternate polarity. To be useful for powering electronic circuits, the rectified voltage must be changed to a constant dc voltage because all circuits require constant power. When connected to the rectifier output, the *filter* nearly eliminates the fluctuations in the rectified voltage and provides a smoothing constant-value dc voltage to the load, which is the electronic circuit. Thus, the capacitors are used as filters in dc power supplies because their ability to store electrical charges.

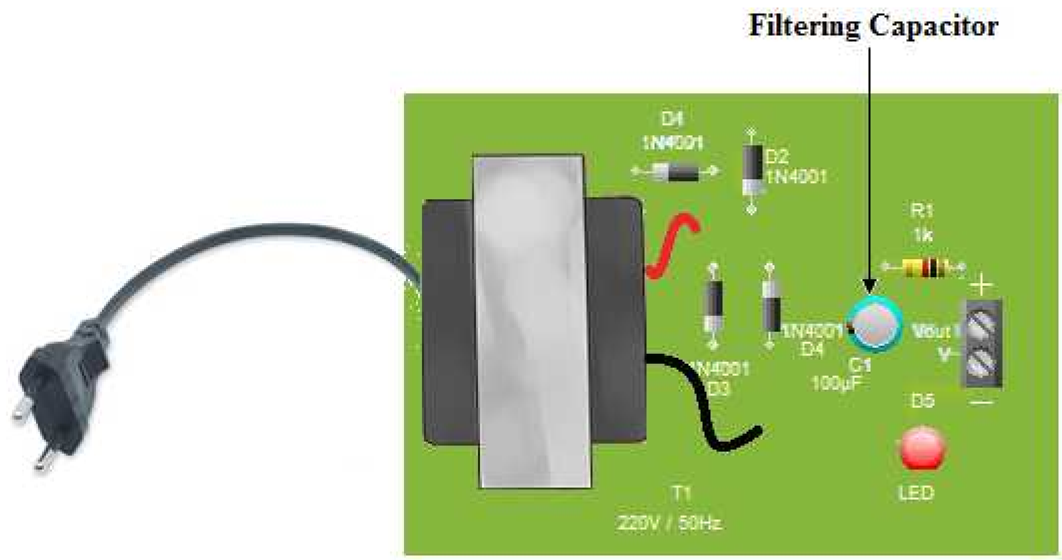


Figure 2-51: Example of typical board of power supply containing a filtering capacitor

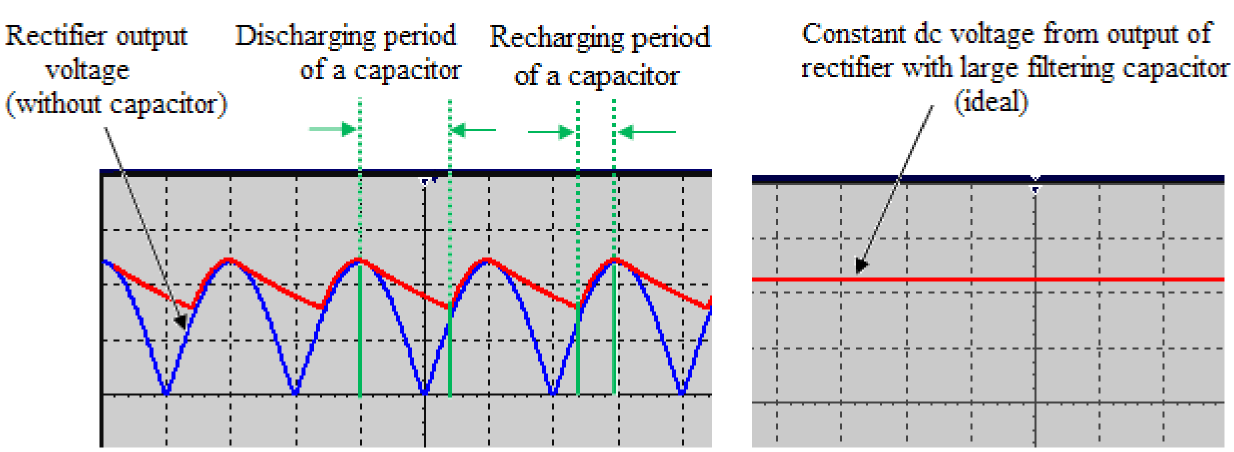


Figure 2-52: Basic operation of a power supply filter capacitor

- **DC Blocking and AC Coupling**

Capacitors are commonly used to block the constant dc voltage in one part of a circuit from getting to another part. As an example a capacitor is connected between two stages of an amplifier to prevent the dc voltage at the output of stage 1 from affecting the dc voltage at the input of stage 2. This type of capacitor is known as a *coupling capacitor*, which ideally appear as an open to dc and short to ac.

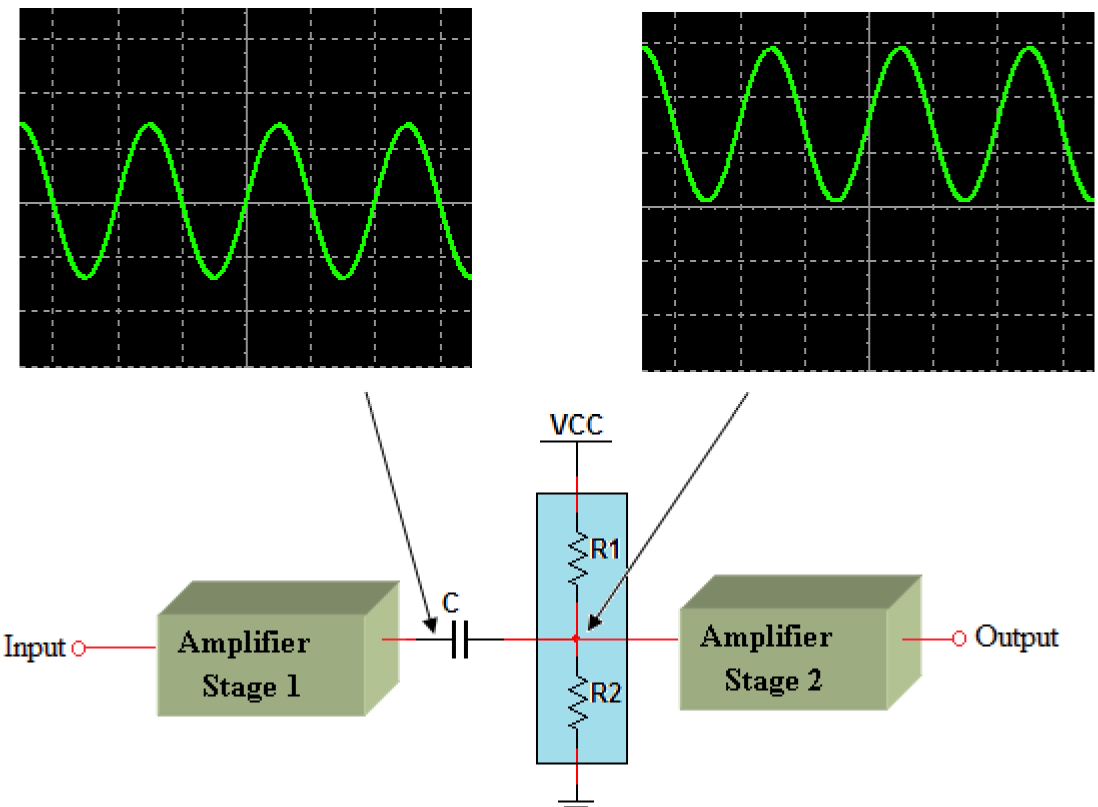


Figure 2-53: An application of a capacitor used to block dc and couple ac in an amplifier

- **Bypassing**

Bypass capacitors are used to bypass an ac voltage around a resistor in a circuit without affecting the dc voltage across the resistor.

- **Power Line Decoupling**

Capacitors connected from the dc supply voltage line to ground are used on circuit boards to decouple unwanted voltage transients or spikes that occur on the dc supply voltage because of fast switching digital circuits.

- **Signal Filters**

Filters are used for selecting one ac signal with a certain specified frequency from a wide range of signals with many different frequencies or for selecting a certain bands of frequencies and eliminating all others. A common example of this application is in radio and television receivers where it is necessary to select the signal transmitted from a given station and eliminating or filtering out the signals transmitted from all other station in the area.

When you tune your radio or TV, you are actually changing the capacitance in the tuner circuit (which is a typical of filter) so that only the signal from the station of channel you want passes through to the receiver circuitry.

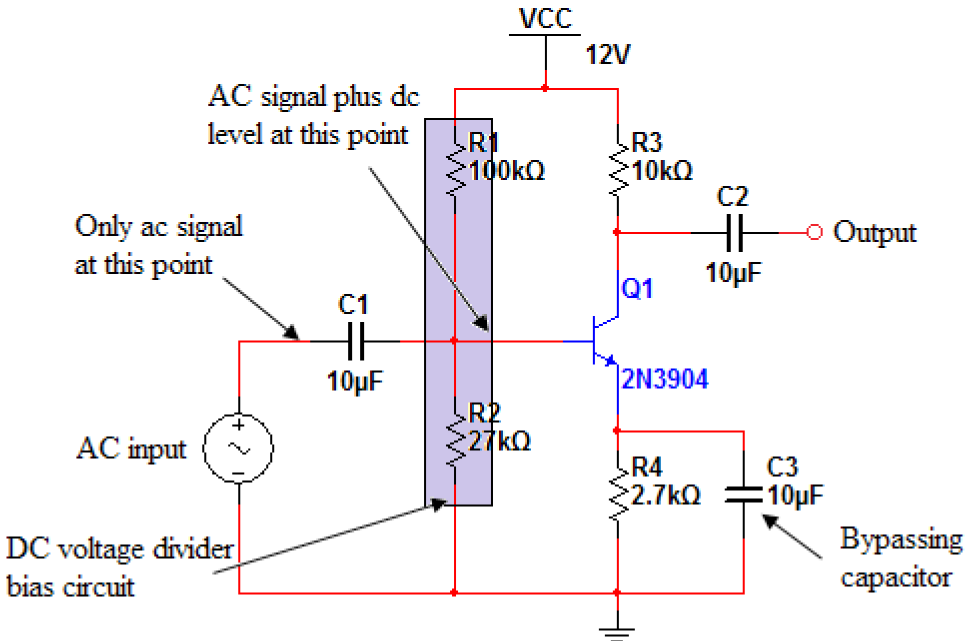


Figure 2-54: Capacitively coupled amplifier

- **Timing Circuits**

Another important area in which capacitors are used is in timing that generates specified time delays or produce waveforms with certain specific characteristics. The charging time of a capacitor can be used as a basic time delay in various types of circuits.

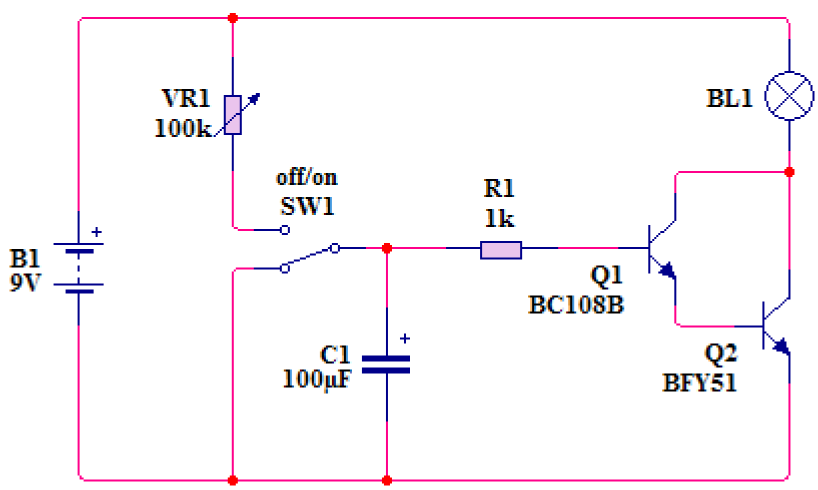


Figure 2-55: Example of delayed switch

- **Computer Memories**

Dynamic computer memories use capacitors as the basic storage element for binary information, which consists of two digits, 1 and 0. A charged capacitor can represent a stored 1 and a discharged capacitor can represent a stored 0.

2. 5. The Basic Inductor

An *inductor* is a passive electrical component formed by a coil of wire and which exhibits the property of inductance.

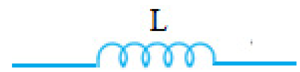


Figure 2-56: Symbol of inductor

Inductance

When there is current through an inductor, an electromagnetic field is established. When the current changes, the electromagnetic field also changes. An increase in current expands the field, and a decrease in current reduces it. Therefore, a changing current produces a changing electromagnetic field around the inductor (also known as *coil* and in some applications, *choke*). In turn, the changing in electromagnetic field causes an *induced voltage* across the coil in direction to oppose the change in current. This property is called *self-inductance* but is usually referred to as simply *inductance*, symbolized by *L*.

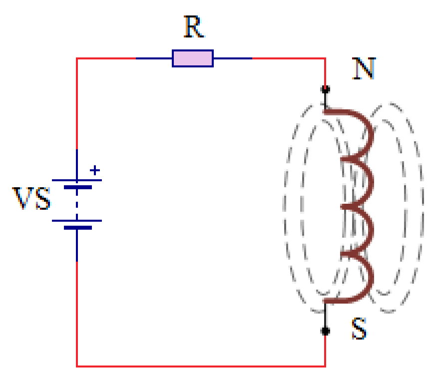


Figure 2-57: A current through a coil creates a three-dimensional electromagnetic field. The resistor limits the current

Inductance is a measure of coil's ability to establish an induced voltage as result of change in its current, and that induced voltage is in direction to oppose that change in current.

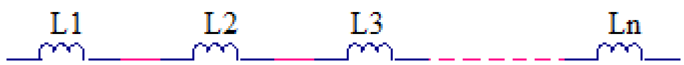
The unit of inductance: The *henry*, symbolized by H, is the basic unit of inductance. By definition, the inductance of a coil is one henry when current through the coil, changing at the rate of one ampere per second, induces one volt across the coil. The henry is a large unit, so in practical applications, millihenry (mH) and microhenry (µH) are more common units.

Energy stored: As inductor stores energy in magnetic field created by the current, the energy stored is expressed as

$$W = 1/2LI^2$$

Series Inductors

When inductors are connected in series, the total inductance increases. When inductors are connected in series, the total inductance, L_T, is the sum of the individual inductances. The formula for L_T is expressed in the following equation for general case of n inductors in series.

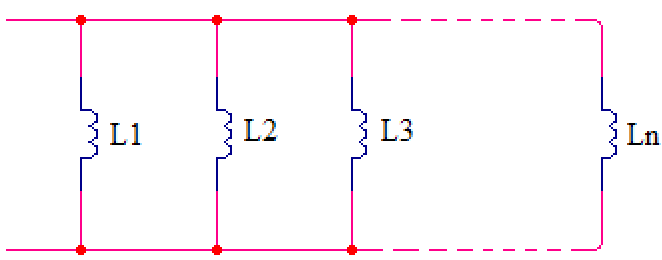


$$L_T = L_1 + L_2 + L_3 + \ldots + L_n \qquad \text{[Equation 2-19]}$$

Notice that the calculation of total inductance in series is analogous to the calculation of total resistance in series and total capacitance in parallel.

Parallel Inductors

When inductors are connected in parallel, the total inductance decreases. When inductors are connected in parallel, the total inductance is less than the smallest inductance. The general formula states that the reciprocal of the total inductance is equal to the sum of the reciprocals of the individual inductances.



$$1/ L_T = 1/ L_1 + 1/ L_2 + 1/ L_3 + \ldots + L_n \qquad \text{[Equation 2-20]}$$

Notice that this calculation of total inductance in parallel is analogous to the calculation of total parallel resistance and total series capacitance.

Applications of Inductors

The primary use for inductors is in filtering circuits, resonance circuits and for current limiting. An inductor can be used in circuits to block or reshape AC current or a range of AC frequencies, and in this role an inductor can be used to "tune" a simple radio receiver or various types of oscillators. It can also protect sensitive equipment from destructive voltage spikes and high inrush currents.

2. 5. 1. Inductor in DC Circuits

Energy is stored in the electromagnetic field of an inductor when it is connected to a dc voltage source. The buildup current through the inductor occurs in a predictable manner, which is dependent on both the time constant determined by the inductance and the resistance in a circuit.

When an inductor is connected to a circuit with dc source, two processes, which are called "storing" and "decaying" energy, will happen in specific conditions. For the circuit below, figure 2-58, the Inductor is connected to the dc source. When the switch is at position P_1, the sudden increase of current in the inductor produces a self induced electromotive force, V_{emf}, opposing the current change. This appears as a voltage across the inductor, $V_L = -V_{emf}$. This $-V_{emf}$ will slow down the current change, and in turn, the slowdown of the current change, will make V_L become smaller. When the current becomes stable, the inductor creates no more opposition and V_L becomes zero, the storage phase is over.

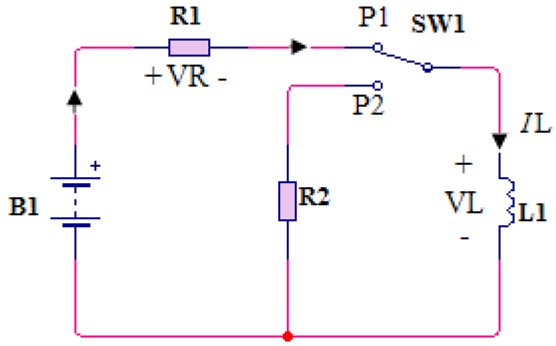


Figure 2-58: Storing and decaying of energy of inductor

An Inductor is equivalent to a short circuit to direct current, because once the storage phase has finished, the current, I_L, that flows through it is stable, $I_L = V / R$, no self induced e.m.f. is produced and V_L is zero. The inductor acts like an ordinary connecting wire, its resistance is zero. The current I_L through an inductor cannot change abruptly.

When the switch changes position from P_1 to P_2, an inductor is disconnected from the Power Supply, V_L reverses polarity and drops instantaneously from zero to a negative value, but I_L maintains the same direction and magnitude. The energy stored in the inductor decays through the resistor R_2. V_L rises gradually to zero and I_L drops gradually to zero.

In this circuit, the resistance of R_1 and R_2 affect the storing rate and the decaying (discharging) rate of the inductor respectively.

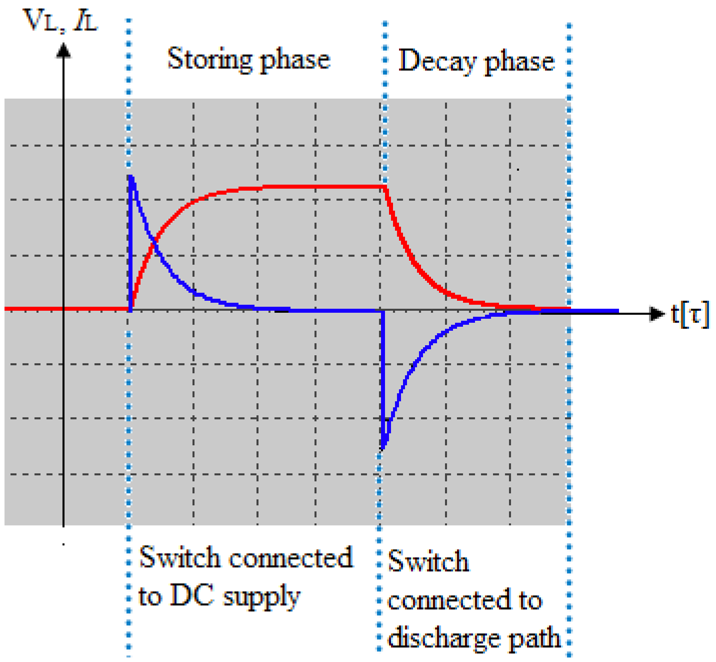


Figure 2-59: The voltage V_L and current I_L during the storage phase and discharge (decay) phase

The RL Time Constant

The quotient of inductance *L* and resistance *R* is called the time constant τ, which characterizes the rate of energy storage and energy decay in the inductor.

$$\tau = L/R \qquad \text{[Equation 2-21]}$$

Where τ is in seconds when inductance *L* is in henries and resistance *R* is in ohms.

The larger the resistance, the smaller the time constant, the faster the inductor stores the energy and decays the energy, and vice versa.

2. 5. 2. Inductor in AC Circuits

Inductors do not behave the same as resistors. Whereas resistors simply oppose the flow of electrons through them (by dropping a voltage directly proportional to the current), inductors oppose changes in current through them, by dropping a voltage directly proportional to the rate of change of current. In accordance with Lenz's Law, this induced voltage is always of such a polarity as to try to maintain current at its present value. That is, if current is increasing in magnitude, the induced voltage will "push against" the electron flow; if current is decreasing, the polarity will reverse and "push with" the electron flow to oppose the decrease. This opposition to current change is called reactance, rather than resistance.

Expressed mathematically, the relationship between the voltage dropped across the inductor and rate of current change through the inductor is as such:

$$e = di/dt$$

The expression *di/dt* is one from calculus, meaning the rate of change of instantaneous current (i) over time, in amps per second. The inductance (L) is in Henrys, and the instantaneous voltage (e), of course, is in volts. Sometimes you will find the rate of instantaneous voltage expressed as "v" instead of "e" (v = L di/dt), but it means the exact same thing.

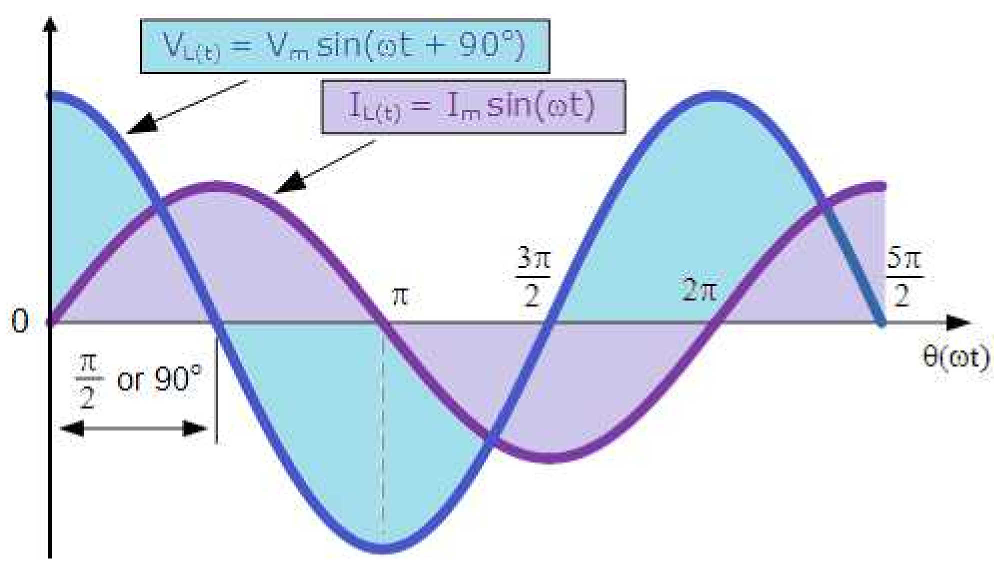


Figure 2-60: Inductor voltage and current phase relationship. The current is always lagging the inductor voltage by 90º

chap 2

Since the voltage dropped across an inductor is a reaction against the change in current through it, the instantaneous voltage is zero whenever the instantaneous current is at a peak (zero change, or level slope, on the current sine wave), and the instantaneous voltage is at a peak wherever the instantaneous current is at maximum change (the points of steepest slope on the current wave, where it crosses the zero line). This results in a voltage wave that is 90^o out of phase with the current wave.

The applied voltage reaches its maximum positive value a quarter ($1/4f$) of a cycle earlier than the current reaches its maximum positive value, in other words, a voltage applied to a purely inductive circuit "LEADS" the current by a quarter of a cycle or 90^o as shown in figure 2-60.

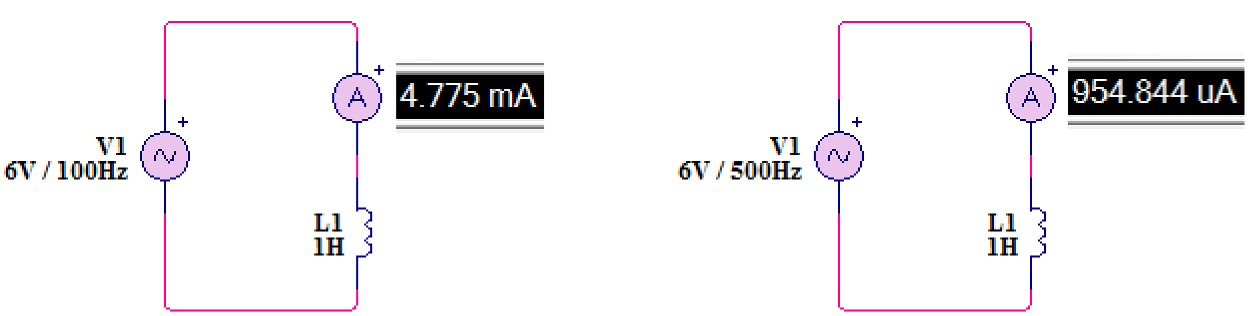
Experiment 2-6: Inductor in AC Circuits

Inductive reactance, XL

An inductor passes ac but with an amount of opposition that depends on the frequency of the ac supply. In circuits figure 2-61 an inductor is shown connected to a sinusoidal voltage source. When the source voltage is held at a constant amplitude value and its frequency is increased, the amplitude of the current decreases. Also, when the frequency of the source is decreased, the current amplitude increases.

Part list

No	Item	Quantity
1	Breadboard	1
2	Step-down power transformer (110-220V / 6 V)	1
3	6VAC source with different frequencies capability (function generator)	1
4	Wire (0.33mm^2 solid wire)	10cm
5	Ammeter (Digital multimeter set in amps range)	1



(a) (b)

Figure 2-61: Behavior of an inductor on changing frequency; (a) Current decreases when the frequency increases, (b) Current increases when the frequency decreases

When the frequency of the source voltage increases, its rate of change also increases, as you already know. Now, if the frequency of the source voltage is increased, the frequency of the current also increases. According to the Faraday's and Lenz's Laws, this increase in frequency induces more voltage across the inductor in a direction to oppose the current and causes it to decrease in amplitude. Similarly, a decrease in frequency will cause an increase in current.

A decrease in the amount of current with an increase in frequency for a fixed amount of voltage indicates that opposition to the current has increased. Thus, the inductor offers opposition to current, and that opposition varies directly with frequency.

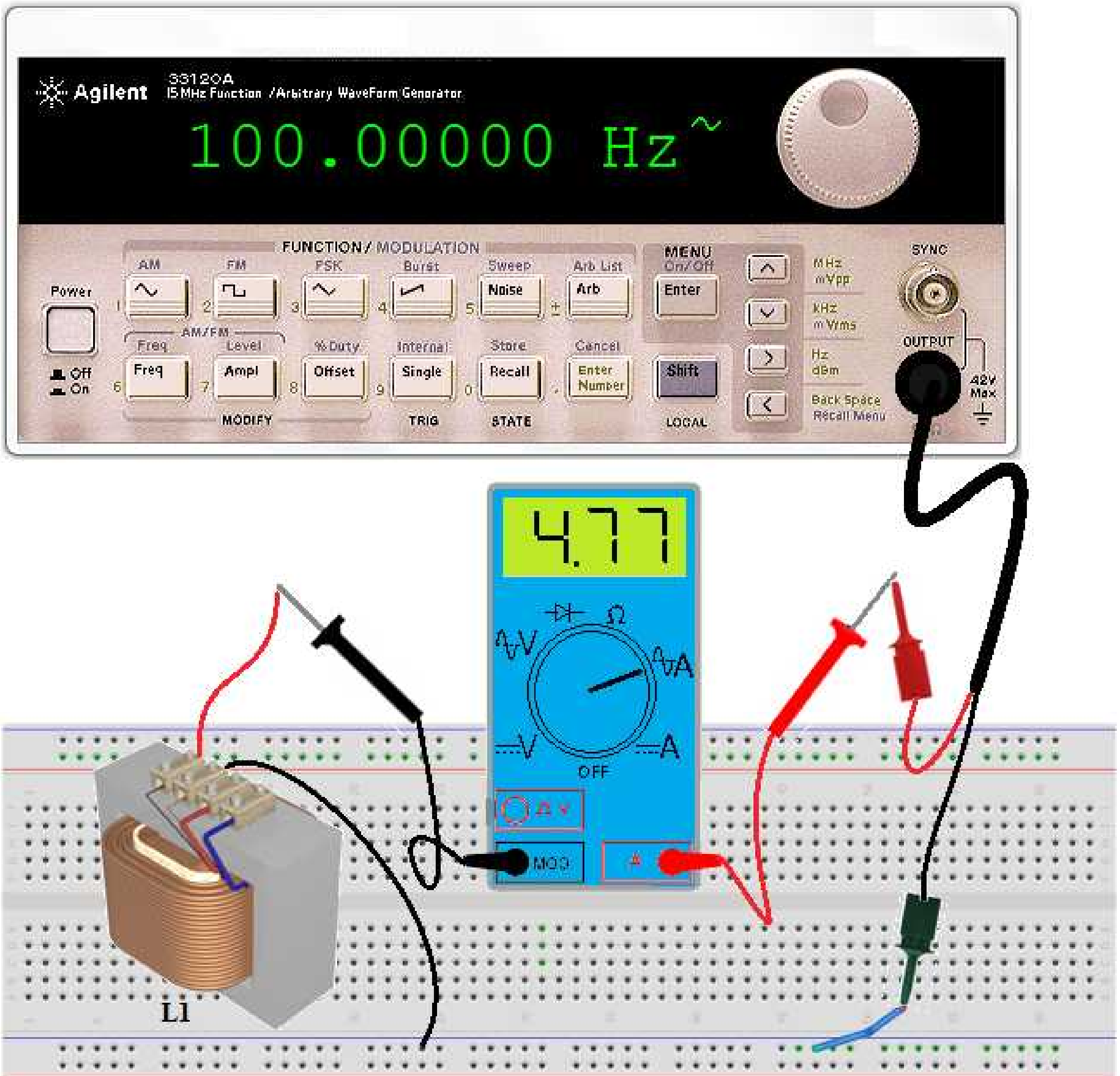


Figure 2-62: Behavior of an inductor on changing frequency while voltage is kept constant, circuit connection

Procedure

1. Preferably use power transformer as an inductor, with only one winding connected. The unused winding should be left open. A simple iron core, single-winding inductor may also be used, but such inductors are more difficult to obtain than power transformers and have

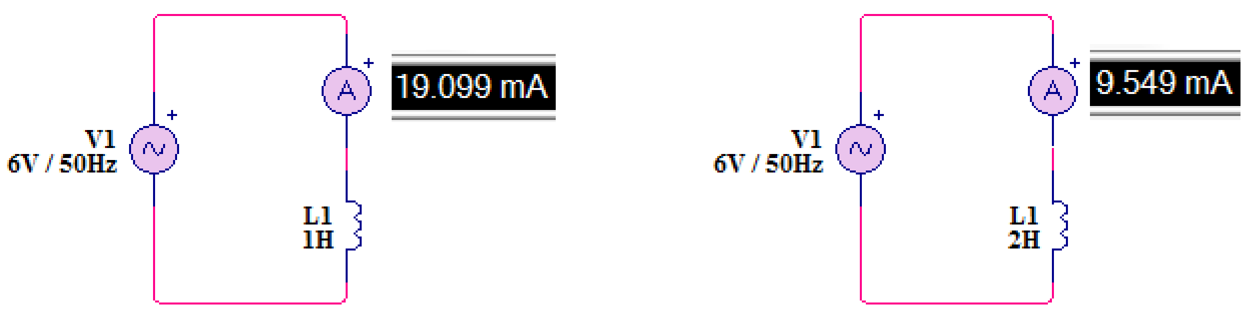
a low internal resistance which can be neglected and cause damage to the ammeter. Power transformers have primary (110 volt) winding inductances of approximately 1H.

2. Connect the circuit as shown in the figure 2-62 (make sure the ammeter is in series with the transformer's inductance)
3. Set multimeter in amp's range 200mA scale
4. Set function generator to produce 6VAC, 100Hz and adjust the frequency up to 500Hz and observe the current amplitude on ammeter

Inductive reactance is the opposition to sinusoidal current in an inductor

The symbol of inductive reactance is XL, and its unit is Ohm (Ω).

You have just seen how frequency affects the opposition to current (inductive reactance) in an inductor. Now, let's see how the inductance, L, affects the reactance. The figure 2-63 in part (a) shows that when a sinusoidal voltage with a fixed amplitude and fixed frequency is applied to a 1H inductor, there is a certain amount of alternating current. When the inductance value increased to 2H, the current decreases as shown in figure 2-63 part (b).



(a) (b)

Figure 2-63: Inductive reactance of an inductor creates an opposition to sinusoidal current; (a) Less inductance more current, (b) More inductance less current

Thus, when the inductance is increased, the opposition to current (inductive reactance) increases. So not only is the inductance reactance directly proportional to the frequency, but it is also directly proportional to inductance. This relationship can be stated as follows:

XL is proportional to f_L

It can be proven that the constant of the proportionality is 2π, so the formula for inductive reactance (XL) is

$$XL = 2\pi f_L \qquad \text{[Equation 2-22]}$$

XL is in Ohms when frequency f is in hertz and L is in henries. As with capacitive reactance, the 2π term comes from the relationship of the sine wave to rotational motion.

Procedure

7. Preferably use power transformer as an inductor, with only one winding having center tapped connected. The unused winding should be left open. It is possible to have the simple iron core, single-winding inductors may be used instead by making sure that they have a desired inductance and not negligible internal resistance as well. Once a power transformer is used, assume that the primary corresponding to 110 volt winding has inductances of approximately 1H while the primary of 220 Volt winding has inductances of approximately 2H.
8. Connect the circuit as shown in the figure 2-64 (make sure the ammeter is in series with the transformer's inductance)
9. Set multimeter in amp's range 200mA scale
10. Set function generator to produce 6VAC at 50Hz
11. Connect the neutral of the function generator to the third wire (3) of the transformer
12. Connect the multimeter's red lead to the line output of function generator and the black lead of the multimeter to the second wire (2) of the transformer and then observe the ammeter reading. The ammeter reading is about 19.099mA.
13. Move the black lead of ammeter to the first wire (1) of the transformer and again observe the ammeter reading. The ammeter reading is about 9.549mA.

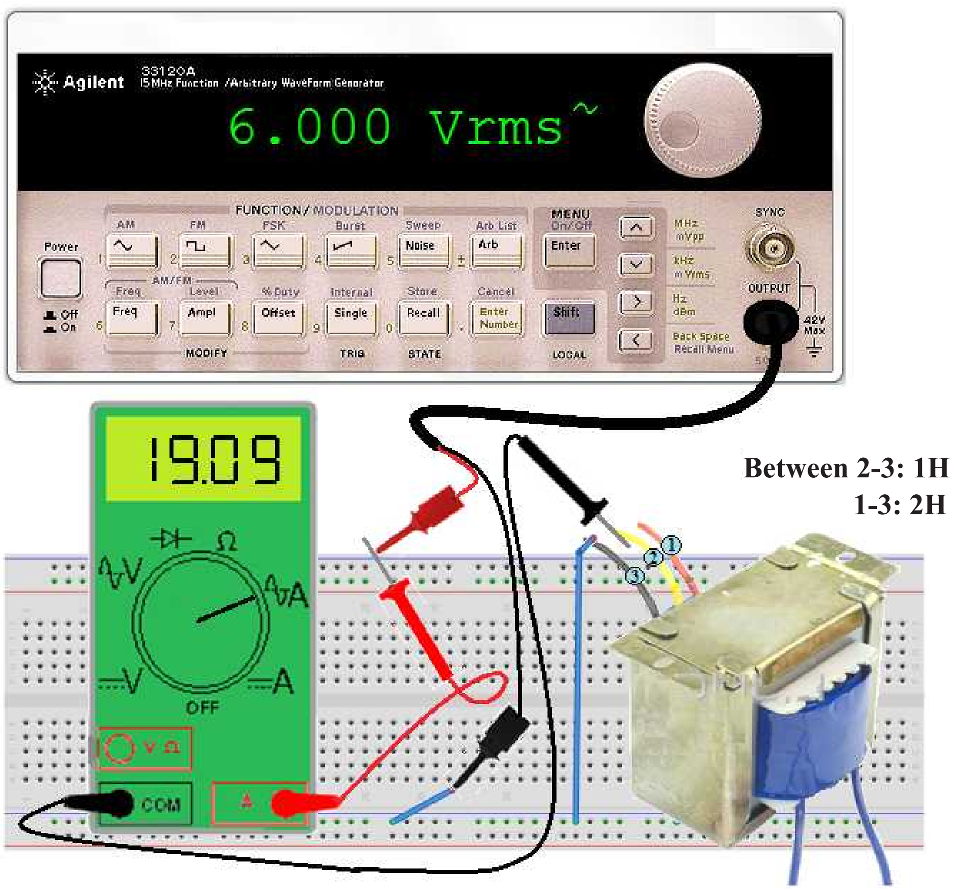


Figure 2-64: Inductive reactance of an inductor creates an opposition to sinusoidal current

Passive Filters and Applications

Chapter three

Objectives

After completing this chapter, you should be able to

- Describe some basic RC, RL and RLC applications
- Describe some characteristics of passive filters
- Design and implement RC low-pass and high-pass filters
- Design and implement RL low-pass and high-pass filters
- Design and implement series and parallel RLC resonant filters

Further reading

Study aids for this chapter are available at

- Williams, Arthur B & Taylor, Fred J (1995). Electronic Filter Design Handbook. McGraw-Hill.
- http://www.learnabout-electronics.org/ac_theory/filters81.php

The *filter* is frequency-selectivity circuit designed to modify, reshape or block all unwanted frequencies of an electrical signal from the input to output and accepts or passes only those signals with wanted frequencies. That is, it "filters-out" unwanted signals and an ideal filter will separate and pass sinusoidal input signals based upon their frequency.

Passive Filters are made up of passive components such as resistors, capacitors and inductors and have no amplifying elements (transistors, op-amps, etc) so have no signal gain, therefore their output level is always less than the input.

Filters are so named according to the frequency range of signals that they allow to pass through them, while blocking or "attenuating" the rest. The most commonly used filter designs are:

- Low-Pass filter
- High-Pass filter
- Band-pass filter
- Band-stop filter

3. 1. RC Circuit as a Filter

Series *RC* circuits exhibit a frequency-selective characteristic. Simple first order passive filters can be constructed by connecting together a single resistor and a single capacitor in series across an input signal, (V_{in}) with the output, (V_{out}) taken from the junction of these two components. Depending on which way around we connect the resistor and the capacitor with regards to the output signal determines the type of filter construction resulting in either a Low-Pass Filter or a High-Pass Filter.

3. 1. 1. RC Low-Pass Filter

The Low-Pass filter only allows low frequency signals from 0 Hz to its cut-off frequency, f_c point to pass while blocking those any higher.

A *low-pass filter circuit* is realized from a series *RC* circuit and by taking the output across the capacitor.

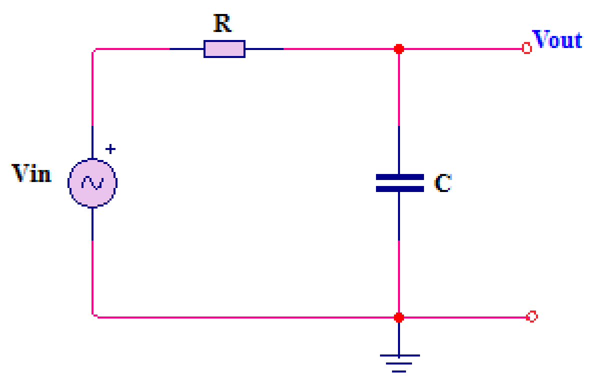


Figure 3-1: Low-Pass filter base circuit

As mentioned previously in the capacitive behaviour in ac circuits, the reactance of a capacitor varies inversely with frequency, while the value of the resistor remains constant as the frequency changes. At low frequencies the capacitive reactance, (Xc) of the capacitor will be very large compared to the resistive value of the resistor, *R* and as a result the voltage across the capacitor, V_c will also be large while the voltage drop across the resistor, V_R will be much lower. At high frequencies the reverse is true with V_c being small and V_R being large.

While the circuit above is that of an *RC* Low Pass Filter circuit, it can also be classed as a frequency variable potential divider circuit similar to the one we looked at in the resistors tutorial. In that tutorial we used the following equation to calculate the output voltage for two single resistors connected in series.

$$V_{out} = V_{in} * (R_2/R_1 + R_2)$$

Where: $R_1+R_2=R_T$, the total resistance of the circuit.

We also know that the capacitive reactance of a capacitor in an AC circuit is given as:

$$Xc = 1/2\pi f C \text{ [Ohm]}$$

Opposition to current flow in an AC circuit is called impedance, symbol Z and for a series circuit consisting of a single resistor in series with a single capacitor, the circuit impedance is calculated as:

$$Z = \sqrt{(R^2 + Xc^2)}$$

Then by substituting our equation for impedance above into the resistive potential divider equation gives us *RC* potential divider equation:

$$V_{out} = V_{in} * Xc / \sqrt{(R^2 + Xc^2)} = V_{in} * Xc/Z$$

Thus, by using the potential divider equation of two resistors in series and substituting for impedance we can calculate the output voltage of an *RC* Filter for any given frequency.

All low-pass filters are rated at a certain cut-off frequency. That is, the frequency above which the output voltage falls below 70.7% of the input voltage. This cut-off percentage of 70.7 is not really arbitrary, all though it may seem so at first glance. In a simple capacitive/resistive low-pass filter, it is the frequency at which capacitive reactance in ohms equals resistance in ohms. In a simple capacitive low-pass filter (one resistor, one capacitor), the cut-off frequency is given as:

$$f_{cutoff} = 1/2\pi RC$$

Experiment 3-1: *RC* Low-Pass filter

Design and examine a *low-pass filter circuit* based on series RC circuit that is realized by taking the output across the capacitor, just as in lag circuit.

Parts and materials

No	Item	Specification	Quantity
1	10VAC, 100Hz-20KHz supply	Agilent Function generator 33120A	1
2	Oscilloscope	54622D Agilent digital oscilloscope	1
3	Breadboard	390-pin Prototyping board	1
4	Connecting wire	22-gaude (0.33mm^2) solid wire	10cm
5	100Ω fixed resistor	100Ω, 1/2watt fixed resistor	1
6	1uF Capacitor	1uF, 16V no polarized capacitor	1

In terms of the filtering action of the series *RC* circuit, the variation in the magnitude of the output voltage as a function of frequency is important. The figure of the circuits below show the filtering action of a series *RC* circuit using a specific series of measurements in which the frequency starts at 100Hz and is increased in increments up to 20KHz. At each value of frequency, the output voltage is measured. As you can see, the capacitive reactance decreases as the frequency increases, thus dropping less voltage across the capacitor while the input voltage is held at a constant 10V throughout each step.

Let us vary the frequency from 100 Hz, 1 KHz, 10 KHz up to 20 KHz at a constant V_{in} of 10V.

At 100 Hz,

$$Xc = 1/2\pi f C$$
$$= 1/2*3.14*100*10^{-6} = 1.590 \text{ K}\Omega$$

$Z = \sqrt{(R^2 + Xc^2)}$
$= \sqrt{(100^2 + 1590^2)} = 1590.314 \approx 1590\ \Omega$
$V_{out} = V_{in} * Xc/Z$
$= 10V*1590/1590.314 = 9.98\ V$
$I = V/Z$
$= 10V/1590\Omega = 0.006289 \approx 6.29\ mA$

At 1 KHz,

$Xc = 1/2\pi f C$
$= 1/2*3.14*1000*10^{-6} = 159\ \Omega$
$Z = \sqrt{(R^2 + Xc^2)}$
$= \sqrt{(100^2 + 159^2)} = 188\ \Omega$
$V_{out} = V_{in} * Xc/Z$
$= 10V*159/188 = 8.46\ V$
$I = V/Z$
$= 10V/188\Omega = 53.2\ mA$

At 10 KHz,

$Xc = 1/2\pi f C$
$= 1/2*3.14*10000*10^{-6} = 15.9\ \Omega$
$Z = \sqrt{(R^2 + Xc^2)}$
$= \sqrt{(100^2 + 15.9^2)} = 101\ \Omega$
$V_{out} = V_{in} * Xc/Z$
$= 10V*15.9/101 = 1.57\ V$
$I = V/Z$
$= 10V/101\Omega = 99.0\ mA$

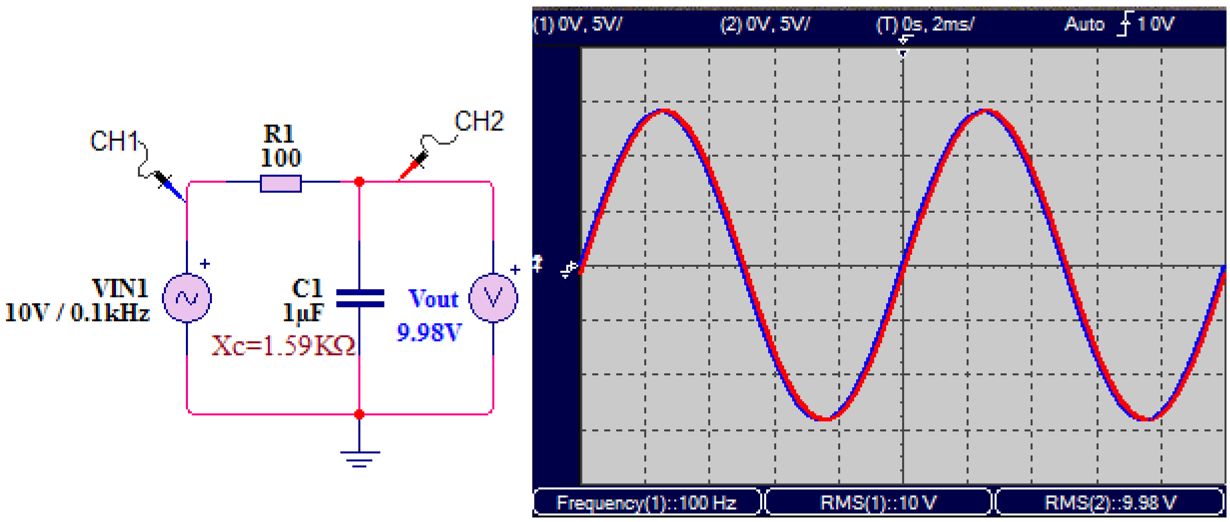
At 20 KHz,

$Xc = 1/2\pi f C$
$= 1/2*3.14*20000*10^{-6} = 7.96\ \Omega$
$Z = \sqrt{(R^2 + Xc^2)}$
$= \sqrt{(100^2 + 7.96^2)} \approx 100\ \Omega$
$V_{out} = V_{in} * Xc/Z$
$= 10V*7.96/100 = 0.79\ V$
$I = V/Z$
$= 10V/100\Omega \approx 100\ mA$

The table 3-1 summarizes the variations of the circuit parameters with frequency.

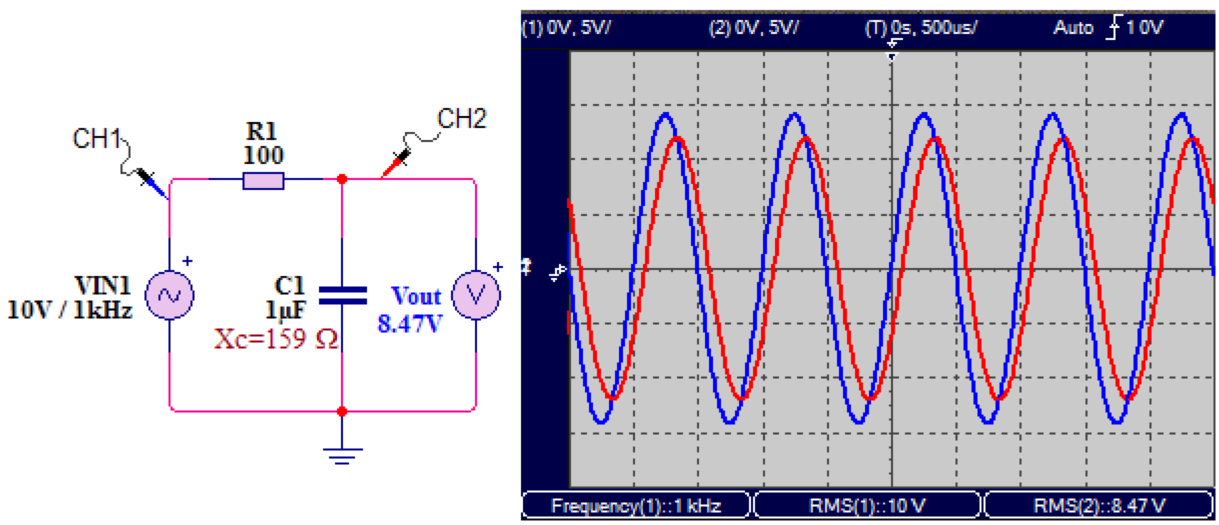
f (KHz)	Xc (Ω)	Z_{tot}(Ω)	I(mA)	V_{out}(V)
0.1	1,590	≈1,590	≈6.29	9.98
1	159	188	53.2	8.46
10	15.9	101	99.0	1.57
20	7.96	≈100	≈100	0.79

Table 3-1: Summarized the variations of the circuit parameters with frequency



(a)

(a) f = 0.1KHz, Xc = 1.59KΩ, V_{out} = 9.98V

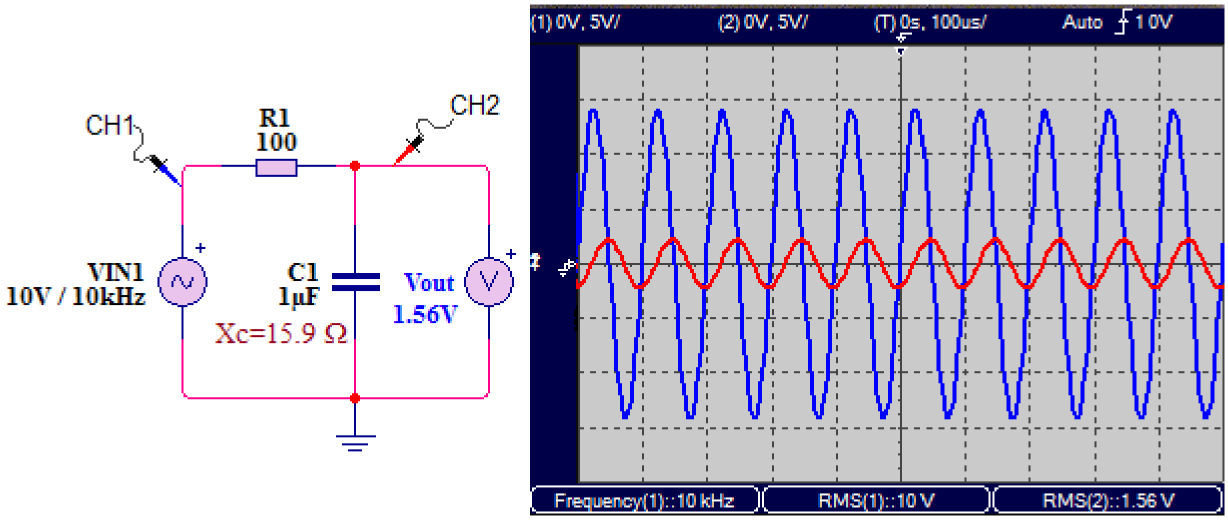


(b)

(b) f = 1KHz, Xc = 159Ω, V_{out} = 8.47V

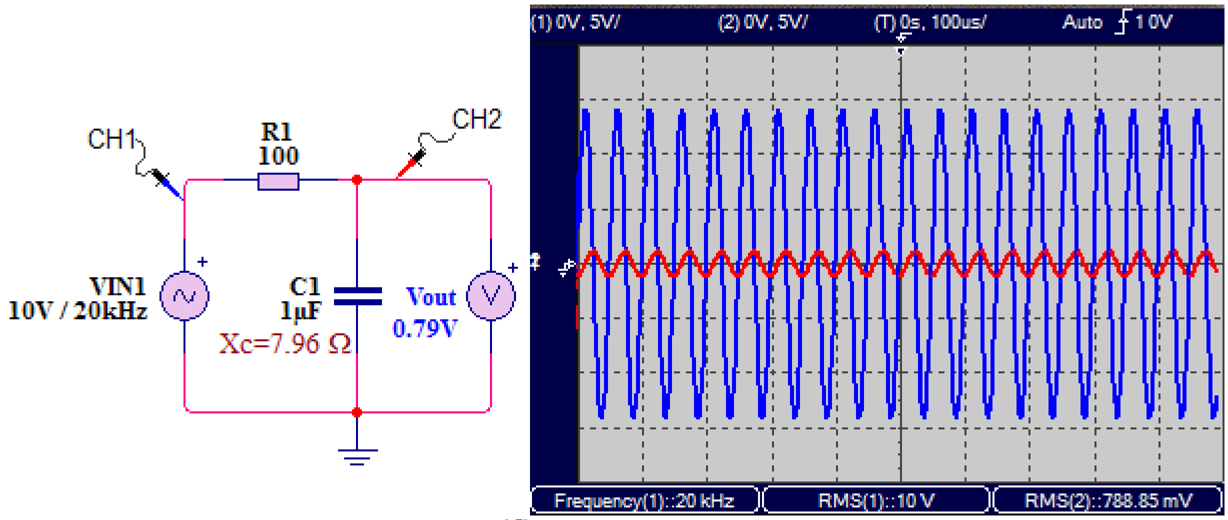
The *frequency response* of the low-pass *RC* filter circuit of figure 3-2, is shown in figure 3-3, where the measured values are plotted on a graph of V_{out} versus f, and a smooth curve is drawn connecting the points. This graph called a *response curve*, shows that the output voltage is greater at the lower frequencies and decreases as the frequency increases. The frequency scale is logarithmic.

The Bode Plot shows the frequency response of the filter to be nearly flat for low frequencies and all of the input signal is passed directly to the output, resulting in a gain of nearly 1, called unity, until it reaches its Cut-off Frequency point (f_c). This is because the reactance of the capacitor is high at low frequencies and blocks any current flow through the capacitor.



(c)

(c) f = 10KHz, Xc = 15.9Ω, V_{out} = 1.56V



(d)

(d) f = 20KHz, Xc = 7.96Ω, V_{out} = 0.79V

Figure 3-2: Low-pass Filter; (a), (b), (c) and (d) are examples of low-pass filtering action. As frequency increases, V_{out} decreases

After this cut-off frequency point the response of the circuit decreases to zero.

Any high frequency signals applied to the low pass filter circuit above this cut-off frequency point will become greatly attenuated, that is they rapidly decrease. This happens because at very high frequencies the reactance of the capacitor becomes so low that it gives the effect of a short circuit condition on the output terminals resulting in zero output.

Then by carefully selecting the correct resistor-capacitor combination, we can create a RC circuit that allows a range of frequencies below a certain value to pass through the circuit unaffected while any frequencies applied to the circuit above this cut-off point to be attenuated, creating what is commonly called a Low Pass Filter.

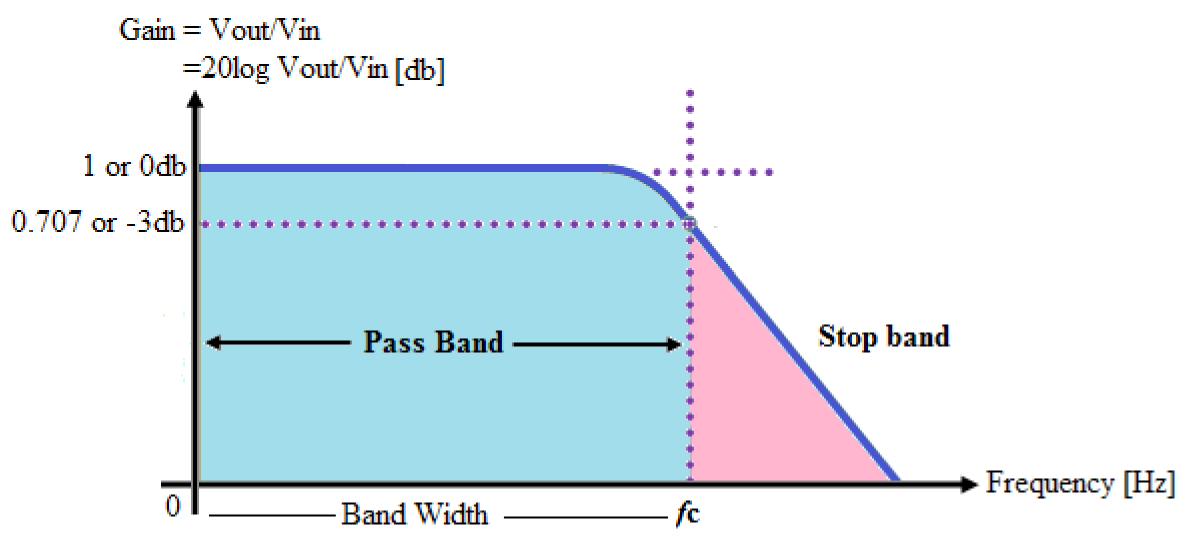


Figure 3-3: Response curve of a Low-pass filter

By plotting the networks output voltage against different values of input frequency, the Frequency Response Curve or Bode Plot function of the low pass filter circuit can be found, as shown in figure 3-4.

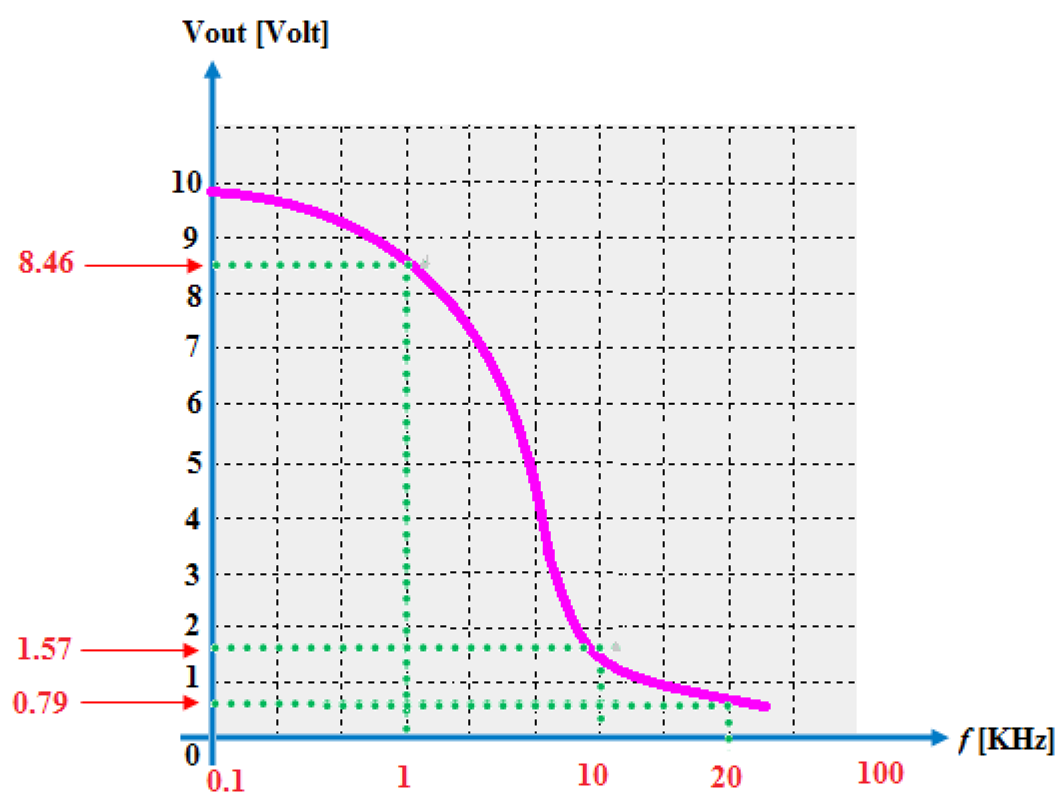


Figure 3-4: Response curve of a Low-pass filter against different values of input frequency

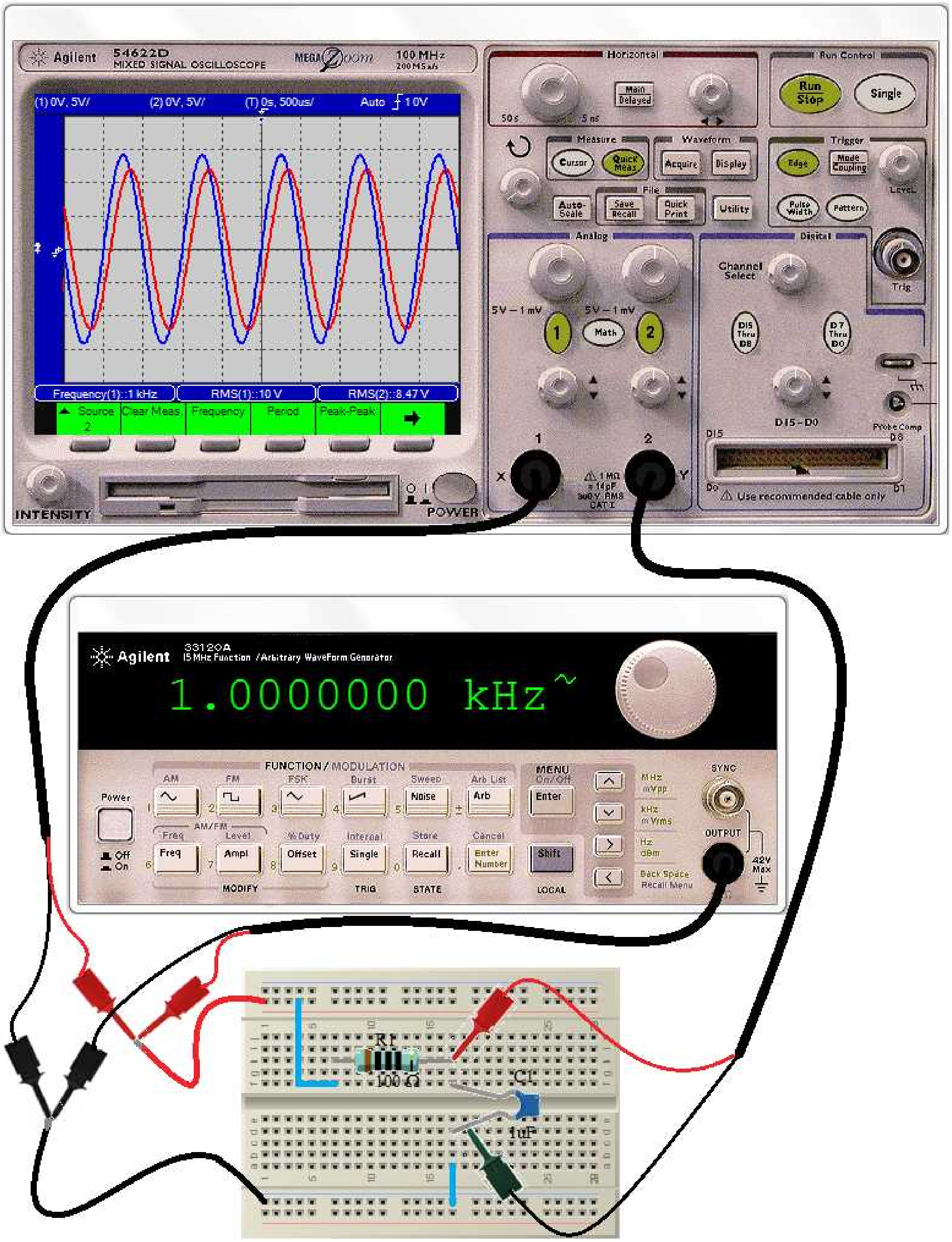


Figure 3-5: RC Low-pass filter, real world circuit connection

chap 3

Procedure

1. Choose the 100 Ω resistor from resistor pack and refer to color-code as the use of ohmmeter can give the value apart of the expected and choose 1uF capacitor.
2. Construct the low pass filter circuit as illustrated in circuit connection figure 3-5.
3. Set the function generator to output a sinusoid with amplitude of 10 V rms or 28.28V peak-to-peak and an initial frequency of 100 Hz.
4. The oscilloscope display may have a natural coarseness, which can be reduced by selecting the Average function. Try to use as few samples as possible to avoid long delays while moving from one display to another.
5. Push the AUTOSCALE button to display both channels. Reposition so that the 0 V is at the midline on the both channels and the waveforms appear to overlap. Adjust the Volts/division to 5V/division for both channels and to 2msec/division to display more than one complete period. At high frequencies, the amplitude of the filtered signal may be too small to be detected by the Auto-scale feature. You may need to set the Volts/division scale manually in order to see the signal. Again as you increase the frequency the period decreases, so you need to reduce the time/division. That is, 500 uSec/division at 1 KHz, 100 uSec/division at 10 KHz, and 50uSec/division or 100 uSec/division at 20 KHz.
6. Press the MEASURE button to determine the voltage RMS amplitudes for both channels, and record in the data sheet for the frequency of 100 Hz, 1 KHz, 10 KHz and 20 KHz. At least one full period of the waveform must be in view to for the oscilloscope measure functions to be accurate.
7. Compare with what you would expect for this circuit based on theoretical calculations.
8. Repeat steps 6 through 7 to make measurements for all frequencies shown and record these values. At very high frequencies AUTOSCALE may not work because the filtered signal is too small. Use manual adjustments of the Volts/division and Time/Division to reveal the signal.

3. 1. 2. RC High-Pass Filter

A *High-Pass filter circuit* only passes signals above the selected cut-off frequency, f_c eliminating or blocking any low frequency signals from the input. A *high-pass filter circuit* is implemented from series *RC* by taking the output across the resistor.

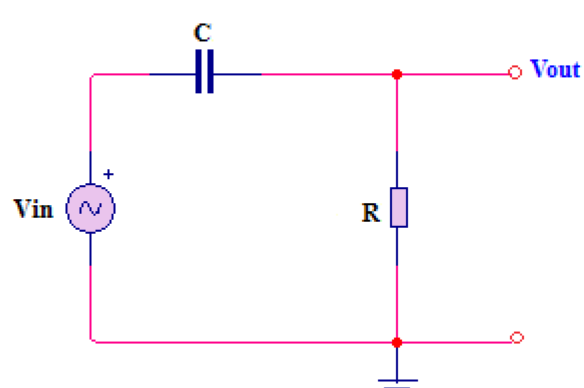


Figure 3-6: Basic RC high-pass filter circuit

In this circuit arrangement, figure 3-6, the reactance of the capacitor is very high at low frequencies so the capacitor acts like an open circuit and blocks any input signals at V_{in} until the cut-off frequency point (f_c) is reached. Above this cut-off frequency point the reactance of the capacitor has reduced sufficiently as to now act more like a short circuit allowing all of the input signal to pass directly to the output as shown in figure 3-7 the High Pass Frequency Response Curve.

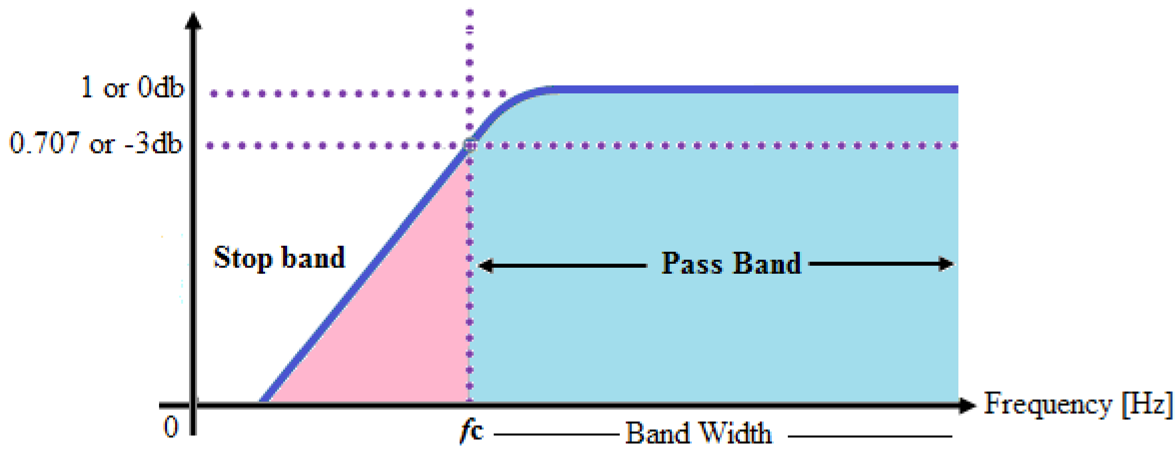


Figure 3-7: High-Pass frequency response curve

The Bode Plot or frequency response Curve above for a High-Pass filter is the exact opposite to that of a low pass filter. Here the signal is attenuated or damped at low frequencies with the output increasing until the frequency reaches the cut-off point (f_c) where again $R = Xc$. It has a response curve that extends down from infinity to the cut-off frequency, where the output voltage amplitude is $1/\sqrt{2}$ = 70.7% of the input signal value or -3dB (20 log (V_{out}/V_{in})) of the input value.

The frequency response curve for a high-pass filter implies that the filter can pass all signals out to infinity. However in practice, the high-pass filter response does not extend to infinity but is limited by the electrical characteristics of the components used.

The cut-off frequency point for a high pass filter can be found using the same equation as that of the low-pass filter and is as follow:

$$f_{cutoff} = 1/2\pi RC$$

The capacitive reactance of a capacitor in an AC circuit is given as:

$$Xc = 1/2\pi f C \text{ [Ohm]}$$

Opposition to current flow in an AC circuit is called impedance, symbolized by Z and for a series circuit consisting of a single resistor in series with a single capacitor, the circuit impedance is calculated as:

$$Z = \sqrt{(R^2 + Xc^2)}$$

Then by substituting our equation for impedance above into the potential divider equation gives *RC* potential divider equation:

$$V_{out} = V_{in} * R / \sqrt{(R^2 + Xc^2)} = V_{in} * R / Z$$

chap 3

Thus, by using the potential divider equation of two resistors in series and substituting for impedance we can calculate the output voltage of an *RC* Filter for any given frequency.

Experiment 3-2: RC High-Pass Filter

Design and examine a *high-pass filter circuit* based on series RC circuit that is realized by taking the output across the resistor, just as in lead circuit.

Parts and materials

No	Item	Specification	Quantity
1	10VAC, 100Hz-20KHz supply	Agilent 33120A Function generator	1
2	Oscilloscope	54622D Agilent digital oscilloscope	1
3	Breadboard	RSR 03MB102 Prototyping board	1
4	Connecting wire	22-gaude (0.33mm^2) solid wire	20cm
5	100Ω fixed resistor	100Ω, 1/2watt fixed resistor	1
6	1uF Capacitor	1uF, 16V no polarized capacitor	1

To illustrate *RC* high-pass filtering action, the circuit of the figure 3-8 shows a series of specific measurements. The frequency starts at 10 Hz and is increased in increments up to 10 KHz. As you can see, the capacitive reactance decreases as the frequency increases, thus causing more of the total input voltage to be dropped across the resistor.

chap 3

Cutoff frequency f_c,

f_c = 1/2πRC = 1/(2*3.14*100*10^{-6}) = 1591 Hz

Let us vary the frequency from 10Hz, 100 Hz, 1 KHz, up to 10 KHz at a constant V_{in} of 10 V sinusoidal.

At 10 Hz,

Xc = 1/2πfC = 1/2*3.14*10*10^{-6} = 15,900Ω = 15.9 KΩ

Z = √ (R^2 + Xc^2) = √ (100^2 + 15900^2) ≈ 15900 Ω

V_{out} = V_{in}*R/Z = 10V*100/15900 = 6.289 mV ≈ 6.3 mV

I = V/Z = 10V/15900Ω = 0.0006289 ≈ 0.629 mA

At 100 Hz,

Xc = 1/2πfC = 1/2*3.14*100*10^{-6} = 1.59 KΩ

Z = √ (R^2 + Xc^2) = √ (100^2 + 1590^2) =1590.314 ≈ 1590 Ω

V_{out} = V_{in}*R/Z = 10V*100/1590.314 = 0.6289 V ≈ 0.63 V

I = V/Z = 10V/1590Ω = 0.006289 ≈ 6.29 mA

At 1 KHz,

Xc = 1/2πfC = 1/2*3.14*1000*10^{-6} = 159 Ω

Z = √ (R^2 + Xc^2) = √ (100^2 + 159^2) =188 Ω

$V_{out} = V_{in}$*R/Z = 10V*100/188 = 5.319 V

I = V/Z = 10V/188Ω = 53.2 mA

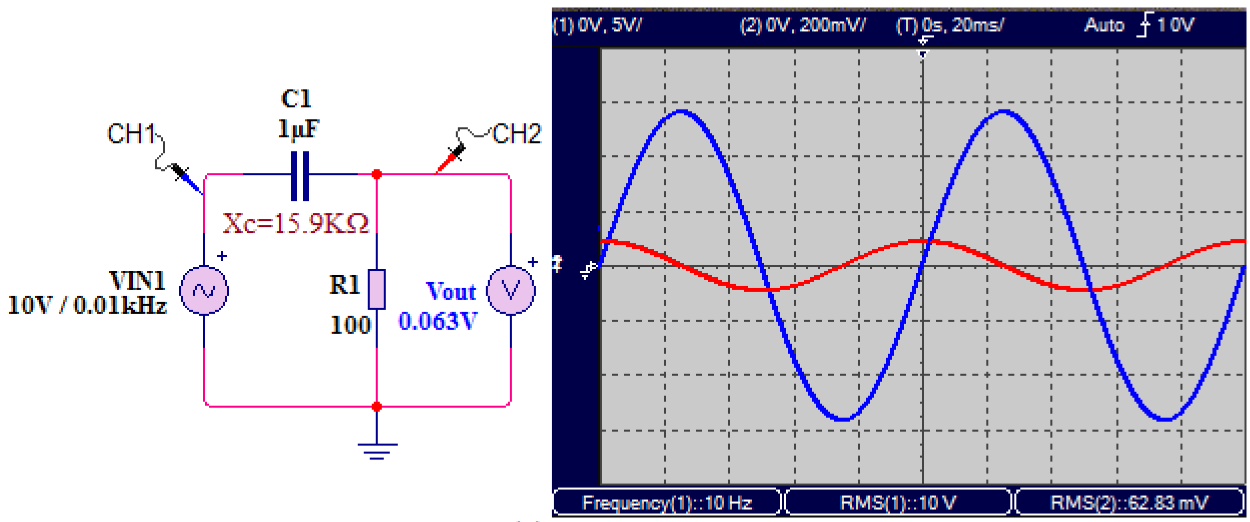
At 10 KHz,

Xc = 1/2πf C = 1/2*3.14*10000*10^{-6} = 15.9 Ω

Z = √ ($R^2 + Xc^2$) = √ ($100^2 + 15.9^2$) = 101.26 Ω

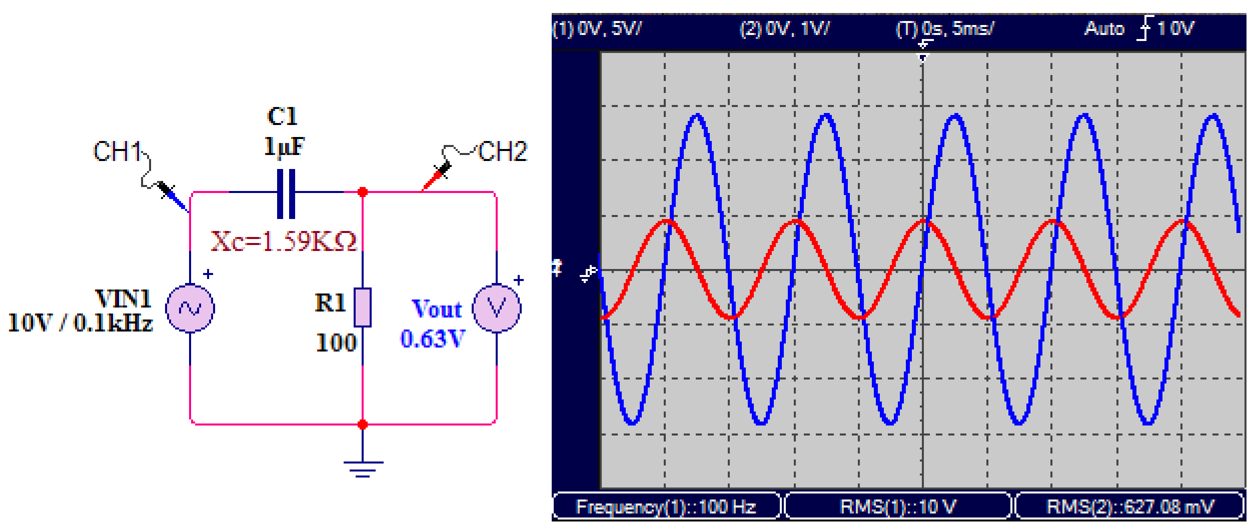
$V_{out} = V_{in}$*R/Z = 10V*100/101 = 9.876 V

I = V/Z = 10V/101.25Ω = 98.8 mA



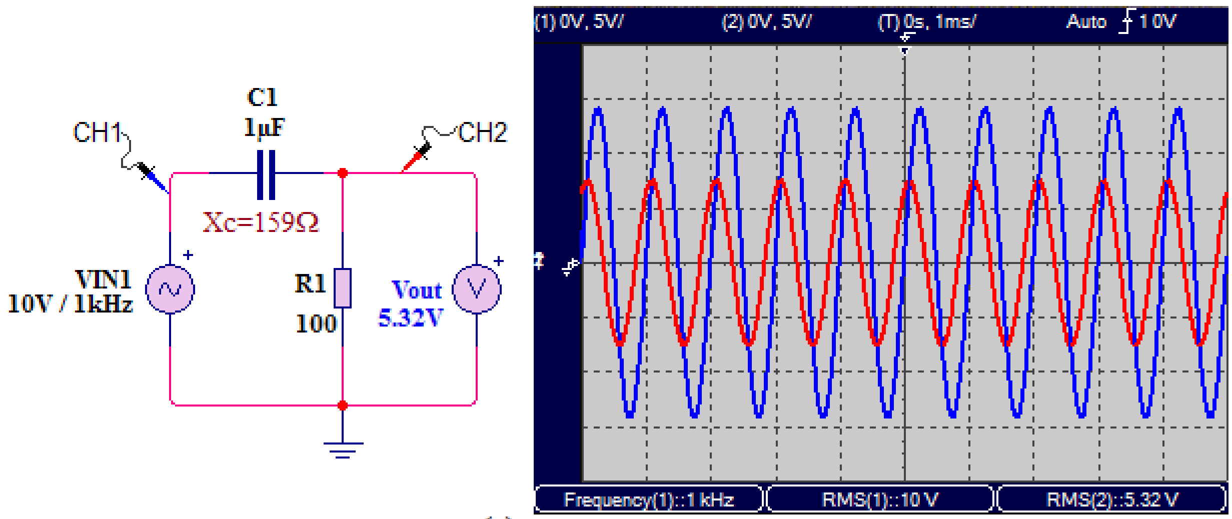
(a)

(a) f = 10 Hz, Xc = 15.9 KΩ, V_{out} = 0.063 V



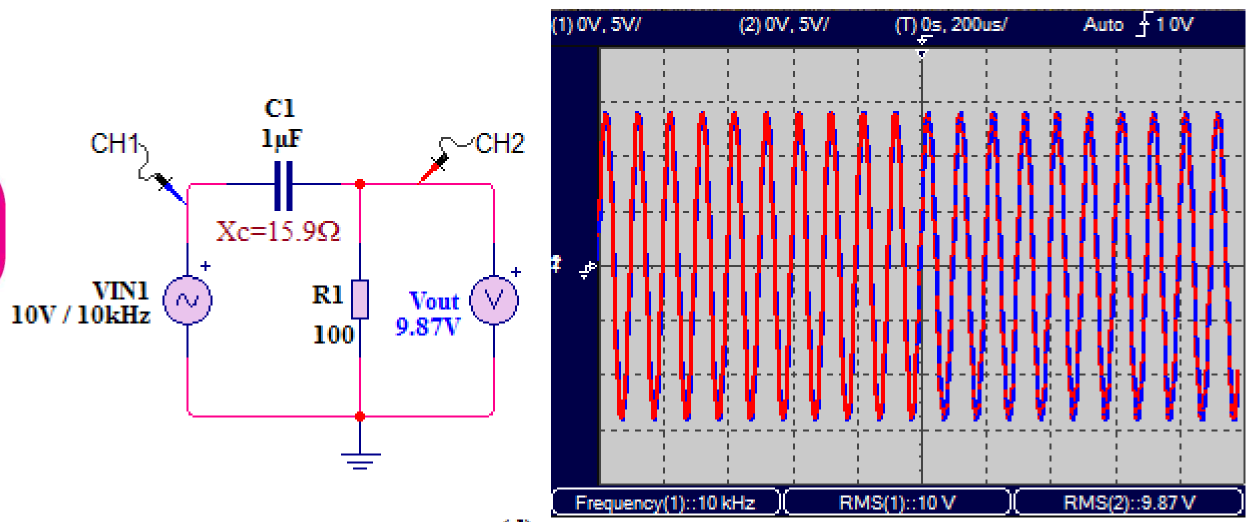
(b)

(b) f = 100 Hz, Xc = 1.59 KΩ, V_{out} = 0.63 V



(c)

(c) f = 1 KHz, Xc = 156 Ω, V_{out} = 5.32 V



(d)

(d) f = 10 KHz, Xc = 159 Ω, V_{out} = 9.87 V

Figure 3-8: High-Pass Filter; (a), (b), (c) and (d) are examples of high-pass filtering action. As frequency increases, V_{out} increases

The table 3-2 summarizes the variation of circuit parameters with frequency.

f (KHz)	Xc (Ω)	Z_{tot} (Ω)	I (mA)	V_{out} (V)
0.01	15,900	≈15,900	0.629	0.063
0.1	1,590	1,593	6.28	0.63
1	159	188	53.2	5.32
10	15.9	101	98.8	9.87

Table 3-2: Summarized variation of circuit parameters with frequency

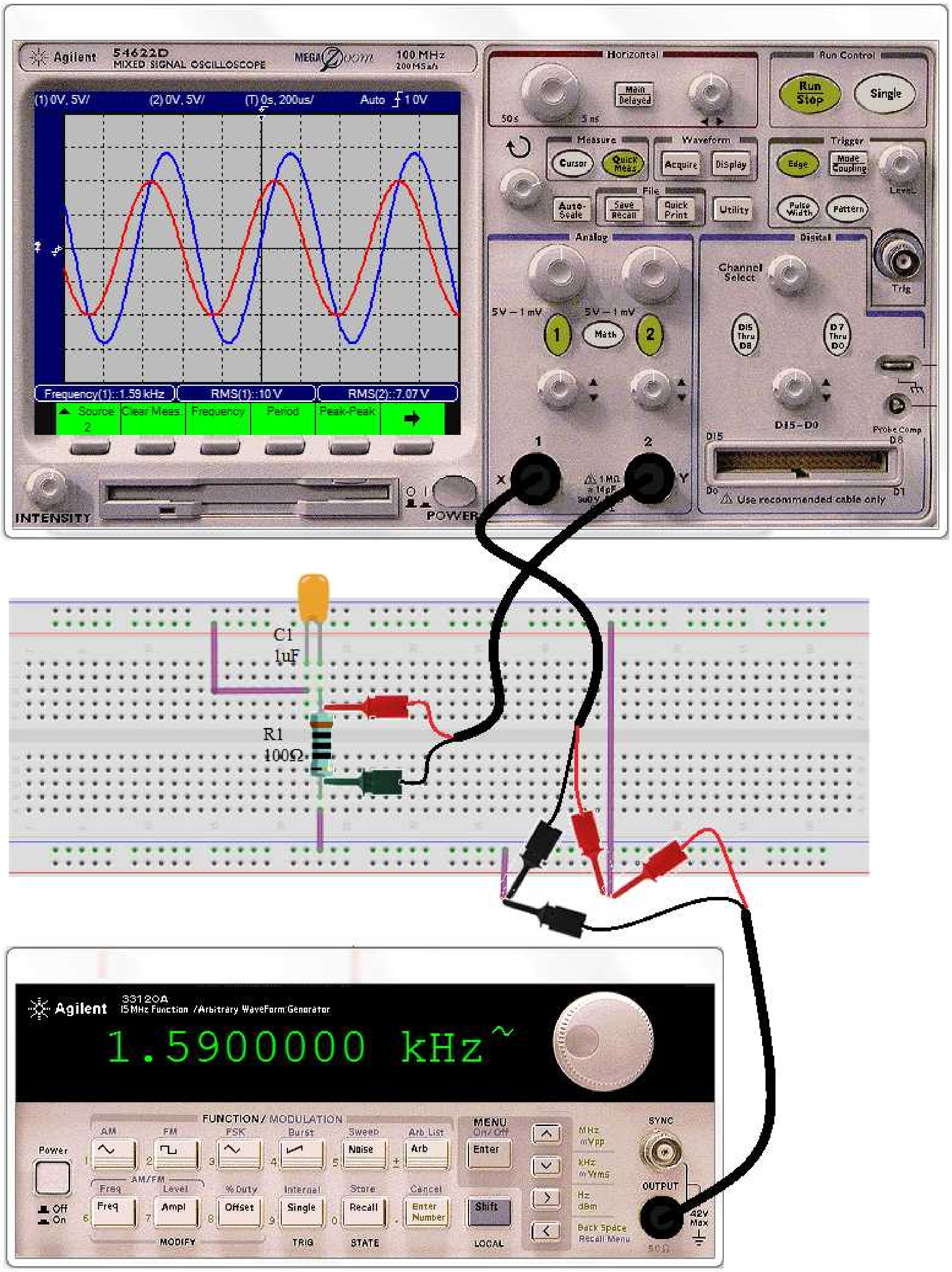
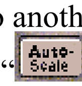


Figure 3-9: RC high-pass filter, real world circuit connection

Procedure

1. Choose the 100 Ω R_1 resistor, using resistor color code, from resistor pack and 1uF C_1 capacitor from capacitor pack.
2. Build the circuit as illustrated in figure 3-9, high-pass filter real world circuit connection.
3. Switch on the function generator. By using any method, set the waveform generator output to a sinusoid with amplitude of 10 V rms and an initial frequency of 10 Hz.
4. The oscilloscope display may have a natural coarseness, which can be reduced by selecting the Average function. Try to use as few samples as possible to avoid long delays while moving from one delay to another.
5. Push the AUTOSCALE button “” to display both channels (Channel 1 and Channel 2). Reposition so that the 0 V is at the midline on the both channels and the waveforms appear to overlap. Adjust the Volts/division to 5V/division for both channels and at 200 usec/division to display more than one complete period. At low frequencies, the amplitude of the filtered signal may be too small to be detected by the Auto-scale feature. You may need to set the Volts/division and time/division scale manually in order to see the signal.
6. Press the MEASURE button “” to determine the Voltage rms amplitudes for both channels, and record in the data sheet for the frequency of 10 Hz, 100 Hz , 1 KHz, 1590 Hz and 10KHz. At least one full period of the waveform must be in view to for the oscilloscope measure functions to be accurate. At low frequencies, the amplitude of the filtered signal may be too small to be detected by the Auto-scale feature.
You may need to set the Volts/Division and Time/Division scale manually in order to see the signal.
7. Do the obtained results agree with what you would expect for a high pass filter?

3. 2. The *RL* Circuit as a Filter

Like *RC* circuits, a series *RL* circuits also exhibit a frequency-selectivity characteristics.

The cut-off Frequency of an *RL* circuit

The frequency at which the inductive reactance equals the reactance in Low-Pass or High-Pass *RL* circuit is called the *cut-off frequency* and is designated f_c, this condition is expressed as $2\pi f_c L=R$. Solving for f_c results in the following formula:

$$f_c = R/2\pi L$$

As with the *RC* circuit, the output voltage is 70.7% of its maximum value at f_c. In a High-Pass circuit, all frequencies above f_c are considered to be passed and all those below to be rejected. The reverse, of course, is true for a Low-Pass circuit. The *bandwidth* as was defined earlier, applies to both *RC* and *RL* series circuits.

3. 2. 1. RL Low-Pass filter

In terms of filtering action of the series *RL* circuit, the variation in the magnitude of the output voltage as a function of frequency is important. Figure 3-10 shows the basic circuit of series *RL* low-pass filter where the output voltage is taken across the resistor.

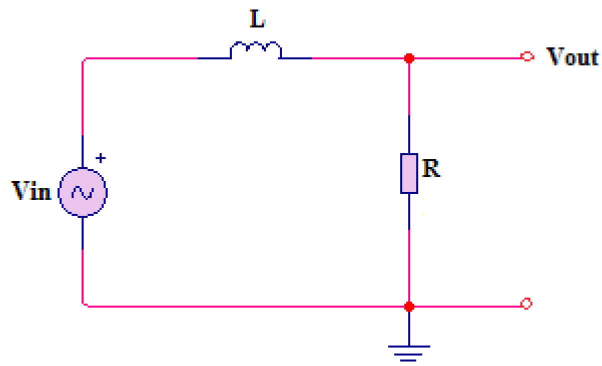


Figure 3-10: Basic circuit for series RL Low-Pass filter

The inductive reactance of an inductor in an AC circuit is given as:

$$X_L = 2\pi f L \text{ [Ohm]}$$

Opposition to current flow in an AC circuit is called impedance, symbolized by Z and for a series circuit consisting of a single resistor in series with a single inductor, the circuit impedance is calculated as:

$$Z = \sqrt{(R^2 + X_L^2)}$$

Then by substituting our equation for impedance above into the potential divider equation gives *RC* potential divider equation:

$$V_{out} = V_{in} * R / \sqrt{(R^2 + XL^2)} = V_{in} * R / Z$$

Accordingly, by using the potential divider equation of two resistors in series and substituting for impedance we can calculate the output voltage of an *RL* Filter for any given frequency.

The *frequency response* curve for these particular values would appear similar to the response curve for the Low-Pass *RC* circuit.

Experiment 3-3: RL Low-Pass filter

Design and examine a low-pass filter based on a series RL circuit.

Parts and materials

No	Item	Specification	Quantity
1	10VAC, 100Hz-20KHz supply	Function generator	1
2	Oscilloscope	TDS 2024 digital oscilloscope	1
3	Breadboard	390-pin Prototyping board	1
4	Connecting wire	22-gaude (0.33mm^2) solid wire	20cm
5	1 KΩ fixed resistor	1 KΩ, 1/2watt fixed resistor	1
6	100mH inductor	100mH inductor	1

The figure 3-11 shows the filter action of a series *RL* circuit using a specific series of measurements in which the frequency starts at 100 Hz and is increased in increments up to 20 KHz. At each value of frequency, the output voltage is measured. As you can see, the inductive reactance increases as frequency increases, thus causing less voltage to be dropped across the resistor while the input voltage is held at constant at 10 V throughout each step.

At 100 Hz,

$X_L = 2\pi f L = 2*3.14*100*0.1 = 62.8\ \Omega$

$Z = \sqrt{(R^2 + XL^2)} = \sqrt{(1000^2 + 62.8^2)} = 1001.9\ \Omega$

$V_{out} = V_{in}*R / Z = 10V*1000/1001.9 = 9.98\ V$

At 1 KHz,

$XL = 2\pi f L = 2*3.14*1000*0.1 = 628\ \Omega$

$Z = \sqrt{(R^2 + XL^2)} = \sqrt{(1000^2 + 628^2)} = 1180.8\ \Omega$

$V_{out} = V_{in}*R / Z = 10V*1000/1180.8 = 8.468\ V$

At 10 KHz,

$XL = 2\pi f L = 2*3.14*10000*0.1 = 6.28\ K\Omega$

$Z = \sqrt{(R^2 + XL^2)} = \sqrt{(1000^2 + 6280^2)} = 6359\ \Omega$

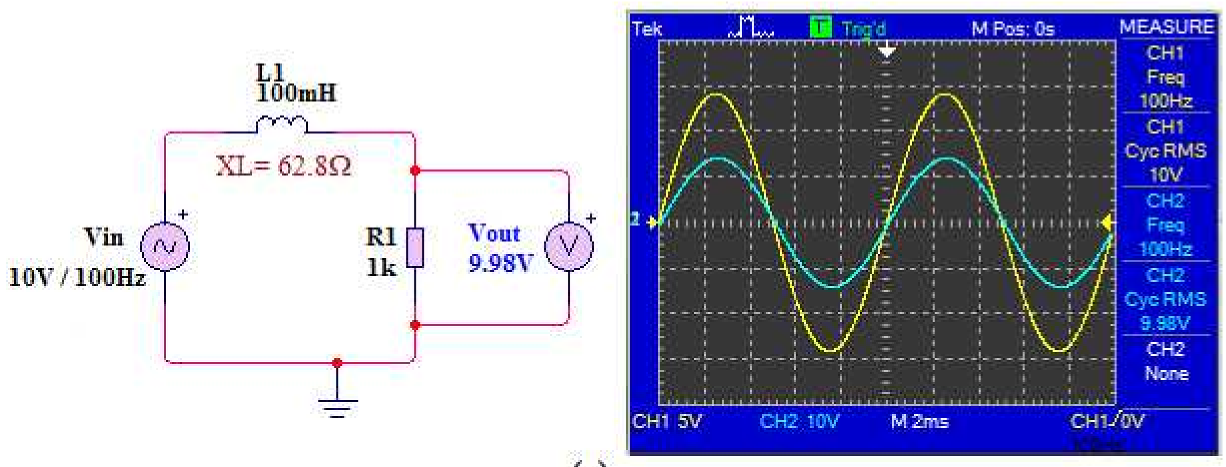
$V_{out} = V_{in}*R / Z = 10V*1000/6359 = 1.57\ V$

At 20 KHz,

$XL = 2\pi f L = 2*3.14*20000*0.1 = 12560\ K\Omega$

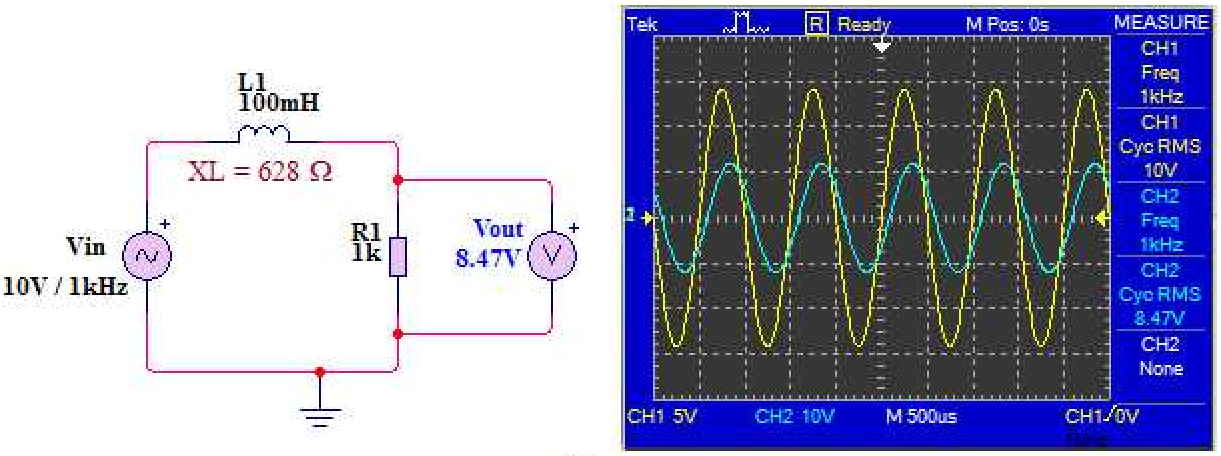
$Z = \sqrt{(R^2 + XL^2)} = \sqrt{(1000^2 + 6280^2)} = 12599\ \Omega$

$V_{out} = V_{in}*R / Z = 10V*1000/12599 = 0.79\ V$



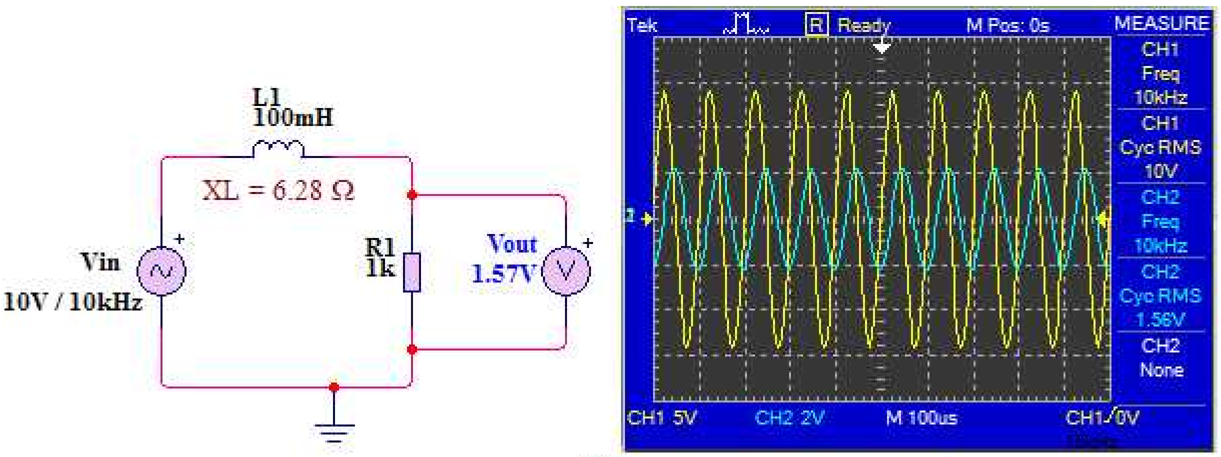
(a)

(a) f = 100Hz, XL = 6.28 Ω, V_{out} = 9.98V



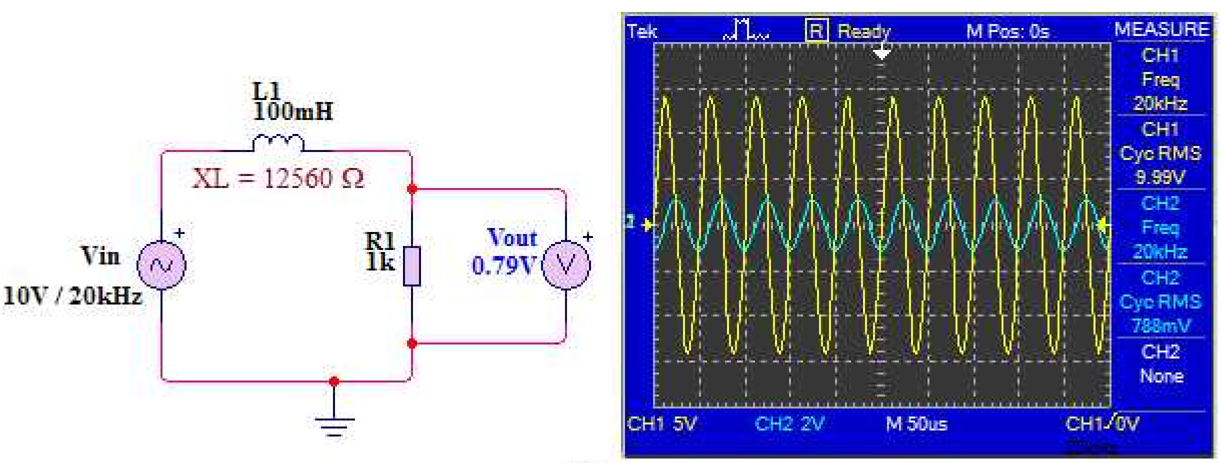
(b)

(b) f = 1 KHz, XL = 628 Ω, V_{out} = 8.47V



(c)

(c) f = 10 KHz, XL = 6.28KΩ, V_{out} = 1.57V



(d)

(d) f = 20 KHz, XL = 12.560KΩ, V_{out} = 0.79V

Figure 3-11: Example of low-pass filtering action. Winding resistance has been neglected. (a), (b), (c) and (d); as the input frequency increases, the output voltage decreases

Procedure

1. Choose the 1 KΩ resistor from resistor pack and 100 mH inductor.
2. Construct RL low pass filter circuit as illustrated in circuit connection figure 3-12.

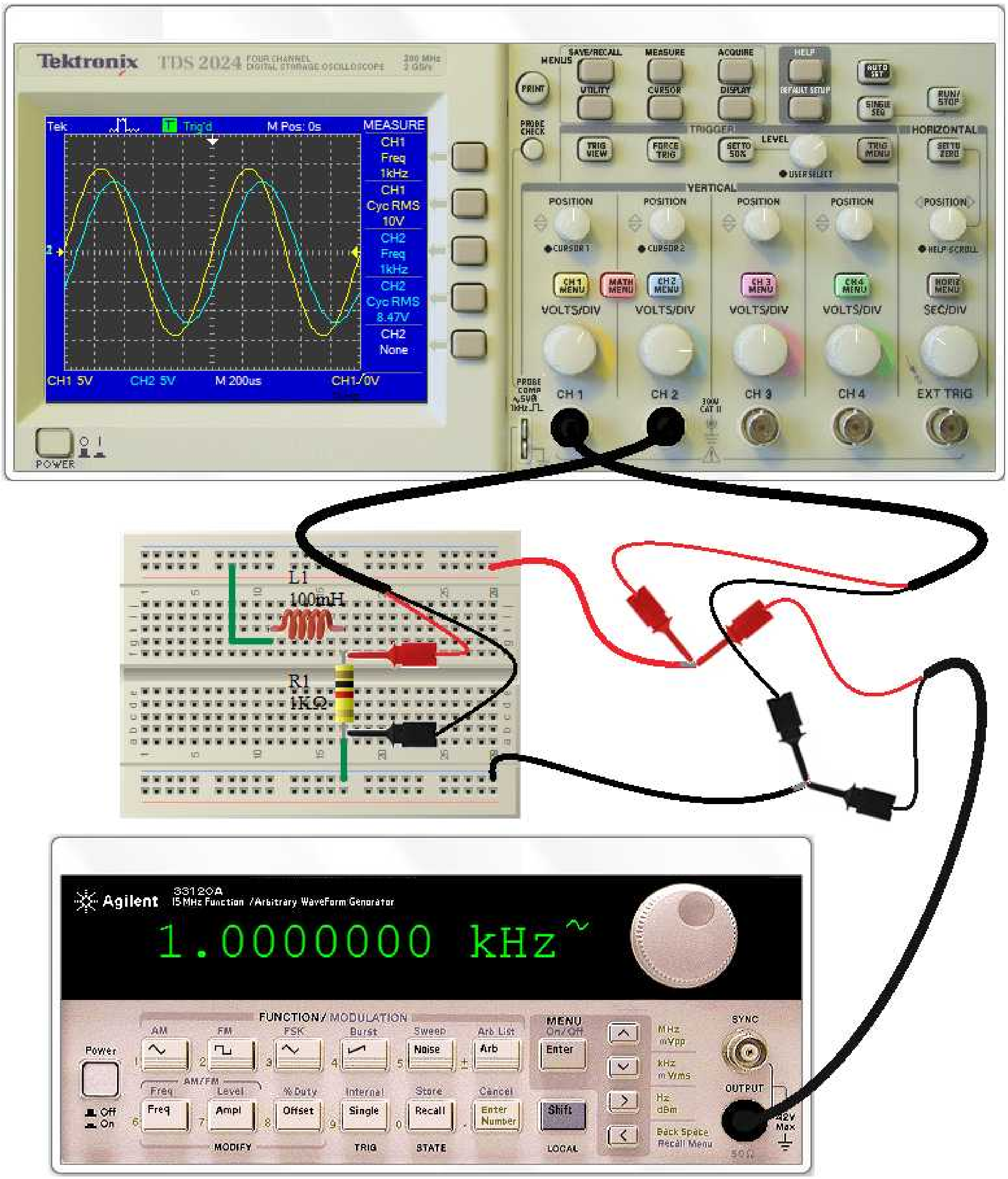


Figure 3-12: RL low-pass filter circuit, real world circuit connection

3. Set the function generator to output a sinusoid with amplitude of 10 V rms or 28.28V peak-to-peak and an initial frequency of 100 Hz.
4. Connect the output of function generator (V_{in} of the circuit) to channel one CH1 of oscilloscope and connect the output of the circuit, taken across the resistor, to channel two CH2 of oscilloscope.
5. The oscilloscope display may have a natural coarseness, which can be reduced by selecting the Average function. Try to use as few samples as possible to avoid long delays while moving from one display to another.
6. Push the AUTO-SET button "

" to display both channels. Reposition so that the 0V is at the midline on the both channels and the waveforms appear to overlap. Adjust the Volts/division to 5V/division for channel one CH1, 10V/division for channel two CH2 and the 2msec/division to display more than one complete period. At high frequencies, the amplitude of the filtered signal may be too small to be detected by the AUTO-SET feature. You may need to set the Volts/division scale manually in order to see the signal. Again as you increase the frequency the period decreases, so you need to reduce the time/division. That is, 500uSec/division at 1 KHz, 100 uSec/division at 10 KHz, and 50 uSec/division or 100uSec/division at 20 KHz.
7. Press the MEASURE button "

" to determine the voltage RMS amplitudes for both channels, and record in the data sheet for the frequency of 100 Hz, 1 KHz, 10 KHz and 20 KHz. At least one full period of the waveform must be in view to for the oscilloscope measure functions to be accurate.
8. Compare with what you would expect for this circuit based on theoretical calculations.
9. Repeat steps 6 through 7 to make measurements for all frequencies shown and record these values. At very high frequencies AUTO-SET may not work because the filtered signal is too small. Use manual adjustments of the volts/division and time/division to reveal the signal.
10. Plot the results (voltage versus frequency) on a graph to make a frequency response curve.

3. 2. 2. RL High-Pass Filter

A series *RL* circuit is also used as a High-Pass filter by blocking the lower frequencies and passing the frequencies above the cut-off frequency. The output voltage is taken across the inductor and the basic circuit is illustrated in the figure 3-13.

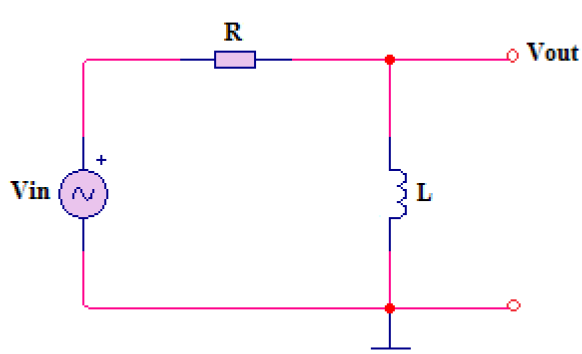


Figure 3-13: Basic circuit for RL high-pass filter

Again, when the values are plotted, the response curve is similar to the one for the High-Pass *RC* circuit that discussed earlier.

The inductive reactance of an inductor in an AC circuit is given as:

$$XL = 2\pi f L \text{ [Ohm]}$$

Opposition to current flow in an AC circuit is called impedance, symbolized by Z and for a series circuit consisting of a single resistor in series with a single inductor, the circuit impedance is calculated as:

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

Then by substituting our equation for impedance above into the potential divider equation gives *RC* potential divider equation:

$$V_{out} = V_{in} * XL / \sqrt{(R^2 + XL^2)} = V_{in} * XL / Z$$

Hence, by using the potential divider equation of two resistors in series and substituting for impedance we can calculate the output voltage of an *RL* Filter for any given frequency.

Experiment 3-4: RL High-Pass Filter

Design and examine a high-pass filter based on series RL circuit.

chap 3

Parts and materials

No	Item	Specification	Quantity
1	10VAC, 10Hz-10KHz supply	Function generator	1
2	Oscilloscope	54622D Agilent digital oscilloscope	1
3	Breadboard	390-pin Prototyping board	1
4	Connecting wire	22-gaude (0.33mm^2) solid wire	20cm
5	1 KΩ fixed resistor	1 KΩ, 1/2watt fixed resistor	1
6	100mH inductor	100mH inductor	1

To illustrate *RL* High-Pass filtering action, the circuits figure 3-14 show a series of specific measurements. The frequency starts at 10 Hz and is increased in increment up to 10 KHz. As you can see, the inductive reactance increases as the frequency increases, thus causing more voltage to be dropped across the inductor.

At 10 Hz,

$$X_L = 2\pi f L = 2*3.14*10*0.1 = 6.28\ \Omega$$
$$Z = \sqrt{(R^2 + XL^2)} = \sqrt{(1000^2 + 6.28^2)} = 1000\ \Omega$$
$$V_{out} = V_{in} * XL/Z = 10V*6.28\ \Omega/1000\ \Omega = 0.06279\ V$$

At 100 Hz,

$$X_L = 2\pi f L = 2*3.14*100*0.1 = 62.8\ \Omega$$
$$Z = \sqrt{(R^2 + XL^2)} = \sqrt{(1000^2 + 6.28^2)} = 1001.9\ \Omega$$

$V_{out} = V_{in}$*XL/Z = 10V*62.8 Ω/1001.9 Ω = 0.627 V

At 1 KHz,

$X_L = 2\pi f L$ = 2*3.14*1000*0.1 = 628 Ω

$Z = \sqrt{}(R^2 + XL^2) = \sqrt{}(1000^2 + 6.28^2)$ = 1180.8 Ω

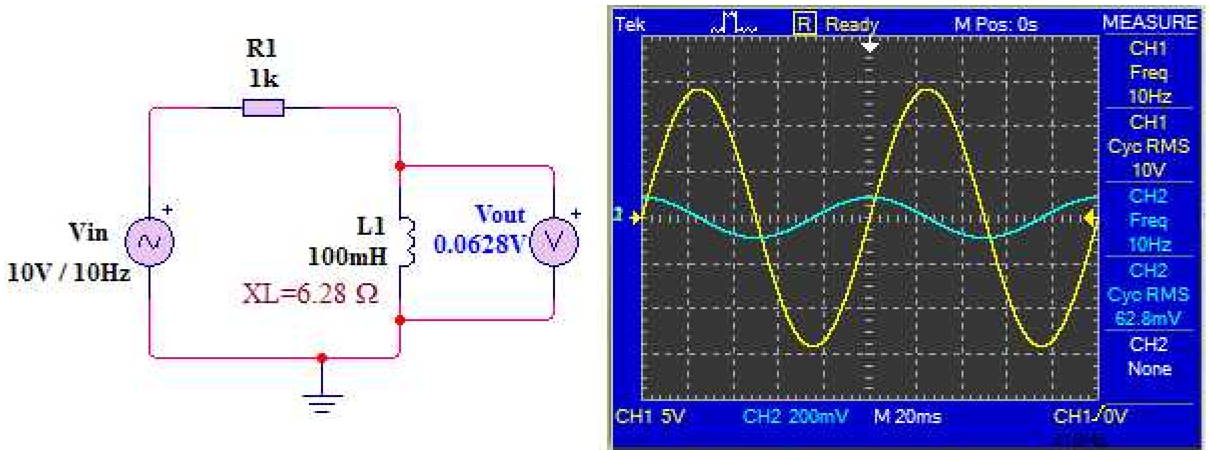
$V_{out} = V_{in}$*XL/Z = 10V*628 Ω/1180.8 Ω = 5.32 V

At 10 KHz,

$X_L = 2\pi f L$ = 2*3.14*10000*0.1 = 6.28 KΩ

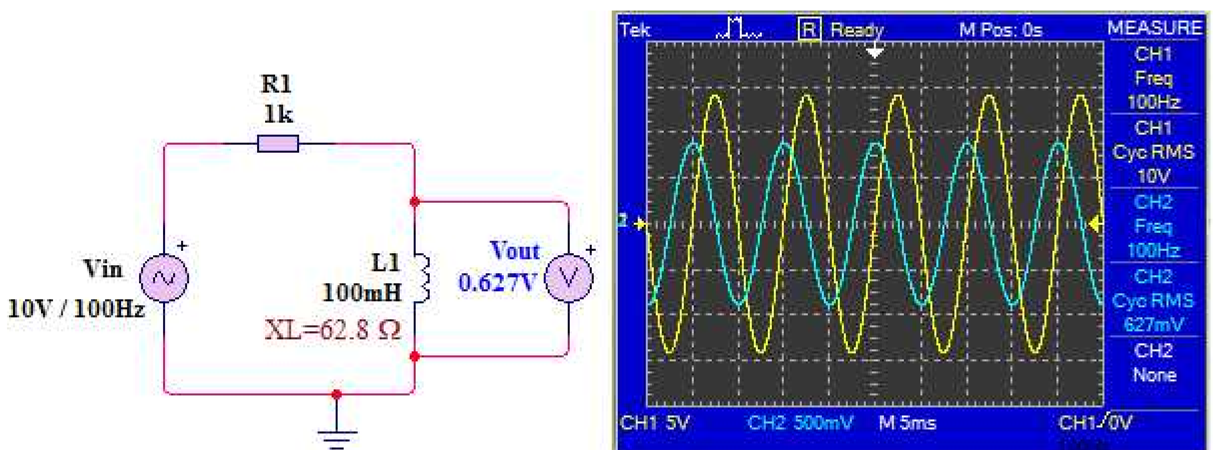
$Z = \sqrt{}(R^2 + XL^2) = \sqrt{}(1000^2 + 6280^2)$ = 6359 Ω

$V_{out} = V_{in}$*XL/Z = 10V*6280 Ω/6359 Ω = 9.88 V



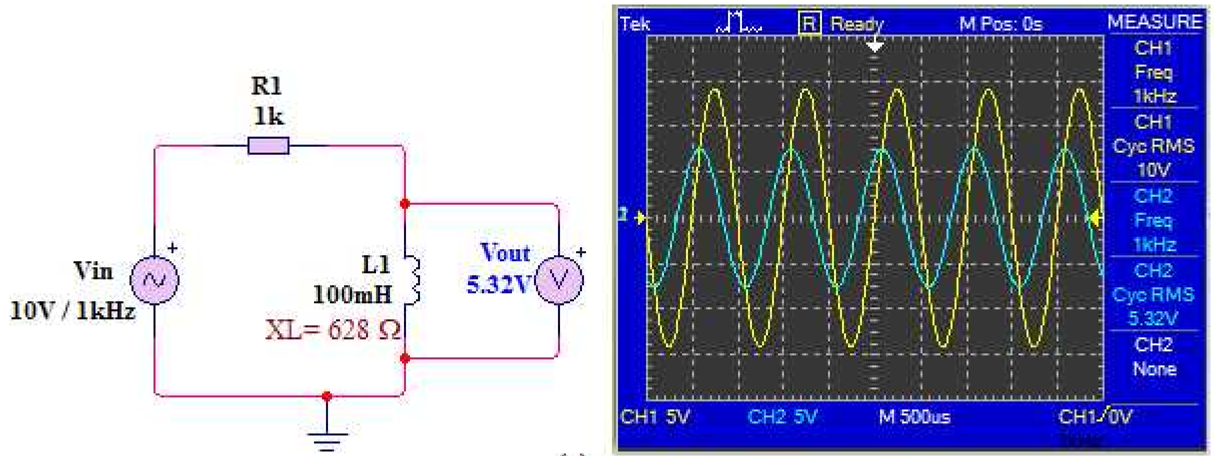
(a)

(a) f = 10Hz, XL = 6.28 Ω, V_{out} = 0.0628 V



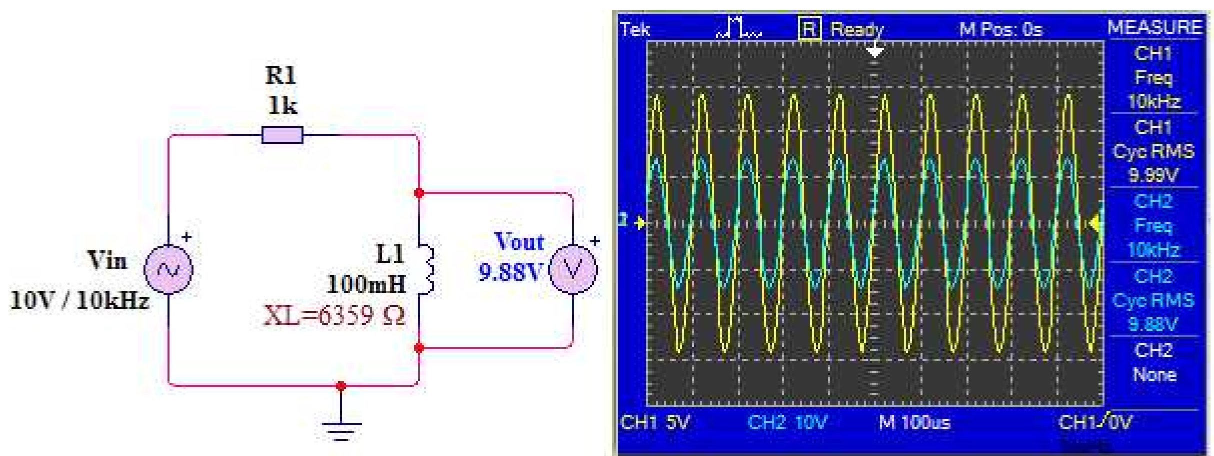
(b)

(b) f = 100 Hz, XL = 62.8 Ω, V_{out} = 0.627 V



(c)

(c) f = 1 KHz, XL = 628 Ω, V_{out} = 5.32 V



(d)

(d) f = 10 KHz, XL = 6359 Ω, V_{out} = 9.88 V

Figure 3-14: Example of high-pass filtering action. Winding resistance has been neglected. (a), (b), (c) and (d); as the input frequency increases, the output voltage increases

Procedure

1. Choose the 1KΩ resistor from resistor pack and 100 mH inductor.
2. Construct RL high pass filter circuit as illustrated in circuit connection figure 3-15.
3. Set the function generator to output a sinusoid with amplitude of 10V rms or 28.28V peak-to-peak and an initial frequency of 10 Hz.
4. Connect the output of function generator (V_{in} of the circuit) to channel one CH1 of oscilloscope and connect the output of the circuit, taken across the inductor, to channel two CH2 of oscilloscope.
5. The oscilloscope display may have a natural coarseness, which can be reduced by selecting the Average function. Try to use as few samples as possible to avoid long delays while moving from one display to another.

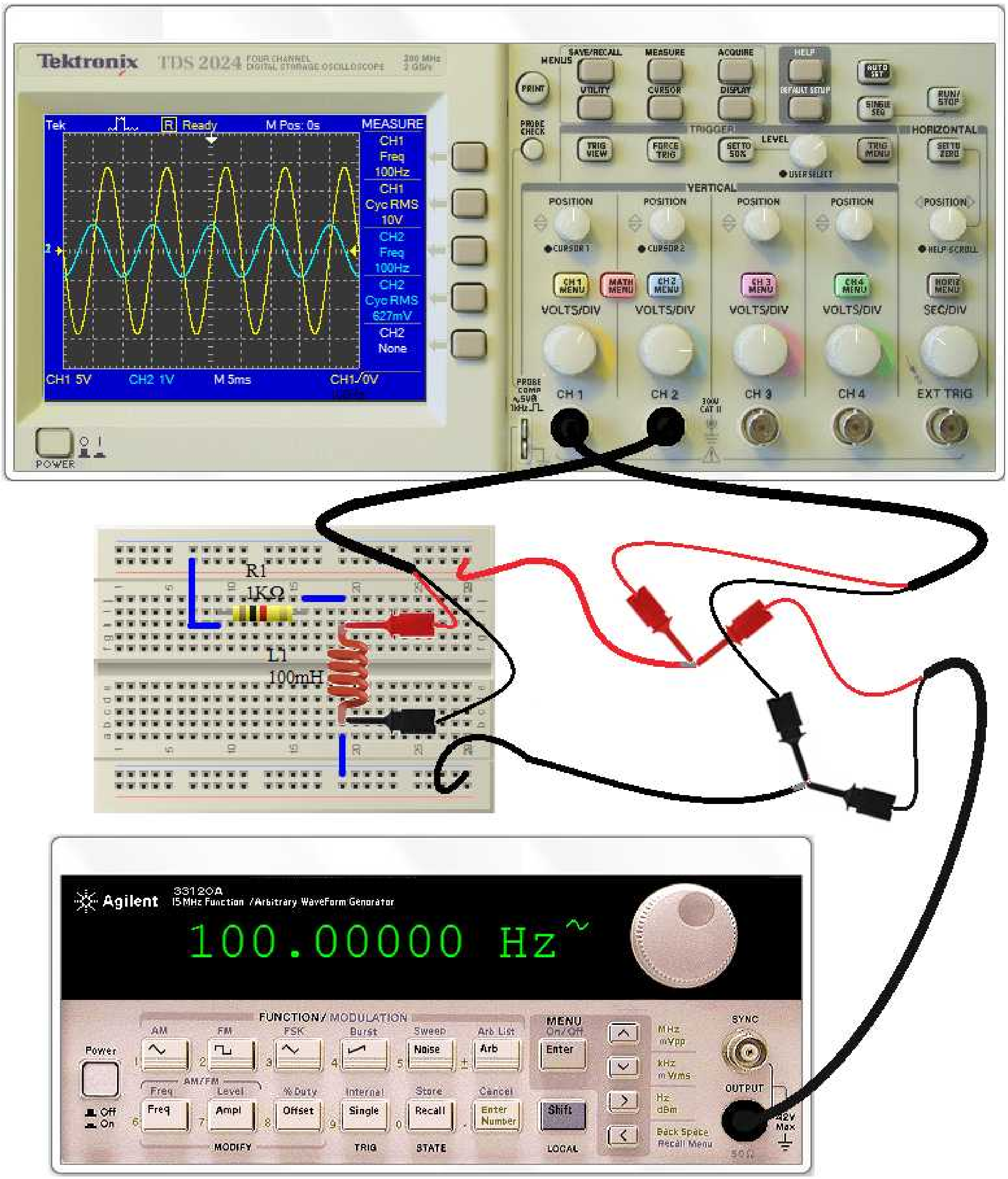
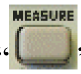


Figure 3-15: RL high-pass filter, real world circuit connection

6. Push the AUTO-SET button "" to display both channels. Reposition so that the 0 V is at the midline on the both channels and the waveforms appear to overlap. Adjust the Volts/division to 5V/division for channel one CH1, 200mV/division for channel two CH2 and the 20msec/division to display more than one complete period. At high frequencies, the amplitude of the filtered signal may be too small to be detected by the AUTO-SET feature.

You may need to set the Volts/division scale manually in order to see the signal. Again as you increase the frequency the period decreases, so you need to reduce the time/division. That is, 5mSec/division at 100 Hz, 500uSec/division at 10 KHz, and 50uSec/division or 100uSec/division at 10 KHz.

7. Press the MEASURE button "

" to determine the voltage RMS amplitudes for both channels, and record in the data sheet for the frequency of 10 Hz, 100 Hz, 1 KHz and 10 KHz. At least one full period of the waveform must be in view to for the oscilloscope measure functions to be accurate.
8. Compare with what you would expect for this circuit based on theoretical calculations.
9. Repeat steps 6 through 7 to make measurements for all frequencies shown and record these values. At very high frequencies AUTO-SET may not work because the filtered signal is too small. Use manual adjustments of the volts/division and time/division to reveal the signal.
10. Plot the results (voltage versus frequency) on a graph to make a frequency response curve.

3. 3. Series Resonant Filters

A common use of series RLC circuits is in filter applications, these include basic band-pass and band-stop filters.

3. 3. 1. The Band-Pass Filter

A band-pass filter allows signal at the resonant frequency and at the frequencies within a certain band (or rang) extending below and above the resonant value to pass from input to output without a significant reduction in amplitude. Signal at frequencies lying outside this specified band (called the bandpass) are reduced in amplitude to below a certain level and are considered to be rejected by the filter.

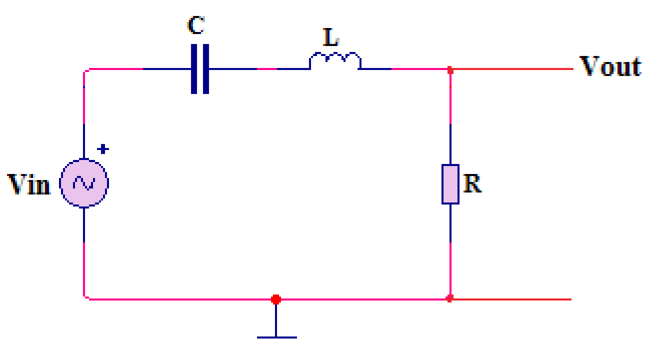


Figure 3-16: Basic passive RLC high-pass filter circuit

The filtering action is the result of the impedance characteristic of the filter. The impedance is minimum at resonance and has increasingly higher value below and above the resonance frequency. At very low frequencies, the impedance is very high and tends to block the current. As the frequency increases, the impedance drops, allowing more current and thus more voltage across the output. At the resonance frequency, the impedance is very low and equal to the winding resistance of the coil. At this point there is a maximum current and the resulting output voltage is maximum.

As the frequency goes above the resonance, the impedance again increases, causing the current and the resulting output voltage to drop.

As seen for a low pass filter that only pass signals of a low frequency range or a high pass filter which pass signals of a higher frequency range, a Band-Pass filters passes signals within a certain "band" or "spread" of frequencies without distorting the input signal or introducing extra noise.

This band of frequencies can be any width and is commonly known as the filters Bandwidth. Bandwidth is defined as the frequency range between two specified frequency cut-off points (f_c), that are 3dB below the maximum centre or resonant peak while attenuating or weakening the others outside of these two points. Then for widely spread frequencies, we can simply define the term "bandwidth", BW as being the difference between the lower cut-off frequency ($f_{c(lower)}$) and the higher cut-off frequency ($f_{c(higher)}$) points. In other words, BW = f_H - f_L.

The "ideal" Band Pass Filter can also be used to isolate or filter out certain frequencies that lie within a particular band of frequencies, for example, noise cancellation. The band-pass filter can be realized by combining *RC* low-pass filter and RC high-pass filter thus, are known generally as second-order filters, (two-pole) because they have "two" reactive component, the capacitors, within their circuit design. One capacitor in the low pass circuit and another capacitor in the high pass circuit. Clearly for type of pass band filter to function correctly, the cut-off frequency of the low pass filter must be higher than the cut-off frequency for the high pass filter.

The center-frequency (resonant frequency) *f*c is given as:

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

The inductive reactance XL is given as:

$$XL=2\pi Lf$$

Quality factor Q of the circuit is the ratio of the reactance power in the inductor to the true power in the winding resistance of the coil and any other resistance in series with the coil.

Quality factor, Q = XL / R

Band width BW is calculated as = f_c / Q

Also BW = f_H - f_L

Experiment 3-5: Series resonant Band-Pass filter

Design a band-pass filter based on series RLC circuit. The series LC portion is placed between the input and the output and the output is taken across the resistor.

Part list

No	Item	Quantity
1	Breadboard	1
2	Inductor 47mH	1
3	10VAC source with different frequencies capability (function generator)	1
4	Wire (0.33mm^2 solid wire)	20cm

5	Resistor 100Ω	1
6	Capacitor 220nF	

The circuits of figure 3-17 illustrate the general frequency response of a series resonant band-pass filter.

For above given resistor, capacitor and inductor let determine band-pass filter related quantities.

Resonant frequency, f_r = =1/(2*3.14*√0.00022*0.047) = 1565 Hz

Inductive reactance, XL = 2πLf = 2*3.14*0.047H*1565Hz = 461.63 Ω

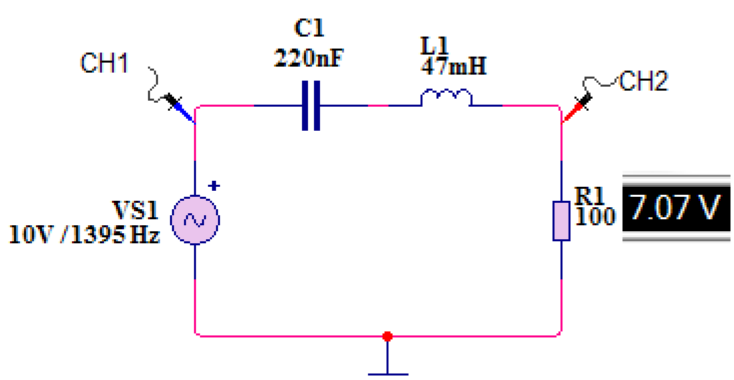
Quality factor, Q = XL/R = 461.63/100 = 4.6163

Band width (BW) of the pass-band filter: this is the range of frequencies for which the current (or output voltage) is equal to or greater than 70.7% of its value at the resonant frequency. The frequency at which the output of a filter is 70.7% of its maximum are called “cut-off frequencies” and are rebelled as f_1 and f_2 alongside (lower and upper) of the resonant frequency f_r. Other names are *-3dB frequencies, critical frequencies and half-power frequencies.*

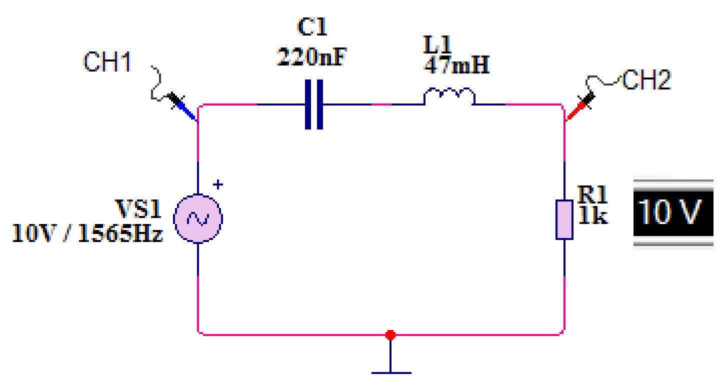
Band width, BW = f_r/Q = 1565/4.6163 = 339Hz; Also BW = f_2-f_1;

f_1 = f_r-BW/2 = 1565Hz-169.5Hz = 1395.5 Hz;

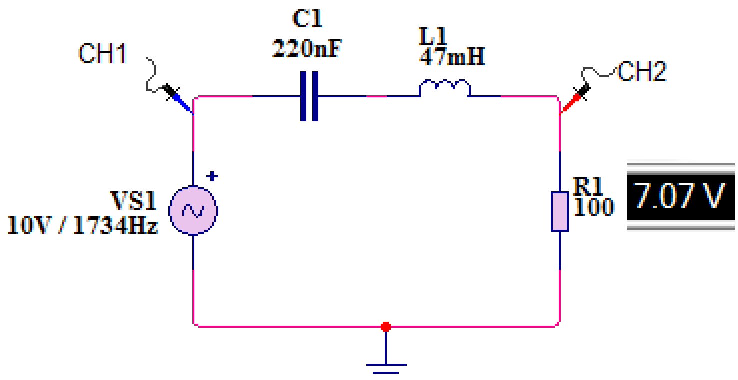
f_2 = f_r + BW/2 = 1565Hz +169.4Hz = 1734.5 Hz



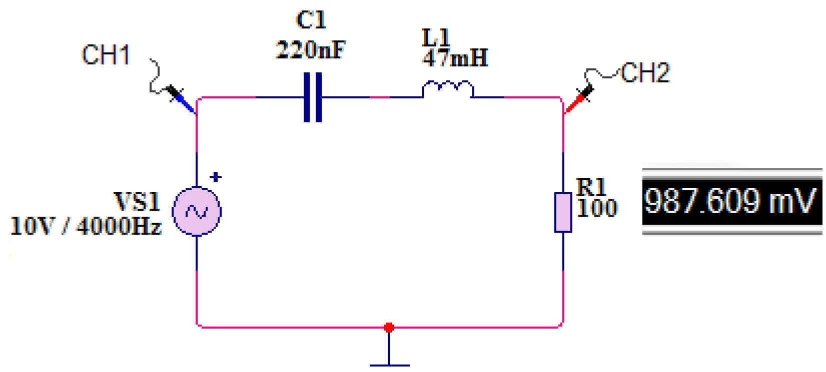
(a) As the frequency increases to f_1, V_{out} increases to 7.07V



(b) As the frequency increases from f_1 to f_r, V_{out} increases from 7.07 V to 10 V



(c) As the frequency increases from f_r to f_2, V_{out} decreases from 10V to 7.07V



(d) As the frequency increases above f_2, V_{out} decreases below 7.07V

Figure 3-17: Series resonant band-pass filter. (a), (b), (c), (d) Are examples of the frequency response of a series resonant band-pass filter with the input voltage at a constant 10 V rms. The winding resistance of the coil is neglected

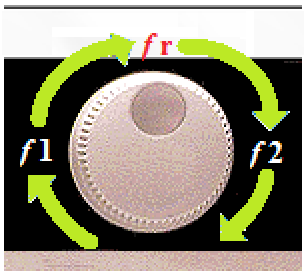
chap 3

Procedure

1. Connect a 100 Ω resistor, 47 mH inductor and (no polarized) 220nF capacitor in series with a signal generator as shown in the circuit figure 3-19.
2. Make sure the oscilloscope ground and the signal generator ground are connected together.
3. Set the function generator to output a 10V rms sine wave.
4. Set both channel 1 (CH1) and channel 2 (CH2) to 5Volt/Division
5. Connect the channel 1 (CH1) to the input of the circuit and the channel 2 (CH2) at the output (on the terminal of the resistor)
6. Change the frequencies by using the knob shown below and take the resulting data to form a table of V_s (input voltage) and V_{out} (voltage on 100Ω resistor R_1) for a range of frequencies, f, from about $f_r/2$ to $2f_r$ or for more exercise from 100Hz to 6Khz with 100Hz interval

Note that for above circuit, f_1 and f_2 are the frequencies where V_{out}/V_s falls to $1/\sqrt{2}$ of the peak value of the input.

7. From obtained result, plot a graph of normalized values of V_{out}/V_s versus frequency. (here normalized means divide all the V_{out}/V_s values by the maximum value which occurs at f_r. This means when the graph is plotted its peak value will appear to be unit.)



Knob used to adjust the value

Frequencies between f_1 and f_2 are passed through the filter with amplitudes not less than 70.7% (≤7.07V≥) of maximum. The frequencies outside pass-band are reduced to less than 70.7% (<7.07V) of the maximum and are considered to be rejected.

We ideally assume that a resonant circuit accepts frequencies within its bandwidth and completely eliminates frequencies outside the bandwidth. Such is not the case, however, because signals with frequencies outside the bandwidth are not completely eliminated. Their magnitude, however, are greatly reduced.

The further the frequencies are from the cut-off frequencies, the greater the reduction as illustrated in graphs figure 3-18.

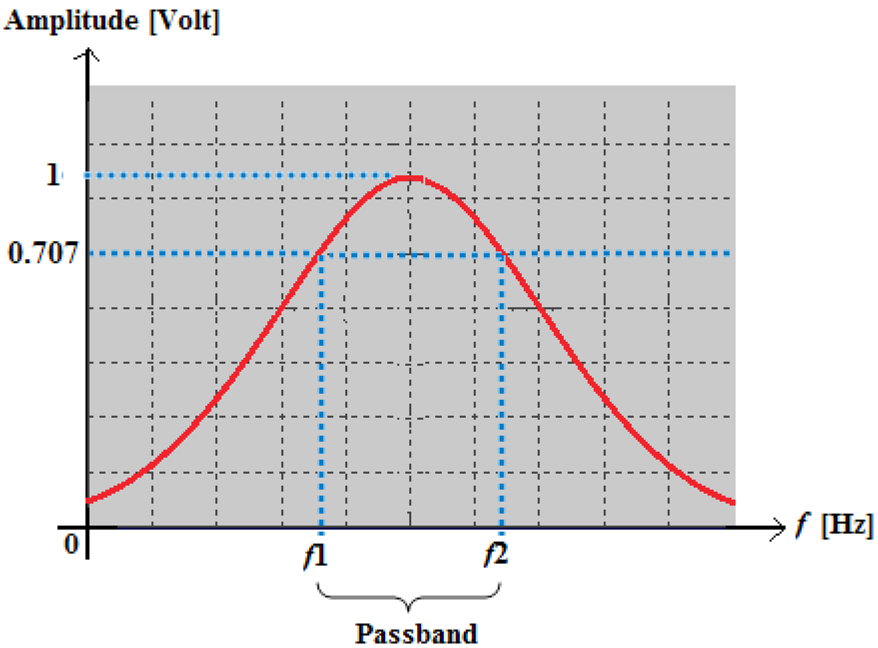


Figure 3-18: Ideal generalized selectivity curve of a band-pass filter

Frequencies from f_1 and f_2 are assumed to be passed equally. All frequencies outside the passband are eliminated.

Compare the practical result to the theory you may well find that these values are not exactly the same. Part of the reason for this difference is the fact that inductor also has a resistance, which you have not taken into account.

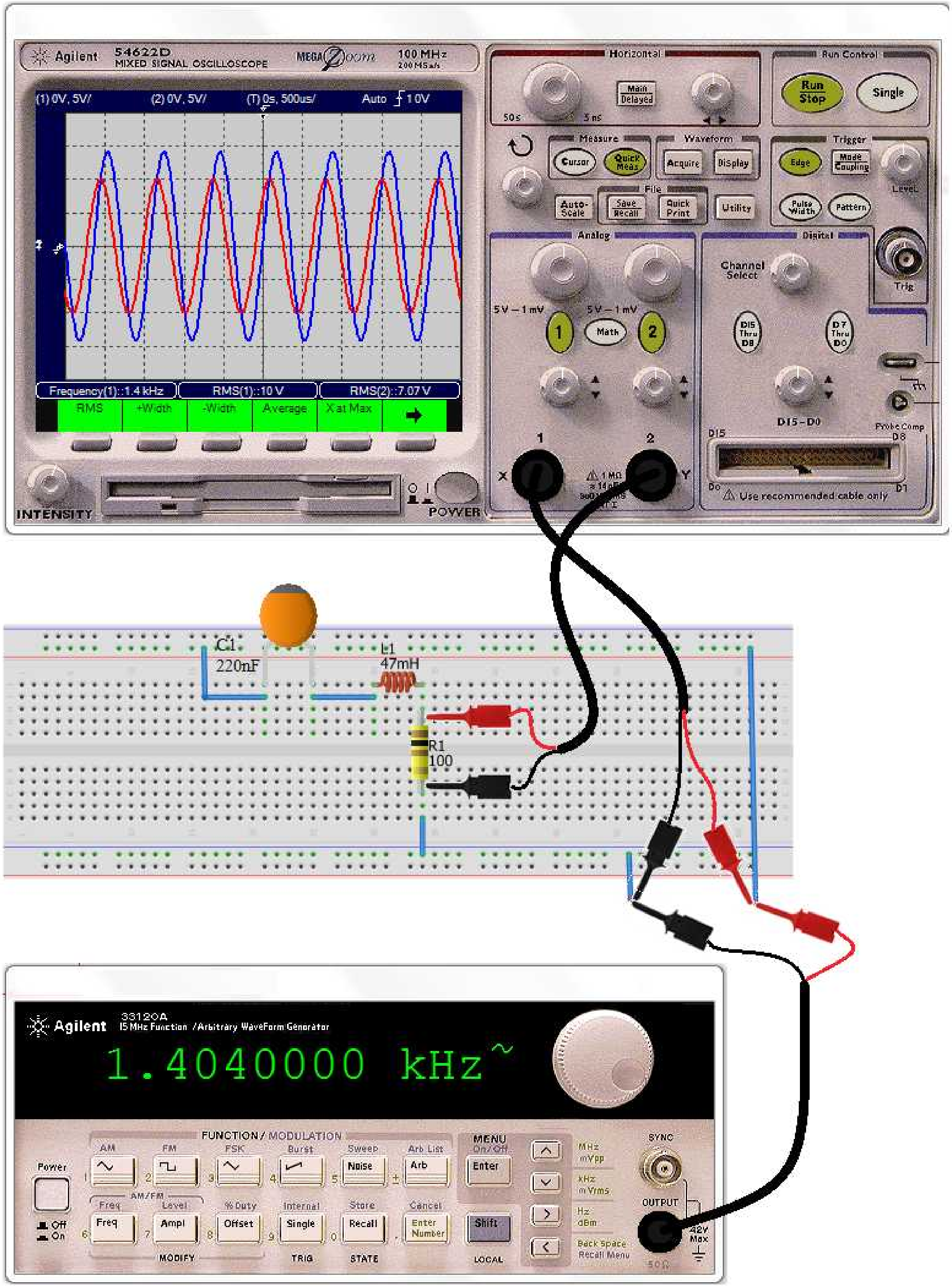


Figure 3-19: Series resonant band-pass filter, real world circuit connection

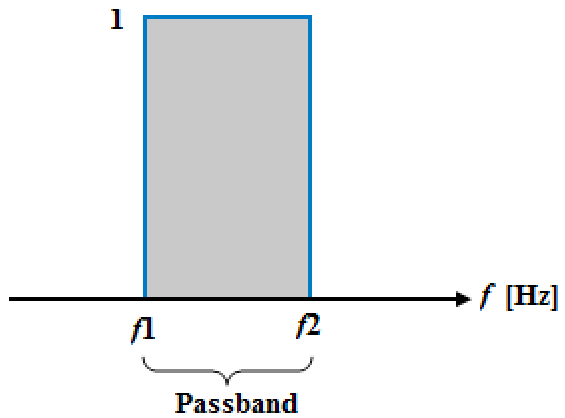


Figure 3-20: Generalized response curve of a series resonant band-pass filter

The other thing, the resistor in this circuit is the resistance of the inductor plus any resistance contributed by the function generator. Normally the function generator has an output impedance of 50Ω. Verify that this is the case by measuring the resistance of the function generator output with the generator turned on but the output voltage set to 0 V.

3. 3. 2. The Band-Stop Filter

The band-stop filter rejects signals with frequencies between the lower and upper cut-off frequencies and passes those signals with frequencies below and above the cut-off values, as shown in response curve figure 3-22. The range of frequencies between the lower and upper cut-off points is called the *stopband*. This type of filter is also referred to as a *band-elimination filter, band-rejection filter,* or a *notch filter*.

The basic series resonant band-stop filter is shown in the figure 3-21. Notice that the output is taken across the *LC* portion of the circuit. This filter is still a series *RLC* circuit, just as the band-pass filter is. The difference is that in this case, the output voltage is taken across the combination of *L* and *C* rather than *R*.

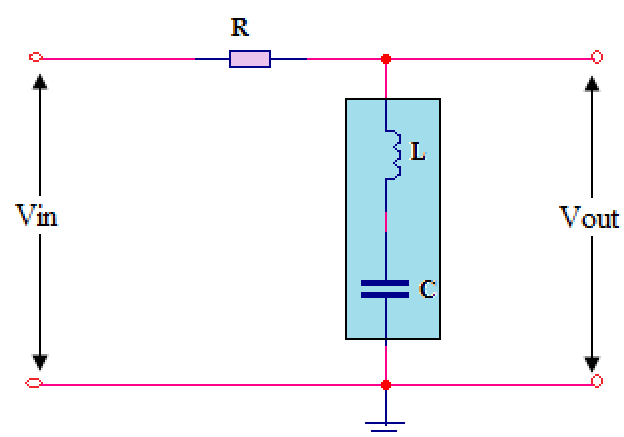


Figure 3-21: A basic circuit of series resonant band-stop filter

All the characteristics that have been discussed in relation to the band-pass filter apply equally to the band-stop filter, with the exception that the response curve of the output voltage is opposite.

For the band-pass filter, V_{out} is maximum at resonance. For the band-stop filter, V_{out} is minimum at resonance.

At very low frequencies, the *LC* combination appears as a near open due to the high Xc, thus allowing most of the input voltage to pass through to the output. As the frequency increases, the impedance of the *LC* combination decreases until, at resonance, it is zero (ideally). Thus, the input signal is shorted to ground, and there is very little output voltage. As the frequency goes above its resonance value, the *LC* impedance increases, allowing an increasing amount of voltage to be dropped across it. The general frequency response of a series resonant band-stop filter is illustrated in figure 3-22.

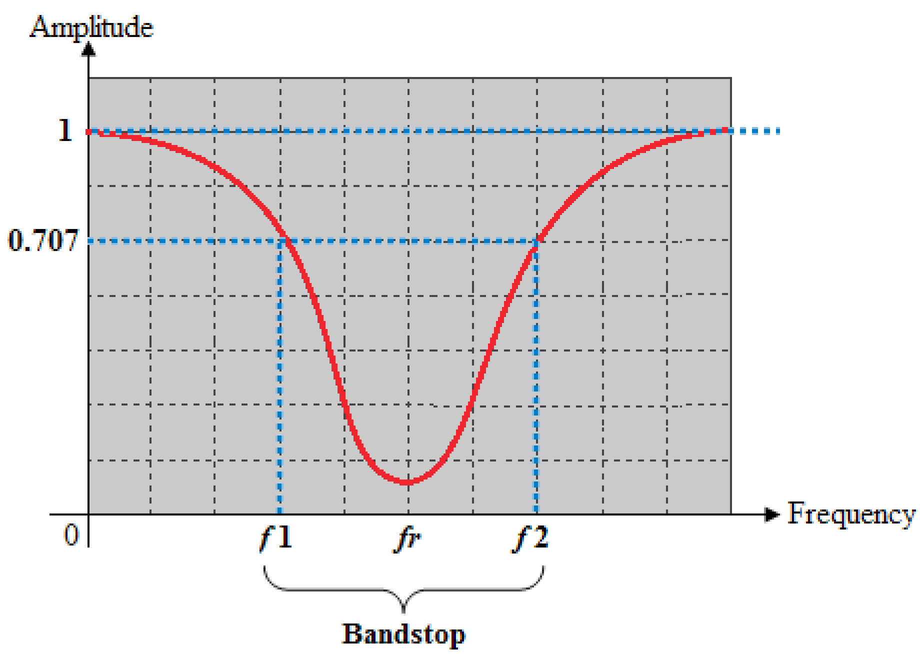


Figure 3-22: Generalized response curve for a band-stop filter

Experiment 3-6: Series RLC Band-stop filter

Design a band-stop filter supplied with V_{in} at a constant 10V rms by neglecting the winding resistance.

Parts and materials

No	Item	Specification	Quantity
1	Oscilloscope	Agilent 54622D digital oscilloscope	1
2	0.01uF Capacitor	0.01uF no polarized capacitor	1
3	Breadboard	Prototyping board (RSR 03MB102)	1
4	Connecting wire	22-gaude solid wire	10cm
5	100Ω and 6.8KΩ resistors	100Ω and 6.8KΩ 1/2watt resistors	1
6	Inductor 100mH	Inductor 100mH	1

Calculations of resonance frequency f_r and bandwidth *BW*

f_r = 1/2π√(LC) = 1/2*3.14*√(100mH*0.01uF) = 5.033KHz

Inductance reactance, XL= 2πfL

$= 2*3.14*5.03KHZ*100mH = 3160\Omega$

Quality factor, Q = XL/R

$= 3160\Omega/47\Omega = 67.234$

Bandwidth, BW = f_r/Q

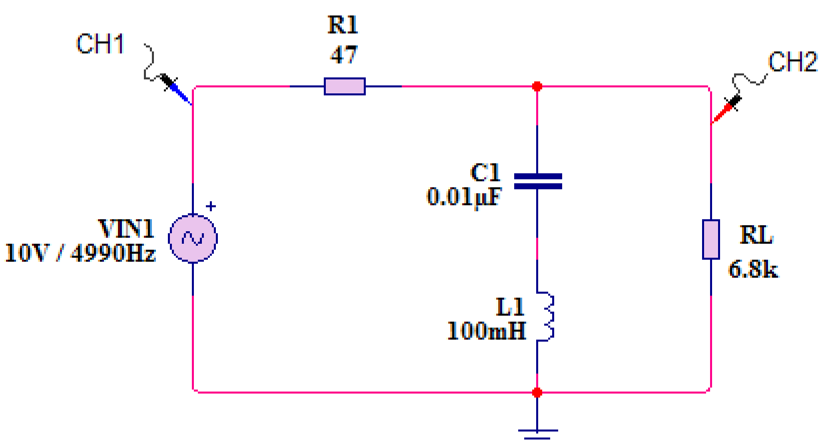
$= 5.033KHZ/67.234 = 74Hz$

$f_1 = f_r - BW/2$

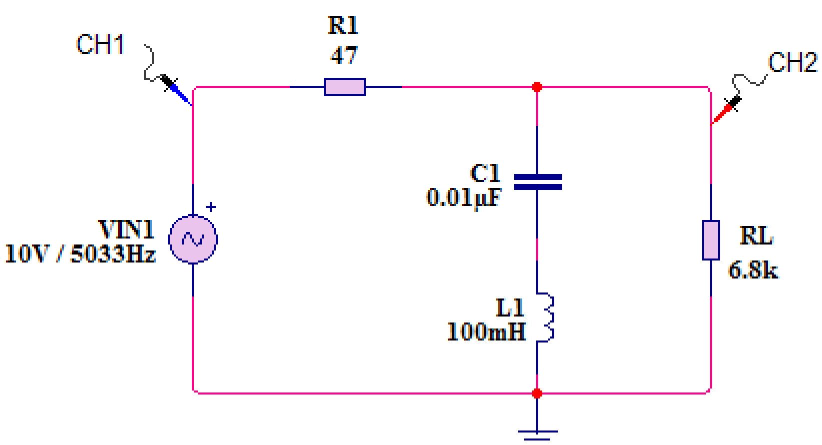
$= 5033-74/2 = 4993Hz$

$f_2 = f_r + BW/2$

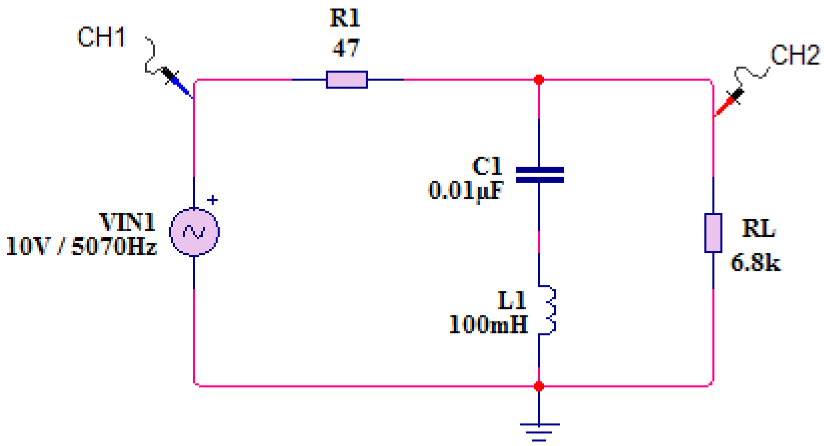
$= 5033+74/2 = 5070Hz$



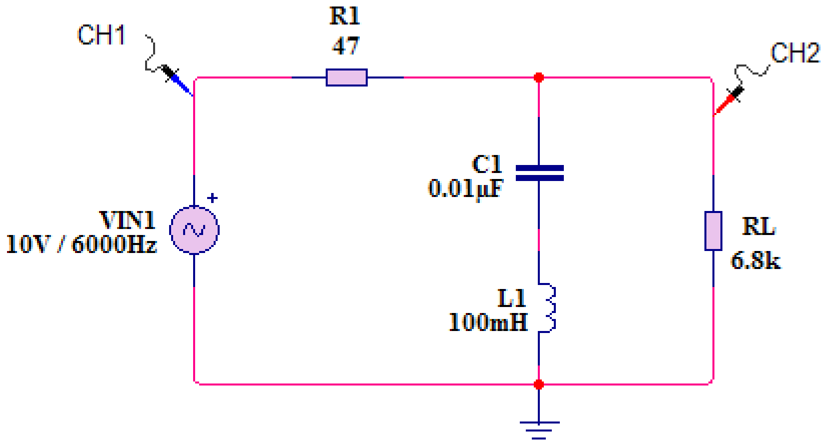
(a) As the frequency increases to f_1, V_{out} decreases from 10V to 7.07V



(b) As frequency increases from f_1 to f_r, V_{out} decreases from 7.07V to 0V



(c) As frequency increases from f_r to f_2, V_{out} increases from 0V to 7.07V



(d) As frequency increases above f_2, V_{out} increases from 7.07V to 10V

Figure 3-23: Series resonant band-stop filter. (a), (b), (c) and (d) are examples of frequency response of a series resonant band-stop filter with V_{in} at a constant 10V rms. The winding resistance is neglected

chap 3

Procedure

1. Connect 47Ω R_1 resistor from ac supply from function generator, 100mH inductor L_1 and 0.01uF (no polarized) capacitor C_1 so that the load resistor R_L 6.8 KΩ be taken across the series combination of C_1 and L_1.
2. Make sure the oscilloscope and the function generator's ground are connected together.
3. Set the function generator to output a 10V rms sine wave.
4. Set both channel 1 (CH1) and channel 2 (CH2) to 5Volt/Division.
5. Connect the channel 1 (CH1) to the input of the circuit and the channel 2 (CH2) at the output (on the terminal of the load resistor R_L)
6. Press on frequency button on the function generator. Change the frequencies by using the knob shown below and note the resulting output voltage V_{out} (voltage on 6.8 KΩ resistor R_L) for a range of frequencies, f, from below about f_1=4990Hz to to above f_2 =5070Hz. Record the frequency at which V_{out} is equal to 7.07V, about 0V and about 10V.

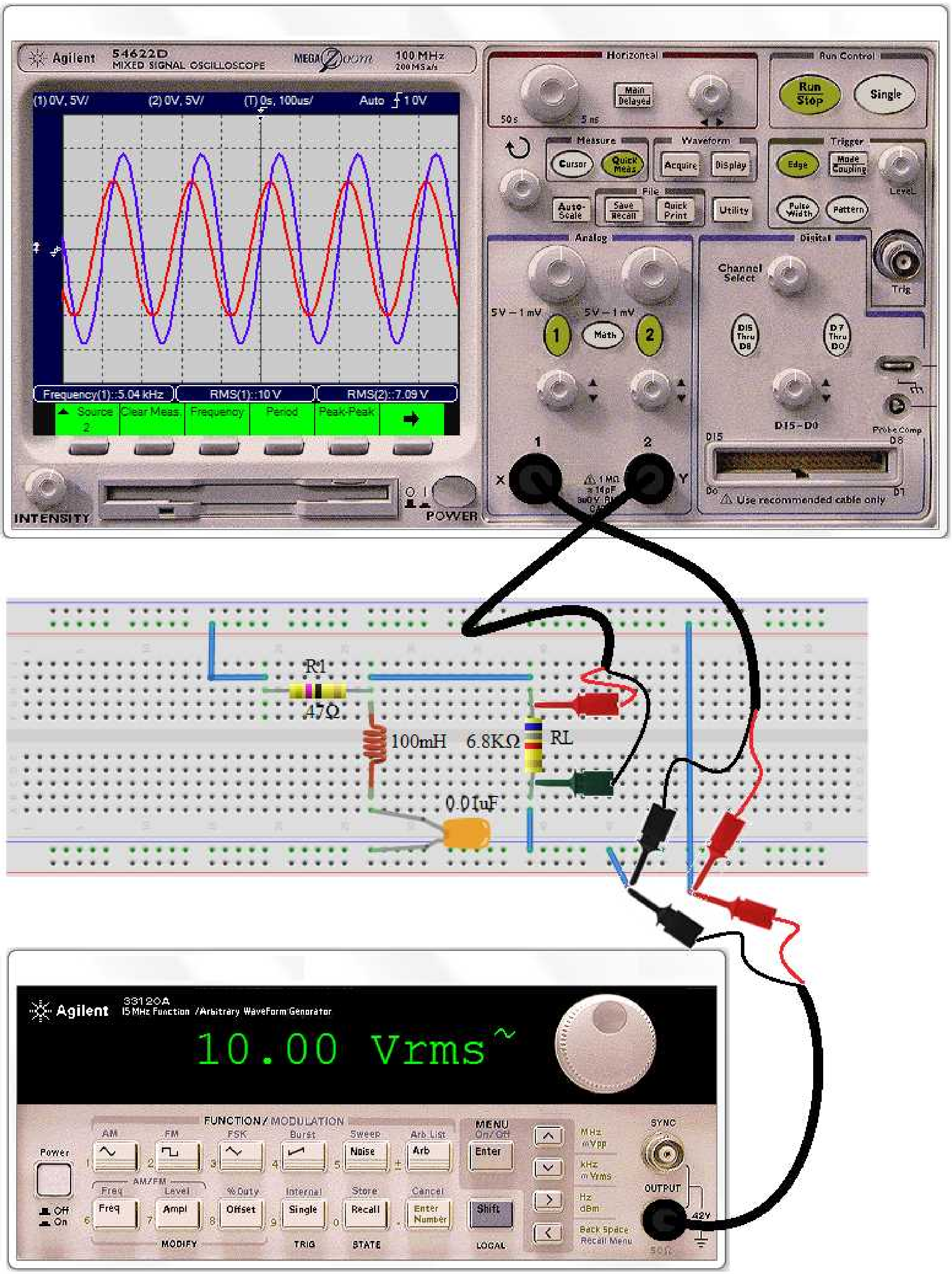
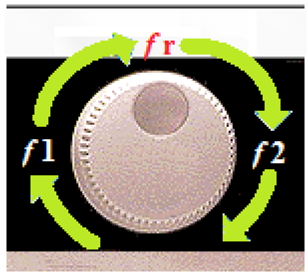


Figure 3-24: Circuit connection for a series resonant band-stop filter

7. From obtained result, plot a graph of normalized values of V_{out}/V_{in} versus frequency. (Here normalized means divide all the V_{out}/V_{in} values by the maximum value which occurs at f_r. This means when the graph is plotted its peak value will appear to be unit).



Knob used to adjust the value

For a given filter circuit f_1 and f_2 are the frequencies where V_{out}/V_{in} falls to $1/\sqrt{2}$ of the peak value of the input.

Frequencies between f_1 and f_2 are passed through the filter with amplitudes no great than 70.7% (≤7.07V≥) of maximum. The frequencies inside band-stop are reduced to less than 70.7% (<7.07V) of the maximum and are considered to be rejected.

We ideally assume that a resonant circuit accepts frequencies outside its bandwidth and completely eliminates frequencies inside the bandwidth.

3. 4. Parallel Resonant Filters

Parallel resonant circuits are commonly applied to band-pass and band-stop filters. In this section we will examine these applications.

3. 4. 1. The Band-Pass Filters

A basic parallel resonant band-pass filter is shown in figure 3-25. Notice that the output is taken across the tank circuit in this application.

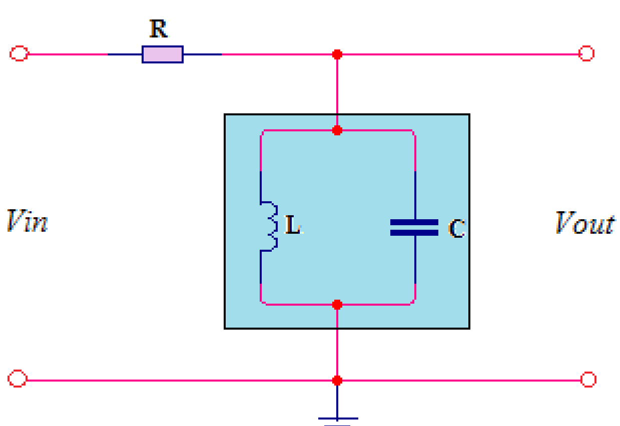


Figure 3-25: A basic parallel resonant band-pass filter circuit

The bandwidth and cut-off frequencies for a parallel resonant band-pass filter are defined in the same way as for the series resonant circuit, and the formulas given in that section still apply. General band-pass frequency response curves showing both V_{out} and I_{tot} versus frequency are given in figure 3-26 part (a) and (b) respectively.

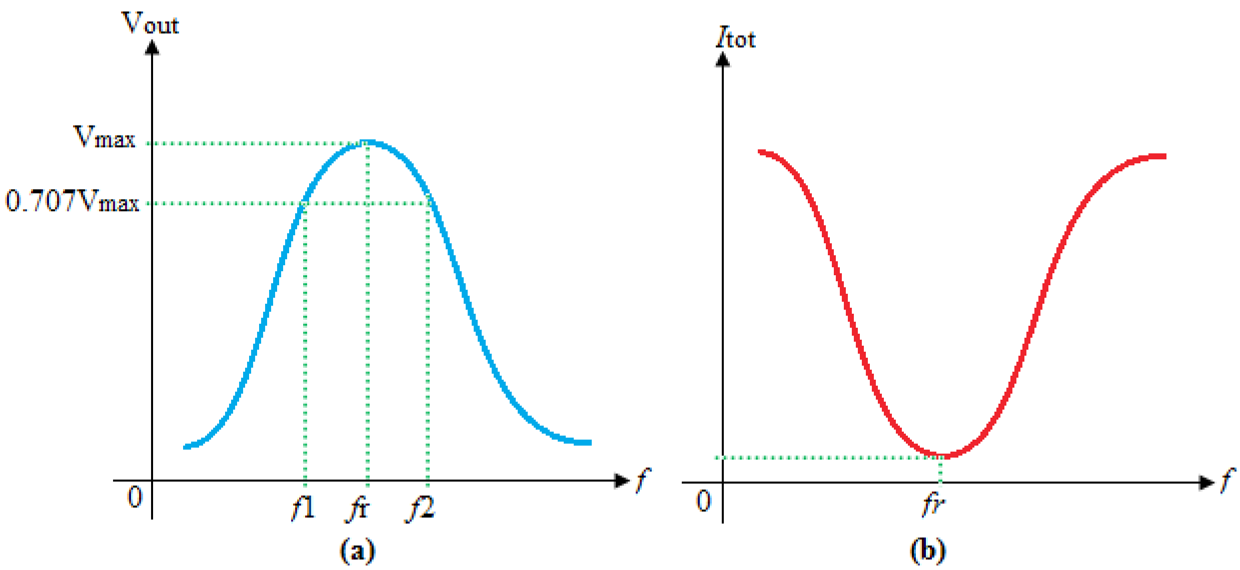


Figure 3-26: Generalized frequency response curve for a parallel resonant band-pass filter. (a) V_{out} versus frequency, I_{tot} versus frequency

The filtering action is as follow: At very low frequencies, the impedance of the tank circuit is very low, and therefore only a small amount of voltage is dropped across it. As the frequency increases, the impedance of the tank circuit increases, and, as a result, the output voltage increases. When the frequency reaches its resonant value, the impedance is at its maximum and so is the output voltage. As the frequency goes above resonance, the impedance begins to decrease, causing the output voltage to decrease.

The expression for the equivalent inductance and equivalent parallel resistance is given as:

$$R_{p(equ)} = R_w(Q^2 + 1)$$

The overall Q (quality factor), designated Q_0, for parallel RLC circuit is expressed differently from the Q of the series circuit.

$$Q_0 = R_{p(total)} / XL_{(eq)}$$

The general response of a parallel resonant band-pass filter is illustrated in the experiment. This illustration shows how the current and the output voltage change with frequency.

Experiment 3-7: Parallel band-stop filter

Design and examine a passive parallel band-stop filter

Parts list

No	Item	Specifications	Quantity
1	Resistor 680Ω	Fixed resistor 680Ω, ½ watt	1
2	Inductor 100mH,	Inductor 100mH, 50 Ω winding resistance	1

3	Capacitor 0.1µF	Ceramic capacitor 0.1µF	1
4	Breadboard	Prototyping board	1
5	Oscilloscope	Digital oscilloscope (Agilent 5462D)	1
6	AC source with variable frequency	Function generator	1
7	Ammeter	Digital multimeter	1
8	Load resistor 47KΩ	Fixed resistor 47KΩ, ½ watt	1

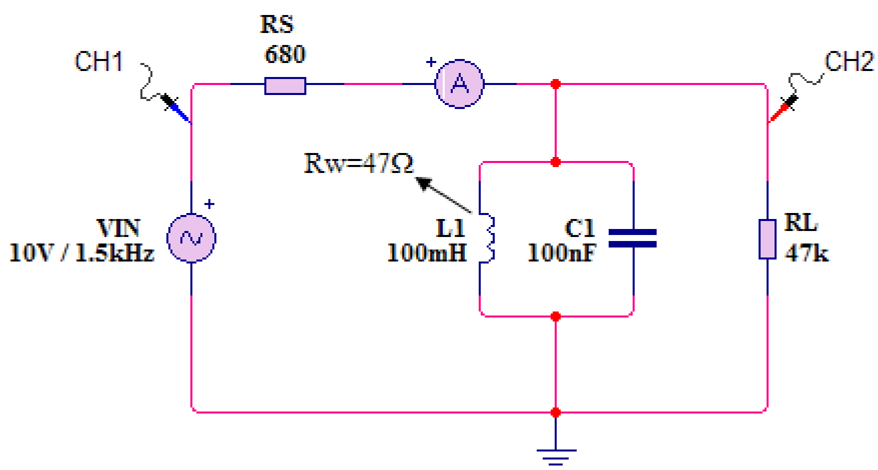


Figure 3-27: Parallel resonant band-pass filter circuit

Calculate the bandwidth at resonance for unloaded filter:

Resonant frequency, $f_r = 1/2\pi\sqrt{LC}$ = 1.591 KHz

Inductance reactance, XL = $2\pi f_r L$

= 2*3.14*1591*0.1 = 999 Ω

Quality factor, Q = XL/R_w

= 999 Ω /50 Ω = 20

Bandwidth, BW = f_r/Q

= 1591/20 = 79.5 Hz

Calculate the output voltage V_{out}:

$R_{p(equ)} = R_w(Q^2 + 1)$

$= 47\Omega\ (20^2 + 1) = 18{,}847\ \Omega$

$V_{out} = V_{in}\ (R_{p(equ)} / R_{p(equ)} + R_s)$

= 10V (18847/18847 + 680) = 9.65V

When a 47 KΩ load resistor is connected, you obtain the following values. The resonant frequency is not affected.

chap 3

$R_{p(total)} = R_{p(equ)} \| R_L$
$= 18847*47000/18847+47000 = 13452\ \Omega$

$Q_0 = R_{p(total)} / XL = 13452 / 999 = 13.46$

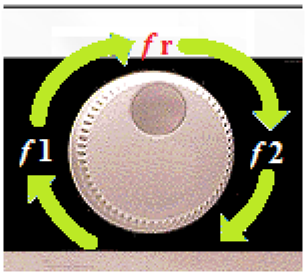
$BW = f_r / Q_0 = 1591Hz/13.46 = 118\ Hz$

$V_{out} = V_{in} (R_{p(total)} / R_{p(total)} + R_s)$
$= 10V (13452/13452+680) = 9.52\ V$

Note: The result of adding the load resistance is in increased bandwidth and a decreased output voltage.

Procedure

1. Connect the circuit as illustrated in figure 3-28.
2. Connect channel one of the oscilloscope to the input of the circuit and channel two to the load resistor R_L. Connect the ammeter in series with the R_S resistor and the tank circuit. Set the ammeter in the ac amp setting.
3. Set the function generator to output 10V sinusoidal.
4. Press to the quick measure on the oscilloscope to measure the input voltage, output voltage and the frequency.
5. Adjust the frequency on the function generator using the knob shown below and observe the output voltage reading on the oscilloscope channel two and also the reading on the ammeter.



6. Note the frequency at which V_{out} is 7.07 V, this is cut-off frequency. Let f_1 be the cut-off frequency less than 1591Hz (resonant frequency) and f_2 the cut-off frequency greater than resonant frequency.
7. As the frequency increase to f_1, V_{out} increases to 7.07V and I_{tot} (current reading on ammeter) decreases.
8. As the frequency increases from f_1 to f_r, V_{out} increases from 7.07 V to 10 V and I_{tot} decreases to its minimum value.
9. As the frequency increases from f_r to f_2, V_{out} decreases from 10V to 7.07V and I_{tot} increases to from its minimum.
10. As the frequency increases above f_2, V_{out} decreases below 7.07V and I_{tot} continue to increase. With obtained V_{out} and I_{tot} with function to the frequency, draw the response curve.

chap 3

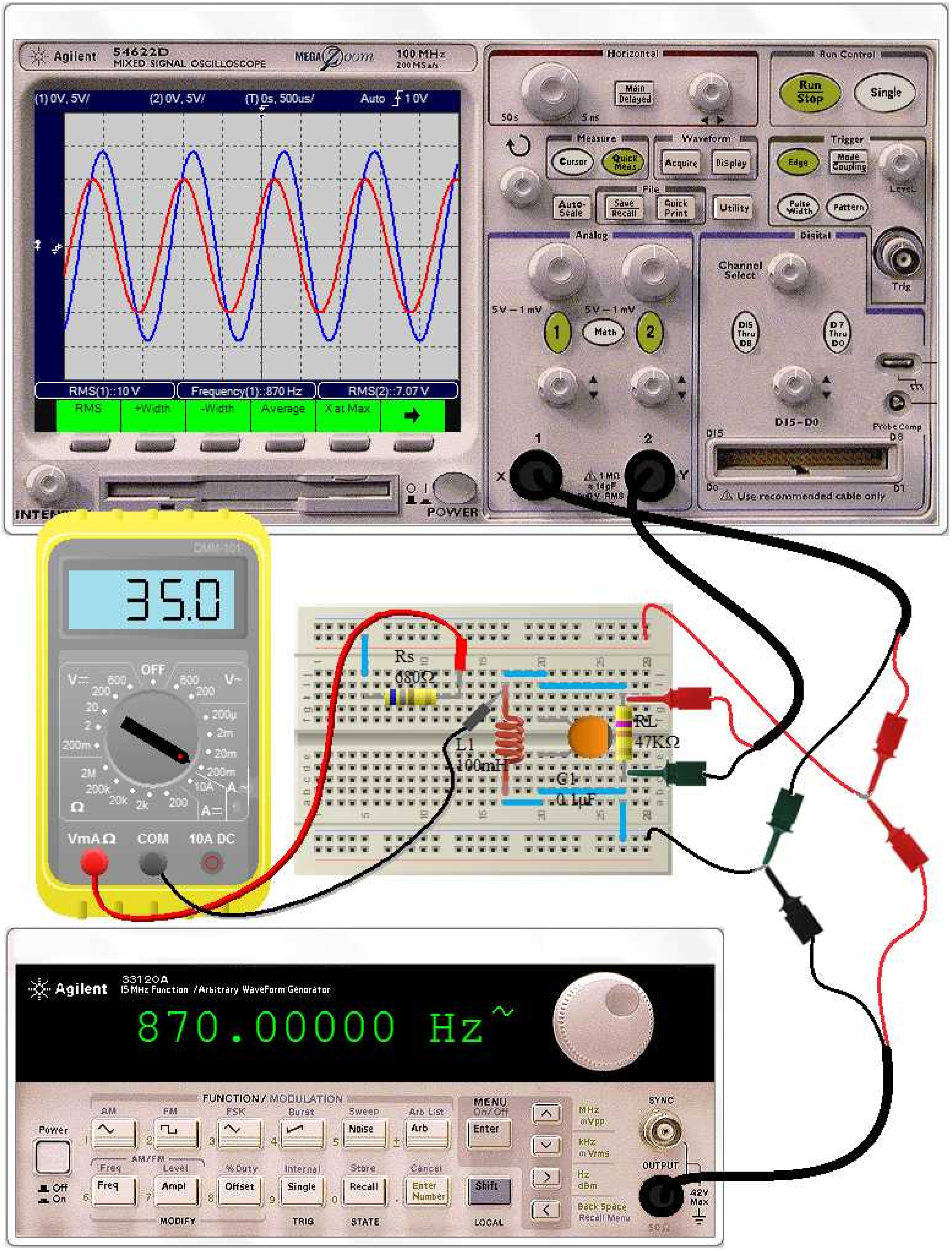


Figure 3-28: Parallel resonant band-pass filter, real world circuit connection

3. 4. 2. The Band-Stop Filter

A basic parallel resonant band-stop filter is illustrated in figure 3-29. The output is taken across a load resistor in series with the tank circuit.

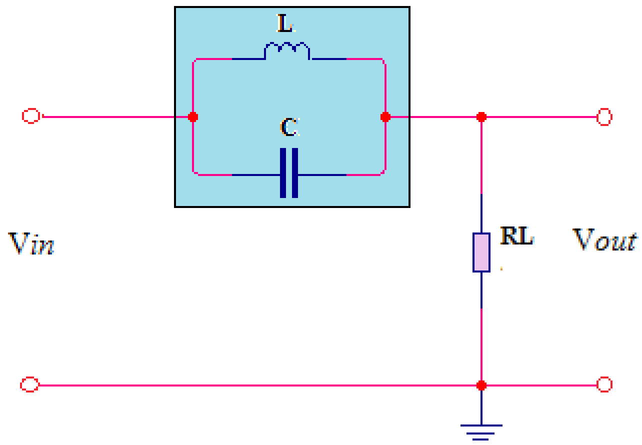


Figure 3-29: A basic parallel resonant band-stop filter

The variation of the tank circuit impedance with frequency produces the familiar current response that has been previously discussed; that is, the current is minimum at resonance and increases on both sides of resonance. Since the output voltage is across the series load resistor, the output voltage follows the current, thus creating the band-stop response characteristic, as indicated in figure 3-30.

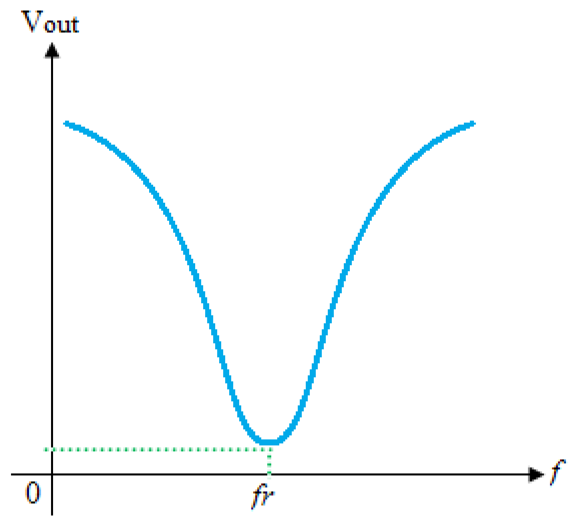


Figure 3-30: Parallel resonant band-stop filter response

Actually, the band-stop filter in figure 3- 30 can be viewed as a voltage divider created by Zr (impedance) of the tank and the load resistance. Thus, the output voltage at f_r is

$$V_{out} = V_{in}(R_L/R_L + Zr);\ \text{with } Zr = R_w(Q^2 + 1) \text{ for non ideal tank circuit at resonance.}$$

Applications of Filters

Resonant circuits are used in a wide variety of applications, particularly in communication system. In this section, we will look briefly at a few common communication systems applications to illustrate the importance of resonant circuits in electronic communication.

- **Tuned Amplifier**

A tuned amplifier is a circuit that amplifies signals with a specified band. Typically, a parallel resonant circuit is used in conjunction with an amplifier to achieve the selectivity. In terms of the general operation, input signals with frequencies that range over a wide band are accepted on the amplifier's input and are amplified. The resonant circuit allows only a relatively narrow band of those frequencies to be passed on. The variable capacitor allows tuning over the range of input frequencies so that a desired frequency can be selected.

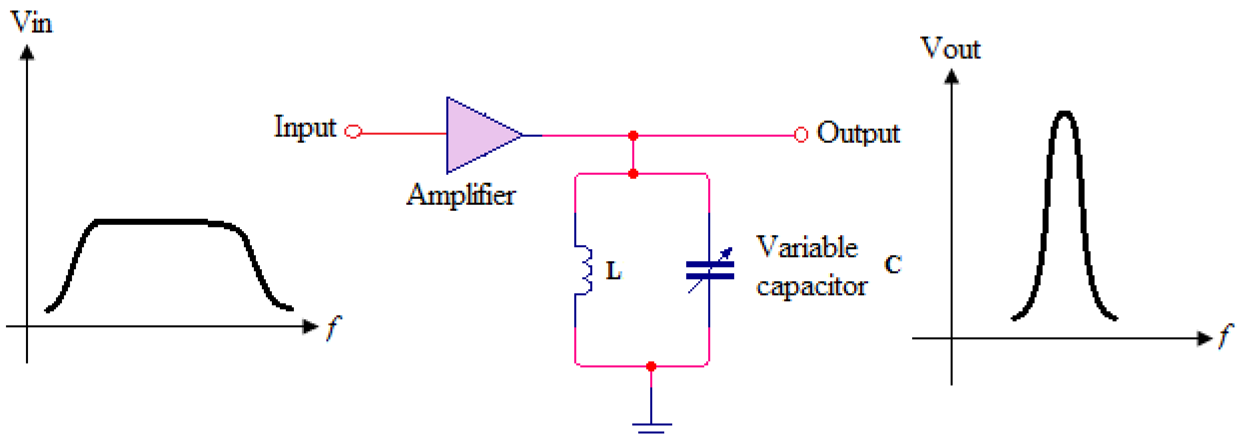


Figure 3-31: A basic tuned band-pass amplifier

- **Antenna Input to a Receiver**

Radio signals are sent out from a transmitter via electromagnetic waves that propagate through the atmosphere. When the electromagnetic waves cut across the receiving antenna, small voltages are induced. Out of the wide range of electromagnetic frequencies, only one frequency or limited band of frequency must be extracted. The figure 3-32 shows typical arrangement of an antenna coupled to the receiver input by a transformer. A variable capacitor is connected across the transformer secondary to form a parallel resonant circuit.

- **Signal Reception and Separation in a TV receiver**

A standard broadcast band television receiver must handle both video (picture) signals and audio (sound) signals. Each TV transmitting station is allocated a 6MHz bandwidth. Channel 2 is allocated 54MHz through 59MH, channel 3 is allocated a band from 60MHz through 65MHz, on up to channel 13, which has a band from 210MHz through 215MHz. you can tune the front end of the TV receiver to select any one of these channels by using tuned amplifier. The signal output of the front end of the receiver has a band-width from 41MHz through 46MHz, regardless of the channel that is tuned in. This band, called the *intermediate frequency* (IF) band, contains both video and audio. Amplifiers tuned to the IF band boost the signal and feed it to the video amplifier.

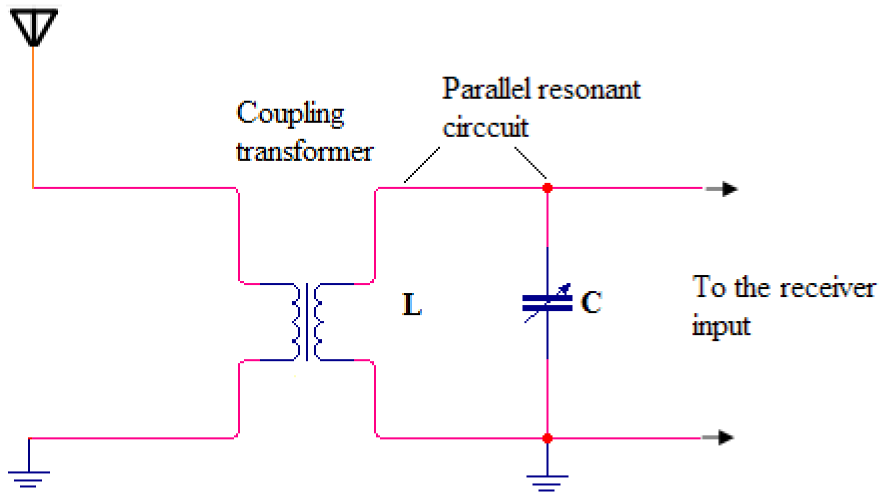


Figure 3-32: Resonant coupling from an antenna

Before the output of the video amplifier is applied to the cathode-ray tube, the audio signal is removed by 4.5MHz band-stop resonant circuit (called a wave trap). This trap keeps the sound signal from interfering with the picture. The video amplifier output is also applied also to band-pass circuits that are tuned to sound carrier frequency of 4.5MHz. The sound signal is then processed and applied to the speaker as indicated in figure 3-33.

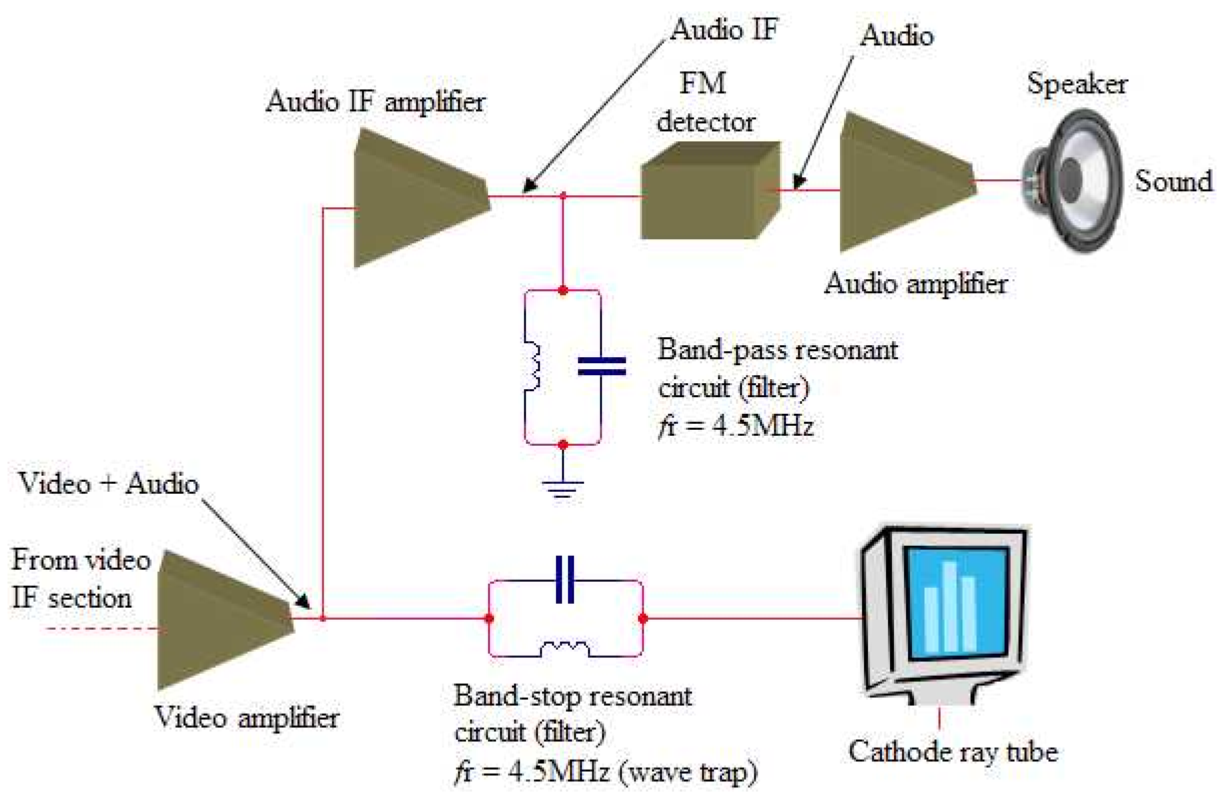


Figure 3-33: A simplified portion of a TV receiver showing filter usage

Chapter four

Electromagnetic Devices and Applications

Objectives

After completing this chapter, you should be able to

- Discuss how speaker works
- Discuss how microphone works
- Differentiate different types of microphones
- Describe and test the transformer
- Have an understanding on DC motor and speed and direction control principle
- Discuss on solenoid
- Discuss and testing the electromechanical relay

Further reading

Study aids for this chapter are available at

- Talbot-Smith, Michael (2013). *Audio Engineer's Reference Book*. CRC Press.
- William H. Yeadon, Alan W. Yeadon. Handbook of small electric motors. McGraw-Hill Professional, 2001.

Many types of useful devices such as tape recorder, electric motors, speaker and microphones, solenoids, and relays are based on electromagnetism. The transformer is another important example and will be covered in this chapter. The operation of many types of electrical devices is based partially on magnetic or electromagnetic principles.

Electromagnetic induction is important in an electrical component called inductor or coil. Two types of magnets are the permanent magnet and the electromagnet. The permanent magnet maintains a constant magnetic field between its two poles with no external excitation. The electromagnet produces a magnet field only when there is current through it. The electromagnet is basically a coil of wire wound around a magnetic core material. This chapter describes most of encountered electromagnetic devices and their applications.

4. 1. The Speaker

A speaker (or loudspeaker) is an electromagnetic device that converts electrical signals to sound waves. Permanent magnet speakers are commonly used in stereos, radios, TVs, and their operation is based on the principle of electromagnetism. Sound can also be used to produce an alert noise or act as an alarm, and loudspeakers, buzzers, horns and sounders are all types of sound transducer that can be used for this purpose with the most commonly used audible type output sound actuator being the "Loudspeaker". Loudspeakers are available in all shapes, sizes and frequency ranges with the more common types being moving coil, electrostatic, isodynamic and piezo-electric. Moving coil type loudspeakers are by far the most commonly used speaker in electronic circuits, kits and toys, and as such it is this type of sound transducer we will examine.

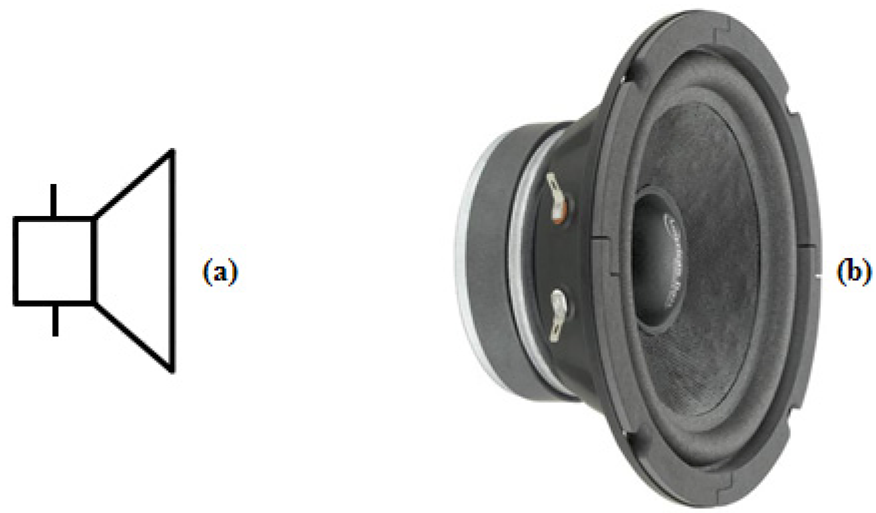


Figure 4-1: Loudspeaker; (a) circuit symbol, (b) typical moving coil loudspeaker

chap 4

The principle of operation of the Moving coil loudspeaker is the exact opposite to that of the "Dynamic Microphone". A coil of fine wire, called the "speech or voice coil", is suspended within a very strong magnetic field, and is attached to a paper or Mylar cone, called a "diaphragm" which itself is suspended at its edges to a metal frame or chassis. Then unlike the microphone which is pressure sensitive input device, this type of sound transducer can be classed as a pressure generating output device.

Working principle of the loudspeaker

A typical speaker is constructed with a permanent magnet and an electromagnet. The cone of speaker consists of a paper-like flexible diaphragm to which is attached a hallow cylinder with a coil around it, forming an electromagnet. One of the poles of the permanent magnet is positioned with the cylinder coil.

When an analog signal passes through the voice coil of the speaker, an electro-magnetic field is produced and whose strength is determined by the current flowing through the "voice" coil, which in turn is determined by the volume control setting of the driving amplifier or moving coil driver.

The electro-magnetic force produced by this field opposes the main permanent magnetic field around it and tries to push the coil in one direction or the other depending upon the interaction between the north and south poles.

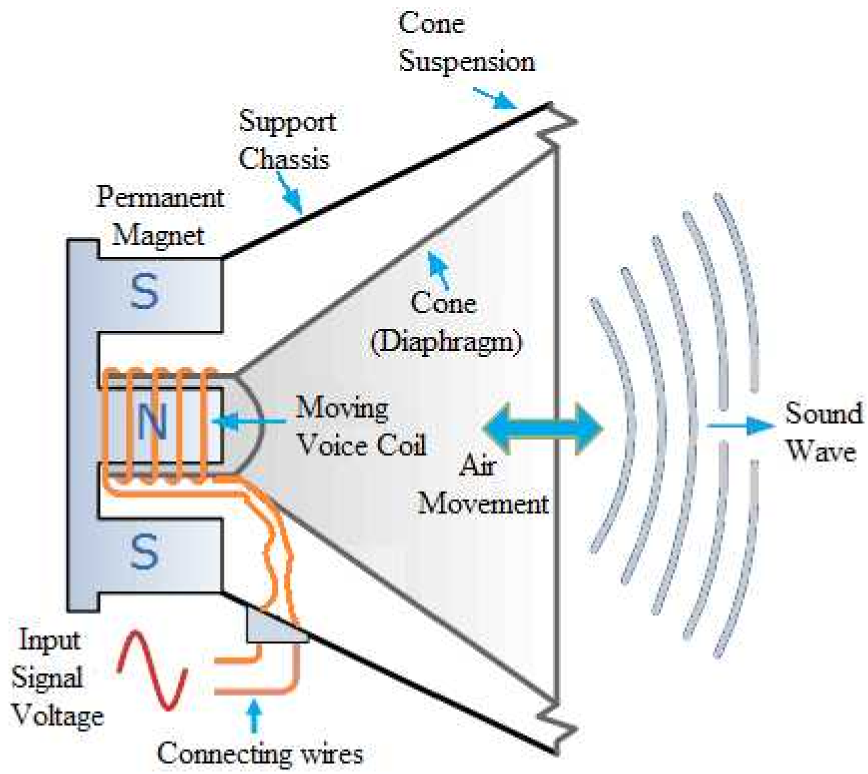


Figure 4-2: Cross-section of the loudspeaker

chap 4

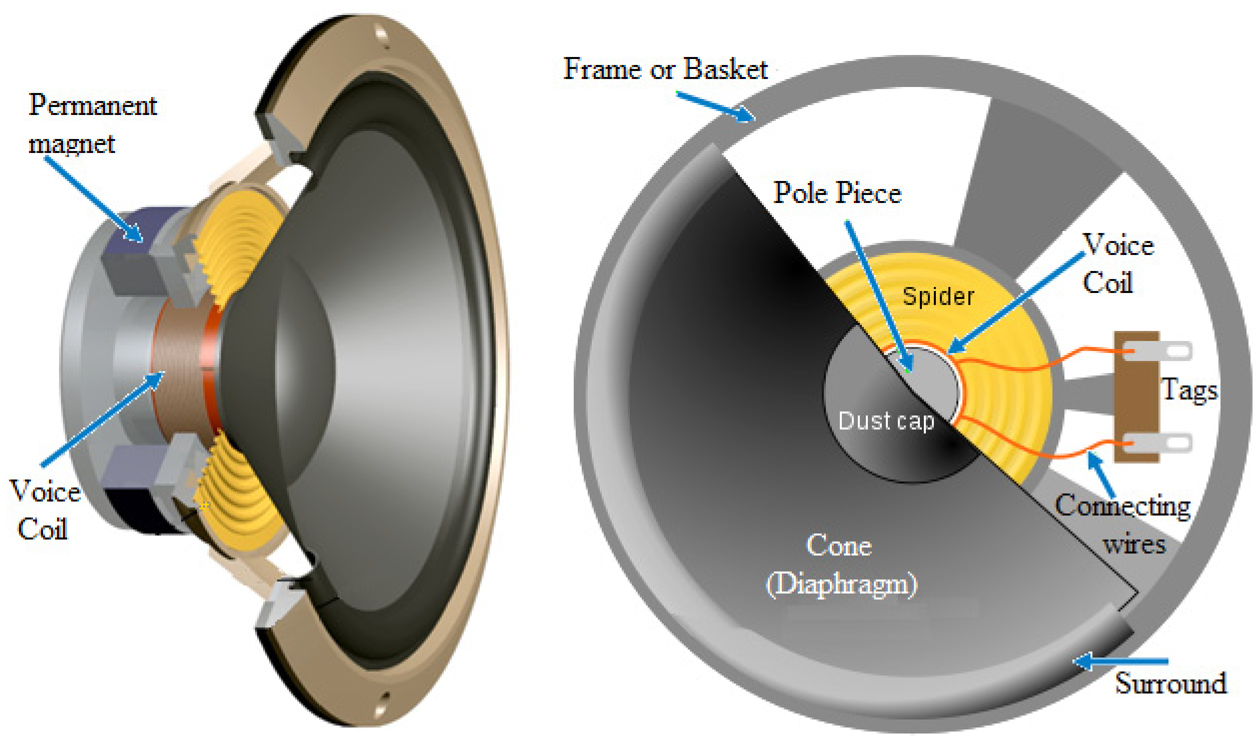


Figure 4-3: Moving coil loudspeaker cutaway view

As the voice coil is permanently attached to the cone/diaphragm this also moves in tandem and its movement causes a disturbance in the air around it thus producing a sound or note. If the input signal is a continuous sine wave then the cone will move in and out acting like a piston pushing and pulling the air as it moves and a continuous single tone will be heard representing the frequency of the signal. The strength and therefore its velocity, by which the cone moves and pushes the surrounding air produces the loudness of the sound. In other words, when an audio signal voltage (music or voice) is applied to the speaker coil, the current varies proportionally in both direction and amount. In response, the diaphragm will vibrate in and out by varying amounts and at varying rates. Vibration in the diaphragm causes the air that is in contact with it to vibrate. These air vibrations move through the air as sound.

Experiment 4-1: Testing loudspeaker cone movement caused by the applied current

When the current passes through the voice coil of the speaker, an electro-magnetic field is produced. The electro-magnetic force produced by this field opposes the main permanent magnetic field around it and tries to push the coil in one direction or the other depending upon the interaction between the north and south poles. This is verified by applying a small DC voltage to the terminals of the loudspeaker and observing the movement of the cone with this by interchanging the battery's polarity.

Parts and material

No	Item
1	Moving coil loudspeaker 4W, 8 Ω
2	3V Battery

Procedure

1. When the positive terminal of the battery is connected to positive marked tag of the loudspeaker and the negative terminal of the battery connected to the negative marked tag of the loudspeaker, as shown in circuit connection figure 4-4, part (a), there is current through the coil in one direction, the interaction of the permanent magnet field with the electromagnet field causes the cone to move to the outward.
2. By interchanging the polarities of the battery as shown in circuit connection figure 4-4, part (b), the current through the coil causes the cone to move to the inwards.
3. The movement of the coil cylinder causes the flexible diaphragm also to move in or out, depending on the direction of the coil current. The amount of coil current determines the intensity of the magnet field which controls the amount that the diaphragm moves.

As the speech or voice coil is essentially a coil of wire it has, like an inductor an impedance value. This value for most loudspeakers is between 4 and 16Ω and is called the "nominal impedance" value of the speaker measured at 0Hz, or DC.

It is important to always match the output impedance of the amplifier with the nominal impedance of the speaker to obtain maximum power transfer between the amplifier and speaker. Most amplifier-speaker combinations have an efficiency rating as low as 1 or 2%.

Although disputed by some, the selection of good speaker cable is also an important factor in the efficiency of the speaker, as the internal capacitance and magnetic flux characteristics of the cable change with the signal frequency, thereby causing both frequency and phase distortion. This has the effect of attenuating the signal. Also, with high power amplifiers large currents are flowing through these cables so small thin bell wire type cables can overheat during extended periods of use, again reducing efficiency.

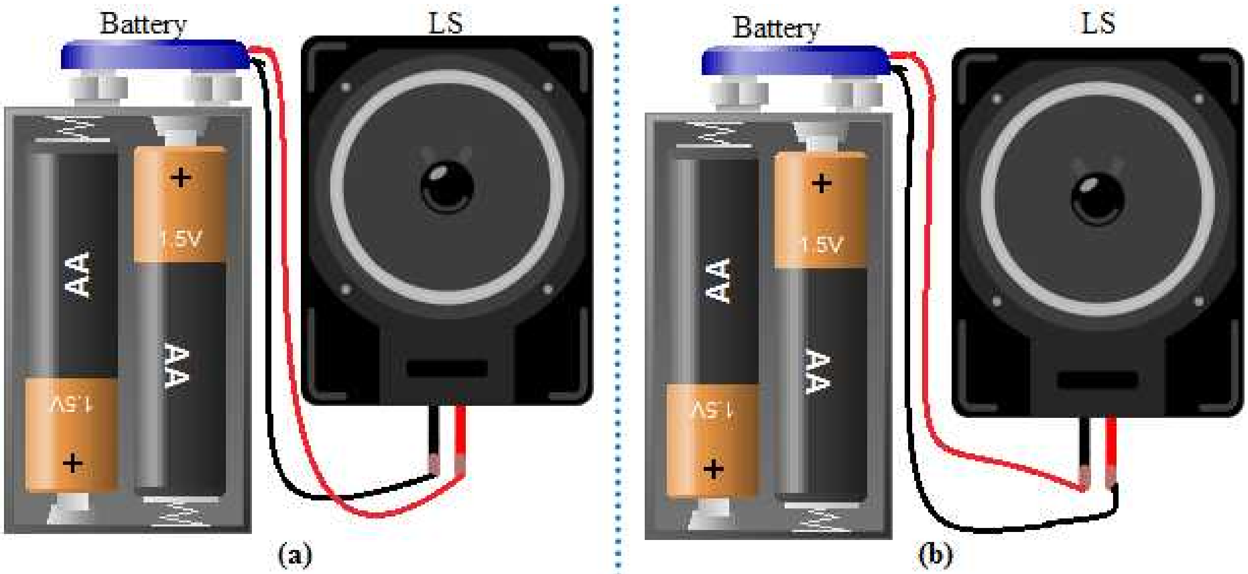


Figure 4-4: Circuit connection to test the loudspeaker cone movement due to the current. (a) The cone moves outward, (b) The cone moves inwards.

The human ear can generally hear sounds from between 20 Hz to 20 KHz, and the frequency response of modern loudspeakers called general purpose speakers are tailored to operate within this frequency range as well as headphones, earphones and other types of commercially available headsets used as sound transducers. However, for high performance High Fidelity (Hi-Fi) type audio systems, the frequency response of the sound is split up into different smaller sub-frequencies thereby improving both the loudspeakers efficiency and overall sound quality as follows:

Generalized Frequency Ranges

Descriptive Unit	Frequency Range
Sub-Woofer	10Hz to 100Hz
Bass	20Hz to 3kHz
Mid-Range	1kHz to 10kHz
Tweeter	3kHz to 30kHz

Table 4-1: Generalized Frequency Ranges

This can be tested by using a function generator connected to the loudspeaker and calibrated to generate different frequencies at fixed amplitude.

Experiment 4-2: Testing and hearing audio frequency range using function generator and loudspeaker

Loudspeakers can be tested using a function generator. As a function generator can generate a small amplitude voltage with variable frequency from 1 Hz up to some mega-hertz, a connected loudspeaker can produce a sound corresponding to the applied frequency from our hearing to ultrasound.

Parts and material

No	Item
1	Moving coil loudspeaker 4W, 8 Ω
2	1V, 1Hz-30KHz supply (Function generator)

Procedure

1. Connect the circuit as shown in the circuit layout figure 4-5.

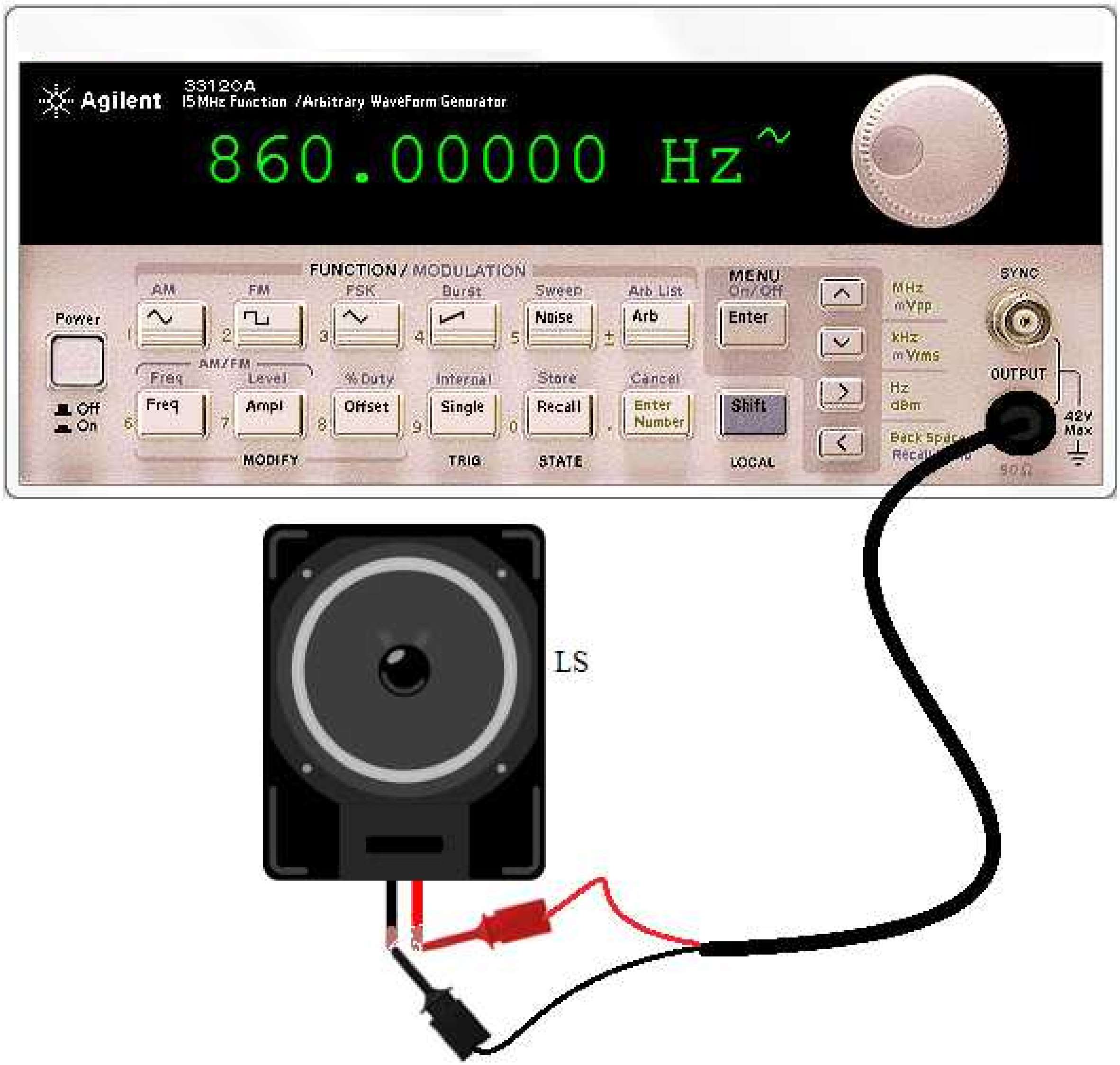
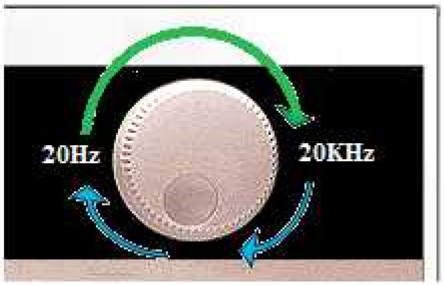


Figure 4-5: Frequency range testing and hearing using function generator and loudspeaker

2. Turn on the function generator and hold the function generator's probes against the loudspeaker's leads.
3. Set the function generator to input about 1Volt and change the frequencies by using the knob as illustrated below. Start at very low frequency of about 1Hz and increase the frequency to about 25 KHz.



4. Note the frequency at which you start to hear a sound and that your hearing fade away. At lowest audio frequency the sound is as bass-sound than that of the high audible frequency.
5. As result, we tend to think of sound as only existing in the range of frequencies detectable by the human ear, from approximately 20Hz up to 20kHz (a typical loudspeaker frequency response), but sound can also extend way beyond these ranges.

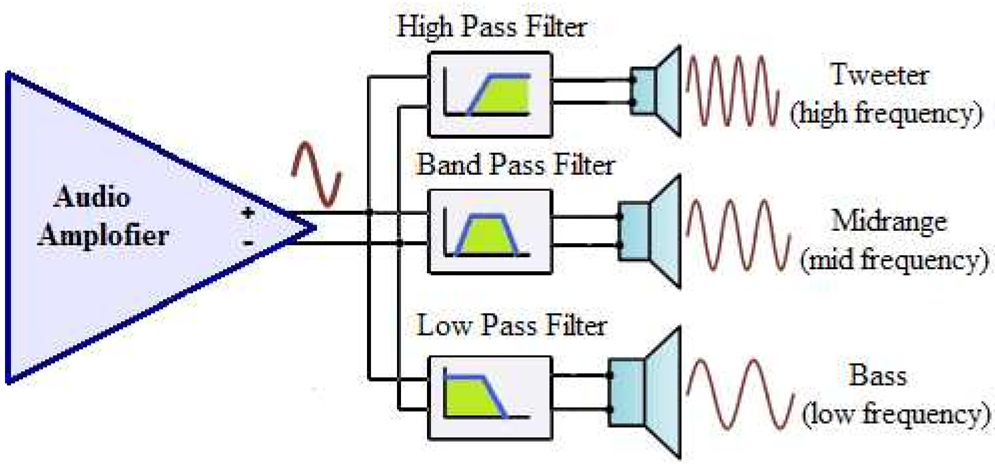


Figure 4-6: Multi-speaker Hi-Fi design

In multi speaker enclosures which have a separate Woofer, Tweeter and Mid-range speakers housed together within a single enclosure, a passive or active "crossover" network is used to ensure that the audio signal is accurately split and reproduced by all the different sub-speakers.

This crossover network consists of Resistors, Inductors, Capacitors, RLC type passive filters or op-amp active filters whose cross-over or cut-off frequency point is finely tuned to that of the individual loudspeakers characteristics and an example of a multi-speaker "Hi-fi" type design is given in figure 4-6.

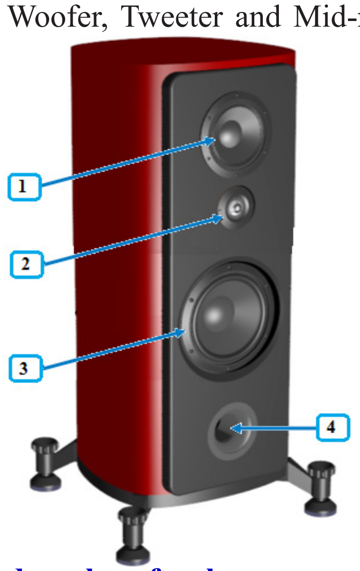


Figure 4-7: Typical loudspeaker for home usage with three types of dynamic drivers; (1) Mid-range driver, (2) Tweeter, (3) Woofers, (4) Bass reflex port

chap 4

Speaker Specifications

With today's Web-savvy, knowledge-thirsty customers comes the challenge of shopping for speakers by specifications and price. This should not be viewed as arduous, but rather as another piece of the conundrum and learning process to make your overall home theatre experience even better. Many squabble that buying a loudspeaker without ever personally auditioning it would be quite thoughtless. However, given the vast number of loudspeakers in today's market understanding specifications and using them to narrow down the field of search is a good idea. The Web can be a valuable resource for such searches, and volumes of information, however beware that not everything you read on the Web is truthful. Stick with companies you know and trust and you will avoid the snares of creative marketing. With this in mind, it is most important to understand the specifications of a loudspeaker, so that you can better weigh where your audition time will be best spent. While many consumers today look first at the "Power Handling" specification of a speaker, this is actually not an important consideration at all. The two most important specifications (and often the least understood) are the Nominal Impedance and Sensitivity Ratings. PSB Speakers takes the Sensitivity Rating a step further by including the Voltage Sensitivity Rating. PSB speakers is a Canadian loudspeaker company founded by Paul Barton.

Sensitivity

Sensitivity is defined as the speakers' ability to effectively convert power into sound. The traditional way of measuring a speakers' sensitivity is using the standard of 1 watt/1 meter. Meaning a microphone is placed 1 meter away from the speaker to measure the sound output (in decibels) with 1 watt of sound played through it. There is much room for error (or imposed error) with this type of measuring system and some manufacturers today take advantage of this.

Voltage Sensitivity

Because today's solid state amplifiers do a good job across the board of maintaining a voltage output of 2.83 volts, many companies consider this as their standard of measurement. Here again, 2.83 volts are inputted and measured at 1 meter.

Note: 2.83 volts into an 8 ohm load is equal to 1 watt.

Ohm's Law: Power (watts) = Voltage (V) x Current (I) or Power = V/R (impedance in Ohms).

Because a speakers' efficiency in transforming (transducer) power into sound is greatly determined by the impedance of a speaker, 2.83 volts becomes greater about 1.5 watts at 6 ohms (as many PSB Speakers are rated.) and 2 watts at 4 ohms — a 3dB increase.

Basically Sensitivity is the speakers' ability to efficiently use the amplifiers energy to create sound. This specification is rated in decibels (dB), but how it's measured is also important to know.

Consider a speaker with a Sensitivity Rating of 87 dB. The standard for achieving this rating is 1 watt/1 meter. This means the speaker will produce sound at the volume of 87 dB, measured by a microphone placed 1 meter away when it's given the input power of 1 watt. (Typically, the input sound is 1 kHz.) A typical PSB loudspeaker has a minimum Sensitivity Rating of 87 dB and this is considered quite efficient. While many of today's mass-marketed loudspeakers range closer to 84 dB, PSB speakers "average" Sensitivity is quite high and some PSB speakers boast a Sensitivity Rating of up to 90 dB which is quite rare and quite good. If you have got a speaker with a Sensitivity Rating of less than 85 dB, you are going to need a fair amount of power to achieve a reasonable sound level.

Impedance

Impedance is the speakers' resistance to power or impeding the flow of power. Generally speaking, the majority of today's loudspeakers are rated at an Impedance of 8 ohms. And this is the standard specification for which many amplifiers are also rated; meaning their power delivered into 8-ohm loads. Many higher-end speakers and amplifiers are rated at 4 to 6 ohms. Understand that some amplifiers cannot drive speakers at less than 8 ohms for any sustained period of time. This will eventually cause the amplifier to fail. PSB speakers all bare a nominal impedance rating of 6 ohms, which on their own make them quite efficient. However, when coupled with their outstanding Sensitivity Ratings, this makes for an incredibly efficient loudspeaker with headroom far greater than most speakers offered today. This is just one of many factors that make PSB Speakers some of the most critically acclaimed, highly sought after brands on the market.

Things to be considered before buying loudspeaker

Apart of specifications related to speaker it is vital to audition speakers in person before you buy them. While there are many factors in which manufacturers account for when designing loudspeakers, it is still more of an art than a science. Several factors will affect a speaker's performance which cannot be accounted for. Consider that nearly every single room, including size, volume, shape, and materials which inhibit or enhance sound, is different. That said, the measurements we discussed cannot account for an anomaly known as "Room Gain." Speakers are tested and measured in anechoic chambers-rooms that are sound proof and have little or no sound reflections whatsoever. No room in any home meets this criterion. This is done (in part) to isolate the speaker during testing.

Direct or reflective sound (reverberation) or sound reflecting off the boundaries of a room is often described as "sound power." Sound power is the sum of all energies combined; including amplifier power, speaker volume, reflective and non reflective sound, and so on. Additionally, this does not take into account the abilities of the listener. Remember, at home, your ears are now the microphone and some are better than others. Expect a sales associate to tell you what to listen for, but not how it should sound to you. When auditioning speakers, it is in your best interest to bring your own source material, if at all possible. After all, this is the music or movie(s) that you are most familiar with and will, therefore, provide the most revealing demonstration. This will also help you to understand the working relationship between speaker and amplifier and better equip you to determine your needs. Take your time and audition as many speakers as you wish until you feel comfortable with your choice. Expect that the speakers will sound better in the comfort of your own home and be sure to make your purchase with a retailer who has a return policy that will allow you to audition them in your own home. Good speakers will last you a couple of decades and should be considered an investment.

4. 2. The Microphone

Microphones are transducers which detect sound signals and produce a voltage or a current which is proportional to the sound signal.

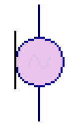


Figure 4-8: Schematic symbol of microphone

Types of microphones

The most common microphones for musical use are dynamic, ribbon, or condenser microphones. Besides the variety of basic mechanisms, microphones can be designed with different directional patterns and different impedances.

4. 2. 1. Dynamic Microphones

Dynamic microphones are versatile and ideal for general-purpose use. They use a simple design with few moving parts. They are comparatively strong and resilient to rough handling. They are also better suited to handling high volume levels, such as from certain musical instruments or amplifiers. They have no internal amplifier and do not require batteries or external power.

Figure 4-9: Typical dynamic microphone

Dynamic Microphones working principle

From Faraday's principle, when a magnet is moved near a coil of wire an electrical current is generated in the wire. Using this electromagnet principle, the dynamic microphone uses a wire coil and magnet to create the audio signal.

The diaphragm is attached to the coil. When the diaphragm vibrates in response to incoming sound waves, the coil moves backwards and forwards past the magnet. This creates a current in the coil which is channelled from the microphone along wires. A common configuration is shown in the figure 4-10.

Similarity between dynamic microphone and loudspeaker

Loudspeakers perform the opposite function of microphones by converting electrical energy into sound waves and that is not just a coincidence. This is demonstrated perfectly in the dynamic microphone which is basically a loudspeaker in reverse. When you see a cross-section of a speaker you will see the similarity with the diagram below, figure 4-10. If fact, some intercom systems use the speaker as a microphone. You can also demonstrate this effect by plugging a microphone into the headphone output of your stereo, although it is not recommend as the high current delivered can damage the microphone.

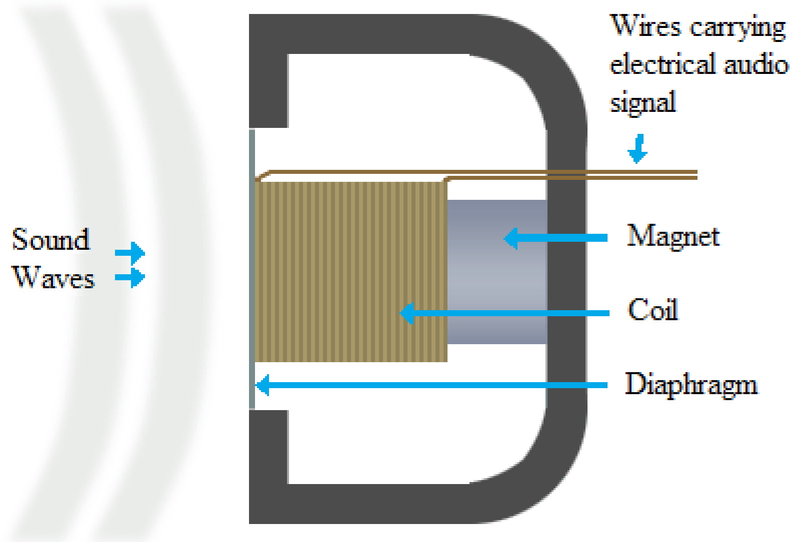


Figure 4-10: Dynamic microphone cross-section

A dynamic microphone is essentially the inverse of a dynamic loudspeaker. In a dynamic microphone, the sound pressure variations move the cone, which moves the attached coil of wire in a magnetic field, which generates a voltage.

In the loudspeaker, the inverse happens; the electric current associated with the electrical image of the sound is driven through the coil in the magnetic field, generating a force on that coil. The coil moves in response to the audio signal, moving the cone and producing sound in the air.

A small loudspeaker can be used as a dynamic microphone, and this fact is exploited in the construction of small intercom systems. Depending upon the position of the Talk-Listen switch, the device on either end of the intercom system can be used as a microphone or a loudspeaker. Of course, this is not a high fidelity process, and for commercial dynamic microphones, the device is optimized for use as a microphone, not a loudspeaker.

Advantages of dynamic microphones

- Relatively cheap and rugged.
- Can be easily miniaturized.
- Do not require batteries or external power.

Disadvantages of dynamic microphones

The uniformity of response to different frequencies does not match that of the ribbon or condenser microphones.

4. 2. 2. Ribbon microphones

A ribbon microphone is based around a thin, corrugated strip of metal (often aluminium) or film suspended between two magnetic poles.

Figure 4-11: Different shapes of typical ribbon microphone

Working principle of ribbon microphone

Unlike traditional moving-coil dynamic microphones, the ribbon element responds to variations in the velocity of air particles, rather than the pressure. The air movement associated with the sound moves the metallic ribbon in the magnetic field, generating an imaging voltage between the ends of the ribbon which is proportional to the velocity of the ribbon. In classic ribbon designs, this level is very low compared to typical dynamic microphones, and a step-up transformer boosts both the output voltage and impedance. Preamp choice is very important when using ribbon microphones.

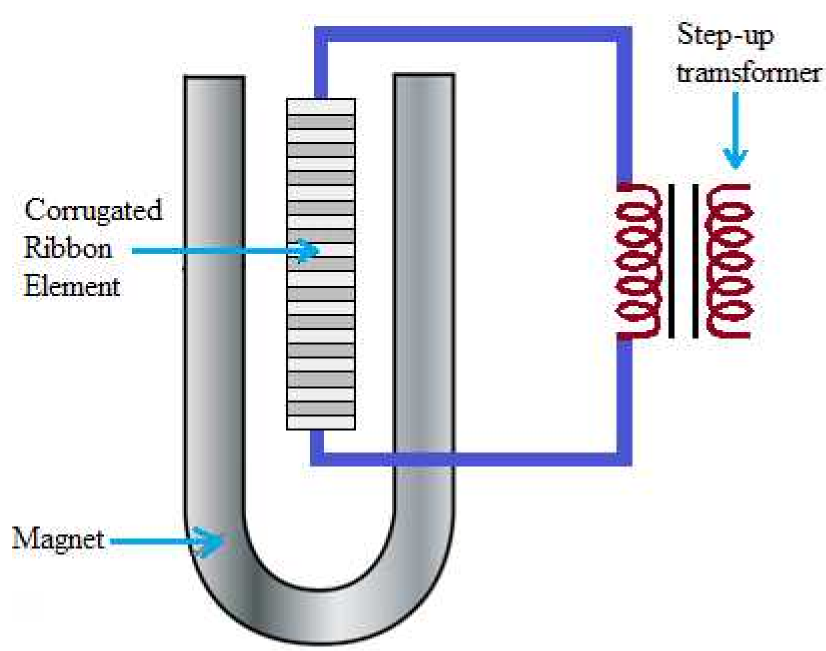


Figure 4-12: Ribbon microphone cross-section

Because a ribbon microphone has an extremely thin, delicate element, it is capable of capturing fast transients. Ribbons microphones have a wide dynamic range, and are capable of handling high SPLs (sound pressure level) at high frequencies. (Give them a try on brass or percussion.) These microphones are bidirectional by design, because the ribbon element responds to sound arriving from the front or back of the microphone, and does not pick up sound arriving on its sides. This natural figure pattern makes them ideal for stereo recording applications, and is useful in applications where you want to eliminate unwanted noise between two sources (i.e. in broadcast).

Ribbon microphones are very sensitive, but they are often quite fragile; delicate older models can be broken by strong gusts of air, voltage spikes or even by being stored on their side.

Advantages of ribbon microphones

- Adds "warmth" to the tone by accenting lows when close-miked.
- Can be used to discriminate against distant low frequency noise in its most common gradient form.

Disadvantages of ribbon microphones

- Accenting lows sometimes produces "boomy" bass.
- Very susceptible to wind noise. Not suitable for outside use unless very well shielded.

4. 2. 3. Condenser Microphone

Condenser refers to an electronic component which stores energy in the form of an electrostatic field; that is "capacitor". The term condenser is actually obsolete but has stuck as the name for this type of microphone, which uses a capacitor to convert acoustical energy into electrical energy.

Condenser microphones require power from a battery or external source. The resulting audio signal is stronger signal than that from a dynamic. Condensers also tend to be more sensitive and responsive than dynamics, making them well-suited to capturing subtle nuances in a sound. They are not ideal for high-volume work, as their sensitivity makes them prone to distort.

Figure 4-13: Different shapes of typical condenser microphone

Working principle of condenser microphones

Sound pressure changes the spacing between a thin metallic membrane and the stationary back plate.

Figure 4-14: Studio condenser microphone recording with a vocal pop filter

The plates are charged to a total charge. A capacitor has two plates with a voltage between them. In the condenser microphone, one of these plates is made of very light material and acts as the diaphragm. The diaphragm vibrates when struck by sound waves, changing the distance between the two plates and therefore changing the capacitance. A change in plate spacing will cause a change in charge Q and force a current through resistance R. This current "images" the sound pressure, making this a "pressure" microphone. Specifically, when the plates are closer together, capacitance increases and a charge current occurs. When the plates are further apart, capacitance decreases and a discharge current occurs.

$$Q = CV = \varepsilon AV/d$$

Where

C: The capacitance,

V: The voltage of the biasing battery,

A: The area of each plate and

D: The separation of the plates.

chap 4

A voltage is required across the capacitor for this to work. This voltage is supplied either by a battery in the microphone or by external phantom power.

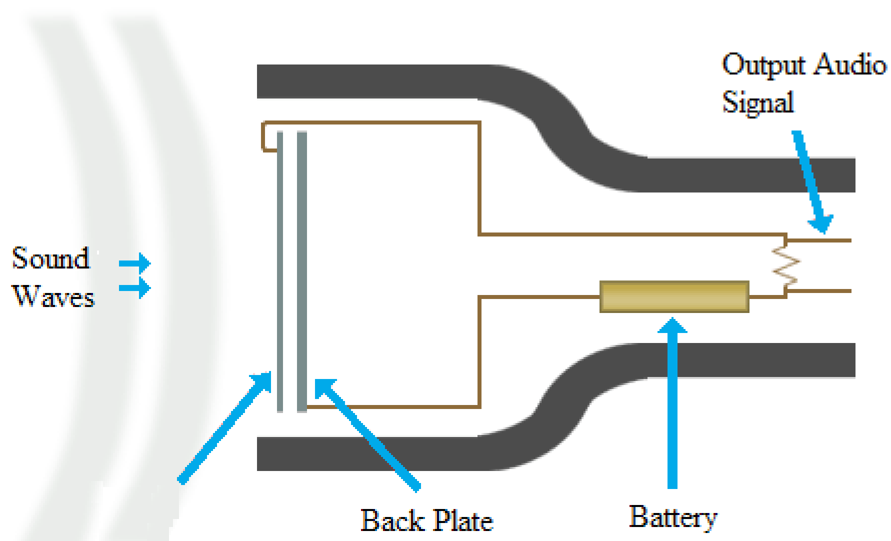


Figure 4-15: Condenser microphone cross-section

The electret condenser microphone

The electret condenser microphone uses a special type of capacitor which has a permanent voltage built in during manufacture. This is somewhat like a permanent magnet, in that it does not require any external power for operation.

Electret microphones, a type of condenser microphone, replace the externally-applied charge with a permanently fixed-charged material. The electret is a stable dielectric material with a permanently-embedded static charge. Electret microphones require no polarizing voltage but, due to their integrated pre-amplifier, require a small amount of power either from microphone inputs as phantom power or from a small battery.

Figure 4-16: Typical electret microphone

Due to mass-production of these devices, electret microphones lack the precision needed for high-quality microphones.

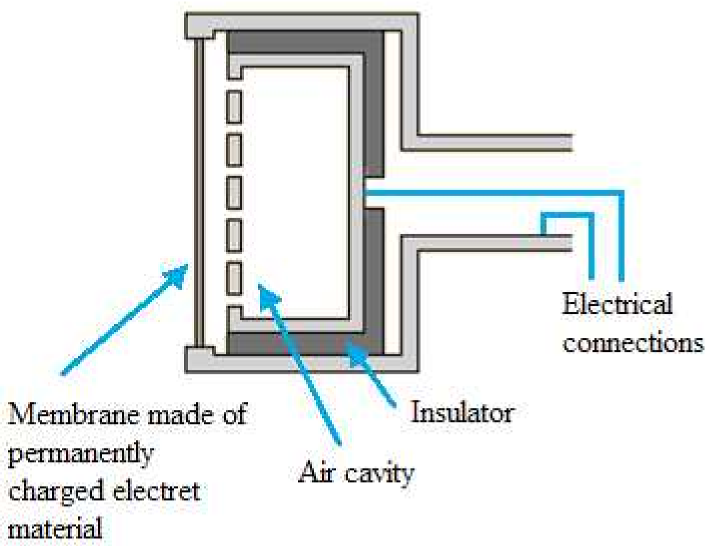


Figure 4-17: Electret microphone cross-section

However good electret condenser microphone usually includes a pre-amplifier which does still require power. Other than this difference, you can think of an electret condenser microphone as being the same as a normal condenser.

Things to be considered while choosing microphone

The following considerations have to be taken when deciding which microphone to use for different applications as each type of microphone has its pros and cons.

Dynamic	Condenser	Ribbon
• Physically durable	• Typically less durable than dynamics	• Generally more fragile than other microphone types, though newer designs are more durable
• Withstanding high sound pressure levels without distorting	• Handle moderate sound pressure levels and may require a "pad" (which reduce the level) to cleanly deliver high-volume signals	• Don't have high sound pressure levels well (at high levels, the ribbon may stretch or even break)
• Moderately wide range frequency response	• Offer an excellent, wide frequency response	• Don't handle bursts of air well -such as from a voice or a kick drum (this may also stretch or break the thin ribbon)
• Generally inexpensive compared to other microphone types of similar quality	• Range in price from inexpensive to super expensive	• Limited frequency response
• Punchy, warm sound	• Usually require an external power (called "phantom"), though some can use batteries.	• Generally expensive , though inexpensive models have appeared recently
	• Use on board tube or solid state electronics	• Sensitive to preamp coloration (though some modern models have electronics that eliminate this)
	• Generally have Hi-Fi sound quality with excellent clarity	• Renowned for warm, natural sound quality

Table 4-2: Comparison between dynamic, condenser and ribbon microphone

Experiment 4-3: Testing microphone's voice to signal transducer property

Parts list

No	Item
1	Microphone
2	Oscilloscope

Procedure

1. Switch on the oscilloscope and connect the probe to one of the channel, i.e CH1
2. Connect the oscilloscope's probe hook end to the terminal of the microphone. Keep the microphone near to the sound or music source

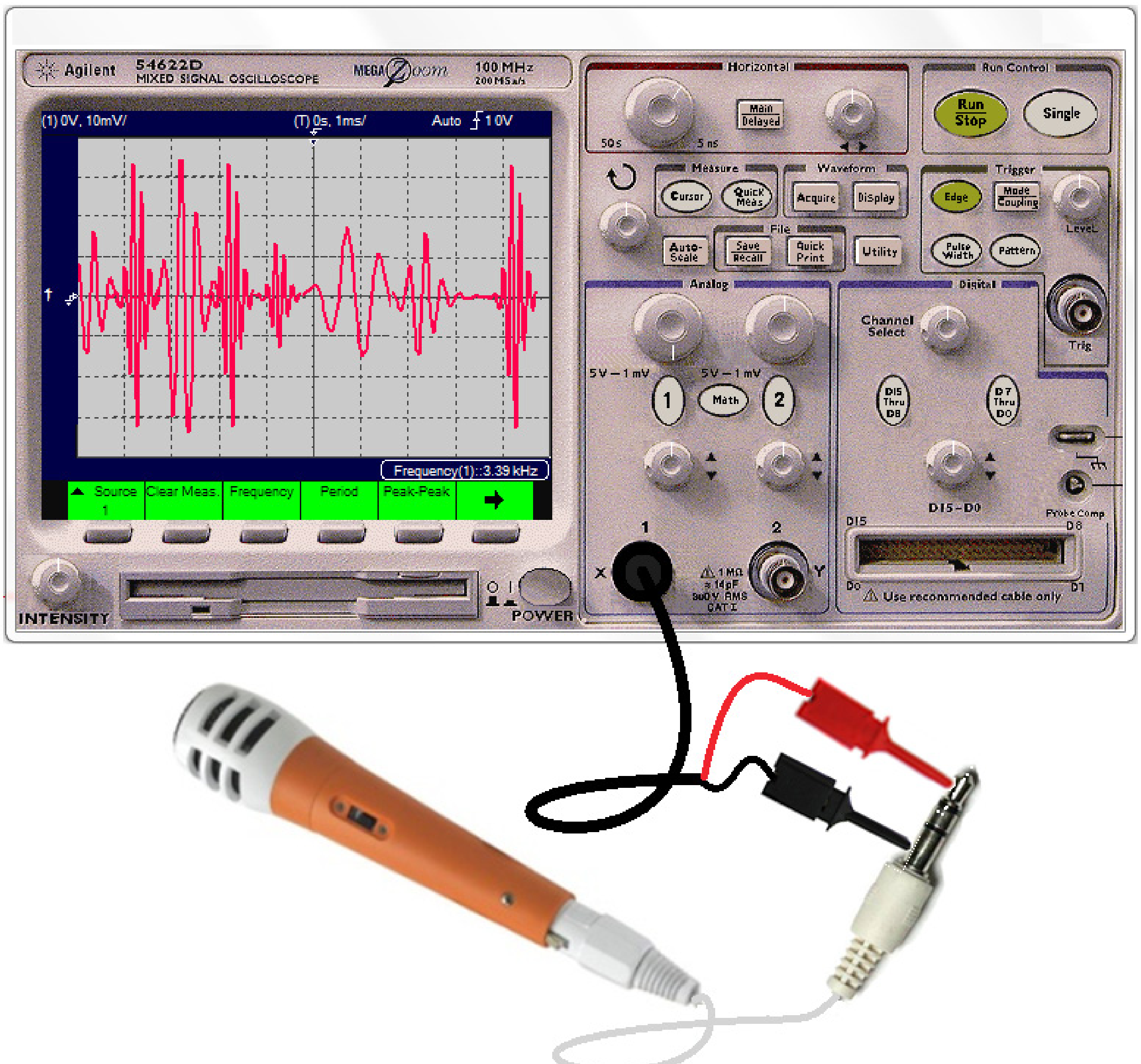


Figure 4-18: Testing microphone's voice to signal transducer property

3. Observe the proportionality between the sound loudness and the signal change. As a result, the higher the loudness, the higher the amplitude (voltage). When you are using digital oscilloscope, at instant of displaying a signal, you can press on run-stop button "  " for easy measuring the voltage and frequency as this change in function of sound.

4. 3. Transformer

A basic transformer is an electrical device constructed of two coils of wire (windings) electromagnetically coupled to each other so that there is a mutual inductance for the transfer of power from one winding to the other. The schematic of a transformer is shown in figure 4-19, part (a). One coil is called the primary winding and the other coil is called the secondary winding. For standard operation, the source voltage is applied to the primary winding, and a load is connected to the secondary winding, as shown in figure 4-19, part (b). The primary winding is the input of the transformer, and the secondary winding is the output winding. It is common to refer to the side of the transformer that has the source voltage as the *primary*, and the side that has the induced voltage as the *secondary.*

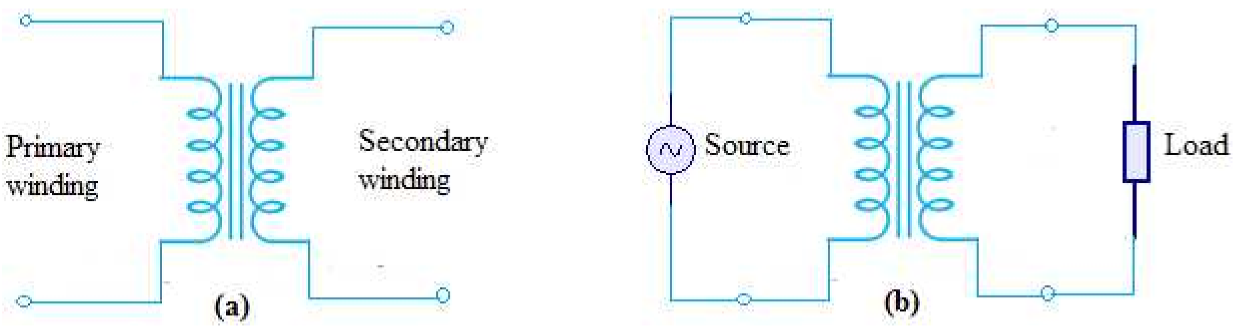


Figure 4-19: Basic transformer, (a) Schematic symbol, (b) source/load connection

The transformer does this by linking together two or more electrical circuits using a common oscillating magnetic circuit which is produced by the transformer itself. A transformer operates on the principals of "electromagnetic induction", in the form of Mutual Induction.

Mutual induction is the process by which a coil of wire magnetically induces a voltage into another coil located in close proximity to it. Then we can say that transformers work in the "magnetic domain", and transformers get their name from the fact that they "transform" one voltage or current level into another.

Transformer working principle

A transformer basically consists of two coils wound around a common soft iron core. When an alternating voltage is applied to the primary coil, current flows through the coil which in turn sets up a magnetic field around itself, called *mutual inductance*, by this current flow according to *Faraday's Law* of electromagnetic induction. The strength of the magnetic field builds up as the current flow rises from zero to its maximum value which is given as $d\Phi/dt$. As the magnetic lines of force setup by this electromagnet expand outward from the coil the soft iron core forms a path for and concentrates the magnetic flux. This magnetic flux links the turns of both windings as it increases and decreases in opposite directions under the influence of the AC supply.

However, the strength of the magnetic field induced into the soft iron core depends upon the amount of current and the number of turns in the winding. When current is reduced, the magnetic field strength reduces. When the magnetic lines of flux flow around the core, they pass through the turns of the secondary winding, causing a voltage to be induced into the secondary coil. The amount of voltage induced will be determined by:

$$E=N.d\Phi/dt \text{ (Faraday's Law)},$$

Where

N: Number of coil turns.

E: Induced voltage

Φ: Flux

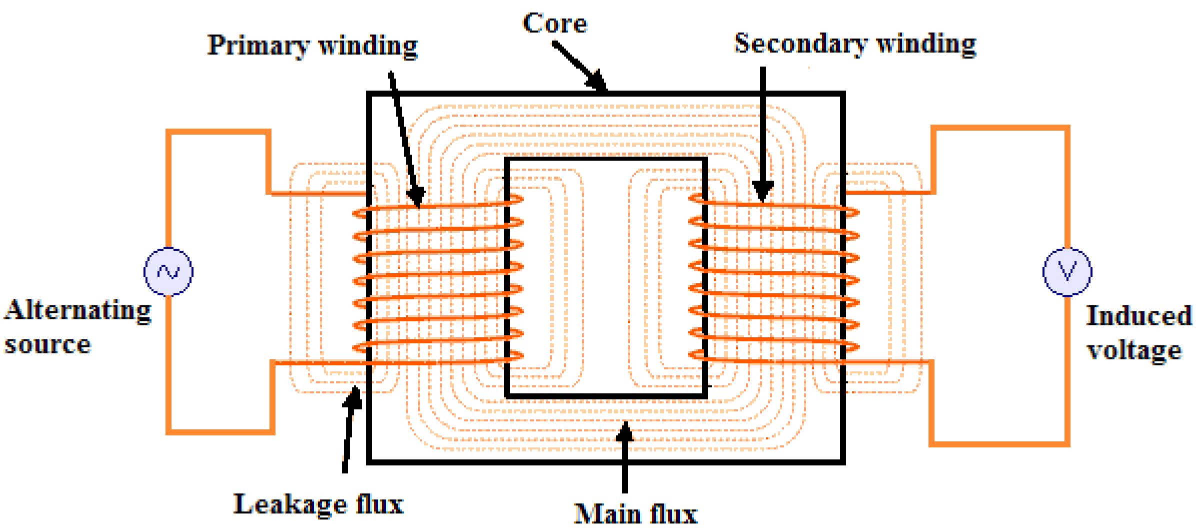


Figure 4-20: Transformer structure

Also this induced voltage has the same frequency as the primary winding voltage. Then the same voltage is induced in each coil turn of both windings because the same magnetic flux links the turns of both the windings together. As a result, the total induced voltage in each winding is directly proportional to the number of turns in that winding. However, the peak amplitude of the output voltage available on the secondary winding will be reduced if the magnetic losses of the core are high.

Types of transformers

The transformers are classified based on voltage levels, Core medium used, winding arrangements used and installation place etc.. In this tutorial, we are going to describe the most transformers used for home appliances and in electronics circuits as well.

Voltage transformer

The Voltage Transformer can be thought of as an electrical component rather than an electronic component. Transformers are capable of either increasing or decreasing the voltage and current levels of their supply, without modifying its frequency, or the amount of Electrical Power being transferred from one winding to another via the magnetic circuit. I.e. if the voltage is high at secondary side then the current drawn from the secondary will low so that the power will be same. Same as in the reverse case when the voltage is low the current drawn will be high. Thus, transformers are classified as step-up and step-down transformers as the voltage ratios from primary to secondary. These are widely used transformer types for all the applications.

One of the main reasons that we use alternating AC voltages and currents in our homes and workplace's is that it can be easily generated at a convenient voltage, transformed into a much higher voltage and then distributed around the country using a national grid of pylons and cables over very long distances. The reason for transforming the voltage is that higher distribution voltages imply lower currents and therefore lower losses along the network.

These high AC transmission voltages and currents are then reduced to a much lower, safer and usable voltage level where it can be used to supply electrical equipment in our homes and workplaces, and all this is possible thanks to the basic Voltage Transformer.

Step-up transformer

As the name specifies the secondary voltage is stepped up with a ratio compared to primary voltage. This is achieved by increasing the number of coil turns in the secondary as shown in figure 4-21.

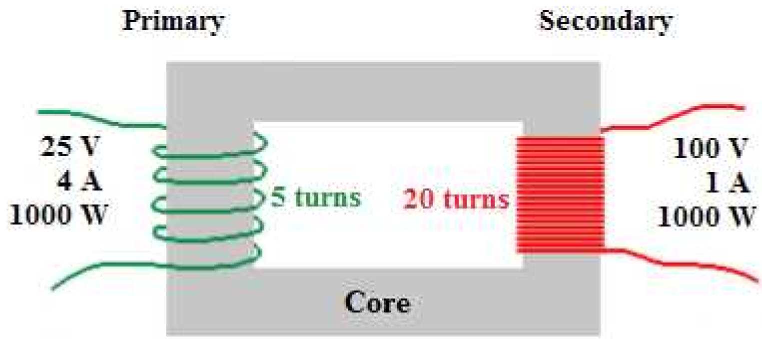


Figure 4-21: Step-up transformer

Generally step-up transformers are used in power plant as connecting transformer of generator to grid. I.e. the Generated low voltage should be suitably stepped up to connect to high voltage grid.

Step-down transformer

The high voltage applied to the primary winding is stepped down to the secondary winding of the transformer so that the secondary voltage is less than the primary where the name step-down transformer. The winding turns will be high at primary side whereas it will be less at secondary side.

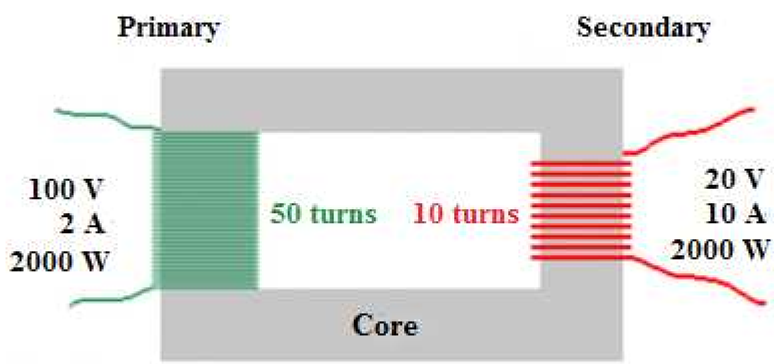


Figure 4-22: Step-down transformer

The step down transformer is widely used to convert the high grid voltage to the low voltage which can be used at house purposes such as radio receivers, television, etc. In power plant the use of this transformer are very high where the grid power supply stepped down and given to corresponding plant auxiliaries during starting of the power plant. Once the plant has started then the voltage stepping down is necessary where the plant auxiliaries will operate at low voltage compared to its generated voltage.

Transformer based on the core medium used

The transformers are divided as air core and iron core under this classification. I.e. the medium placed between the primary and secondary air in air core type transformer and iron in iron core type transformer. The core provides both a physical structure for placement of the windings and a magnetic path so that the magnetic flux is concentrated close to the coils. Three general categories of core material are air, ferrite and iron. The schematic symbol for each type is shown in figure 4-23.

Air core transformer

The primary and secondary windings wounded on non magnetic strip where the flux linkage between primary and secondary is through the air. The mutual inductance effect is less in air core compared to iron core i.e. the reluctance offered to the generated flux is high in the air medium whereas in the iron core it is less. But the hysteresis and eddy current losses which are dominant in iron core type are less or completely eliminated in air core type transformer.

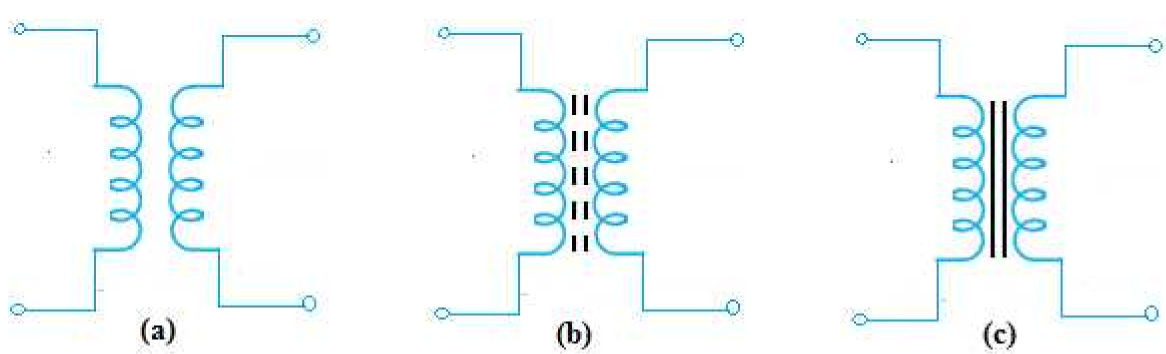


Figure 4-23: Transformer based on core medium; (a) Air core transformer, (b) ferrite core transformer, (c) Iron core transformer

Air-core and ferrite-core transformers generally are used for high-frequency applications and consist of windings on an insulating shell that is hollow (air) or constructed of ferrite. In figure 4-23(a), the windings are loosely coupled because they are separated, and in part (b) they are tightly coupled because they are overlapping. The tighter the coupling, the greater the induced voltage in the secondary for a given current in the primary.

Iron core transformer

The two windings are wounded on iron plates which provide a perfect linkage path to the generated flux. Due to the conductive or magnetic property of the iron it offers less reluctance to the linkage flux. These are widely used transformers in which is efficiency is high compared to air core type transformer. Iron-core transformers generally are used for audio-frequency (AF) and power applications.

Transformer Construction of the Core

Generally, the name associated with the construction of a transformer is dependent upon how the primary and secondary windings are wound around the central laminated steel core. The two most common and basic designs of transformer construction are the Closed-core Transformer and the Shell-core Transformer.

Core type

In the “closed-core” type (core form) transformer, the primary and secondary windings are wound outside and surround the core ring.

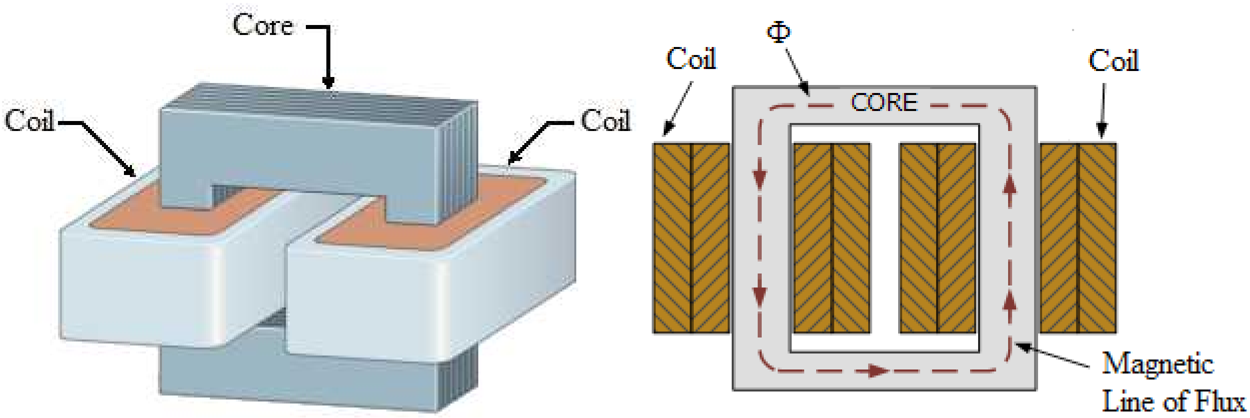


Figure 4-24: Figure: Core type transformer has each winding on a separate leg

chap 4

Shell type

In the “shell type” (shell form) transformer, the primary and secondary windings pass inside the steel magnetic circuit (core) which forms a shell around the windings.

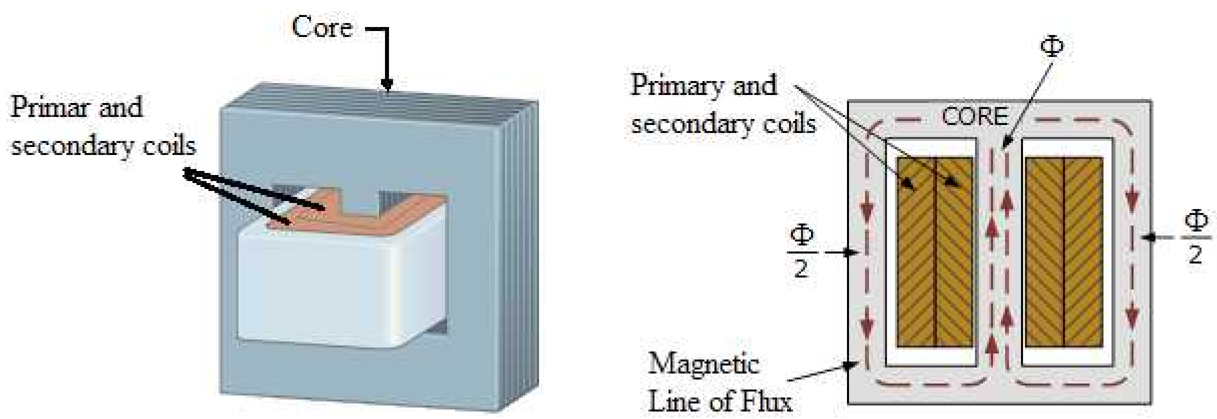


Figure 4-25: Shell type transformer has both windings on the same leg

The construction of a simple two-winding transformer consists of each winding being wound on a separate limb or core of the soft iron form which provides the necessary magnetic circuit. This magnetic circuit, know more commonly as the “transformer core” is designed to provide a path for the magnetic field to flow around, which is necessary for induction of the voltage between the two windings.

The coils are firstly wound on a former which has a cylindrical, rectangular or oval type cross section to suit the construction of the laminated core. In both the shell and core type transformer constructions, in order to mount the coil windings, the individual laminations are stamped or punched out from larger steel sheets and formed into strips of thin steel resembling the letters “E’s”, “L’s”, “U’s” and “I’s”

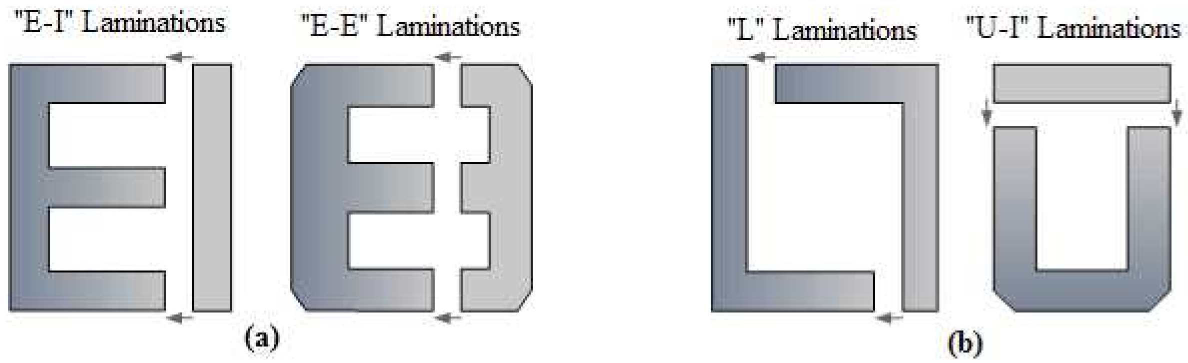


Figure 4-26: Transformer Laminations, (a) Shell-type Laminations, (b) Core-type Laminations

In all types of transformer construction, the central iron core is constructed from of a highly permeable material made from thin silicon steel laminations assembled together to provide the required magnetic path with the minimum of losses. The resistivity of the steel sheet itself is high reducing the eddy current losses by making the laminations very thin.

Transformer based on winding arrangements

Auto Transformer

Unlike the previous voltage transformer which has two electrically isolated windings called, i.e. primary and secondary, but in auto transformer, both the windings that is primary and secondary windings are connected to each other both physically and magnetically to form only one single voltage winding which is common to both sides. This single winding is “tapped” at various points along its length to provide a percentage of the primary voltage supply across its secondary load. Then the autotransformer has the usual magnetic core but only has one winding, which is common to both the primary and secondary circuits. The basic autotransformer construction is shown in figure 4-27.

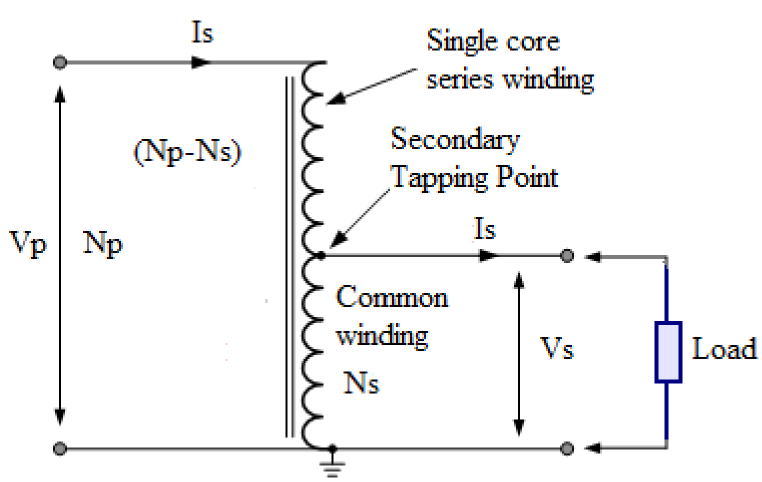


Figure 4-27: Autotransformer basic schematic

Where

V_P: is the Primary Voltage
V_S: is the Secondary Voltage
N_P: is the Number of Primary Windings
N_S: is the Number of Secondary Windings

The Autotransformer can also be constructed with more than one single tapping point. Autotransformers can be used to provide different voltage points along its winding or increase its supply voltage with respect to its supply voltage. The basic autotransformer with multiple tapping points is shown in figure 4-28.

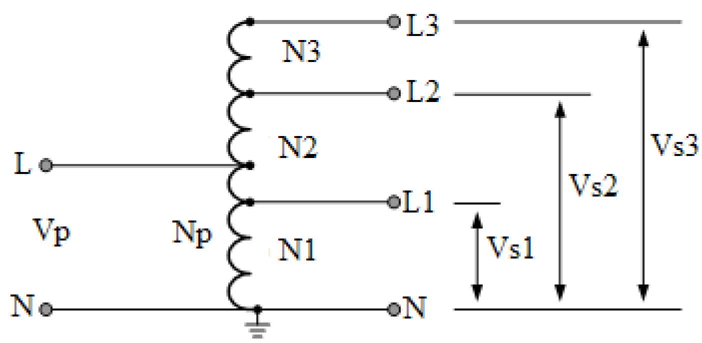


Figure 4-28: Autotransformer with multiple tapping points

Therefore in an autotransformer the primary and secondary windings are linked together both electrically and magnetically. The main advantage of this type of transformer design is that it can be made a lot cheaper for the same VA rating, but the biggest disadvantage of an autotransformer is that it does not have the primary/secondary winding isolation of a conventional double wound transformer.

The section of winding designated as the primary part of the winding is connected to the AC power source with the secondary being part of this primary winding. An autotransformer can also be used to step the supply voltage up or down by reversing the connections. If the primary is the total winding and is connected to a supply, and the secondary circuit is connected across only a portion of the winding, then the secondary voltage is "stepped-down".

4. 3. 1. Transformer Dot Orientation

The two coil windings can have a distinct orientation of one with respect to the other. Either coil could be wound around the core clockwise or anticlockwise so to keep track of their relative orientations "dots" are used to identify a given end of each winding.

This method of identifying the orientation or direction of a transformer's windings is called the "dot convention". Then a transformer's windings are wound so that the correct phase relations exist between the winding voltages with the transformers polarity being defined as the relative polarity of the secondary voltage with respect to the primary voltage.

The first transformer shows its two "dots" side by side on the two windings. The current leaving the secondary dot is "in-phase" with the current entering the primary side dot. Thus the polarities of the voltages at the dotted ends are also in-phase so when the voltage is positive at the dotted end of the primary coil, the voltage across the secondary coil is also positive at the dotted end.

The second transformer shows the two dots at opposite ends of the windings which means that the transformers primary and secondary coil windings are wound in opposite directions. The result of this is that the current leaving the secondary dot is 180^{o} "out-of-phase" with the current entering the primary dot. So the polarities of the voltages at the dotted ends are also out-of-phase so when the voltage is positive at the dotted end of the primary coil, the voltage across the corresponding secondary coil will be negative.

Then the construction of a transformer can be such that the secondary voltage may be either "in-phase" or "out-of-phase" with respect to the primary voltage. In transformers which have a number of different secondary windings, each of which is electrically isolated from each other it is important to know the dot polarity of the secondary windings so that they can be connected together in series-aiding (secondary voltage is summed) or series-opposing (the secondary voltage is the difference) configurations.

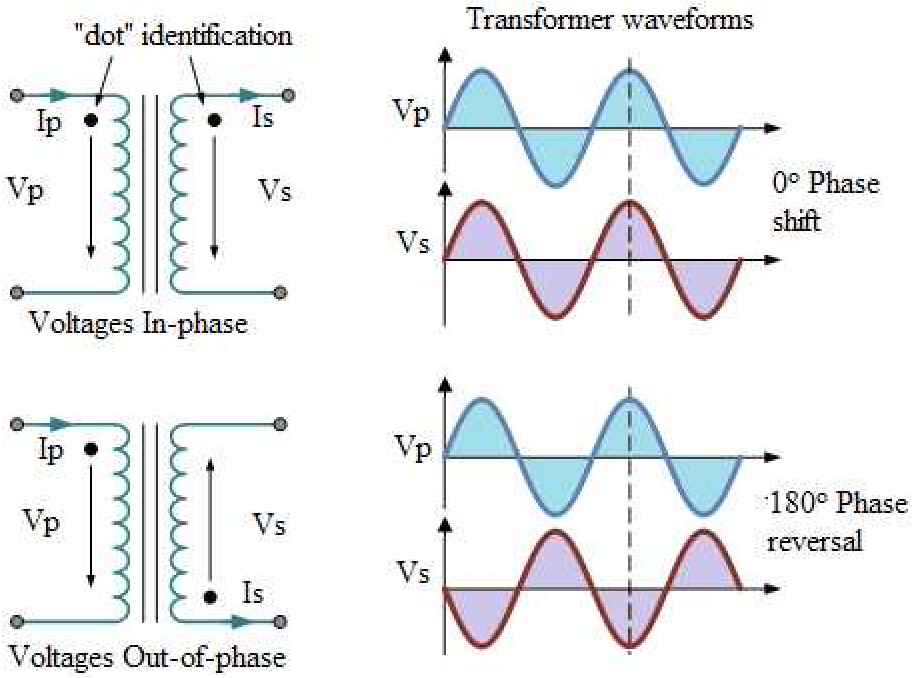


Figure 4-29: Transformer dot orientation

Transformer ratio

The turns ratio of a transformer states that the total induced voltage in each winding is proportional to the number of turns in that winding and also that the power output and power input of a transformer is equal to the volts times amperes, (V x *I*). Therefore

$$\text{Power}_{(\text{Primary})} = \text{Power}_{(\text{secondary})}$$

$$V_p * I_p = V_s * I_s$$

$$\text{Then } V_p * I_p / V_s = I_s$$

$$V_p / V_s = I_s / I_p$$

But the voltage ratio of a transformer is equal to the turn ratio of a transformer.

"voltage ratio = turns ratio" Then the relationship between the voltage, current and number of turns in a transformer can be linked together and is therefore given as:

$$n = N_p / N_s = V_p / V_s = I_s / I_p$$

Where:

$N_P/N_S = V_P/V_S$ - represents the voltage ratio

$N_P/N_S = I_S/I_P$ - represents the current ratio

Note that the current is inversely proportional to both the voltage and the number of turns. This means that with a transformer loading on the secondary winding, in order to maintain a balanced power level across the transformers windings, if the voltage is stepped up, the current must be stepped down and vice versa. In other words, “higher voltage — lower current” or “lower voltage — higher current”.

Series and parallel connected secondary transformer

Voltage transformers can be connected to operate from power supplies of different voltage levels. For example, let’s say that the primary winding could have a voltage rating of 220/110V on the primary and 12/24V on the secondary. To achieve this, each of the two primary windings is, therefore, rated at 110V, and each secondary winding is rated at 12V. The transformer must be connected so that each primary winding receives the proper voltage. Consider the circuits of the figure 4-30 and figure 4-31.

Series connected secondary transformer

In this example, figure 4-30, the two 110V rated primary windings are connected together in series across a 220V supply as the two windings are identical, half the supply voltage, namely 110V, is dropped across each winding and the same primary current flows through both. The two secondary windings rated at 12V, 2A each are connected in series with the secondary terminal voltage being the sum of the two individual winding voltages giving 24 Volts.

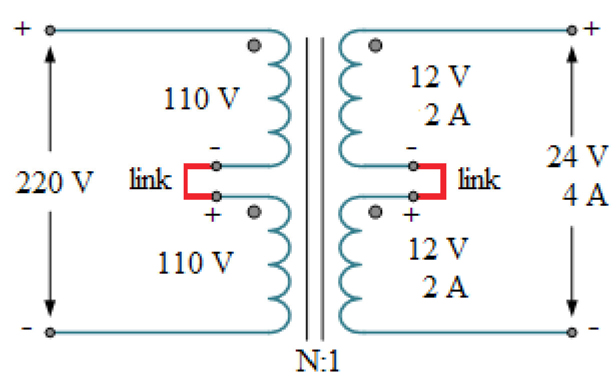


Figure 4-30: Series connected secondary transformer

As the two windings are connected in series, the same amount of current flows through each winding, then the secondary current is the same at 2 Amps. So for a series connected secondary, the output in the above example, figure 4-30, is rated at 24 Volts, 2 Amps. Consider the parallel connected transformer below, in figure 4-31.

Parallel connected secondary transformer

Here we have kept the two primary windings the same but the two secondary windings are now connected in a parallel combination. As before, the two secondary windings are rated at 12V, 2A each, therefore the secondary terminal voltage will be the same at 12 Volts but the current adds.

Then for a parallel connected secondary, the output in our example above is rated at 12 Volts, 4 Amps.

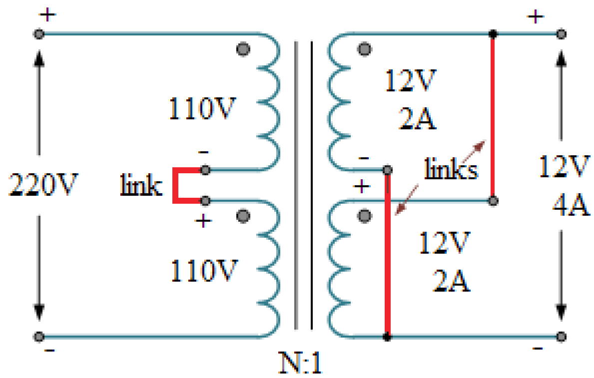


Figure 4-31: Parallel connected secondary transformer

Transformer testing

Transformers are key electrical components in power supplies and electronic circuits. Though they are reliable and have no moving parts, sometimes transformers fail, rendering the equipment it powers inoperable where the need to be tested. Different methods can be used to test home appliances' transformer functionality. Some simple and reliable methods are described in the experiment 4-4 through experiment 4-6.

Experiment 4-4: Test a transformer by using a function generator (ac source)

chap 4

If you suspect a miss function of a transformer in a part of equipment, you can carry out a simple test with a function generator that checks the transformer's basic functions. The function generator's ac signal tests the integrity of the primary and secondary windings; broken wires or shorts inside the transformer will produce no output. The method is safe as it avoids the high voltages usually present inside electronic equipment.

Parts list

No	Item	Specification
1	5V/50Hz ac supply	Function generator
2	Voltmeter	Digital multimeter

Procedure

1. Disconnect the equipment containing the transformer from the wall outlet, if the transformer is in a circuit.
2. Turn the function generator on. Set its frequency to 50 Hz.
3. Touch the digital multimeter's probe tips to the function generator's output. Set the multimeter to read AC volts.
4. Set the function generator's signal amplitude to 5 volts by monitoring the multimeter's display as you adjust the function generator's amplitude control. Disconnect the multimeter from the function generator.
5. Connect the function generator's output to the transformer's two primary windings wires.

6. Touch the multimeter's probe tips to the transformer's two secondary wires. A good isolation transformer reads about 5 volts AC on the meter.
7. A good step-down transformer produces from 100 millivolts to 1 volt AC, depending on its rated output voltage. If the output reads only a few millivolts or less on the multimeter, the transformer is faulty.

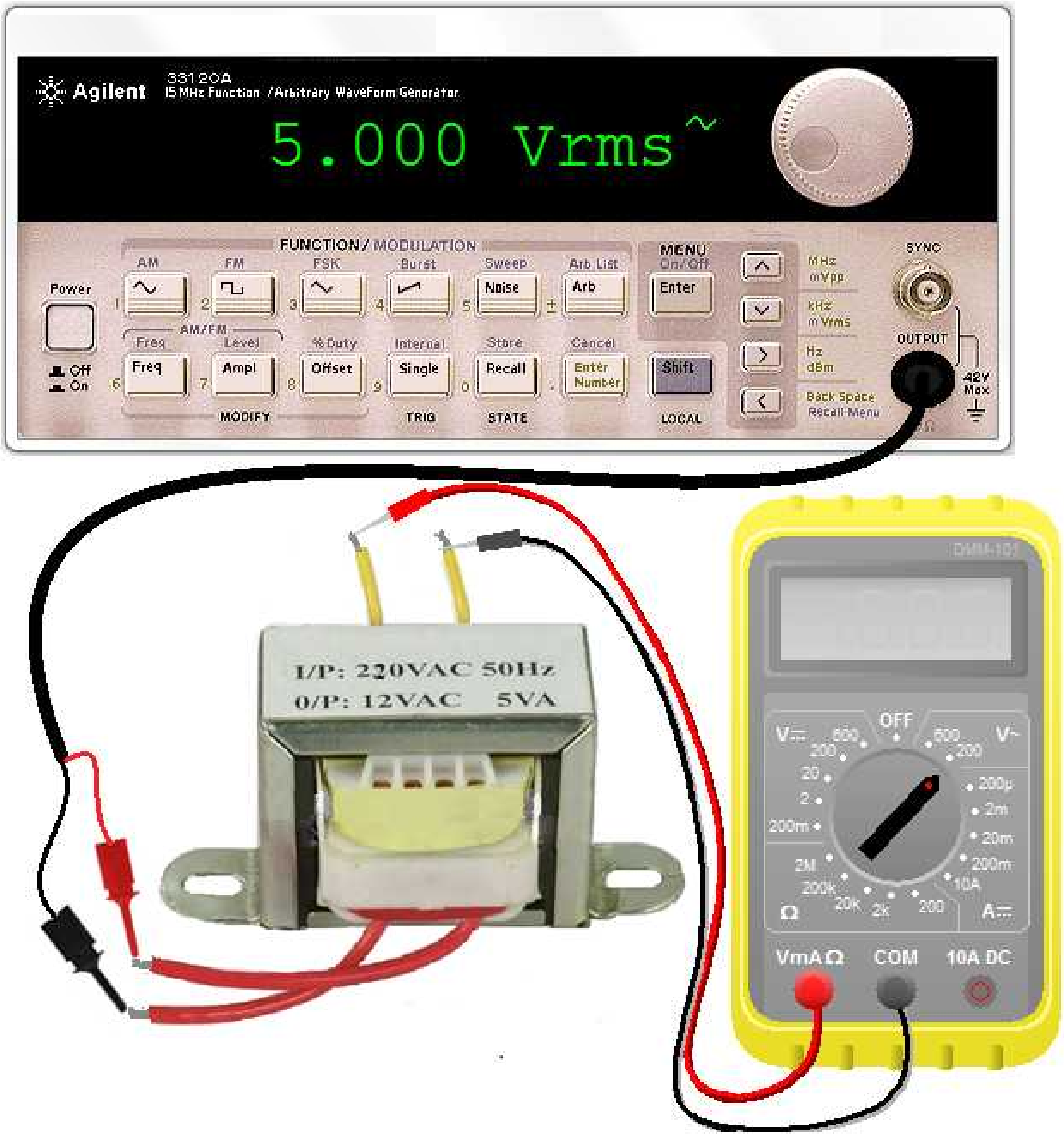


Figure 4-32: Testing transformer with a function generator

Experiment 4-5: Test transformer from wall outlet socket

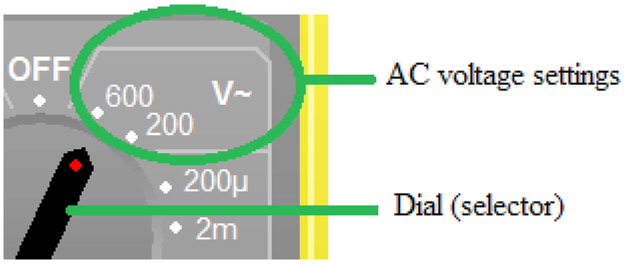
Checking a transformer for its proper operation can be conducted by using voltmeter to test the voltage presence. The transformer can be divided into two parts. The supply voltage or primary which is usually high voltage for step-down transformer, and the secondary or low voltage. What we need to determine is that we are getting correct supply voltage according to the transformer specifications and the supply voltage from network.

Parts list

No	Item	Specification
1	Voltmeter	Digital multimeter
2	Transformer	220V/12V, 50Hz, 1A Transformer

Procedure

1. Set the multimeter to read AC volts at a scale equivalent to the highest voltage you are expecting to measure. The next scale higher than 220 volts AC on a lot of meters is 600. Most digital multimeters have a dial to turn for the different settings and scales then, use this dial to set to the desired range except if you are using auto range multimeter. Make sure that the red lead of multimeter is connected on VΩ jack and black lead to the COM jack of multimeter.



2. Test incoming voltage from wall outlet socket. If no voltage is present in either case you have a bad breaker or voltage has become interrupted somewhere along the way and needs to be repaired. If voltage is present you should read about 220 volts ac on your meter. Then you have proper supply voltage and now you can test your transformer.
3. Locate the transformer wires. You should be able to read transformer main plate information in order to know which wires are for primary and that of the secondary windings otherwise you can damage you transformer. Some transformer have color-code on their lead which can help to indentify the wires but let us refer on a transformer where the color code has faded to the point as to be unreadable. The supply voltage wires coming from the transformer can be many colors; the most common for a 220 volt ac transformer supply voltage is black and white, the black being the live wire and the white being the neutral common wire or, black and red with red as live wire. If the indication on main plate is +, 0, -, (220V/6-0-6 for example) this is a centre tapped transformer where the 3 wires which have continuity are for secondary windings and the remaining two wires having the continuity are for primary winding. If the indication on the main plate of a transformer is not readable refer to the winding section. The winding with thin (small) section is for primary supplying voltage 220V while the winding with big section is for secondary, 12VAC for this experiment.

   Transformer windings have a phase relationship, but it is typically not important for power supplies. Unless one of the wires has continuity to the transformer core, the polarity doesn't matter.

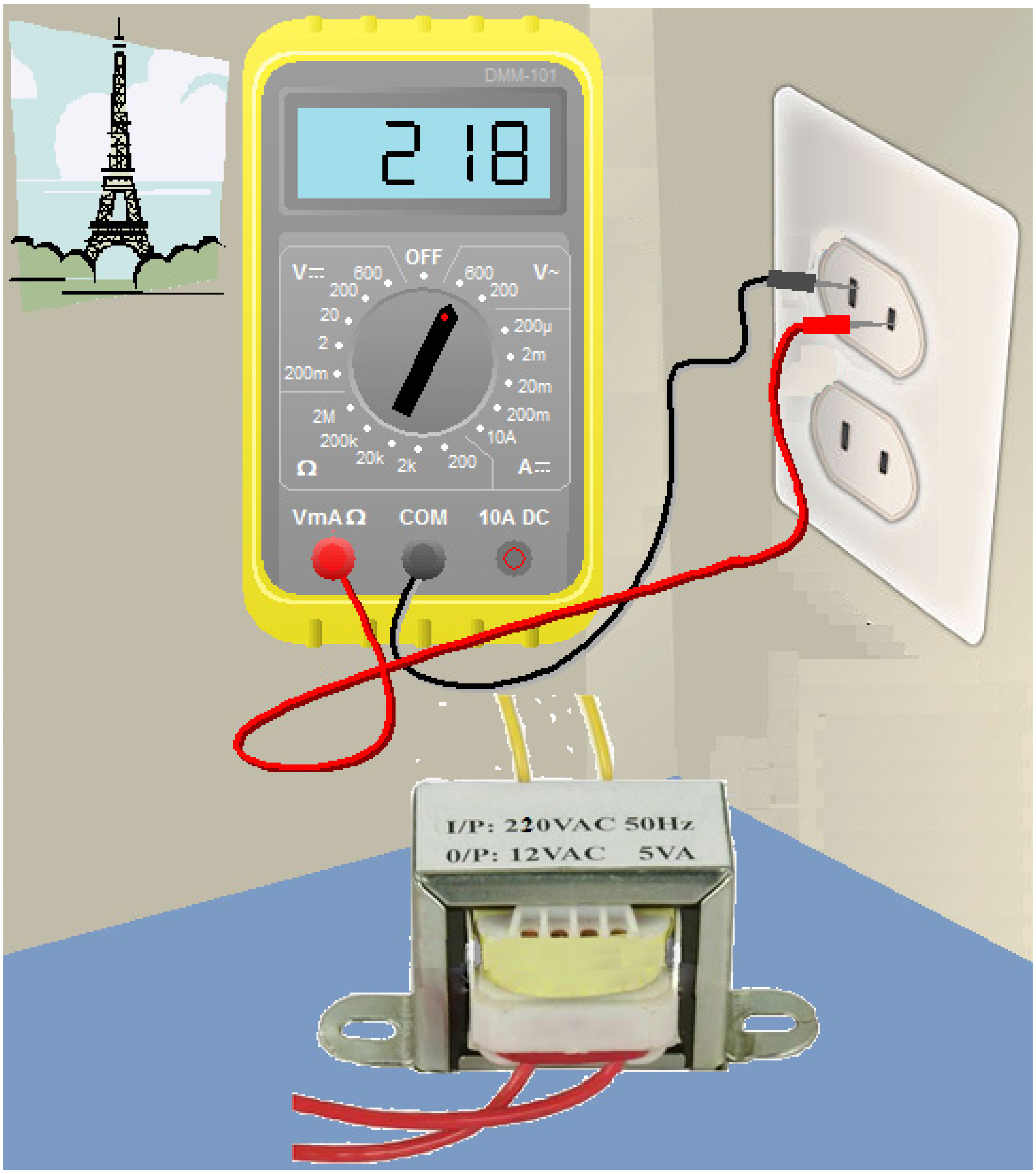


Figure 4-33: Testing transformer from wall outlet socket

4. Connect the primary of a transformer on the wall outlet socket.
5. Set the multimeter to read 12 volts AC and connect the leads of multimeter to the secondary winding of a transformer and observe the reading on multimeter. The reading should be at about 12V. If the reading is 11V or 13V there everything is good up to the transformer. If there is no power go further back on the line and measure the voltage. Keep going until you find voltage. Look for the problem between the point with the voltage and the last point checked that had no voltage. If there is no voltage there then the transformer itself is faulty and you need to replace the transformer.

A word of caution: This test should only be done by an experienced person familiar with electricity.

Inspect transformer visually

Overheating, which causes the internal wiring of the transformer to run at elevated temperatures, is a common cause of transformer failure. Whenever a winding has shorted turns, applying full input voltage may spell disaster in the form of a smoking transformer. If the transformer exterior is swelling or shows what appear to be burn marks, don't waste your time by testing the transformer. This is the sign that shows you that the transformer is burnt and cannot work.

Experiment 4-6: Test transformer from a circuit

Transformers are key electrical components in power supplies. Transformers change the voltage of the system to suit the electronics equipments. Step-down transformers are found in electric power company systems and in a multitude of appliances for household and industrial uses. Size ranges are found from in a cell phone charger up to one the size of power house equipment to the end of a high voltage transmission line. For standard household transformer testing a digital multimeter (DMM) makes a good test.

Part list

No	Item
1	Voltmeter (Digital multimeter)
2	DC power supply motherboard with 220V/12V, 50Hz Transformer

Procedure

1. Determine the wiring of the transformer. The transformer should be clearly labelled. However, it is always best to obtain a schematic of the circuit containing the transformer to determine how the transformer is connected. The schematic for the circuit will be available in the product documentation or on the website of the circuit manufacturer.
2. Identify the transformer inputs and outputs. The first electrical circuit, which is generating the magnetic field, will be connected to the primary of the transformer. The voltage being supplied to the primary should be on both the transformer label and the schematic. The second circuit that is receiving power from the magnetic field will be connected to the secondary of the transformer. The voltage being generated by the secondary should be on both the transformer label and the schematic.
3. Determine the output filtering. It is common to attach capacitors and diodes to the transformer secondary to shape the AC power from the output into DC power. This filtering and shaping will not be available from the transformer label. The filtering and shaping will be shown on the schematic.

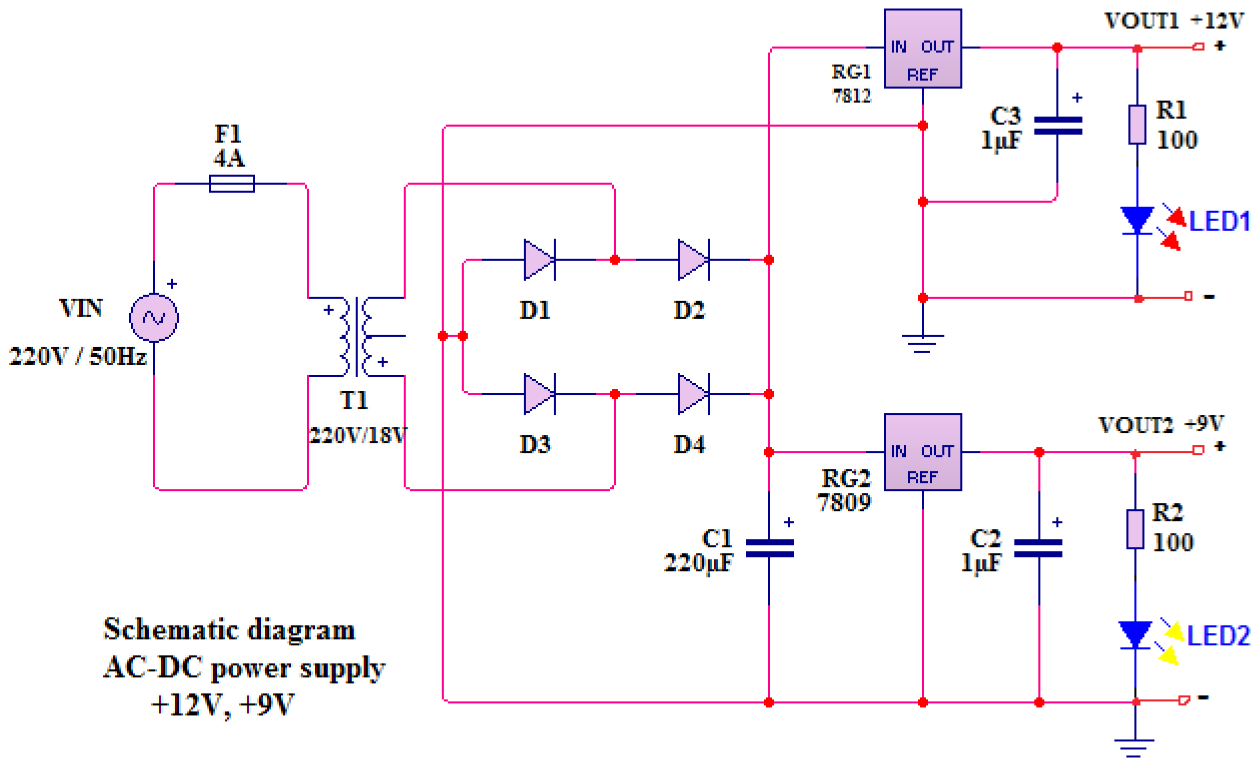


Figure 4-34: Schematic circuit diagram of an AC-DC power supply +12VDV and +9VDC

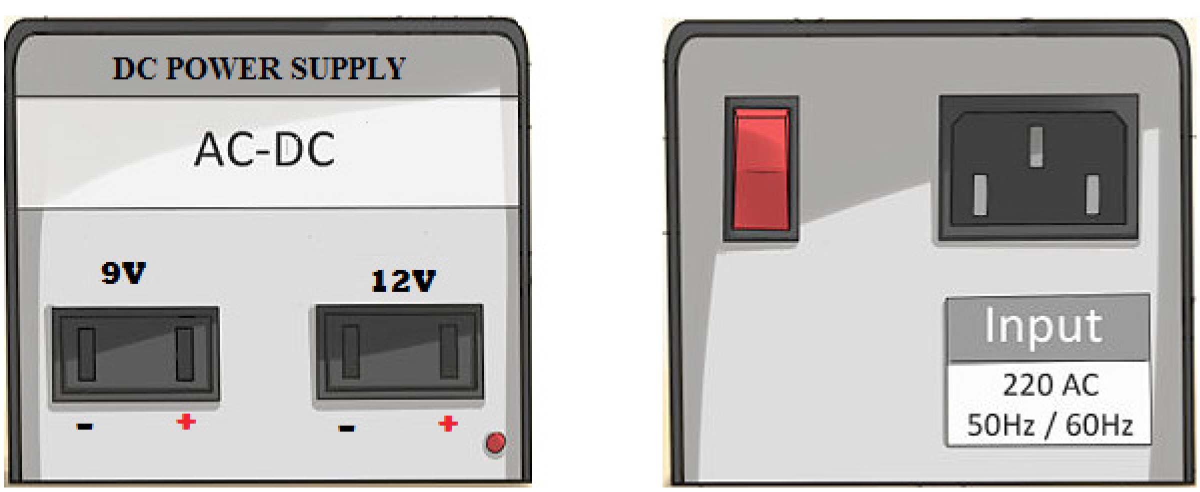


Figure 4-35: Input and output of an AC-DC +12VDC, +9VDC supply

4. Prepare to measure circuit voltages. Remove covers and panels as necessary to gain access to the circuits that contain the transformer. Acquire a digital multimeter (DMM) to take the voltage readings. Arrange the multimeter's lead with red lead connected to VΩ jack and black lead connected to COM jack of multimeter.

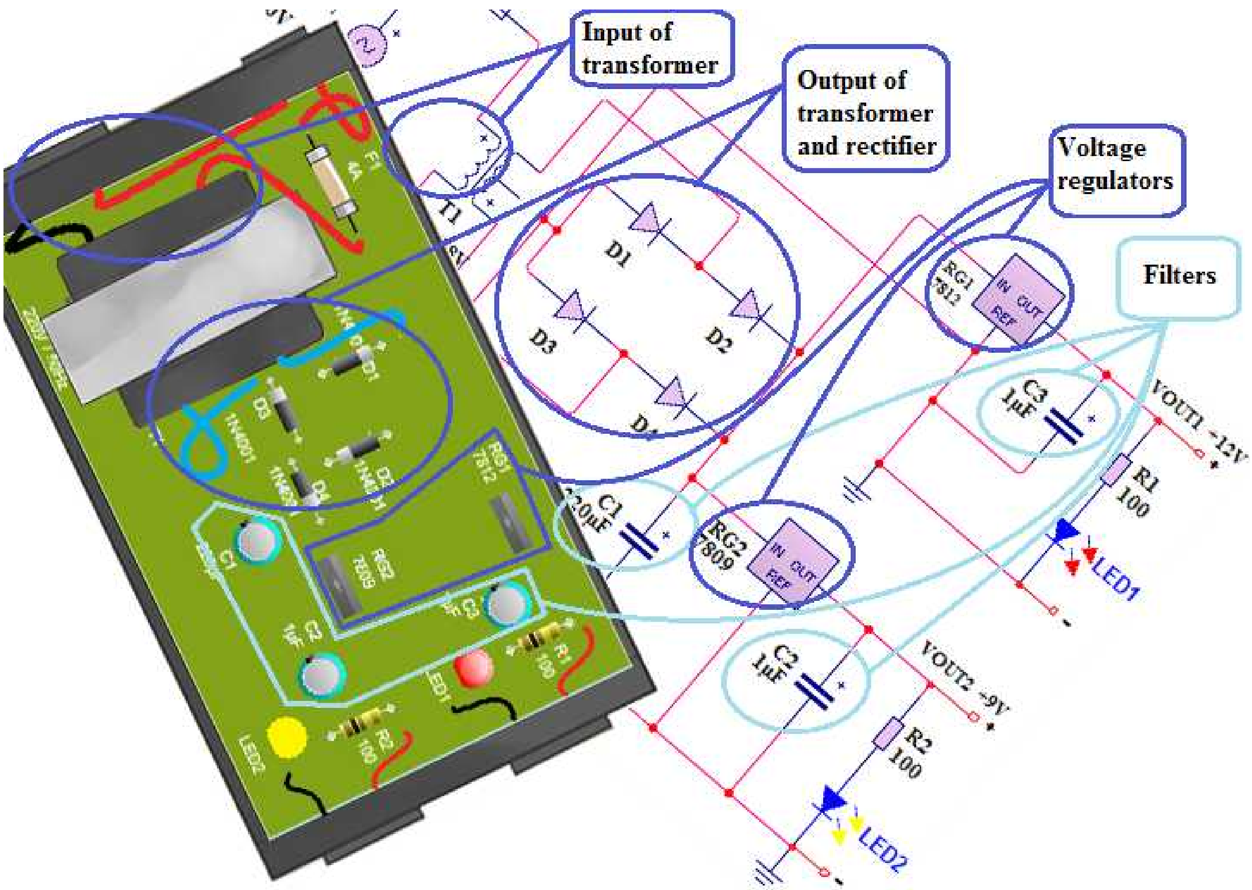


Figure 4-36: Locating different parts of an AC-DC power supply's motherboard

5. Confirm proper input to the transformer. Apply power to the circuit. Use the DMM in AC mode to measure the transformer primary voltage.

   Set multimeter to read AC voltage at 600V scale. If the measurement is less than 80 percent of the expected voltage, the fault could lie in either the transformer or the circuitry providing the primary with power. In this case, the primary must be separated from the input circuit.

   If the input power (transformer primary not disconnected) climbs to the expected value, then the primary of the transformer is faulty. If the input power does not go up to the expected value, then the problem lies not with the transformer, but with the input circuitry.

6. Measure the secondary output of the transformer and regulators. If you have determined that there is no filtering or shaping being performed by the secondary circuitry, use the AC mode of the DMM. If there is filtering and shaping in the secondary circuitry, use the DC mode of the DMM. Set the multimeter in 20V DC scale.

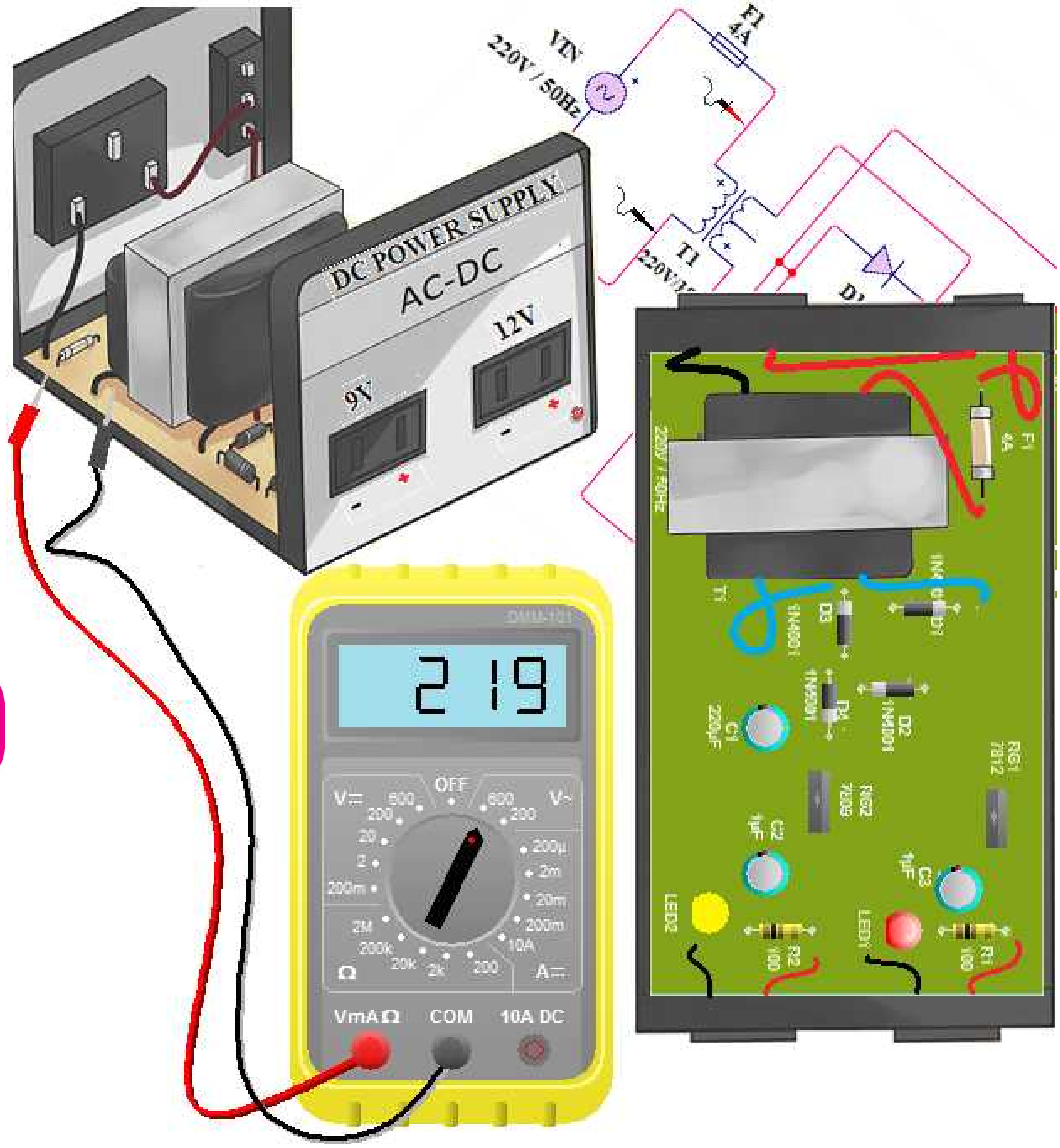


Figure 4-37: Measuring the input AC supply from motherboard of an AC-DC supply real world connection.

If the expected voltage is not present on the secondary, either the transformer or a filtering or shaping component is faulty. Filtering (smoothing) capacitor is located directly at the output of diode rectifier and the shaping capacitor at the output of regulators. Test the filtering and shaping components separately. If the testing of the filtering and shaping components shows no problems, then the transformer is faulty and needs to be replaced.

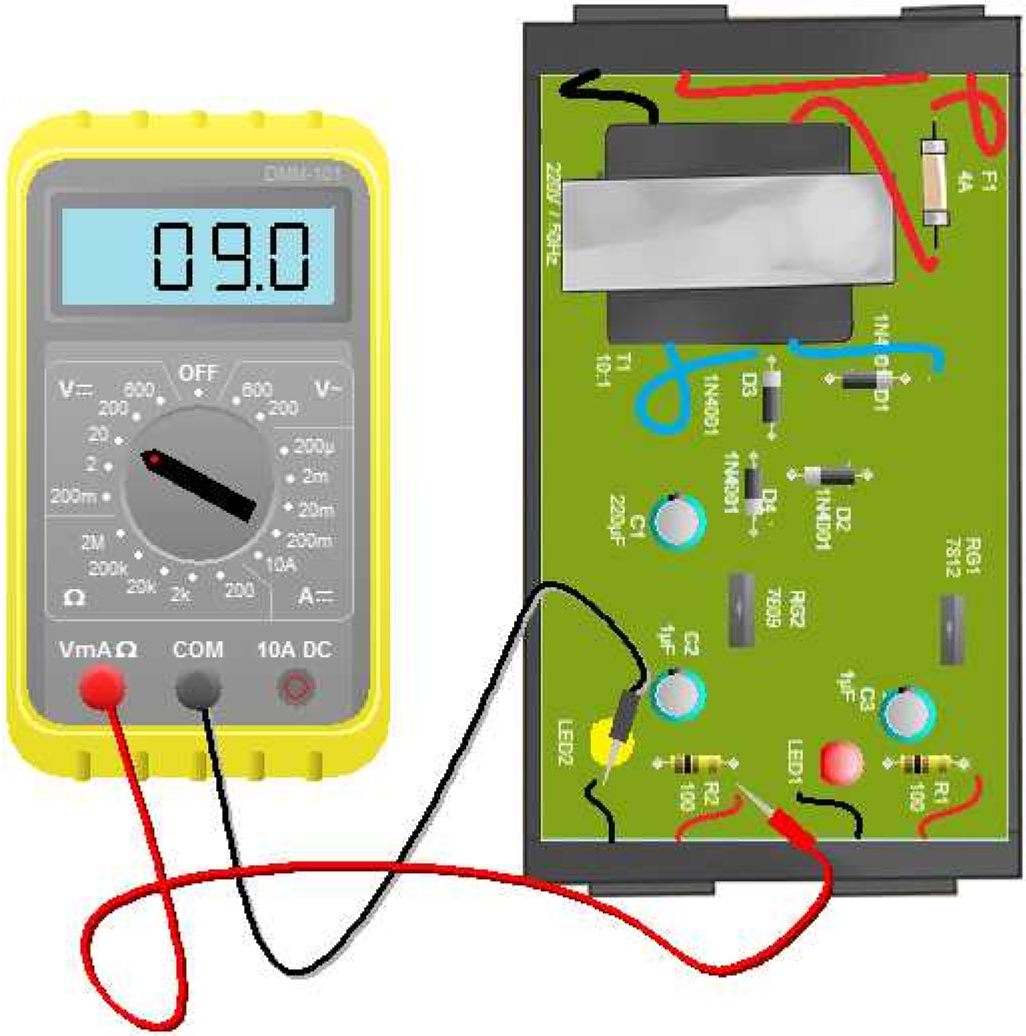


Figure 4-38: Measuring the output from motherboard of an AC-DC supply real world connection.

Identifying Transformer Windings

Most of technician who repair vintage electronic equipment it is not unusual to have several power transformers that have either been removed from some old radio or other piece of equipment lying around. Due to the expense and availability of new power transformers, it would be nice to identify the windings and voltages of these spare units so that they could be reused.

Most of power transformers come with color coding on their leads. If the color coding on the leads is still readable then you only have to refer to the chart to determine the windings. But often the color coding on the leads of these transformers has faded to the point it is difficult or impossible to read the color code and know which winding is which. By using the procedures outlined here, you should be able to identify the windings so the next time you discover the transformer in the set you are about to restore is defective, perhaps one of those spare transformers will qualify as a replacement unit. On a transformer where the color code has faded to the point as to be unreadable, there is a technique to identify the various windings. The ohmmeter is the test instrument we use to accomplish this task.

Below, in figure 4-39, is a diagram of a typical power transformer. This one has a primary winding with a center tapped and a secondary winding also with a center tapped. The diagram gives the color code of the leads for each winding.

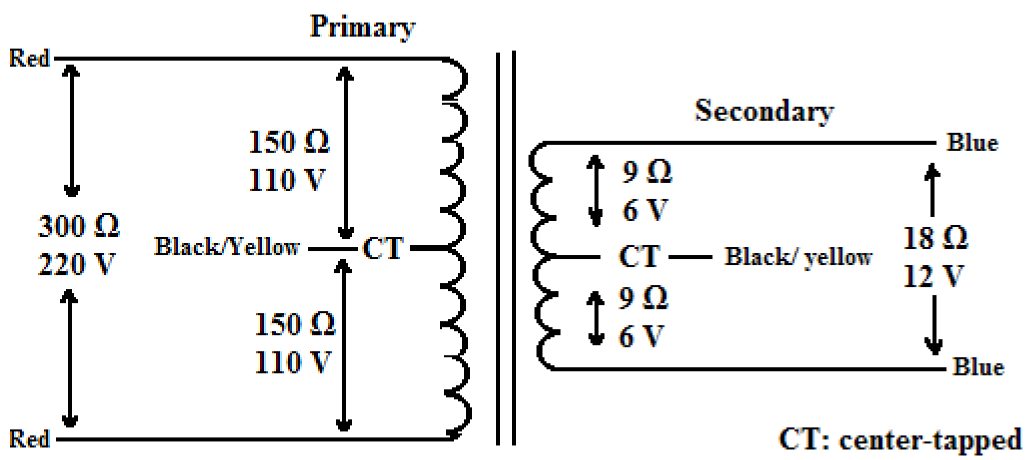


Figure 4-39: Identifying transformer windings

As it can be seen from the diagram, each winding will have continuity from one lead of the winding to the other, but there should be no continuity between different windings, i.e. primary and secondary winding. It is possible to have a secondary with multiple windings which provide different voltages. Also, each set of windings will have a resistance value determined by the number of turns and size of the wire. So we can use the ohmmeter to determine which leads have continuity and use the resistance reading to determine which winding is primary and that of secondary (low and high voltage windings), broken, etc.

Experiment 4-7: Identifying transformer windings using ohmmeter

Parts list

No	Item	Specification
1	Ohmmeter	Digital multimeter
2	Transformer	220-110V/6-0-6V, 50Hz, 2A Transformer

Procedure

1. Set the ohmmeter to 200Ω scale that gives the lowest ohms reading, and calibrate the meter so when the test leads are shorted the meter reads zero ohms. The lowest reading scale is used for the windings having a very low resistance with exception of the high voltage winding. Now pick one of the transformer leads and clip one test lead of the meter to that lead. With the other meter lead, begin checking the other transformer winding leads. Make sure that the transformer is not connected to the supply system. When you check the other lead of the winding under test, the ohmmeter will show continuity. Make a note of the resistance reading. With the first lead still attached, check all other leads to see if any others also have continuity with the first lead. If so, make a note of that reading also. If three leads have continuity, then this winding has a center-tap. Tag these leads as being checked, and proceed to check the other leads in the same manner, noting the resistance readings.

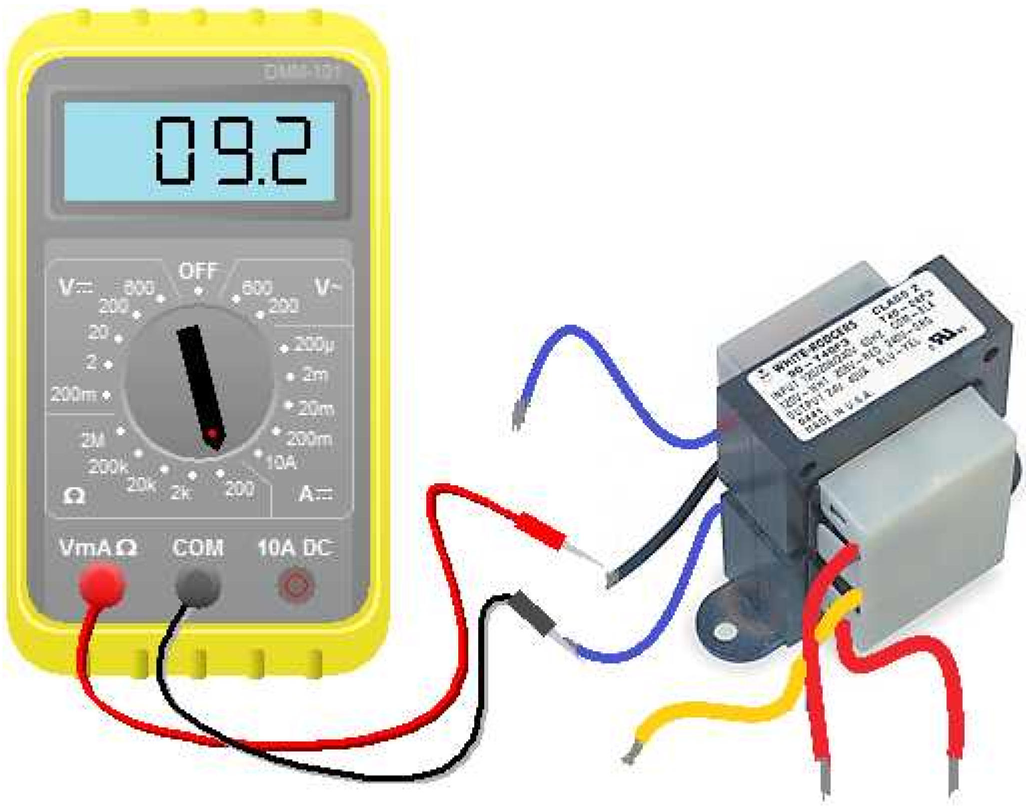


Figure 4-40: Identifying transformer windings using Ohmmeter

2. After all leads have been checked and tagged, it can be determined which windings are which from these readings. Since the high voltage winding has the most turns of wire, it will have the highest reading; secondary winding have only a few turns of larger wire and will have very low readings, with the primary winding having a resistance somewhere in between. A lead that has no continuity with any other lead indicates an open winding.
3. A center-tapped winding will give a resistance reading between the center-tap and the outer windings equal to approximately one-half of the resistance of the two outer windings; however, on secondary windings the resistance may be so low that it may be hard to tell the center-tap from the outer leads.

   We will be able to determine which is the center-tap when we take voltage readings as outlined later. The table 4-3 shows the typical readings for a transformer shown in the above schematic diagram, figure 4-40.

Note: These resistances are approximate. Readings may vary by several ohms from one transformer to the next. The idea is not to be concerned with the exact resistance of the windings, but to identify the various windings by the difference in the resistance of the windings

Winding		Resistance	Corresponding Voltage
Primary	1st Ending terminal - center tapped	150 Ω	110 V
	Center-tapped – 2nd ending terminal	150 Ω	110 V
	1st Ending terminal - 2nd ending terminal	300 Ω	220 V
Secondary	1st Ending terminal - center tapped	9 Ω	6 V
	Center-tapped – 2nd ending terminal	9 Ω	6 V
	1st Ending terminal - 2nd ending terminal	18 Ω	12 V

Table 4-3: Examples on identifying the various windings by the difference in the resistance of the windings

Transformer Replacement

When replacing a transformer in household electronic equipments, consideration must be given to the power handling capabilities of the replacement unit. One rule of thumb is, if the replacement unit's physical core size is as large or larger than the original, it will probably work. The true test however will be if the secondary voltages are up to specifications and the unit does not run to warm when under operating conditions. A transformer should not be so warm that you cannot hold your hand on it after it has been operating for 20 to 30 minutes.

RMA (Radio Manufacturers Association) Standard Power Transformer Lead Color Code	
Winding	**Color code**
Primary	Both black if not tapped, if tapped other winding may be red
Primary tap (optional)	Black/Yellow
High-voltage secondary	Red
High-voltage Center-tap	Red/Yellow
No. 1 Secondary	Yellow
No. 1 Center-tap	Yellow/Black
No. 2 Secondary	Green
No. 2 Secondary Center-tap	Green/Yellow
No. 3 Secondary	Brown
No. 3 Secondary Center-tap	Brown/Yellow
No. 4 Secondary	Slate
No. 4 Secondary Center-tap	Slate/Yellow

Table 4-4: Standard Power Transformer Lead Color Code based on Radio Manufacturers Association.

Note: The number of secondary windings can vary, and may or may not be center-tapped. Windings that are center-tapped, the center-tap will carry the same color as the winding plus a yellow tracer. The exception is the five-volt rectifier winding where the center-tap will carry some other color tracer such as black or blue.

RCA Power Transformer Lead Color Codes Old and New		
Winding	**Old RCA Color Code**	**New Color Code**
Primary	Red	Black
Tapped primary	Start: Red	Black
	Tap: Red and Black	Black and Yellow
	Finish: Black and Red tracer	Black and Red
Rectifier filament	Green and Red tracer	Yellow
Filament (secondary)	Blue	Brown
High voltage	Brown	Red
High voltage Center tap	Brown and black	Red and Yellow

Table 4-5: Old and new power transformer lead color codes

4. 4. Electrical DC Motors

Electrical DC motors, or Direct Current Motors to give their full name, are continuous actuators that convert electrical energy into mechanical energy. The DC motors produce a continuous angular rotation that can be used to rotate pumps, fans, compressors, wheels, etc. As well as conventional rotary DC motors, linear motors are also available which are capable of producing a continuous liner movement.

There are basically three types of conventional electrical motor available: AC type motors, DC type motors and Stepper motors.

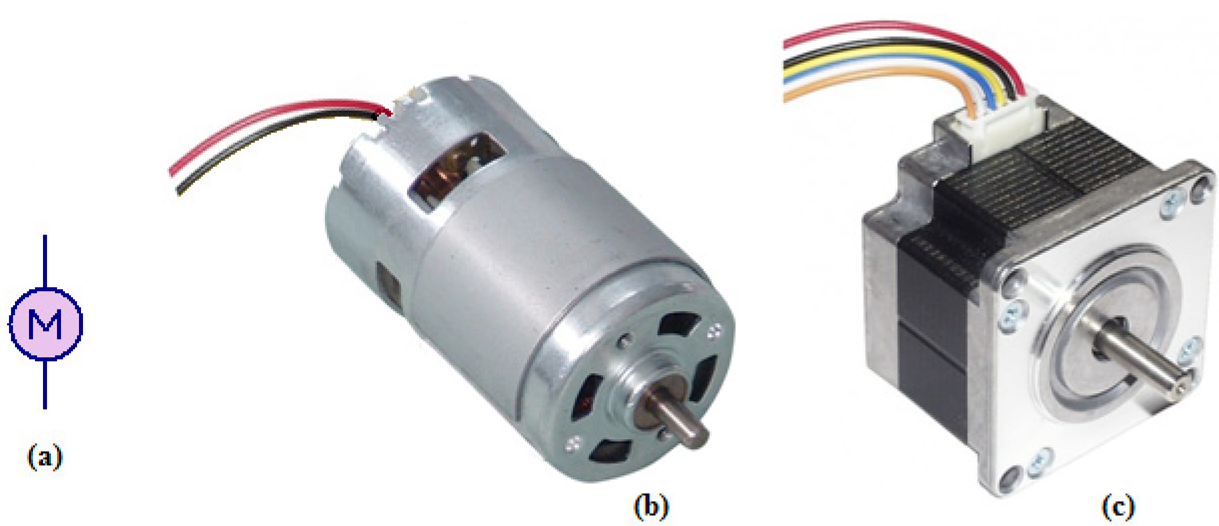


Figure 4-41: DC Motor; (a) DC motor circuit symbol, (b) Typical small DC motor, (c) Typical stepper motor.

AC motors are generally used in high power single or multi-phase industrial applications where a constant rotational torque and speed is required to control large loads such as fans or pumps.

Stepper motors are electromechanical actuators that convert a pulsed digital input signal into a discrete (incremental) mechanical movement. Stepper motors are used widely in industrial control applications. A stepper motor is a type of synchronous brushless motor in that it does not have an armature with a commutator and carbon brushes but has a rotor made up of many, some types have hundreds of permanent magnetic teeth and a stator with individual windings.

In this tutorial on Electrical Motors we will look only at simple light duty DC motors which are used in many electronics, positional control and robotic circuits.

4. 4. 1. The Basic DC Motor

Normal DC motors have almost linear characteristics with their speed of rotation being determined by the applied DC voltage and their output torque being determined by the current flowing through the motor windings. The speed of rotation of any DC motor can be varied from a few revolutions per minute (rpm) to many thousands of revolutions per minute making them suitable for electronic, automotive or robotic applications. By connecting them to gearboxes or gear-trains their output speed can be decreased while at the same time increasing the torque output of the motor at a high speed.

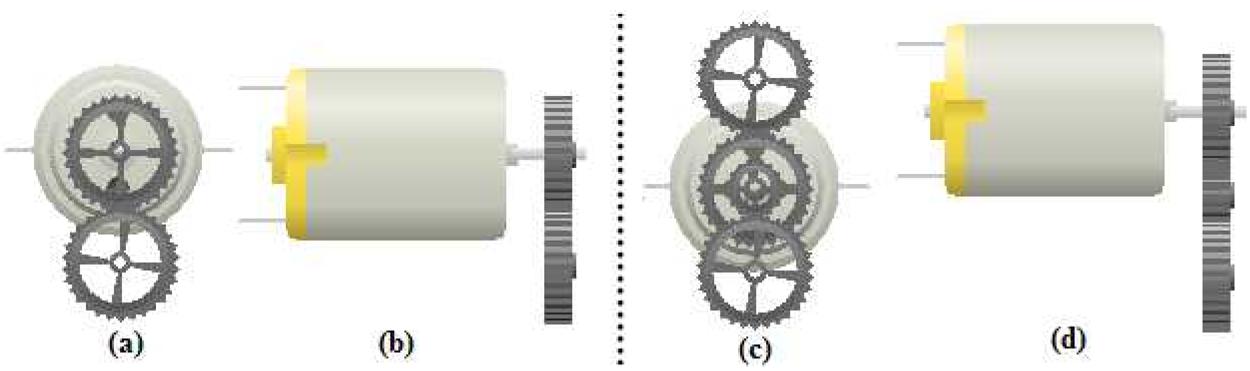


Figure 4-42: Small DC motor with gears; (a) Simple gear train plan view, (b) Simple gear train side view, (c) Compound gear train plan view, (d) Compound gear train side view

The DC Motor is the most commonly used actuator for producing continuous movement and whose speed of rotation can easily be controlled, making them ideal for use in applications were speed control, servo type control, and/or positioning is required. A DC Motor consists of two parts, a "Stator" which is the stationary part and a "Rotor" which is the rotating part. The result is that there are basically three types of DC Motor available.

- **The "Brushed" DC Motor**

This type of motor produces a magnetic field in a wound rotor (the part that rotates) by passing an electrical current through a commutator and carbon brush assembly, hence the term "Brushed". The stators (the stationary part) magnetic field is produced by using either a wound stator field winding or by permanent magnets.

Generally brushed DC Motors are cheap, small and easily controlled.

A conventional brushed DC Motor consists basically of two parts, the stationary body of the motor called the Stator and the inner part which rotates producing the movement called the Rotor or "Armature" for DC machines.

The motors wound stator is an electromagnet circuit which consists of electrical coils connected together in a circular configuration to produce the required North-pole then a South-pole then a North-pole etc, type stationary magnetic field system for rotation, unlike AC machines whose stator field continually rotates with the applied frequency. The current which flows within these field coils is known as the motor field current.

Series and Shunt Connected DC Motor

The electromagnetic coils which form the stator field can be electrically connected in series, parallel or both together (compound) with the motors armature. A series wound DC Motor has its stator field windings connected in *series* with the armature. Likewise, a shunt wound DC motor has its stator field windings connected in *parallel* with the armature as shown in figure 4-43.

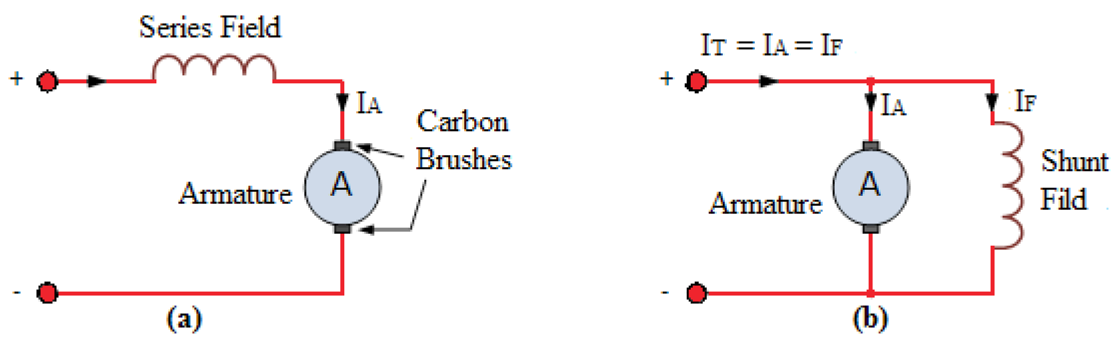


Figure 4-43: Connection of the electromagnetic coil of the brushed DC Motor.
(a) stator field windings connected in *series* with the armature,
(b) Stator field windings connected in *parallel* with the armature

The rotor or armature of a DC machine consists of current carrying conductors connected together at one end to electrically isolated copper segments called the commutator. The commutator allows an electrical connection to be made via carbon brushes (hence the name "Brushed" motor) to an external power supply as the armature rotates.

The magnetic field setup by the rotor tries to align itself with the stationary stator field causing the rotor to rotate on its axis, but cannot align itself due to commutation delays. The rotational speed of the motor is dependent on the strength of the rotors magnetic field and the more voltage that is applied to the motor, the faster the rotor will rotate. By varying this applied DC voltage the rotational speed of the motor can also be varied.

The Permanent magnet brushed DC motor is generally much smaller and cheaper than its equivalent wound stator type DC Motor as they have no field winding. In permanent magnet DC Motors these field coils are replaced with strong rare earth (i.e. Samarium Cobolt, or Neodymium Iron Boron) type magnets which have very high magnetic energy fields.

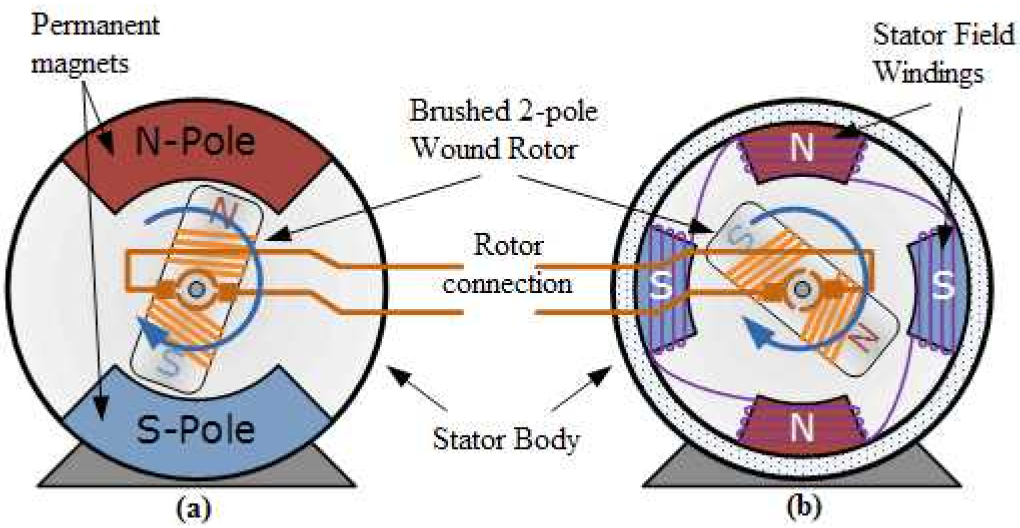


Figure 4-44: Conventional (Brushed) DC Motor; (a) 2-Pole permanent magnet, (b) 4-Pole wound field motor

Advantage and disadvantage of brushed DC motor

Although DC brushed motors are very efficient and cheap, problems associated with the brushed DC motor is that sparking occurs under heavy load conditions between the two surfaces of the commutator and carbon brushes resulting in self generating heat, short life duration and electrical noise due to sparking, which can damage any semiconductor switching device such as a MOSFET or transistor. To overcome these disadvantages, Brushless DC Motors were developed.

Application of brushed DC motor

The use of permanent magnets gives the DC motor a much better linear speed and torque characteristic than the equivalent wound motors because of the permanent and sometimes very strong magnetic field, making them more suitable for use in models, robotics and servos.

- **The "Brushless" DC Motor**

This type of motor produce a magnetic field in the rotor by using permanent magnets attached to it and commutation is achieved electronically. They are generally smaller but more expensive than conventional brushed type DC Motors because they use "Hall effect" switches in the stator to produce the required stator field rotational sequence but they have better torque and speed characteristics, are more efficient and have a longer operating life than equivalent brushed types.

The brushless DC Motor is very similar to a permanent magnet DC Motor, but does not have any brushes to replace or wear out due to commutator sparking. Therefore, little heat is generated in the rotor increasing the motors life. The design of the brushless motor eliminates the need for brushes by using a more complex drive circuit where the rotor magnetic field is a permanent magnet which is always in synchronization with the stator field allows for a more precise speed and torque control.

The construction of a brushless DC motor is very similar to the AC motor making it a true synchronous motor but one disadvantage is that it is more expensive than an equivalent "brushed" motor design.

The control of the brushless DC Motors is very different from the normal brushed DC Motor, in that this type of motor incorporates some means to detect the rotors angular position (or magnetic poles) required to produce the feedback signals required to control the semiconductor switching devices. The most common position and pole sensor is the "Hall Effect Sensor", but some motors also use optical sensors.

Using Hall Effect sensors, the polarity of the electromagnets is switched by the motor control drive circuitry. Then the motor can be easily synchronized to a digital clock signal, providing precise speed control. Brushless DC Motors can be constructed to have an external permanent magnet rotor and an internal electromagnet stator or an internal permanent magnet rotor and an external electromagnet stator.

Advantage of Brushless DC Motor

Advantages of the Brushless DC Motor compared to "brushed" is higher efficiencies, high reliability, low electrical noise, good speed control and more significantly, no brushes or commutator to wear out producing a much higher speed. However their disadvantage is that they are more expensive and more complicated to control.

- **The DC Servo Motor**

This type of motor is basically a brushed DC motor with some form of positional feedback control connected to the rotor shaft. They are connected to and controlled by a PWM (Pulse Width Modulation) type controller and are mainly used in positional control systems and radio controlled models.

DC Servo motors are used in closed loop type applications were the position of the output motor shaft is fed back to the motor control circuit. Typical positional "Feedback" devices include Resolvers, Encoders and Potentiometers as used in radio control models such as aeroplanes and boats etc..

A servo motor generally includes a built-in gearbox for speed reduction and is capable of delivering high torques directly. The output shaft of a servo motor does not rotate freely as do the shafts of DC motors because of the gearbox and feedback devices attached.

DC Servo Motor Block Diagram

A servo motor consists of several devices in one package such as a DC Motor, reduction gearbox, positional feedback device and some form of error correction for controlling position, direction or speed. The speed or position is controlled in relation to a positional input signal or reference signal applied to the device.

The error detection amplifier looks at this input signal and compares it with the feedback signal from the motors output shaft and determines if the motor output shaft is in an error condition and, if so, the controller makes appropriate corrections either speeding up the motor or slowing it down. This response to the positional feedback device means that the servo motor operates within a "Closed Loop System".

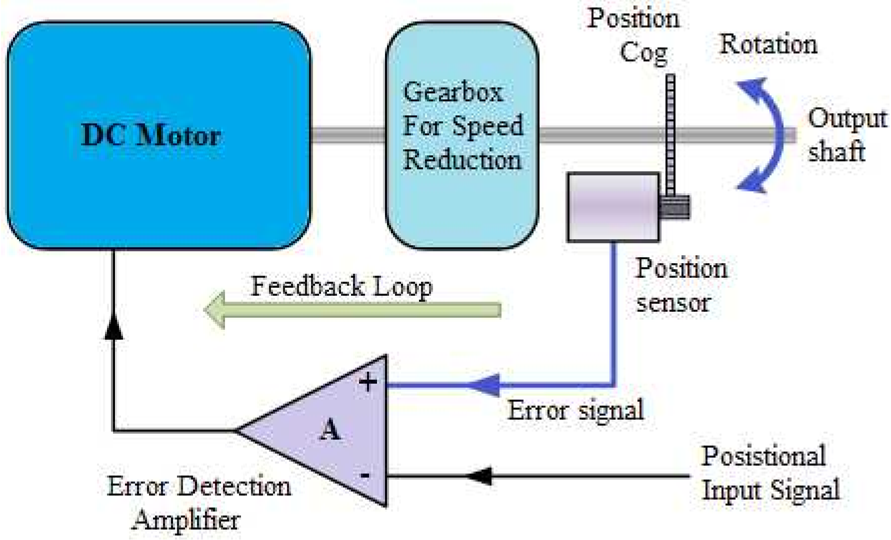


Figure 4-45: DC servo motor block diagram

Application of Servo Motor

Servo Motors are widely used in robotics and small models as they are easily controlled using just three wires, Power, Ground and Signal Control. As well as large industrial applications, servo motors are also used in small remote control models and robotics, with most servo motors being able to rotate up to about 180 degrees in both directions making them ideal for accurate angular positioning. However, these type are unable to continually rotate at high speed like conventional DC motors unless specially modified.

4. 4. 2. DC Motor Switching and Control

Small DC motors can be switched "On" or "Off" by using of switches, relays, transistors or MOSFET circuits.

Transistor Switching

DC motor transistor switching can use either bipolar transistor (BJT) or MOSFET (metal oxide semiconductor field effect transistor) as shown in circuits of figure 4-46.

Figure 4-46 part (a) shows a basic type of circuit using a bipolar transistor as a switch (A Darlington transistor may also be used were a higher current rating is required) to control the motor from a single power supply. By varying the amount of base current flowing into the transistor the speed of the motor can be controlled for example, if the transistor is turned on "half way", then only half of the supply voltage goes to the motor. If the transistor is turned "fully ON" (saturated), then all of the supply voltage goes to the motor and it rotates faster. Then for this linear type of control, power is delivered constantly to the motor. Figure 4-46 part (b) shows motor switching by means of MOSFET.

A continuous logic "1" or logic "0" from a control circuit is applied to the input of the circuit to turn the motor "ON" (saturation) or "OFF" (cut-off) respectively.

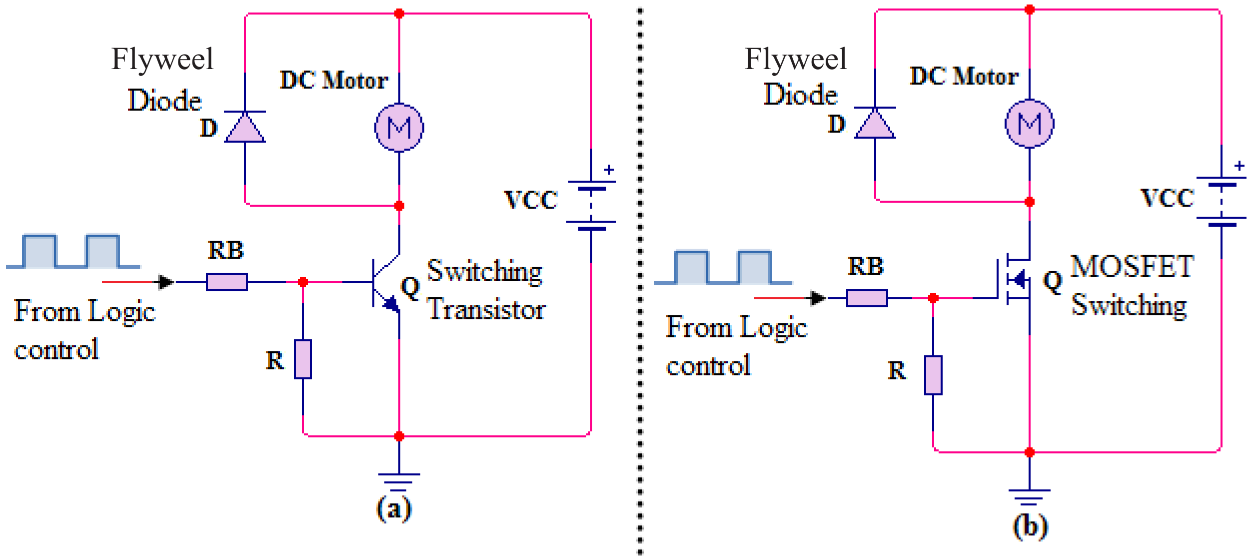


Figure 4-46: DC motor transistor switching; (a) Bipolar transistor switching, (b) MOSFET switching

A flywheel diode is connected across the motor terminals to protect the switching bipolar transistor or MOSFET from any back electromotive force (emf) generated by the motor when the transistor turns the supply "OFF". In case of high power applications, a relay may be used as shown in the figure 4-47.

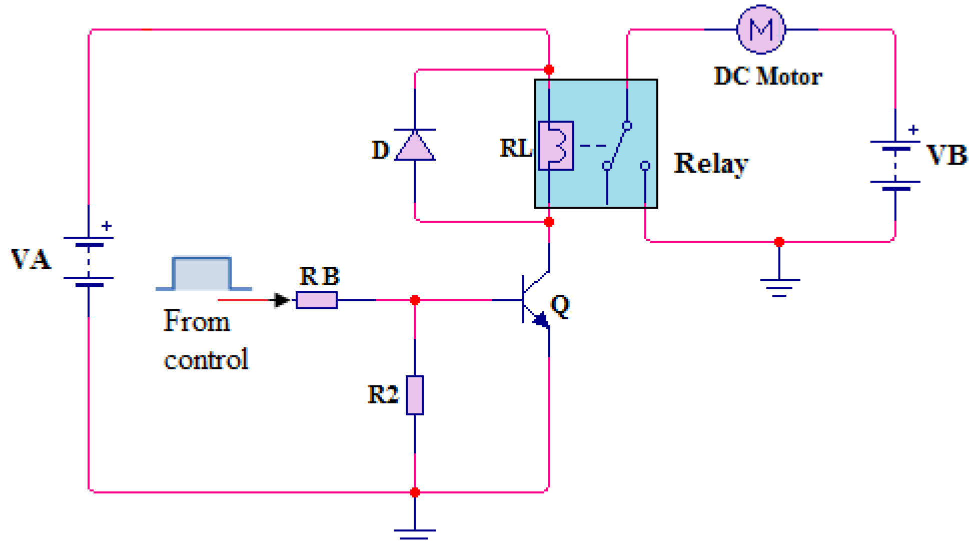
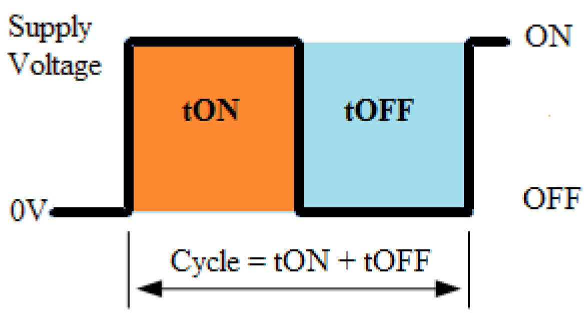


Figure 4-47: Transistor motor switching using relay, VA is assumed to be less than VB

As well as the basic "ON/OFF" control, the same circuit can also be used to control the motors rotational speed. By repeatedly switching the motor current "ON" and "OFF" at a high enough frequency, the speed of the motor can be varied between stand still (0 rpm) and full speed (100%). This is achieved by varying the proportion of "ON" time (t_{ON}) to the "OFF" time (t_{OFF}) and this can be achieved using a process known as Pulse Width Modulation.

Pulse Width Speed Control

The rotational speed of a DC motor is directly proportional to the mean (average) value of its supply voltage and the higher this value, up to maximum allowed motor volts, the faster the motor will rotate. In other words more voltage more speed. By varying the ratio between the "ON" (t_{ON}) time and the "OFF" (t_{OFF}) time durations, called the "Duty Ratio", "Mark/Space Ratio" or "Duty Cycle", the average value of the motor voltage and hence its rotational speed can be varied. For simple unipolar drives the duty ratio β is given as:



$$\text{Duty ration } \beta = \frac{tON}{tON+tOFF}$$

The mean DC output voltage fed to the motor is given as: $V_{mean} = \beta \times V_{supply}$. Then by varying the width of pulse t_{ON}, the motor voltage and hence the power applied to the motor can be controlled and this type of control is called Pulse Width Modulation or PWM.

Another way of controlling the rotational speed of the motor is to vary the frequency (and hence the time period of the controlling voltage) while the "ON" and "OFF" duty ratio times are kept constant. This type of control is called Pulse Frequency Modulation or PFM.

With pulse frequency modulation, the motor voltage is controlled by applying pulses of variable frequency for example, at a low frequency or with very few pulses the average voltage applied to the motor is low, and therefore the motor speed is slow. At a higher frequency or with many pulses, the average motor terminal voltage is increased and the motor speed will also increase.

Then, Transistors can be used to control the amount of power applied to a DC motor with the mode of operation being either "Linear" (varying motor voltage), "Pulse Width Modulation" (varying the width of the pulse) or "Pulse Frequency Modulation" (varying the frequency of the pulse). This method, PWM, will be discussed in chapter seven.

Experiment 4-8: DC motor speed control using different dc voltage source

Part lit

No	Item	Quantity
1	DC motor 9V, 1A, 10000rpm	1
2	Battery of 4.5V	1
3	Battery of 9V	1
4	Breadboard	1
5	Push button (NO)	1

Procedure

1. Connect the DC motor as illustrated in figure 4-48 and figure 4-49.

2. Start with 4.5V supply and press on push button PB, hold a while until the motor reaches the maximum speed.
3. Observing the rotation speed of the motor's propeller.
4. As indicated in figure 4-48, a 9V dc motor supplied by 4.5V rotates at 1,701.59rpm.

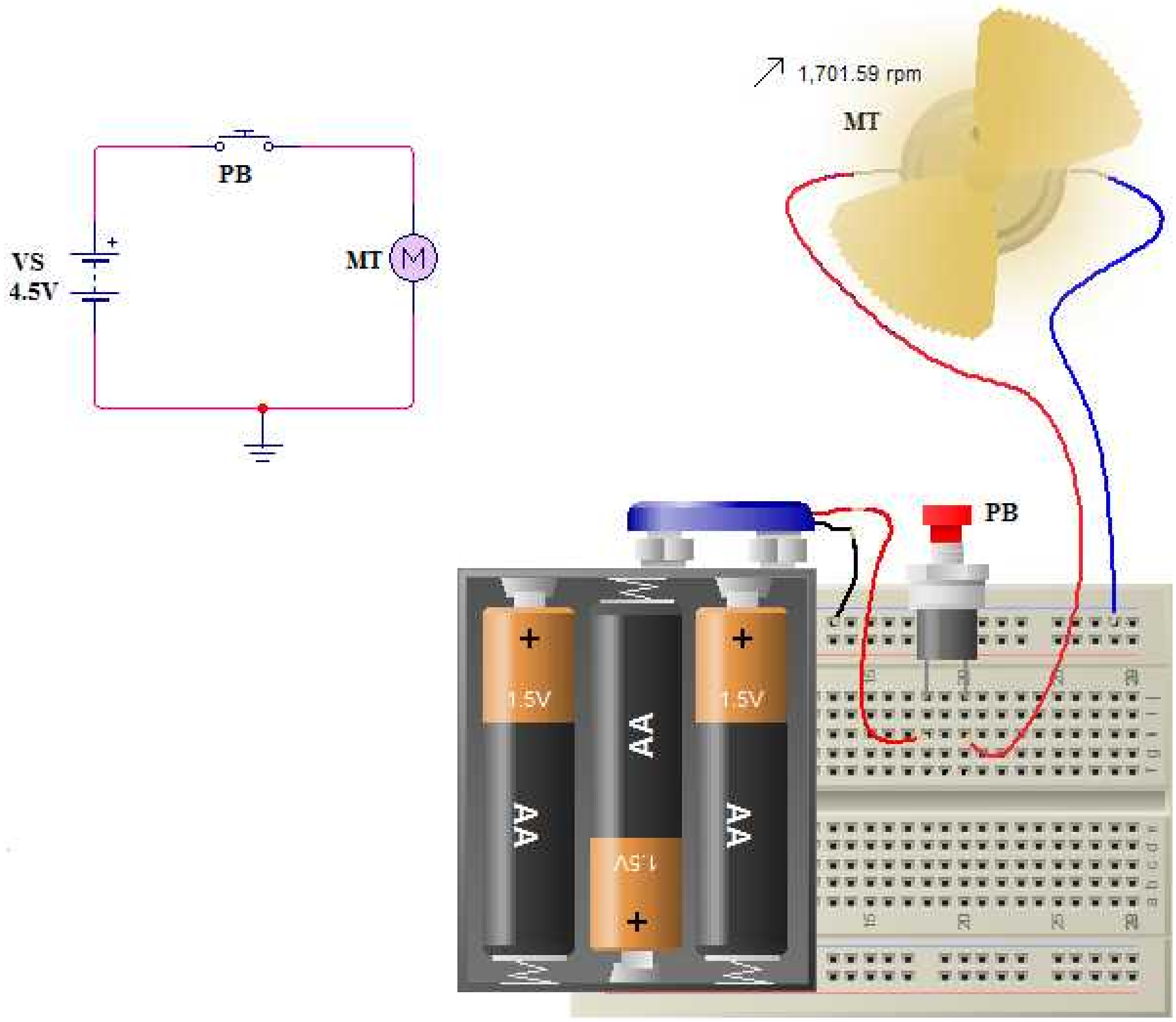


Figure 4-48: DC motor supplied with low dc voltage rotate at low revolution per minute (rpm)

5. Then after, replace 4.5 V battery with 9 V battery and again observe the rotation speed of the motor's propeller.
6. As indicated in figure 4-49, a 9V dc motor is supplied with its nominal voltage thus, the motor rotates at its maximum rpm, i.e. 10,000 rpm.
7. You may see that as the voltage of the dc motor increases, its rotation speed also increases.
8. If you have instrument to measure rotation speed you can record the speed corresponding to each voltage.

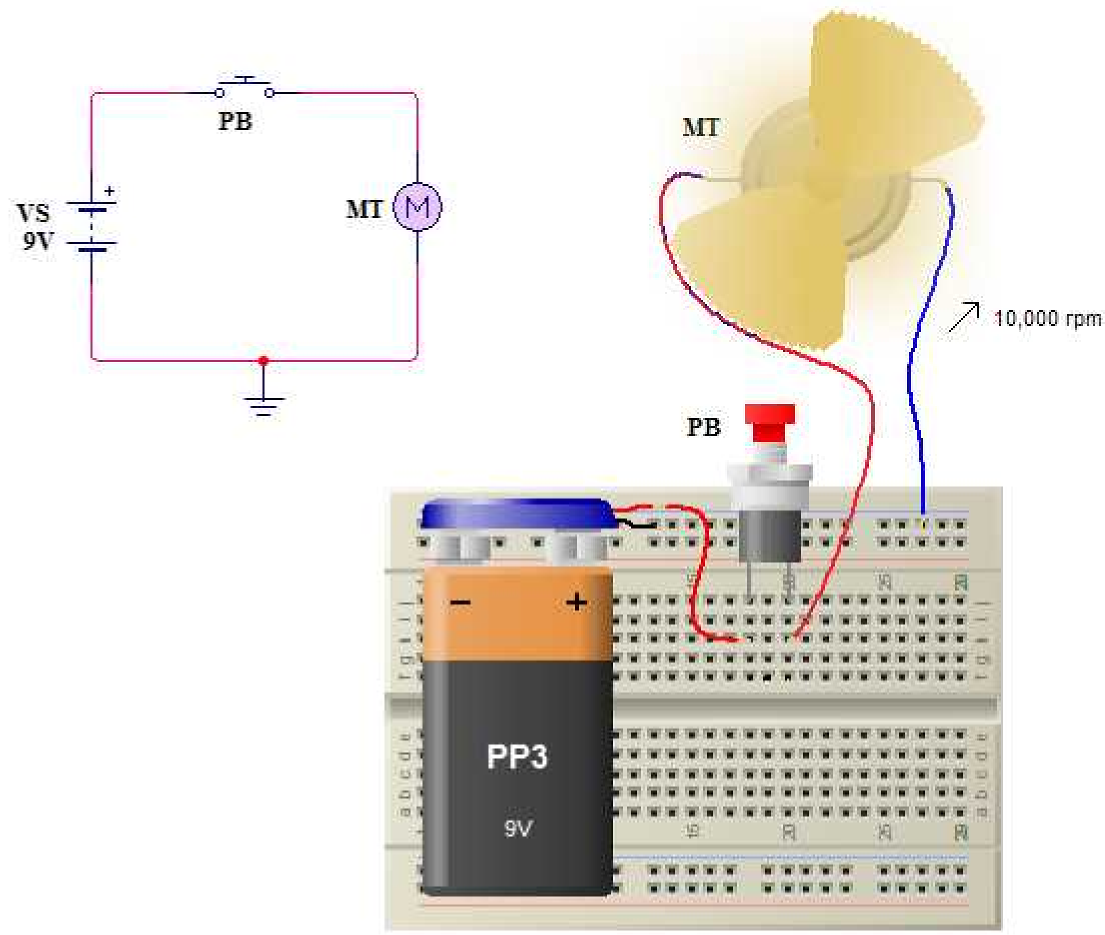


Figure 4-49: DC motor supplied with dc voltage equivalent to its specification rotate at its nominal revolution per minute (rpm)

4. 4. 3. Reversing the Direction of a DC Motor

While controlling the speed of a DC motor with a single transistor has many advantages it also has one main disadvantage, the direction of rotation is always the same, it is a "Unidirectional" circuit. In many applications we need to operate the motor in both directions forward and back.

To control the direction of a DC motor, the polarity of the DC power applied to the motor's connections must be reversed allowing its shaft to rotate in the opposite direction. One very simple and cheap way to control the rotational direction of a DC motor is to use different switches arranged in the following manner:

DC Motor Directional Control Using Switches

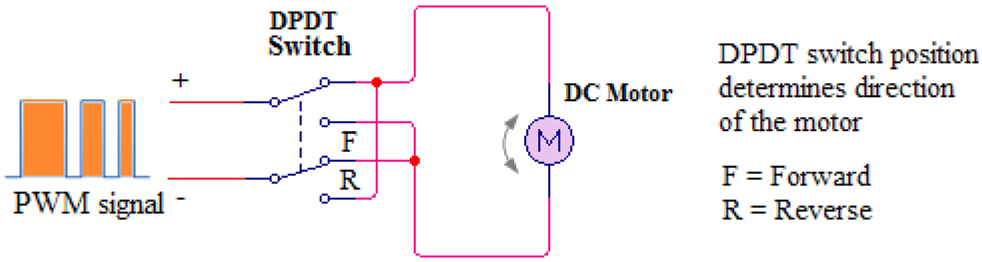


Figure 4-50: Controlling direction of DC motor using double pole double throw (DPDT) switch

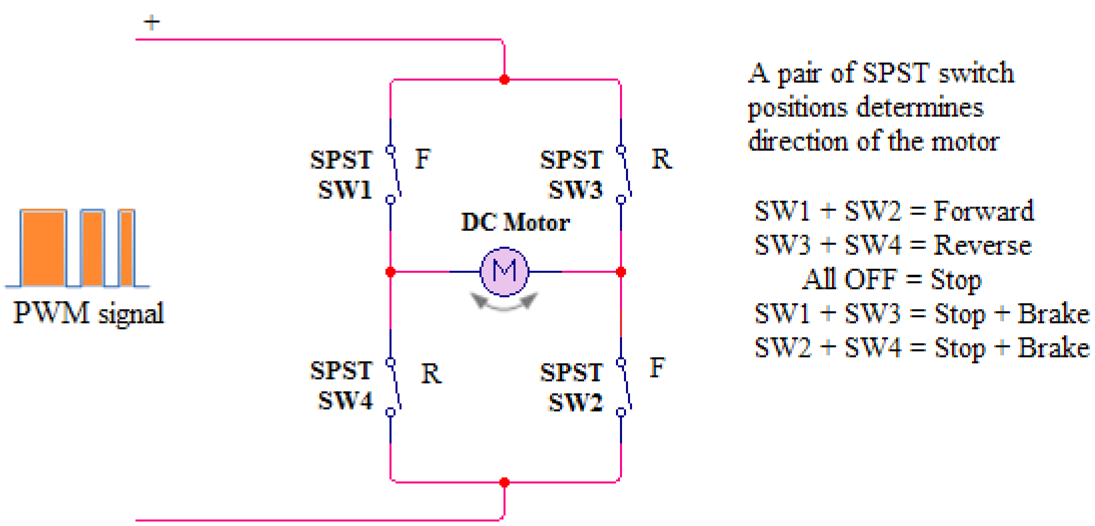


Figure 4-51: Controlling direction of DC motor using a pair of single pole, single throw (SPST) switches

The first circuit, figure 4-50, uses a single double-pole, double-throw (DPDT) switch to control the polarity of the motor's connections. By changing over the contacts the supply to the motors terminals is reversed and the motor reverses direction. The second circuit, figure 4-51, is slightly more complicated and uses four single-pole, single-throw (SPST) switches arranged in an "H" configuration.

The mechanical switches are arranged in switching pairs and must be operated in a specific combination to operate or stop the DC motor. For example, switch combination $SW_1 + SW_2$ controls the forward rotation while switches $SW_3 + SW_4$ control the reverse rotation as shown. Switch combinations $SW_1 + SW_3$ or $SW_2 + SW_4$ shorts out the motor terminals causing it to brake quickly until it stops.

However, using switches in this manner has its danger as operating switches $SW_1 + SW_4$ or $SW_3 + SW_2$ together would short out the power supply. These conditions have to be avoided.

The two circuits above would work very well for most small DC motor applications and no reason to operate different combinations of mechanical switches to reverse the direction of the motor.

chap 4

We could change the manual switches for set of electromechanical relays and have a single forward-reverse button or switch or even use a solid state CMOS 4066B quad bilateral switch.

Experiment 4-9: Reversing the direction of the dc motor by reversing supply polarity

Part list

No	Item	Quantity
1	9V dc motor	1
2	9V dc supply (battery)	1

Procedure

1. Connect the circuit as illustrated in schematic circuit figure 4-52
2. A is positive terminal of the dc motor, and B is the negative terminal of the dc motor

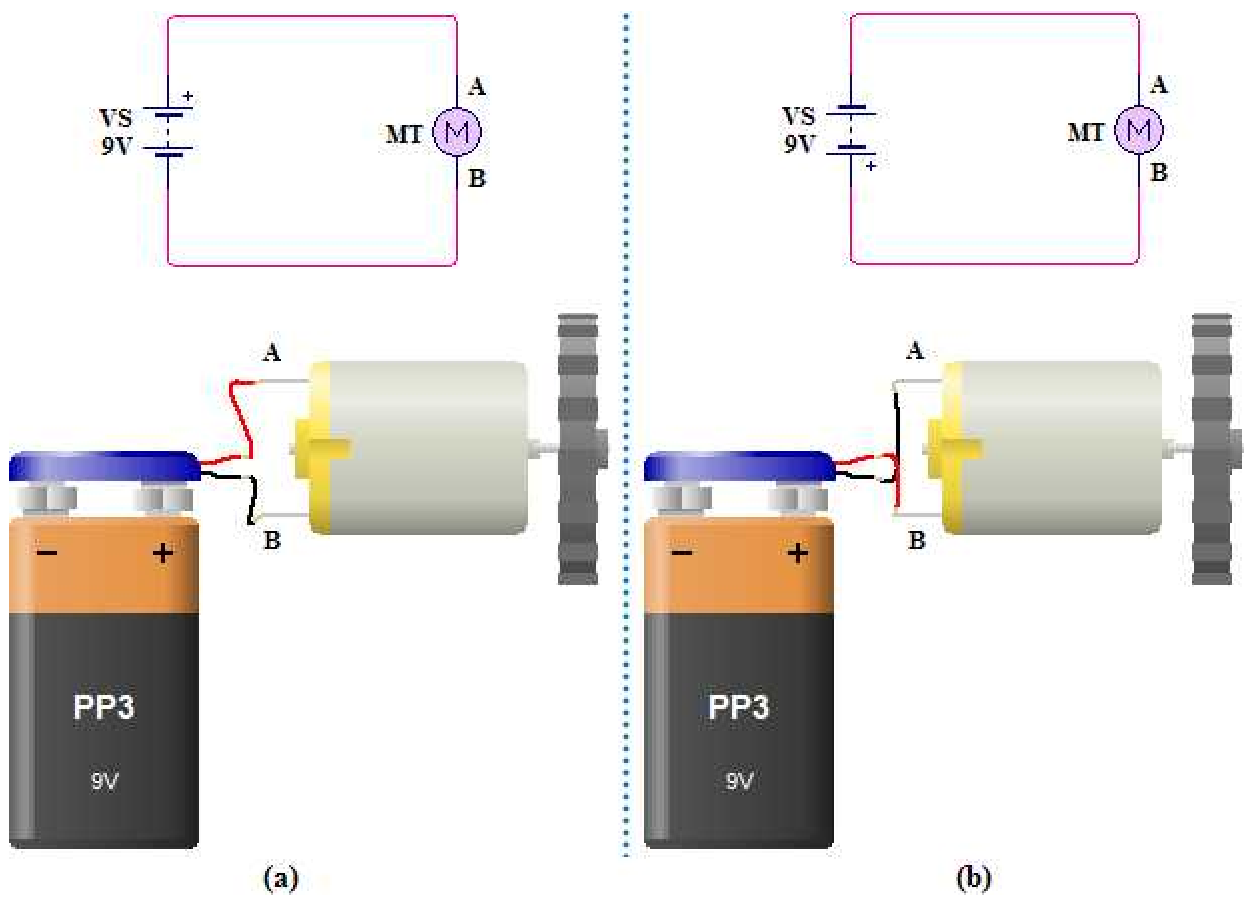


Figure 4-52: Reversing motor rotation by reversing dc source terminals, (a) Terminal A is positive with respect to terminal B and motor rotate in clockwise, (b) Terminal B is positive with respect to terminal A and motor rotate in anticlockwise.

3. Connect the positive of the battery on terminal A of the motor and negative of the battery on terminal B of the dc motor as indicated in figure part (a). Then, terminal A is positive with respect to terminal B with a result, motor rotates in clockwise (forward).

4. Disconnect the motor for it to stop and interchange the polarity connection.
5. Connect the positive of the battery to terminal B of the dc motor and the negative terminal of the battery to terminal A of the motor as indicated in figure 4-52 part (b). Therefore, terminal A is negative with respect to terminal B of the dc motor, as result, the motor rotates in anticlockwise (reverse).

Another very good practical way of achieving bi-directional control of a motor (as well as its speed) is to connect the motor into a transistor H-bridge type circuit arrangement.

DC Motor Directional Control Using H-bridge

H-bridge is an electronic circuit that enables a voltage to be applied across a load in either direction. These circuits are often used in robotics and other applications to allow DC motors to run forwards and backwards (Bi-directional). Figure 4-53 shows basic circuit arrangement of H-bridge using complementally bipolar transistor.

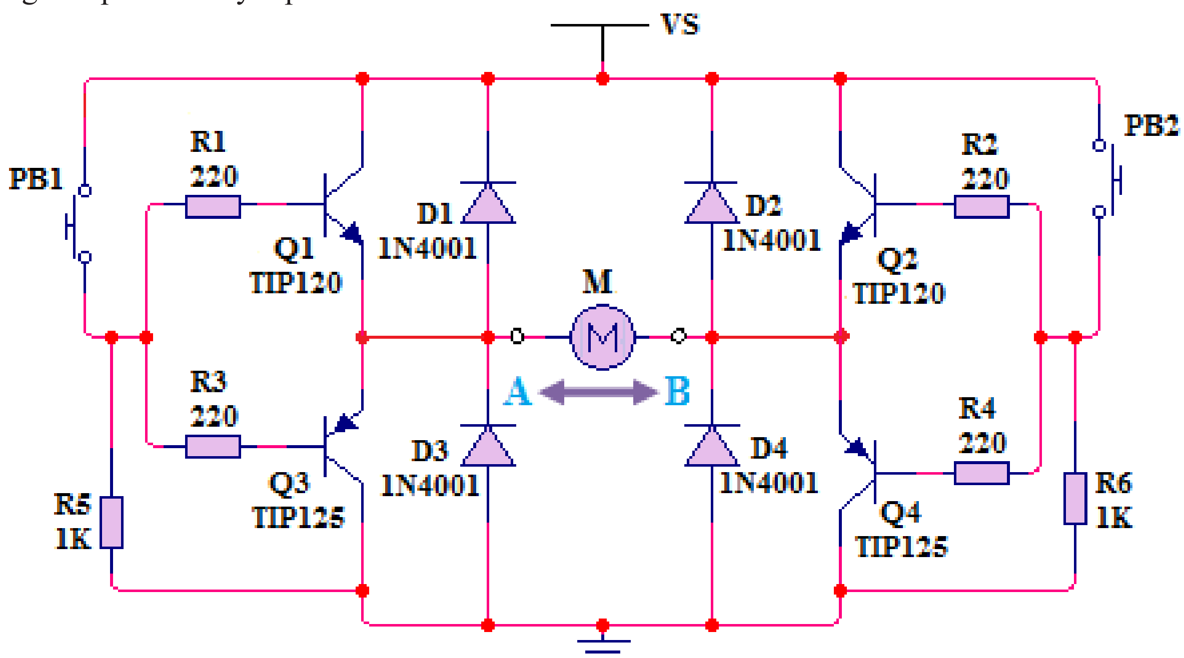


Figure 4-53: Basic circuit for H-bridge

The H-bridge circuit is so named because the basic configuration of the four switches, either electro-mechanical relays or transistors resembles that of the letter "H" with the motor positioned on the centre bar. The Transistor or MOSFET H-bridge is probably one of the most commonly used types of bi-directional DC motor control circuits. It uses “complementary transistor pairs” both NPN and PNP, such as TIP120 and TIP125, 2SD313 and 2SB507 and many more, in each branch with the transistors being switched together in pairs to control the motor.

Control PB1 operates the motor in one direction i.e., Forward rotation while PB2 operates the motor in the other direction i.e., Reverse rotation. Then by switching the transistors “ON” or “OFF” in their “diagonal pairs” results in directional control of the motor.

In this circuit normally PB_1 and PB_2 are open. So the bases of the transistors are grounded. Hence Q_3 and Q_4 are ON, Q_1 and Q_2 are OFF. The voltage at both the motor terminals is the same and hence the motor is OFF. Similarly when both PB_1 and PB_2 are “PUSH” the motor is OFF.

chap 4

When PB1 is pushed, Q_1 becomes ON since it is NPN. This type of transistor needs high potential at the base to turn ON. But the Q_3 is OFF because it is PNP and it needs a low voltage to turn on.

PB2 is still OPEN so the transistor Q_2 is OFF and Q_4 is ON because low potential appear at the base of the transistors. So the current flows from VS through Q_1, M, Q_4 and to GND. So motor rotates one direction, i.e. forward. Normally when the transistor Q_1 is "ON" and transistor Q_2 is "OFF", point A is connected to the supply voltage (+VS) and if transistor Q_3 is "OFF" and transistor Q_4 is "ON" point B is connected to 0 volts (GND). Then the motor will rotate in one direction corresponding to motor terminal A being positive and motor terminal B being negative. At this time the motor rotate in forward.

When PB2 is ON and PB1 OFF similarly Q_2 and Q_3 are ON and Q_1 and Q_4 are OFF. Now current flows from VS through Q_3, M, Q_2 and to GND. Now motor rotates another direction, i.e. reverse.

The transistors are emitter followers and the voltage on the motor will be less than the voltages of the circuit because the output voltage will be determined by the slight drop across the 1KΩ+220Ω and the voltage drop across the base-emitter junction of the transistor. The total voltage drop to the motor (due to both sides of the bridge) will be about 2V.

This circuit has advantage that when both inputs are high or low this emitter follower H-bridge circuit has not any current flow.

Then, by applying opposite logic levels "1" or "0" to the push buttons PB_1 and PB_2 the motors rotational direction can be controlled as follows.

chap 4

Push button PB1 Q1 and Q4	Push button PB2 Q2 and Q3	Motor Function
0	0	Motor Stopped (OFF)
1	0	Motor Rotates Forward
0	1	Moto Rotates Reverse
1	1	Motor brakes

Table 4-6: H-bridge truth table

It is important that no other combination of inputs are allowed as this may cause the power supply to be shorted out, i.e. both transistors, Q_1 and Q_2 switched "ON" at the same time.

With unidirectional DC Motor control as seen above, the rotational speed of the motor can also be controlled using Pulse Width Modulation (PWM). Then by combining H-bridge switching with PWM control, both the direction and the speed of the motor can be accurately controlled.

Commercial of the shelf decoder integrated circuits (IC) such as the SN754410 Quad Half H-Bridge IC or the L298N which has 2 H-Bridges are available with all the necessary control and safety logic built in are specially designed for H-bridge bi-directional motor control circuits.

4. 5. The Solenoid

The solenoid is a type of electromagnetic actuator that has a movable iron core called a *plunger* and converts an electrical signal into a magnetic field. The movement of this iron core depends on both an electromagnetic field and mechanical spring force.

Types of Solenoid

Solenoids are available in a variety of formats with the more common types being the linear solenoid also known as the Linear Electromechanical Actuator (LEMA) and the rotary solenoid. Both types of solenoid, linear and rotational are available as either a holding (continuously energized) or as a latching type (ON-OFF pulse) with the latching types being used in either energized or power-off applications.

Working Principle of the Solenoid

When electrical current flows through a conductor it generates a magnetic field, and the direction of this magnetic field with regards to its North and South Poles is determined by the direction of the current flow within the wire. This coil of wire becomes an "Electromagnet" with its own north and south poles exactly the same as that for a permanent type magnet.

The strength of this magnetic field can be increased or decreased by either controlling the amount of current flowing through the coil or by changing the number of turns or loops that the coil has. Here below, figure 4-54, is an example of an "Electromagnet".

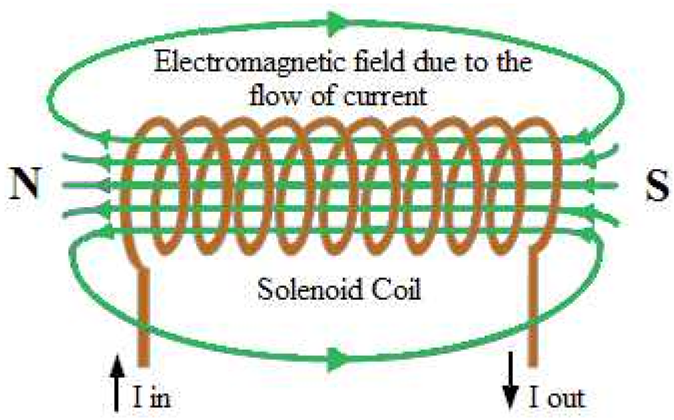


Figure 4-54: Magnetic field produced by a Coil

When an electrical current is passed through the coil windings, it behaves like an electromagnet and the plunger, which is located inside the coil, is attracted towards the centre of the coil by the magnetic flux setup within the coils body, which in turn compresses a small spring attached to one end of the plunger. The force and speed of the plungers movement is determined by the strength of the magnetic flux generated within the coil.

When the supply current is turned "OFF" (de-energized) the electromagnetic field generated previously by the coil collapses and the energy stored in the compressed spring forces the plunger back out to its original rest position. This back and forth movement of the plunger is known as the solenoids "Stroke", in other words the maximum distance the plunger can travel in either an "IN" or an "OUT" direction, for example, 0 – 30mm.

4. 5. 1. The Linear Solenoid

A Linear Solenoid is an electromagnetic device that converts electrical energy into a mechanical pushing or pulling force or motion. The linear solenoid works on the same basic principal as the electromechanical relay and just like relays that will be discussed later.

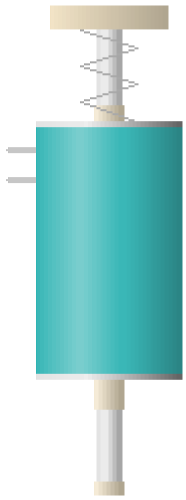


Figure 4-55: Typical linear solenoid

Push-type Linear Solenoid Construction

The basic construction of linear solenoid consists of a cylindrical coil of wire wound around a nonmagnetic hollow form. A stationary iron core is fixed in position at the end of the shaft, and a sliding iron core is attached to the stationary core with a spring. The ferromagnetic actuator or "plunger" is free to move or slide "IN" and "OUT" of the coils body.

In the at-rest (unenergized) state, the plunger is retracted as shown in the figure 4-56 part (a), the solenoid is energized by current through the coil, as shown in the figure 4-56 part (b). The current sets up an electromagnetic field that magnetizes both iron cores. The south pole of the stationary core attracts the north pole of the movable core, which causes it to slide outward, thus extending the plunger and compressing the spring. As long as there is a coil current, the plunger remains extended by the attraction force of the magnetic fields. When the current is cut off, the magnetic fields collapses; and the force of the compressed spring pushes the plunger back in.

This type of solenoid is generally called a Linear Solenoid due to the linear directional movement and action of the plunger. Linear solenoids are available in two basic configurations called a "Pull-type" as it pulls the connected load towards itself when energized, and the "Push-type" that act in the opposite direction pushing the load away from itself when energized.

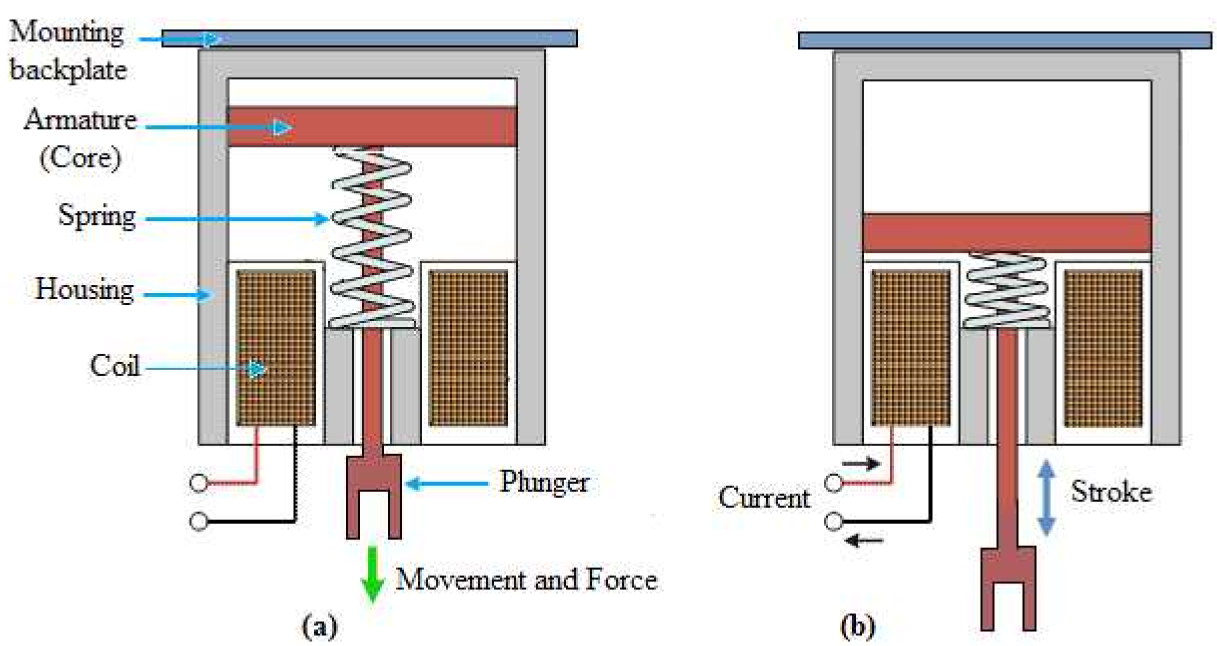


Figure 4-56: Basic linear solenoid construction; (a) Unenergized-plunger retracted, (b) Energized-plunger extended.

Both push and pull types are generally constructed the same with the difference being in the location of the return spring and design of the plunger.

Experiment 4-10: Energizing and de-energizing solenoid

Part lit

No	Item	Quantity
1	12V DC battery 100Ah	1
2	12V solenoid	1
3	Push button NO	1
4	Breadboard	1
5	Connecting wire	50Cm

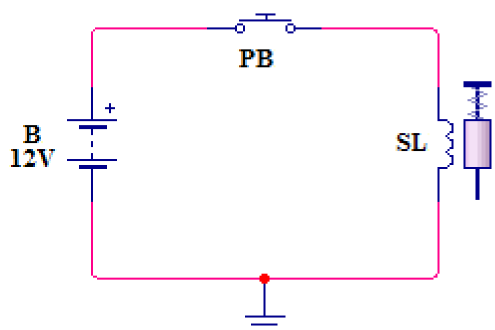
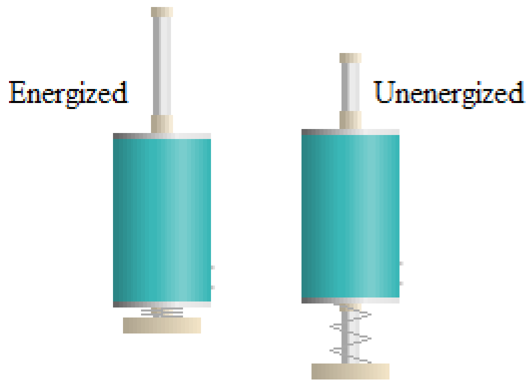


Figure 4-57: Circuit for energizing and de-energizing the solenoid

Procedure

1. Connect the circuit as illustrated in figure 4-58.
2. Press the push button PB to energize the solenoid. As result, the plunger moves outward.
3. Release the push button PB for de-energizing the solenoid SL. At this time, the plunger moves back to its rest point.
4. Example of energized and unenergized solenoid plunger is shown below.



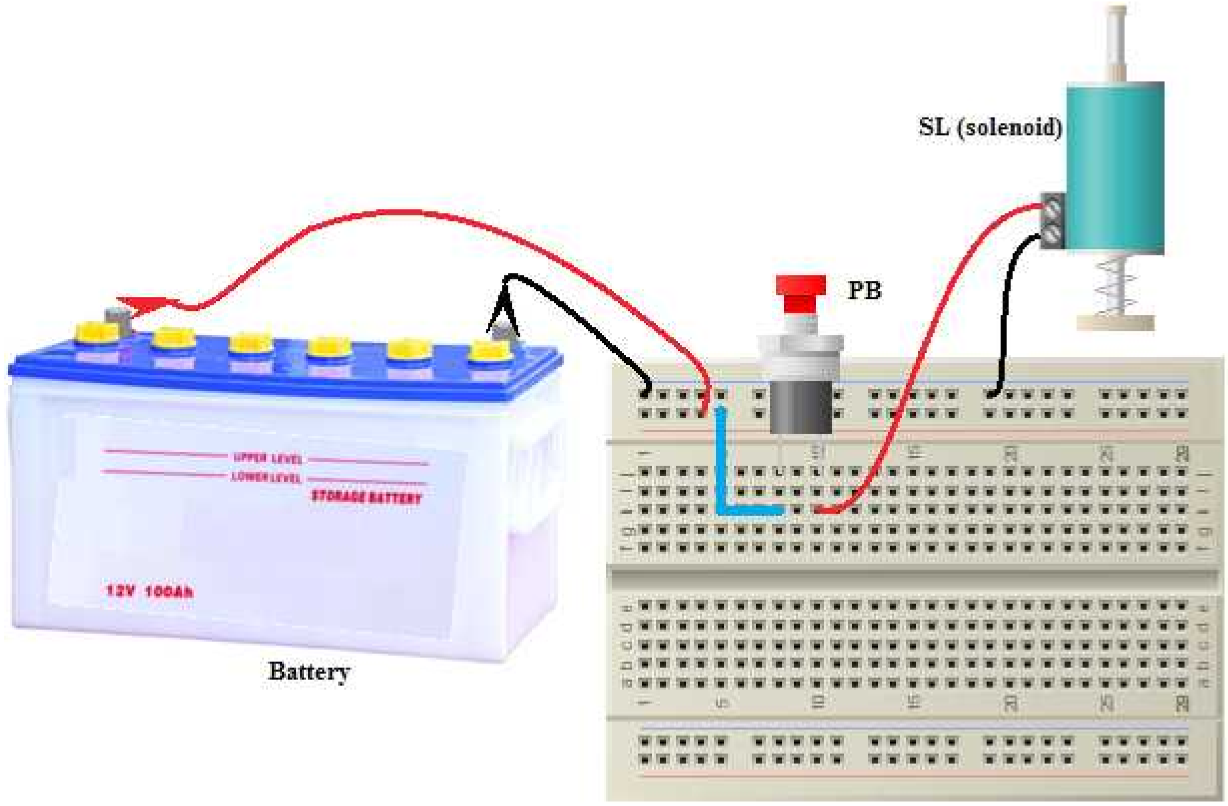


Figure 4-58: Circuit connection for energizing and de-energizing solenoid

Application of Linear Solenoid

Linear solenoids are useful in many applications that require an open or closed (in or out) type motion such as electronically activated door locks, pneumatic or hydraulic control valves, robotics, automotive engine management, irrigation valves to water the garden and even the "Ding-Dong" door bell has one. They are available as open frame, closed frame or sealed tubular types.

4. 5. 2. Rotary Solenoid

The majority electromagnetic solenoids are linear devices producing a linear back and forth force or motion. On the other hand, rotational solenoids also exist which produce an angular or rotary motion from a neutral position in either clockwise, anticlockwise or in both directions (bi-directional).

Rotary solenoids can be used to replace small DC motors or stepper motors where the angular movement is very small with the angle of rotation being the angle moved from the start to the end position.

Commonly available rotary solenoids have movements of 25°, 35°, 45°, 60° and 90° as well as multiple movements to and from a certain angle such as a 2-position self restoring or return to zero rotation, for example 0°-to-90°-to-0°, 3-position self restoring, for example 0° to +45° or 0° to -45° as well as 2-position latching.

Rotary solenoids produce a rotational movement when either energized, de-energized, or a change in the polarity of an electromagnetic field alters the position of a permanent magnet rotor.

Their construction consists of an electrical coil wound around a steel frame with a magnetic disk connected to an output shaft positioned above the coil. When the coil is energized the electromagnetic field generates multiple north and south poles which repel the adjacent permanent magnetic poles of the disk causing it to rotate at an angle determined by the mechanical construction of the rotary solenoid.

Application of Rotary Solenoids

Rotary solenoids are used in vending or gaming machines, valve control, camera shutter with special high speed, low power or variable positioning solenoids with high force or torque are available such as those used in dot matrix printers, typewriters, automatic machines or automotive applications etc..

4. 5. 3. Solenoid Switching

Generally solenoids either linear or rotary operate with the application of a DC voltage, but they can also be used with AC sinusoidal voltages by using full wave bridge rectifiers to rectify the supply which then can be used to switch the DC solenoid. Small DC type solenoids can be easily controlled using Transistor or MOSFET switches and are ideal for use in robotic applications.

However, as with electromechanical relays, linear solenoids are also "inductive" devices so some form of electrical protection is required across the solenoid coil to prevent high back emf (electromotive force) voltages from damaging the semiconductor switching device. In this case the standard "Flywheel Diode" is used, but you could equally use a zener diode or small value varistor.

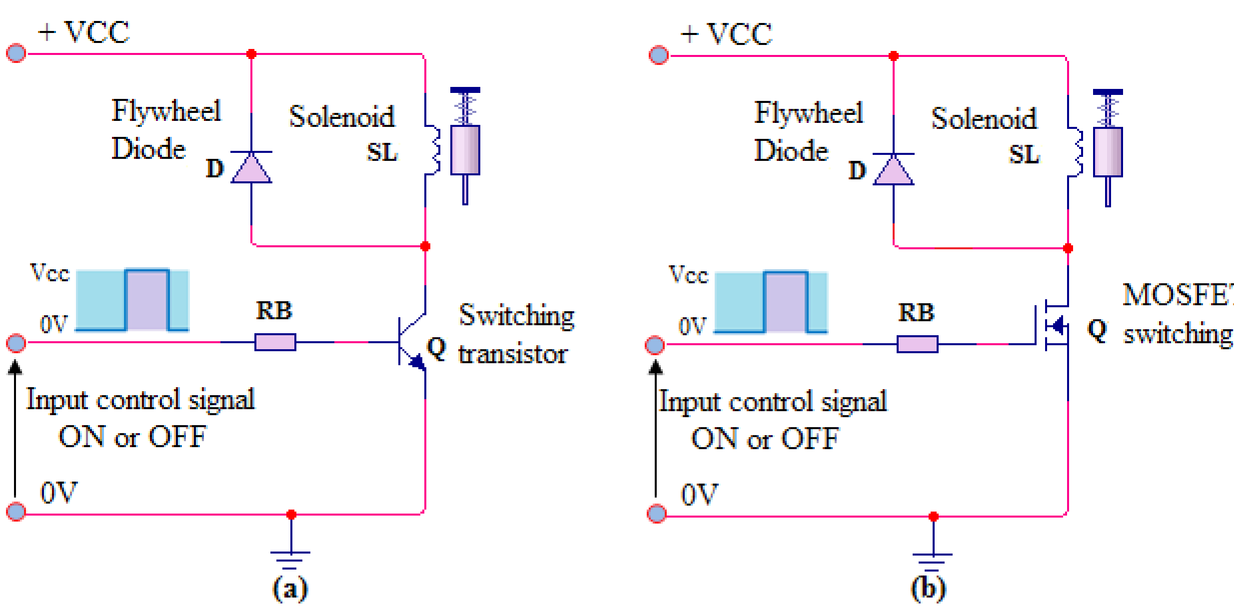


Figure 4-59: Basic circuit for switching the solenoid using transistor; (a) BJT switching, (b) MOSFET switching

4. 5. 4. Reducing energy consumption

One of the main disadvantages of solenoids and especially the linear solenoid is that they are "inductive devices". This means that their solenoid coil converts some of the electrical energy used to operate them into "HEAT". In other words when connected for long periods of time to an electrical supply they get hot, and the longer the time that the power is applied to a solenoid coil, the hotter the coil will become. Also as the coil heats up, its electrical resistance also changes allowing more current to flow increasing its temperature.

With a continuous voltage input applied to the coil, the solenoids coil does not have the opportunity to cool down because the input power is always on. In order to reduce this self generated heating effect it is necessary to reduce either the amount of time the coil is energized or reduce the amount of current flowing through it.

1. Holding resistor

One method of consuming less current is to apply a suitable high enough voltage to the solenoid coil so as to provide the necessary electromagnetic field to operate and seat the plunger but then once activated to reduce the coils supply voltage to a level sufficient to maintain the plunger in its seated or latched position. One way of achieving this is to connect a suitable "holding" resistor in series with the solenoids coil.

From circuit arrangement shown in figure 4-60, the switch contact is closed shorting out the resistance and passing the full supply current directly to the solenoid coils windings. Once energized the contacts which can be mechanically connected to the solenoids plunger action open connecting the holding resistor, RH in series with the solenoids coil. This effectively connects the resistor in series with the coil.

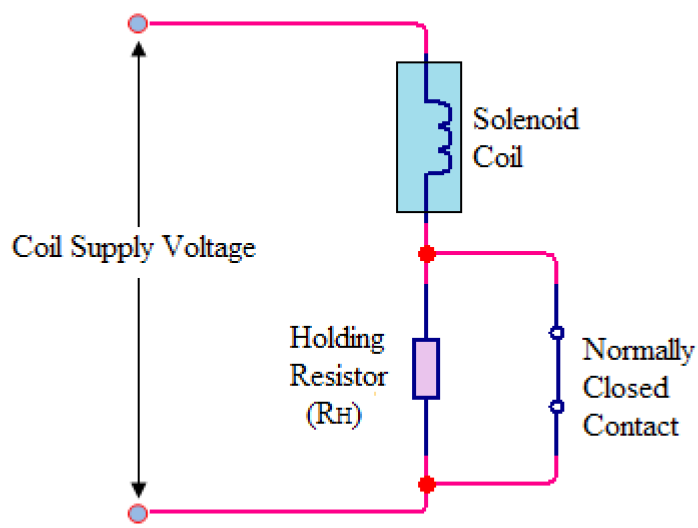
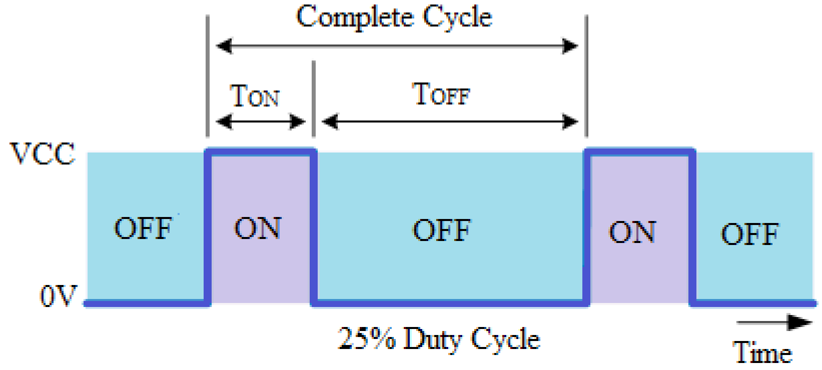


Figure 4-60: Reducing energy consumption of solenoid circuit principles

By using this method, the solenoid can be connected to its voltage supply indefinitely (continuous duty cycle) as the power consumed by the coil and the heat generated is greatly reduced, which can be up to 85 to 90% using a suitable power resistor. However, the power consumed by the resistor will also generate a certain amount of heat, I^2R (Ohm's Law) and this also needs to be taken into account.

2. Solenoid Duty Cycle

Another more practical way of reducing the heat generated by the solenoids coil is to use an "intermittent duty cycle". An intermittent duty cycle means that the coil is repeatedly switched "ON" and "OFF" at a suitable frequency so as to activate the plunger mechanism but not allow it to de-energize during the OFF period of the waveform. Intermittent duty cycle switching is a very effective way to reduce the total power consumed by the coil.



The Duty Cycle of a solenoid is the portion of the "ON" time that a solenoid is energized and is the ratio of the "ON" time to the total "ON" and "OFF" time for one complete cycle of operation. In other words, the cycle time equals the switched-ON time plus the switched-OFF time. Duty cycle is expressed as a percentage, for example:

Duty Cycle = ("ON" time/ "ON" time + "OFF" time)*100%

Then if a solenoid is switched "ON" or energized for 20 seconds and then switched "OFF" for 60 seconds before being re-energized again, one complete cycle, the total "ON/OFF" cycle time would be 80 seconds, (20+60) so the solenoids duty cycle would be calculated as 20/80 seconds or 25%. This means that you can determine the solenoids maximum switch-ON time if you know the values of duty cycle and switch-OFF time.

For example, the switch-OFF time equals 4 seconds; duty cycle equals 60%; therefore switch-ON time equals 6 seconds. A solenoid with a rated Duty Cycle of 100% means that it has a continuous voltage rating and can therefore be left "ON" or continuously energized without overheating or damage.

4. 6. The Electromechanical Relay

The term Relay generally refers to a device that provides an electrical connection between two or more points in response to the application of a control signal. The most common and widely used type of electrical relay is the electromechanical relay or EMR.

The most fundamental control of any equipment is the ability to turn it "ON" and "OFF". The easiest way to do this is using switches to interrupt the electrical supply. Although switches can be used to control something, they have their disadvantages. The biggest one is that they have to be manually (physically) turned "ON" or "OFF". Also, they are relatively large, slow and only switch small electrical currents.

Electrical Relays however, are basically electrically operated switches that come in many shapes, sizes and power ratings suitable for all types of applications. Relays can also have single or multiple contacts within a single package with the larger power relays used for mains voltage or high current switching applications being called "Contactors".

Figure 4-61: Typical electromechanical relay

In this tutorial about electrical relays we are just concerned with the fundamental operating principles of "light duty" electromechanical relays that we can use in motor control or robotic circuits. Such relays are used in general electrical and electronic control or switching circuits either mounted directly onto PCB boards or connected free standing and in which the load currents are normally fractions of an ampere up to 20 amperes. The relay circuits are common in Electronics applications.

Working Principle of Electromechanical Relays

As their name implies, electromechanical relays are electro-magnetic devices that convert a magnetic flux generated by the application of a low voltage electrical control signal either AC or DC across the relay terminals, into a pulling mechanical force which operates the electrical contacts within the relay. The most common form of electromechanical relay consists of an energizing coil called the "primary circuit" wound around a permeable iron core.

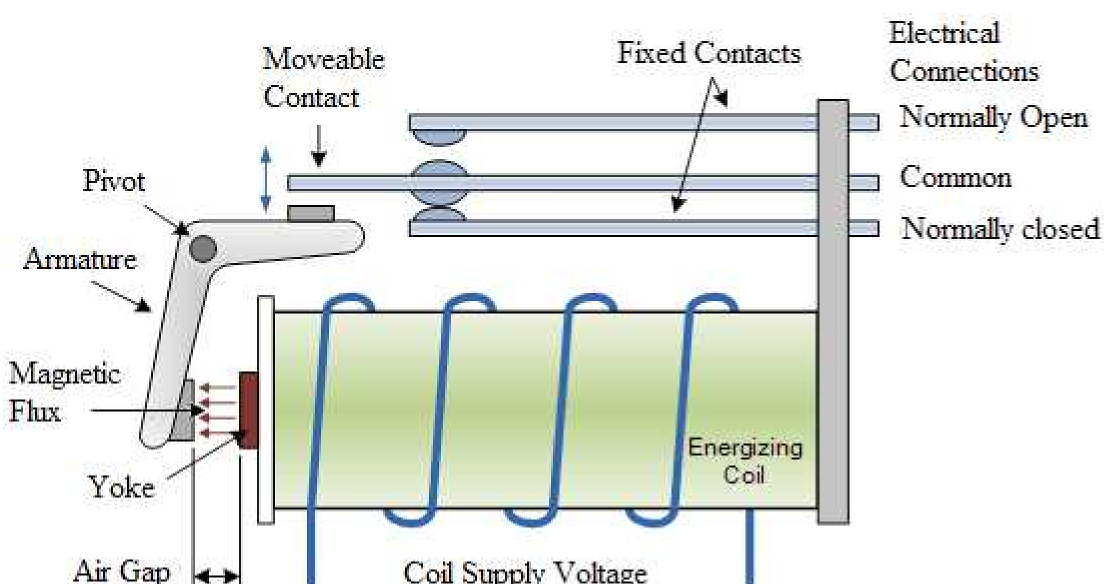


Figure 4-62: Electromechanical Relay construction

This iron core has both a fixed portion called the yoke, and a moveable spring loaded part called the armature, that completes the magnetic field circuit by closing the air gap between the fixed electrical coil and the moveable armature. The armature is hinged or pivoted allowing it to freely move within the generated magnetic field closing the electrical contacts that are attached to it. Connected between the yoke and armature is normally a spring (or springs) for the return stroke to "reset" the contacts back to their initial rest position when the relay coil is in the "de-energized" condition, i.e. turned "OFF".

In the relay of figure 4-62, we have two sets of electrically conductive contacts. Relays may be "Normally Open", or "Normally Closed". One pair of contacts is classed as Normally Open, (NO) or make contacts and another set which is classed as Normally Closed, (NC) or break contacts. In the normally open position, the contacts are closed only when the field current is "ON" and the switch contacts are pulled towards the inductive coil.

In the normally closed position, the contacts are permanently closed when the field current is "OFF" as the switch contacts return to their normal position. These terms Normally Open, Normally Closed or Make and Break Contacts refer to the state of the electrical contacts when the relay coil is "de-energized", i.e. no supply voltage connected to the inductive coil. An example of this arrangement is given in figure 4-63.

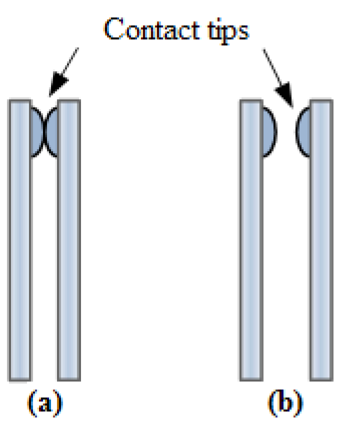


Figure 4-63: Relay contact tips; (a) Normally Closed contacts (NC), (b) Normally Open contacts (NO)

The relays contacts are electrically conductive pieces of metal which touch together completing a circuit and allow the circuit current to flow, just like a switch. When the contacts are open the resistance between the contacts is very high in the Mega-Ohms, producing an open circuit condition and no circuit current flows.

When the contacts are closed the contact resistance should be zero, a short circuit, but this is not always the case. All relay contacts have a certain amount of "contact resistance" when they are closed and this is called the "On-Resistance", similar to field effect transistor FET's. With a new relay and contacts this ON-resistance will be very small, generally less than 0.2 Ω because the tips are new and clean, but over time the tip resistance will increase.

With a new relay and contacts the ON-resistance will be very small, generally less than 0.2 Ω because the tips are new and clean, but over time the tip resistance will increase. For example, if the contacts are passing a load current of say 10A, then the voltage drop across the contacts using Ohms Law is 0.2 x 10 = 2 volts, which if the supply voltage is say 12 volts then the load voltage will be only 10 volts (12 – 2).

As the contact tips begin to wear, and if they are not properly protected from high inductive or capacitive loads, they will start to show signs of arcing damage as the circuit current still wants to flow as the contacts begin to open when the relay coil is de-energized.

This arcing or sparking across the contacts will cause the contact resistance of the tips to increase further as the contact tips become damaged. If allowed to continue the contact tips may become so burnt and damaged to the point where they are physically closed but do not pass any or very little current.

If this arcing damage becomes too severe the contacts will eventually "weld" together producing a short circuit condition and possible damage to the circuit they are controlling. If now the contact resistance has increased due to arcing to say 1 Ω the volt drop across the contacts for the same load current increases to 1 x 10 = 10 volts dc. This high voltage drop across the contacts may be unacceptable for the load circuit especially if operating at 12 or even 24 volts, then the faulty relay will have to be replaced.

4. 6. 1. Extending the Life of Relay Tips

Relay manufacturers data sheets give maximum contact ratings for resistive DC loads only and this rating is greatly reduced for either AC loads or highly inductive or capacitive loads. In order to achieve long life and high reliability when switching AC currents with inductive or capacitive loads some form of arc suppression or filtering is required across the relay contacts.

Extending the life of relay tips by reducing the amount of arcing generated as they open is achieved by connecting a Resistor-Capacitor network called an RC Snubber network electrically in parallel with an electrical relay contact tips. The voltage peak, which occurs at the instant the contacts open, will be safely short circuited by the RC network, thus suppressing any arc generated at the contact tips.

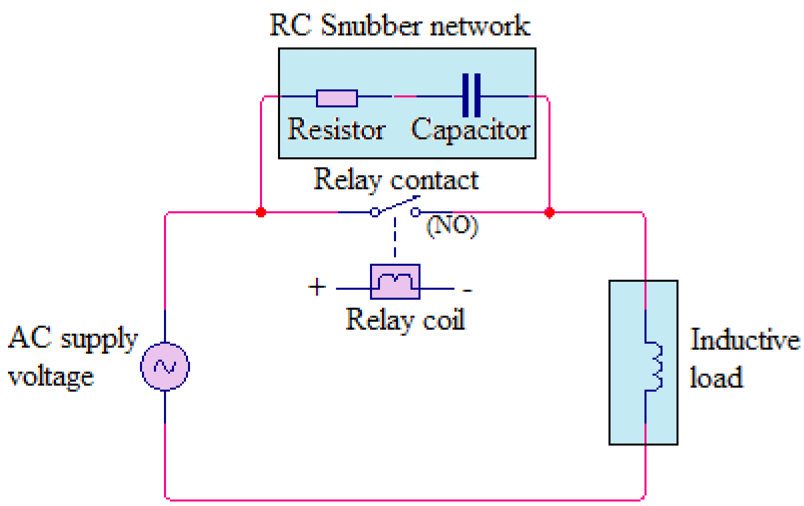


Figure 4-64: Electrical Relay Snubber Circuit

Electrical Relay Contacts Types

As well as the standard descriptions of Normally Open, (NO) and Normally Closed, (NC) used to describe how the relays contacts are connected, relay contact arrangements can also be classed by their actions. Electrical relays can be made up of one or more individual switch contacts with each "contact" being referred to as a "pole". Each one of these contacts or poles can be connected or "thrown" together by energizing the relays coil and this gives rise to the description of the contact types as being:

- SPST – Single Pole Single Throw
- SPDT – Single Pole Double Throw
- DPST – Double Pole Single Throw
- DPDT – Double Pole Double Throw

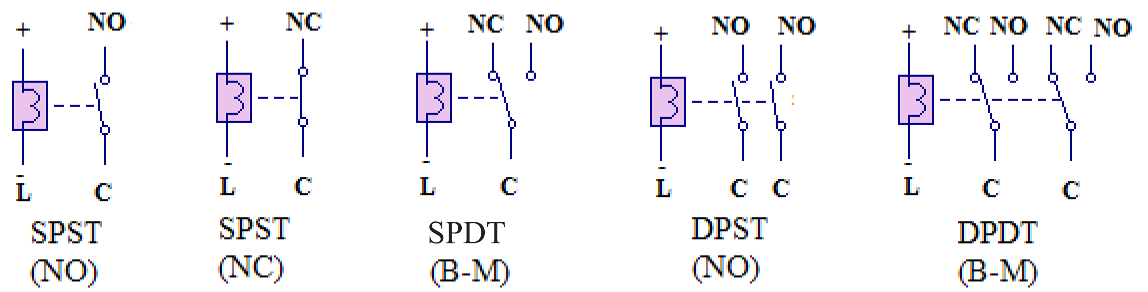


Figure 4-65: Electrical Relay contact configurations

Where

- C is the Common terminal
- NO is the Normally Open contact
- NC is the Normally Closed contact
- L is the relay coil

With the action of the contacts being described as "Make" (M) or "Break" (B). Then a simple relay with one set of contacts as shown above can have a contact description of:

"Single Pole Double Throw – (Break before Make)", or SPDT – (B-M)

Examples of some of the more common diagrams used for electrical relay contact types to identify relays in circuit or schematic diagrams is given in figure 4-65 but there are many more possible configurations.

4. 6. 2. Relay Configuration, Installation Tips and Switching

It is not advisable at all to connect relay contacts in parallel to handle higher load currents. For example, never attempt to supply a 10A load with two relays in parallel that have 5A contact ratings each, as the mechanically operated relay contacts never close or open at exactly the same instant of time. The result is that one relay contact will always be overloaded even for a brief instant in time resulting in premature failure of the relay over time.

Also, while electrical relays can be used to allow low power electronic or computer type circuits to switch relatively high currents or voltages both "ON" or "OFF"; never mix different load voltages through adjacent contacts within the same relay such as for example, high voltage AC (220V) and low voltage DC (12V), always use separate relays for safety.

One of the more important parts of any electrical relay is its coil. This converts electrical current into an electromagnetic flux which is used to mechanically operate the relay contacts. The main problem with relay coils is that they are "highly inductive loads" as they are made from coils of wire. Any coil of wire has an impedance value made up of resistance (R) and inductance (L) in series (LR Series Circuit).

As the current flows through the coil a self induced magnetic field is generated around it. When the current in the coil is turned "OFF", a large back electromotive force (emf) voltage is produced as the magnetic flux collapses within the coil, as we have seen in transformer theory. This induced reverse voltage value may be very high in comparison to the switching voltage, and may damage any semiconductor device such as a transistor, FET or micro-controller used to operate the relay coil. One way of preventing damage to the transistor or any switching semiconductor device, is to connect a reverse biased diode across the relay coil.

When the current flowing through the coil is switched "OFF", an induced back emf is generated as the magnetic flux collapses in the coil. This reverse voltage forward biases the diode which conducts and dissipates the stored energy preventing any damage to the semiconductor transistor.

When used in this type of application the diode is generally known as a Flywheel Diode, Free-wheeling Diode and even Fly-back Diode, but they all mean the same thing. Other types of inductive loads which require a flywheel diode for protection are solenoids, motors and inductive coils also discussed previously.

As well as using flywheel Diodes for protection of semiconductor components, other devices used for protection include RC Snubber Networks, Metal Oxide Varistors (MOV) and Zener Diodes.

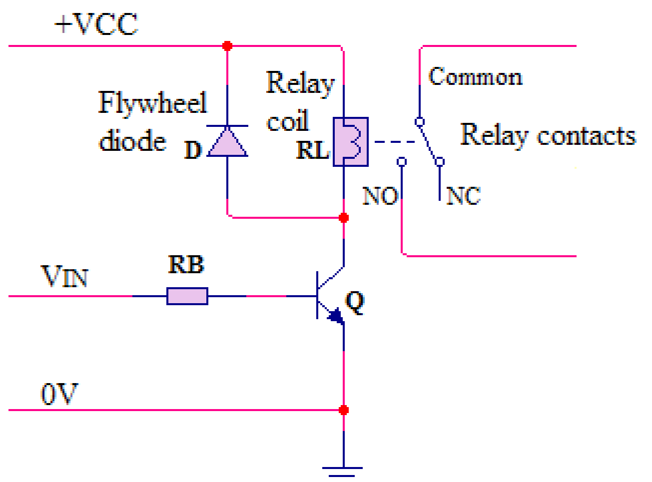
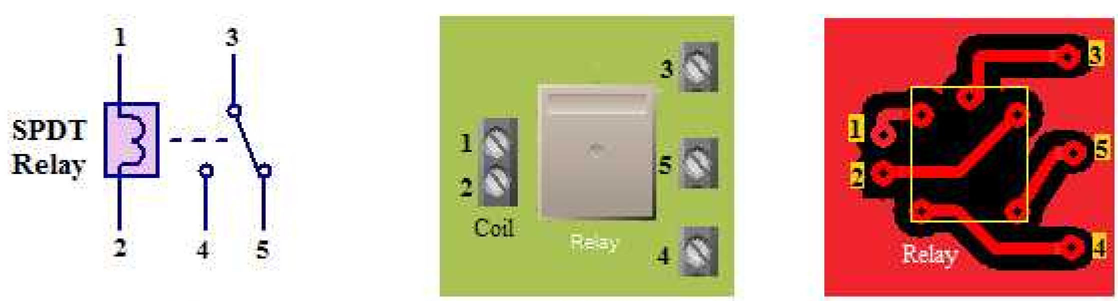


Figure 4-66: Circuit configuration with a flywheel diode used to protect switching device

Experiment 4-11: Testing and identifying relay terminals by using multimeter

Part list

No	Item	Quantity
1	Multimeter with continuity settings and Ohm's range	1
2	Relay 9V, SPDT	1
3	9V battery	1



(a) (b) (c)

Figure 4-67: SPDT relay. (a) Schematic symbol, (b) SPDT relay on PCB top view, (c) SPDT relay on PCB solder side.

Procedure

1. Set multimeter in continuity settings or Ohm's range

2. Hold one multimeter's lead to one relay terminal, i.e. terminal 3 and touch other lead to other relay's terminals, i.e. terminal 4 and 5.
3. Between terminal 3 and terminal 4 there is no continuity, means no beep sound from multimeter and once set in ohm's range, the ohmmeter reading is infinity, i.e. OL.

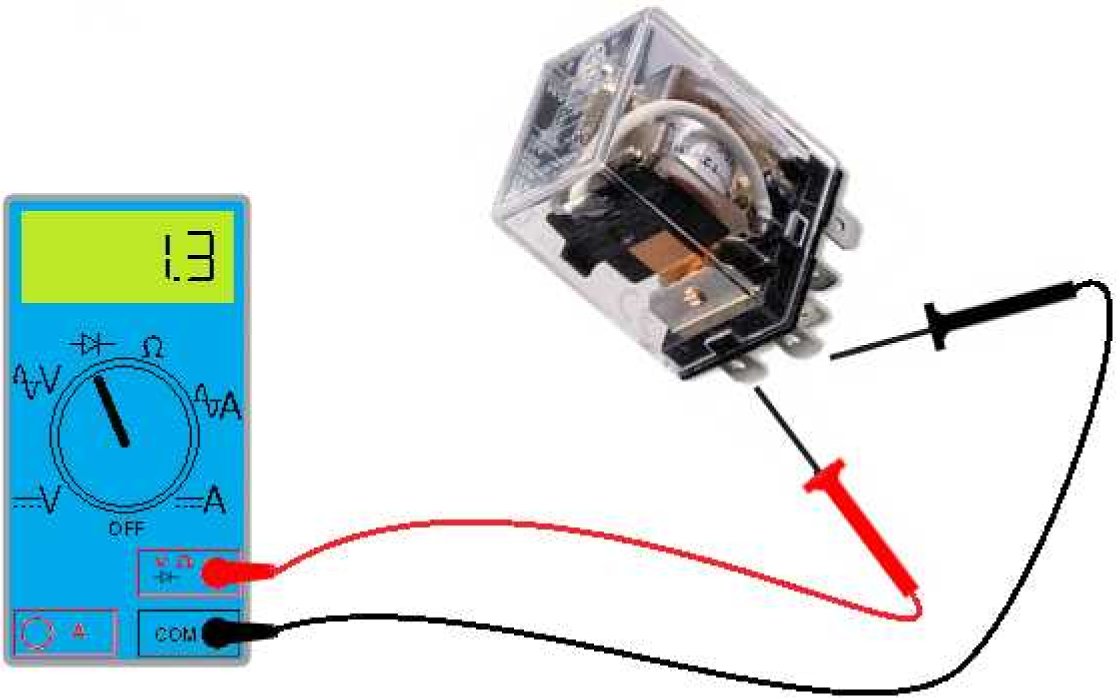


Figure 4-68: Testing and identifying relay terminals

4. Between terminal 3 and terminal 5 there is continuity justified by a beep sound from multimeter. For ohm's settings, the ohmmeter reading is 0 Ω.
5. As result, contact of terminal 3 is common, terminal 4 is normally open contact (NO), and terminal 5 is normally closed contact (NC). Terminal 3, 4 and 5 are reserved for high power commutation.
6. Once more, hold one multimter's lead to terminal 1 and other lead to terminal 2. If the multimeter is in continuity settings, the reading is about 20 to 100 range. For the value less than 30 you may hear the sound from multimeter. When you are using ohm's range lower scale such as 200 Ω, the multimeter will read also the value ranging in 20Ω up to 100Ω. This implies that terminal 1 and terminal 2 is coil contact.
7. Supply the coil by connecting the battery on terminal 1 and 2, you may hear movable contact. Then set in ohm's range and touch the multimeter's lead to terminal 3 and 4; the ohmmeter reading is 0 Ω since the relay is now energized and closed contact between terminal 3 and 4 while terminal 3 and 5 is open and reading is infinity.
8. Therefore, if the multimter is in continuity settings, when the reading gives a value less than 200 but not zero these two contacts are the coil terminals. When the multimter leads are touching on the three terminals and the reading gives a very high value or infinity or OL, that contact is NO; where the reading gives 0 Ω, this is NC contact, the unchanged terminal among these three is common contact. Care has to be taken since burnt coil also will read a very high value or infinity (OL) and at this time the relay has to be replaced.

Sensors and Transducers

Chapter five

Objectives

After completing this chapter, you should

- Be able to describe basic electronic sensors
- Have understanding on light sensors and their applications
- Have understanding on temperature sensors and their applications
- Have understanding on position sensors and their applications

chap 5

Further reading
Study aids for this chapter are available at
• Enss, Christian (Editor) (2005). *Cryogenic Particle Detection*. Springer, Topics in applied physics 99.

A *sensor* is a device that carries out an input function to perform any useful task or function. It senses a physical change in some characteristic that changes in response to some excitation and converts that into an electrical signal. Devices which perform an *output* function are generally called *actuators* and are used to control some external device, for example movement or light.

A *transducer* is a device which converts one physical quantity into another. The word transducer is the collective term used for both sensors which can be used to sense a wide range of different energy forms such as movement, electrical signals, radiant energy, thermal or magnetic energy etc, and actuators which can be used to switch voltages or currents.

There are many different types of sensors and transducers, both analog and digital and input and output available to choose from. The type of input or output transducer being used, really depends upon the type of signal or process being *Sensed* or *Controlled*.

This chapter presents the most commonly used sensors and transducers to make electronic circuit project.

5. 1. Light Sensors

A Light Sensor generates an output signal indicating the intensity of light by measuring the radiant energy that exists in a very narrow range of frequencies basically called “light”, and which ranges in frequency from “Infra-red” to “Visible” up to “Ultraviolet” light spectrum.

The Light Sensor is a passive device that converts the “light energy” whether visible or in the infra-red parts of the spectrum into an electrical signal output. Light sensors are more commonly known as “Photoelectric Devices” or “Photo Sensors” because they convert light energy (photons) into electricity (electrons).

Photoelectric devices can be grouped into two main categories, those which generate electricity when illuminated, such as *Photo-voltaic* or *Photo-emissive* etc., and those which change their electrical properties in some way such as *Photo-resistors* or *Photo-conductors*. This leads us to the following classification of devices.

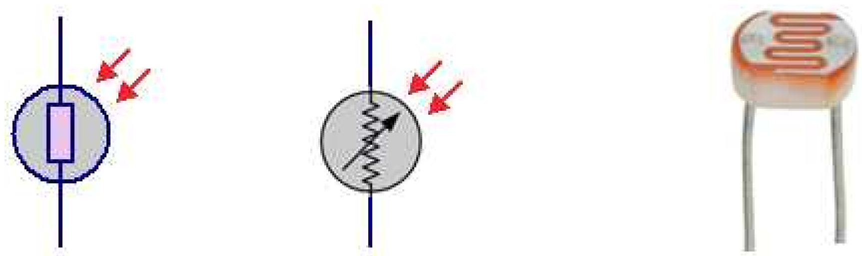
- Photo-emissive Cells – These are photodevices which release free electrons from a light sensitive material such as caesium when struck by a photon of sufficient energy. The amount of energy the photons have depends on the frequency of the light and the higher the frequency, the more energy the photons have converting light energy into electrical energy.
- Photo-conductive Cells – These photodevices vary their electrical resistance when subjected to the light. Photoconductivity results from light hitting a semiconductor material which controls the current flow through it. Thus, more light increase the current for a given applied voltage. The most common photoconductive material is Cadmium Sulphide used in LDR photocells.
- Photo-voltaic Cells – These photodevices generate an emf (electromotive force) in proportion to the radiant light energy received and is similar in effect to photoconductivity. Light energy falls on to two semiconductor materials sandwiched together creating a voltage of approximately 0.5V. The most common photovoltaic material is Selenium used in solar cells.
- Photo-junction Devices – These photodevices are mainly true semiconductor devices such as the photodiode or phototransistor which use light to control the flow of electrons and holes across their PN-junction. Photojunction devices are specifically designed for detector application and light penetration with their spectral response tuned to the wavelength of incident light.

5. 1. 1. The Photoconductive Cell

A Photoconductive light sensor does not produce electricity but simply changes its physical properties when subjected to the light energy. The most common type of photoconductive device is the *Photoresistor* which changes its electrical resistance in response to changes in the light intensity.

Photoresistors are semiconductor devices that use light energy to control the flow of electrons, and hence the current flowing through them. The commonly used *Photoconductive Cell* is called the Light Dependent Resistor or LDR.

A photoresistor or light-dependent resistor (LDR) or photocell is a light-controlled variable resistor. The Light Dependent Resistor has a resistance that varies according to the amount of light falling on its surface. The resistance of a photoresistor decreases with increasing incident light intensity vice versa. It changes its electrical resistance from several thousand Ohms in the dark to only a few hundred Ohms when light falls upon it by creating hole-electron pairs in the material; in other words, it exhibits photoconductivity.



(a) (b)

Figure 5-1: Light Dependent Resistor; (a) American and European symbol, (b) Typical LDR

Experiment 5-1: Light dependent resistor (LDR) under light condition resistance measuring

With LDR resistor, source of light and the ohmmeter, verify that the resistance of LDR varies with respect to the change in the light intensity falling on the LDR. With this by obstaculing and allowing the light to reach the LDR resistor and observing the reading on ohmmeter.

Part list

No	Items	Specifications	Quantity
1	Light source	Torch	1
2	LDR	ORP12	1
3	Ohmmeter	Digital multimeter	1

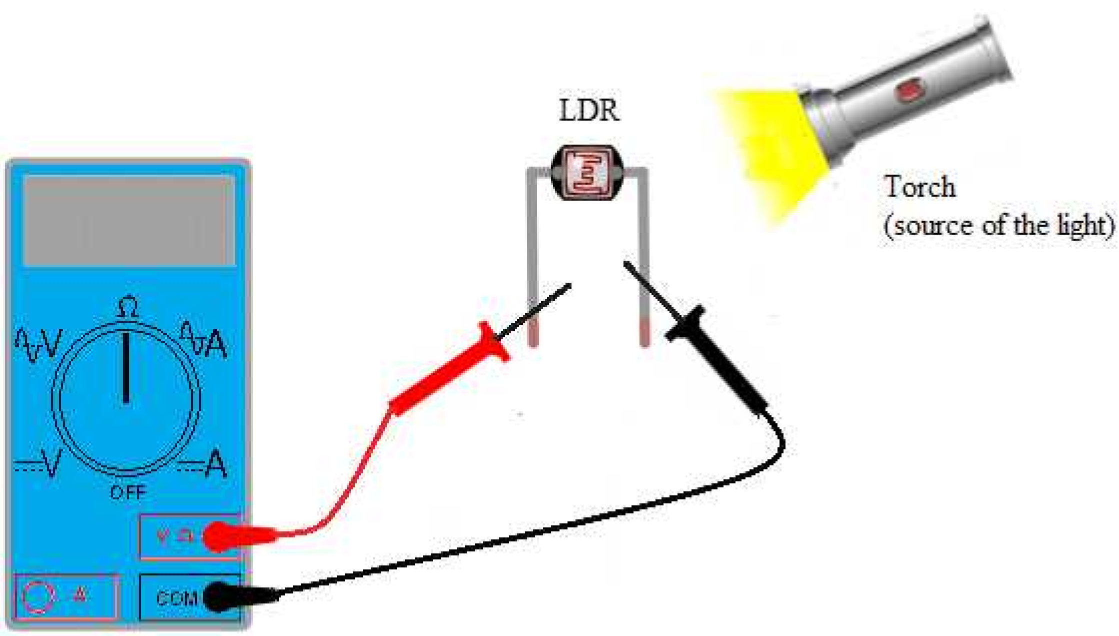


Figure 5-2: Measuring the resistance of LDR under the source of light condition

Procedure

1. Set the multimeter in ohm's range.
2. Hold the multimeter's probes against LDR's legs (the red probe in VΩ plug and the black probe in COM of the multimeter). Polarity does not matter when measuring resistance.
3. Switch on the torch and let the light intensity fall on the LDR.
4. Observe the reading of the multimeter and change the ohm's scale.
5. Obstacle the light and change the scale to Kilo-ohm or Mega-ohm and observe the ohm-meter reading.
6. It may be noted that as the light intensity falling on LDR increases, its resistance also increases.

The net effect is an improvement in its conductivity with a decrease in resistance for an increase in illumination. Also, photoresistive cells have a long response time requiring many seconds to respond to a change in the light intensity.

Cadmium sulphide is used in the manufacture of photoconductive cells because its spectral response curve closely matches that of the human eye and can even be controlled using a simple torch as a light source. Typically then, it has a peak sensitivity wavelength (λp) of about 560nm to 600nm in the visible spectral range.

The most commonly used photoresistive light sensor is the ORP12 Cadmium Sulphide photoconductive cell. This light dependent resistor has a spectral response of about 610nm in the yellow to orange region of light. The resistance of the cell when unilluminated (dark resistance) is very high at about 10 MΩ which falls to about 100 Ω when fully illuminated (lit resistance).

To increase the dark resistance and therefore reduce the dark current, the resistive path forms a zigzag pattern across the ceramic substrate. The CdS photocell is a very low cost device often used in auto dimming, darkness or twilight detection for turning the street lights "ON" and "OFF", and for photographic exposure meter type applications.

Photoresistor applications

- The photoresistor or light dependent resistor is used in various electronic circuit designs because of its low cost, simple structure and rugged features. While it may not have some of the features of the photo-diode and photo-transistor, it is ideal for many applications. As a result the photo-resistor is widely used in circuits such as photographic meters, flame or smoke detectors, thief alarms, card readers, controls for street lighting, clock radios, alarm devices, outdoor clocks, solar street lamps and solar road studs, etc..

The properties of photoresistors can vary quite widely dependent upon the type of material used. Some have very long time constants, for example. It is therefore necessary to carefully choose the type of photoresistor for any given circuit or application.

Connecting a light dependant resistor in series with a standard resistor like in figure 5-3 across a single DC supply voltage has one major advantage; a different voltage will appear at their junction for different light levels.

The amount of voltage drop across series resistor, R_2 is determined by the resistive value of the light dependant resistor, LDR (R_1). This ability to generate different voltages produces a very handy circuit called a "Potential Divider" or Voltage Divider Network. The current through a series circuit is common and as the LDR changes its resistive value due to the light intensity, the voltage present at V_{out} will be determined by the voltage divider formula. An LDR's resistance can vary from about 100 Ω in the sun light, to over 10 MΩ in absolute darkness with this variation of resistance being converted into a voltage variation at V_{out}.

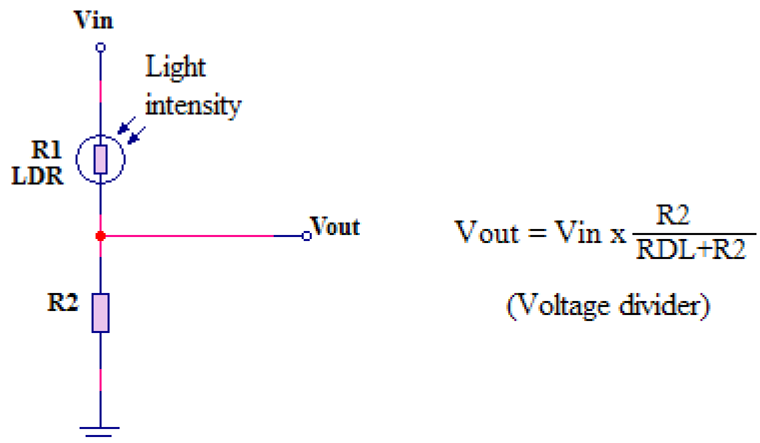


Figure 5-3: LDR in voltage divider network

- One simple application of a Light Dependent Resistor, is as a light sensitive switch as shown in figure 5-4.

chap 5

This basic light sensor circuit is of a relay output light activated switch. A potential divider circuit is formed between the photoresistor, LDR and the resistor R_1. When no light is present, i.e. in darkness, the resistance of the LDR is very high in the Megaohms (MΩ) range so zero base bias is applied to the transistor Q1 and the relay is de-energised or "OFF".

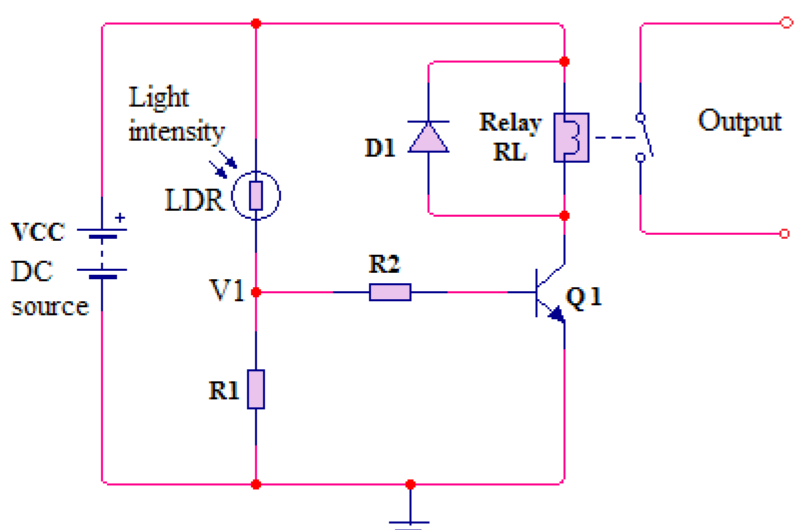


Figure 5-4: LDR Switch

As the light level increases the resistance of the LDR starts to decrease causing the base bias voltage at V_1 to rise. At some point determined by the potential divider network formed with resistor R_1, the base bias voltage is high enough to turn the transistor Q1 "ON" and thus activate the relay which in turn is used to control some external circuitry. As the light level falls back to darkness again the resistance of the LDR increases causing the base voltage of the transistor to decrease, turning the transistor and relay "OFF" at a fixed light level determined again by the potential divider network.

By replacing the fixed resistor R_1 with a potentiometer VR_1, the point at which the relay turns "ON" or "OFF" can be pre-set to a particular light level.

This type of simple circuit shown in figure 5-4 has a fairly low sensitivity and its switching point may not be consistent due to variations in either temperature or the supply voltage. A more sensitive precision light activated circuit can be easily made by incorporating the LDR into a "Wheatstone Bridge" arrangement and replacing the transistor with an Operational Amplifier as shown in figure 5-5.

- Light Level Sensing

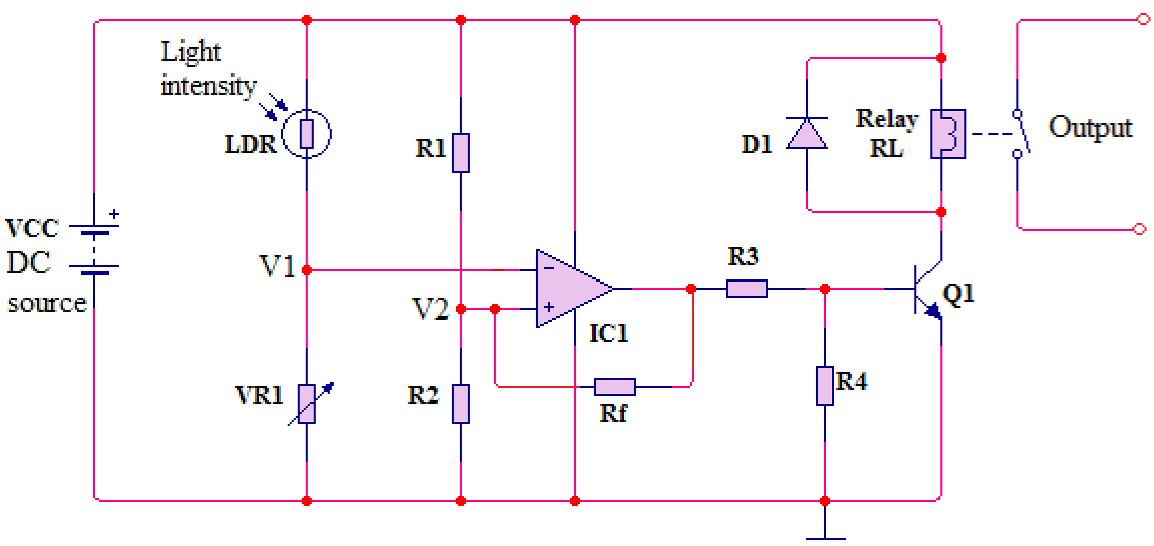


Figure 5-5: Light Level Sensing Circuit

In this basic dark sensing circuit of figure 5-5, the light dependent resistor LDR_1 and the potentiometer VR_1 form one adjustable arm of a simple resistance bridge network, also known commonly as a *Wheatstone bridge*, while the two fixed resistors R_1 and R_2 form the other arm. Both sides of the bridge form potential divider networks across the supply voltage whose outputs V_1 and V_2 are connected to the non-inverting and inverting voltage inputs respectively of the operational amplifier.

The operational amplifier is configured as a Differential Amplifier also known as a voltage comparator with feedback whose output voltage condition is determined by the difference between the two input signals or voltages, V_1 and V_2. The resistor combination R_1 and R_2 form a fixed voltage reference at input V_2, set by the ratio of the two resistors. The LDR and VR_1 combination provides a variable voltage input V_1 proportional to the light level being detected by the photoresistor.

As with the previous circuit the output from the operational amplifier is used to control a relay, which is protected by a flywheel diode, D_1. When the light level sensed by the LDR and its output voltage falls below the reference voltage set at V_2 the output from the op-amp changes state activating the relay and switching the connected load.

Likewise as the light level increases the output will switch back turning "OFF" the relay. The hysteresis of the two switching points is set by the feedback resistor R_f which can be chosen to give any suitable voltage gain of the amplifier.

The operation of this type of light sensor circuit can also be reversed to switch the relay "ON" when the light level exceeds the reference voltage level and vice versa by reversing the positions of the light sensor LDR and the potentiometer VR_1. The potentiometer can be used to "pre-set" the switching point of the differential amplifier to any particular light level making it ideal as a simple light sensor project circuit.

- The LDR is used as internal components of a photoelectric control for a typical streetlight. The photoresistor is facing rightwards, and controls whether current flows through the heater which opens the main power contacts. At night, the heater cools, closing the power contacts, energizing the street light.

5. 2. Photo-junction Devices

Photo-junction Devices are basically PN-Junction light sensors or detectors made from silicon semiconductor PN-junctions which are sensitive to light and which can detect both visible light and infra-red light levels. Photo-junction devices are specifically made for sensing light and this class of photoelectric light sensors includes the *Photodiode* and the *Phototransistor*.

5. 2. 1. Photodiode

A photodiode consists of an active P-N junction which is operated in reverse bias. When light falls on the junction, a reverse current flows and is proportional to the illuminance. The current is generated when photons are absorbed in the photodiode junction. In other words, a photodiode is a semiconductor device that converts light into current.

Recall that when reverse-biased, a rectifier diode has a very small leakage current. The same is true for the photo diode. The reverse biased current is produced by thermally generated electron hole pairs in the depletion region, which are swept across the junction by the electric field created by the reverse voltage.

(a) (b)

Figure 5-6: Photo-diode; (a) Photodiode symbol, (b) Typical photodiodes

The photodiode has a small parent window that allows light to strike the PN junction.

In a rectifier diode, the reverse current increases with increase in the number of electron hole pairs. In photodiode, the reverse current increases with the light intensity at the exposed PN junction. When there is no incident light, the reverse current (I_λ) is almost negligible and is called the *dark current*. An increase in the amount of the light intensity, expressed as irradiance (mW/cm^2), produces an increase in reverse current as shown by the graph in figure 5-7.

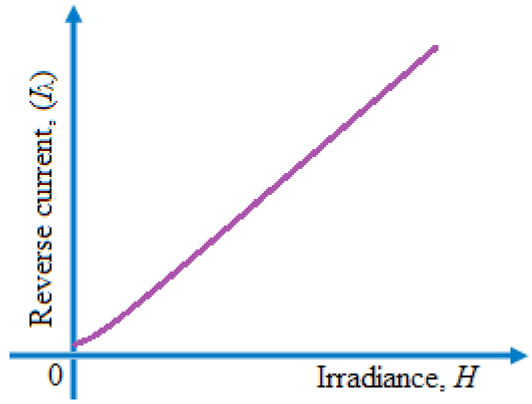


Figure 5-7: General graph of reverse current versus irradiance for a photodiode

Thus, the photodiodes current is directly proportional to light intensity falling onto the PN-junction. One main advantage of photodiodes when used as light sensors is their fast response to changes in the light levels, but one disadvantage of this type of photodevice is the relatively small current flow even when fully lit.

The following circuit, figure 5-8, shows a photo-current-to-voltage convertor circuit using an operational amplifier as the amplifying device.

chap 5

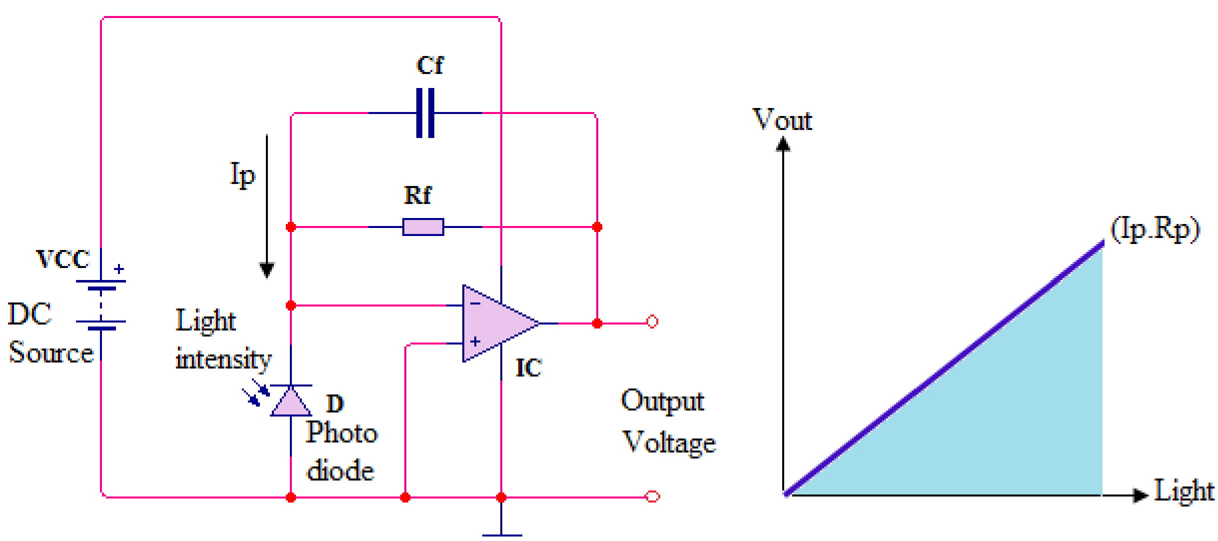


Figure 5-8: Photo-diode Amplifier Circuit

The output voltage (V_{out}) is given as $V_{out} = Ip \times R_f$ and is proportional to the light intensity characteristics of the photodiode.

This type of circuit also utilizes the characteristics of an operational amplifier with two input terminals at about zero voltage to operate the photodiode without bias. This zero-bias op-amp configuration gives a high impedance loading to the photodiode resulting in less influence by dark current and a wider linear range of the photocurrent relative to the radiant light intensity.

Capacitor C_f is used to prevent oscillation or gain peaking and to set the output bandwidth ($1/2\pi RC$).

A photodiode operates in two different modes: Photovoltaic mode and photoconductive mode.

When used in zero bias or *photovoltaic mode*, the illuminated photodiode generates a voltage which can be measured. The diode becomes forward biased and "dark current" begins to flow across the junction in the direction opposite to the photocurrent. This mode is responsible for the photovoltaic effect, which is the basis for solar cells. In fact, a traditional solar cell is just a large area photodiode.

Photodiode operating in *photoconductive mode* is often reverse biased (with the cathode driven positive with respect to the anode). This reduces the response time because the additional reverse bias increases the width of the depletion layer, which decreases the junction's capacitance. The reverse bias also increases the dark current without much change in the photocurrent.

Applications of Photodiode

- Photodiodes are very versatile light sensors that can turn its current flow both "ON" and "OFF" in nanoseconds and are commonly used in cameras, light meters, CD and DVD-ROM drives, TV remote controls, scanners, fax machines and copiers etc, and when integrated into operational amplifier circuits as infrared spectrum detectors for fiber optic communications, burglar alarm motion detection circuits and numerous imaging, laser scanning and positioning systems etc..

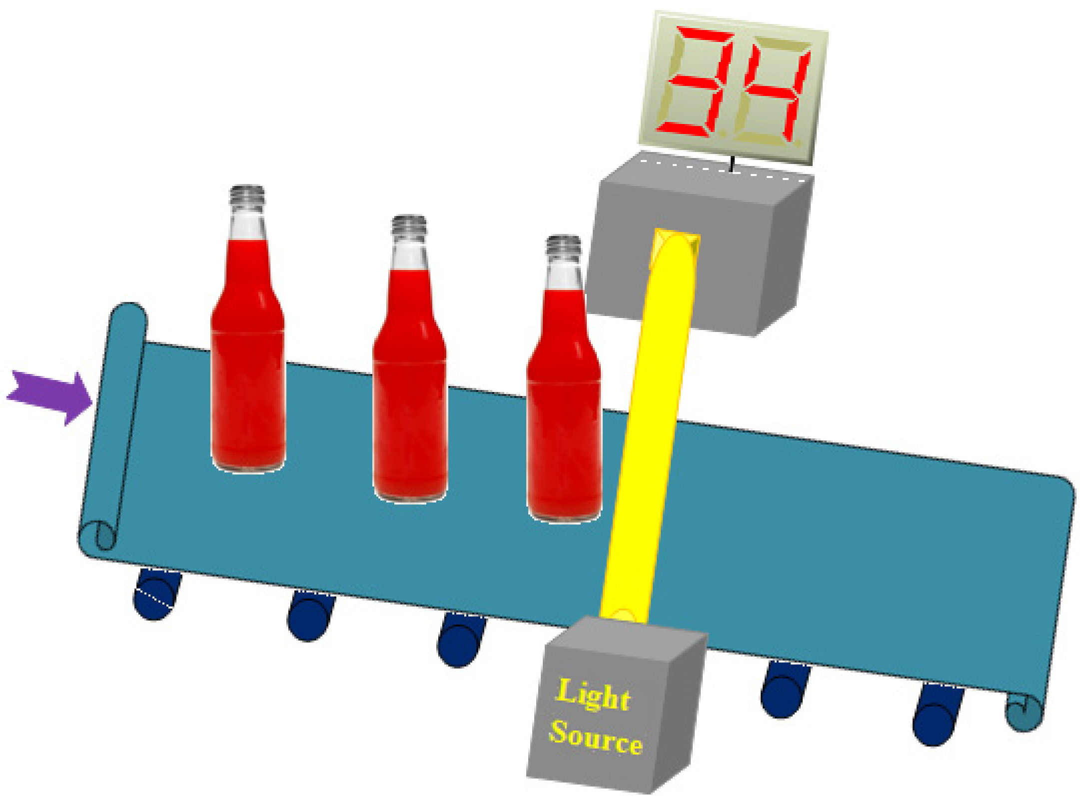
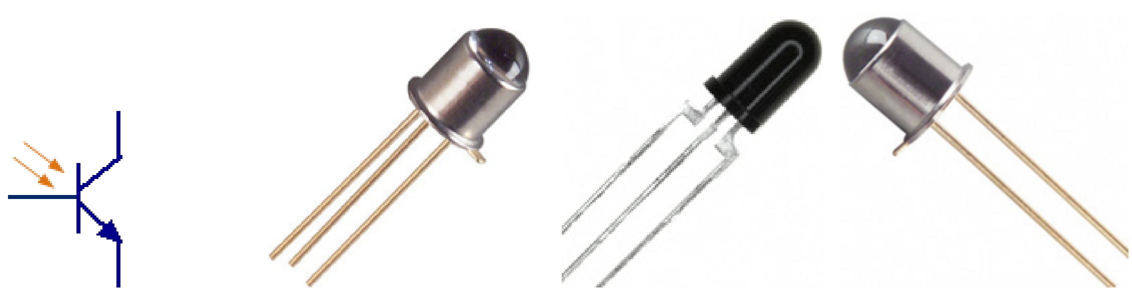


Figure 5-9: A photodiode circuit used in a system that counts objects as they pass on a conveyor belt

- Photodiodes may be used to generate an output which is dependent upon the illumination (analog; for measurement and the like), or to change the state of circuitry (digital; either for control and switching, or digital signal processing).
- Photodiode like other photosensor, may be used for light measurement, as in camera light meters, or to respond to light levels, as in switching on street lighting after dark.
- A photodiode is often combined into a single component with an emitter of light, usually a light-emitting diode (LED), either to detect the presence of a mechanical obstruction to the beam (slotted optical switch), or to couple two digital or analog circuits while maintaining extremely high electrical isolation between them, often for safety (optocoupler).
- A simple photodiode application is depicted in figure 5-9. Here a beam of light continuously passes across a conveyor belt and into a transparent window behind which is a photodiode circuit. When the light beam is interrupted by an object passing by a conveyor belt, the sudden reduction current activates a control circuit that advances a counter by one. The total count of objects that have passed that point is displayed by the counter. This basic concept can be extended and used for production control, shipping, and monitoring of activity on production line.

5. 2. 2. The Phototransistor

A phototransistor is a light-sensitive transistor. Although all transistors are light-sensitive, Phototransistors are designed specifically to take advantage of this fact. A common type of phototransistor, called a photobipolar transistor, is in essence a bipolar transistor encased in a transparent case so that light can reach the base-collector junction. Here, light striking the base replaces what would ordinarily be voltage applied to the base. An alternative photo-junction device to the photodiode is the Phototransistor which is basically a photodiode with amplification.



(a) (b)

Figure 5-10: Phototransistor; (a) Phototransistor symbol, (b) Typical phototransistors

The Phototransistor light sensor has its collector-base PN-junction reverse biased exposing it to the radiant light source. The electrons that are generated by photons in the base–collector junction are injected into the base, and this photodiode current is amplified by the transistor's current gain (β or hfe). If the emitter is left unconnected, the phototransistor becomes a photodiode. While phototransistors have a higher responsivity for light they are not able to detect low levels of light any better than photodiodes.

Phototransistors also have significantly longer response times. Field-effect phototransistors, also known as photoFETs, are light-sensitive field-effect transistors. Unlike photobipolar transistors, photoFETs control drain-source current by creating a gate voltage. Phototransistors are generally encased in an opaque or clear container in order to enhance light as it travels through it and allow the light to reach the phototransistor's sensitive parts.

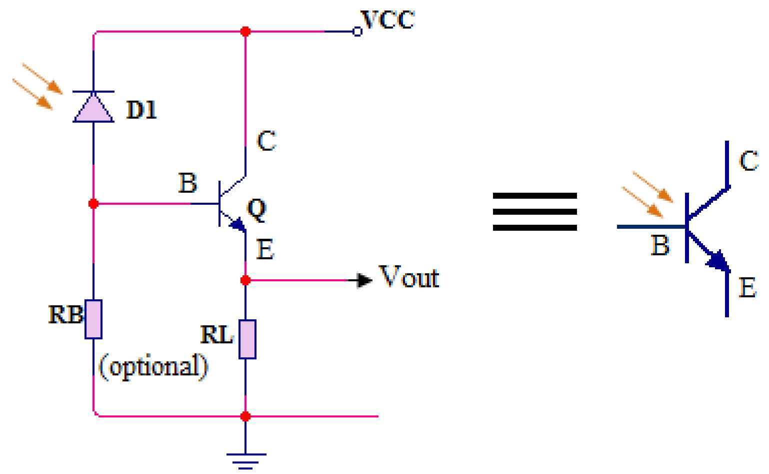


Figure 5-11: Photo-transistor equivalent circuit

A phototransistor generally has an exposed base, although some phototransistors allow a base connection to control the sensitivity, that amplifies the light that it comes in contact with which in turn causes a collector to emitter current to flow. This causes a relatively high current to pass through the phototransistor. As the current spreads from the base to the emitter, the current is concentrated and converted into voltage. Most phototransistors are NPN.

Phototransistors operate the same as the photodiode except that they can provide current gain and are much more sensitive than the photodiode with currents are 50 to 100 times greater than that of the standard photodiode and any normal transistor can be easily converted into a phototransistor light sensor by connecting a photodiode between the collector and base.

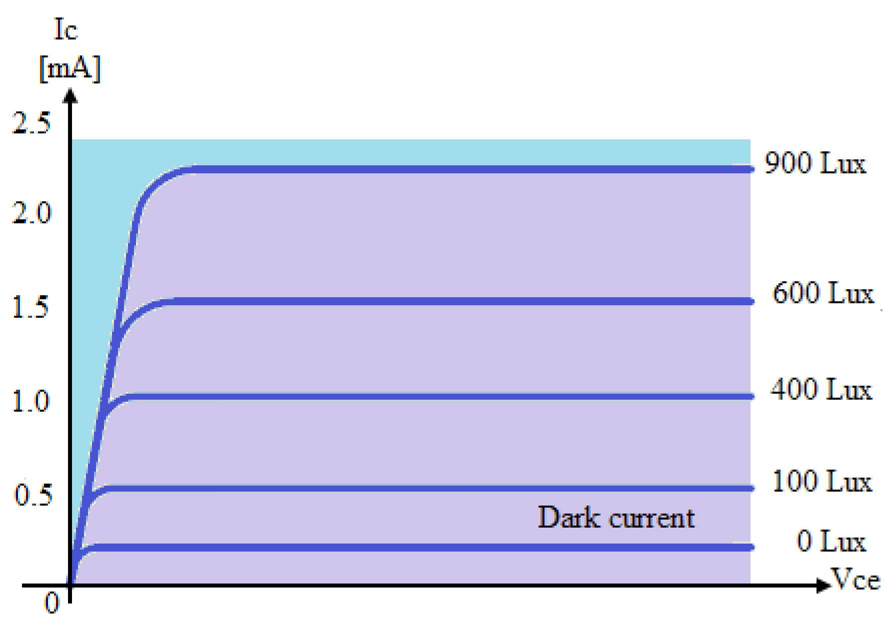


Figure 5-12: Graph representing collector current for a given light intensity

In the NPN transistor the collector is biased positively with respect to the emitter so that the base-collector junction is reverse biased. Therefore, with no light on the junction normal leakage or dark current flows which is very small. When light falls on the base more electron/hole pairs are formed in this region and the current produced by this action is amplified by the transistor.

Advantages of Phototransistor

Phototransistors have several important advantages that separate them from other optical sensors. They produce a higher current than photodiodes and also produce a voltage, something that photoresistors cannot do. Phototransistors are very fast and their output is practically instantaneous. They are relatively inexpensive, simple, and so small that several of them can fit onto a single integrated computer chip.

Disadvantages of Phototransistor

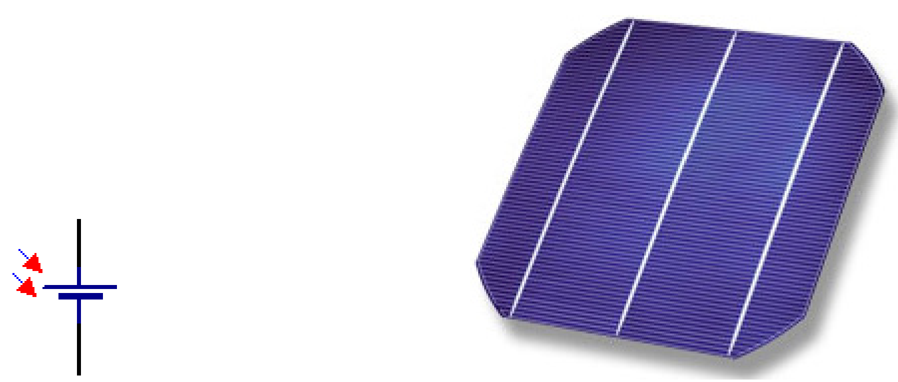
While phototransistors can be advantageous, they also have some disadvantages. Phototransistors do not allow electrons to move as freely as other devices, such as electron tubes, do. Also, phototransistors are also more vulnerable to electrical surges/spikes and electromagnetic energy.

Application of Phototransistor

Typical applications of Phototransistors light sensors are in opto-isolators, slotted opto-switches, light beam sensors, fiber optics and TV type remote controls. Phototransistors are also used in smoke detectors, infrared receivers (Infrared filters are sometimes required when detecting visible light), and CD players. Phototransistors can also be used in astronomy, night vision, and laser range-finding.

5. 3. Photovoltaic Cells

Solar cells or photovoltaic cells convert light energy directly into DC electrical energy in the form of a voltage or current to power a resistive load such as a light, battery or motor. Then photovoltaic cells are similar in many ways to a battery because they supply DC power. It is a form of photoelectric cell, defined as a device whose electrical characteristics, such as current, voltage, or resistance, vary when exposed to light. Solar cells are the building blocks of photovoltaic modules, also known as solar panels



(a) (b)

Figure 5-13: Photovoltaic cell; (a) Solar cell symbol, (b) Typical photovoltaic cell

However, unlike the other photo devices which use light intensity even from a torch to operate, photovoltaic solar cells work best using the suns radiant energy.

Solar cells are described as being photovoltaic irrespective of whether the source is sunlight or an artificial light. They are used as a photodetector (for example infrared detectors), detecting light or other electromagnetic radiation near the visible range, or measuring light intensity.

Solar cells are used in many different types of applications to offer an alternative power source from conventional batteries, such as in calculators, satellites and now in homes offering a form of renewable power.

Photovoltaic cells are made from single crystal silicon PN junctions, the same as photodiodes with a very large light sensitive region but are used without the reverse bias. They have the same characteristics as a very large photodiode when in the dark.

When illuminated the light energy causes electrons to flow through the PN junction and an individual solar cell can generate an open circuit voltage of about 0.58V (580mV). Solar cells have a "Positive" and a "Negative" side just like a battery.

Individual solar cells can be connected together in series to form solar panels which increases the output voltage or connected together in parallel to increase the available current. Commercially available solar panels are rated in Watts, which is the product of the output voltage and current (Volts times Amps) when fully lit.

The amount of available current from a solar cell depends upon the light intensity, the size of the cell and its efficiency which is generally very low at around 15 to 20%. To increase the overall efficiency of the cell commercially available solar cells use polycrystalline silicon or amorphous silicon, which have no crystalline structure, and can generate currents of between 20 to 40mA per cm^2.

Applications of Photovoltaic cells

Assemblies of photovoltaic cells are used to make solar modules which generate electrical power from sunlight, as distinguished from a "solar thermal module" or "solar hot water panel". The electrical energy generated from solar modules, colloquially referred to as solar power, is an example of solar energy.

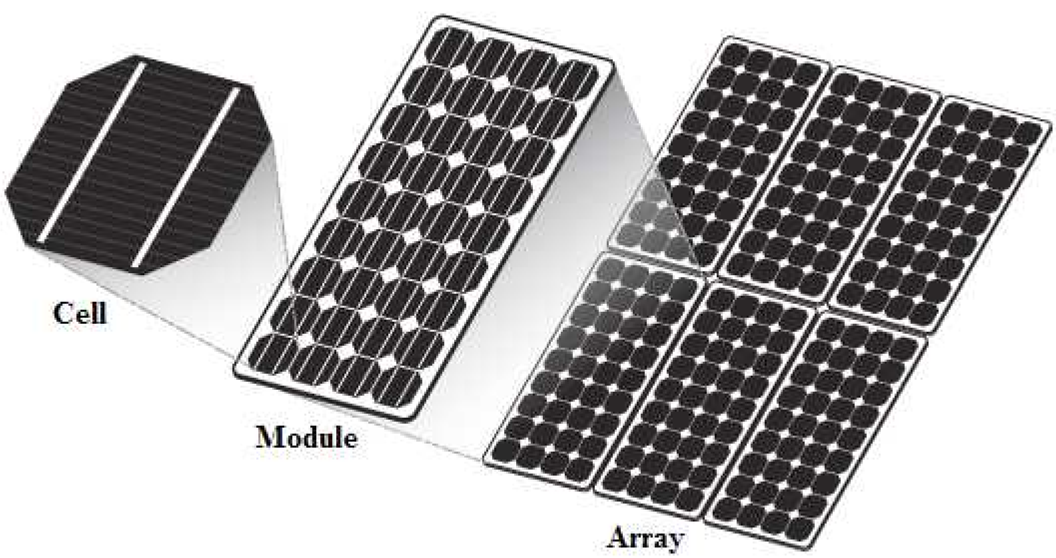


Figure 5-14: Photovoltaic cells are attached in different combinations with each other to make a module or array in order to supply a required voltage

Multiple solar cells in an integrated group, all oriented in one plane, constitute a solar photovoltaic panel or solar photovoltaic module. Photovoltaic modules often have a sheet of glass on the sun-facing side, allowing light to pass while protecting the semiconductor wafers. Solar cells are usually connected in series in modules, creating an additive voltage. Connecting cells in parallel yields a higher current; however, problems such as shadow effects can shut down the weaker (less illuminated) parallel string (a number of series connected cells) causing substantial power loss and possible damage because of the reverse bias applied to the shadowed cells by their illuminated partners.

Calculators with solar cells, devices that never need batteries and in some cases don't even have an off button. As long as there is enough light, they seem to work forever. Solar photovoltaic panels are also used on emergency road signs, call boxes, in homes offering a form of renewable power, orbiting our planet on satellites, and even in parking lots to power the lights.

5. 4. Temperature Sensor

Sensors which detect Temperature or heat are the most frequently used type of all the sensors. Temperature sensors vary from simple ON/OFF thermostatic devices which control a domestic heating system to highly sensitive semiconductor types that can control complex process control heater plants.

From the principle of the movement of molecules and atoms produces heat (kinetic energy) and the greater the movement, the more heat that is generated. Temperature sensors measure the amount of heat energy or even coldness that is generated by an object or system, allowing us to "sense" or detect any physical change to that temperature producing either an analogue or digital output.

There are many different types of temperature sensor available and all have different characteristics depending upon their actual application. A Temperature Sensor consists of two basic physical types:

- Contact Temperature Sensor Types: These types of temperature sensor are required to be in physical contact with the object being sensed and use conduction to monitor changes in temperature. They can be used to detect solids, liquids or gases over a wide range of temperatures.
- Non-contact Temperature Sensor Types: These types of temperature sensor use convection and radiation to monitor changes in temperature. They can be used to detect liquids and gases that emit radiant energy as heat rises and cold settles to the bottom in convection currents or detect the radiant energy being transmitted from an object in the form of infra-red radiation (the sun).

The two basic types of contact or even non-contact temperature sensors can also be sub-divided into the following three groups of sensors, *Electro-mechanical*, *Resistive* and *Electronic* and all three types are discussed below.

5. 4. 1. The Thermostat

The Thermostat is a contact type of electro-mechanical temperature sensor or switch, that basically consists of two different metals such as nickel, copper, tungsten or aluminium etc, that are bonded together to form a Bi-metallic strip. The different linear expansion rates of the two dissimilar metals produce a mechanical bending movement when the strip is subjected to heat.

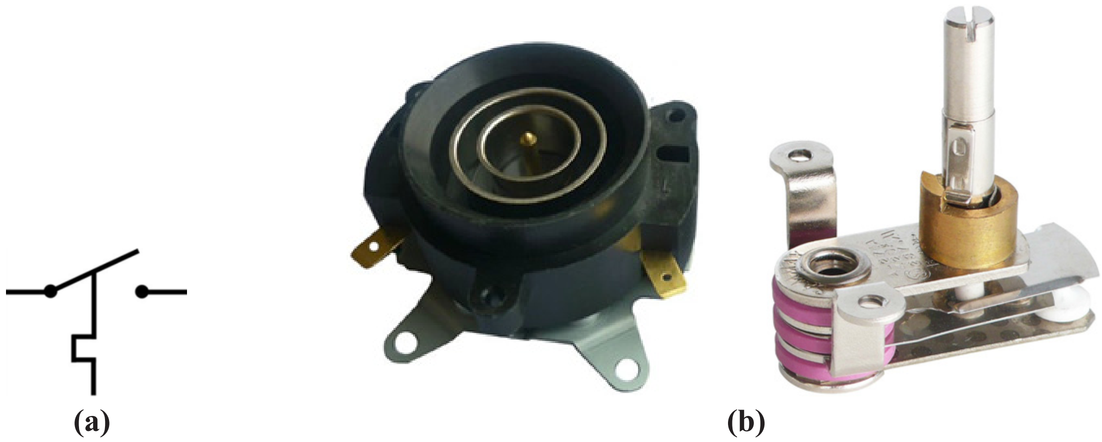


Figure 5-15: Thermostat; (a) Thermostat symbol, (b) Typical On/Off Thermostat used in kettle and in iron

The bi-metallic strip can be used itself as an electrical switch or as a mechanical way of operating an electrical switch in thermostatic controls and are used extensively to control hot water heating elements in boilers, furnaces, hot water storage tanks as well as in vehicle radiator cooling systems.

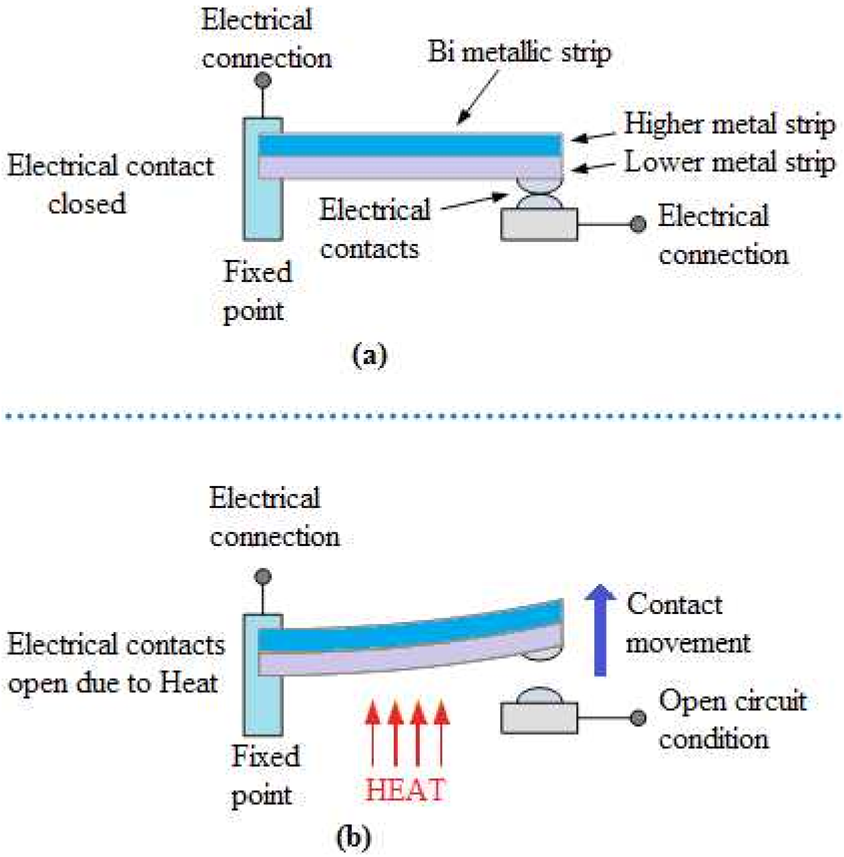


Figure 5-16: Thermostat structure; (a) Thermostat without heat contacts are closed, (b) Thermostat under heat bi-metallic strips bends and open contacts

The thermostat consists of two thermally different metals stuck together back to back. When it is cold the contacts are closed and current passes through the thermostat. When it gets hot, one metal expands more than the other and the bonded bi-metallic strip bends up (or down) opening the contacts preventing the current from flowing. There are two main types of bi-metallic strips based mainly upon their movement when subjected to temperature changes. There are the "snap-action" types that produce an instantaneous "ON/OFF" or "OFF/ON" type action on the electrical contacts at a set temperature point, and the slower "creep-action" types that gradually change their position as the temperature changes.

Although very cheap and are available over a wide operating range, one main disadvantage of the standard snap-action type thermostats when used as a temperature sensor, is that they have a large hysteresis range from when the electrical contacts open until when they close again. For example, it may be set to 20ºC but may not open until 22ºC or close again until 18ºC.

So the range of temperature swing can be quite high. Commercially available bi-metallic thermostats for home use do have temperature adjustment screws that allow for a more precise desired temperature set-point and hysteresis level to be preset.

Applications of Thermostat

Snap-action type thermostats are commonly used in our homes for controlling the temperature set point of ovens, irons, immersion hot water tanks and they can also be found on walls to control the domestic heating system.

Creeper types generally consist of a bi-metallic coil or spiral that slowly unwinds or coils-up as the temperature changes. Generally, creeper type bi-metallic strips are more sensitive to temperature changes than the standard snap ON/OFF types as the strip is longer and thinner making them ideal for use in temperature gauges and dials etc..

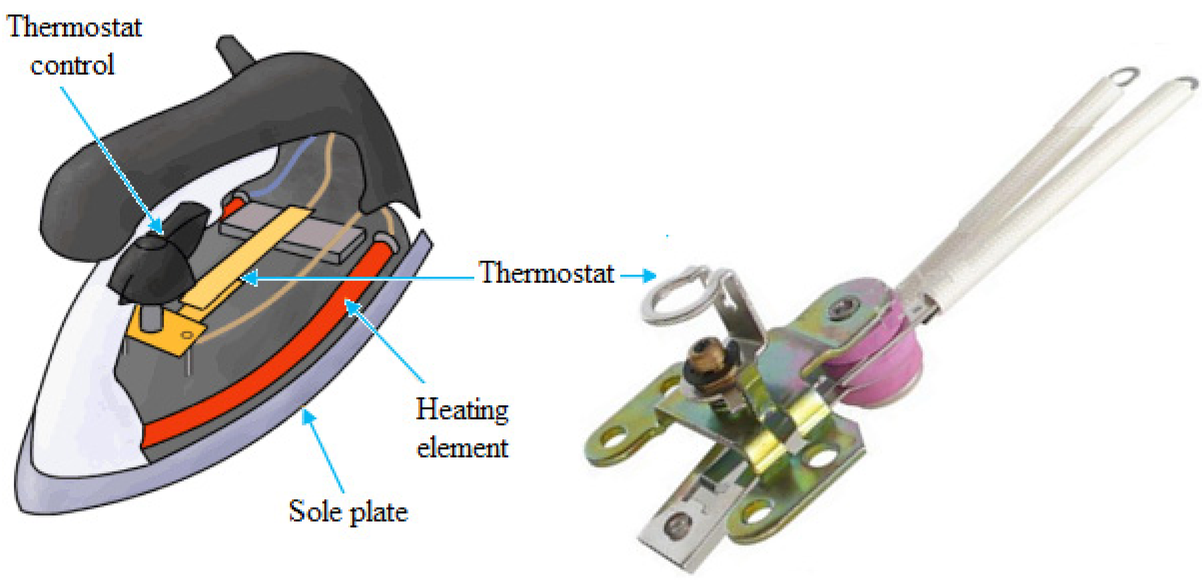
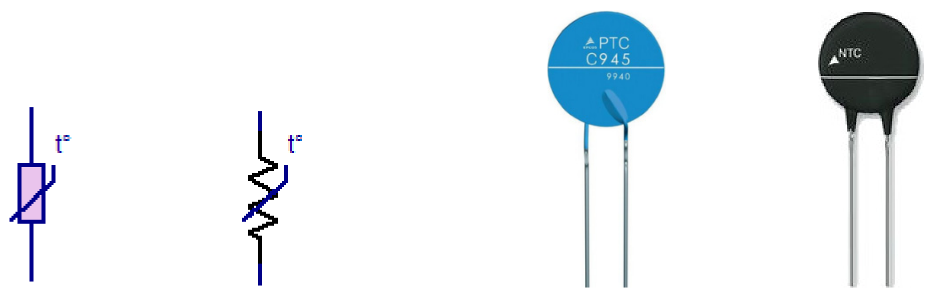


Figure 5-17: Thermostat in iron

5. 4. 2. The Thermistor

Thermistors are temperature sensitive resistors. A thermistor is a type of resistor whose resistance varies significantly with temperature, more than in standard resistors. Thermistor name is a combination of the words THERM-ally sensitive res-ISTOR. All resistors vary with temperature, but thermistors are constructed of semiconductor material with a resistivity that is especially sensitive to temperature.



(a) (b)

Figure 5-18: Thermistor; (a) American and European symbol, (b) Typical disc termistor

Thermistors are generally made from ceramic materials such as oxides of nickel, manganese or cobalt coated in glass which makes them easily damaged. Their main advantage over snap-action types is their speed of response to any changes in temperature, accuracy and repeatability.

Basic operation and types of thermistors

Assuming, as a first-order approximation, that the relationship between resistance and temperature is linear, then:

$$\Delta R = k * \Delta T$$

Where

ΔR: Change in resistance

ΔT: Change in temperature

k: First-order temperature coefficient of resistance

Thermistors can be classified into two types, depending on the sign of k. Most types of thermistors have a Negative Temperature Coefficient of resistance or (NTC), that is their resistance value goes down with an increase in the temperature, and of course there are some which have a Positive Temperature Coefficient, (PTC), in that their resistance value goes up with an increase in temperature.

1. Positive temperature coefficient (PTC) thermistor

If k is positive, the resistance increases with increasing temperature, and the device is called a positive temperature coefficient (PTC) thermistor, or posistor.

Most PTC thermistors are of the "switching" type, which means that their resistance raises suddenly at a certain critical temperature.

Experiment 5-2: Positive temperature coefficient resistor (PTC) thermistor under temperature resistance measuring

Measure and observe the resistance change of a PTC resistor held near a variable temperature source.

Part list

No	Items	Specifications	Quantity
1	Iron solder	40W iron solder	
2	Positive temperature coefficient	PTC 10KΩ	
3	Ohmmeter	Digital multimeter	

Procedure

1. Connect the soldering iron to the supply and allow it to rise its temperature. The other source of heat can be used instead.
2. Set the multimeter in Ohm's range to about 10K scale.
3. Hold the multimeter leads against the PTC resistor's legs and come within reach of temperature.
4. Observe and note the Ohmmeter reading. It may be seen that as temperature increases, the resistance of the PTC also increases.

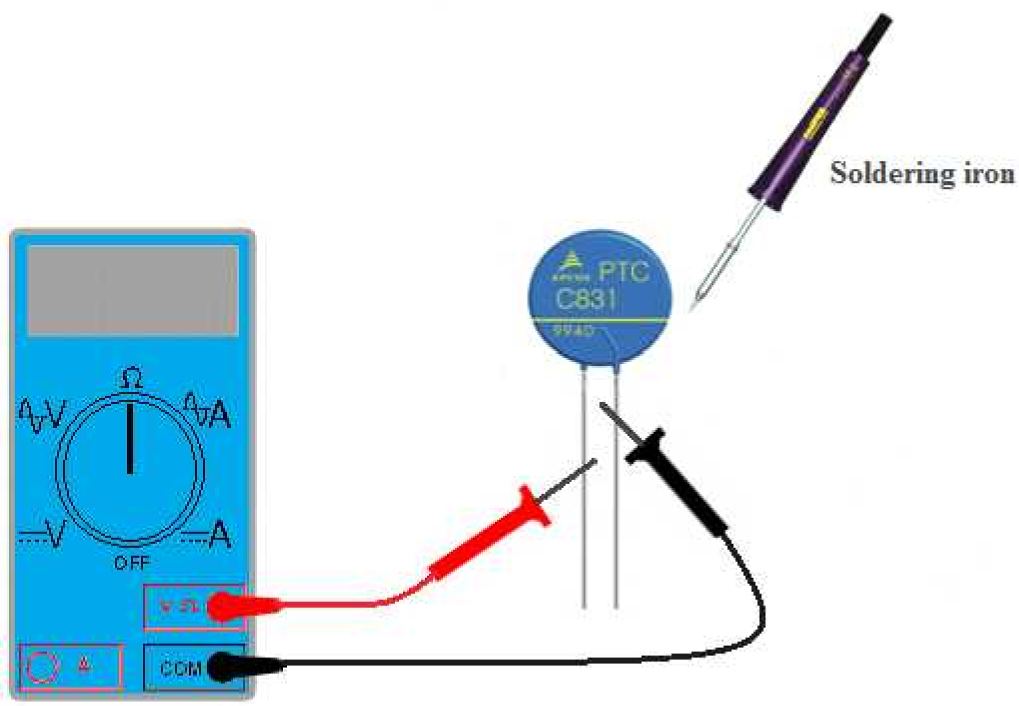


Figure 5-19: Measuring the resistance of PTC thermistar under temperature

2. Negative temperature coefficient (NTC) thermistor

If k is negative, the resistance decreases with increasing temperature, and the device is called a negative temperature coefficient (NTC) thermistor.

Many NTC thermistors are made from a pressed disc or cast chip of a semiconductor such as a sintered metal oxide. They work because raising the temperature of a semiconductor increases the number of electrons able to move about and carry charge; it promotes them into the conduction band. The more charge carriers that are available, the more current a material can conduct.

Experiment 5-3: Measuring resistance of negative temperature coefficient (NTC) thermistor under temperature condition

Measure and observe the resistance change of a NTC resistor held near a variable temperature source.

Part list

No	Items	Specifications	Quantity
1	Iron solder	40W iron solder	1
2	Negative temperature coefficient	NTC 2.2KΩ	1
3	Ohmmeter	Digital multimeter	1

Procedure

5. Connect the soldering iron to the supply and allow it to rise its temperature. The other source of heat can be used instead.
6. Set the multimeter in Ohm's range to about 2K scale.
7. Hold the multimeter leads against the NTC resistor's legs and come within reach of temperature.
8. Observe and note the Ohmmeter reading. It may be seen that as temperature increases, the resistance of the NTC decreases.

chap 5

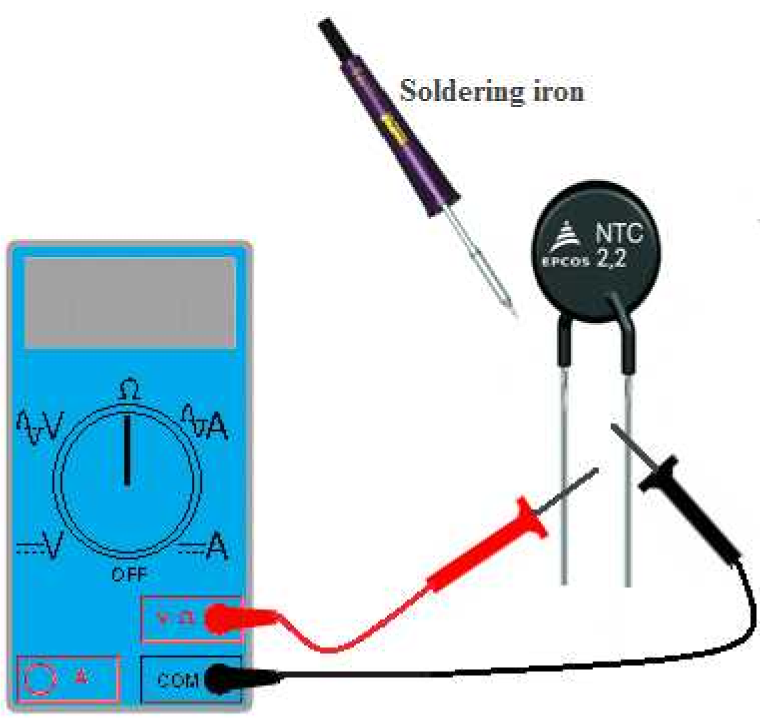


Figure 5-20: Measuring the resistance of NTC thermistar under temperature

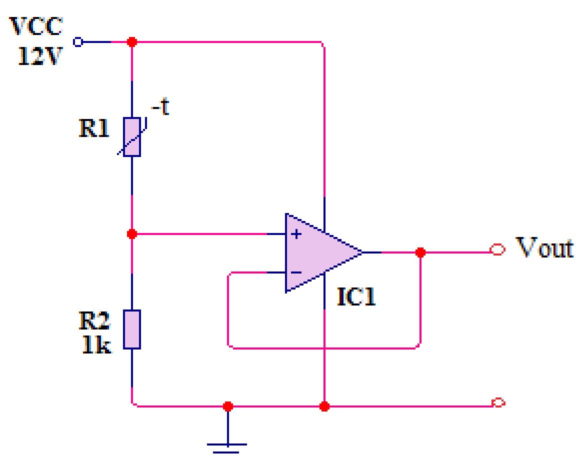
Thermistors are constructed from a ceramic type semiconductor material using metal oxide technology such as manganese, cobalt and nickel, etc.. The semiconductor material is generally formed into small pressed discs or balls which are hermetically sealed to give a relatively fast response to any changes in temperature.

Thermistors are rated by their resistive value at room temperature (usually at 25ºC), their time constant (the time to react to the temperature change) and their power rating with respect to the current flowing through them. Like resistors, thermistors are available with resistance values at room temperature from 10's of MΩ down to just a few Ohms, but for sensing purposes those types with values in the kilo-ohms are generally used.

Thermistors are passive resistive devices which means we need to pass a current through it to produce a measurable voltage output. Then thermistors are generally connected in series with a suitable biasing resistor to form a potential divider network and the choice of resistor gives a voltage output at some pre-determined temperature point or value for example:

Temperature Sensors Circuit Example

A thermistor has a resistance value of 5KΩ at 25ºC and a resistance value of 200Ω at 80ºC. Calculate the voltage drop across the thermistor and hence its output voltage (V_{out}) for both temperatures when connected in series with a 1kΩ resistor across a 12V power supply. An amplifier IC_1 is an emitter follower with a unit gain.



At 25ºC

$$V_{out}=V*(R_2/R_1+R_2)=12V*(1000/5000+1000)=2V$$

At 80ºC

$$V_{out}=V*(R_2/R_1+R_2)=12V*(1000/200+1000)=10V$$

By changing the fixed resistor value of R_2 (in our example 1kΩ) to a potentiometer or preset, a voltage output can be obtained at a predetermined temperature set point for example, 5V output at 60ºC and by varying the potentiometer a particular output voltage level can be obtained over a wider temperature range.

It needs to be noted however, that thermistors are non-linear devices and their standard resistance values at room temperature is different between different thermistors, which is due mainly to the semiconductor materials they are made from. The Thermistor have an exponential change with temperature and therefore have a Beta temperature constant (β) which can be used to calculate its resistance for any given temperature point.

However, when used with a series resistor such as in a voltage divider network or Whetstone Bridge type arrangement, the current obtained in response to a voltage applied to the divider/bridge network is linear with temperature. Then, the output voltage across the resistor becomes linear with temperature.

Applications of thermistor

- PTC thermistors can be used as current-limiting devices for circuit protection, as replacements for fuses.
- PTC thermistors are used as timers in the degaussing coil circuit of most CRT (cathode ray tube) displays.
- PTC thermistors are used as heater in automotive industry to provide additional heat inside cabin with diesel engine or to heat diesel in cold climatic conditions before engine injection.
- NTC thermistors are used as resistance thermometers in low-temperature measurements of the order of 10 K.
- NTC thermistors can be used as inrush-current limiting devices in power supply circuits.
- NTC thermistors are regularly used in automotive applications
- NTC thermistors can be also used to monitor the temperature of an incubator.
- Thermistors are also commonly used in modern digital thermostats and to monitor the temperature of battery packs while charging.
- Thermistors are often used in the hot ends of 3D printers; they monitor the heat produced and allow the printer's control circuitry to keep a constant temperature for melting the plastic filament.
- NTC thermistors are used in the food handling and processing industry, especially for food storage systems and food preparation.
- NTC thermistors are used throughout the consumer appliance industry for measuring temperature. Toasters, coffee makers, refrigerators, freezers, hair dryers, etc. all rely on thermistors for proper temperature control.

Inrush Current Limiting Power NTC Thermistors

Inrush current, input surge current or switch-on surge, is the maximum input current drawn by an electrical device when first turned on. Inrush-current limiters power NTC thermistor protect circuits from undesirably high currents, suppressing high inrush current surges, while its resistance remains negligible low during continuous operation. Thanks to their low resistance in the operating state, power thermistor have considerably lower power dissipation than the fixed resistors frequently used for this application. As the inrush current limiter self-heats, the current begins to flow through it and warm it. Its resistance begins to drop and a relatively small current flow. The self heated inrush current limiter offers little resistance in the circuit, with a low voltage drop with respect to the total voltage drop of the circuit. A disadvantage is that immediately after the device is switched off, the NTC resistor is still hot and has a low resistance. It cannot limit the inrush current unless it cools for more than 1 minute to get a higher resistance. Another disadvantage is that the NTC thermistor is not short circuit proof.

Inrush Current Limiting Power NTC Thermistor Advantages

- Low cost solid state device for inrush current suppression.

- Minimize line current distortion and radio noise.
- Protect switches, rectifier diodes and smoothing capacitors against premature failures.
- Prevent fuse from blowing in error.

Inrush Current Limiting Power NTC Thermistor Features

- Resin coated disk thermistor with uninsulated lead-wires.
- Suitable for both AC and DC circuits up to a voltage of 265 V(rms).
- Wide range of resistance, current and dimension.
- Excellent mechanical strength.
- Suitable for PCB mounting.

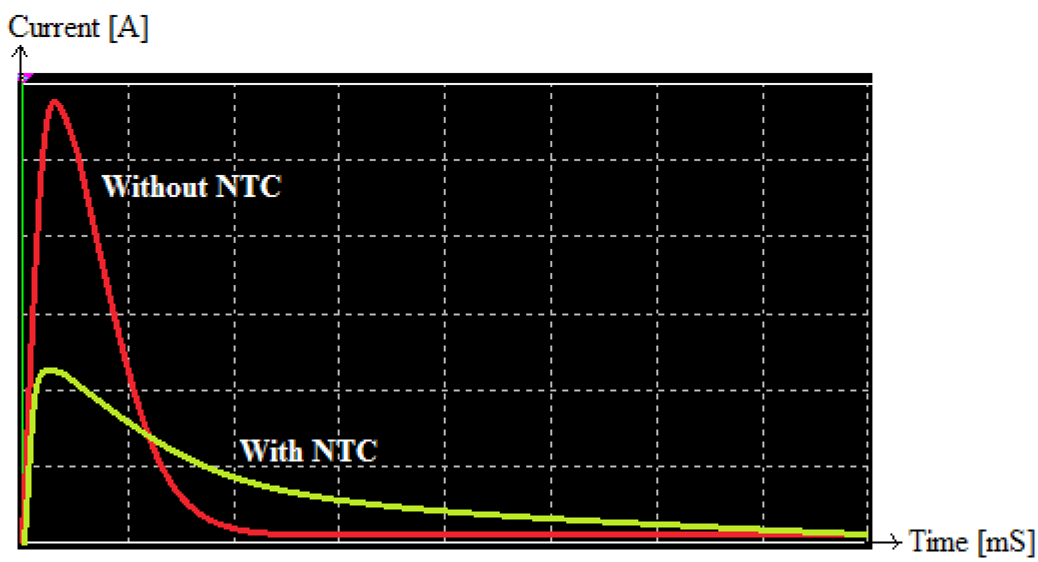


Figure 5-21: Comparison curve with and without Inrush Current Limiting power NTC thermistor

Basic circuit diagram with NTC thermistor for diode protection is shown in figure 5-22.

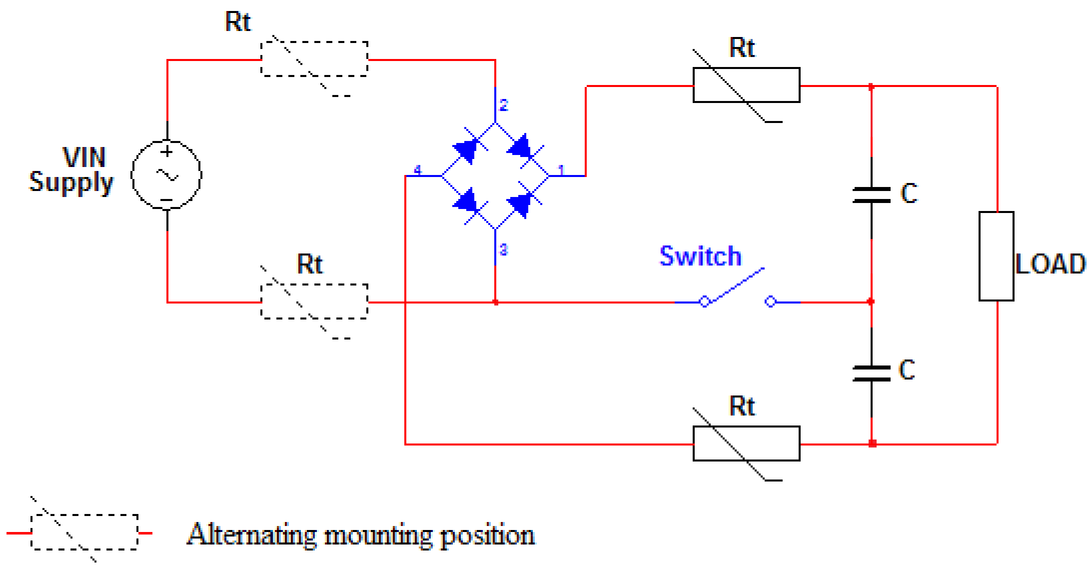


Figure 5-22: Basic circuit diagram with NTC thermistor for diode protection

Typical Application of Inrush Current Limiting Power NTC thermistors for Circuit Protection is shown in figure 5-23.

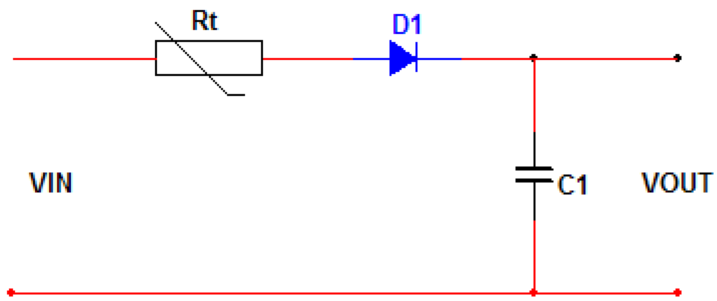


Figure 5-23: Application of Inrush Current Limiting Power NTC thermistors for Circuit Protection

Inrush Current Limiting Power NTC Thermistor Applications

limiting surge current, suitable for the protection of switch mode power supply, UPS power, transformers, motors, various electric heating utensil, energy saving lights, ballast, various power circuit, amplifiers, colored displayer, monitors, color TV, filament protection, etc..

Power thermistor components can also be used for the soft starting of motors, for example in vacuum cleaners with continuous currents of up to 20 A.

chap 5

5. 4. 3. Resistive Temperature Detectors (RTD)

Resistance Temperature Detector or RTD are precision temperature sensors made from high-purity conducting metals such as platinum, copper or nickel wound into a coil and whose electrical resistance changes as a function of temperature, similar to that of the thermistor. These devices have a thin film of platinum paste deposed onto a white ceramic substrate.

Figure 5-24: Typical resistive temperature detector

Resistive temperature detectors have positive temperature coefficients but unlike the thermistor their output is extremely linear producing very accurate measurements of temperature.

However, they have very poor thermal sensitivity, that is a change in temperature only produces a very small output change for example, 1Ω/°C.

The more common types of RTD are made from platinum and are called Platinum Resistance Thermometer or PRT with the most commonly available of them all the Pt100 sensor, which has a standard resistance value of 100Ω at 0°C. The downside is that Platinum is expensive and one of the main disadvantages of this type of device is its cost.

Like the thermistor, RTD are passive resistive devices and by passing a constant current through the temperature sensor it is possible to obtain an output voltage that increases linearly with temperature. A typical RTD has a base resistance of about 100Ω at 0°C, increasing to about 140Ω at 100°C with an operating temperature range of between -200 and + 600°C.

An RTD takes a measurement when a small DC current is supplied to the sensor. The current experiences the impedance of the resistor, and a voltage drop is experienced over the resistor. Depending on the nominal resistance of the RTD, different supply currents can be used. To reduce self-heating on the sensor the supply current should be kept low. In general, around 1mA or less of current is used. Because the RTD is a resistive device, we need to pass a current through them and monitor the resulting voltage. However, any variation in resistance due to self heat of the resistive wires as the current flows through it, I^2R , (Ohms Law) causes an error in the readings. To avoid this, the RTD is usually connected into a Whetstone Bridge network which has additional connecting wires for lead-compensation and/or connection to a constant current source.

Advantage and Disadvantage of RTD

RTDs (resistance temperature detectors) are one of the most common temperature sensor types used in industrial applications. Thermocouples and thermistors are popular temperature sensors as well, but RTD sensors are more accurate over a wide temperature range, interchangeability over a wide temperature range, and more stable over time, making them an excellent choice for many applications.

Even though the RTD are advantageous in temperature measuring but have some drawback include: Low sensitivity, higher cost than thermocouple, no point sensing, RTD are affected by chock and vibration, require three or four wires operation.

Application of RTD

Resistor temperature detectors (RTD) are used in air conditioning and refrigeration servicing, food processing, stoves and grills, textile production, plastics processing, petrochemical processing, micro electronics, air or gas and liquid temperature measurement, exhaust gas temperature measurement

chap 5

5. 4. 4. The Thermocouple

The Thermocouple is by far the most commonly used type of all the temperature sensor types. Thermocouples are popular due to its simplicity, ease of use and their speed of response to changes in temperature, due mainly to their small size. Thermocouples also have the widest temperature range of all the temperature sensors from below -200°C to well over 2000°C.

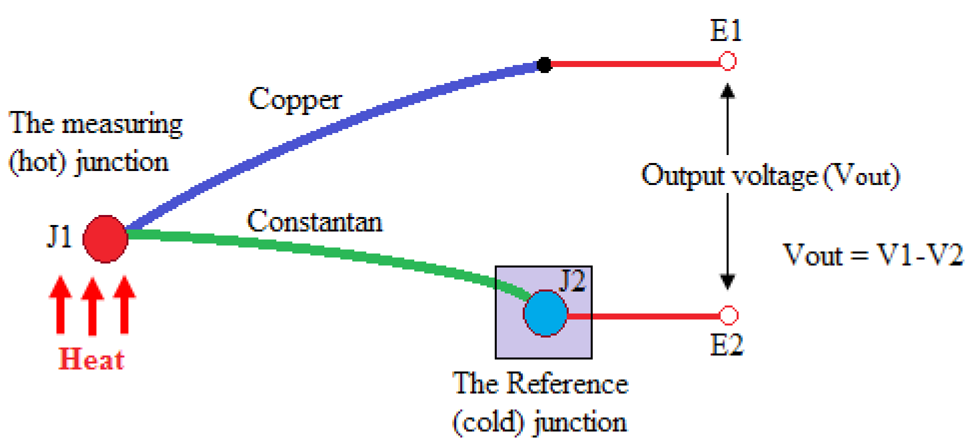


Figure 5-25: Thermocouple construction. The voltage output being the temperature difference between the two dissimilar junctions

Thermocouples are thermoelectric sensors that basically consist of two junctions of dissimilar metals, such as copper and constantan that are welded or crimped together. One junction is kept at a constant temperature called the reference (Cold) junction, while the other the measuring (Hot) junction. When the two junctions are at different temperatures, a voltage is developed across the junction which is used to measure the temperature sensor as shown in figure 5-25.

The operating principal of a thermocouple is as follow: When fused together the junction of the two dissimilar metals such as copper and constantan produces a "thermo-electric" effect which gives a constant potential difference of only a few millivolts (mV) between them. The voltage difference between the two junctions is called the "Seebeck effect" as a temperature gradient is generated along the conducting wires producing an emf (electromotive force). Then the output voltage from a thermocouple is a function of the temperature changes.

If both the junctions are at the same temperature the potential difference across the two junctions is zero in other words, no voltage output as $E_1 = E_2$. However, when the junctions are connected within a circuit and are both at different temperatures a voltage output will be detected relative to the difference in temperature between the two junctions, $E_1 - E_2$. This difference in voltage will increase with temperature until the junction's peak voltage level is reached and this is determined by the characteristics of the two dissimilar metals used.

Thermocouples can be made from a variety of different materials enabling extreme temperatures of between -200ºC to over +2000ºC to be measured. With such a large choice of materials and temperature range, internationally recognised standards have been developed complete with thermocouple colour codes to allow the user to choose the correct thermocouple sensor for a particular application.

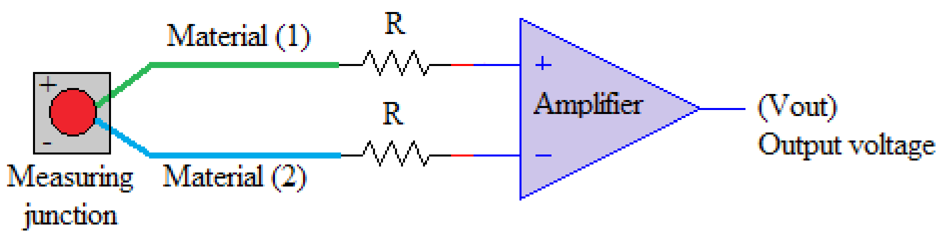


Figure 5-26: Thermocouple with amplifier

The type of amplifier, either discrete or in the form of an Operational Amplifier needs to be carefully selected, because good drift stability is required to prevent recalibration of the thermocouple at frequent intervals. This makes the chopper and instrumentation type of amplifier preferable for most temperature sensing applications.

Advantage of Thermocouple

One of the advantage of the thermocouples is that are able of being used to directly measure temperatures up to 2600°C. The other benefit is that the thermocouple junction may be grounded and brought into direct contact with the material being measured and also have high resolution.

Disadvantage of Thermocouple

Temperature measurement with a thermocouple requires two temperatures be measured, the junction at the work end (the hot junction) and the junction where wires meet the instrumentation copper wires (cold junction). To avoid error the cold junction temperature is in general compensated in the electronic instruments by measuring the temperature at the terminal block using with a semiconductor, thermistor, or RTD.

Thermocouples operation is relatively complex with potential sources of error. The materials of which thermocouple wires are made are not inert and the thermoelectric voltage developed along the length of the thermocouple wire may be influenced by corrosion etc..

The relationship between the process temperature and the thermocouple signal (millivolt) is not linear.

Application of Thermocouple

Thermocouples are widely used in science and industry; applications include temperature measurement for kilns, gas turbine exhaust, diesel engines, and other industrial processes. Thermocouples are also used in homes appliances, offices and businesses as the temperature sensors in thermostats, and also as flame sensors in safety devices for gas-powered major appliances.

5. 5. Position Sensor

As their name implies, Position Sensors detect the position of something which means that they are referenced either to or from some fixed point or position. These types of sensors provide a "positional" feedback.

One method of determining a position is to use either "distance", which could be the distance between two points such as the distance travelled or moved away from some fixed point, or by "rotation" (angular movement). For example, the rotation of robots wheel to determine its distance travelled along the ground. Either way, Position Sensors can detect the movement of an object in a straight line using Linear Sensors or by its angular movement using Rotational Sensors.

The Potentiometer

The most commonly used of all the "Position Sensors", is the potentiometer because it is an inexpensive and easy to use position sensor. It has a wiper contact linked to a mechanical shaft that can be either angular (rotational) or linear (slider type) in its movement, and which causes the resistance value between the wiper/slider and the two end connections to change giving an electrical signal output that has a proportional relationship between the actual wiper position on the resistive track and its resistance value. In other words, resistance is proportional to position.

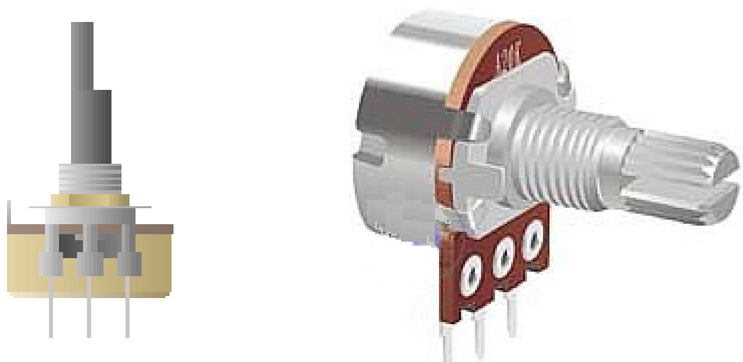


Figure 5-27: Typical potentiometers

Potentiometers come in a wide range of designs and sizes such as the commonly available round rotational type or the longer and flat linear slider types. When used as a positional sensor the moveable object is connected directly to the rotational shaft or slider of the potentiometer.

A DC reference voltage is applied across the two outer fixed connections forming the resistive element. The output voltage signal is taken from the wiper terminal of the sliding contact as shown in figure 5-28.

This configuration produces a potential or voltage divider type circuit output which is proportional to the shaft position. Then for example, if you apply a voltage of say 10V across the resistive element of the potentiometer the maximum output voltage would be equal to the supply voltage at 10 volts, with the minimum output voltage equal to 0 volts. Then the potentiometer wiper will vary the output signal from 0 to 10 volts, with 5 volts indicating that the wiper or slider is at its half-way or centre position.

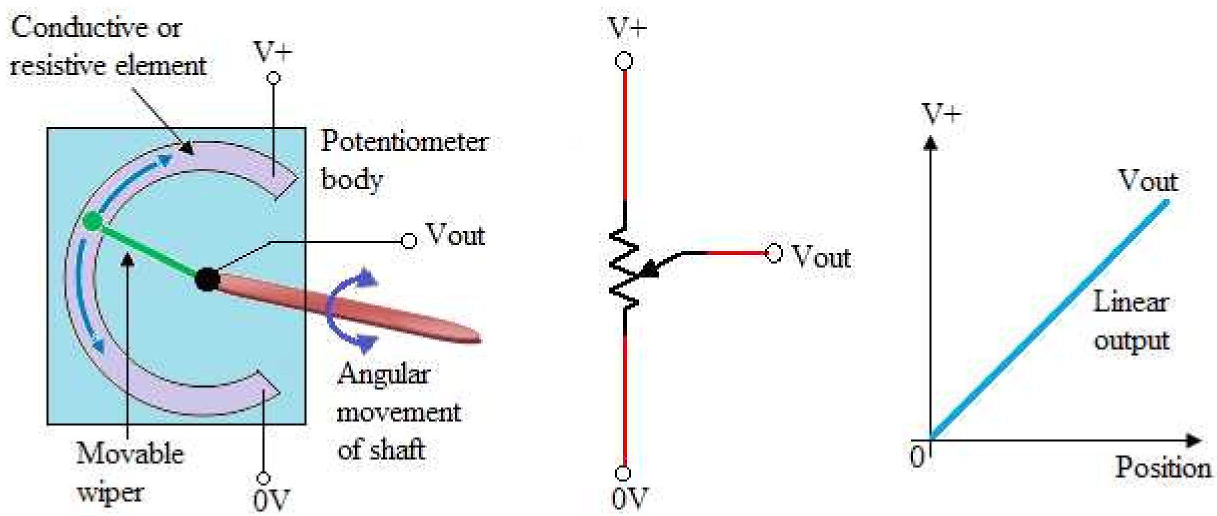


Figure 5-28: Potentiometer construction, symbol and graph representing the output voltage corresponding to the position of wiper

The output signal (V_{out}) from the potentiometer is taken from the centre wiper connection as it moves along the resistive track, and is proportional to the angular position of the shaft. While resistive potentiometer position sensors have many advantages: low cost, low tech, easy to use etc, as a position sensor they also have many disadvantages: wear due to moving parts, low accuracy, low repeatability, and limited frequency response. But there is one main disadvantage of using the potentiometer as a position sensor. The range of movement of its wiper or slider (and hence the output signal obtained) is limited to the physical size of the potentiometer being used.

For example a single turn rotational potentiometer generally only has a fixed mechanical rotation of between 0° and about 240° to 330° maximum. However, multi-turn pots of up to 3600° (10 x 360°) of mechanical rotation are also available.

Most types of potentiometers use carbon film for their resistive track, but these types are electrically noisy (the crackle on a radio volume control), and also have a short mechanical life.

Wire-wound pots also known as rheostats, in the form of either a straight wire or wound coil resistive wire can also be used, but wire wound pots suffer from resolution problems as their wiper jumps from one wire segment to the next producing a logarithmic (LOG) output resulting in errors in the output signal. These too suffer from electrical noise.

For high precision low noise applications conductive plastic resistance element type polymer film or cermet type potentiometers are now available. These pots have a smooth low friction electrically linear (LIN) resistive track giving them a low noise, long life and excellent resolution and are available as both multi-turn and single turn devices. Typical applications for this type of high accuracy position sensor is in computer game joysticks, steering wheels, industrial and robot applications.

Typical Application of Potentiometer as Positional Sensor

A typical application of voltage divider which depicts a potentiometer voltage divider as a level sensor is a storage tank.

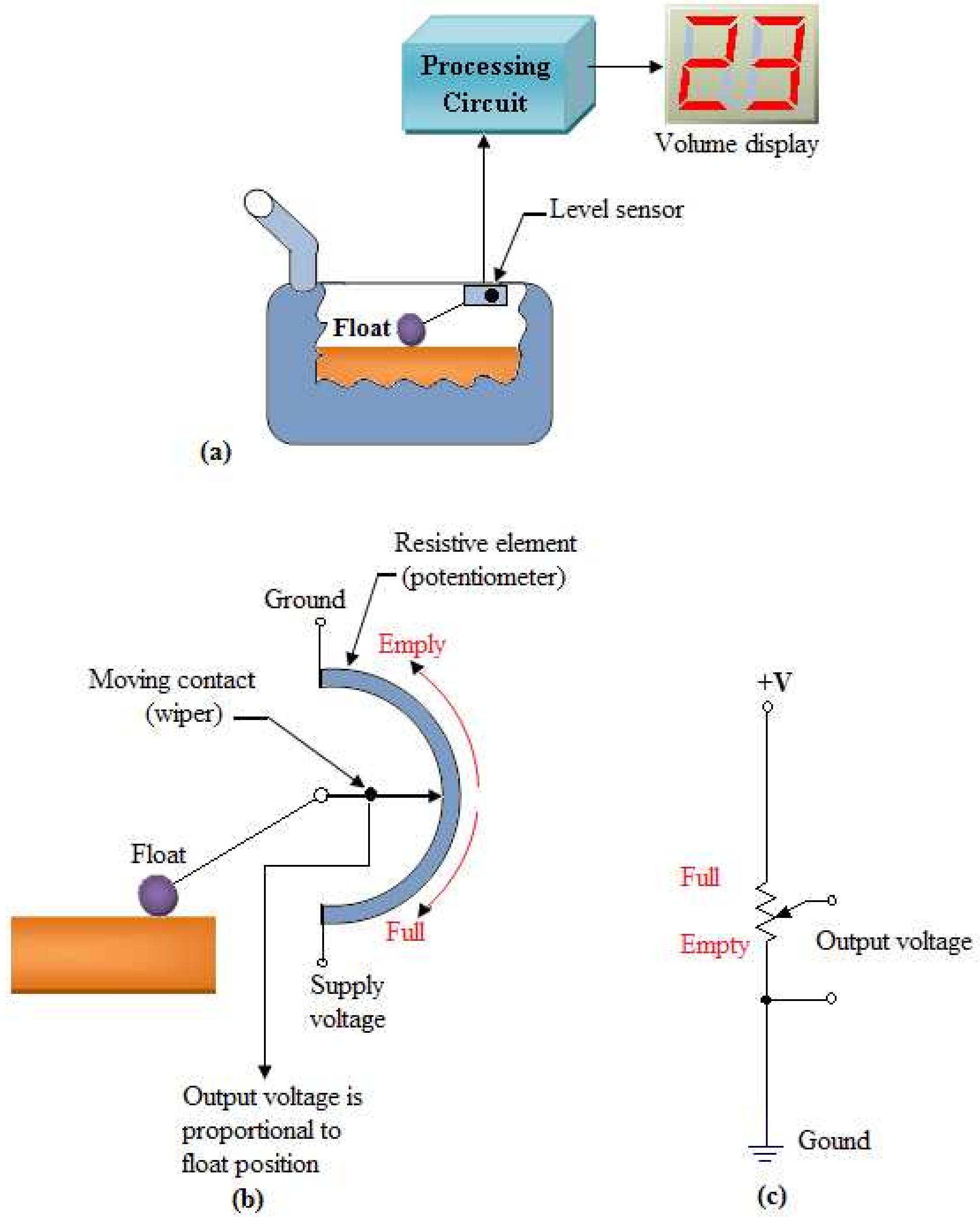


Figure 5-29: A potentiometer voltage divider used as level sensor; (a) Storage tank, (b) Detail of level sensor, (b) Schematic of level sensor

The float moves up as the tank is filled and moves down as the tank empties. The float is mechanically linked to the wiper arm of potentiometer. The output varies proportionally with the position of the wiper arm. As the liquid in the tank decreases, the sensor output voltage also decreases. The output voltage goes to the indicator circuitry, which controls a digital readout to show the amount of liquid in the tank.

5. 6. Light Emitting Diodes

Light emitting diode, LED for short, is a specialized type of PN junction diode that emits the light when forward biased. When the diode is forward biased, electrons from the semiconductors conduction band recombine with holes from the valence band releasing sufficient energy to produce photons which emit a monochromatic (single color) of light. Because of this thin layer a reasonable number of these photons can leave the junction and radiate away producing a colored light output. Then we can say that when operated in a forward biased direction Light Emitting Diodes are semiconductor devices that convert electrical energy into light energy.

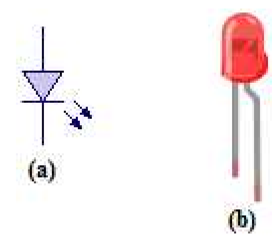


Figure 5-30: Light emitting diode, (a) Symbol, (b) Typical light emitting diode

Unlike normal incandescent lamps and bulbs which generate large amounts of heat when illuminated, the light emitting diode produces a «cold» generation of light which leads to high efficiencies than the normal «light bulb» because most of the generated energy radiates away within the visible spectrum. Because LEDs are solid-state devices, they can be extremely small and durable and provide much longer lamp life than normal light sources.

5. 6. 1. Light Emitting Diode Colors

Ordinary diodes are made for detection or power rectification, and are made from either germanium or Silicon semiconductor materials. On the other hands light emitting diodes are made from exotic semiconductor compounds such as Gallium Arsenide (GaAs), Gallium Phosphide (GaP), Gallium Arsenide Phosphide (GaAsP), Silicon Carbide (SiC) and Gallium Indium Nitride (GaInN) all mixed together at different ratios to produce a distinct wavelength of color.

Different LED compounds emit light in specific regions of the visible light spectrum and therefore produce different intensity levels. The exact choice of the semiconductor material used will determine the overall wavelength of the photon light emissions and therefore the resulting color of the light emitted.

Thus, the actual color of a light emitting diode is determined by the wavelength of the light emitted, which in turn is determined by the actual semiconductor compound used in forming the PN junction during manufacture. Therefore the color of the light emitted by an LED is not determined by the coloring of the LED's plastic body although these are slightly colored to both enhance the light output and to indicate its color when it is not being illuminated by an electrical supply.

Typical LED Characteristics			
Semiconductor Material	Wavelength	Color	Forward Voltage at 20mA
GaAs	850-940nm	Infra-Red	1.2v
GaAsP	630-660nm	Red	1.8v
GaAsP	605-620nm	Amber	2.0v
GaAsP:N	585-595nm	Yellow	2.2v
AlGaP	550-570nm	Green	3.5v
SiC	430-505nm	Blue	3.6v
GaInN	450nm	White	4.0v

Table 5-1: Various LED characteristics

5. 6. 2. LED Testing and Anode/Cathode Identification

When designing a circuit containing LED it is recommended first and foremost to locate the anode and cathode and test proper working of LED using methods shown in the experiment 5-4.

Experiment 5-4: LED Testing and Anode/Cathode Identification

- Light emitting diode (LED) is a small component used in almost every electronic device. LED has two terminals or legs. The longer leg is the anode or positive terminal and shorter leg is cathode or negative terminal.

- The method of identifying the leads from their length will not always work. Consider the case in which the legs of the LEDs are already cut into same length for using in a circuit. Even in such case the following method works. The one side is rounded shape and the other side has been made a little straighter. The leg in the straight side is always negative and the leg in the rounded side is always positive.

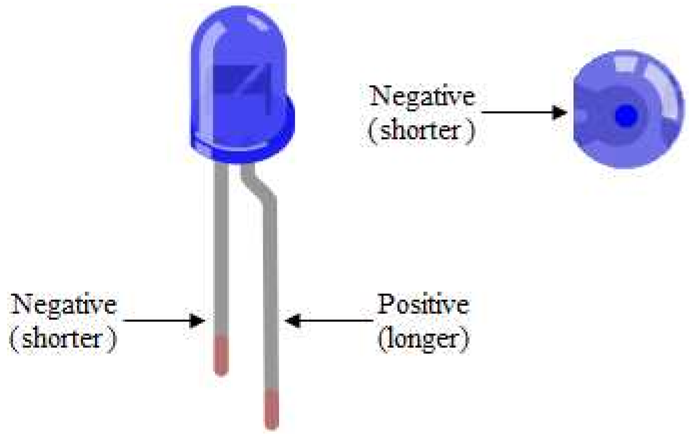


Figure 5-31: Identifying the anode and cathode of LED from length of legs and straighter side

- You can also look inside of the LED and see the larger electrode, which is cathode and the smaller electrode is the anode.

chap 5

- **Using digital multimeter**

A digital multimeter can be used to test LEDs. This must be equipped with a diode test function. For digital mustimeter to test different LEDs colors has to apply enough voltage potential at the meter leads to prevail over the forward voltage requirement of the LED. In their normal operation the diode test function typically applies 2 mA of current to the diode under test. With this current, enough voltage to overcome the forward bias requirement is usually enough to illuminate most LEDs.

Red, green, and yellow LED have the relatively low forward voltage thus, an ordinary digital multimeter can easily forward biases these color LEDs. On the other hands, blue and white LEDs require more applied voltage to overcome the higher barrier voltage potential. As seen in table, a typical red has a forward voltage of 1.8V and yellow, green have voltage requirements close to this value; that is 2.2V. A white LED requires 3.5V but it can actually begin to conduct at about 2.5 volts. For that reason as long as the digital multimeter applies at least 2.5 volts at the meter leads when in diode test mode, it may be used to test most of LEDs. This applied voltage may be verified by using a second digital multimeter in Voltage mode to measure the output of the meter set to diode test mode. The LED is then placed across the test leads (observing the correct polarity) and see if the LED is illuminated. At low voltage outputs the LED may only illuminate very dimly. Most modern digital multimeters are able of supplying sufficient voltage to forward bias most LEDs and therefore may be used to test individual LEDs

Procedure

1. Set the digital multimeter on the continuity mode. If you touch the testing leads to each other the multimeter will give a continuous beep sound. The beep sound means that the multimeter is working perfectly. Testing leads should be plugged into the multimeter as shown in the figure 5-32.

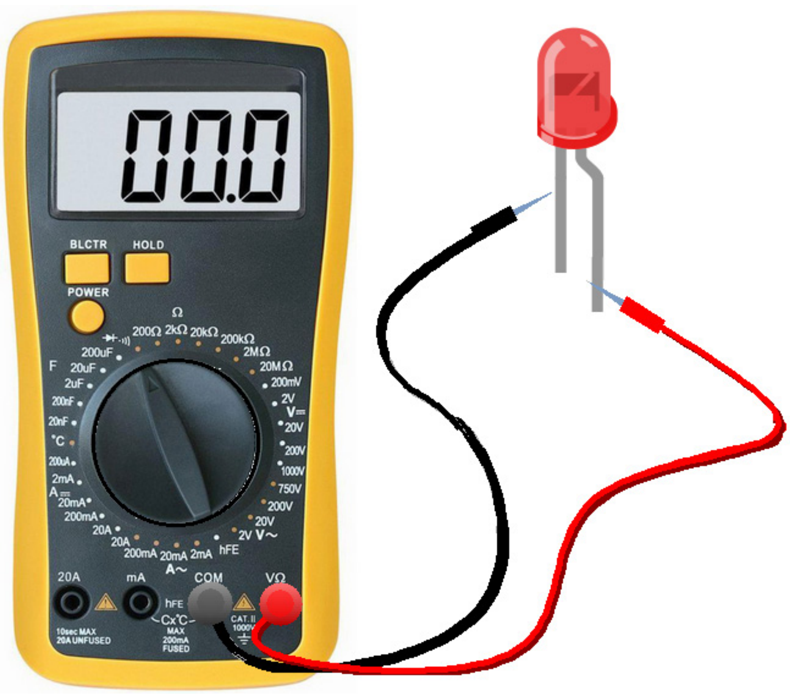


Figure 5-32: Figure: Testing LED using digital multimeter set in diode check

2. Longer leg is connected to the positive (red lead) and shorter to negative (black lead).
3. Switch on the multimeter and set the knob on the continuity mode.
4. LED will glow indicating that it is in working condition
5. If the LED doesn't glows then check the connections else the LED is faulty.

- **Using DC voltage source**

When digital multimeter is not delivering sufficient voltage to forward bias LED you can use a dc voltage source, just greater than 3.5V, and limiting resistor to test LED. A simple circuit as shown in figure 5-33 can perform such task. With anode connected to positive of the battery through a series resistor and cathode connected to negative of the battery the LED will light up. If the LED doesn't lit then check the connections else the LED is faulty.

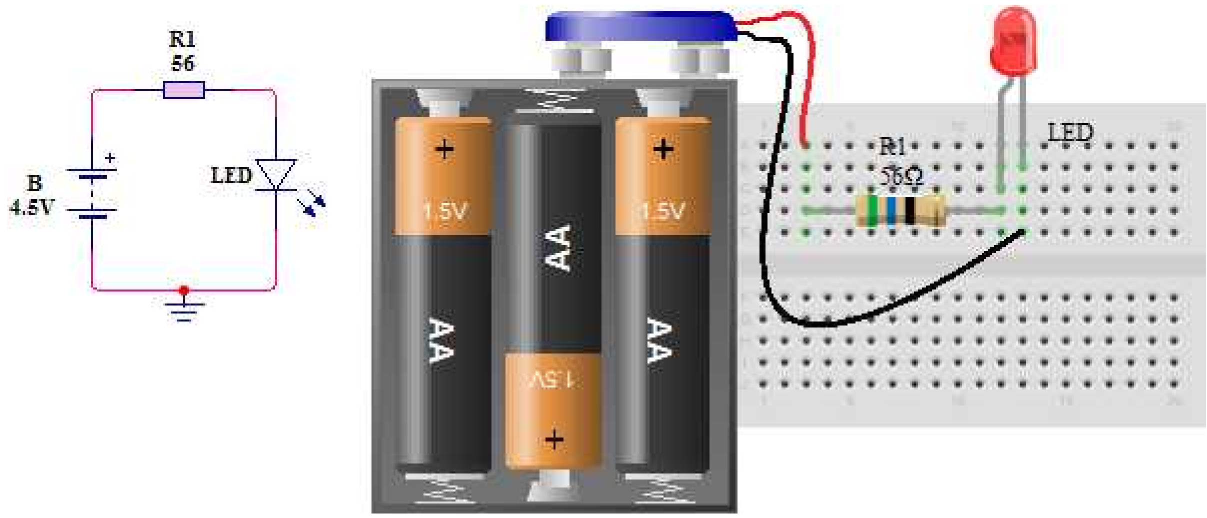


Figure 5-33: Testing LED from DC supply

Advantages of LED

Light Emitting Diodes have many advantages over traditional bulbs and lamps, with the main ones being,

- Their small in size, and hence can be regarded as point source of light. Because of their small size, several thousand LEDs can be packaged in 1 square metre area.
- The brightness of light emitted by LED depends on the current flowing though LED. Hence the brightness of light can be smoothly controlled by varying the current. This makes possible to operate LED displays under different amount of lighting conditions.
- Fast operating devices. They can be turned on and off in time less than 1 microsecond.
- Light in weight
- Available in various colours
- Long life
- Cheapness and readily available
- Easily to be interfaced with various other electronic circuits

- Some LEDs radiate infrared light which is invisible but still useful in some applications like thief alarm system
- Easily dimmed using pulse width modulation or by controlling the forward current
- Useful for the applications which are subjected to frequent on-off cycling. The fluorescent lamps burn out more quickly when cycled.
- Shock resistant and difficult to damage due to external shocks
- Not contain toxic material like mercury which is used in fluorescent lamps

Disadvantages of LED

The various disadvantages of LED are,

1. It draws considerable current requiring frequent replacement of battery in low power battery operated devices
2. Luminous efficiency of LEDs is low which is about 1.5lumen/watt
3. The characteristics are affected by temperature
4. Need large power for the operation compared to normal p-n junction diode

Applications of LED

Due to the advantages like low voltage, long life, cheap, reliable, fast on-off switching etc, the LEDs are used to many applications.

- The main advantage of light emitting diodes is that because of their small die size, several of them can be connected together within one small and compact package producing what is generally called a 7-segment display and all kind of visual displays i.e. alpha numeric displays. Such displays are commonly used in the watches and calculators
- LEDs are used in the optical devices such as optocouplers
- LEDs are also used as on-off indicator in various types of electronic circuits

5. 7. Optocoupler

The optocoupler, also called optoisolator, is a device that contains light emitting element and a photodetector (such as a photodiode, phototransistor, Darlington pair, SCR or triac) combined in one package. Optocoupler works basically as an interface between two circuits which operate at (usually) different voltage levels. The key advantage of an optocoupler is the electrical isolation between the input and output circuits. With an optocoupler, the only contact between the input and the output is a beam of light. Because of this it is possible to have an insulation resistance between the two circuits in the thousands of megohms. Isolation like this is useful in high voltage applications where the potentials of two circuits may differ by several thousand volts. Both the LED and photo-sensitive device are enclosed in a light-tight body or package with metal legs for the electrical connections as shown in figure 5-34.

The optical coupling method eliminates the need for a relay-controlled contact or an isolating transformer, which are traditional methods of providing electrical isolation between circuits. The optical coupling method is superior in many applications, because it gets rid of some of the less desirable features of relays and transformers.

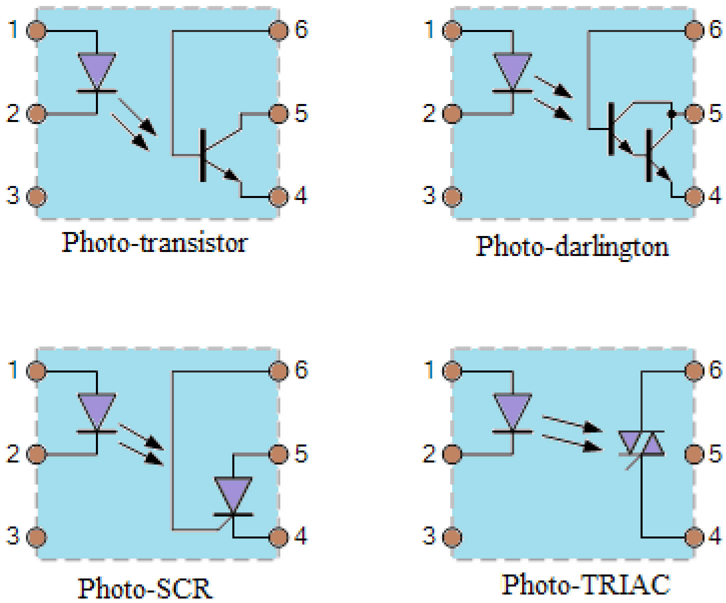


Figure 5-34: Symbols of optocouplers

Figure 5-35: Typical optocoupler IC

The optocouplers works well on either ac or dc high-voltage signals. For this reason, signal converters employing optical coupling are sometimes referred to the universal signal converters. Optocouplers typically come in a small 6-pin or 8-pin IC package, but are essentially a combination of two distinct devices: an optical transmitter and an optical receiver.

Some examples of optocouplers are given in table 5-2.

No	Part number	Description
1	PS2702-1-F3-A	PS2702 Series Single channel 40 V 3750 Vrms Darlington Photocoupler - SOIC-4
2	PS2701-1-F3-A	PS2701 Series Single Channel 40 V 3750 Vrms High Isolation Photocoupler - SOIC-4
3	MOC3023SR2M	DIP6 SMT 1 Channel 400 V 7500 V (Pk) Random Phase Triac Optoisolator
4	MOC3063SR2M	DIP6 SMT 1 Channel 600 V 7500 V (Pk) Zero Cross Phototriac Driver Optocoupler
5	FOD817A	DIP4 Through Hole Single Channel 70 V 5000 Vrms Phototransistor Optocoupler
6	MOCD217R2M	SOIC8 SMT Dual Channel 30 V 2500 Vrms Phototransistor Optocoupler
7	4N35M	DIP6 Through Hole Single Channel 30 V 7500 VAC (Pk) Phototransistor Optocoupler
8	H11AA1M	DIP6 Through Hole Single Channel 30 V 7500 VAC (Pk) Phototransistor Optocoupler

9	H11L1M	DIP6 Through Hole Single Channel High Speed 1 MHz 7500 VAC (Pk) Optocoupler
10	CNY17F2M	DIP6 Through Hole Single Channel 70 V 7500 Vrms Phototransistor Optocoupler
11	IL420	IL420 Series Single Channel 600 V 5300 Vrms Phototriac Optocoupler - DIP-6
12	MOC223	

Table 5-2: Some of the common optocouplers

5. 8. 7-segment display

The 7-segment display, also written as "seven segment display", or seven-segment indicator is a form of electronic display device that consists of seven LEDs (hence its name) arranged in a rectangular fashion for displaying decimal-numerals. Each of the seven LEDs is called a segment because when illuminated the segment forms part of a numerical digit (both Decimal and Hex) to be displayed. An additional 8th LED is sometimes used within the same package thus allowing the indication of a decimal point, (DP) when two or more 7-segment displays are connected together to display numbers greater than ten.

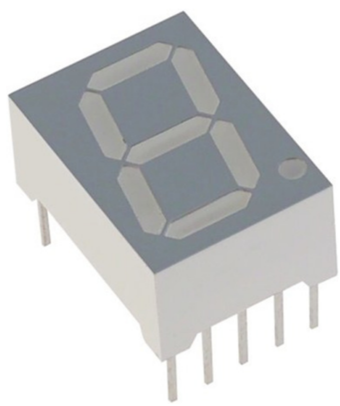


Figure 5-36: Typical 7-segment display

Each one of the seven LEDs in the display is given a positional segment with one of its connection pins being brought straight out of the rectangular plastic package. These individually LED pins are labelled from *a* to *g* representing each individual LED. The other LED pins are connected together and wired to form a common pin.

So by forward biasing the appropriate pins of the LED segments in a particular order, some segments will be light and others will be dark allowing the desired character pattern of the number to be generated on the display. This then allows us to display each of the ten decimal digits 0 through to 9 on the same 7-segment display.

The display common pin is generally used to identify which type of 7-segment display is. As each LED has two connecting pins, one called the "Anode" and the other called the "Cathode", there are therefore two types of LED 7-segment display called: Common Cathode (CC) and Common Anode (CA).

The difference between the two displays, as their name suggests, is that the common cathode has all the cathodes of the 7-segments connected directly together and the common anode has all the anodes of the 7-segments connected together.

- **The Common Cathode (CC)**

In the common cathode display, all the cathode connections of the LED segments are joined together to logic "0" or ground. The individual segments are illuminated by application of a "HIGH", or logic "1" signal via a current limiting resistor to forward bias the individual Anode terminals (a-g).

Example of the digital IC, commonly known as a 7-segment decoder/driver, which is used to drive the common cathode 7-segment is 45HC11.

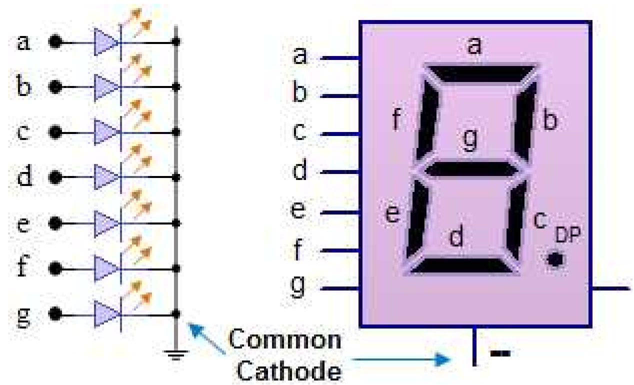


Figure 5-37: Common cathode 7-segment

chap 5

- **The Common Anode (CA)**

In the common anode display, all the anode connections of the LED segments are joined together to logic "1". The individual segments are illuminated by applying a ground, logic "0" or "LOW" signal via a suitable current limiting resistor to the Cathode of the particular segment (a-g). Example of the digital IC, commonly known as a 7-segment decoder/driver, which is used to drive the common anode 7-segment is 74HC47.

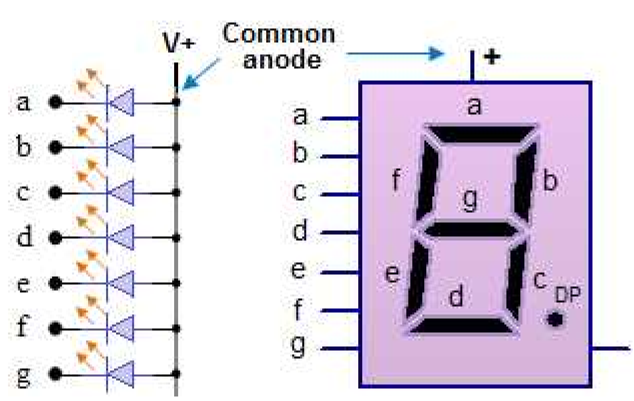


Figure 5-38: Common anode 7-segment

In general, common anode displays are more popular as many logic circuits can sink more current than they can source. Also note that a common cathode display is not a direct replacement in a circuit for a common anode display and vice versa, as it is the same as connecting the LEDs in reverse, and hence light emission will not take place.

Depending upon the decimal digit to be displayed, the particular set of LEDs is forward biased. For instance, to display the numerical digit 0, we will need to light up six of the LED segments corresponding to *a*, *b*, *c*, *d*, *e*, *f* and *g*. Then the various digits from 0 through 9 can be displayed using a 7-segment display as shown in figure 5-39.

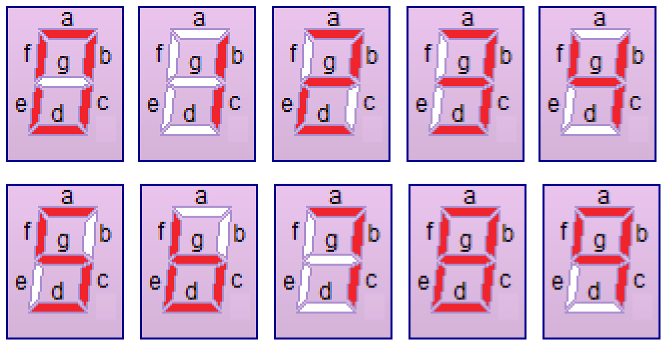


Figure 5-39: 7-Segment Display Segments for all Numbers

Thus for a 7-segment display, we can produce a truth table giving the individual segments that need to be illuminated in order to produce the required decimal digit from "0" through "9" as shown in table 5-3.

Decimal number	Binary number (input of decoder)	Individual Segments Illuminated						
		a	b	c	d	e	f	g
0	0000	on	on	on	on	on	on	off
1	0001	off	on	on	off	off	off	off
2	0010	on	on	off	on	on	off	on
3	0011	on	on	on	on	off	off	on
4	0100	off	on	on	off	off	on	on
5	0101	on	off	on	on	off	on	on
6	0110	off	off	on	on	on	on	on
7	0111	on	on	on	off	off	off	off
8	1000	on	on	on	on	on	on	on
9	1001	on	on	on	off	off	on	on

Table 5-3: 7-segment truth table

Although a 7-segment display can be thought of as a single display, it is still seven individual LEDs within a single package and as such these LEDs need protection from over current. A series resistor of a convenient value is required for a segment protection. LEDs produce light only when it is forward biased with the amount of light emitted being proportional to the forward current. So this forward current must be controlled and limited to a safe value by an external resistor to prevent damage to the LED segments. The forward voltage drop across a red LED segment is very low at about 2-to-2.2 volts, (blue and white LEDs can be as high as 3.6 volts) so to illuminate correctly, the LED segments should be connected to a voltage source in excess of this forward voltage value with a series resistance used to limit the forward current to a desirable value. Typically for a standard red coloured 7-segment display, each LED segment can draw about 15 mA to illuminated correctly, so on a 5 volt digital logic circuit, the value of the current limiting resistor would be about 200 (5V – 2V)/15mA, or 220 to the nearest higher preferred value.

Controlling a 7-segment display

The 7-segment display can be driven by individual switches or by special IC commonly called BCD to 7-segment display decoder.

Refer to the circuit example figure 5-40 where the segments of a common anode display are illuminated using the switches. If switch "a" is closed, current will flow through the "a" segment of the LED to the current limiting resistor connected to pin "a" and to 0 volts, making the circuit. Then only segment "a" will be illuminated. So a LOW condition (switch to ground) is required to activate the LED segments on this common anode display.

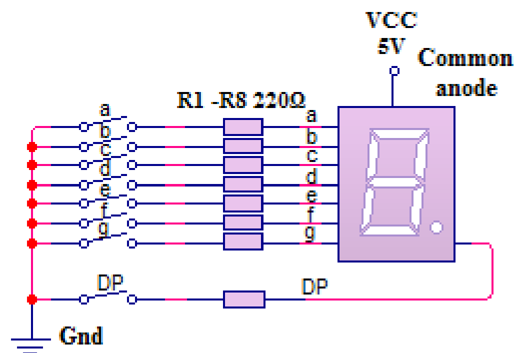


Figure 5-40: Driving a 7-segment display using switches

chap 5

In case we want the decimal number "5" to illuminate on the display the switches "a", "f", "g", "c" and "d" would be closed to light the corresponding LED segments. Likewise for a decimal number "3", switches "a", "b", "c", "d" and "g" would be closed. But illuminating 7-segment displays using individual switches is not very practical. In case of digital counters a special IC is used to drive 7-segment.

7-segment Displays are usually driven by a special type of integrated circuit (IC) commonly known as a 7-segment decoder/driver.

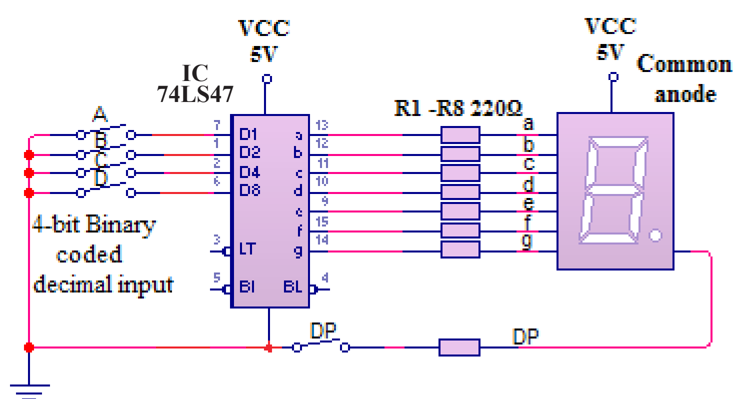


Figure 5-41: 7-segment driven by a BCD to 7-segment decoder

The 7-segment display driver is known as a Binary Coded Decimal or BCD to 7-segment display decoder or driver and there are those designed to drive common anode or common cathode displays. A BCD-to-7 segment decoder/driver takes a four-bit BCD input labelled A, B, C and D for the digits of the binary weighting of 1, 2, 4 and 8 respectively, has seven outputs that will pass current through the appropriate segments to display the decimal digit of the numeric LED display.

Testing and identifying 7-segment's pin

When you purchase a 7-segment display from market and you want to check that your display is working properly or not, or sometime when we do projects based on seven segment displays and our assembled circuit is alright but display doesn't work so in this case you should test the 7-segment display.

Experiment 5-5: Testing and identifying 7-segment's pins

In principle before starts project on s7-segment display, the same to all components made up the circuit, you should check if the 7-segment display is working properly or not. Here below are guidelines to test the 7-segment display.

Part list

No	Items	Quantity
1	Digital multimeter with diode check settings	1
2	7-segment display	1

Procedure

1. Identify the common pin (3, 8) and input pins (1, 2, 4, 5, 6, 7, 9, and 10) of seven segment display.

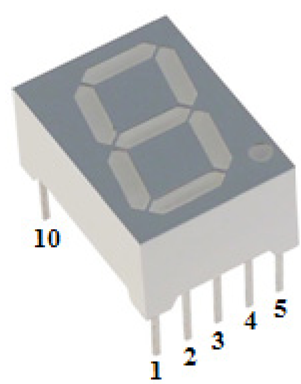


Figure 5-42: 7-segment pin numbering scheme

2. Set the multimeter in diode check mode and assume that red lead is positive and black lead is negative.
3. Connect the black lead to common pin (3 or 8) of multi-meter because both pin are connected each other internally. And connect the red lead to input pin (1, 2, 4, 5, 6, 7, 9 and 10) of multimeter.

4. If any of the LED segment is glow then your display is common cathode. And check all the LED segment of common cathode to ensure that your display is working properly or not.
5. If none of the seven segment glows then interchange the multi-meter lead connection. It means connect the red lead to common pin (3 or 8) of display and black lead to input pin (1, 2, 4, 5, 6, 7, 9, and 10) of display.
6. If any of the LED segment is glow then your display is common anode. And check all the segment of common anode to ensure that your display is working properly.
7. If none of the LED segment glow then your display is faulty or not working properly.

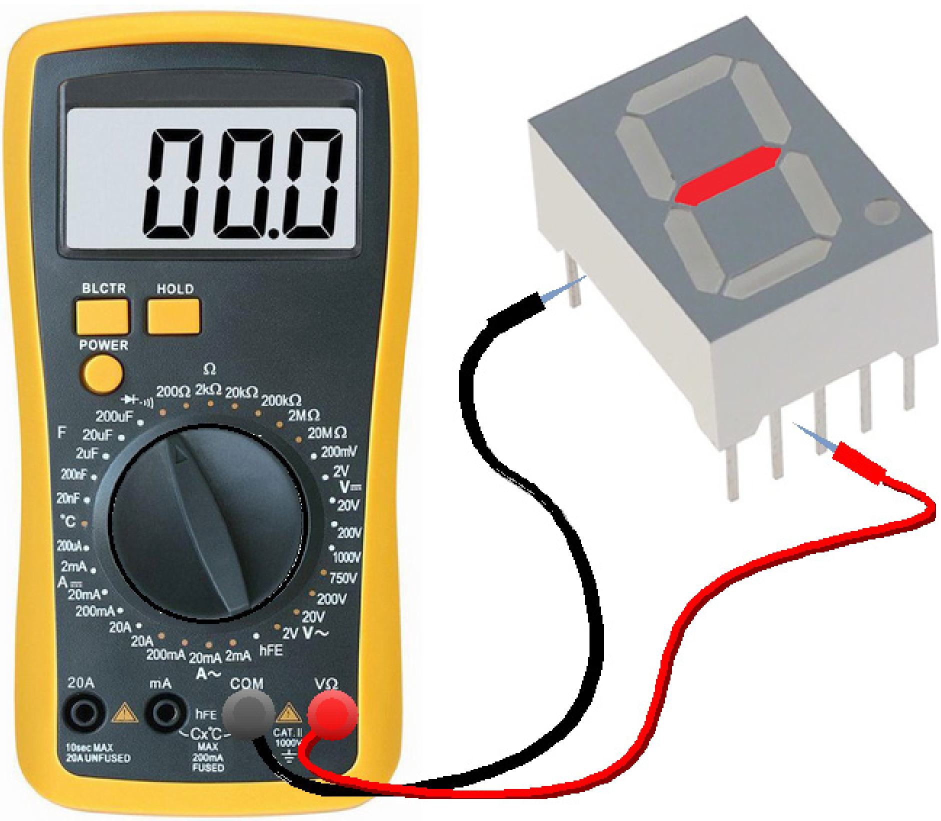


Figure 5-43: Testing 7-segment using digital multimeter's diode check

Note that for testing the 7-segment that requires voltage greater than produced by digital multimeter, a dc voltage source can be used in place of multimeter; here by connecting a resistor in series with the pin to be tested and by considering the positive terminal of the battery as the red lead of the digital multimeter, and the negative terminal of the battery as the black lead of the digital multimeter.

Transistor and Applications

Chapter six

Objectives

After completing this chapter, you should

- Have an understanding of transistor operation, classification and transistor testing
- Have an understanding of transistor working as switch and amplifier
- Have an understanding of transistor pulse generators, (astable, bistable and monostable) and their applications
- Be able to design different circuit projects based around transistor as switch
- Be able to design transistor amplifier, (CE, CC and CB amplifier) and their applications

chap 6

Further reading

Study aids for this chapter are available at

- Amos S W & James M R (1999). *Principles of Transistor Circuits*. Butterworth-Heinemann.

The transistor is the most important example of an active component, a device that can amplify, producing an output signal with more power in it than the input signal, and that can work as switch. The transistor input power comes from an external source of power (the power supply and signal generator, to be exact). Note that voltage amplification isn't what matters, since, for example, a step-up transformer, a "passive" component just like resistor or capacitor, has voltage gain but no power gain. It is interesting to note that the property of power amplification seemed very important to the inventors of transistor. The transistor is essential ingredient of every electronic circuit, from simplest amplification or oscillator to the most elaborated digital computer. Integrated circuits (ICs), which have largely replaced circuits constructed from discrete transistors, are themselves merely arrays of transistors and other components built from a single chip of semiconductor material. A good understanding of transistors is very important, even if nowadays most of circuits are made from ICs, because you need to understand the input and output properties of the IC in order to connect it to the rest of the circuit and to the outside the world. In addition, the transistor is the single most powerful resource for interfacing, whether between ICs and other circuitry or between one subcircuit and other. Finally, there are frequent situation where the right IC just doesn't exist, and you have rely on discrete transistor circuitry to do the job. So this chapter will describe the basic operation of transistor which will give an openness to the use of it to make a desired circuit project.

6. 1. Transistor Overview

A *transistor* is a semiconductor device that controls current between two terminals based on the current or voltage at a third terminal and is used for the amplification or switching of electrical signals. There are basically two types of transistor: Bipolar Junction Transistor (BJT) and Field Effect Transistor (FET).

(a) (b)

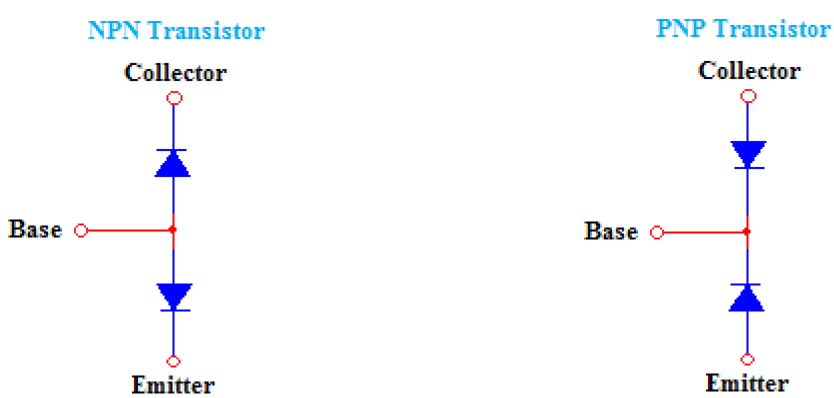
Figure 6-1: Transistor symbol; (a) Bipolar transistor symbols, NPN and PNP, (b) Field effect transistor symbols, N-channel and P-channel

Bipolar junction transistor has three terminals: Base, Collector and Emitter

- Emitter (E): It is more heavily doped than any of the other regions because its main function is to supply majority charge carries (either electrons or holes) to the base.
- Base (B): It forms the middle section of the transistor. It is very thin as compared to either the emitter or collector and is very lightly-doped.
- Collector (C): Its main function (as indicated by its name) is to collect majority charge carriers coming from the emitter and passing through the base.

The Bipolar Transistor basic construction consists of two PN-junctions.

A bipolar junction transistor is analogous to just pair of connected back to back diodes (PN junction) producing three connecting terminals with each terminal being given a name to identify it from the other two. These three terminals are known and labelled as the Emitter (E), the Base (B) and the Collector (C) respectively. Thus there are PNP and NPN BJT transistors.



(a) (b)

Figure 6-2: BJT diode equivalent; (a) NPN transistor, (b) PNP transistor

The symbol does not indicate if the transistor is small-signal, high power or the type of package. It just indicates the transistor is PNP or NPN. The arrow always indicates the emitter, and the arrow pointing toward the base for NPN and pointes out of the base for PNP transistors. This is how to remember the symbol.

The principle of operation of the two transistor types PNP and NPN, is exactly the same the only difference being in their biasing and the polarity of the power supply for each type. For NPN, emitter is connected to ground and collector to positive of the supply while for PNP emitter is connected to positive of the supply and collector to ground of the supply.

6. 1. 2. Transistor Biasing

For proper working of a transistor, it is essential to apply voltages of correct polarity across its two junctions. It is meaningful to remember that for normal operation;

- *Emitter-base junction* is always forward biased, and
- *Collector-base junction* is always reverse- biased.

6. 1. 3. Basic Transistor Parameters

When searching for an appropriate transistor replacement some of the basic transistor parameters that need to be considered include the following:

1. Semiconductor material used: Most transistors will either be germanium or silicon. Other types are normally only used in very specialist applications. It is important to know what type the transistor is because there is a difference in the base emitter forward bias voltage drop. For germanium it is around 0.2 - 0.3 volts and for silicon it is around 0.7 volts. The circuit will be designed around a particular voltage drop.

2. Polarity: It is absolutely imperative to find out whether the transistor is either NPN or PNP variety. Install the incorrect type and it experience the inverse of all the voltages it would expect and is likely to be destroyed.

3. General application: Although it is not always necessary to exactly match the intended purpose for the transistor, a variety of areas of its performance will be tailored to its intended applications. Possible application types may include: switching, analog, low power, RF amplifier, low noise, etc. Put in the correct type and it may not perform well. For example a low power general-purpose transistor is unlikely to work well in a switching application even if it has a high *ft* or frequency limit.

4. Package and pin-out: Transistors have many packages. It is often necessary to match the replacement transistor package as closely as possible to enable the transistor to physically fit. Also the package may give an indication of other parameters.

5. Voltage breakdown: It is necessary to make sure that the transistor is able to withstand the voltages it is likely to see. Transistor parameters such as V_{ceo}, etc. need to be checked.

6. Current gain: The current gain parameter of a transistor normally has a very wide spread. This is normally quoted as β or hfe. Although they are slightly different, for all circuit equivalences of this nature these transistor parameters are the same. Choosing a replacement transistor with approximately the same current gain is necessary. Normally it is not a problem to choose a replacement transistor with a higher gain. Often a lower current gain may be acceptable.

7. Frequency limit: The upper frequency limit for a transistor is normally quoted as its *ft*. It is normally important to ensure that the transistor can meet any frequency limits.

8. Power dissipation: It is necessary to ensure that the replacement transistor can dissipate sufficient power. Often the package type is a good indication of this.

These are the main parameters that are of importance in most applications, but be on the lookout for any other transistor parameters that may need to be included in the selection of the replacement transistor.

6. 2. Transistor Replacement

When choosing a suitable replacement transistor for use within an electronic circuit, there are several stages that must be considered when making the choice. These can be progressed in a logical order to narrow down the choice and enable the best alternative for the replacement transistor to be made.

Step by step instructions in replacement of a transistor

1. Choose a transistor of the same polarity: The first major selection criterion is whether the transistor is PNP or NPN.

2. Select a replacement transistor of the same material: Most transistors are either silicon or germanium. As bias voltages and other features are different it is necessary to select a replacement transistor with the same material.

3. Select the same functional type of transistor: Transistors are normally given an indication of their application in the datasheets. The replacement should have the same application if possible.

4. Choose a replacement with the same package: Choosing a replacement transistor with the same package and pin-out will mean that many of the characteristics including power capability are the same. Ensuring with pin-out is the same (most but not all transistors have their leads in order - EBC) will save many problems with fitting.

5. Select a replacement transistor with the same breakdown voltage: Ensure that figures for V_{CEO} and V_{CBO} etc. are at least as high as the original transistor.

6. Check if it can take the current: Ensure that the replacement transistor can pass the required current. It should have an IC_{max} greater than or equal to the original transistor.

7. Select a transistor with a similar hfe: It is necessary to ensure that the current gain of the replacement transistor is about the same as the original. Current gain values normally vary widely even for transistors of the same type so some variation will be acceptable.

8. Select a replacement transistor with equivalent *ft*: It is necessary to ensure that the replacement transistor will be able to operate at the relevant frequencies, so a similar or slightly higher Ft is advisable. Do not go for a transistor with a much higher *ft* as this may increase the risk of oscillation.

9. Choose a transistor with similar power dissipation: It is necessary to ensure that the replacement transistor can handle the power that it will dissipate within the circuit. Choosing a replacement transistor with a similar can style will often mean that both transistors have similar power dissipation.

10. Check for any special features: While ensuring the features above are selected, there may be some additional features that need to be considered. These are normally required when transistors are used in specialist applications.

6. 3. Transistor Choice Preference

Nowadays the bipolar junction transistor (BJT) is the most commonly used transistor even if FETs and MOSFETs are becoming widely available the BJT remain the transistor of choice for many analog circuits such as amplifiers because of their greater linearity and ease of manufacture. In integrated circuits, the desirable properties of MOSFETs allowed them to capture nearly all market shares for digital circuits. Discrete MOSFETs can be applied in transistor applications, including analog circuits, voltage regulators, amplifiers, power transmitters and motor drivers. Consequently, on market we find different types of transistors and the choice depend on the desired application and the characteristics of transistor. Transistors are categorized by:

- **Semiconductor material used**
- **Structure:** BJT, JFET, IGFET (MOSFET), insulated-gate bipolar transistor, etc.
- **Electrical polarity** (positive and negative): NPN, PNP (BJTs); N-channel, P-channel (FETs)
- **Maximum power rating:** low, medium, high
- **Maximum operating frequency:** low, medium, high, radio (RF), microwave frequency (the maximum effective frequency of a transistor is denoted by the term f_T , an abbreviation for transition frequency. The frequency of transition is the frequency at which the transistor yields unity gain)
- **Application:** switch, general purpose, audio, high voltage, super-beta, matched pair
- **Physical packaging:** through-hole metal, through-hole plastic, surface mount, ball grid array, power modules.
- **Amplification factor** h_{fe}, β_F (transistor beta) or g_m (transconductance).

chap 6

6. 4. Transistor Part Number Specifications

Semiconductor devices are classified by the manufacturer using a unique part numbering system. The types of some transistors can be parsed from the part number. There are many systems in use but here are three major semiconductor naming standards; the American based JEDEC system, the European based Pro-electron system and the Japanese based JIS system; in each the alphanumeric prefix provides clues to type of the device. Major manufacturers introduce their own schemes as well.

6. 4. 1. JEDEC Numbering System

JEDEC (Joint Electron Devices Engineering Council) has the following format:

Digit, Letter, Serial number, [Suffix]

- **Digit**

The first digit designates the amount of P-N junctions in the device. So a device starting with "2" would contain 2 P-N junctions and would most likely be either a BJT transistor or a FET. The JEDEC EIA370 transistor device numbers usually start with "2N", indicating a three-terminal device (dual-gate field-effect transistors are four-terminal devices, so begin with 3N), then a 2, 3 or 4-digit sequential number with no significance as to device properties (although early devices with low numbers tend to be germanium).

Common part numbers are listed below:

1. Diodes
2. Bipolar transistors or Field Effect Transistors
3. Double Gate MOSFETS, SCRs
4. Opto-Couplers

- **Letter**

The letter is always "N", and the remaining figures contain the device serial number.

- **Serial Number**

The serial number runs from 100 to 9999 and indicates nothing about the transistor.

- **Suffix**

A letter suffix (such as "A") is sometimes used to indicate a newer variant, but rarely gain groupings. If a suffix is present then this indicates the gain group as below:

A: Low gain
B: Medium gain
C: High gain

No suffix: Ungrouped (any gain).

So for example, 1N4001 is a diode, 2N2102 is amplifier NPN silicon transistor; 2N2222 is an NPN switching transistor, 2N5655 is a plastic NPN high-voltage power transistor and 3N201 would be a double gate MOSFET.

6. 4. 2. Pro-electron Numbering System

This system has the following format:

Two letters, [letter], Serial number, [Suffix]

- **The first letter indicates the semiconductor material type**

A: Germanium
B: Silicon
C: Gallium Arsenide
R: Compound Materials

- **The second letter specifies the type of device**

A: Diode, low power or signal
B: Diode, variable capacitance
C: Transistor, audio frequency low power
D: Transistor, audio frequency power

E: Diode, tunnel
F: Transistor, high frequency low power
G: Miscellaneous devices
H: Diode, sensitive to magnetism
K: Hall Effect device
L: Transistor, high frequency power
N: Photocoupler
P: Light detector
Q: Light emitter
R: Switching device, low power e.g. thyristor, diac, unijunction etc.
S: Transistor, low power switching
T: Switching device power, e.g. thyristor, triac, etc.
U: Transistor, switching power
W: Surface acoustic wave device
X: Diode, multiplier, e.g. varactor
Y: Diode, rectifying
Z : Diode, voltage reference

- **Third Letter**

If present this indicates that the device is intended for industrial or professional rather than commercial applications. It is usually a W, X, Y or Z. Examples- BFY51.

- **Serial Number**

The serial number runs from 100 to 9999. A 3-digit sequence number (or one letter then 2 digits, for industrial types) follows. With early devices this indicates the case type.

- **Suffix**

Suffixes may be used, with a letter.

If a suffix is present then this indicates the gain group as below:

A: Low gain
B: Medium gain
C: High gain (such as in: BC548C)
No suffix: Ungrouped (any gain)

Other codes may follow to show gain (e.g. BC327-25) or voltage rating (e.g. BUK854-800A).

More common prefixes are shown in table 6-1.

Prefix class	Type and usage	Example
AC	Germanium small-signal AF transistor	AC126
AD	Germanium AF power transistor	AD133
AF	Germanium small-signal RF transistor	AF117
AL	Germanium RF power transistor	ALZ10
AS	Germanium switching transistor	ASY28
AU	Germanium power switching transistor	AU103
BC	Silicon, small-signal transistor (“general purpose”)	BC548
BD	Silicon, power transistor	BD139
BF	Silicon, RF (high frequency) BJT or FET	BF245
BS	Silicon, switching transistor (BJT or MOSFET)	BS170
BL	Silicon, high frequency, high power (for transmitters)	BLW60
BU	Silicon, high voltage (for CRT horizontal deflection circuits)	BU2520A
CF	Gallium Arsenide small-signal Microwave transistor (MESFET)	CF739
CL	Gallium Arsenide Microwave power transistor (FET)	CLY10

Table 6-1: Common transistor prefixes in pro-electron numbering system

6. 4. 3. JIS Numbering System

The Japanese Industrial Standard has the following format:

chap 6

Digit, Two letters, Serial number, [Suffix]

- **Digit**

This indicates the amount of PN junctions as in the JEDEC code. The JIS-C-7012 specification for transistor part numbers starts with “2S”.

- **Letters**

The letters indicate the intended application for the device according to the following code:

Letter code	Intended application	Letter code	Intended application
SA	PNP HF transistor	SK	N-channel FET/MOSFET
SB	PNP AF transistor	SM	Triac
SC	NPN HF transistor	SQ	LED
SD	NPN AF transistor	SR	Rectifier
SE	Diodes	SS	Signal diodes
SF	Thyristors	ST	Diodes
SG	Gunn devices	SV	Varicaps
SH	UJT	SZ	Zener diodes
SJ	P-channel FET/MOSFET		

Table 6-2: The letter codes indicating the intended application for the device in JIS numbering system

- **Serial Number**

The serial number runs from 10 to 9999.

- **Suffix**

This series sometimes has suffixes (such as “R”, “O”, “BL”... standing for “Red”, “Orange”, “Blue” etc.) to denote variants, such as tighter h_{FE} (gain) groupings. Also the (optional) suffix indicates that the type is approved for use by various Japanese organizations.

For example 2SC2873 is NPN silicon high performance switching transistor. Nevertheless, sometimes the “2S” prefix is not marked on the package; a 2SD965 might only be marked “D965”; a 2SC1015 might be listed by a supplier as simply “C1015”.

6. 5. Transistor testing and PIN Identification

When testing transistor you have to assume that a transistor is just a pair of connected diodes. Therefore it can be tested for shorts, open or leakage with analog or digital multimeter. These are frequently tests but you can also test the gain, frequency response, etc. with specialized instruments.

Things to remember: Some power transistors have built in damper diodes connected across collector-emitter and resistors connected across base-emitter which will confuse the reading. In addition, a few small signal transistors have built-in resistors in series with the base or other leads, making this simple test method useless. Darlington transistors can also show unusual voltage drops and resistances.

When testing a transistor of this type you will need to compare with a known good transistor or check the specifications to be sure.

6. 5. 1. Diode Check Function

Take the transistor out of circuit and set digital multimeter in diode check mode. This test is for already know base.

- Connect the red (positive) lead to the base of the transistor. Connect the black (negative) lead to the emitter. A good NPN transistor will read a junction drop voltage of 0.4V to 0.9V. A good PNP transistor will read open.
- Leave the red meter lead on the base and move the black meter lead to the collector, the reading should be almost the same as the previous test, open for PNP and a slightly lower voltage drop for NPN transistors.
- Reverse the meter leads and repeat the test. This time, connect the black meter lead to the base of the transistor and the red lead to the emitter. A good PNP transistor will read a junction drop voltage of 0.4V to 0.9V. A good NPN transistor will read open.
- Leave the black meter lead on the base and move the red lead to the collector. The reading should be almost the same as the previous test, open for NPN and a slightly lower voltage drop for PNP transistors.
- Place one meter lead on the collector, the other on the emitter, then reverse. Both tests should read open for both NPN and PNP transistors.

It is also practical to test a bipolar transistor with an analog multimeter using the low ohms scale. Only two of the six possible combinations (the B-E and B-C junctions in forward bias) should show a low resistance, anywhere from 100 ohms to several Kilo-ohms, and none of the resistances should be near 0 Ohms.

If you read a short circuit (zero ohms or a voltage drop of zero) between two leads, or the transistor fails any of the tests described above, it is bad and must be replaced.

If you get readings that do not make sense, try to compare them with measurements done on a good transistor of the same type.

Some analog multimeters have their probe colors reversed since this makes the internal circuitry easier to design. So, it is a good idea to confirm and label the lead polarity of your instrument by making a few measurements in resistance or diode test mode (DMM) using a known good diode. This will also show you what to expect for a reading of a forward biased junction.

Experiment 6-1: Identify collector and emitter of transistor using diode check of DMM and with analog multimeter

Identify the collector and emitter of C1815 and 2N3905 knowing that the base for C1815 is pin 3 and for 2N3905 is pin 2.

Part list

No	Item	Quantity
1	Digital multimeter DMM with diode check function	1
2	C1815 transistor	1
3	2N3905 transistor	1
4	Analog multimeter	1

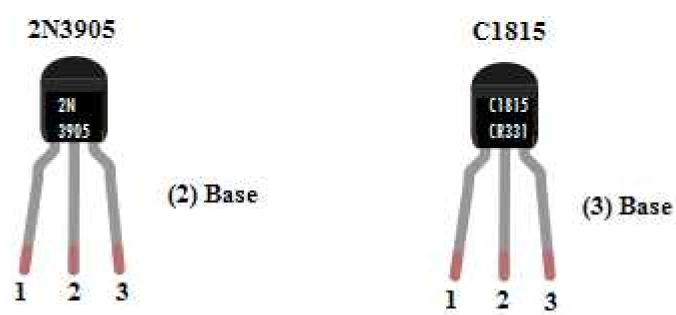


Figure 6-3: Typical 2N3905 and C1815 transistor

- **Determining Collector and Emitter with DMM**

Procedure

1. Set digital multimeter in diode check mode (continuity tester) and connect the red lead of multimeter to pin 3 and black lead to pin 2 of C1815 transistor. The reading is approximately 0.727. Maintain red lead of multimeter to pin 3 and move black lead to pin 1, now the reading is slightly greater than the prior and is about 0.730, this is the emitter.

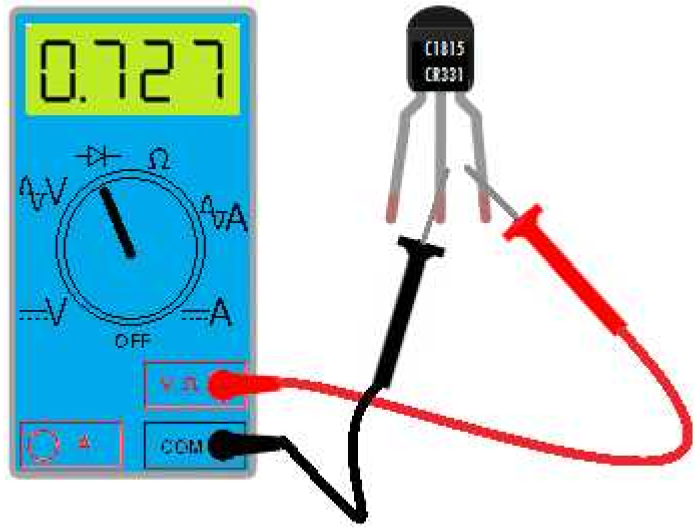


Figure 6-4: Identifying collector and emitter of C1815 using diode check mode

2. Change the transistor to 2N3905 and repeat step 1 but maintain the red lead to pin 2 (base). As result, the meter reading is 0L (open). Therefore, C1815 is NPN, pin 3 is the base, pin 2 is collector and pin 3 is emitter.
3. Since in step 2 we got open (0L) while testing 2N3905, we need to reverse the meter leads and hold black lead for two readings. The base for this transistor is pin 2.
4. Connect the black lead of multimeter to pin 2 and the red lead to pin 3 of 2N3905 transistor. The meter reading is about 0.682. Maintain the black lead to pin 2 and move the red lead of meter to pin 1, the reading is a little bit greater than the previous and is 0.686. This is the emitter.
5. Change the transistor to C1815 and repeat step 4 considering pin 3 as the base. Now the meter reading is 0L (open) for two combinations. As result, 2N3905 is a PNP transistor with pin 2 the base, pin 3 the collector and pin 1 the emitter. Note that the complement of C1815 is C1015 where you need only to reverse the meter leads without changing the connection.

- **Determine the base and the type of transistor with analog multimeter**

All analog multmeters have one or more Ohms ranges. The lowest range is called the “Ohms Range” as the scale on the meter is read directly. For instance, “500” on the scale is 500 Ohms (500R). The other range is the x1k range. “500” on the scale is read 500k. This is the setting we use for the diode tests.

Remember that inside the multimeter there is a battery (1.5V or 3V) and this provides the energy to move the needle. Note that the red probe of a multimeter is connected to negative of the battery (inside the multimeter) and the black probe is connected to the positive of the battery (via a set of resistors and the meter-movement itself).

When the black probe is connected to the anode of a diode and the red probe to the cathode, as shown in the figure 6-5, the needle moves about 90% across the dial. (It does not move fully across because the multimeter is actually detecting the voltage-drop of 0.7v of the diode and not its actual resistance.

When the red probe is connected to the anode and the black probe to the cathode, the needle does not move at all.

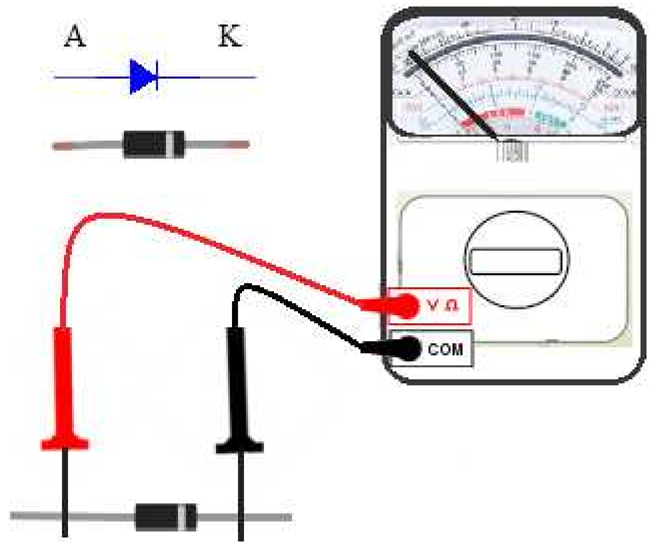


Figure 6-5: Analog multimeter set to determine anode and cathode of diode

In the first case the diode is forward biased and current flows. In the second case the diode is reverse biased and no current flows. The pointer (needle) clearly indicates these two states. These are the two conditions we need to remember. Also remember to check the ohm's scale the multimeter is set, this should be on "x1K" scale.

The above test gives idea on how analog multimeter reacts to a diode in forward and reverse bias now we can test a transistor and determine the base pin.

Procedure:

1. Place the black probe on any pin of C1815 transistor. Then place the red probe on each of the other leads. If the needle moves across the dial, the transistor is NPN. If the needle moves for only one test, try the black probe on another lead. This may take up to 6 tests to get a final answer. The pin at which you maintain the black probe for needle movement on other two pins is the base; that is pin 3.

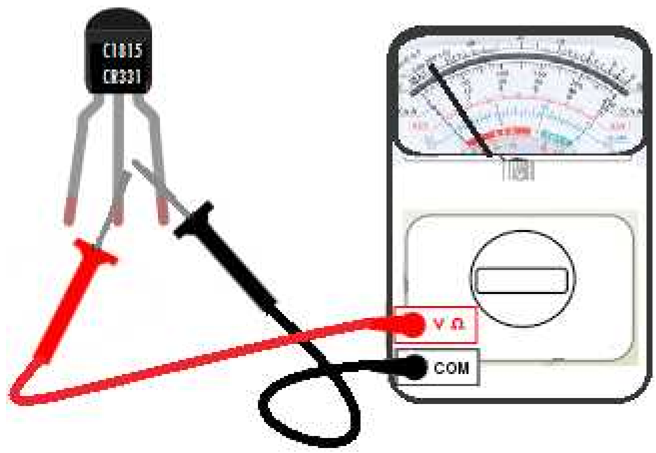


Figure 6-6: Determining the base and type of transistor using analog multimeter

2. If the pointer doesn't move twice, place the red probe on any pin and repeat the above. When the needle moves for both the other pins, the transistor is PNP and the pin at which you held red probe for needle movement on other two pins is the base; that is pin 2 for 2N3905 transistor.

Note: - If the needle does not move for the two other leads, the transistor is faulty. It is "OPEN."
- If the needle moves for all tests, the transistor is faulty. It is "SHORTED."
- If the needle moves slightly for one of the tests, the transistor is "LEAKY."

Other faults such as heat stress, over-voltage breakdown, high-frequency failure, or intermittent breakdown can occur and special testing equipments are required.

6. 5. 2. Testing Unknown Transistor

By assuming a bipolar transistor as a pair of connected diodes, the type (NPN or PNP) and the pin arrangement of unmarked transistors can be determined straightforwardly using a digital or analog multimeter. The collector and emitter can be identified knowing the fact that the doping for the base-emitter (B-E) junction is always much higher than for the base-collector (B-C) junction, therefore, the forward voltage drop for B-E junction will be slightly higher. This will show up as a couple of millivolts difference on a digital multimeter's diode test mode or a slightly higher resistance on an analog Volt-Ohm-Meter.

First make a few measurements between various pins. Soon you will identify a pin (the Base) that will show a forward voltage drop on digital multimeter or a low resistance on an analog multimeter combined with other two pins (the Emitter and Collector). Now that the Base is identified, observe carefully the voltage drops across B-E and B-C. The B-C junction will have a slightly less voltage drop on digital multimeter or a slightly lower resistance when using an analog ohmmeter.

From six combination tests leads, you already know the polarity of the transistor under test. Remember that the base is shared terminal between the emitter and collector. For NPN the base is positive while negative for PNP. Thus, if the negative lead (black lead connected to the COM on most digital multimeters) is placed on the base when measuring the B-C and B-E voltage drops you have a PNP transistor. In similar way, if the positive (red) meter lead is placed on the base, you have a NPN transistor.

It is a good habit to test every transistor before placing it into the circuit, as the datasheet is not always at hand, and misplacing the pins can have devastating results.

Things to remember: For every degree the transistor increases in temperature, the diode drops will decrease by a few millivolts. This change can be confusing when determining the B-E and B-C junctions. So, make sure you do not hold the transistor under test in your hand and leave enough time for it to cool down to room temperature after soldering.

Experiment 6-2: Transistor pinout

Identify the pins and the type of an unlabeled transistor. This experiment is important because transistor packaging, unfortunately, is not standardized. All bipolar transistors have three wires, of course, but the positions of the three wires on the actual physical package are not arranged in any universal, standardized order.

Part list

No	Item	Quantity
1	Digital multimeter with diode check function	1
2	Bipolar transistor	1

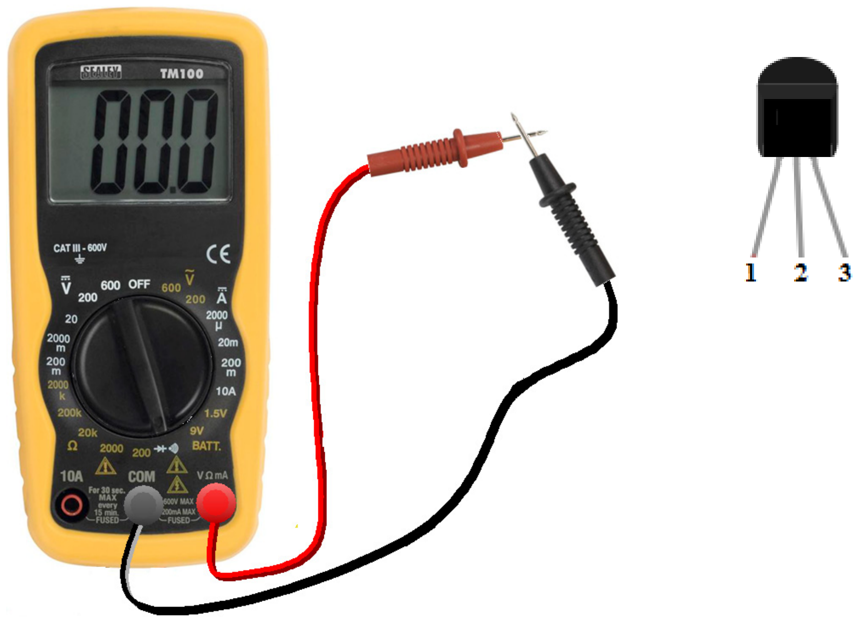


Figure 6-7: Multimeter set in diode check to identify transistor pins and type

To test a transistor with a multimeter's "resistance" or "diode check" function we need to consider transistor as being two diodes connected back-to-back. Make sure you connect the red lead in "VΩ" jack and black lead in "COM" jack of multimeter. Meter readings with PNP will be exactly opposite, of course, for an NPN transistor, with both PN junctions facing the other way. The base-emitter (B-E) junction possesses a slightly greater forward voltage drop than the base-collector (B-C) junction. This forward voltage difference is due to the disparity in doping concentration between the emitter and collector regions of the transistor. The emitter is a much more heavily doped piece of semiconductor material than the collector, causing its junction with the base to produce a higher forward voltage drop.

Expect you find a bipolar transistor and you proceed to measure continuity with a multimeter set in the "diode check" mode. Measure between pairs of pins of transistor and recording the values displayed by the meter and you obtain:

1. Meter touching pin 1 (black lead) and pin 2 (red lead): "OL"
2. Meter touching pin 1 (red lead) and pin 2 (black lead): 0.647 V
3. Meter touching pin 1 (black lead) and pin 3 (red lead): "OL"
4. Meter touching pin 1 (red lead) and pin 3 (black lead): 0.632 V
5. Meter touching pin 2 (black lead) and pin 3 (red lead): "OL"
6. Meter touching pin 2 (red lead) and pin 3 (black lead): "OL"

From the above records the only combinations of test points giving conducting meter readings are between pin 1 and pin 2 (red test lead on pin 1 and black test lead on pin 2), and pin 1 and pin 3 (red test lead on 1 and black test lead on 3). These two readings indicate forward biasing of the base- to-emitter (B-E) junction (0.647 volts) and the base-to- collector (B-C) junction (0.632 volts).

The one pin common to both sets of conductive readings must be the base connection of the transistor, because the base is the only layer of the three-layer device common to both sets of PN junctions (base-emitter and base-collector). In this example, that pin is number 1, being common to both the 1-2 and the 1-3 test point combinations. In both those sets of meter readings, the red (+) meter test lead was touching pin 1, which tells us that the base of this transistor is made of P-type semiconductor material (red = positive). Thus, the transistor is a NPN with base on pin 1, emitter on pin 2 and collector on pin 3.

For PNP, the above polarities are reversed and the black lead being common for both readings and identifies the base.

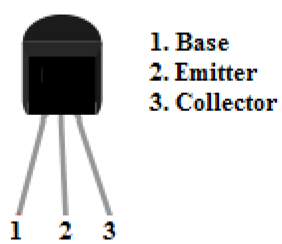


Figure 6-8: Identified pins and type of transistor after testing

Note that If you measure continuity in any more than two or any less than two of the six test lead combinations, have immediately to conclude that the transistor is defective or else that it is not a bipolar transistor but rather something else (if no part numbers can be referenced for sure identification). You don’t have to assume that the middle pin is always the base, as one might expect from the three-layer “sandwich” model of a bipolar transistor. This is quite often the case, and tends to confuse new students of electronics.

6. 5. 3. Testing MOSFETs

To test FET (MOSFET) requires expensive specialized and sophisticated test equipments but with digital multimeter it is possible to identify good or dead MOSFET. Most of dead mosfet have their pins, gate-source, gate-drain and drain-source, shorted. In other words, all the mostfet pins are connected together. You will know a MOSFET is good when the gate has infinite resistance to both drain and source. Exception to this rule, FETs with protection circuitry may act like there is a diode shunting gate and source, a diode drop for gate reverse bias. Connecting gate to source should cause the drain to source act like a diode. Forward biasing gate-source with 5V and measuring drain-source in forward bias should yield very low ohms. In reverse bias, it will still act like a diode.

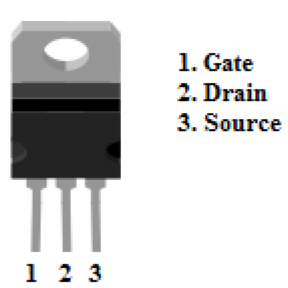


Figure 6-9: MOSFET pin configuration

When measuring a MOSFET, hold it by the case or the tab and don't touch the metal parts of the test probes with any of the other MOSFET's terminals unless needed. Do not allow a MOSFET to come in contact with your clothes, plastic, etc. because of the high static voltages they can generate.

Experiment 6-3: Testing MOSFET

Test a MOSFET using multimeter's diode check function and by using a small battery source

Part list

No	Item	Quantity
1	Digital multimeter with diode check function	1
2	N-MOSFET	1
3	1K resistor	1
4	NO push button	1
5	100nF capacitor	1
6	toggle switch	1
7	9V battery	

1st method: Testing MOSFET with multimeter's diode check function

Procedure

1. Set digital multimeter in diode check "". Connect multimeter's black lead in "COM" and red lead in "VΩ" jacks.
2. Keep the MOSFET on a dry wooden table on its metal tab, with the printed side facing you and pins pointed towards you.
3. With a piece of insulated wire stripped at the ends, short the gate and the source pins of MOSFET. You can touch these pins with your finger and will discharge through you. This will initially keep the internal capacitance of the device completely discharged.
4. Now touch the meter black probe to drain pin and the red probe to source pin of the MOSFET. You should see an open circuit (0L) indication on the meter.
5. Now keeping the black probe touched to the centre pin, lift the red probe from the source pin and touch it to the gate pin of the circuit momentarily and bring it back to the source pin of the MOSFET. This time the meter will show a short circuit.
6. The results from step 4 to 5 show that the MOSFET is normal. You can repeat these above procedures many times for proper confirmation. Remember which type of MOSFET (n-channel or p-channel) you are testing, depending on the type you will need to reverse the multimeter leads.

Figure 6-10: Testing MOSFET with digital multimeter set in diode check mode

2nd method: Testing MOSFET with dc voltage source and a capacitor

Most digital multimeters have a diode test range. On most models multimeters set in diode check put about 3 to 4V across the leads under test. This is enough to turn on most MOSFETs, at least partially, and enough to test. The meters which use a lower open-circuit test voltage (sometimes 1.5V) cannot make this test. This voltage works as pinch-off voltage: the highest voltage that can be put on the MOSFET's gate without it starts to conduct.

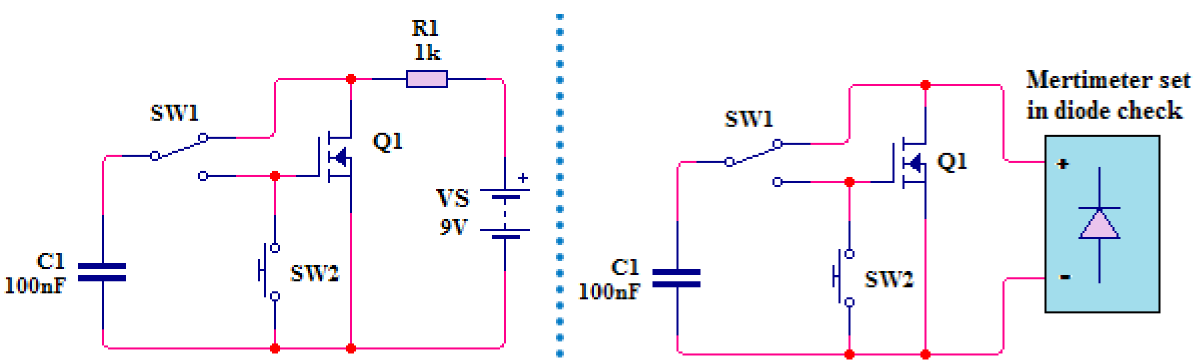


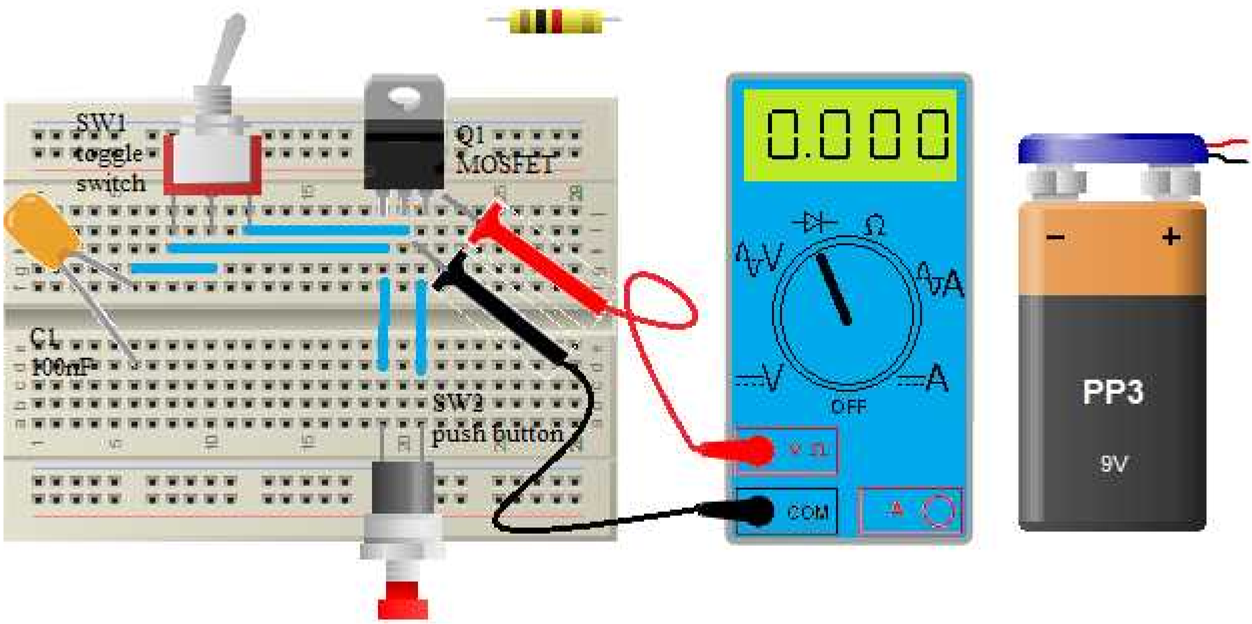
Figure 6-11: Testing MOSFET with dc voltage source and digital multimeter's diode check

In the above circuit, figure 6-11, there are two push switches, SW_1 is a changeover switch, SW_2 is a push to make switch (normally open). The test can be achieved by using the diode test of a multimeter, or you could use any power supply, or a 9V battery, with a resistor R_1 in series with the MOSFET to limit the current.

Procedure

1. With both switches in the normal position, the capacitor C_1 charges up to the open circuit voltage of the diode tester. There is no recommended value of capacitor, 10n-100n is fine.
2. When Sw_1 changed position, the charged capacitor is disconnected from the meter leads and reconnect to the MOSFET gate. The MOSFET should turn fully on, so the diode tester will indicate a short circuit.
3. Turning Sw1 to its normal position the MOSFET gate is still charged. Only leakage will discharge it, so the MOSFET should still stay conducting for quite some time.
4. Push SW_2 to short the MOSFET's gate to source (ground), to discharge it. The meter should show an open circuit.
5. The result from step 2 to step 4 confirm that the most is fine.
6. When you use the battery or any other dc voltage source you have to disconnect it after charging the capacitor and just measure using diode check mode as described in step 2 to step 4. Avoid connecting multimeter set in diode check when the battery is connected.

Alternatively you can connect the capacitor to red lead (+) via a diode, to allow the capacitor to charge, now when the capacitor is connected to the gate, the MOSFET will conduct but the diode will stop the conducting MOSFET from discharging the capacitor.



chap 6

Figure 6-12: Testing MOSFET with dc voltage source and digital multimeter's diode check, circuit connection

6. 6. Most Useful Transistors PIN Configuration

There are a great number of manufacturers around the world who produce semiconductor electronic devices where transistor belongs to this family of components, so there are literally thousands of different types. The most common type of transistor is called bipolar junction transistor and these are divided into NPN and PNP types.

The bipolar junction transistor construction material is most commonly silicon (their marking has the letter B) or germanium (their marking has the letter A). Original transistor was made from germanium, but they were very temperature-sensitive. Silicon transistors are much more temperature tolerant and much cheaper to manufacture.

Bipolar transistors have three leads namely: base (B), emitter (E), and collector (C). Sometimes, HF transistors have another lead which is connected to the metal housing. This lead is connected to the ground of the circuit, to protect the transistor from possible external electrical interference. Four leads emerge from some other types, such as two-gate FETs. High power transistors are different from low-to-medium power, both in size and in shape.

It is important to have the manufacturer's catalogue or a datasheet to know which lead is connected to what part of the transistor. These documents hold the information about the component's correct use (maximum current rating, power, amplification, etc.) as well as a diagram of the pin configuration. Placement of leads and different housing types for some commonly used transistors are in diagrams below, table 6-3 through 6-7.

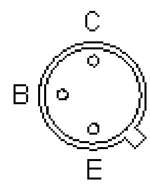
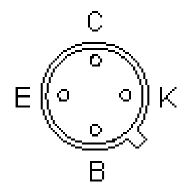
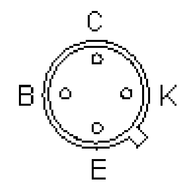
PACKAGE	TO5, TO18		T072			
PIN CONFIGURATION						
TYPE OF TRANSISTOR	**NPN**	**PNP**	**NPN**	**PNP**	**NPN**	**PNP**
LEBEL	BC 107	BC 177	BF 180	AF 139	BF 115	AF 124
	BC 108	BC 178	BF 181	AF 178	BF 167	AF 125
	BC 109	BC 179	BF 182	AF 179	BF 173	AF 126
	BFX84	BC 186	BF 183	AF 180	BF 184	AF 127
	BFY50	BC 187	BF 200	AF 181	BF 185	
	2N 706	BFX88				
	2N 2369	BCY71				
	BC 286	BC 287				
		2N2904				

Table 6-3: Pin configuration of TO-5, TO18 and TO72

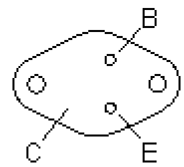
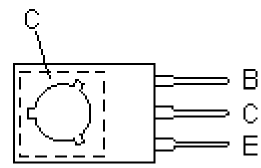
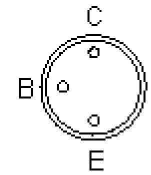
PACKAGE	T03 and Similar		TO126		TO-1	
PIN CONFIGURA-TION						
TYPE OF TRAN-SISTOR	NPN	PNP	NPN	PNP	NPN	PNP
LEBEL	2N3055	PNP3055	BD135	BD136	AC 176	AC 128
	BDY20	BDX18	BD131	BD132	AC 187	AC 188
	BD 121	OC26	BD437	BD438		
	BD 123	AD149	BUP41			
	AD 161	AD162				

Table 6-5: Pin configuration of TO-3, TO-126 and TO-1

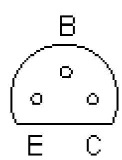
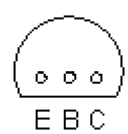
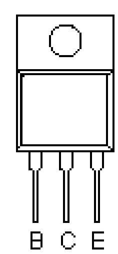
PACKAGE	X-55		TO92		TO 220	
PIN CONFIGURA-TION						
TYPE OF TRANSIS-TOR	NPN	PNP	NPN	PNP	NPN	PNP
LEBEL	BC 182	BC 212	BC183L	BC213L	BD 539	BD 540
	BC 183	BC 213	BC237B	BC 557	BD 743	BD 744
	BC 184	BC 214	BC 546	BC307B	TIP 29C	TIP30C
	2N3707	2N3702	BC 547	2N 4402	BU 407	BD240C
	2N3710	2N3703	BC 548		BUP 30	BD242C
			2N 3705		2 6099	BD244C
			2N 3903		BD243C	
					D44C10	
					BD241C	

Table 6-4: Pin configuration of X-55, TO-92 and TO-220

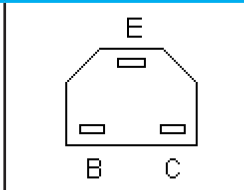
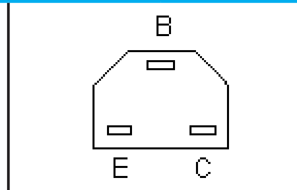
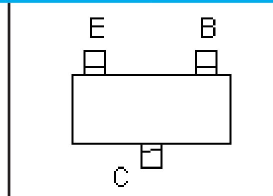
PACKAGE	SOT25			SOT23	
PIN CONFIGURA-TION					
TYPE OF TRAN-SISTOR	NPN	NPN	PNP	Surface mount transistor NPN	PNP
LEBEL	BF 194	BC 157	BC 147	BC846B	BC856B
	BF 195	BC 158	BC 148	BC847B	BC857B
	BF 196	BC 159	BC 149	BC848B	BC858B
	BF 197	BC X35	BC X31	BC849B	BC859B

Table 6-6: Pin configuration of SOT25 and SOT23

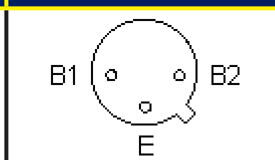
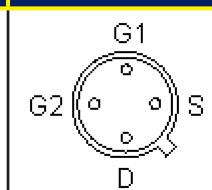
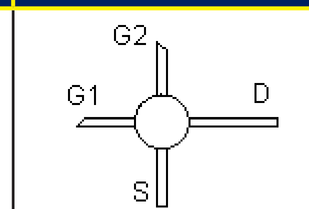
PACKAGE	SOT25		SOT 103
PIN CONFIGURATION			
TYPE OF TRANSISTOR	Unijuction Tran-sistor		
LEBEL	2N 2646	3N 140	BF 960
	2N 2647	3N 141	BF 961
	2N 4870	40673	BF 981
	2N 4871		3SK81

Table 6-7: Pin configuration of SOT25 and SOT103

It might be helpful to remember the pin configuration for TO-1, TO-5, TO-18 and TO-72 packages and compare them with the schematic drawing symbols. These transistors are the ones you will come across frequently in everyday work.

The TO-3 package, which is used to house high-power transistors, has only two pins, one for base, and one for emitter. The collector is connected to the package, and this is connected to the rest of the circuit via one of the screws which fasten the transistor to the heat-sink. Transistors used with very high frequencies (like BFR14) have pins shaped differently.

One of the breakthroughs in the field of electronic components was the invention of SMD (surface mount devices) circuits. This technology allowed manufacturers to achieve tiny components with the same properties as their larger counterparts, and therefore reduce the size and cost of the design.

One of the SMD housings is the SOT23 package. There is, however, a trade-off to this, SMD components are difficult to solder to the printed circuit board (PCB) and they usually need special soldering equipment.

As there are literally thousands of different transistors, many of them have similar characteristics, which make it possible to replace a faulty transistor with a different one. The characteristics and similarities can be found in comparison charts. If you do not have one these charts, you can try some of the transistors you already have. If the circuit continues to operate correctly, everything is alright. You can only replace an NPN transistor with an NPN transistor. The same goes if the transistor is PNP or FET. It is also necessary to make sure the pin configuration is correct, before you solder it in place and power up the circuit.

As a helpful guide, there is a chart in this chapter which shows a list of replacements for some frequently used transistors.

6. 7. TUN and TUP

TUN stand for Transistor Universal NPN whereas TUP stand for Transistor Universal PNP. Many electronic devices work perfectly even if the transistor is replaced with a similar device. Because of this, many publications use the identification TUN and TUP in their schematics. These are general purpose transistors. TUN identifies a general purpose NPN transistor, and TUP is a general purpose PNP transistor.

These transistors have following characteristics:

Parameter	Value
UCE_{max}	20V
IC_{max}	100mA
hFE_{min}	110
PC_{max}	100mW
fT_{min}	100MHz

Table 6-8: TUN and TUP characteristics

Some of the TUNs are:

TUN				
BC107	BC147	BC207	BC239	BC173
BC108	BC148	BC208	BC382	BC437
BC109	BC149	BC209	BC184	BC184
BC547	BC317	BC347	2N3856A	2N3856A
BC548	BC318	BC348	BC383	BC438
BC549	BC319	BC349	BC284	BC439
BC237	BC171	BC182	2N3904	2N3947
BC278	BC172	BC183	2N385	2N3860
				2N4124 etc.

Table 6-9: TUN transistors

Some of the TUPs are:

TUP				
BC157	BC177	BC204	BC512	BC309
BC158	BC178	BC205	BC513	BC322
BC159	BC179	BC506	BC514	BC352
BC212	BC251	BC261	2N3906	BC416
BC213	BC252	BC262	BC557	2N2412
BC214	BC253	BC263	BC558	2N3251
BC307	BC320	BC350	BC559	2N4291
BC308	BC321	BC351	2N4126	

Table 6-10: TUP transistors

Bipolar transistors have the ability to operate within three different regions:

1. Cut-off: The transistor is "fully-OFF" operating as a switch and $Ic = 0$
2. Saturation: The transistor is "fully-ON" operating as a switch and $Ic = I_{(saturation)}$
3. Active Region: The transistor operates as an amplifier and $Ic = \beta.Ib$

6. 8. Transistor as Switch

chap 6

When used as an electronic switch, a transistor normally is operated alternatively in cut-off region and saturation.

- **Cut-off Region**

The transistor is in the cut-off region if the base-emitter junction is not forward biased. For the cut-off region, the operating conditions of the transistor are zero input base current (I_B), zero output collector current (I_C) and maximum collector voltage ($VCE_{(max)}$) which results in a large depletion layer and no current flowing through the device. Therefore the transistor is switched "Fully-OFF". In this condition, there is, ideally, an open between collector and emitter and can be indicated by switch equivalent.

Cut-off Characteristic

- The input and Base are grounded (0V)
- Base-Emitter voltage (V_{BE}) is less than 0.7V for silicon and less than 0.3V for germanium transistors.
- Base-Emitter junction is reverse biased
- Base-Collector junction is reverse biased
- Transistor is "fully-OFF" (Cut-off region)
- No Collector current flows ($I_C = 0$)
- $V_{OUT} = V_{CE} = VCC$ = "logic 1"
- Transistor operates as an "open switch"

The equivalent circuit of transistor in cut-off is shown in figure 6-13.

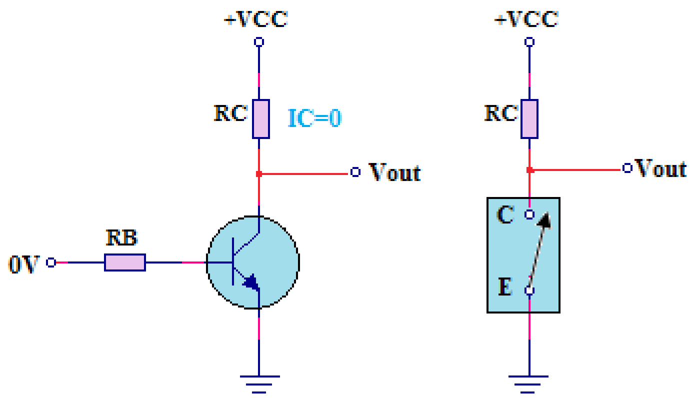


Figure 6-13: Cut-off region of transistor

Then we can define the "cut-off region" or "OFF mode" when using a bipolar transistor as a switch as being, both junctions reverse biased, $V_B < 0.7V$ and $I_C = 0$. For a PNP transistor, the Emitter potential must be negative with respect to the Base. Neglecting leakage current, as the current are approximately zero, and V_{CE} is approximately equal to VCC.

$$V_{CE(cutoff)} \approx VCC$$

- **Saturation Region**

The transistor is in the saturation region if the base-emitter junction and the base-collector junction are forward biased, and the base current is made larger enough to cause the collector current to reach its saturated value. Here the transistor will be biased so that the maximum amount of base current is applied, producing maximum collector current lead in the minimum collector emitter voltage drop which results in the depletion layer being as small as possible and maximum current flowing through the transistor. Therefore the transistor is switched "Fully-ON". In this condition there is, ideally, a short between collector and emitter as indicated by the switch equivalent figure 6-14. Actually, a voltage drop of up to a few tenths of a volt normally occurs, which is the saturation voltage, $V_{CE(sat)}$. Since $V_{CE(sat)}$ is very small compared to VCC, it can usually be neglected, so the collector current is:

$$I_{C(sat)} \approx VCC/R_C$$

Saturation Characteristics

- The input and Base are connected to VCC
- Base-Emitter voltage (V_{BE}) is approximately equal to 0.7V for silicon and equal to 0.3V for germanium transistors
- Base-Emitter junction is forward biased
- Base-Collector junction is forward biased
- Transistor is "fully-ON" (saturation region)
- Max Collector current flows ($I_C = VCC/R_C$)
- $V_{CE} = 0$ (ideal saturation)

- $V_{OUT} = V_{CE}$ = "0 logic"
- Transistor operates as a "closed switch"

The equivalent circuit of transistor in saturation is shown in figure 6-14.

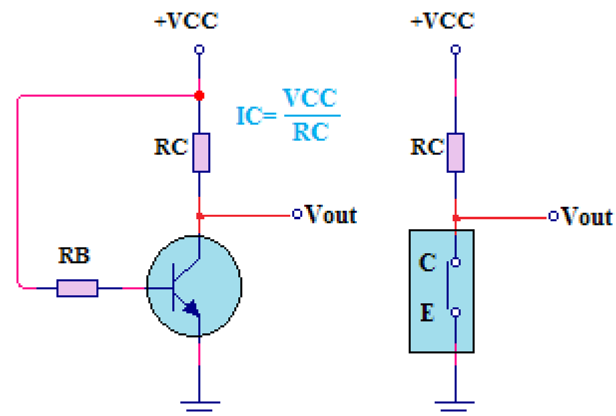


Figure 6-14: Saturation region of transistor

Then we can define the "saturation region" or "ON mode" when using a bipolar transistor as a switch as being, both junctions forward biased, $V_B > 0.7v$ and I_C = Maximum. For a PNP transistor, the Emitter potential must be positive with respect to the Base.

As result, the transistor as switch operates as a "single-pole single-throw" (SPST) solid state switch. With a zero signal applied to the Base of the transistor it turns "OFF" acting like an open switch and zero collector current flows. With a positive signal applied to the Base of the transistor it turns "ON" acting like a closed switch and maximum circuit current flows through the device.

Experiment 6-4: Transistor as switch

Design a circuit based on transistor as switch that turns on/off the bulb using the potentiometer connected as voltage divider on the base of transistor. Measure the voltage between the base and emitter, between the collector and emitter and the current through the load (bulb) connected on collector of transistor.

Part list

No	Items	Specifications	Quantity
1	Resistor	Fixed resistor 82Ω 1/2watt	1
2	Potentiometer	Potentiometer or trimmer 100KΩ	1
3	Voltmeter	Digital multimeter	2
4	Ammeter	Digital multimeter	1
5	Breadboard	RSR 03MB102 Breadboard	1
6	Connecting wires	22 gauge wire	50Cm
7	Transistor	2N2222	1
8	Lamp (bulb)	9VDC bulb	1
9	Battery	9V battery or DC power supply	1

If potentiometer POT_1 moves toward its lower position, voltage on the base decreases current through the base decreases, current through the collector decreases, and the brightness of the bulb decreases. The light intensity disappears at nearest 0% of the potentiometer, at this time the transistor is fully OFF. The ammeter reading is 0 A, voltmeter connected between the base and emitter reading is nearly 0V and voltmeter connected between the collector and the emitter reading is 9V (VCC).

If potentiometer POT_1 moves toward its upper position, voltage on the base increases current through the base increases, current through the collector increases, the brightness of the bulb increases. The ammeter reads the value greater than 0A, voltmeter connected between the base and emitter reads the voltage of about 0.7V and voltmeter connected between the collector and the emitter reads nearly 0V.

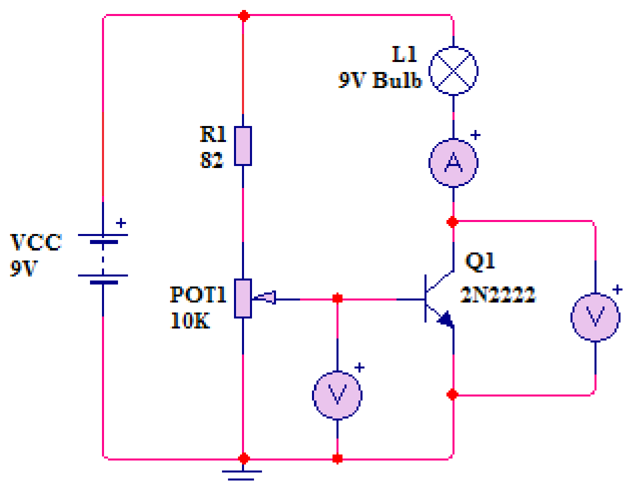


Figure 6-15: Transistor connected as switch to control the bulb

Working Principle of the Circuit

Resistor (R_1) is not really necessary, but if you do not use it, you must not turn the potentiometer (pot or trimmer) to its high position, because that would destroy the transistor. This is because the DC voltage V_{BE} (voltage between the base and the emitter), should not be higher than 0.7V, for silicon transistors.

Turn the potentiometer to its lowest position. This brings the voltage on the base (or more correctly between the base and ground) to zero volts ($V_{BE} = 0$). The bulb does not light, which means there is no current passing through the transistor.

As we already mentioned, the potentiometer's lowest position means that V_{BE} is equal to zero. When we turn the knob from its lowest position V_{BE} gradually increases. When V_{BE} reaches 0.7V, current starts to enter the transistor and the bulb starts to emit the light. As the potentiometer is turned further, the voltage on the base remains at 0.7V but the current increases and this increases the current through the collector-emitter circuit. If the POT_1 is turned fully, the base voltage will increase slightly to about 0.75V but the current will increase significantly and the bulb will glow brightly.

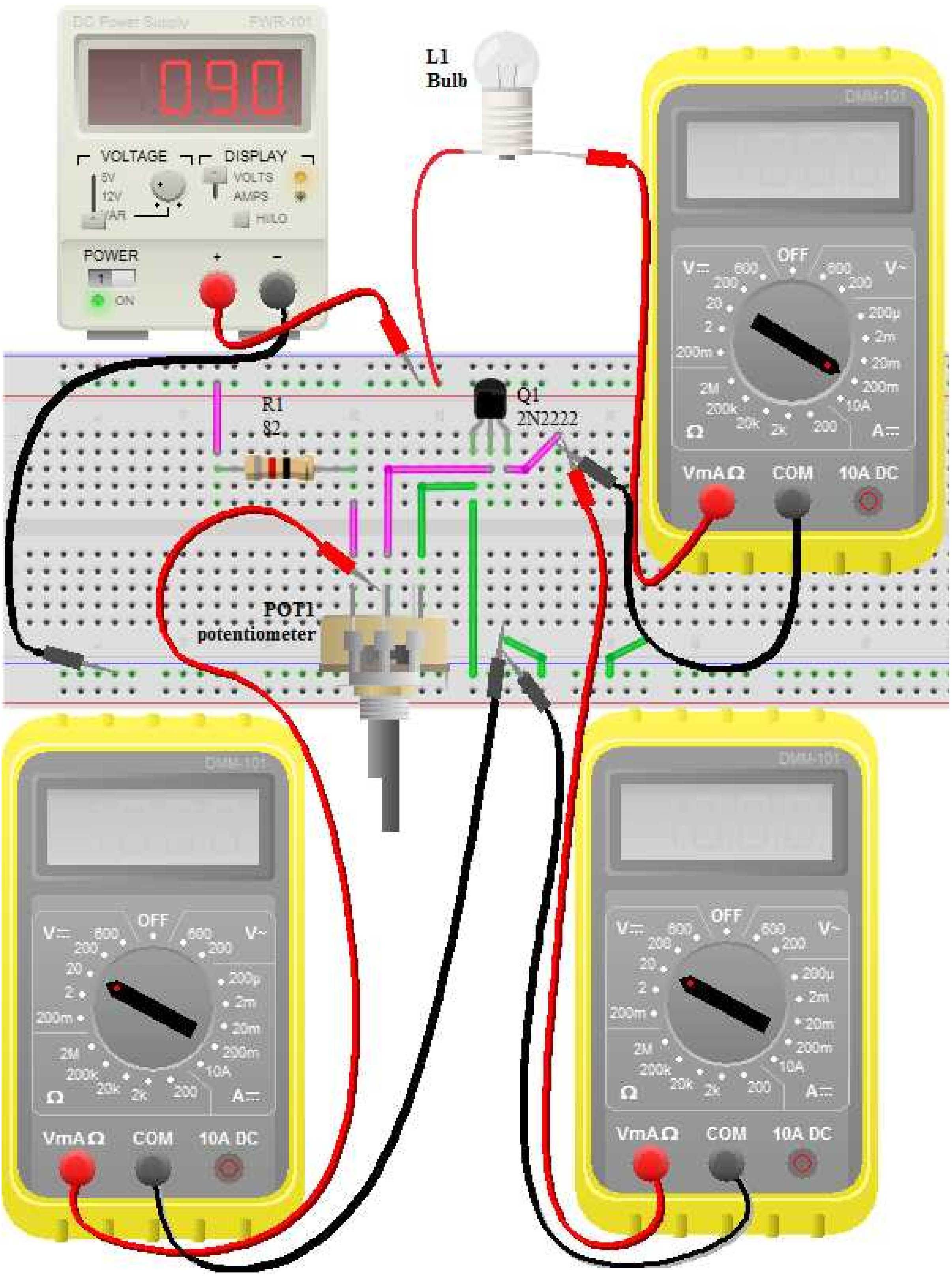


Figure 6-16: Transistor connected as switch to control the bulb, circuit connection

If we connect an ammeter between the collector and the bulb (to measure I_C), another ammeter between the potentiometer and the base (for measuring I_B), and a voltmeter between the ground and the base and repeat the whole experiment, we will find some interesting data. When the POT_1 is in its low position V_{BE} is equal to 0V, as well as currents I_C and I_B. When the POT_1 is turned, these values start to rise until the bulb starts to glow when they are: $V_{BE} = 0.6V$, $I_B = 0.8mA$ and $I_C = 36$ mA (if your values differ from these values, it is because the 2N2222 the writer used does not have the same specifications as the one you use, which is common when working with transistors).

The end result we get from this experiment is that when the current on the base is changed, current on the collector is changed also.

6. 8. 1. Light activated transistor switch

An example of an NPN Transistor as a switch being used to operate a relay is given in experiment 6-5.

Experiment 6-5: Transistor as switch controlled by the light sensor

Parts and materials

No	Item	Specification	Quantity
1	Lamp	Bulb 220Vac/100Watt	1
2	Transistor	ZTX337	1
3	Resistor	47Ω / ½ watt Fixed resistor	1
4	Potentiometer	100KΩ potentiometer or trimmer	1
5	Light dependent resistor	ORP12	1
6	Relay	9VDC/220VAC 1A	1
7	Battery	9VDC battery or DC power supply	1
8	Breadboard	Prototyping breadboard	1
9	Connecting wire	22-gauge ($0.33mm^2$) solid wire	40cm
10	Lamp supply wire	$2.5mm^2$	1m

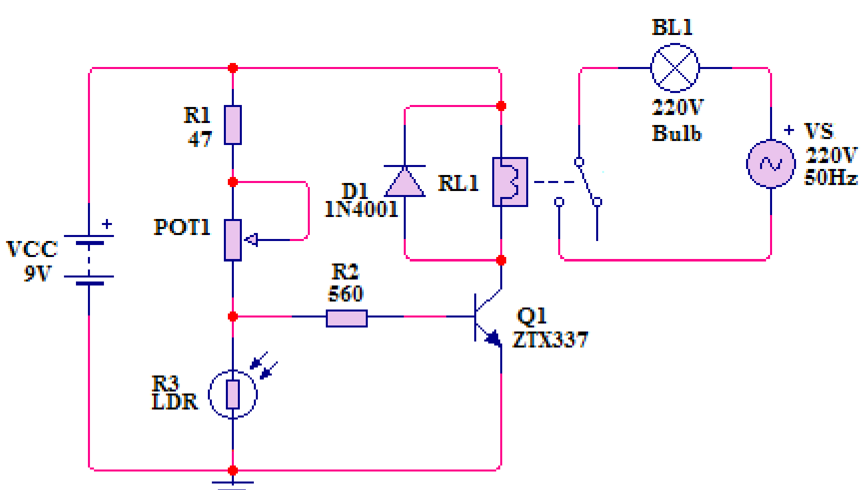


Figure 6-17: Transistor as switch controlled by the light sensor to drive a relay

Automatic dark detector senses darkness as the light level decreases and LDR meets the maximum threshold resistance, the circuit automatically switches on the lamp BL_1. The resistor R_1, POT_1 and LDR form voltage divider network. The resistor R_2 is added to determine threshold voltage other than 0.7V and prevents the current through the base of the transistor Q_1 getting too large when the voltage on LDR gets high.

A dark sensing circuit is made using light dependent resistor R_3 (LDR). The sensitivity of the circuit can be adjusted with a variable resistor (potentiometer POT_1). High resistance POT_1, more darkness to switch on the lamp BL_1. Low resistance POT_1, less darkness to switch on the Lamp BL_1. When transistor is driven to saturation, the relay RL_1 is energized and closes the contact and the current from the supply VS flows through the load (lamp BL_1). The relay is added so that the circuit can be more applicable to our day to day living necessity such as on-street lights.

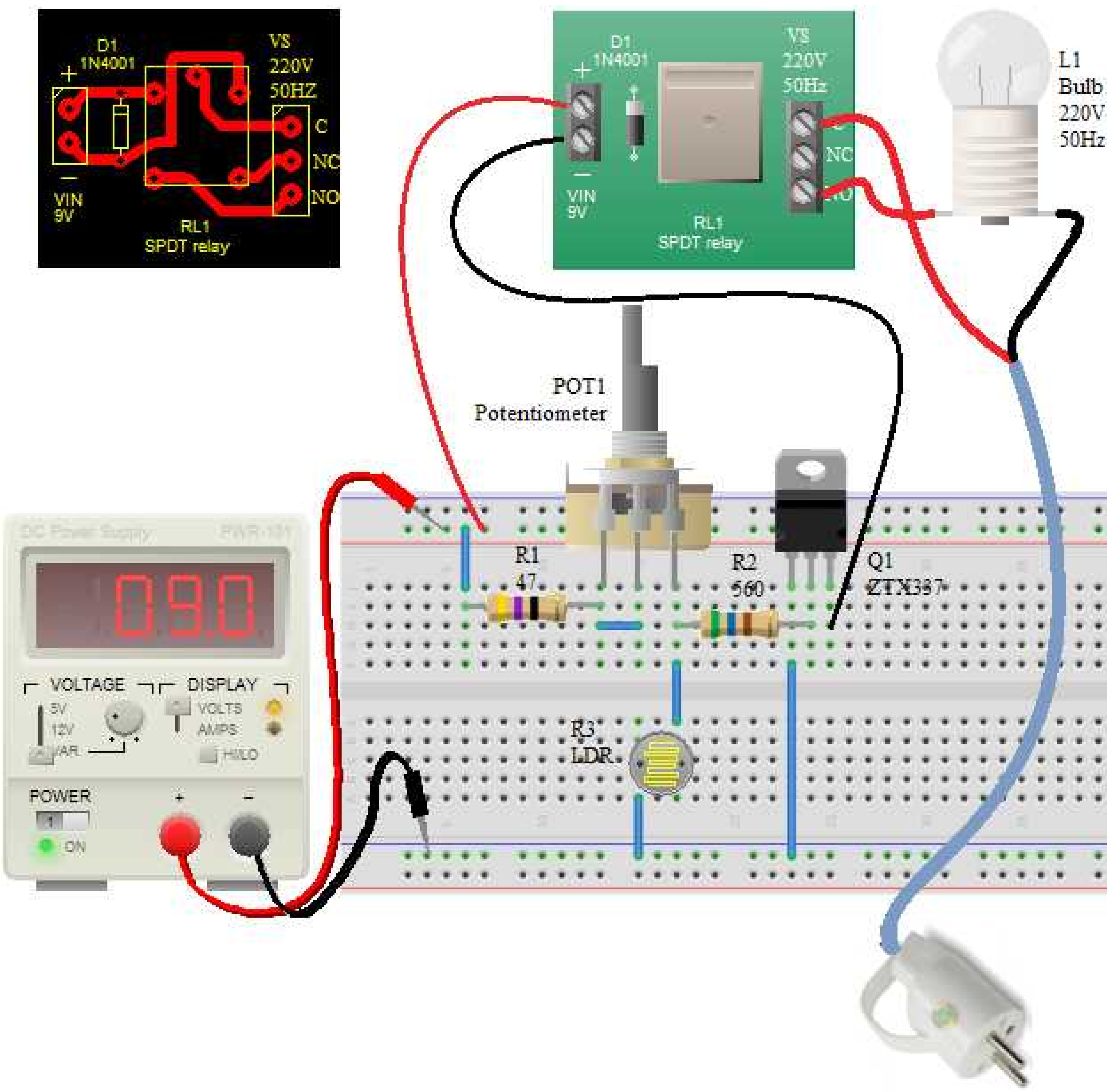


Figure 6-18: Transistor as switch controlled by the light sensor to drive a relay, circuit connection

Procedure

1. Connect the circuit as illustrated in figure 6-18.
2. The relay connection is shown on PCB real word on the right with its artwork connection on the left. This is because breadboard is not reserved for high current application. Remember to check common, normal open and normal close on relay. Reversing NO with NC the circuit that was sensing the light will sense the darkness.
3. Power on the circuit and adjust the potentiometer to set light sensing level. Turning the POT_1 in clockwise, high resistance will make more voltage to be developed across R_1 and POT_1, thus less light intensity (more darkness) on LDR will be required to switch on the lamp BL_1. Adjusting the potentiometer POT_1 in anticlockwise less darkness on LDR to switch on the bulb. You can hear relay switching.
4. You can use the light source such as the torch to let the light fall on LDR. Put the bulb BL_1 away the LDR to avoid light intrusion.
5. Plug (connect) the power cable in 220V/50Hz socket outlet to supply the bulb.
6. Measure the voltage on terminal of LDR that makes the lamp to switch on and record it. This voltage is equal to V_{BE} plus voltage drop on R_2.

With inductive loads such as relays or solenoids a flywheel diode is placed across the load to dissipate the back *emf* generated by the inductive load when the transistor switches "OFF" and so protect the transistor from damage. If the load is of a very high current or voltage nature, such as motors, heaters etc., then the load current can be controlled via a suitable relay.

chap 6

Note: You can swap the potentiometer POT_1 and LDR to convert a dark sensing circuit to a light sensor.

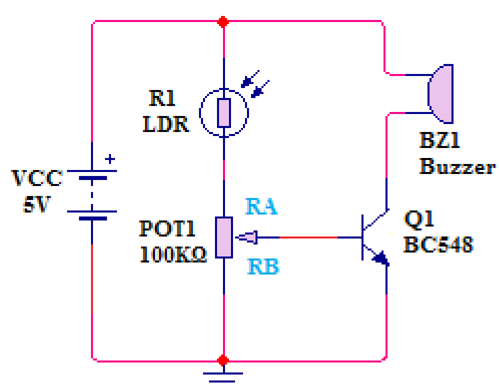


Figure 6-19: Transistor as switch used to make light alarm

For the circuit figure 6-19, LDR and POT_1 form voltage divider network. The threshold voltage V_{th} required to turn on transistor Q_1 is determined by the resistance set on POT_1 from wiper to ground (R_B). For this circuit, threshold voltage V_{th} is equal to the voltage of the base (V_{BE}) 0.7V. To set the potentiometer light sensibility level you need to let desired light intensity fall on LDR and adjust the potentiometer until the load (buzzer) is turned on. Adjust the POT_1 from the minimum to maximum until the buzzer is turned on. Assume that the LDR has resistance of 50KΩ at a given light intensity and calculate the voltage from the wiper to ground (V_B).

Resistance of $POT_1 = R_A + R_B = 100K\Omega$

1st case: POT$_1$ turned to 10%

R_B = 10% of POT$_1$= 100KΩ*10/100= 10KΩ,

Thus R_A =100KΩ-10KΩ = 90KΩ

V_B= VCC*R_B/R_B+R_A+LDR =5V *10KΩ/10KΩ+90KΩ+50KΩ

= 0.33V

Since V_A (0.067) is less than V_{BE} 0.7 (threshold voltage) transistor Q_1 will be in cut-off and no current flows through the buzzer (buzzer is off).

2nd case: POT$_1$ turned to 20%

R_B = 20% of POT$_1$= 100KΩ*20/100 = 20KΩ,

Thus R_A =100KΩ-20KΩ = 80KΩ

V_B= VCC*R_B/R_B+R_A+LDR =5V *20KΩ/20KΩ+80KΩ+50KΩ

= 0.67V

Since V_A (0.67) is less than V_{BE} 0.7 (threshold voltage) transistor Q_1 will be in cut-off and no current flows through the buzzer (buzzer is off).

3rd case: POT$_1$ turned to 21%

R_B = 21% of POT1= 100KΩ*21/100 = 21KΩ,

Thus R_A =100KΩ-21KΩ = 79KΩ

V_B= VCC*R_B/R_B+R_A+LDR = 5V *21KΩ/41KΩ+79KΩ+50KΩ

= 0.7V

Therefore, V_B is equal to threshold voltage and is equal to 0.7 and transistor Q_1 is saturated and current flows through the load (buzzer is on).

4th case: POT$_1$ turned to 25%

R_B = 30% of POT$_1$= 100KΩ*25/100 = 25KΩ,

Thus R_A =100KΩ-25KΩ = 75KΩ

V_B= VCC*R_B/R_B+R_A+LDR = 5V *25KΩ/25KΩ+75KΩ+50KΩ

= 0.833V

Now V_B (0.833V) is greater than 0.7 (threshold voltage) and transistor Q_1 is saturated and current flows through the load (buzzer).

Note: As the potentiometer is turned to the scale greater than 21%, the voltage on the base remains at 0.7V but the current increases and this increases the current through the collector-emitter circuit. If the POT$_1$ is turned fully (100%), the base voltage will increase slightly to about voltage greater than 0.7V but the current will increase significantly and the buzzer sound louder.

After setting light sensitivity level you can illuminate and obstruct the light on LDR to activate and deactivate the buzzer from the influence of the light.

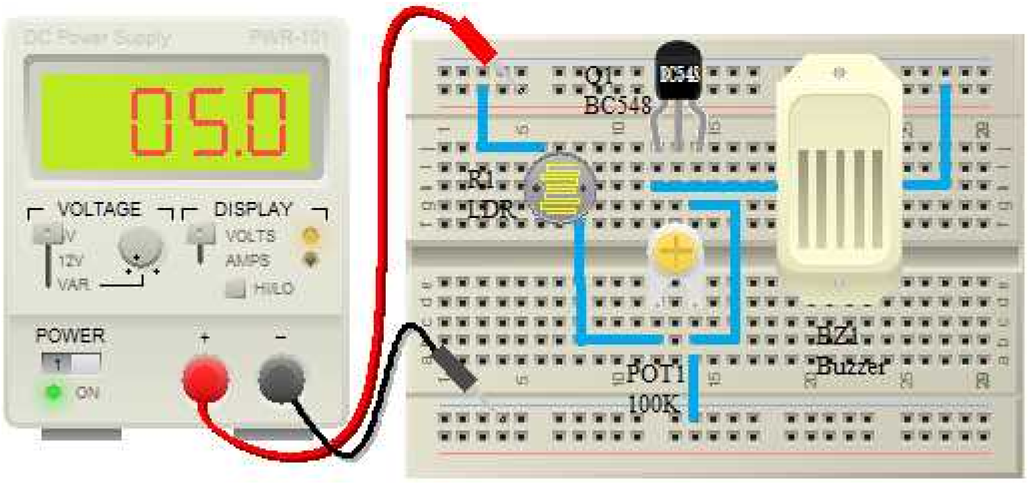


Figure 6-20: Transistor as switch used to make light alarm circuit connection

6. 8. 2. Transistor as switch with temperature sensor

The transistor can be used in a circuit with temperature sensor such as thermistor to switch on/off the fan. A thermistor is a component that has a resistance that changes with temperature. There are two types of thermistor. Those with a resistance that increase with temperature (Positive Temperature Coefficient – PTC) and those with a resistance that falls with temperature (Negative Temperature Coefficient – NTC). The amount by which the resistance increases/decreases as the temperature decreases is not constant. It varies with temperature. A formula can be used to calculate the resistance of the thermistor at any given temperature and information can be found on device datasheet. With thermistor (NTC or PTC) it is possible to construct the circuit, based around the transistor as switch, which turns on or off when it gets hot. The decrease or increase in resistance of the thermistor in relation to the other resistor which is fixed as the temperature rises will cause the transistor to turn on. The value of the fixed resistor will depend on the thermistor used, the transistor used and the supply voltage. The circuit can be heat or cold activated option depending on the type of thermistor used or position of it in a circuit with respect to switching transistor. The experiments 6-6 describe the way transistor working as switch combined with thermistor can be used to design hot or cold activated switch.

Experiment 6-6: Transistor as switch controlled by the temperature sensor

Design cold and heat activated switch based on transistor as switch to turn on/off the load as the temperature increases. List of materials is given below.

Part list

No	Item	Specification	Quantity
1	Transistor	2N6488	1
2	9V DC motor	9V DC motor	1
3	Potentiometer or preset resistor 50K	50KΩ preset resistor	1
4	PTC and NTC Thermistor	NTC of 10K@25°C, PTC of 2K@25°C	2
5	Resistor	270Ω 1/2watt	2

6	Heating device	40watt soldering iron	1
7	Breadboard	Prototyping breadboard	1
8	Breadboard connecting wire	22-gauge	40cm
9	9V DC supply	DC power supply	1
10	Light emitting diode (LED)	Red LED	1

- **Cold activated transistor switch (frost alarm)**

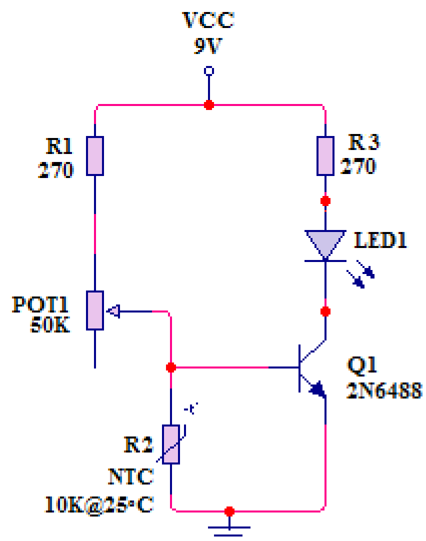


Figure 6-21: Cold activated switch

The circuit of figure 6-20 uses NTC thermistor to turn on the LED_1. The resistor R_1, potentiometer POT_1, and NTC R_2 form voltage divider network and the base of transistor Q_1 is fed from connecting point of POT_1 and NTC thermistor. The voltage at the base of Q_1 is determined by the resistance of thermistor R_2 from the ratio of the voltage divider network. When the voltage on thermistor is equal or greater than 0.7V (V_{BE}) the output is turned on. When fire or other source of heat is lit, the thermistor R_2 gets hot, the resistance of the thermistor deceases and the voltage on thermistor R_2 also decreases below threshold voltage required to turn on the transistor Q_1. In other hand when NTC thermistor gets cold its resistance increases and the voltage on the base of transistor also increase. The voltage across the thermistor is calculated as follow:

$$V_{thermistor} = VCC \times [R_{thermistor}/(R_1+POT_1+R_{thermistor})]$$

Normally it requires 0.7V to turn on a transistor it is also worth noting that the output, when turned on, will be lower than the supply voltage VCC. This is because of the voltage drop across the collector and emitter pins of transistor. R_1 is present to protect the transistor from excessive base current in case the potentiometer should be set to zero.

The point at which the circuit is triggered is set by the 50KΩ potentiometer resistor POT_1. By varying the value of this resistor the ratio of the resistance of R_2 and POT_1 plus R_1 can be varied to a point where a centre voltage (trip point) of 0.7V is achieved at the desired cold level.

If LED_1 and R_3 are fitted the LED will light at this point. The value of R_3 should be selected for the relevant supply voltage on LED used. A standard red LED would require around 1.8V, 20mA

(0.02A) producing a normal brightness. As stated a 9V supply would give about 8V across LED_1 and R_3. The LED_1 (red) would use 1.8V leaving around 6.2V (8V-1.8V) across R_3.

$$\text{Using Ohm's law } R = V/I_{R3} = 6.2V / 0.02A;\ R_3 = 310\Omega$$

Then from resistor pack the preferred value is 330Ω or we choose 270Ω for high brightness.

Procedure

1. Connect the circuit as illustrated in figure 6-22. Turn the potentiometer POT_1 fully clockwise (high resistance = 50KΩ). At this point the output should be off and the LED_1 is off.
2. Now turn the potentiometer POT_1 anticlockwise until the output turns on and the LED lit.
3. Turn the potentiometer back clockwise. Note the point at which the output and the LED_1 turn back off. This is the trip point for the current temperature.
4. If you want the circuit to trip at a lower temperature, then adjust POT_1 forward in the clockwise direction. If you want the circuit to trip at a higher temperature, then adjust POT_1 back in the anticlockwise direction.
5. With cooling device let the cold reach on NTC thermistor. With enough cold on R_2, the LED_1 should light up.

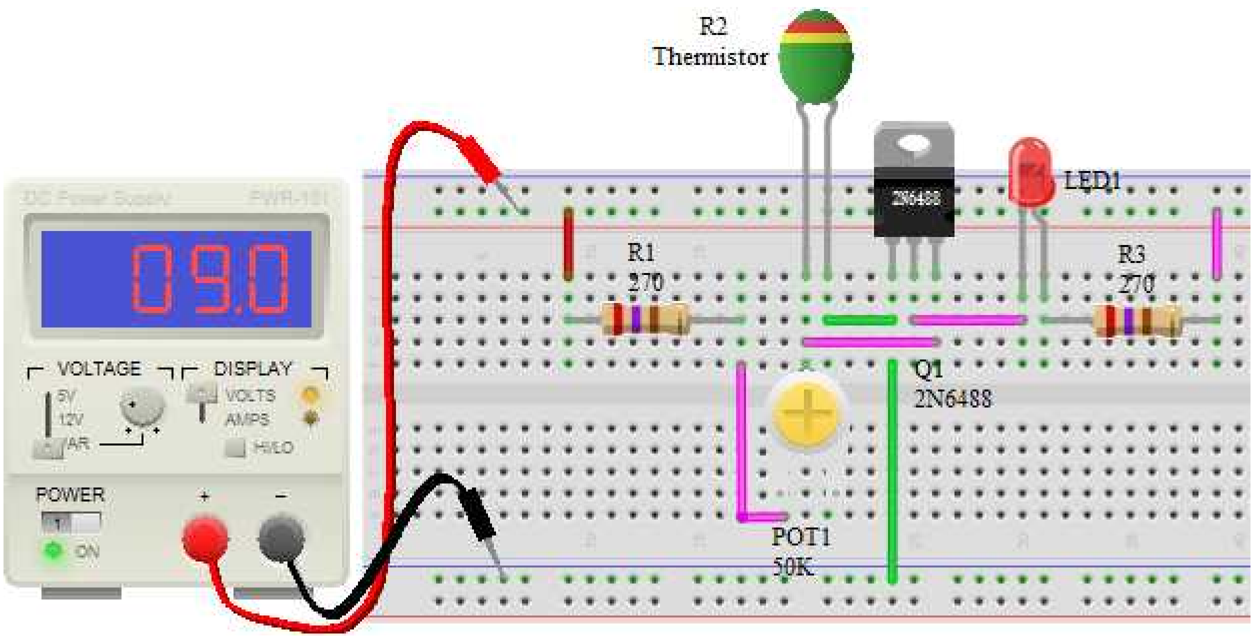


Figure 6-22: Cold activated switch circuit connection

Replacing the NTC thermistor resistor (R_2) with PTC thermistor in the above circuit and swamp R_2 with potentiometer POT_1 the circuit will remain working as cold activated switch.

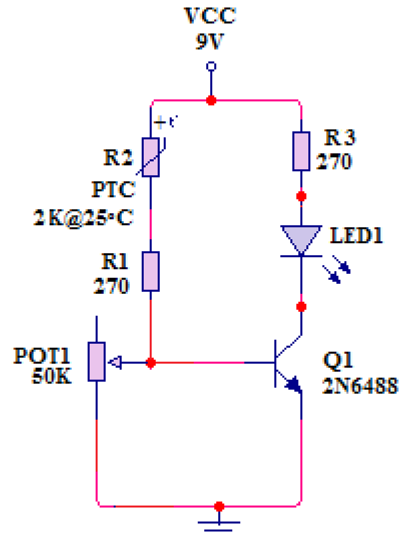


Figure 6-23: Cold activated switch using PTC thermistor

- **Heat activated transistor switch (warmth alarm)**

The circuit figure 6-24 uses PTC thermistor as temperature sensor. R_1, POT_1 and R_2 form the potential divider network. The voltage on the base of transistor Q_1 is determined by the resistance of R_2. The resistance of R_2 increases as the temperature increases. The voltage across R_2 (PTC) gradually increases as the temperature increases. But the voltage across the transistor (V_{CE}) does not change smoothly as the voltage across the thermistor resistor R_2 goes up. Instead the voltage across the transistor V_{CE} drops quite suddenly when the voltage at the base of the transistor reaches 0.7 V.

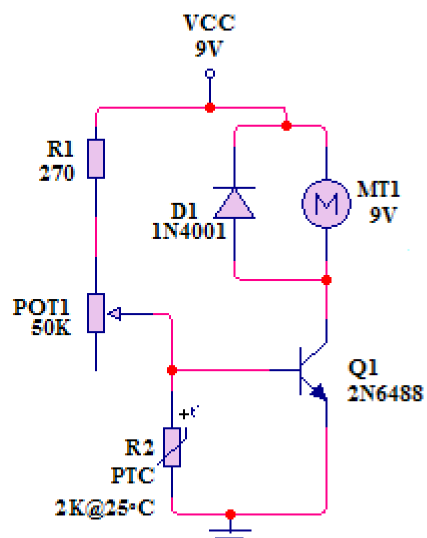


Figure 6-24: Transistor as switch with PTC thermistor for fan control

When the voltage across the transistor V_{CE} drops, the voltage across the DC motor MT_1 goes up. This means that the transistor Q_1 is full on and the current now is flowing through the motor MT_1. Because the voltage across the transistor V_{CE} drops suddenly at one particular temperature, the motor MT1 switches on quite sharply at that temperature. The thermistor R_2 resistor controls the temperature at which the motor switches on. With the potentiometer resistor POT_1, the MOT will switch on at a specific determined temperature. Changing the value of this resistor will alter the temperature at which the motor MT_1 switches on. The voltage across the thermistor is given by:

$$V_{thermistor} = VCC \times [R_{thermistor}/(R_1+POT_1+R_{thermistor})]$$

Assume at a particular temperature given in a table and calculate the voltage across the thermistor R_2 and determine the state of the DC motor MT_1 when the potentiometer POT_1 is set to 40KΩ.

No	Temperature	Resistance of R2	Voltage on R2	MT1 state
1	10°C	2KΩ	0.4258V	Stop
2	20°C	3KΩ	0.624V	Stop
3	30°C	4KΩ	0.8132V*	Run
4	40°C	5KΩ	0.99V*	Run
5	50°C	6KΩ	1.1671V*	Run

* Marks the voltages greater than 0.7V which, from a given circuit, cannot happen. The voltage on the base of transistor Q_1 remains at 0.7V for thermistor resistance greater than 4KΩ but the current increases and this increases the current through the collector-emitter circuit.

Table 6-11: As the temperature on thermistor increases, the voltage on the base increases and the motor start to run

Procedure

1. Connect the circuit as illustrated in figure 6-25. Turn the potentiometer resistor POT_1 fully clockwise (high resistance = 50K). At this point the output should be off and the motor MT_1 stops.
2. Now turn the potentiometer resistor POT_1 anti-clockwise until the output turns on and the motor MT_1 starts to run.
3. Turn the potentiometer resistor POT back clockwise. Note the point at which the motor MT_1 turns back off and stop. This is the trip point for the current temperature.
4. If you want the circuit to trip at a high temperature, then adjust POT_1 forward in the clockwise direction.
5. If you want the circuit to trip at a lower temperature, then adjust POT_1 back in the anticlockwise direction.
6. With heating device, let temperature reaches on PTC until the motor starts to run. Remove the thermistor from the circuit and measure its resistance at the point the motor starts to run. Remove also the potentiometer and measure the set resistance and calculate the voltage on thermistor R_2. Compare the obtained voltage to already calculated in table 6-11.

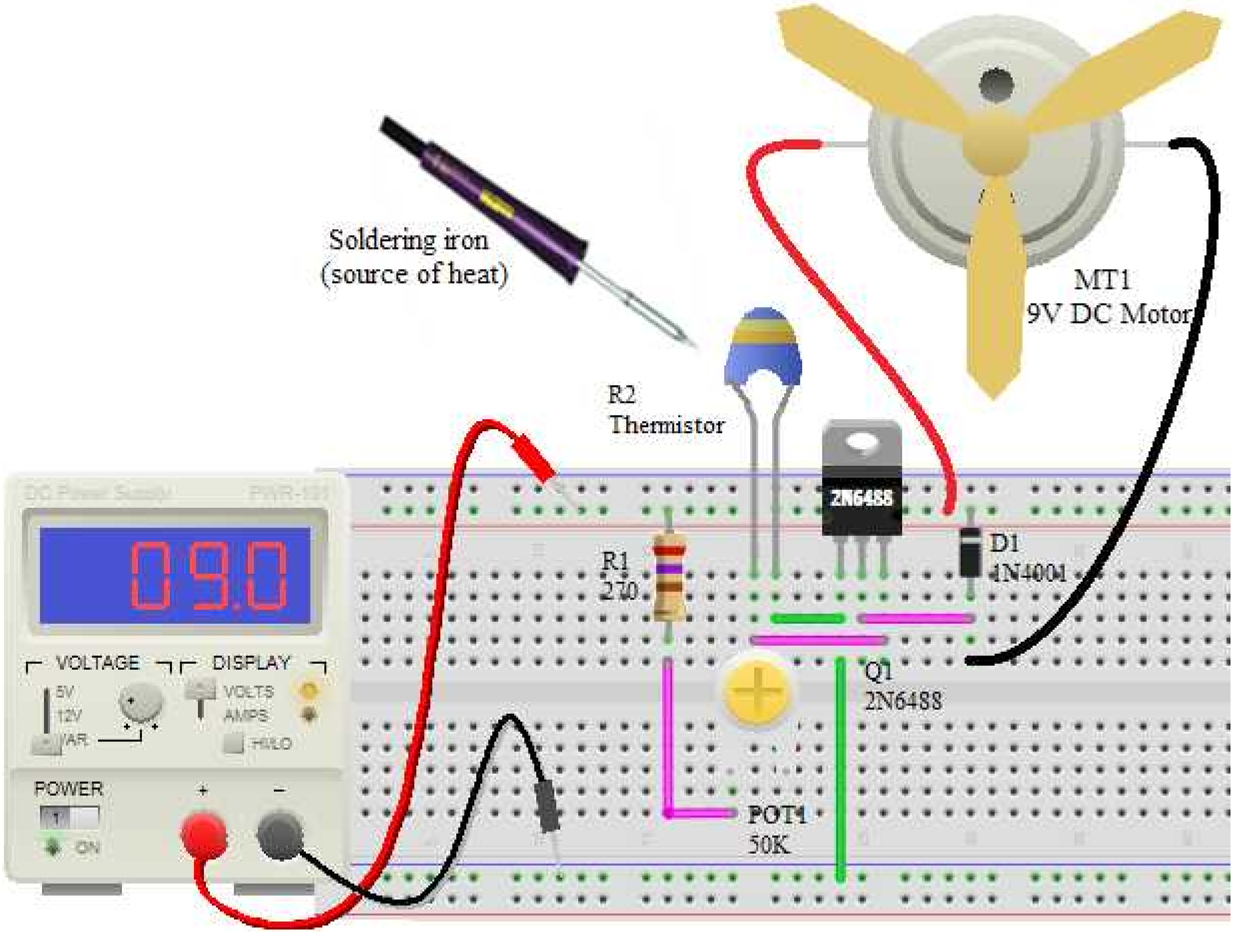


Figure 6-25: Transistor as switch with PTC thermistor for fan control circuit connection

Replacing the PTC thermistor resistor (R_2) with NTC thermistor in the above circuit and swamp R_2 with potentiometer POT_1 the circuit will remain working as heat activated switch.

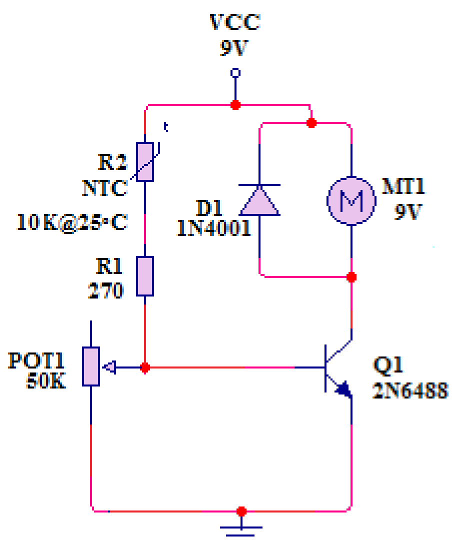


Figure 6-26: Heat activated switch using NTC thermistor

6. 9. Pulse Generators

Pulse generators are circuits that generate a pulse waveform directly and most of them use the relaxation principle (charging and discharging of capacitor). The most common type is the multivibrator, which consists of two stages, resistance coupled amplifier with the output of each stage coupled resistively to the other. In generation collector current of one stage is a maximum when collector current of the other stage is at cut-off at regular intervals. Multivibrators are classified into three types: Astable, bistable, and monostable.

6. 9. 1. Astable Multivibrator

Astable Multivibrator based on transistors is a cross coupled transistor network capable of producing sharp continuous square wave. It is free running oscillator or simply a regenerative switching circuit using positive feedback. It uses two transistors switching on/off alternatively in regular time interval and has no stable state. Astable multivibrator switches continuously between its two unstable states without the need for any external triggering. Time period of astable multivibrator can be controlled by changing the values of feedback components such as coupling capacitors and resistors.

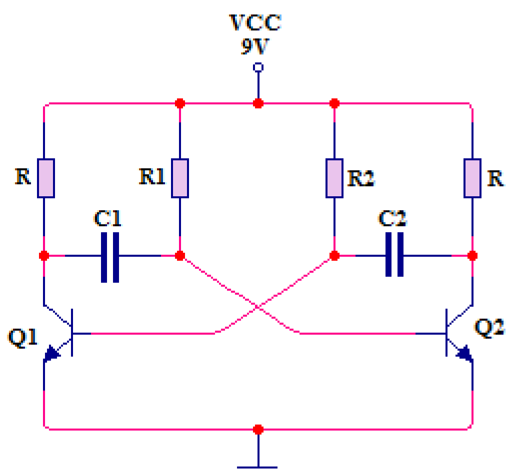


Figure 6-27: Astable multivibrator

Working Principle of the Circuit

Assume anyone of the transistors Q_1 or Q_2 turns ON due to parameter variation or due to some switching transients, let it be Q_2. Then the collector voltage of $Q_2=V_{ce(sat)}\approx 0V$, it is cross coupled to the base terminal of Q_1 through C_2, then Q_1 remains in OFF state. During Q_2 ON, the current path through R_2 charges the capacitor C_2, the capacitor C_1 voltage is coupled to the base of transistor Q_1. While charging of C_2, when the capacitor voltage exceeds 0.7V, Q_1 becomes turned ON. As soon as Q_1 is ON, its collector voltage falls to $V_{ce(sat})\approx 0V$, it is coupled to the base terminal of Q_2 then Q_2 becomes OFF. At the same time capacitor C_1 starts charging through R_1, when the C_1 voltage exceeds 0.7V, Q_2 turns on due to cross coupling. This process continues. The resistors "R" on the collector of transistors Q_1 and Q_2 are considered as loads. We can also take the output from collector terminals and ground.

If we refer to the transistor Q_1, time period of positive and negative half cycle of astable multi-vibrator is given by:

$$T_{ON}=0.69\ R_1*C_1\ [Sec]$$
$$T_{OFF}=0.69R_2*C_2\ [Sec]$$

The output can also be taken from transistor Q_2, if so;

$$T_{ON}=0.69\ R_2*C_2\ [Sec]$$
$$T_{OFF}=0.69R_1*C_1\ [Sec]$$

The time period (T) is given by:

$$T=T_{ON}+T_{OFF}$$
$$=0.69\ (R_1C_1+R_2C_2)\ \ [Sec]$$

The frequency (*F*) of transistorized astable multivibrator is calculated as:

$$F=1/T=1/0.69\ (R_1C_1+R_2C_2)\ \ [Hz]$$

Duty cycle D percentage is calculated as

$$D = 100*\ T_{ON}/\ T_{ON}+\ T_{OFF}$$

For square wave output, the resistor values should satisfy: $R_1/R<hfe$, where *hfe* is the transistor's current gain (same for R_2 and R). As hfe is usually around 80 to 300 for most of transistors, you have to make sure that the resistors you are using are close to the margin.

Experiment 6-7: Astable multivibrator

Design an astable multivibrator using transistors and display the output waveform on oscilloscope. Calculate the period, time ON, time OFF, and frequency. Measure using oscilloscope and compare the practical results to theoretical results.

Parts and materials

No	Item	Specifications	Quantity
1	5V DC supply	DC power supply	1
2	Resistors	2x100KΩ, ½ watt and 2x220Ω, ½ watt resistors	4
3	Capacitors	2x10uF, 16V electrolytic capacitors	2
4	Breadboard	RSR 03MB102 breadboard	1
5	Connecting wires	22-gauge solid wire	50cm
6	Oscilloscope	Digital oscilloscope (Agilent 54622D)	1
7	LEDs	Red and green LEDs	2

- Calculation of T_{ON} and T_{OFF}

Given that $R_2= R_3=100K\Omega$, and $C_1=C_2=10uF$, Thus;

$$T_{ON} = 0.69R_2{*}C_1 = 0.69{*}100{*}10^3{*}10{*}10^{-6}$$
$$= 0.69Sec$$

$$T_{OFF} = 0.69R_3{*}C_2 = 0.69{*}100{*}10^3{*}10{*}10^{-6}$$
$$= 0.69Sec$$

- Calculation of period T,

$$T = T_{ON} + T_{OFF} = 0.69\ Sec + 0.69\ Sec$$
$$= 1.38\ Sec$$

- Calculation of frequency *F*,

$$F = 1/T = 1/1.38Sec = 0.725Hz$$

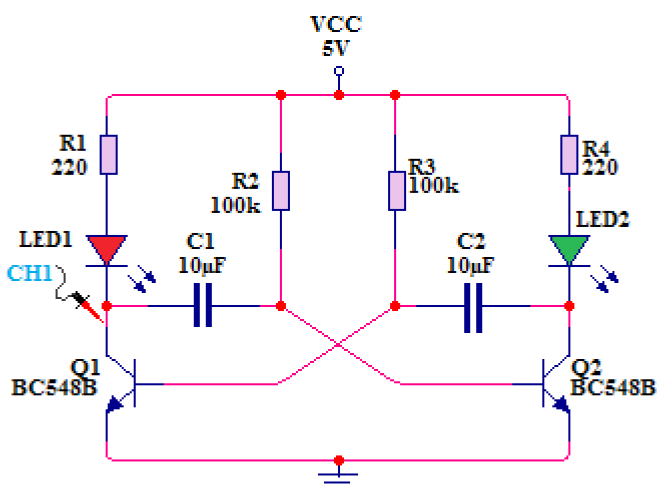


Figure 6-28: Astable multivibrator oscillator

Procedure

1. Connect the circuit as illustrated in figure 6-29. Remember to identify the anode and cathode of LEDs. Anodes are connected to positive of supply through the limiting resistors R_1 and R_4. Remember also to identify the base, collector and emitter of transistors Q_1 and Q_2.
2. Power on the circuit and observe the alternation lighting of LED_1 and LED_2.
3. Connect the oscilloscope's probe on collector of transistor Q_1 and observe the waveform on the screen of oscilloscope. This should be the square wave.
4. With oscilloscope press on quick measure and measure T_{ON}, T_{OFF}, period and frequency.
5. Compare the practical result from the oscilloscope to the theoretical calculated results.

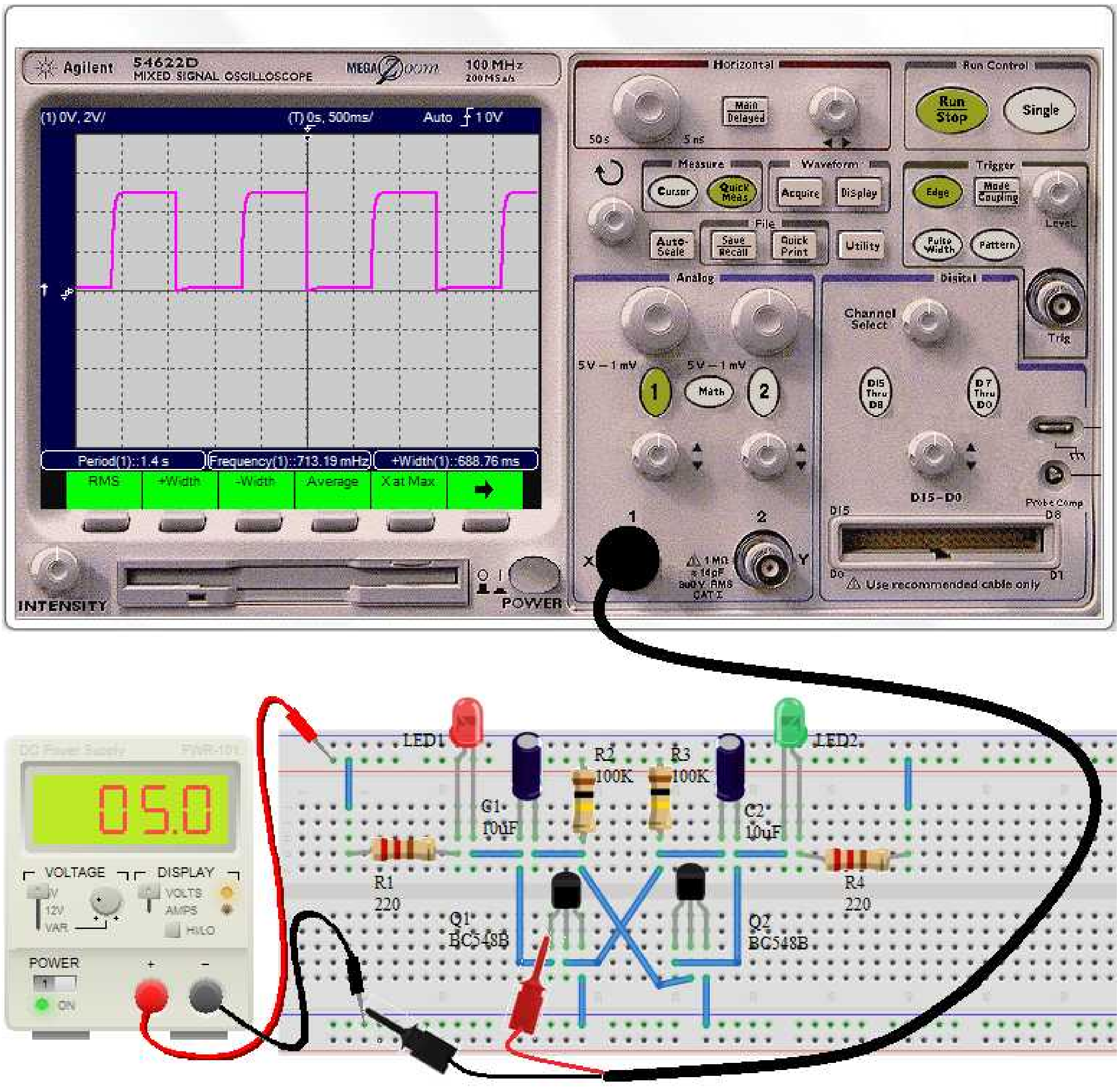


Figure 6-29: Astable multivibrator oscillator circuit connection

You should have found that this circuit does indeed oscillate between its two states, neither of which is stable. That's why this circuit is known as an astable multivibrator. With these component values, each LED remained on for just about 0.69 seconds before the circuit switched states. This continued for as long as you left power on. With smaller capacitors to increase the frequency of oscillation, this type of circuit is sometimes used as a clock generator for sequential digital circuits that do not need to operate at some precise frequency.

6. 9. 2. Bistable Multivibrator

As the name indicates, the bistable multivibrator has two stable states. Bistable multivibrator requires external triggering pulse of proper amplitude and polarity to change the state. Because the bistable has two stable states, it is ideally suited for several applications in electronics. The closest mechanical analogy to a bistable multivibrator is a toggle switch. Momentary pressure in the form of a trigger pulse causes the output of the bistable to change the states. That means the output changes from high to low, or low to high. In electronics, a toggle is a latch that maintains a condition once pressure or a signal is received. As the output stays at one state until another trigger pulse is received, the circuit remembers. That means the circuit can function as a memory storage device. Each time a trigger is applied to a bistable, it changes states.

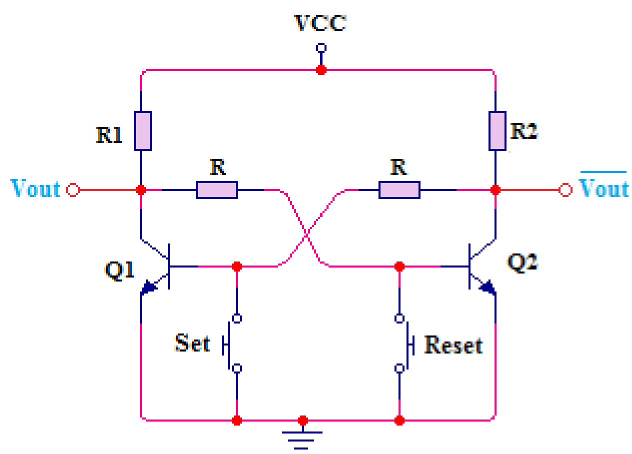


Figure 6-30: Bistable multivibrator

The circuit is constructed based on two basic RTL (resistor transistor logic) inverters cross-couple them so that each has its input connected to the output of the other. At first thought this may seem meaningless, since such a circuit would seem to be locked forever into one condition, or state. One transistor would be turned on while the other would be turned off, and each would continue to enforce the state of the other. But as we mentioned earlier, an external triggering signal is required for bistable multivibrator to change the state. Either transistor can be on while the other is off, and the circuit will retain its state until it is changed by an external signal or power is turned off.

When the power is applied on the circuit let, for instance, suppose at any particular instant, transistor Q_1 is conducting and transistor Q_2 is at cut-off. The output taken on the collector of Q_2 will be at logic 1 "VCC". This will be applied to the base of transistor Q_1 through the input resistor, keeping Q_1 turned on so that its output taken on the collector will be a logic 0. This logic 0 will be applied back to the input resistor of the base of Q_2 keeping Q_2 turned off and hold the entire circuit locked into this state. If left to itself, the bistable multivibrator will stay in this position for ever. If the "set button" is pressed, 0V will be applied on the base of transistor Q_1 and turns off. The output of Q_1 and the base of Q_2 will rises toward VCC. This will make transistor Q_2 to be driven in saturation and the output of Q_2 will be pulled to ground. A logic 0 from the collector of Q_2 will be applied back to the base of transistor Q_1, keeping Q_1 turned off so that the logic 1 will be taken on its collector. The circuit will remain in this state until a reset button is pressed. If a "reset button" is pressed, a reverse operation will be applied and keep a logic 0 on collector of Q_1 while the logic 1 will be applied on the collector of Q_2.

Experiment 6-8: Bistable multivibrator

Design a transistor based bistable multivibrator and change the LEDs states using the switches.

Parts and materials

No	Item	Specification	Quantity
1	5V DC power supply	DC power supply	1
2	Transistors	2N2222 transistor	2
3	LEDs	Red LED and green LED	2
4	Push button	NO push button	2
5	Resistors	2x330Ω ½ watt and 2x6.8KΩ resistors	4
6	Breadboard	RSR 03MB102 breadboard	1
7	Connecting wires	22-guage solid wire	30cm

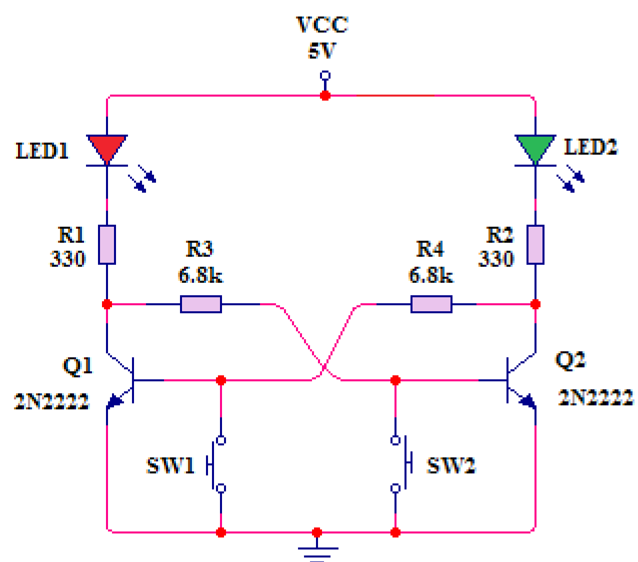


Figure 6-31: Bistable multivibrator

Procedure

1. Connect the circuit as illustrated in figure 6-32. Make sure you have identified the anode and cathode of light emitting diodes. The anodes have to be connected to the positive of the supply and the cathode to the collector of the transistors Q_1 and Q_2 through the resistors R_1 and R_2 respectively. Locate the emitter of the transistors and connect them to ground. Connect the push button switches SW_1 and SW_2 between the base of transistor Q_1 and Q_2 respectively and ground. Connect the resistor R_3 between the collector of Q_1 and the base of Q_2. Connect the resistor R_4 between the collector of Q_2 and the base of Q_1.
2. Power on the circuit. One of the LED must light up whilst the other is off. Push button switches SW_1 and SW_2 are used to change the state of either transistor by applying a required triggering pulse.
3. Note the state of LED_1 and LED_2. What does this tell you about the state of transistor in

your experimental circuit? Turn off power and wait a while, about 1 min and turn power back again. Now what are the state of LED_1 and LED_2? Repeat this action five more times. Do LED_1 and LED_2 always indicate the same state at power-up or they came up at random?

4. If the red LED is lightened up first, press the SW_1 to turn it off and turn on the green LED. If it is the green LED which was lightened up at first, press SW_2 to turn it off and switch on the red LED.
5. You will find out that if one of the LED is on, it will continue to be on until a corresponding push button used to change the state is pressed. As result, when you first turned on power, either LED_1 or LED_2 turned on and stayed on. In nearly every case, turning power off long enough and then turning it back on again made no difference; the two transistors are not identical and one turned on faster than the other. That transistor kept its LED on, while the other transistor remained turned off and therefore turned its connected LED off to indicate a logic one at that collector. Both states are completely stable, meaning that only the application of an external signal (grounding the base) can cause this circuit to change states. It will not spontaneously changes states so long as the circuit is powered.

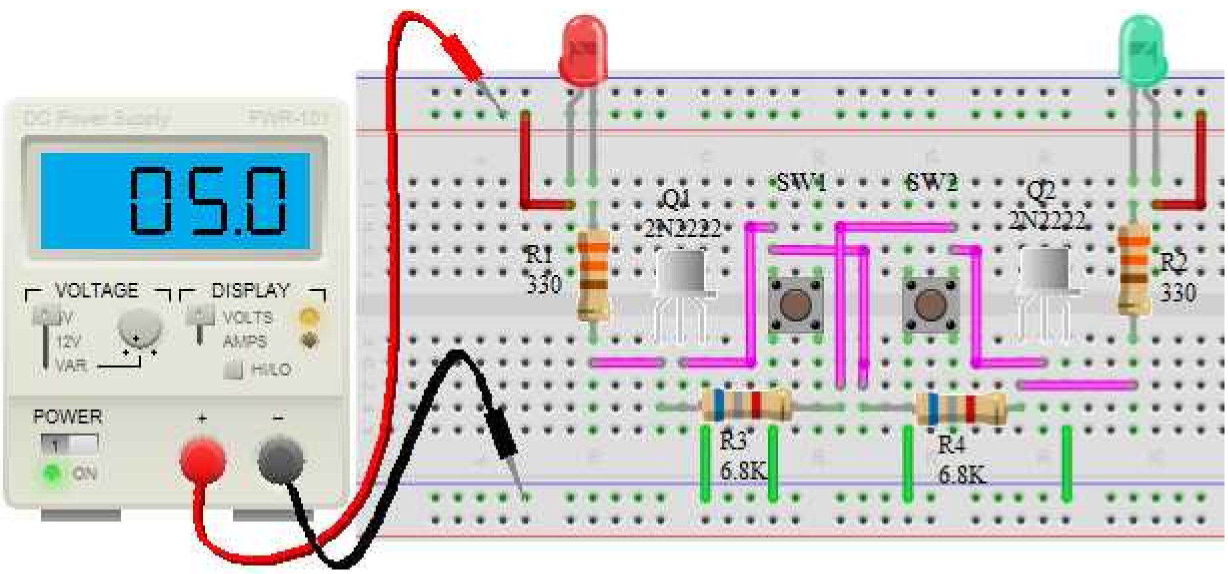


Figure 6-32: Bistable multivibrator circuit connection

Applications of Bistable Multivibrator

The major uses for the bistable multivibrator are as a memory storage device, timing circuit, frequency divider, and an electronic toggle switch.

6. 9. 3. Monostable Multivibrator

Monostable multivibrators have only one stable state (hence their name: "Mono"), and produce a single output pulse of a specified width, either "HIGH" or "LOW" when a suitable external trigger signal or pulse is applied. The timing cycle of the monostable is determined by the time constant of the timing capacitor *C* and the resistor *R* until it resets or returns itself back to its original (stable) state. The trigger signal initiates a timing cycle which causes the output of the monostable to change its state at the start of the timing cycle and back to its original stable state. The monostable multivibrator will then remain in this original stable state indefinitely until another input pulse or trigger signal is received.

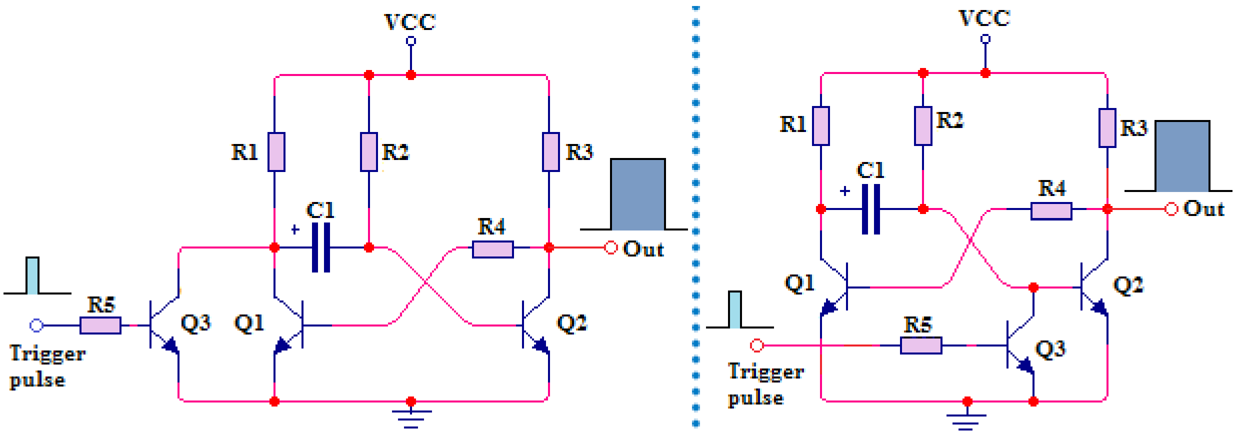


Figure 6-33: Monostable multivibrator

The circuits of the figure 6-33 show the basic transistor monostable multivibrators with their associated waveforms. When power is firstly applied and assuming there is no triggering pulse, the base of transistor Q_2 will be connected to VCC through a biasing and timing resistor R_2, thereby driving the transistor into saturation "full-on". This in turn pulls the collector of Q_2 (output) to ground, "logic 0". This output is also applied to the base of transistor Q_1 at the same time turning the transistor Q_1 "full-off". This then represents the circuit stable state with zero (LOW) output. Of course during this process, the capacitor C_1 will take a certain amount of time to charge, but once it does so it will carry no current. The current through the base of Q_2 is equal to $I_B = (VCC-0.7)/R_2$.

At this point, the capacitor C_1 is charged to just about VCC volts (less V_{BE} of Q_2) with the terminal connected to the collector of Q_1 being positive.

For the left circuit, when a positive trigger pulse is received on the base of transistor Q_3, it turns on and pulls the left end of the capacitor down to ground. Since the capacitor voltage cannot change instantaneously, this forces the right end of capacitor to $-$VCC, immediately turning Q_2 off. When Q_2 is off, its collector will be pulled to VCC (logic 1). This in turn applies a logic 1 to the base of transistor Q_1 through R_4, turning Q_1 on. In case the trigger is held to high, the output (collector of Q_2) is logic 1 and Q_1 will remain in saturation "fully-on". At this time, the left end of capacitor C_1 will remain locked to ground through the collector of Q_1, but the right end gradually charges through timing resistor R_2, toward VCC. With a triggering pulse, as soon as the capacitor voltage reaches 0.7V, the base of Q_2 will be forward biased and turns on, in the other hand turning off again the transistor Q_1. This returns the circuit to its stable state.

For the right circuit, when a trigger pulse is received on the base of transistor Q_3, the transistor will work as a closed switch shorting the base of Q_2 to ground. This will induce the transistor Q_2 from saturation to cut-off and the output taken to collector of Q_2 goes to the high state "logic 1". This high output when coupled to the base of transistor Q_1, drives it into saturation and turns Q_1 on. The collector voltage on Q_1 falls by I_{CQ1}*R_1 and the base voltage on Q_2 falls by the same amount, since the voltage across the capacitor C_1 cannot change instantaneously. Now, from the moment we applied the trigger, Q_2 went to cut-off and Q_1 into saturation. The capacitor C_1 charges to about VCC less V_{BE} from supply through timing resistor R_2 and the conducting transistor. The moment the base voltage of Q_2 reaches 0.7V it will be driven in saturation, turning on, which, owing to coupling through R_4 turns Q_1 off. The circuit will remain in this state, which represents the original stable state, until another trigger pulse is applied.

The output time duration is given by: $T_{ON} = 0.693 * R_2 * C_1$

The time on delay can be adjusted by replacing the resistor R_2 by potentiometer. The collector resistors have to be well chosen as high collector resistor will not allow the transistor to go in saturation and low collector resistor will allow the current to pass through the collector-ground connected load even when the transistor is in saturation.

Experiment 6-9: Monostable multivibrator

Design a monostable multivibrator using transistors that turns on an LED for about seven second when a push button is pressed/released.

Parts and materials

No	Item	Specification	Quantity
1	5V dc supply	5V DC power supply	1
2	Transistors	2N6714 transistor	3
3	Resistors	2x680Ω ½ watt, 100KΩ, 3x1KΩ ½ watt ,resistors	6
4	Breadboard	RSR03MB102 breadboard	1
5	Connecting wires	22-gauge solid wire	50cm
6	LED	Red LED	1
7	Push button	NO push button	1
8	Capacitor	100uF/50V electrolytic capacitor	1

chap 6

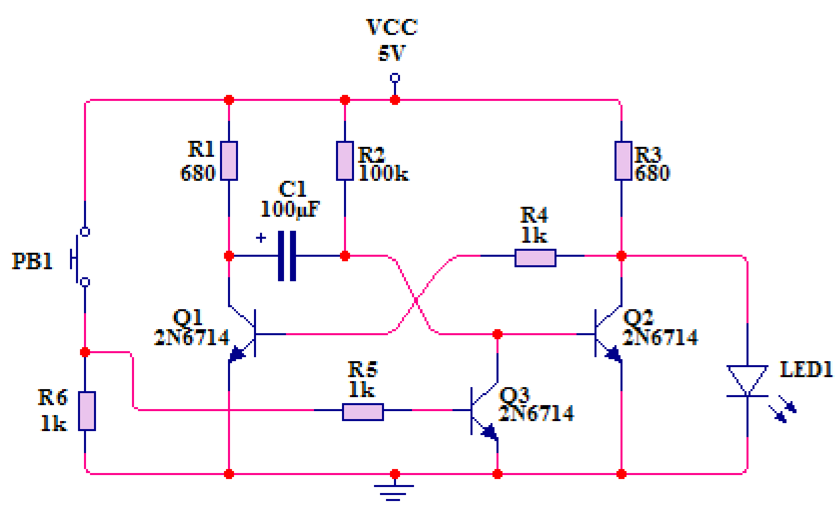


Figure 6-34: Monostable multivibrator experimental circuit

The circuit of the figure 6-22 shows a monostable multivibrator based around the transistors. The push button PB1 is used as trigger. The base of transistor Q_3 is normally at ground via a pull down resistor R_6. When the push button is pressed for a short time, the base of Q_3 will be connected to

VCC (5V) through the base resistor R_5 and turns on. This will make the base of Q_2 to be grounded and turns off. As a result, the current will flow through the LED_1 and light up for a period determined by the timing RC network (R_2 and C_1). The time the LED_1 will be on is calculated as:

$$T_{ON} = 0.693 * R_2 * C_1 = 0.693 * 100 * 10^3 * 100 * 10^{-6} Sec$$
$$= 6.93 Sec$$

Procedure

1. Connect the circuit as illustrated in figure 6-35.
2. Power on the circuit and note the state of LED_1. This should be off. Initially capacitor C_1 was full discharge and as its voltage cannot change instantaneously it has no effect on holding Q_2 off, therefore, LED_1 turns ON briefly and back off when you firstly apply power to the circuit.
3. Press on the push button for a short time as possible. At this time LED_1 should light up for about 6.93 Seconds. After this time the LED should have turned back off. If not you may check the pin polarity otherwise the LED was damaged. Also make sure the base, collector and emitter of transistors are well identified and located.
4. Press and old the bush button for about 2minitues. The LED should be on all along the time you have pressed plus 0.693 seconds. This is long enough to prevent contact bounce in the switch from being a problem, but not long enough to exercise manual control of the circuit timing.

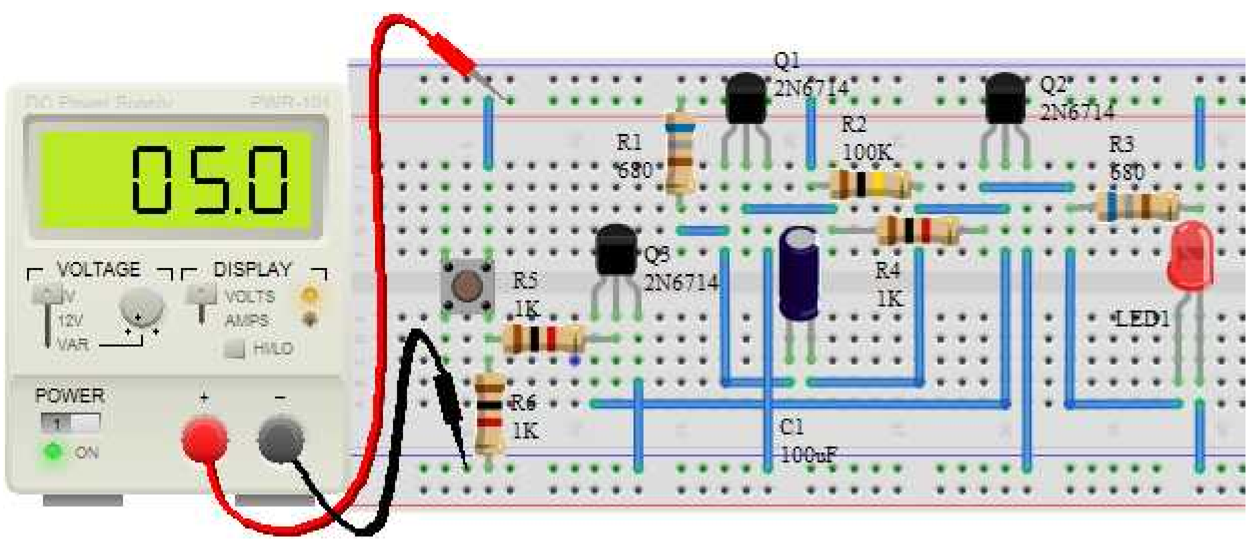


Figure 6-35: Monostable multivibrator experimental circuit connection

Basically, each time you triggered this circuit, LED_1 remained on for about 7 seconds, then turned off. You could not extend this interval, nor could you shorten it. Thus, this circuit generates a single, fixed-duration output pulse each time it is triggered, and then waits for the next trigger pulse. Since the duration of the pulse produced by this circuit (the "pulse width") is controlled by R_2 and C_1, we can set the pulse width by modifying these components. Of course, R_2 must have a value suitable for the correct biasing of Q_2. Therefore, the value of C_1 is selected to get the pulse width as close as possible to the desired value, and then R_2 is adjusted to "fine tune" the pulse width. If exact timing is critical, a potentiometer is used.

There are some of integrated circuits (ICs) that can be used as monostable multivibrators and among of them can combine two monostable in-one. Those ICs are 74121 (single monostable multivibrator), 74122 (single retriggerable monostable multivibrator), 74221 (dual monostable multivibrator), and 74123 (dual retriggerable monostable multivibrator).

6. 10. Transistor as Amplifier

Transistors have ability to increase the strength of weak signals where they work as amplifier. Transistor working as amplifier operates in active region. Amplifier is a circuit that is used for amplifying a signal. The input signal to an amplifier will be a current or voltage and the output will be an amplified version of the input signal. An amplifier circuit which is purely based on a transistor or transistors is called a transistor amplifier. Transistors amplifiers are commonly used in applications like RF (radio frequency), audio, OFC (optic fiber communication) etc. Anyway the most common application we see in our day to day life is the usage of transistor as an audio amplifier. Basically, there are three transistor configurations that are used: −common base (CB), −common collector (CC) and −common emitter (CE). In common base configuration has a gain less than unity and common collector configuration (emitter follower) has a gain almost equal to unity). Common emitter has a gain that is positive and greater than unity. So, common emitter configuration is most commonly used in audio amplifier applications.

Transistor Amplifier Design

The importance of amplifier designing process is to make an amplifier capable to increase small signal of all of the input with minimum amount of distortion possible to the output signal so that the output be an exact reproduction of the input signal but only increased amplitude. To achieve this, dc operating point have to be well selected. Normally an input signal of transistor amplifier is an ac current which swings above and below a presetting value determined by a biasing voltage. To build a transistor amplifier, all you need are a transistor, a power source, resistors, and capacitors. There are the ways to mix these components together in order to build an amplifier. We give some basic conditions and assumptions to work with and then walk through the design.

6. 10. 1. Common Emitter Amplifier

The common emitter amplifier is one of the transistor amplifier in which the base is the input, the collector is the output, and the "common" or ground is the emitter. The emitter is common for both the input and the output where the name *common emitter*.

The common emitter amplifier is characterized by:

- A moderately low input impedance
- A moderately large output impedance
- A high current gain (β is high, 50-300)
- It has a high voltage gain
- It produces very high power gain
- It produces phase reversal of input. Input and output signals are 180^0 out of phase.

Common-Emitter Amplifier Design

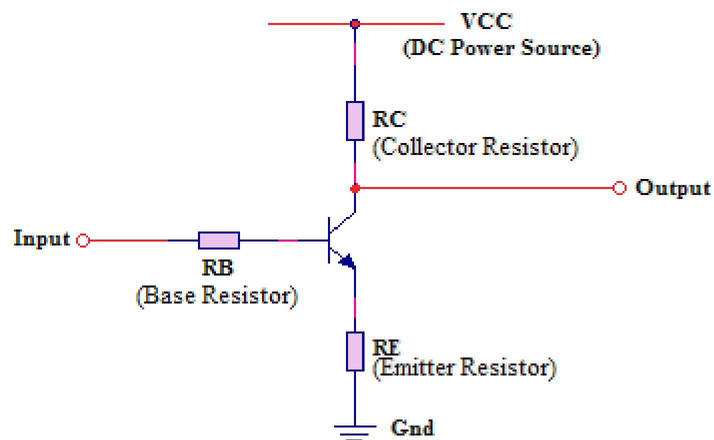


Figure 6-36: Common emitter configuration representation

1. Deciding the DC supply voltage

DC supply voltage for transistor amplifier should be less than the maximum V_{CEO} voltage for the transistor you intend to use (For example BC337 transistor has V_{CEO} of 45V) and will also depend on the available supply; this may be a bench power supply or a battery. Values of 6 to 12 volts are common for a common emitter voltage amplifier. For our circuit design we will use 9VDC.

chap 6

2. Choosing transistor

The type of transistor to be used will depend on the type of supply and the function a transistor is reserved for. This can be found on datasheet of transistor which can be downloaded from www.datasheetcatalog.com where you will find different electrical characteristics for a device. If you are using positive voltage you have to choose NPN, if you are using negative voltage you have to choose PNP transistor. For our design we will use BC337 NPN transistor.

3. Decide on a suitable quiescent collector current I_C

I_C is the Collector current when no signal is applied. The maximum value must be less than the maximum $I_{C(max)}$ for a given transistor. However using a high value of current will waste power as the circuit is supposed to be a voltage amplifier so current should be kept quite low, but the lower the current you choose, the higher the value of R_L will be. This increases the output impedance of the amplifier (which will be approximately the value of the load resistor) and ideally this should be low. A compromise figure of 10 to 20% of the transistor's $I_{C(max)}$ figure shown on the datasheet should be fine for a maximum value of I_C but a commonly selected current of around 1mA would be typical for a small general purpose amplifier transistor.

4. Choosing the value for collector resistor R_C

The value of collector resistor R_C can be calculated after deciding on the supply voltage and collector current. The transistor quiescent collector voltage needs to be half of VCC (supply) so that the output signal alternates by equal amount in positive and negative around this value without driving the transistor into saturation. Generally, the quiescent Q-point of the amplifier is with zero input signal applied to the Base, so the collector sits about half-way along the load line between zero volts and the supply voltage, (Vcc/2). RC will therefore be half of VCC divided by I_C. For this experiment RC is equal to:

$$R_C=(VCC/2)/I_C=(9V/2)/1mA=4.5/0.001A= 4.5K\Omega$$

Note that whenever a component value has been calculated, it is unlikely that the result of the calculation will match any of the available preferred values of real resistors. Therefore you will need to choose the nearest preferred value.

From resistor pack, the next preferred value of 4.5KΩ is equal to 4.7KΩ, thus R_C is equal to 4.7KΩ.

5. Choosing the value of emitter resistor R_E

To provide efficient bias stabilization, the emitter voltage V_E should be about 10% to 15% of VCC. So choosing a value of 10% to 15% of VCC for V_E and assuming that I_E is the same as I_C ($I_E=I_C+I_B$, $I_C=\beta I_B$, transistors of the same type and part number will have large variations in their Beta β value this is because Beta is a characteristic of their construction and not their operation and 100 is a reasonable average value for low power signal transistors, thus I_B is neglected) a value for the resistor R_E can be calculated by dividing the emitter voltage V_E by the emitter current I_E then choosing the nearest preferred value. Also Scherz recommends a V_E of 1 V to stabilize the transresistance instability.

$$R_E=V_E/I_E= 1V/1mA=1K\Omega$$

6. Calculating the gain A_V

The gain as it stands at ΔVc/ΔVe which is equal to the ratio of Rc/re. We have already set R_C = 4.7KΩ, and *re* is already built into transistor. Re is called the transresistance, which is calculated as:

$$re = 0.026V/I_E=0.026V/0.001A=26\Omega$$

Then the gain A_V is calculated as:

$$A_V=R_C/re=4700\Omega/26\Omega=180$$

Nevertheless we have to consider the emitter resistor since the transresistance can be unstable in the transistor, so we need to add our own resistor in addition to the transresistance. Now the gain becomes

$$A_V=R_C/R_E+re= 4700\Omega/1000\Omega+26\Omega= 4.6$$

As you see, the gain becomes much smaller with emitter resistor. One way to get the original gain is to bypass the emitter resistor by putting a capacitor in parallel with R_E to form a high pass filter.

7. Choosing bypassing capacitor C_E and coupling capacitors C_1 and C_2

In Common Emitter Amplifier circuits, capacitors C_1 and C_2 are used as coupling capacitors to separate the AC signals from the DC biasing voltage. This ensures that the bias condition set up for the circuit to operate correctly is not affected by any additional amplifier stages, as the capacitors will only pass AC signals and block any DC component. The output AC signal is then superimposed on the biasing of the following stages. Also a bypass capacitor, C_E is included in the emitter leg circuit. C_1 and C_2 are chosen to be 1μF.

This capacitor is an open circuit component for DC bias meaning that the biasing currents and voltages are not affected by the addition of the capacitor maintaining good operating point stability. However, this bypass capacitor short circuits the emitter resistor at high frequency signals and only R_L plus a very small internal resistance acts as the transistors load increasing the voltage gain to its maximum. In most of the case, the value of the bypass capacitor, C_E is chosen to provide a reactance of at most, $1/10^{th}$ the value of R_E at the lowest operating signal frequency. Let's assume for our circuit we only want to measure signals with components above 200 Hz. We already have R_E equal to 1KΩ, from cut-off frequency formula F=1/2ΠRC we can extract the capacitance. Therefore, C_E is calculated as:

$$F=1/2\pi R_E C_E$$

$$C_E=1/2\pi R_E F=1/2*3.14*1000*200=0.7uF \text{ or } 0.68uF$$

By considering bypassing capacitance as being the $1/10^{th}$ of R_E at lowest operating frequency, C_E is given as:

$$1000*10/100=1/2\pi C_E F$$

$$C_E=1/100*2\pi*F=1/100*2*314*200= 7.9*10^{-6}=7.9\mu F \text{ or } 6.8\mu F \text{ from capacitor standard values.}$$

8. Estimating the value of base current I_B

chap 6

The base current I_B is found by dividing the collector current I_C by the transistor's current gain β (h_{fe}) obtained from the device datasheet. Because the h_{fe} varies from one transistor to another, even of the same type, it may be quoted as a typical value or as a range between minimum and maximum values, h_{fe} also varies with collector current so whatever value you choose for h_{fe}, the result of calculating I_B will be an approximation so the base voltage will probably not be accurate. However, this can be "fine tuned" when the amplifier is being constructed. For this transistor amplifier design we are using β equal to 100, thus $I_B=I_C/\beta=0.001A/100=10\mu A$.

9. Calculating base voltage V_B

The base voltage should be about 0.7V (700mV) higher than V_E to ensure that the input signal is biased on the linear part of the transistor input characteristic.

Therefore,

$$V_B=V_{BE}+V_E=0.7V+1V= 1.7V.$$

10. Calculating the DC bias network current I_T

The current flowing through the potential divider circuit has to be large compared to the actual base current, I_B, so that the voltage divider network is not loaded by the base current flow. To ensure adequate bias stability, the current flowing through R_2 should be about 10 times greater than the base current I_B so the current flowing through R_1 and R_2 (I_T) will be simply I_B x 11.

$$I_T= I_B*11=10\mu A*11= 110\mu A$$

11. Calculating R_1 and R_2

A general rule of thumb is a value of at least 10 times I_B flowing through the resistor R_2. Transistor base/emitter voltage, V_{BE} is fixed at 0.7V (silicon transistor) then this gives the value of R_2 as:

$$R_2 = (V_E+V_{BE})/10*I_B = 1V+0.7V/10*10\mu A = 17000\Omega = 17K\Omega,$$

The next preferred value of 17KΩ from resistor pack is 18KΩ

If the current flowing through resistor R_2 is 10 times the value of the Base current, then the current flowing through resistor R_1 in the divider network must be 11 times the value of the Base current. The voltage across resistor R_1 is equal to VCC – 1.7V (V_{RE} + 0.7 for silicon transistor) which is equal to 7.3V, therefore R_1 can be calculated as:

$$R_1 = (VCC-(V_E+V_{BE}))/11*I_B = (9V-1.7V)/11*I_B = 66363\Omega = 66.363K\Omega$$

The next preferred value of 66.363KΩ from resistor pack is 68KΩ

In addition to the values of components we have calculated above and the cable this experiment design requires to have the following equipments:

Parts list

No	Items	Specifications	Quantity
1	Resistors	4.7KΩ, 1KΩ, 18KΩ, 68KΩ resistors	4
2	Microphone or function generator	10mV/2KHz AC supply	1
3	Transistor	NPN transistor BC337	2
4	Capacitors	2x1µF and 6.8µF capacitors	2
5	Oscilloscope	Analogue or digital oscilloscope	1
6	Breadboard	Prototyping board	1
7	Connecting wires	22-gauge solid wire	50Cm
8	Connector	Battery connector	1
9	Battery	9V battery or DC power supply	1
10	Speaker or 100KΩ fixed resistor	8Ω,4watts speaker or 100KΩ resistor	1
11	Voltmeter	Digital multimeter	1

After calculating and choosing devices, connect the circuit without capacitors as shown in the figure 6-37. Then after a thorough visual check that everything is correctly connected, connect the power supply, switch on and use a multimeter set in volt mode to check the voltages on collector, base and emitter of the transistor. If the measured voltages are exactly equal to that we have calculated above you have successfully designed the DC conditions. If there are any considerably wrong voltages, (e.g. more than 20% high or low) check that all the connections on the amplifier are correct, and that you have read the resistor color codes correctly. Any smaller differences may need the value of one or more of the resistors changing.

Try to make collector voltage V_C exactly half of the supply voltage VCC. If it already is, that is great. If not, which is most likely, the first thing to check is that you have correctly calculated the values of R_C and R_E and fitted the nearest preferred value of resistor in both positions. If these resistors are correctly chosen, the base voltage probably needs correcting, as mentioned in "estimate a value for base current” above. If the collector voltage is high, the base voltage will need increasing slightly (try changing R_2 to the next higher preferred value). If the collector voltage is low, decrease the value of R_2. This unfitting of voltage is due to the use of preferred values of resistors rather than the exact calculated values. Make a record of any change you make, if you are changing resistor to increase the voltage, check if really the voltage also increases. After all this process, you will understand the role and effect of each component in transistor amplifier circuit.

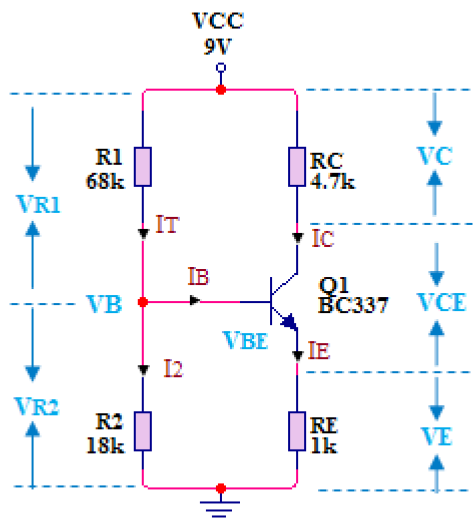


Figure 6-37: DC operating points of common emitter amplifier

chap 6

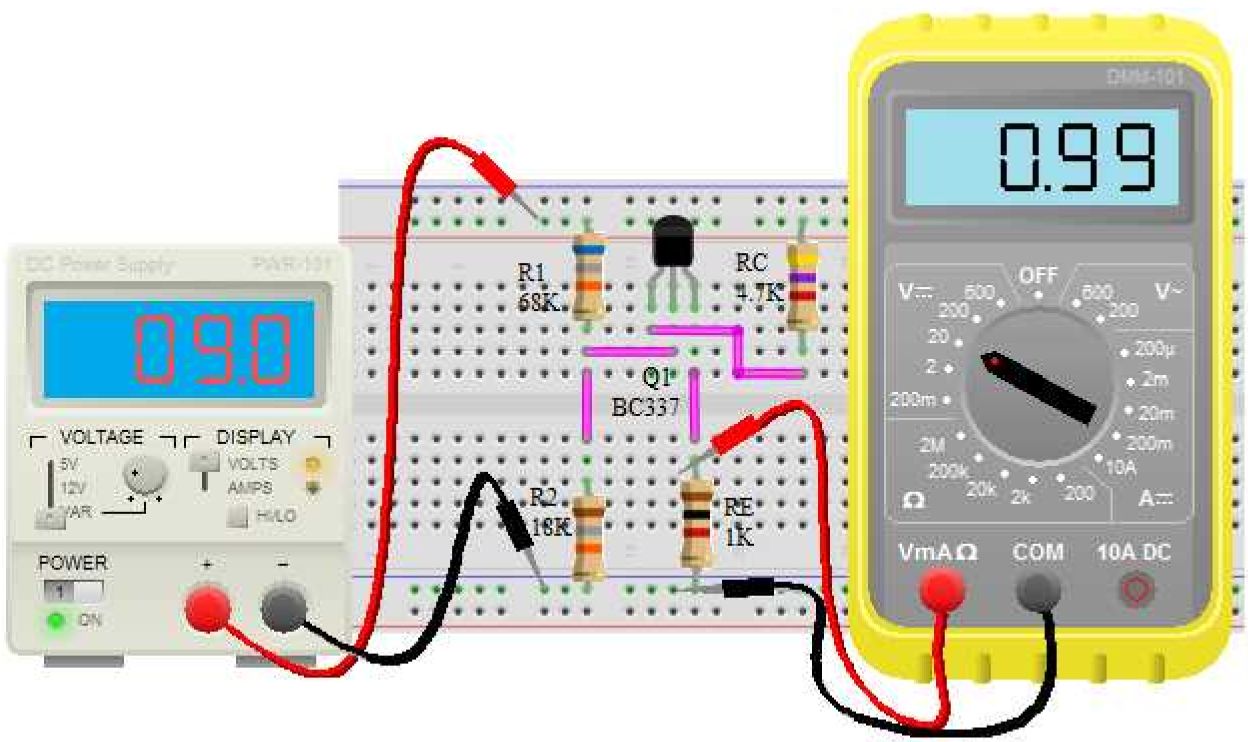


Figure 6-38: Measuring dc operating points of an amplifier

After found out that the DC conditions are fitting, now add the capacitors as shown in the circuit figure 6-39 and apply an ac signal with adjustable amplitude and voltage.

Role of each Component

Within the circuit there are a number of components that provide different functions to enable the overall circuit to operate in the required mode:

- C_1 and C_2: These capacitors provide AC coupling between stages and are used as decoupling capacitors to separate the AC signals from the DC biasing voltage. This ensures that the bias condition set up for the circuit to operate correctly is not affected by any additional amplifier stages, as the capacitors will only pass AC signals and block any DC components. They need to be chosen to provide negligible reactance at the frequencies of operation.
- R_1 and R_2: These resistors provide the bias for the base of the transistor.
- R_C: This is the collector load resistor within the common emitter amplifier.
- C_E: This is a bypass capacitor. The effect of R_E is to reduce the gain of the circuit. Bypassing the resistor enables greater levels of AC gain to be achieved.
- R_E: This resistor in the common emitter amplifier provides a measure of DC feedback to ensure that the DC conditions within the circuit are maintained.

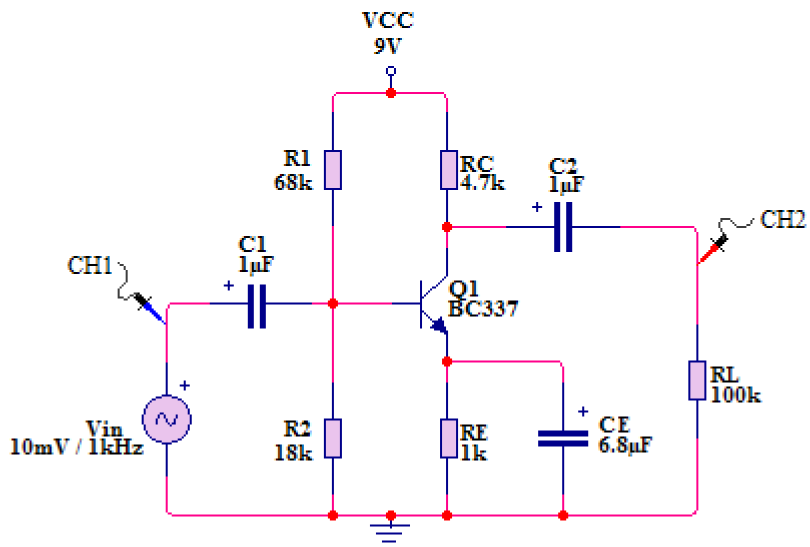


Figure 6-39: Transistor Common Emitter amplifier

Performing the Circuit

Set the function generator to output 10mV and adjust the frequency from 200Hz up to 10KHz. You will find out that at low frequency, the output voltage is low and the voltage increases as the frequency increases. This is because at low frequency signals the total resistance in the emitter leg is equal to R_E+re. At high frequency, the bypass capacitor shorts out the emitter resistor leaving only the internal resistance *re* in the emitter leg resulting in a high gain.

You can calculate the gain A_V by taking the output voltage measured on oscilloscope channel 2 (CH2) divide by the input voltage also measured on channel 1 (CH1) of oscilloscope. Make a record of output voltage at each 200Hz frequency change and after calculate the gain with and without the load resistor R_L.

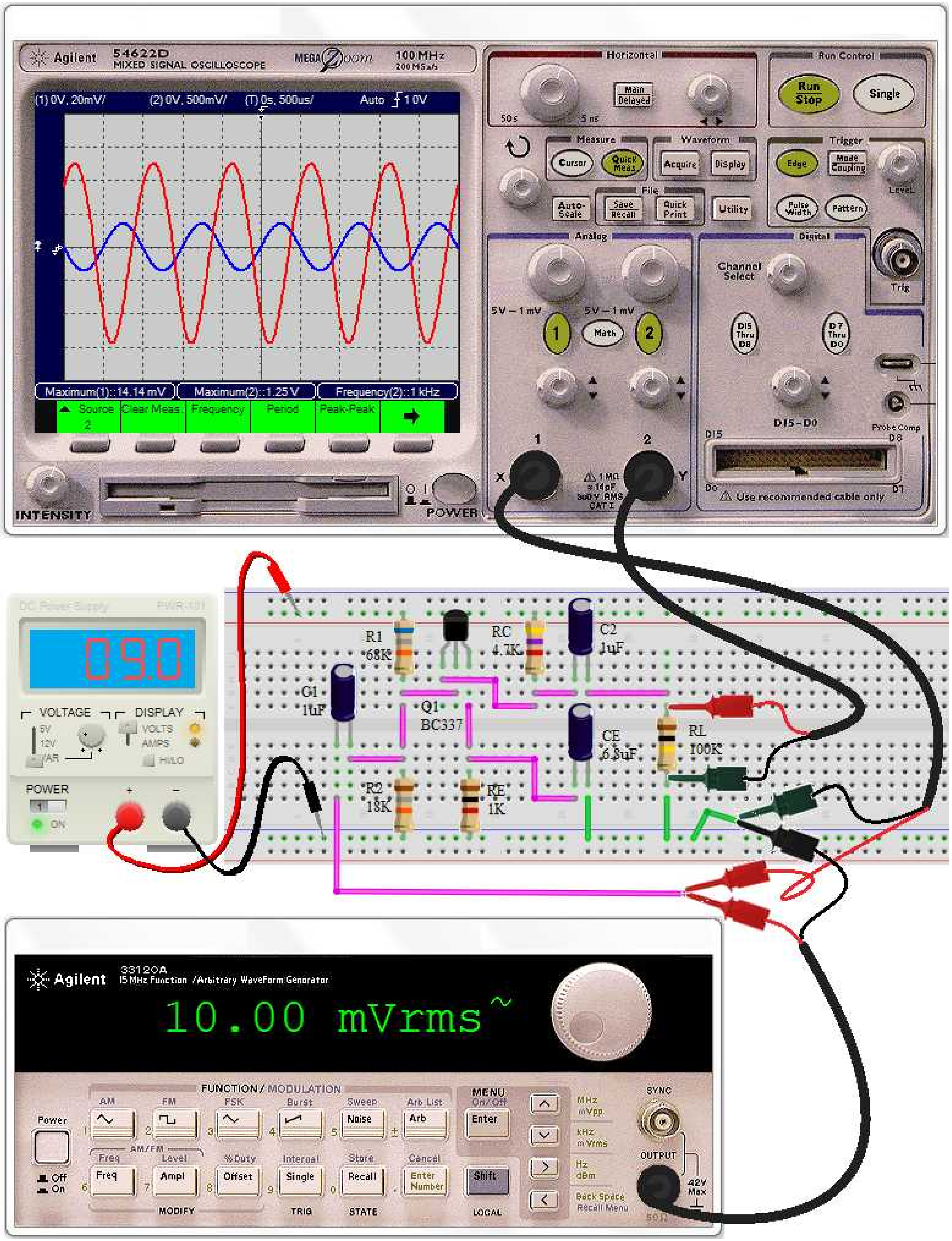


Figure 6-40: Transistor CE amplifier, circuit connection

The output with load resistor will be a little bit less than the output without load resistor. This is due to the influence of the load resistor R_L on the gain since the collector resistor is in parallel with load resistor and the gain becomes:

$$A_V = rc/re = (R_C \| R_L)/re$$

- **Amplifier Cascading**

When you want to increase the gain much more, you can put different amplifier stages in cascade where the output of the first stage becomes the input of the second and so on. When you are using microphone as an input and loudspeaker as output allow the voice to be captured by microphone and turn the speaker up all the way. You may hear a very faint response, but sometimes it can be buried in the little noise. You can amplify the voice signal some more. You can create a second stage of amplification, much like we did with our previous circuit and let output of the first stage go into the input of the next.

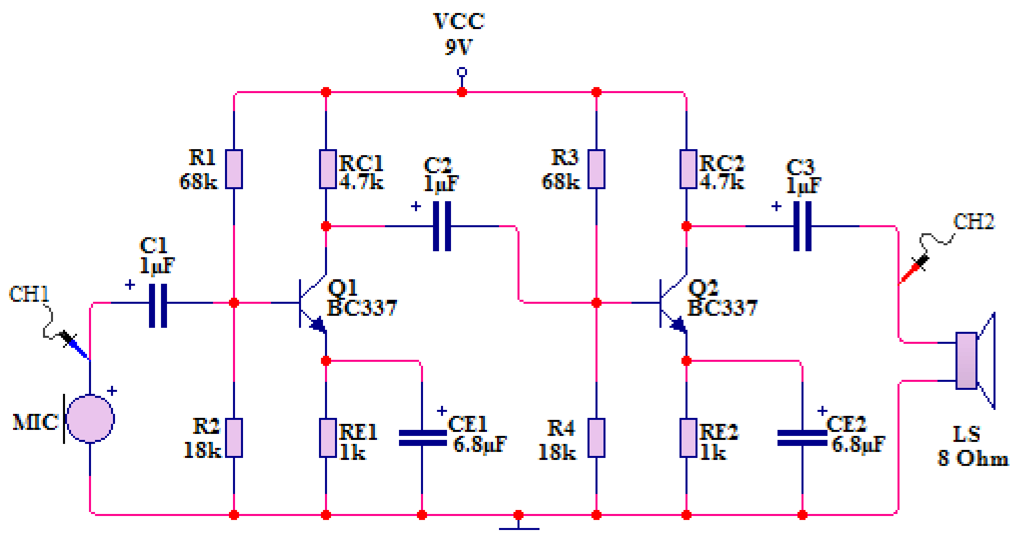


Figure 6-41: Two stages transistor amplifier cascading

However, this amplifier cascading makes the circuit slightly unstable, you can simply drop the gain a bit on the second stage by adding a resistor with low resistance in parallel with R_E or simply a capacitor with low capacitance to the emitter to lower the gain of the second stage a bit, even if will still make for louder and a little bit noisy. This makes transistor amplifiers less useful for day to day application.

- **Application of Common Emitter Amplifier**

Most of the transistor amplifiers are of common emitter type because of large gains in voltage, current and power. Moreover, their input and output impedance characteristics are suitable for many applications.

6. 10. 2. Common Collector Amplifier

The common collector (CC) amplifier, commonly referred to as an emitter-follower, is another type of BJT configuration. The input is applied to the base and the output is taken at the emitter.

There is no collector resistor. The voltage gain of a CC amplifier is approximately 1 (just a little less than unit since the emitter voltage is constrained at the diode drop of about 0.6 volts below the base), but the current gain is always greater than 1. Its function is not voltage gain but current or power gain and impedance matching. Its input impedance is much higher than its output impedance so that a signal source does not have to work so hard. This can be seen from the fact that the base current in on the order of 100 times less that the emitter current. The low output impedance of the emitter follower matches a low impedance load and buffers the signal source from that low impedance.

A common collector amplifier is characterized by:

- High input impedance
- Low output impedance
- High current gain $\gamma=\beta+1$
- Unit voltage gain
- A moderately high power gain
- No phase reversal of the input signal to the output

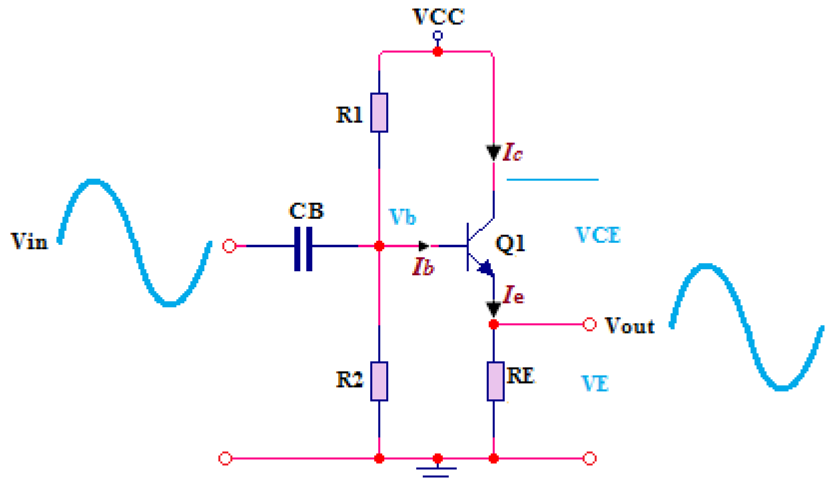


Figure 6-42: Common collector amplifier

Voltage gain A_V: As in all amplifiers, the voltage gain in a CC amplifier is $A_V = V_{out}/V_{in}$. For the emitter-follower, V_{out} is IeR_E, and V_{in} is *Ie(re+Re)*. Therefore, the gain is *IeRe/Ie(re+Re)*. The currents cancel, and the gain expression simplifies to:

$$A_V= R_E/re+R_E$$

It is important to notice here that *the gain is always less than 1*. Because *re* is much less than R_E, then a good approximation is $A_V=1$.

Since the output voltage is the emitter voltage, it is in phase with the base or input voltage. As a result, and because the voltage gain is closer to 1, the voltage follows the input voltage; thus, the common-collector amplifier is also known as *emitter-follower.*

Input Resistance: The emitter-follower is characterized by a high input resistance, which makes it very useful circuit. Because of the high input resistance, the emitter-follower can be used as buffer to minimize loading effects when on circuit is driving another.

The derivation of the input resistance viewed from the base Z_b is similar to that for CE amplifier. In this case, however, the emitter resistor is not bypassed.

$$Z_b=V_b/I_b=Ie(re+R_E)/I_b\approx\beta_{ac}I_b(re+R_E)=\beta_{ac}(re+R_E)$$

In case the emitter resistor R_E is very high compared to the transresistance re, R_E>>re

$$Z_b=\beta_{ac}R_E$$

The input impedance Z_{in} is determined by:

$$Z_{in}=R_B\,\|\,Z_b,\text{ whre } R_B=R_1\,\|\,R_2$$

Experiment 6-10: Common Collector Amplifier Design

Split bias voltage drops about equally across the transistor V_{CE} (or V_{CB}) and V_{RE} (or V_B).

Choose V_B=VCC/2, this means that $R_1=R_2$.

Set to 9 Volts and assume β to 100 and we want our amplifier to work from 10Hz.

Choose I_E=1mA and assume that $I_C = I_E$

Set emitter voltage (V_E) at (VCC/2) −0.7 = (9V/2) −0.7 = 3.8 Volts

$R_E = V_E/I_E$ = 3.8V/0.001A = 3.8KΩ; from resistor standard values we choose 3.9KΩ

$$V_B=[VCC/(R_1+R_2)]\,R_2$$
$$=V_E+0.7=VCC/2$$
$$=4.5\text{ Volts}$$

For an assumed β = 100; as with common emitter bias design, stable operating point, $R_B << (\beta+1)R_E$, i.e. $R_B = R_1\,\|\,R_2 = R_1/2 = (\beta+1)R_E/10 \approx 10R_E$; Therefore $R_1=20R_E$= 76KΩ

$$V_B=[VCC/(R_1+R_2)]\,R_2$$

$$4.5=[9/(76+R_2)]\,R_2 \text{ Therefore } R_2=342/4.5=76K\Omega$$

$R_1=R_2$=76KΩ; from resistor standard values pack the next preferred value we choose 82KΩ.

Choose input capacitor C_B such that its reactance is ≤1/10th of input impedance Z_{in} at min frequency.

$$1/w_{min}C_B=Z_{in}/10;\ C_B=10/w_{min}Z_{in}$$

$$R_B=R_1\,\|\,R_2=(82K\Omega*82K\Omega)/(82K\Omega+82K\Omega)$$
$$=41K\Omega$$

$$re=26mV/I_E=26mV/1mA$$
$$=26\Omega$$

$Z_b = \beta(re+R_E)$

$=100(26\Omega+3900\Omega)=392600\Omega = 392.600K\Omega$

$Z_{in}=R_B \| Z_b = (41K\Omega*392.6K\Omega)/(41K\Omega+392.6K\Omega)=37.1232K\Omega$

$w_{min}=2\Pi f_{min}=2*3.14*10$

$=62.8rad$

$C_B=10/w_{min}\ Z_{in}$

$=10/68.2*37123.2=42.89*10^{-7}=4.289\mu F$

From standard capacitor values, the next preferred value is 4.7µF.

Set the emitter coupling capacitors C_E to 1µF, hence the part list below.

Part list

No	Items	Specifications	Quantity
1	Fixed Resistors	2x82KΩ, 3.9KΩ, 15KΩ 1/2watt	4
2	Transistor	MPS3704 Silicon Amplifier Transistor	1
3	Capacitors	4.7µF and 1µF capacitor	1
4	Breadboard	Prototyping board	1
5	Connecting wires	22 gauge solid wire	50Cm
6	Battery	9V battery or DC power supply	1
7	Function generator	FGN with sinusoidal wave application	1
8	Oscilloscope	Analog or digital oscilloscope	1

chap 6

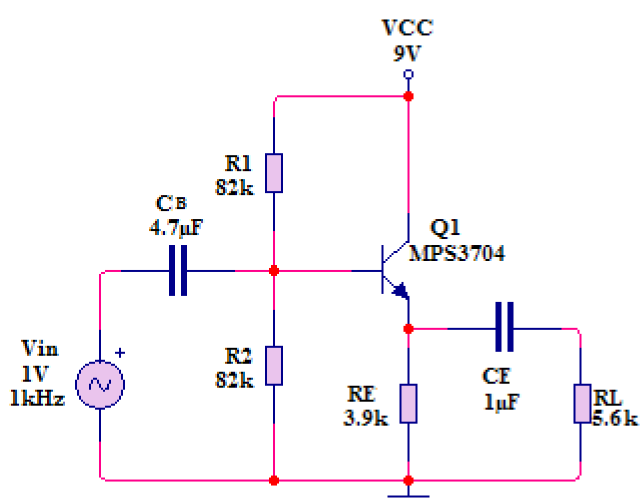


Figure 6-43: Common-collector (emitter-follower) amplifier

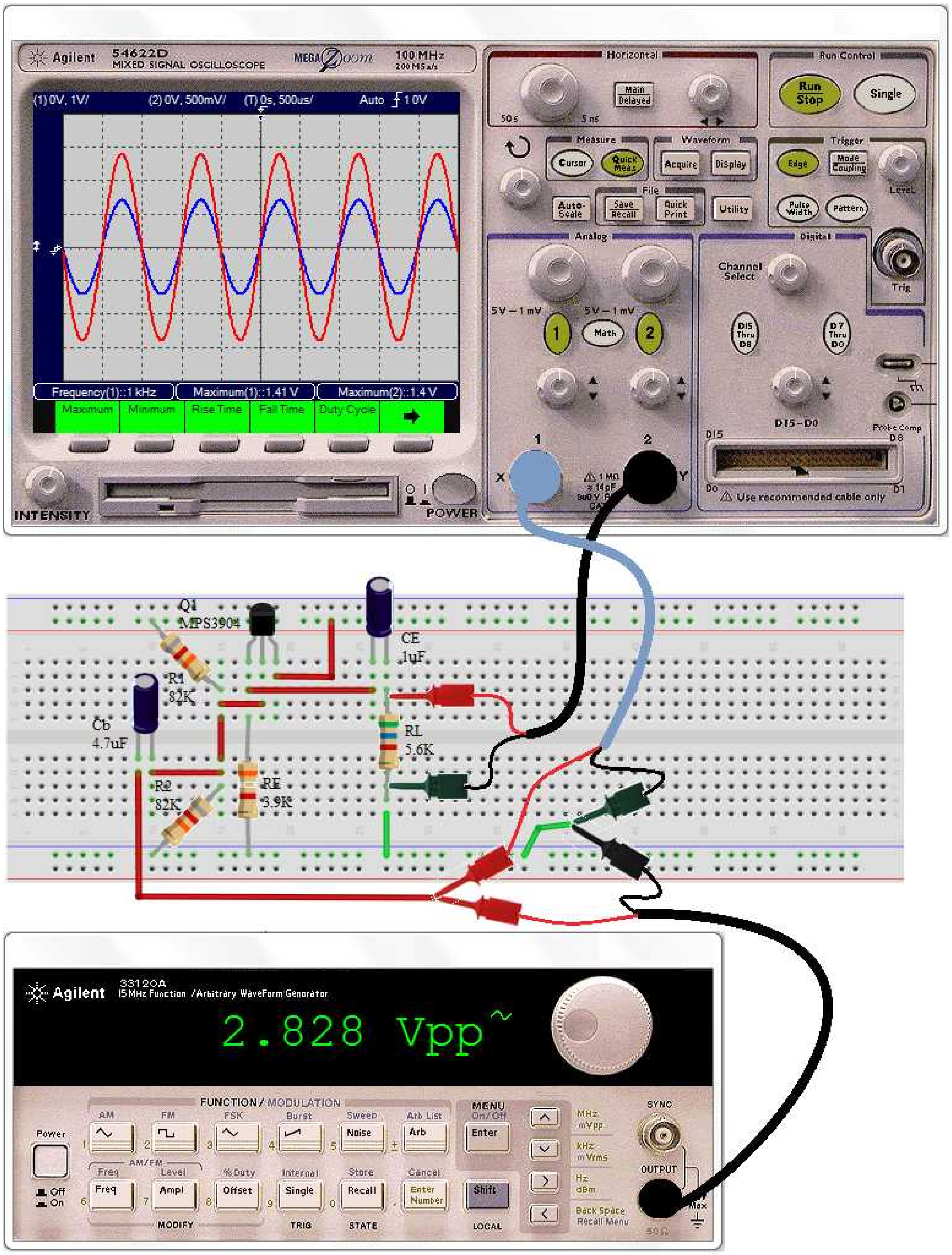


Figure 6-44: Common-collector (emitter-follower) amplifier

chap 6

Procedure

1. Connect the circuit as illustrated in figure 6-44. Connect the power supply after double-checking all connections, especially the transistor leads.
2. Use a voltmeter to measure the dc biasing voltages from the positive of supply (VCC) to the base of transistor (voltage on R_1) and from the base to ground (voltage on R_2). These should be a half of supply for each, i.e. 4.5V on R_1 and on R_2. Measure also the voltage across the emitter resistor R_E (this should be 4.5V−0.7V, that is 3.8V) and the collector to emitter voltage (it should be about VCC-V_E, that is 5.2V).
3. Set channel one (CH1) to 1V/div and channel two (CH2) to 500mV at 500µSec, AC coupling and adjust both ground levels to be at x-axis of the screen. This way, you will be able to observe both the input and the output characteristics at the same time. Setting both channels on the same volt/div you will mess things up by overlapping both input and the output signal and no way to identify each. Connect CH1 to the input, just direct to the output of function generator, and CH2 to the output on terminal of load resistor.
4. Set the signal generator to output a 1 KHz sine wave at 2.828 Vp-p (1V rms), then connect it to the input capacitor C_B. You should see a sine wave at the output (on terminal of load resistor R_L) with amplitude of about 1.4V peak and in phase with the input (a voltmeter will measure 1V rms at the input and output). You may adjust the frequency from below 100Hz up to above 1MHz and the input and the output will be on almost the same amplitude.
5. Compare the input voltage to the output voltage. You will notice that V_E simple tracks or follows V_B, just 0.7 V under in DC. The AC signal voltages are the same for both (signal voltage gain ≈ 1), just the DC is different. This is the reason why the *common-collector* circuit is also known as an *emitter follower* because the emitter voltage follows the base voltage.
6. Increase the input signal to 2.5V rms (7Vp-p) and above and observe the output signal on the oscilloscope. As you lower the collector current (V_B decreasing), you will see the output waveform clip on negative peaks as the collector current is cut off. Raising collector current will eventually result in distortion on positive peaks as the transistor enters saturation.

chap 6

In this circuit of common-collector amplifier, figure 6-44, you will find out that as you replace the load resistor with a lower value the output voltage is still not big that is because the load is so small. The circuit is being overloaded. For example, substitute 1 KΩ, 100 Ω, and 10 Ω resistors (10 Ω resistor is close to the loudspeaker resistance, i.e 8Ω or 16 Ω) for R_{load}, reducing the input voltage at each value, so that the output waveform remains undistorted. Lower resistance loads can only be driven at lower voltages because the ac currents in the transistor are much higher at lower values of load resistance. You will also see the output signal begin to "lag" behind the input signal at these low load values. This is because the impedance of the output coupling capacitor at 1KHz becomes significant for loads below 100, introducing phase shift in a series RC circuit. As seen, lowering the load distorts the output. As the main application of the common-collector is not to overload the input (impedance matching) because of its high input impedance Z_{in}, using low load needs to change the biasing conditions such as reducing also the value of R_E, and both R_1 and R_2. As the emitter coupling capacitor affects the signal for low load this needs to be increased so that the gain remains to be the unit.

Applications of Common Collector Amplifier

The characteristics of common collector amplifier allow it to be used:

- For the impedance matching for connecting a circuit having high output impedance to one having low impedance of input.
- For circuit isolation
- As a two way amplifier since it can pass a signal in either connection
- For switching circuits

6. 10. 3. Common Base Amplifier

A common base amplifier is a type of transistor amplifier where the base terminal is common for both input and output and is characterized as having a relatively low input and a high output impedance and current gain less than 1. The voltage gain, however, can be quite large. The standard configurations appear in figure 6-45.

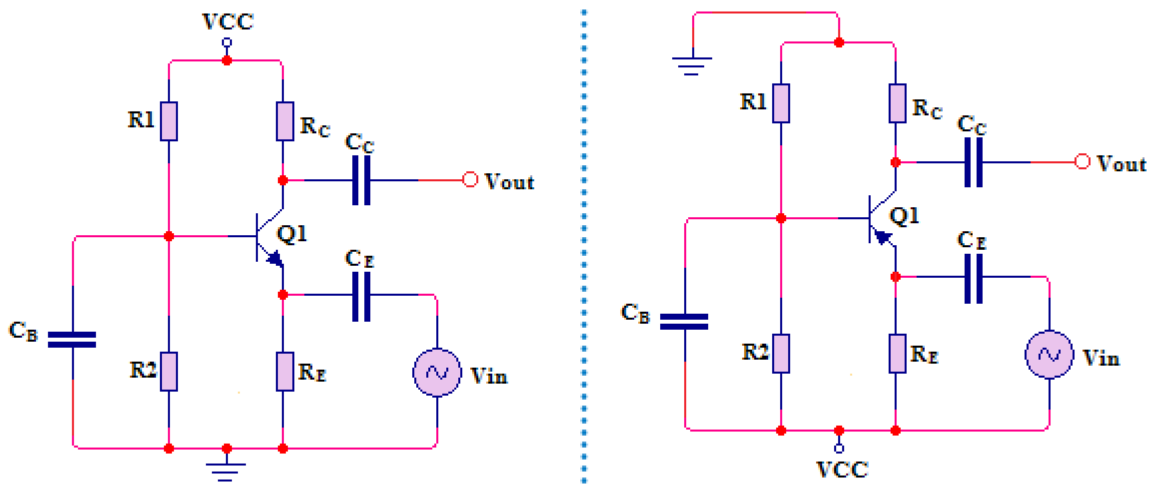


Figure 6-45: Common base amplifier, (a) CB amplifier with NPN transistor, (b) CB amplifier with PNP transistor

For both NPN and PNP circuits, it can be seen that for the common base amplifier circuit, the input is applied to the emitter, and the output is taken from the collector. The common terminal for both input and output circuits is the base. The base is biased by a voltage divider resistor network composed by R_1 and R_2 and the emitter is biased through an emitter resistor R_E. The capacitors C_B, C_C, and C_E are used, in the circuit, to block the DC bias currents and pass only the AC signal currents. At high frequency or in AC mode, the capacitor C_B shorts the base of transistor to ground and open for DC mode.

The transistor output impedance r_o, is typically in the mega-ohm range, can be ignored because is in parallel with collector resistor R_C, this is sufficiently larger than the parallel resistance R_C to permit the approximation $r_o \| R_C \approx R_C$. As result, the output impedance of common-base amplifier is given as:

$$Z_{out}=R_C$$

$$Z_{out}=R_C \| R_L \text{ with load connected}$$

The input impedance Z_{in} is given as:

$$Z_{in}=R_E \| re$$

Because the transiresistance *re* is always very low, the input impedance is approximated to be:

$$Z_{in} \approx re$$

The voltage gain A_v of common base amplifier is calculated as:

$$V_o=-I_oR_c=-(-I_C)R_C=\alpha R_C$$

With $I_e=V_{in}/re$; so that $V_o=\alpha(V_{in}/re)R_C$

Thus, $A_V= V_o/V_{in}= \alpha\, R_C/re \approx R_C/re$

The current gain Ai by assuming that the emitter resistor R_E is very large to the transiresistance *re*, $R_E \gg re$, yields

$$I_e=Ii$$

$$I_o= -\alpha I_e= -\alpha Ii$$

$$Ai= I_o/Ii = -\alpha \approx -1$$

chap 6

The fact that A_V is a positive number shows that V_o, output, and V_{in}, input, are in phase for common-base configuration.

A common base amplifier is such amplifier which has the following characteristics

- High voltage gain but maximum current gain α of 1. $\alpha=I_E/I_C\approx 1$.
- It has a voltage gain equal to 10 times of common emitter amplifier
- Has a high output impedance and low input impedance
- Input is given to emitter via a coupling capacitor
- Output is capacitively coupled from collector
- Base is at ac ground
- There is no phase inversion from input to output and the output is approximately the same amplitude as input
- As the current gain α is equal to 1 so the power is approximately equal to voltage gain

Experiment 6-11: Common-base amplifier design

Although the common-base amplifier is not widely used but is still have some application as in high frequencies operations. To design a common-base amplifier you must know the input signal size (amplitude and frequency range) and the required amplification factor. Also, you have to know the DC voltage that will be used for the transistor bias (VCC). A common base amplifier has no current amplification; therefore, we only need to calculate the voltage amplification. The design can begin with the DC equivalent circuit by considering that the capacitors are open for DC and short for AC current.

- Choosing biasing voltage, VCC, and calculating voltage, V_C, across the collector resistor

We want the circuit to operate on 12V battery at frequency of 1KHz. We want also the voltage gain to be 100. Let choose BC548 transistor and the transistor amplification factor β to be 100.

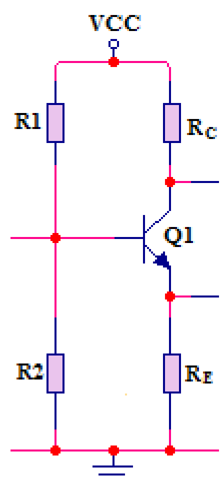


Figure 6-46: CB amplifier DC equivalent circuit

The calculation of dc biasing conditions is similar to the common-emitter amplifier. The Q-point or dc working conditions are determined by DC batteries along with resistors R_E and R_C. In other words, values of I_E, I_B and V_{CE} are decided by V_{CC}, R_E and R_C. When no signal is applied to the input circuit, the output just sits at the Q-point so that there is no output signal.

Usually, we place the Q-point in the middle of V_{CE}. This means that:

$$V_{CE} = VCC/2 = 12V/2 = 6V$$

We know that the AC input must be able to provide enough current for the emitter and the collector circuit. Therefore, we want to maintain the emitter current as low as possible. We can safely choose a large resistor such as 4.7KΩ for R_E. It can be proved that the voltage gain A_V is calculated from this formula:

chap 6

$$Av = R_C/re$$

To calculate the gain we must first calculate the internal emitter resistor *re*. But we cannot calculate it since we do not know the emitter current. So we must find a way to calculate the emitter voltage first. Let assume the collector current is equal to the emitter current, $I_C = I_E$

$$re = 25mV/I_C = 25mV / I_E$$

We replace this formula to the gain amplification formula,

$$A_V = R_C /(25mV/I_E)$$
$$A_V = I_E{*}R_C/25mV = I_C{*}R_C/25mV = V_C/25mV$$

Thus, $V_C = I_C{*}R_C = A_V{*}\ 25mV = 100{*}25mV = 2.5V$

- Calculating the emitter voltage V_E

Now we can use the Kirchhoff's law to calculate the emitter voltage V_E.

$$VCC = V_C + V_{CE} + V_E$$
$$V_E = VCC - V_C - V_{CE} = 12V - 2.5 - 6V = 3.5V$$

- Calculating the emitter current I_E and estimating the collector current

Since we already know the value of emitter resistor, 4.7KΩ, now we can calculate the emitter current I_E.

$$I_E = V_E/R_E = 3.5V/4.7K\Omega$$
$$= 0.000745A$$

- Choosing the value of the collector resistor R_C

From the formula $V_C = I_C * R_C$ and by considering the emitter current to be equal to the collector current we can calculate the collector resistor R_C,

$R_C = V_C / I_C = 2.5V/0.000745A = 3357\Omega$, we choose 3.3KΩ from the next preferred standard resistor values.

- Calculating transistor transresistance *re*

Now we can calculate the emitter internal resistance *re*.

$$re = 25mV / I_E = 0.025V/0.000745 = 33.6\ \Omega$$

- Calculating the base voltage

Now we can calculate the base voltage;

chap 6

$$V_B = V_{BE} + V_E = 0.7 + V_E = 0.7 + 3.5V = 4.2V$$

From the voltage divider formula, we can now calculate the resistor ratio needed to get this voltage, 4.2V:

$$V_B = VCC * R_2/(R_1 + R_2)$$

- Calculating the DC bias voltage divider network current I_{VD} and the base current

Let's assume that we choose to design a firm voltage divider. This means that the current through the voltage divider (I_{VD}) must be at least 10 times greater than the base current. In other words:

$$I_{VD} = 10 * I_B$$

To ensure adequate bias stability, the current flowing through R_2 should be about 10 times greater than the base current I_B so the current flowing through R_1 and R_2 (I_{VD}) will be simply I_B x 11. For I_C equal to 0.000745A and transistor amplification factor β equal to 100, I_{VD} is equal to:

$$I_B = I_C/\beta = 0.000745/100 = 0.00000745A = 7.45\mu A$$
$$I_{VD} = I_B * 11 = 7.45\mu A * 11 = 81.95\mu A$$

- Calculating R_1 and R_2

A general rule of thumb is a value of at least 10 times I_B flowing through the resistor R_2. Transistor base/emitter voltage, V_{BE} is fixed at 0.7V (silicon transistor) then this gives the value of R_2 as:

$$R_2 = (V_E + V_{BE})/10 * I_B = 3.5V + 0.7V/10 * 7.45\mu A = 56375\Omega = 56.375K\Omega$$

We choose 56KΩ as the next preferred standard resistor value from resistor pack.

If the current flowing through resistor R_2 is 10 times the value of the base current, then the current flowing through resistor R_1 in the divider network must be 11 times the value of the base current. The voltage across resistor R_1 is equal to VCC–V_B (V_{RE} + 0.7 for silicon transistor) which is equal to 7.8V, therefore R_1 can be calculated as:

$$R_1 = (VCC-(V_E+V_{BE}))/11*_{IB} = (12V-(3.5V-0.7V))/11*I_B = 95179\Omega = 95.179K\Omega$$

We choose the next preferred value of 95.179KΩ from resistor standard resistor pack which is 100KΩ.

- Choosing coupling capacitors C_E, C_B, and C_C

The coupling capacitors are chosen to provide no reactance to the circuit. The input signal source like a function generator also provide a resistance which in most of the case is 50Ω or 75Ω which has to be taken in consideration for ac mode. The input signal is considered to have a changing voltage and frequency, thus the coupling capacitors are taken by assumptions. Let's choose all capacitors to be 1μF.

Part list

No	Items	Specifications	Quantity
1	Emitter and collector resistors	4.7KΩ ½ watt, 3.3KΩ ½ watt	2
2	R_1 and R_2	56KΩ ½ watt, 100KΩ ½ watt	2
3	C_E, C_B, C_C coupling capacitors	1μF capacitor	3
4	12V dc supply	12V dc supply	1
5	0.1V/1KHz ac supply	FGN with sinusoidal wave application	1
6	Breadboard	Breadboard	1
7	Oscilloscope	100MHz Tektronix oscilloscope	1
8	Load resistor	10KΩ ½ watt resistor	1

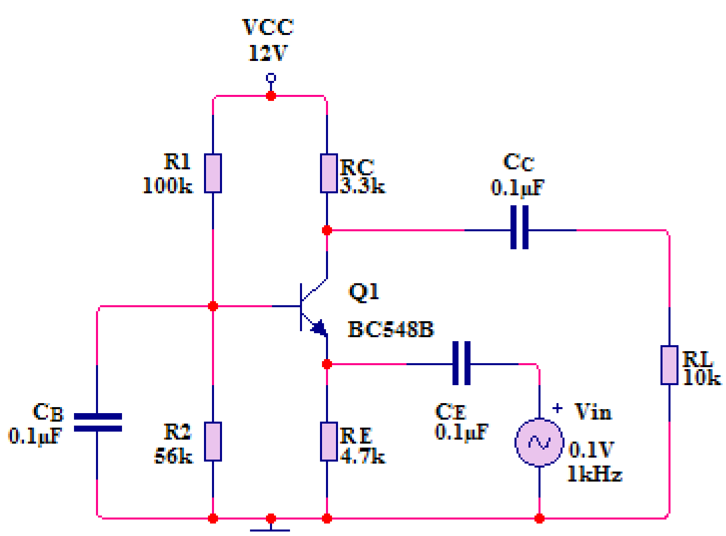


Figure 6-47: Circuit experiment for common-base amplifier

Procedure

1. Connect the circuit as illustrated in figure 6-50. Connect the power supply after double-checking all connections, especially the transistor leads.
2. Use a voltmeter to measure the dc biasing voltages from the positive of supply (VCC) to the base of transistor (voltage on R_1) and from the base to ground (voltage on R_2). These should be 7.8V on R_1 and 4.2V on R_2. Measure also the voltage across the emitter resistor R_E (this should be 3.5V) and the voltage across the collector resistor RC (this should be 2.5V) and collector to emitter voltage V_{CE} (it should be about VCC-V_E, that is 6V).

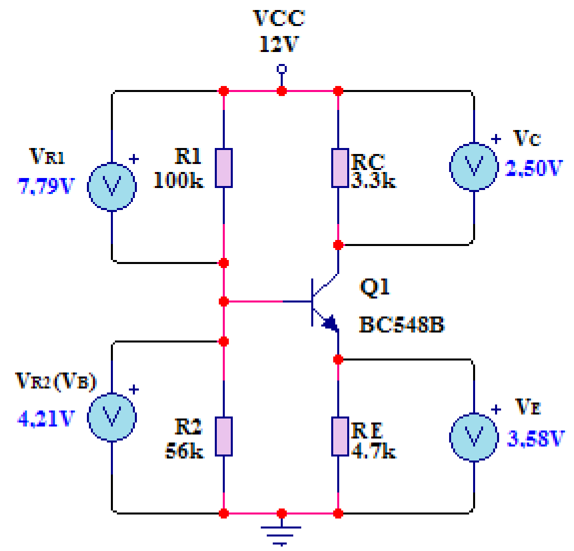


Figure 6-48: Circuit for measuring biasing dc conditions of CB amplifier

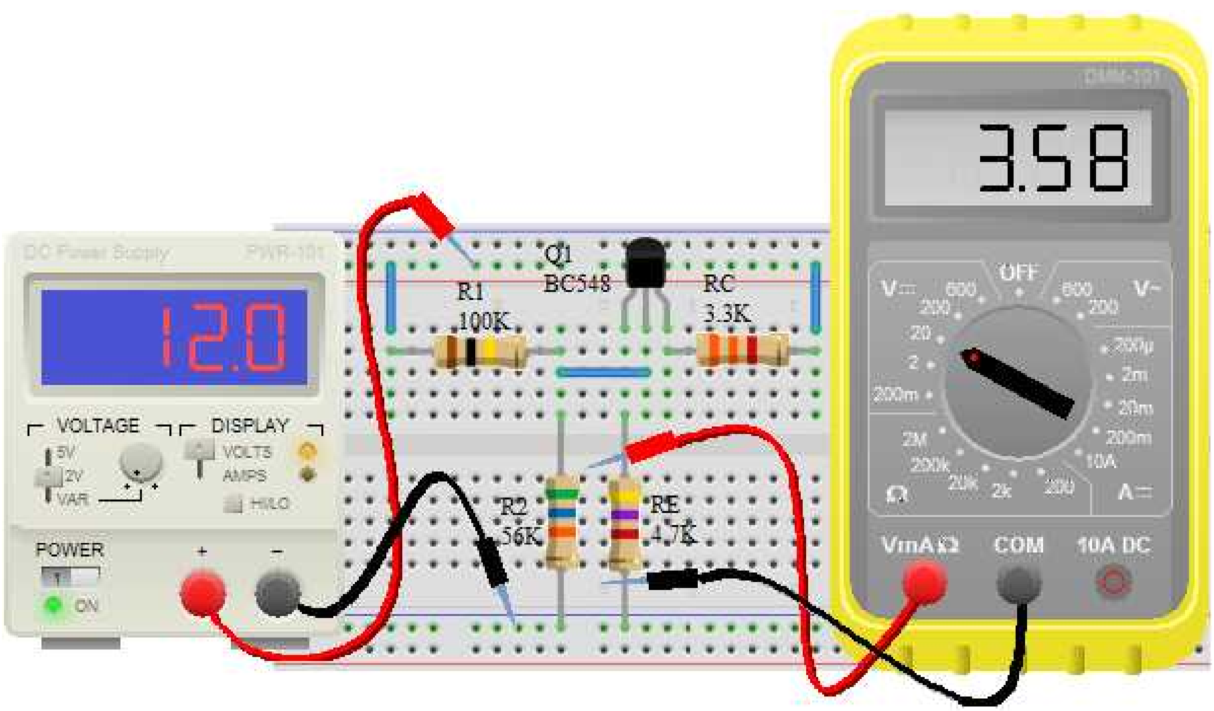


Figure 6-49: Circuit connection for measuring biasing dc conditions of CB amplifier

chap 6

3. Set both channel one (CH1) and channel two (CH2) to 100mV/div at 200µSec time/div, AC coupling and adjust both ground levels to be at x-axis of the screen. You may be able to observe both the input and the output characteristics at the same time.

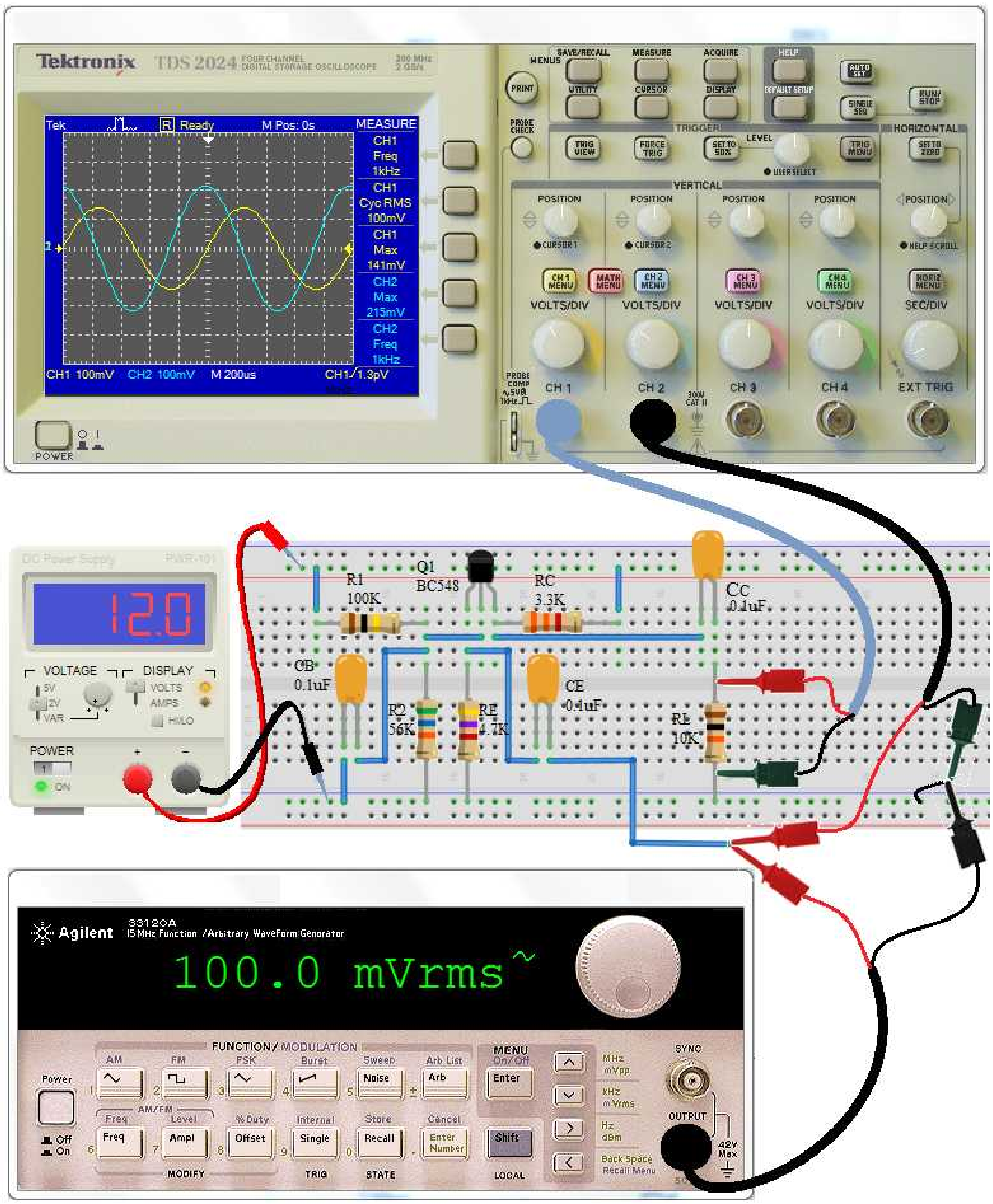


Figure 6-50: Common-base amplifier circuit connection

4. You can change volt/div and time/div as you change the input voltage and frequency in such way that you can view both input and the output characteristics. At this time, make sure that at least one period is visible on screen of oscilloscope for easy calculations. Connect CH1 to the input, just direct to the output of function generator, and CH2 to the output on the terminal of the load resistor.
5. Set the signal generator to output a 1 KHz sine wave at 282.8mVp-p (0.1V rms), then connect it to the input capacitor C_E. You should see a sine wave at the output (on terminal of load resistor R_L) which is in phase with the input. You may adjust the frequency from below 100Hz up to above 1MHz and the output voltage will be increased significantly. As you continue to increase the frequency, the output will be increased to the level where the transistor will be driven into saturation and the output signal will be clipped off. Also by maintaining the frequency and changing the input voltage there will be a level where the transistor will be driven into saturation and the output will be clipped off.
6. Compare the input voltage to the output voltage. For any voltage and frequency setting divide the output, measured just on channel two, with the input, measured on channel one to get the gain.
7. Open load resistor and connect multimeter set in amps range between the load resistor and ground to measure current through the emitter terminal and record it. Connect also ammeter at the input, just between the positive of function generator and the capacitor, to measure the input current and record it. You will notice that as you increase the input voltage the ratio of the current through the load resistor and the input current is almost equal to 1 for a wide range of input voltage variation.

Application of Common Base Amplifier

One of the uses of common base amplifier is in matching a low-impedance circuit to a high-impedance circuit. The common base amplifier configuration is not used as widely as other transistor amplifier configurations. However it does find uses with amplifiers that require low input impedance levels. One application is for moving-coil microphones preamplifiers; these microphones have very low impedance levels. Another application is within VHF (very high frequency) and UHF (ultra high frequency) RF (radio frequency) amplifiers where the low input impedance allows accurate matching to the feeder impedance which is typically 50Ω or 75Ω.

This circuit is usually found in high-frequency amplifiers because its input capacitance does not suffer from the Miller effect, which degrades the bandwidth of the common emitter configuration, and because of the relatively high isolation between the input and output.

It is also used as current buffer since it has a current gain of approximately unity. When the circuit is preceded by a common emitter stage, it is called a cascode circuit. The cascade circuit has the benefits of both configurations, such as high input impedance and isolation.

Operational Amplifier Circuits

Chapter seven

Objectives

After completing this chapter, you should

- Be able to explain the basic operation of comparator circuit
- Be able to analyze summing amplifiers, averaging amplifiers, and scaling amplifiers
- Have an understanding the operation of op-amp integrator and differentiators
- Be able to design light activated switch
- Be able to design temperature activated switch
- Be able to design the controller for speed and direction of DC motor

Further reading
Study aids for this chapter are available at
• Design with Operational Amplifiers and Analog Integrated Circuits; 4th Ed; Sergio Franco; McGraw Hill; 672 pages; 2014.

Early operational amplifier (op-amps) were used primarily to perform mathematical operations such as addition, subtraction, integration, and differentiation; hence the term *operational*. These early devices were constructed with vacuum tubes and worked with high voltages. Today's op-amps are linear integrated circuits (ICs) that are relatively low dc supply voltages and are reliable and inexpensive.

In this chapter, linear integrated circuits (ICs), in which many transistors, diodes, resistors, and capacitors are fabricated on a single silicon chip and packaged in a single case to form an operational amplifier (Op-amp), are introduced. The manufacturing process for ICs is complex and beyond the scope of this coverage.

In our study of ICs, we will treat the entire circuit as a single device. That is, you will be more concerned with what the circuit does from external point of view and will be less concerned about the internal, component-level operation.

7. 0. Introduction to the Integrated Circuit

An integrated circuit (also referred to as monolithic integrated circuit, an IC, a chip, or a microchip) is a set of electronic circuits on one small plate (chip) of semiconductor material, normally silicon. This can be made much smaller than a discrete circuit made from independent components.

An integrated circuit is defined as: A circuit in which all or some of the circuit elements are inseparably associated and electrically interconnected so that it is considered to be indivisible for the purposes of construction and commerce.

Integrated circuits are used in virtually all electronic equipment today and have revolutionized the world of electronics.

Computers, mobile phones, and other digital home appliances are now inextricable parts of the structure of modern societies, made possible by the low cost of producing integrated circuits.

7. 0. 1. IC Classifications

Integrated circuits can be classified into analog, digital and mixed signal (both analog and digital on the same chip).

Digital integrated circuits can contain anything from one to millions of logic gates, flip-flops, multiplexers, and other circuits in a few square millimetres. The small size of these circuits allows high speed, low power dissipation, and reduced manufacturing cost compared with board-level integration. These digital ICs, typically microprocessors, DSPs (Digital Signal Processors), and micro controllers, work using binary mathematics to process "one" and "zero" signals.

Analog ICs, such as sensors, power management circuits, and operational amplifiers, work by processing continuous signals. They perform functions like amplification, active filtering, demodulation, and mixing. Analog ICs ease the burden on circuit designers by having expertly designed analog circuits available instead of designing a difficult analog circuit from scratch.

ICs can also combine analog and digital circuits on a single chip to create functions such as analog to digital converter and digital to analog converters. Such mixed-signal circuits offer smaller size and lower cost, but must carefully account for signal interference.

Modern electronic component distributors often further sub-categorize the huge variety of integrated circuits now available:

- Digital ICs are further sub-categorized as logic ICs, memory chips, interface ICs (level shifters, serializer/deserializer, etc.), power management ICs, and programmable devices.
- Analog ICs are further sub-categorized as linear ICs and RF ICs.
- mixed-signal integrated circuits are further sub-categorized as data acquisition ICs (including Analog to Digital converters, Digital to Analog converter, digital potentiometers) and clock/timing ICs.

7. 0. 2. IC Labelling and Manufacture

Most integrated circuits large enough to include identifying information include four common sections: the manufacturer's name or logo, the part number, a part production batch number and/or serial number, and a four-digit code that identifies when the chip was manufactured. Extremely small surface mount technology parts often bear only a number used in a manufacturer's lookup table to find the chip characteristics.

The manufacturing date is commonly represented as a two-digit year followed by a two-digit week code, such that a part bearing the code 8341 was manufactured in week 41 of 1983, or approximately in October 1983.

Silicon Labelling and Graffiti

To allow identification during production most silicon chips will have a serial number in one corner. It is also common to add the manufacturer's logo. Ever since ICs were created, some chip designers have used the silicon surface area for surreptitious, non-functional images or words. These are sometimes referred to as Chip Art, Silicon Art, Silicon Graffiti or Silicon Doodling.

IC Chip Identification

"IC" stands for "integrated circuits," are used in a number of applications, including digital signal processing, timing, and signal amplification. Once identified, you will know what applications the IC is designed for.

Instructions

- Read the serial number from the top side of the IC. The top side of the IC is facing up when the chip is standing on its pins. You may require a magnifying glass while reading the IC serial information.
- Open your internet browser and enter the IC's serial number into either the Google or Yahoo search engines. Once searched you should find a link from the manufacturer's website with a PDF datasheet available for download. Download and open it.
- Read over the datasheet and you will find technical information on the IC's performance and voltage characteristics. These characteristics are necessary information if a user is to implement the IC into a design scheme. For an example, the LM324 datasheet is available for download.

How to Read an IC Part Number

Reading an IC (integrated circuit) part number is process which will allow the reader to determine the chip's manufacturer and technical specifications. All IC chips have a two-part serial number. The first part of the serial number delineates the manufacturer's information. The second part of the serial number indicates the IC's technical specifications. Many IC manufacturers produce identical chips with the same technical specifications. In the case of the serial number "LM324," the "LM" field indicates the manufacturer National Semiconductor and the "324" field indicates that the chip is a dual op-amp IC. The manufacturer and maker of an IC can be referenced easily by obtaining a datasheet that corresponds to the IC's serial number.

Integrated Circuit "Chip" Identification

No	IC Prefix	Manufacturer
1	ALDxxx	Advanced Linear Devices
2	ADxx, OPxx	Analog Devices (PMI)
3	OPAxx, INAxx	Burr-Brown

4	ELxxx	Elantec
5	uAxx, RCxx, KAxx, LMxx	Fairchild
6	MBxxx	Fujistu
7	CAxx, HAxx, HCxx, HFxx	Intersil
8	LTxx, LMxx, LFxx, OPxx	Linear Technology
9	MAXxx, ICLxx	Maxim
10	MICxxx	Micrel
11	MCxxxx, MCxxxxx, etc	Motorola
12	LMxx, LFxx, LHxx, CLCxx	National Semiconductor
13	NExx, SAxx	Philips Semiconductor
14	BAxxx	Rohm
15	LMxx, MCxx, TSxx, etc.	ST Microelectronics
16	TLxx, TLCxx, TLVxx	Texas Instruments (TI)
17	TAxxx	Toshiba

Table 7-1: Some examples of IC manufactures

7. 0. 3. Sinking and Sourcing

IC is said "Sinking" when its output goes low and the current from the supply flows through the load and switch it on. This is called "sinking" current because the current is sourced from VCC and flows through the load and to 0V from the IC.

IC is said "Sourcing" When the output of IC goes high, current will flow through the load to the ground and switch it on. This is called "sourcing" current because the current is sourced from the IC and flows through the load to 0V (ground).

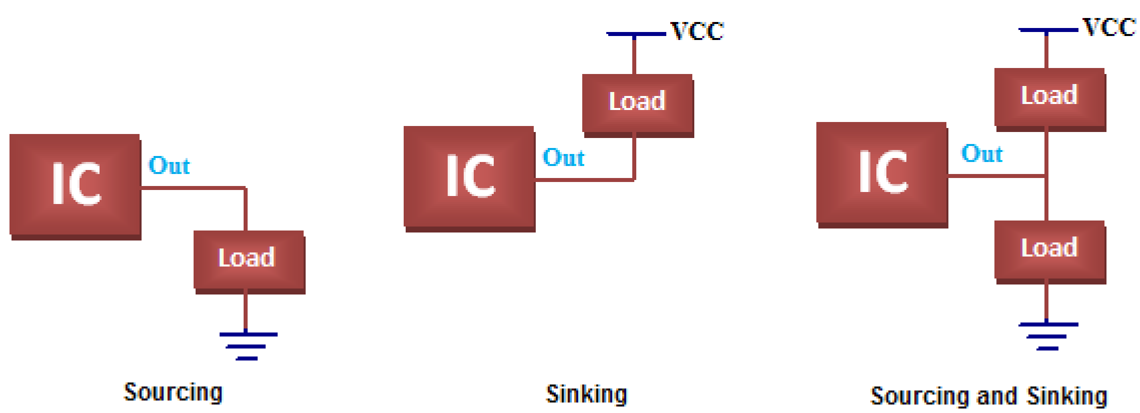


Figure 7-1: Integrated circuit sinking and sourcing

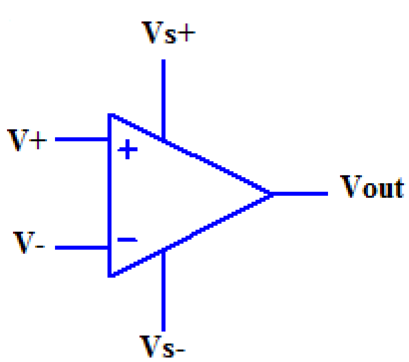
Sinking and sourcing can also be used together so that two connected loads can be alternately switched on and off.

The load(s) could be anything that can be switched on and off, such as LEDs, lamps, relays, motors or electromagnets.
Unfortunately, these devices have to be connected to the output in different ways because the output of the IC can only source or sink a small current. In practice you have to make sure that the power supply can deliver enough current for both the load and IC otherwise you need to interface it with other switching devises such as transistor.

7. 1. Operational Amplifier

An operational amplifier (op-amp) is a DC-coupled high-gain electronic voltage amplifier with a differential input and, usually, a single-ended output. An op-amp produces an output voltage that is typically hundreds of thousands of times larger than the voltage difference between its input terminals.
Operational amplifiers had their origins in analog computers, where they were used to do mathematical operations in many linear, non-linear and frequency-dependent circuits. Characteristics of a circuit using an op-amp are set by external components with little dependence on temperature changes or manufacturing variations in the op-amp itself, which makes op-amps popular building blocks for circuit design.
Op-amps are among the most widely used electronic devices today, being used in a vast array of consumer, industrial, and scientific devices.

The circuit symbol for an op-amp is shown in figure 7-2, where:



- V_+: non-inverting input
- V_-: inverting input
- V_{out}: output
- V_{S+}: positive power supply
- V_{S-}: negative power supply

Figure 7-2: Op-amp Circuit symbol

The power supply pins (V_{S+} and V_{S-}) can be labelled in different ways. Often these pins are left out of the diagram for clarity, and the power configuration is described or assumed from the circuit. The amplifier's differential inputs consist of a V_+ input and a V_- input, and ideally the op-amp amplifies only the difference in voltage between the two, which is called the differential input voltage.

7. 1. 1. Powering up the Op-Amp

In order to function, the op-amp must be connected to an external power supply. Since we want to produce both positive and negative output voltages, we need both positive and negative voltages for the power supply.

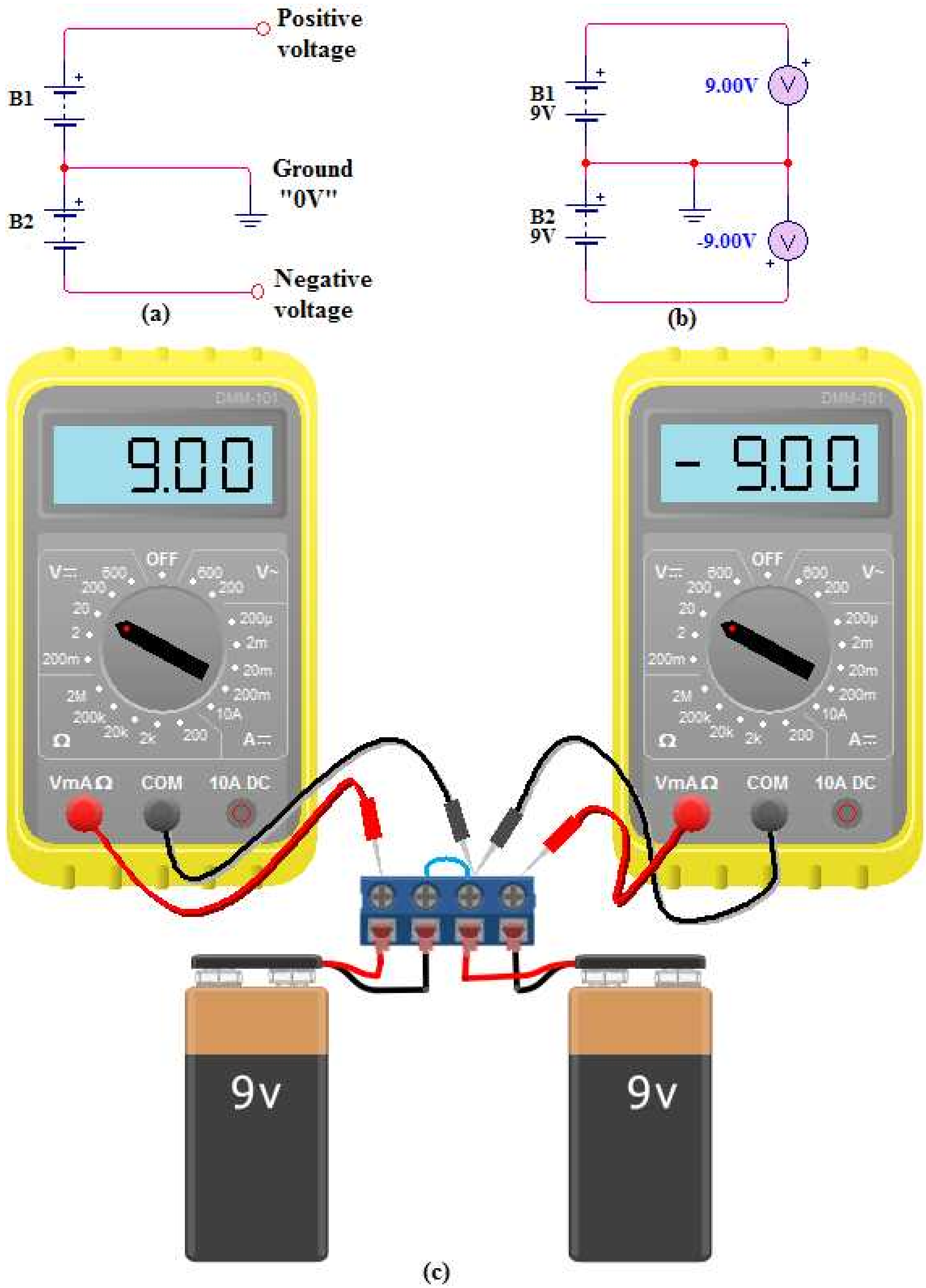


Figure 7-3: Dual polarity power supply using two dc sources; (a) Circuit principle, (b) Circuit example, (c) Circuit example real world connection

chap 7

These are labelled VCC+ and VCC- on the diagram. For a 741, the nominal values are VCC+ =15 V and VCC– = –15 V. To avoid clutter, we will show the power supply terminals (pin 4 for VCC– and pin 7 for VCC+) on any of the subsequent circuit diagrams. Yet, they must be connected or your amplifier will not operate.

Note that there is no ground terminal on the op-amp. The zero reference point is established by the external circuit and is not important to the op-amp itself.

Dual polarity power supply consists of a circuit capable of powering both positive voltage and negative voltage. This can be achieved by using two sources of DC voltage connected in series and by taking ground terminal at junction point of positive and negative terminals of the two batteries thereafter, the remaining terminals serve as positive supply and negative supply as shown in the circuit figure 7-3.

The other method for getting dual polarity is using both positive and negative voltage regulator (78xx and 79xx voltage regulators) having the same supply (rectifier for example). Principally when pair equal capacitors are connected in series and supplied by a dc source, their point of connection serves as ground and the positive voltage and negative voltage are taken directly to the terminal of the main supply respectively. It is with this property used to get dual polarity when the main dc source is rectifier but has the main disadvantage that the voltage reaching on any polarity side is halved with respect to the input main supply. This principle is shown in the circuit figure 7-4.

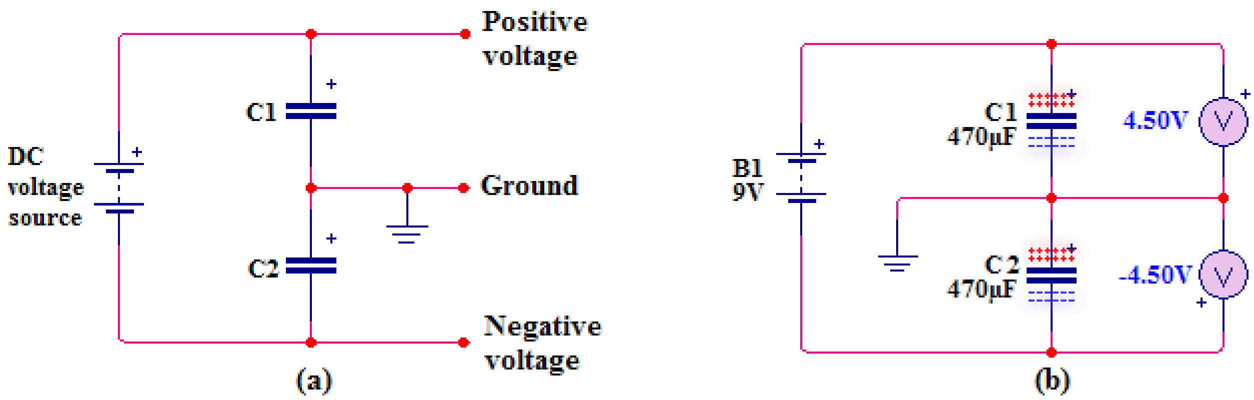


Figure 7-4: Designing dual polarities power supply using equal pair capacitors; (a) Circuit principle, (b) Circuit example.

7. 1. 2. Comparison of Op-Amp Parameters

There are several op-amp ICs on the market and the user's choice depend on the features and parameters of each. Table 7-2 provides a comparison of values of some of the parameters just described for several popular IC op-amps. Any values not listed were not given on the manufacturer's datasheet. A representative op-amp datasheet is given in appendix.

OP-AMP	INPUT OFFSET VOLTAGE (mV) (MAX)	INPUT BIAS CURRENT (nA)(MIN)	INPUT IMPEDANCE (MΩ) (MIN)	OPEN-LOOP GAIN (TYP)	SLEW RATE (V/µS) (TYP)	CMRR (db) (MIN)	COMMENT
LM741C	6	500	0.3	200,000	0.5	70	Industry standard
LM101A	7.5	250	1.5	160,000	—	80	General purpose
OP113E	0.075	600	—	2,400,000	1.2	100	Low noise, low drift
OP177A	0.01	1.5	26	12,000,000	0.3	130	Ultra precision
OP184E	0.065	350	—	240,000	2.4	60	Precision, rail-to-rail*
AD8009AR	5	150	—	—	5500	50	BW=700MHz, ultra fast, low distortion, current feedback
AD8041A	7	2000	0.16	56,000	160	74	BW=160MHz, rail-to-rail
AD8055A	5	1200	10	3500	1400	82	Very fast voltage feedback

*rail-to-rail means that the output voltage can go as high as the supply voltage

Table 7-2: Comparison of some commercial op-amp parameters

7. 1. 3. Using Negative Feedback

Negative feedback is one of the most useful concepts in electronics, particularly in op-amp applications. Negative feedback is the process whereby a portion of the output voltage of an amplifier is returned to the input with a phase angle that opposes (or subtracts from) input signal.

As you have seen, the inherent open-loop gain of a typical op-amp is very high (usually greater than 100,000). Therefore, an extremely small difference in the two input voltages drives the op-amp into its saturated output state. In fact, even the input offset voltage of the op-amp can drive it into saturation. For example, assume V_{in}= 1mV and A_{ol} = 100,000. Then,

$$V_{in} * A_{ol} = (1mV)(100{,}000) = 100V$$

Since the output level of an op-amp can never reach 100V, it is driven into saturation and the output is limited to its maximum output levels, as illustrated in figure 7-5 for both a positive and negative input voltage is 1mV.

The usefulness of an op-amp operated in this manner is severely restricted and is generally limited to comparator applications. With negative feedback, the overall closed-loop voltage gain (A_{cl}) can be reduced and controlled so that the op-amp can function as linear amplifier. In addition to providing a controlled, stable voltage gain, negative feedback also provides for control of the input and output impedance and amplifier bandwidth.

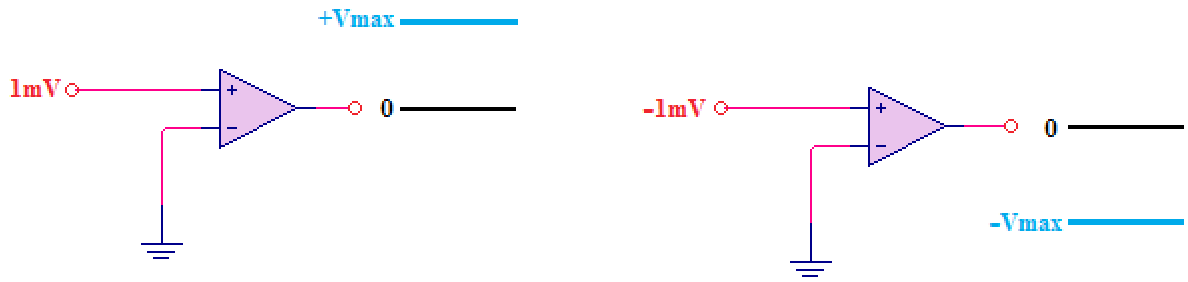


Figure 7-5: Without negative feedback, an extremely small difference in the two input voltages drives the op-amp to its limits and it becomes nonlinear

7. 1. 4. The 741 Operational Amplifier

The 741 is the godfather of all operational amplifiers (amplifiers on a chip). Although most up-to-date designs beat it for speed, low noise, etc., it still works well as a general purpose device. One of its advantages is that it is compensated (its frequency response is tailored) to ensure that under most circumstances it will not produce unwanted spurious oscillations. This means it is easy to use, but the down-side of this is the poor speed/gain performance compared to more modern op-amps.

Basic Op-Amp Circuits

Op-amps are used in such a wide variety of applications that it is impossible to cover all of them in one chapter or even in one book. Therefore, in this chapter we examine some of the more fundamental applications to illustrate the versatile of the op-amp and to give you a foundation in basic op-amp circuits.

7. 2. Op-Amp Comparator

Operational amplifiers are often used to compare the amplitude of one voltage with another. In this application, the op-amp is used in the open-loop configuration, with the input voltage on one input and a reference voltage on the other.

7. 2. 1. Zero-Level Detection

One application of the op-amp used as a *comparator* is to determine when an input voltage exceeds a certain level.

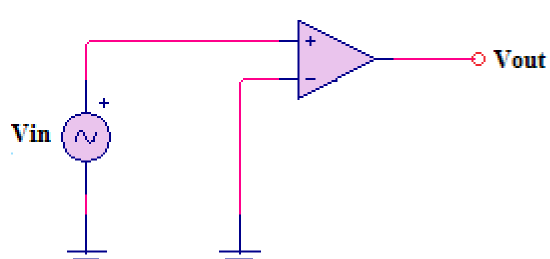


Figure 7-6: The op-amp as zero-level detector base circuit

The figure 7-6 shows a zero-level detector. Notice that the inverting (–) input is grounded and the input signal is applied to the noninverting (+) input. Because of the high open-loop voltage gain, a very small difference voltage between the two inputs drives the amplifier into saturation, causing the output voltage to go to its limit. For example, consider an op-amp having A_{ol} =100,000. A voltage difference of only 0.25mV between the inputs could produce an output voltage of (0.25mV)(100,000) = 25V if the op-amp were capable. However, since most op-amps have output voltage limitations of less than ±15V, the device would be driven into saturation. For many comparison applications, special op-amp comparators are selected.

Experiment 7-1: Op-amp comparator as zero-level detector

Design a zero-level detector circuit based on op-amp comparator. Supply the circuit with 6Vrms/50Hz sinusoidal and use the oscilloscope to measure and view the resulting output shape.

Part list

No	Items	Specifications	Quantity
1	Oscilloscope	Agile oscilloscope 54622D	1
2	6V/50Hz sinusoidal source	Agilent function generator 33120A	1
3	Op-amp IC	LM741	1
4	Breadboard	Prototyping board	1
5	Connecting wire	22-gauge solid wire	40cm
6	±9V to ±15V dc supply	9V battery	2
7	1KΩ Load resistor	1KΩ/1/2watt Load resistor	1

chap 7

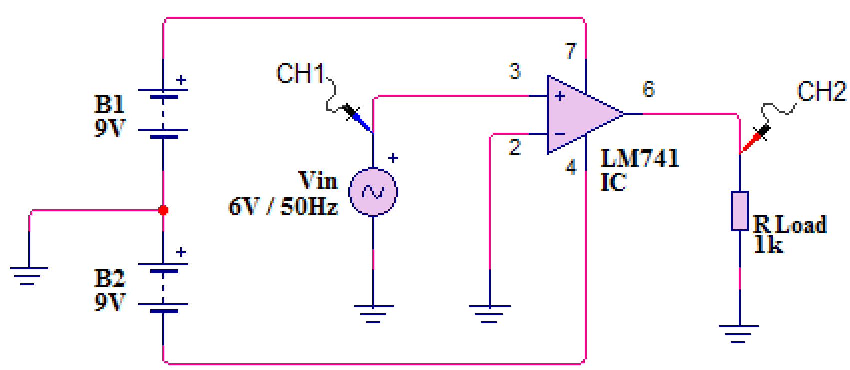


Figure 7-7: Zero-level detector op-amp

Working Principle

As shown on the circuit figure 7-7, a sinusoidal input voltage is applied to the noninverting input of the zero-level detector. When the sine wave is negative, the output is at its maximum negative level. When the sine wave crosses 0, the amplifier is driven to its opposite state and the output goes to its maximum positive level. As you can see on the resulting waveform, the zero-level detector can be used as a squaring circuit to produce a square wave from a sine wave.

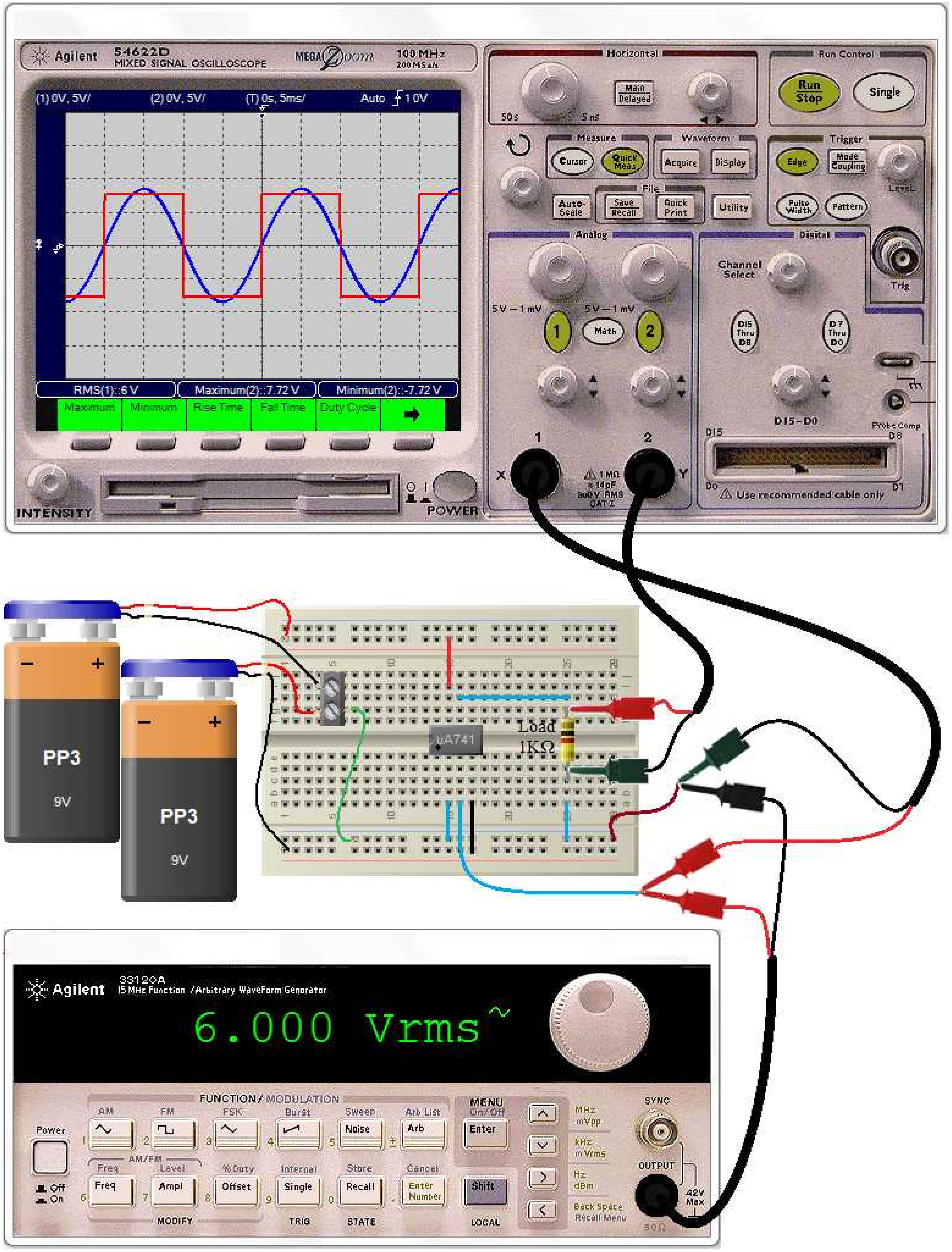


Figure 7-8: Zero-level detector op-amp circuit connection

Procedure

1. Wire the bus strips on your breadboard to provide positive power, negative power and ground buses. Whatever color scheme you have chosen for your wires, you should use the green binding post for ground, the black for –9 V, and the red for +9 V.
2. If you look closely at the package of dual inline op-amp, you will find a notch at one end or a dot in one corner. This tells us how to find Pin 1: the dot is located next to Pin 1 and the notch is located between Pins 1 and 8. Plug an op-amp into the breadboard so that it straddles the gap between the top and bottom sections of the socket strip. If you have wired the power buses as suggested above, Pin 1 should be to the left.
3. Do not try to unplug the op-amp with your thumb and forefinger. It is a good way to end up with the op-amp plugged into your fingertip. Use the IC extractor from your toolkit.
4. Set the function generator to produce a 6Vrms, 50 Hz sine wave. Connect the signal lead to pin 3 of LM741 IC op-amp and ground lead to common ground of the circuit.
5. Connect CH1 of the scope to and V_{in}, just to pin 3, and CH2 to V_{out}, at pin 6. Set both channels Volts/Div to 5 and 5mS Time/Div. Connect Pin 2 to ground. Make sure both channels of the scope are in DC coupling.
6. Connect Pin 4 (VCC–) to the negative power supply bus (–9V). Connect Pin 7 (VCC+) to the positive power supply bus (+9 V). For this experiment connect pin 4 to the positive of B1 and pin 7 to negative of B2. Take the ground at the interconnection of the two batteries.

7. 2. 2. Nonzero-Level Detection

The zero-level detection can be modified to detect voltages other than zero by connecting a fixed reference voltage to the inverting (–) input as shown in figure 7-9 part (a). A more practical arrangement is shown in figure 7-9 part (b) using voltage divider to set the reference voltage as follow:

$$V_{REF} = (+VCC)R_2/R_1+R_2$$

Where +VCC is the positive op-amp supply voltage. As long as the input voltage (V_{in}) is less than V_{REF}, the output voltage remains at the maximum negative level.

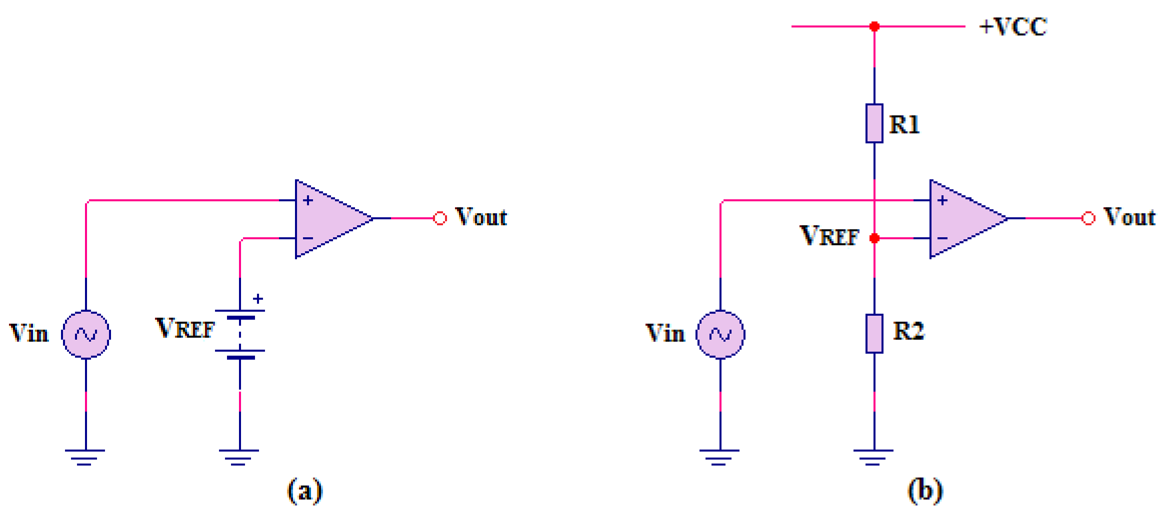


Figure 7-9: Nonzero-level detector circuit; (a) Battery reference, (b) Voltage-divider reference

When the input voltage exceeds the reference voltage, the output goes to its maximum positive state.

Experiment 7-2: Nonzero-level detector using op-amp comparator

Design a nonzero-level detector circuit based on op-amp comparator with two resistors voltage-divider network to make voltage reference. Use 1KΩ as load, 8.2KΩ and 2.7KΩ for voltage divider, and 5Vrms/50Hz sinusoidal.

Part list

No	Items	Specifications	Quantity
1	Oscilloscope	Agile oscilloscope 54622D	1
2	5V/50Hz ac source	Agilent function generator 33120A	1
3	Op-amp IC	LM741	1
4	Breadboard	Prototyping board	1
5	Connecting wire	22-gauge solid wire	40cm
6	±9V to ±15V dc supply	9V battery	2
7	2.7KΩ,1KΩ, 8.2KΩ resistors	2.7KΩ8.2KΩ ½ watt, and 1KΩ ¼ resistor	3

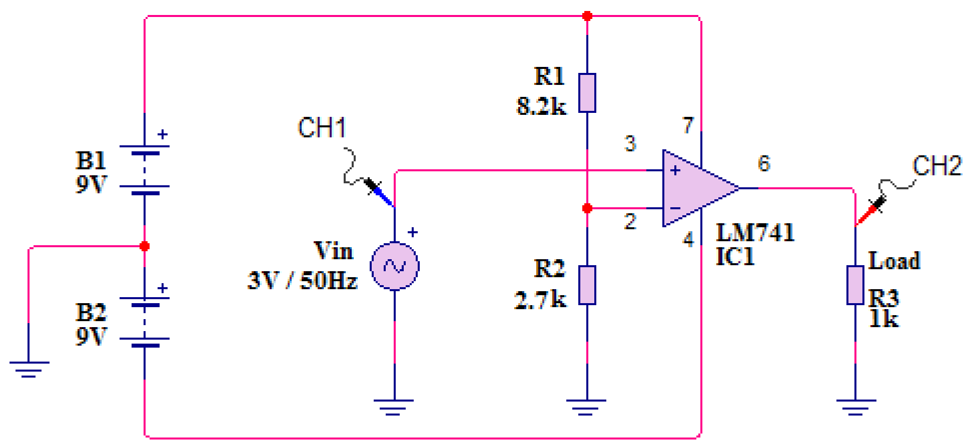


Figure 7-10: Nonzero-level detector circuit

The reference voltage is set by R_1 and R_2;

$$V_{REF} = (+VCC)R_2/R_1+R_2 = (9V)2.7K\Omega/8.2K\Omega+2.7K\Omega = 2.229V$$

This implies that each time the input voltage V_{in} exceeds +2.229V, the output voltage switches to its +VCC level, that's +9V, and each time the input signal goes below +2.229V, the output voltage switches back to its –VCC level, this is –9V.

Procedure

1. Wire the bus strips on your breadboard to provide positive power, negative power and ground buses.

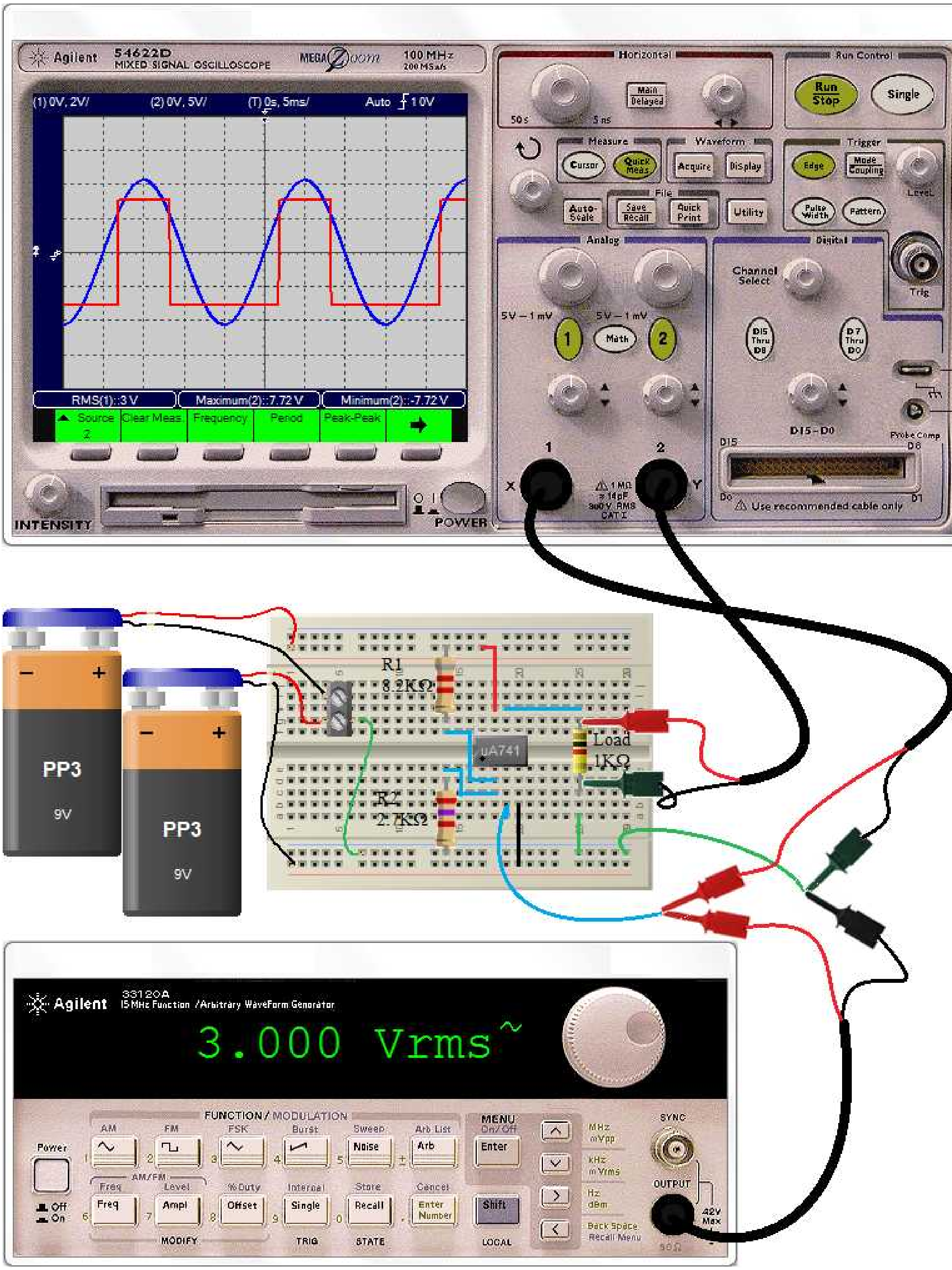


Figure 7-11: Nonzero-level detector circuit connection

2. If you look closely at the package of dual inline op-amp, you will find a notch at one end or a dot in one corner. This tells us how to find Pin 1: the dot is located next to Pin 1 and the notch is located between Pins 1 and 8. Plug an op-amp into the breadboard so that it straddles the gap between the top and bottom sections of the socket strip. Pin 1 should be to the left.
3. Do not try to unplug the op-amp with your thumb and forefinger. It is a good way to end up with the op-amp plugged into your fingertip. Use the IC extractor from your toolkit.
4. Set the function generator to produce a 3Vrms, 50 Hz sine wave. Connect the signal lead to pin 3 of LM741 IC op-amp and ground lead to common ground of the circuit.
5. Connect CH1 of the scope to V_{in}, just to pin 3, and CH2 to V_{out}, at pin 6. Set CH1 Volts/Div to 2V and CH2 Volts/Div to 5V and 5mS Time/Div. Make a series network resistor with 8.2KΩ (R_1) at +VCC,9V, and 2.7KΩ (R_2) at ground and connect the point of interconnection of the two resistors to Pin 2 of the op-amp. Make sure both channels of the scope are in DC coupling.
6. Connect Pin 4 (VCC–) to the negative power supply bus (–9V). Connect Pin 7 (VCC+) to the positive power supply bus (+9 V). For this experiment connect pin 4 to the positive of B1 and pin 7 to negative of B2. Take the ground at the interconnection of the two batteries.

7. 2. 3. Comparator With different on and off Voltages (Schmitt Trigger)

The Schmitt trigger is an inverting comparator with positive feedback. This circuit converts an irregular-shaped waveform to a square wave or pulse. The circuit is known as the Schmitt trigger or squaring circuit. The input voltage V_{in} triggers (change the state of) the output V_{out} every time it exceeds certain voltages levels called the upper threshold voltage V_{ut} and lower threshold voltage V_{lt}, as shown in figure 7-12.

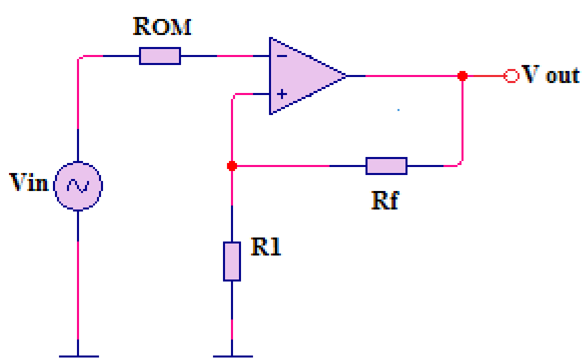


Figure 7-12: Schmitt trigger base circuit

These threshold voltages are obtained by using the voltage divider R_1 and R_f, where the voltage across R_1 is feedback to the non-inverting (+) input. The voltage across R_1 is a variable reference threshold voltage that depends on the value and polarity of the output voltage V_{out}. When V_{out} = $+V_{sat}$, the voltage across R_1 is called the upper threshold voltage, V_{ut}. The input voltage V_{in} must be slightly more positive than V_{ut} in order to cause the output V_{out} to switch from $+V_{sat}$ to $–V_{sat}$. As long as $V_{in}<V_{ut}$, V_{out} is at $+V_{sat}$. Using the voltage-divider rule:

$$V_{ut} = (+V_{sat})R_1/R_1 + R_f$$

On the other hand, when $V_{out} = –V_{sat}$,the voltage across R_1 is referred to as lower threshold voltage,

V_{lt}. V_{in} must be slightly more negative than V_{lt} in order to cause V_{out} to switch from $-V_{sat}$ to $+V_{sat}$. In other words, for V_{in} values greater than V_{lt}, V_{out} is at $-V_{sat}$.

$$V_{lt} = (-V_{sat})R_1/R_1 + R_f$$

Resistance $R_{OM} \approx R_1 || R_2$ is used to minimize the offset problems.

Experiment 7-3: Schmitt trigger (squaring circuit)

Design a circuit based on Schmitt trigger capable to output a square wave controlled from different sinusoidal input voltage levels.

Part list

No	Items	Specifications	Quantity
1	Oscilloscope	Agile oscilloscope 54622D	1
2	4V peak/50Hz ac source	Agilent function generator 33120A	1
3	Op-amp IC	LM741	1
4	Breadboard	Prototyping board	1
5	Connecting wire	22-gauge solid wire	40cm
6	±9V to ±15V dc supply	9V battery	2
7	4.7KΩ,1KΩ, 2x10KΩ resistors	4.7KΩ, 2x10KΩ, 1KΩ ½ watt resistors	3

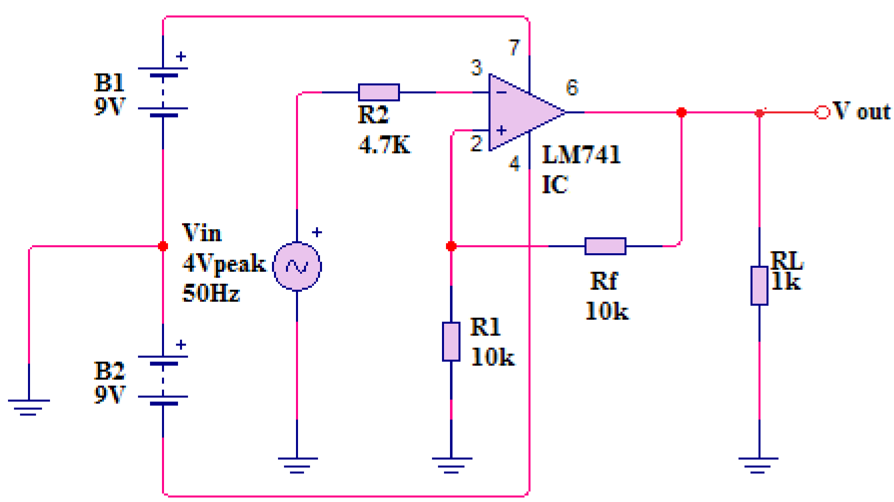


Figure 7-13: Schmitt trigger as squaring circuit

In this circuit R_f and R_1 form a voltage divider. For this circuit $R_f = R_1$ to make this explanation easier to follow.

If the Op-Amp output voltage (V_{out}) is high (say 9 Volts). The non inverting input will have 4.5V volts on it. The inverting input must rise above 4.5 Volts to make the op amp output go low.

The Op Amp output is now low (say –9 Volts).

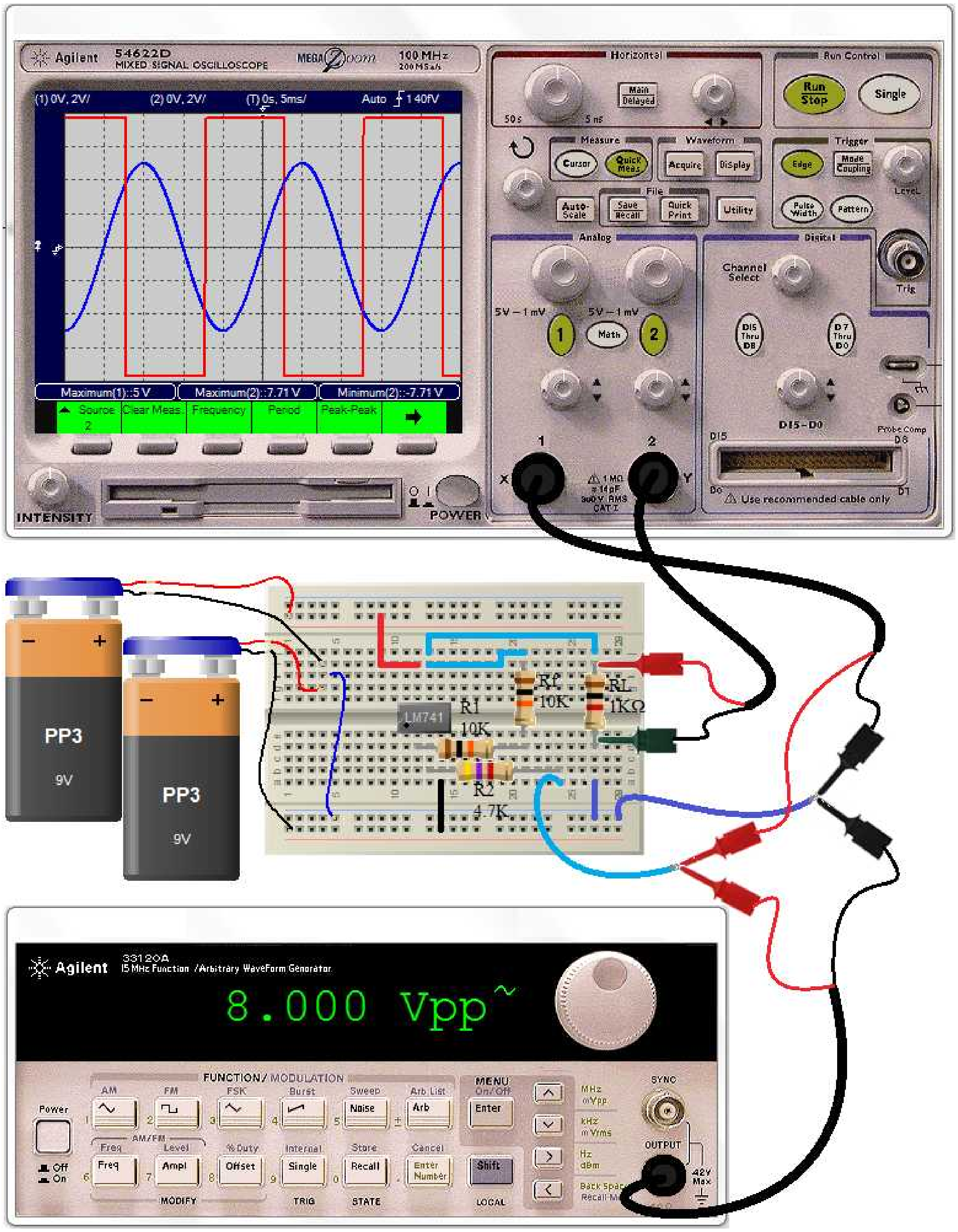


Figure 7-14: Schmitt trigger as squaring circuit connection

chap 7

The non inverting input will have –4.5 volts on it. The inverting input must fall below –4.5 Volts to make the op-amp output voltage (V_{out}) to go high again. 4.5V and –4.5V are V_{ut} and V_{lt} threshold voltages and 9V and –9V are $+V_{sat}$ and $-V_{sat}$.

Thus, if the threshold voltages V_{ut} and V_{lt} are made larger than the input noise voltages, the positive feedback will eliminate the false output transitions. Also the positive feedback, because of its regenerative action, will make V_{out} to switch faster between $+V_{sat}$ and $-V_{sat}$.

By altering R_f and R_1, the threshold voltages can be adjusted.

Procedure

1. Wire the bus strips on your breadboard to provide positive power, negative power and ground buses.
2. Locate the pins of IC. The dot is located next to pin 1 and the notch is located between pins 1 and 8. Plug an op-amp into the breadboard so that it straddles the gap between the top and bottom sections of the socket strip. Pin 1 should be to the left.
3. Set the function generator to output 8V peak-to-peak/50 Hz sine wave. Connect the signal lead to pin 3, through 4.7KΩ R_2, of LM741 IC op-amp and ground lead to common ground of the circuit.
4. Connect CH1 of the oscilloscope to V_{in}, just on the terminal of the function generator, and CH2 to V_{out} at pin 6, on the terminal of the RL. Set both CH1 and CH2 Volts/Div to 2V and Time/Div to 5mS. Make sure both channels of the oscilloscope are in DC coupling.
5. Connect Pin 4 (VCC–) to the negative power supply bus (–9V). Connect Pin 7 (VCC+) to the positive power supply bus (+9 V). For this experiment connect pin 4 to the positive of B1 and pin 7 to negative of B2. Take the ground at the interconnection of the two batteries.
6. In practical, saturation voltages are not exactly equal to ±VCC, in most of the case are 1.5V to 3V lower to VCC. This also makes a change on upper and lower threshold voltages. For this experiment, $+V_{sat}$ is equal to 7.71V and $-V_{sat}$ is equal to –7.71V. The point of intersection between the input voltage (sine wave) and square wave determines the threshold voltage. This is at 1.9 divisions.

   Thus, the upper threshold voltage (V_{ut}) is:

   $$1.9\text{Division}*2\text{Volt/Division} = 3.8\text{V}.$$

   The lower threshold voltage (V_{lt}) is:

   $$-(1.9\text{Division}*2\text{Volt/Division}) = -3.8\text{V}.$$

chap 7

7. 3. Op-Amp Comparator as Switch

The gain of the op-amp without feedback (so called "open loop" mode) is too high to be useful as an amplifier. But this high gain provides a very sharp threshold if we use the op-amp as a switch or comparator (so called because it compares one voltage to another and gives an output signifying which is greater). In operation the op-amp goes into positive or negative saturation dependent upon the input voltages. As the gain of the operational amplifier will generally exceed 100,000 the output will run into saturation when the inputs are only fractions of a millivolt apart.

A typical comparator circuit will have one of the inputs held at a given voltage. This may often be a potential divider from a supply or reference source. The other input is taken to the point to be sensed. Let's consider the circuit of figure 7-15 where the op-amp IC is used as comparator. A fixed voltage (V_Y) is connected to the inverting input at pin 2 of LM324 IC and is determined by voltage-divider network of resistor R_1 and R_2. Thus, V_Y is calculated as:

$V_Y = V_{B1}*(R_2/R_1+R_2) = 9V*(5.6/6.8+5.6) = 4.0645V$ [equation 7-1]

A variable voltage (V_X) is connected to noninverting input at pin 3 of IC and the amount of this voltage depend on the resistance of VR_3. The voltage V_X is found by applying voltage divider:

- Expect VR_3 is set at 35% of its maximum value; that is 10KΩ*35/100=3.5KΩ, thus

$V_X = V_{B1}*(VR_3/R_4+VR_3) = 9V*(3.5/4.7+3.5) = 3.841V$ [equation 7-2]

At this case the output voltage V_z is low ≈ 0V (practically 0.00508V in considering leakage) because the voltage on noninverting is less than the voltage on inverting ($V_Z<V_Y$) so the LED D_1 is "OFF".

- Let's increase variable resistor VR_3 to 45% of its maximum value; 10KΩ*45/10 = 4.5KΩ, so

$V_X = V_{B1}*(VR_3/R_4+VR_3) = 9V*(4.5/4.7+4.5) = 4.402V$

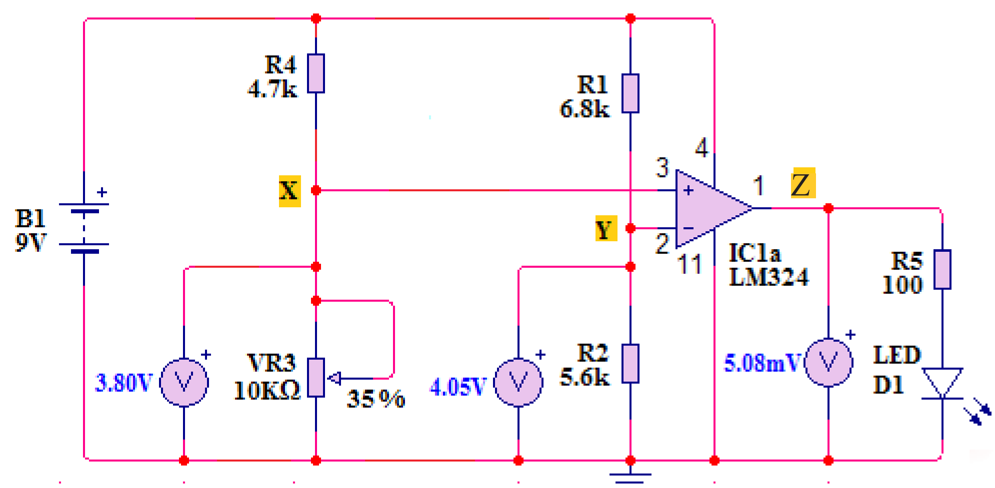


Figure 7-15: Op-amp comparator circuit principle

At this case the output voltage V_z is high ≈ V_{B1} (practically 7.8V) because the voltage on noninverting is greater than the voltage on inverting ($V_Z>V_Y$) so the LED D_1 is "ON".
Remember that resistor tolerance affects the expected voltage value, theoretical value differ in many case from practical value.
The higher the resistance of variable resistor VR_3, the higher the voltage is on noninverting terminal at pin 3.
Depending on the desired response, the variable resistor can be interchanged with R_4 so that the output goes high as the resistance of VR_3 goes low or be connected to inverting terminal to reverse the previous conditions. In many applications, the manual variable resistor is replaced by sensor.

Note: In most of the practical cases, two LEDs are needed because, when the op-amp pin 1 voltage is low, there may still be as much as two volts present. This could be enough to light a single LED. With two LEDs, four volts are needed. The op-amp pin 1 voltage will be well below 4V so the LEDs will be off.

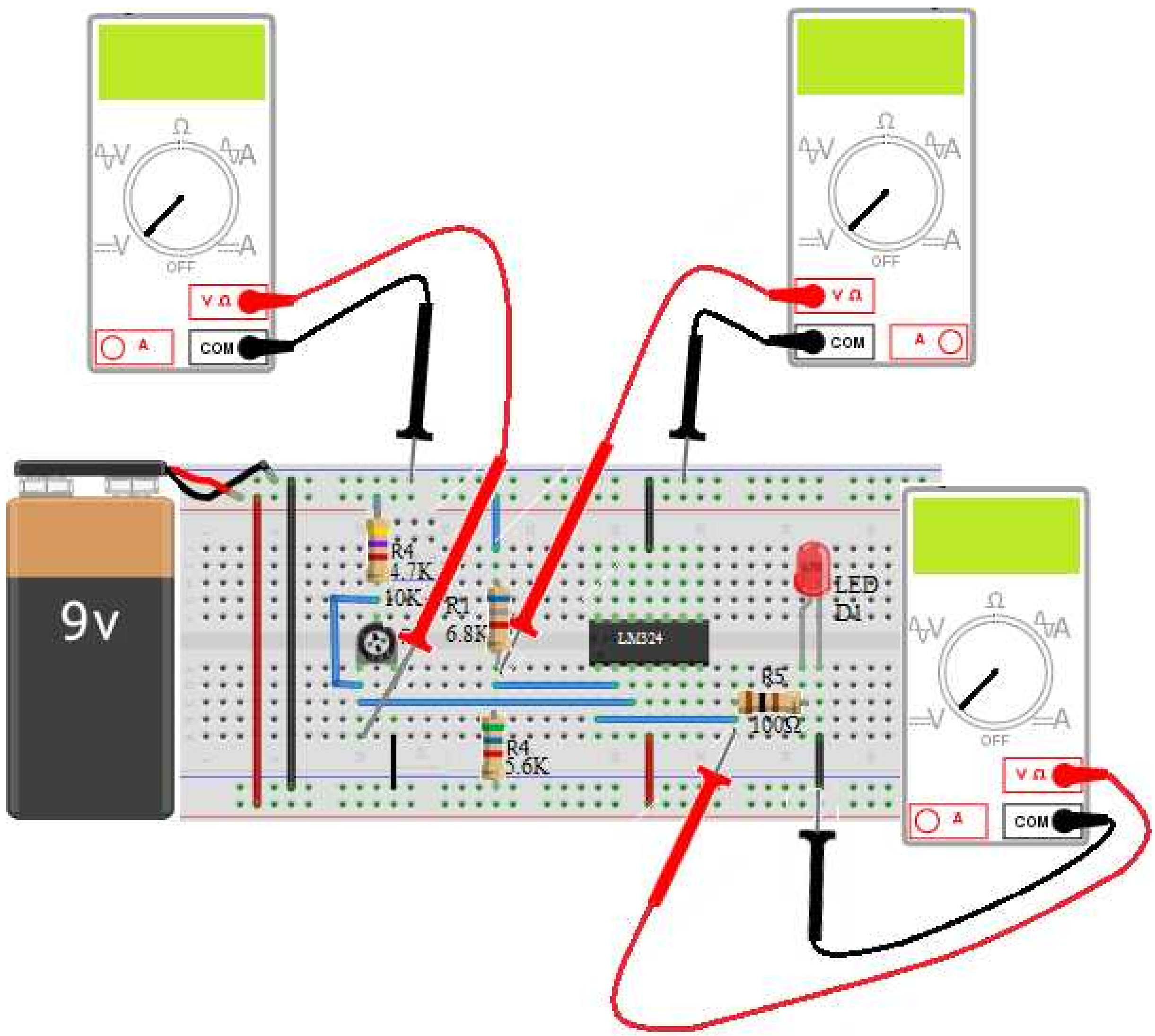


Figure 7-16: Op-amp comparator circuit principle connection

There are a number of points to remember when using comparator circuits. As there is no feedback the two inputs to the circuit will be at different voltages. Accordingly it is necessary to ensure that the maximum differential input is not exceeded. Again as a result of the lack of feedback the load will change. Particularly as the circuit changes there will be a small increase in the input current. For most circuits this will not be a problem, but if the source impedance is high it may lead to a few unusual responses.

The main problem with this circuit is that new changeover point, even small amounts of noise will cause the output to switch back and forth. Thus near the changeover point there may be several transitions at the output and this may give rise to problems elsewhere in the overall circuit. The solution to this is to use a Schmitt Trigger.

Most of comparator circuits find many uses as detectors. They are often used to sense voltages where a reference voltage is kept on one input, and a voltage that is being detected on another. While the detected voltage is above the reference, the output of the comparator will be in one state. If the detected voltage falls below the reference then it will change the state of the comparator, and this could be used to ensign the condition. Below list some practical projects examples of many for which comparators can be used.

7. 4. Light Activated Switch

A Light activated switch generates an output signal indicating the intensity of light falling on the light sensor. The light sensor may respond to the radiant energy that exists in a very narrow range of frequencies basically called "light", and which ranges in frequency from "Infra-red" to "Visible" up to "Ultraviolet" light spectrum. In light activated switch, an op-amp as comparator generates an output signal based on the condition state sensed by a light sensor. A light sensor has to be configured with other electronics components to control its sensibility. The output signal of the comparator is used to activate an actuator for a specific task. Flame or smoke detectors, thief alarms, controls for street lighting, clock radios and alarm devices are examples of light activated switches.

Experiment 7-4: Light-level alarm

Part list

No	Items	Quantity
1	Light dependent resistor (LDR) ORP12	1
2	9 VDC supply (battery)	1
3	Op-amp IC (LM324)	1
4	Breadboard	1
5	Connecting wire, 22-gauge ($0.33mm^2$) solid wire	40cm
6	Buzzer	1
7	3.9KΩ, 3.3KΩ, 1KΩ, 100Ω, 270Ω resistors	5
8	Transistor 2N2222	1
9	Potentiometer 100KΩ	1

Here the circuit figure7-17 acts as a light-activated switch which turns the buzzer either "ON" or "OFF" as the light level detected by the LDR resistor exceeds or falls below a pre-set value at V_2 determined by the position of potentiometer VR_3. A fixed voltage reference V_1 is applied to the inverting input terminal via the R_1 and R_2 voltage divider network and the variable voltage (proportional to the light level) applied to the non-inverting input terminal V_2. When the LDR receives the light its resistance decreases as a result, the voltage on non-inverting input (V_2) increases. When there is obstacle of the light to fall on LDR, its resistance will be increased making the voltage V_2 to decrease. The values of V_1 and V_2 are calculated by using equation 7-1 and equation 7-2. For this circuit example, supply voltage is set to B_1 =9V, resistance R_1 =3.9KΩ and R_2=3.3KΩ, therefore a reference (V_1) voltage on inverting terminal is:

$$V_1= V_{B1}*(R_2/R_1+R_2) = 9V*(3.3K\Omega/3.3K\Omega+3.9K\Omega) = 4.125V$$

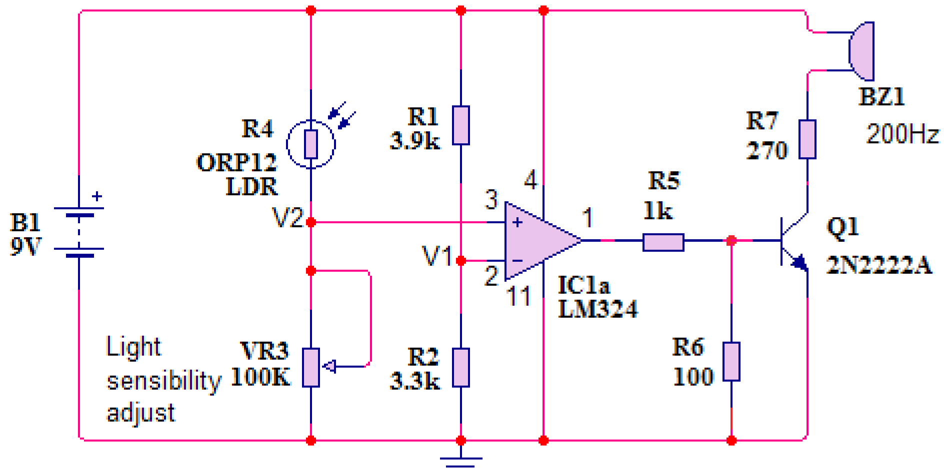


Figure 7-17: Light-level detector circuit (non-inverting comparator)

Thus, when voltage applied on non-inverting input is greater than 4.125V (voltage applied on inverting input) the transistor Q_1 goes in saturation and the buzzer turns ON telling the person that it is the time to get up as the sun shines for example. This is when the LDR is illuminated but in case there is no light intensity falling on LDR, the transistor Q_1 will go in cut-off turning the buzzer OFF as the voltage on non-inverting terminal of op-amp is less than the reference voltage (4.125V). R_5 and R_6 make a voltage divider and determine the voltage on the base of transistor Q_1. R_7 limits current through the buzzer BZ_1. By interchanging the positions of potentiometer and the LDR, the circuit can be used to detect either light or dark.

chap 7

Procedure

1. Connect the circuit as illustrated in figure 7-18. Wire the positive terminal of the battery to power bus of breadboard and negative terminal of the battery to ground bus of breadboard.
2. If you look closely at the package of dual inline op-amp, you will find a notch at one end or a dot in one corner. This tells us how to find pin 1: the dot is located next to pin 1 and the notch is located between pins 1 and 14. Plug an op-amp into the breadboard so that it straddles the gap between the top and bottom sections of the socket strip. Pin 1 should be to the left.
3. Do not try to unplug the op-amp with your thumb and forefinger. It is a good way to end up with the op-amp plugged into your fingertip. Use the IC extractor from your toolkit.
4. Connect pin 11 (VCC–) to the negative terminal of power supply (ground bus of breadboard). Connect pin 4 (VCC+) to the positive terminal of power supply (power bus of breadboard). For this experiment, use dc voltage source of 5V to 9V for the safe of the buzzer.
5. Let the light intensity from the torch fall on the LDR and adjust the potentiometer for light sensitivity until you get a fine response. As expected result, when the light falls on LDR the buzzer will sound just as an alarm, and will stop buzzing on the absence of the light.

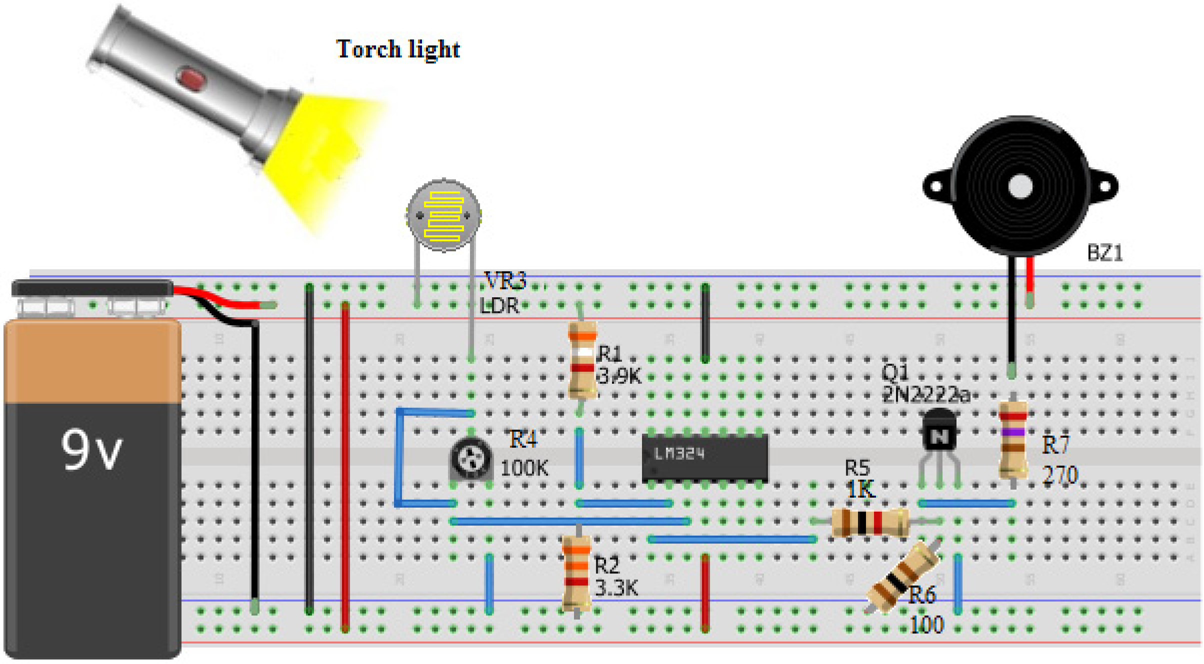


Figure 7-18: Light-level detector circuit connection (non-inverting comparator)

Note that by interchanging the position of VR_3 with that R_4 (light sensor) the circuit becomes dark activated. To maintain the same working for light-level detector we can interchange the input to be sensed with reference input and also the position of the light sensor as shown in figure 7-19.

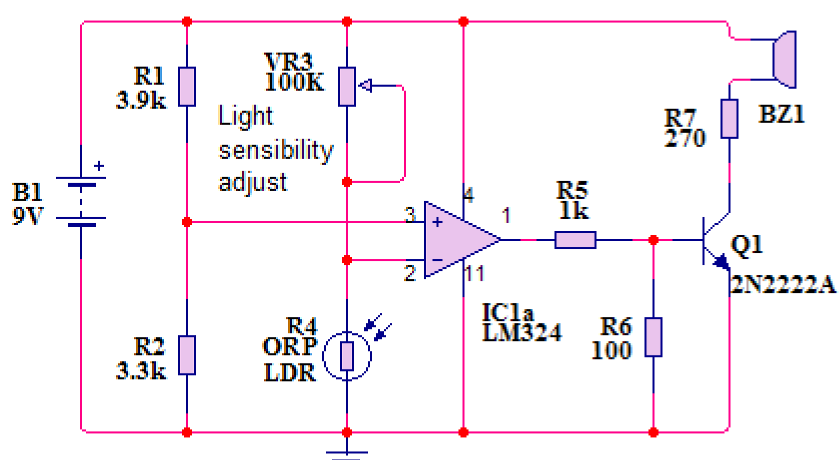


Figure 7-19: Light-level detector circuit (inverting comparator)

The circuit of figure 7-19 acts as a light-activated switch which turns the sound-buzzer either "ON" or "OFF" depending on the light level sensed by R_4 (LDR). This circuit is using the first op-amp of LM324 IC which contains four 741 op-amps. When the light falls on LDR its resistance decreases and the voltage applied on inverting input also decreases. When the light is obstructed to fall on LDR its resistance increases and as a result the voltage on inverting input is also increased. The light sensitivity is adjusted on potentiometer VR_3. A fixed voltage reference is applied to the non-inverting input terminal via the R_1 and R_2 voltage divider network and the variable voltage (proportional to the light level) applied to the inverting input terminal. Voltage on inverting and non-inverting terminal is calculated by using equation 7-1 and equation 7-2. For B_1 =9V, R_1= 3.9KΩ and R_2= 3.3KΩ, a reference voltage is:

$$V_{REFERENCE} = V_{B1}*(R_2/R_1+R_2) = 9V*(3.3K\Omega/3.3K\Omega + 3.9K\Omega) = 4.125V$$

When voltage applied on inverting input is less than 4.125V (light falling on LDR), a reference voltage applied on non-inverting input, the transistor Q_1 goes in saturation and the buzzer turns ON, telling the person that it is the time to get up as the sun shines for example, otherwise the transistor goes in cutoff and the buzzer is OFF. R_5 and R_6 make a voltage divider and determine the voltage on the base of transistor Q_1. R_7 limits the current through the buzzer BZ_1.

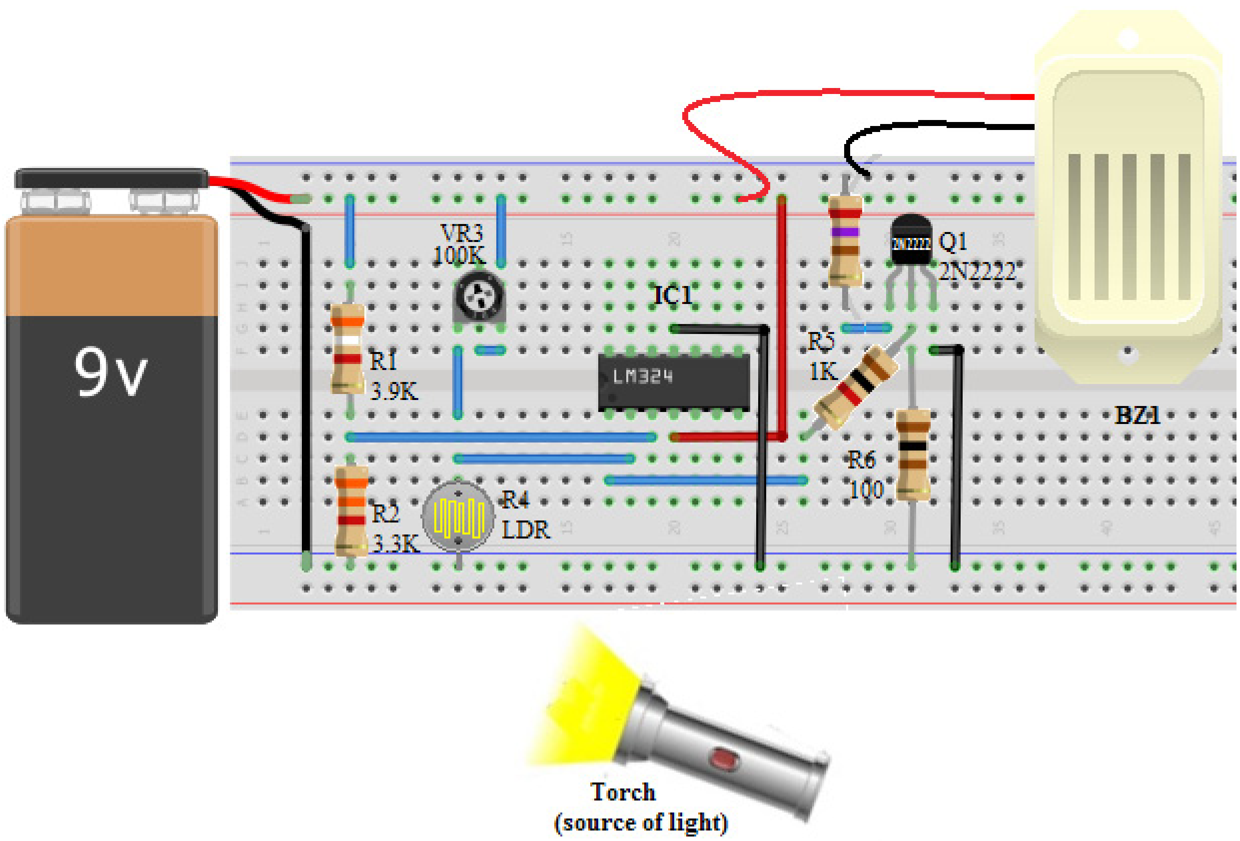


Figure 7-20: Light-level detector circuit connection (inverting comparator)

Experiment 7-5: Dark (smoke) detector

Part list

No	Items	Quantity
1	Light dependent resistor (LDR) ORP12	1
2	9 VDC supply (battery)	1
3	Op-amp IC (LM324)	1
4	Breadboard	1
5	Connecting wire, 22-gauge (0.33mm²)x50cm and 1.5 mm²x100cm solid wire	150cm

6	220V/50Hz Bulb	1
7	6.8KΩ, 270Ω, 150Ω, 470Ω resistors	4
8	Transistor 2N2222	1
9	Potentiometer 100KΩ, potentiometer 10KΩ	2
10	9Vdc/220Vac, 5A SPDT relay	1

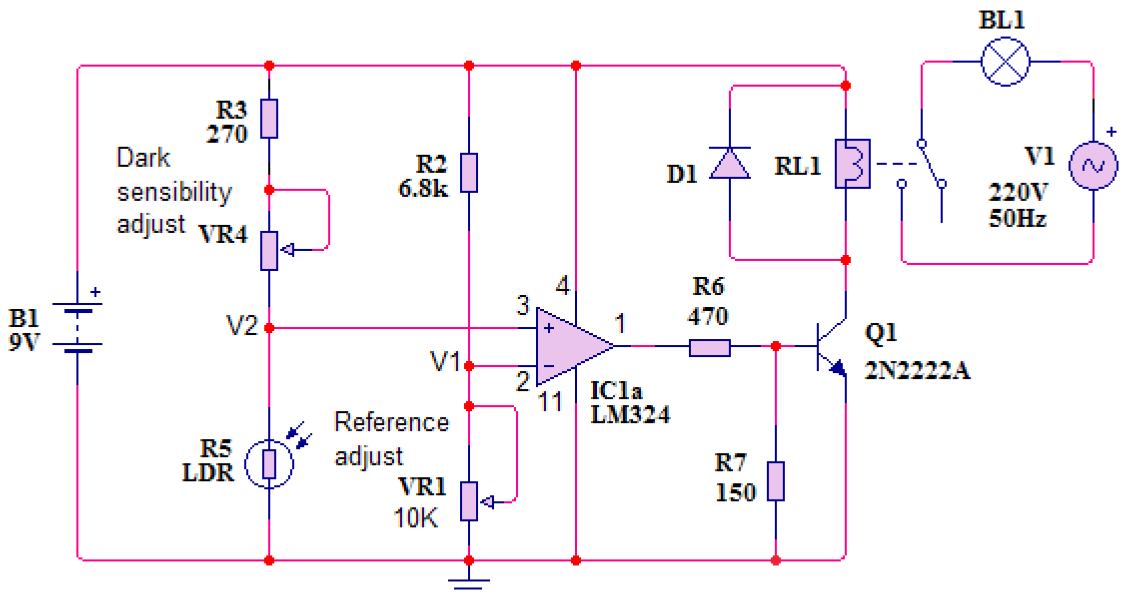


Figure 7-21: Dark (smoke) detector circuit (non-inverting op-amp)

In this dark sensing circuit, figure 7-21, the light dependent resistor LDR (R_5) is used to sense dark level. The potentiometer V_{R4} is used to adjust darkness sensitivity of LDR by forming an adjustable arm resistance bridge network. Resistor R_3 fixes adjustable limits in case the potentiometer could be set to 0%. Also resistor R_2 and potentiometer VR_1 form other arm which determines a reference voltage, adjustable with a help of potentiometer VR_1. Both sides of arms form potential divider networks across the supply voltage whose outputs V_1 and V_2 are connected to the inverting and non-inverting voltage inputs respectively of the operational amplifier. The voltage V_1 and V_2 are calculated by using equation 7-1 and equation 7-2. The operational amplifier is configured as a voltage comparator whose output voltage is determined by input signal V_1 and a reference voltage V_2. The resistor combination R_2 and VR_1 form a voltage reference at input V_1, set by the ratio of the two resistors. The LDR and VR_1 combination provides a variable voltage input V_2 proportional to the light level being detected by the photo-resistor R_5. So if voltage V_2 is greater than reference voltage V_1 the output of comparator is high driving transistor Q_1 in saturation. Assume LDR at 600lux has resistance of 200KΩ with potentiometer VR_4 set to 50% has 50KΩ and potentiometer VR_1 set to 50% has 5KΩ, therefore V_1 and V_2 are calculated as:

Reference voltage $V_1 = V_{B1}*(VR_2/VR_2+R_1)$

$= 9V*(5K\Omega/5K\Omega + 6.8K\Omega) = 3.81356V$

Signal voltage $V_2 = V_{B1}*(R_5/R_5+R_3 + R_3)$

$= 9V*(200K\Omega/200K\Omega + 50K\Omega + 0.27K\Omega) = 7.1922V$

chap 7

Thus, at this time V_2 is greater than V_1 (7.1922V>3.81356V) and the output of comparator is high (almost equal to V_{B1}, 9V) making transistor Q_1 to be in saturation. When transistor is in its saturation region the relay RL_1 is energized and close the contact.

The output from the operational amplifier is used to control a relay, which is protected by a fly-wheel diode, D_1. When the darkness is sensed by the LDR, V_2 becomes high enough, once exceeds the reference voltage set at V_1 the output from the op-amp changes state activating the relay and switching the connected load lamp BL_1.

Likewise as the darkness decreases the output will switch back turning “OFF” the relay. The operation of this type of dark sensor circuit can also be reversed to switch the relay “ON” when the light level exceeds the reference voltage level and vice versa by reversing the positions of the light sensor LDR and the potentiometer VR_4 as shown in prior circuits. The potentiometer can be used to “pre-set” the switching point of the differential amplifier to any particular dark level making it ideal as a simple dark sensor project circuit.

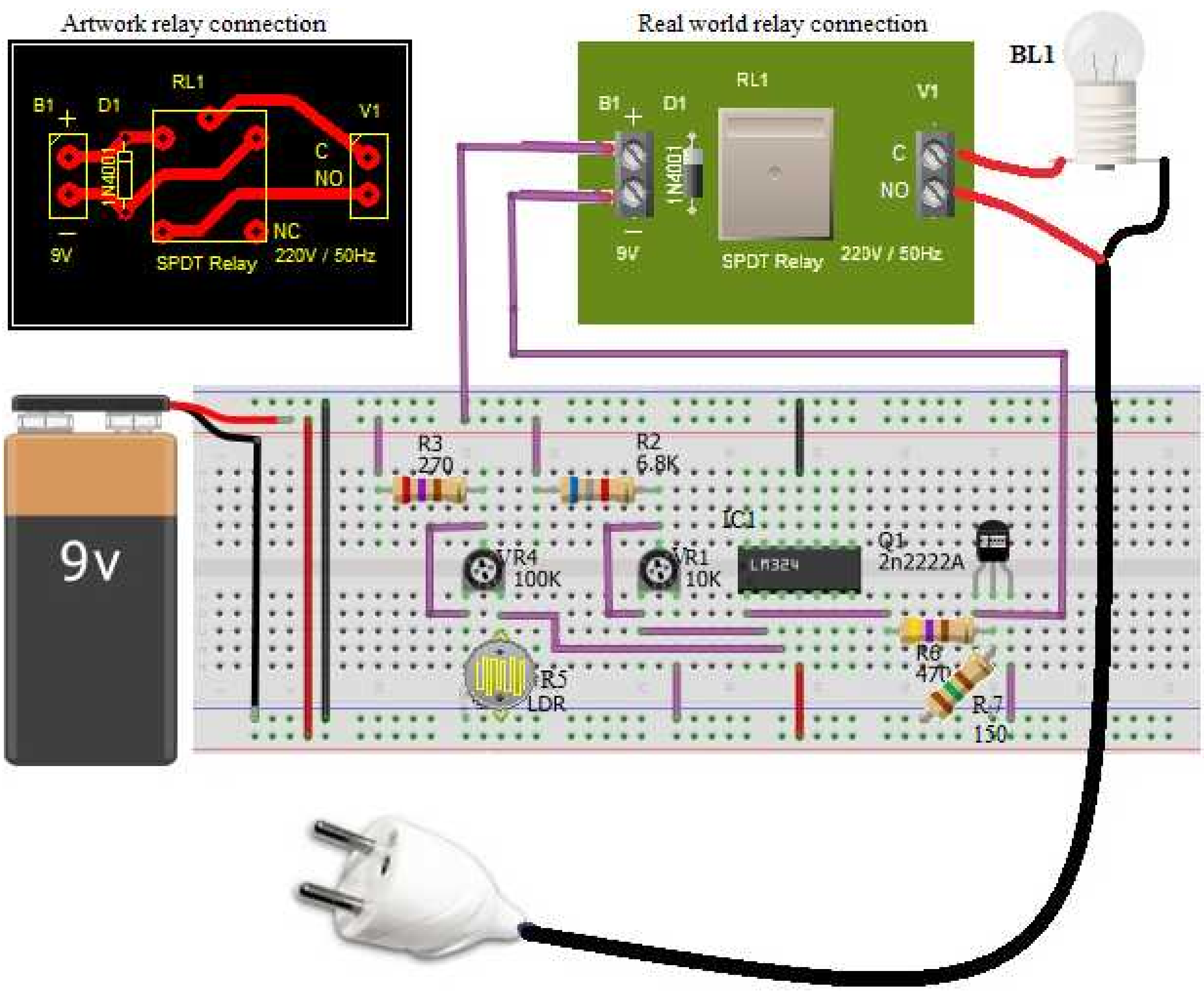


Figure 7-22: Dark (smoke) detector circuit connection (non-inverting op-amp)

Procedure

1. Connect the circuit as illustrated in figure7-22. Wire the positive terminal of the battery to power bus of breadboard and negative terminal of the battery to ground bus of breadboard.
2. Locate the pin 1 of IC. If you look closely at the package of dual inline op-amp, you will find a notch at one end or a dot in one corner. This tells us how to find pin 1: the dot is located next to pin 1 and the notch is located between pins 1 and 14. Plug an op-amp into the breadboard so that it straddles the gap between the top and bottom sections of the socket strip. Pin 1 should be to the left.
3. Connect pin 11 (VCC–) to the negative terminal of power supply (ground bus of breadboard). Connect pin 4 (VCC+) to the positive terminal of power supply (power bus of breadboard).
4. Adjust the potentiometer VR_4 for dark sensitivity of LDR. You can change reference voltage by just adjusting on the potentiometer VR_1. Get in the way of LDR to obstruct the light so dark access to LDR. In some of the case the output of op-amp is not exactly 0V at off state, if this occurs try to modify R_7 resistor and use a low value. As expected result, when there is darkness on LDR the bulb BL_1 will be on, and will be off on the presence of the light.

Note that it is possible to modify the circuit arrangement of figure 7-21 and working principle of circuit remain unchanged. This is by interchanging the position of LDR and potentiometer VR_4 and by interchanging input terminals of the op-amp by keeping the same reference as shown in figure 7-23.

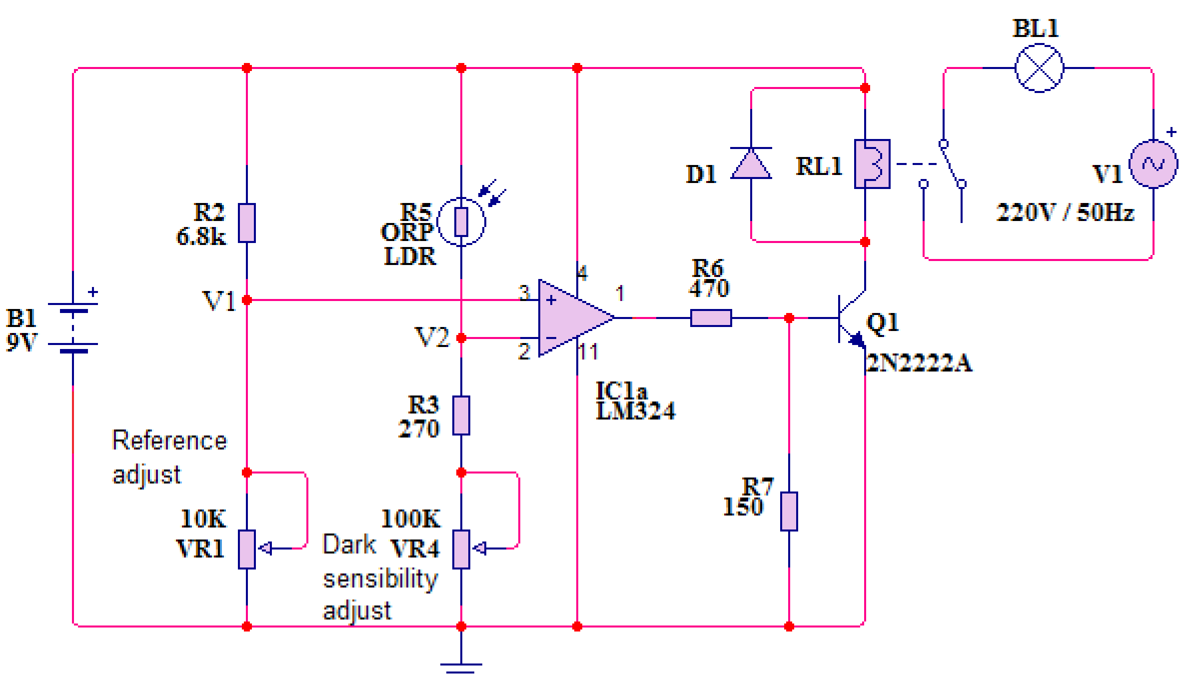


Figure 7-23: Dark (smoke) detector circuit (inverting op-amp)

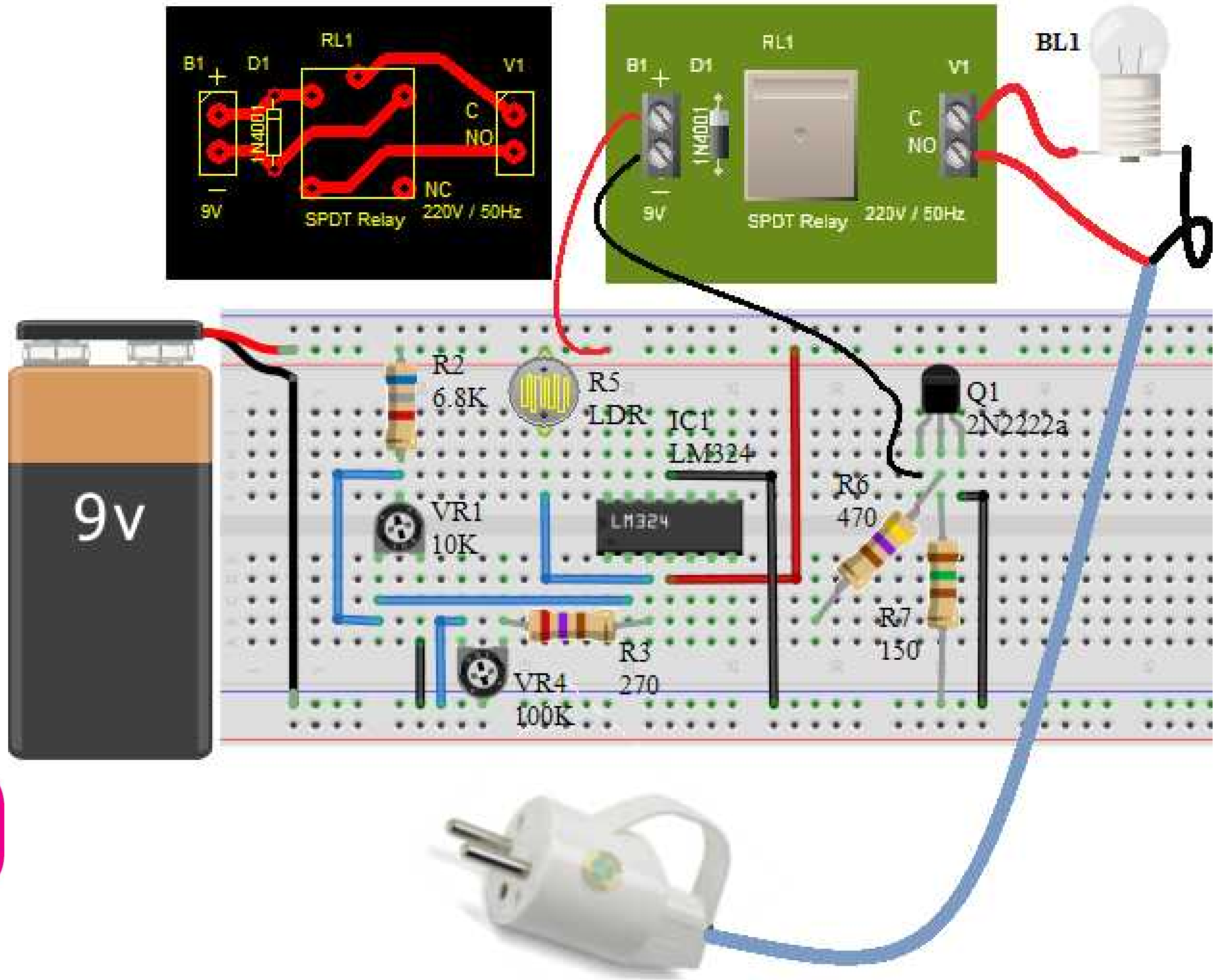


Figure 7-24: Dark (smoke) detector circuit connection (inverting op-amp)

Uses of Positive Feedback

A comparator with positive feedback, referred to as Schmitt trigger, is used to solve problem that happen when using comparator as switch. Suppose you have built a circuit to turn on some lights when it gets dark. But when the lights turn on, it gets brighter so the lights go off again. But then it is too dark so they come on again, flashing on and off rapidly. The Schmitt trigger solves this problem by having different threshold voltages for turning on and off.

When it gets dark, the lights come on. This makes it lighter but not lights enough to turn the lights off again. Later when it gets light again, the lights go off. This makes it darker but not dark enough to turn the lights on again.

This behaviour is called hysteresis. The Schmitt trigger uses positive feedback to achieve this effect.

In this circuit figure 7-24, if the output is low, R_f is in parallel with R_2. The combined resistance can be calculated as: $R_{f2} = R_f * R_2 / R_f + R_2$.

The lower threshold voltage V_{lt} is found using the voltage divider formula with R_1 and R_2 in parallel with R_f. $V_{lt} = VCC * R_{f2} / R_{f2} + R_1$.

If the output is high, R_f is in parallel with R_1. The combined resistance can be calculated as:

$$R_{f1}=R_f*R_1/\ R_f+R_1.$$

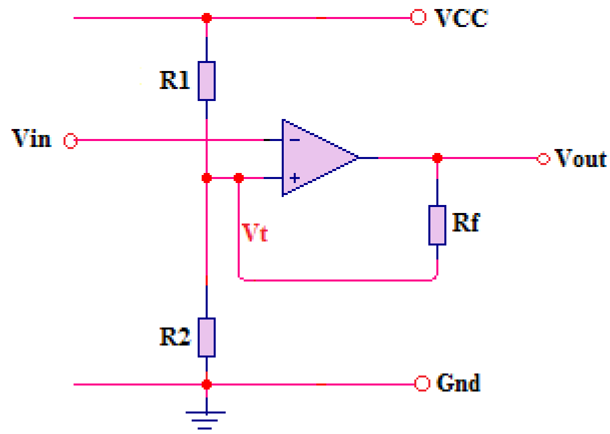


Figure 7-25: Comparator converted into a Schmitt trigger by adding one positive feedback resistor to give different on and off switching levels

The upper threshold voltage V_{ut} is found using the voltage divider formula with R_2 and R_1 in parallel with R_f. V_{ut} = VCC* $R_{f1}/\ R_{f1}+R_2$.
As it is an inverting Schmitt trigger, a low input gives a high output and vice versa.

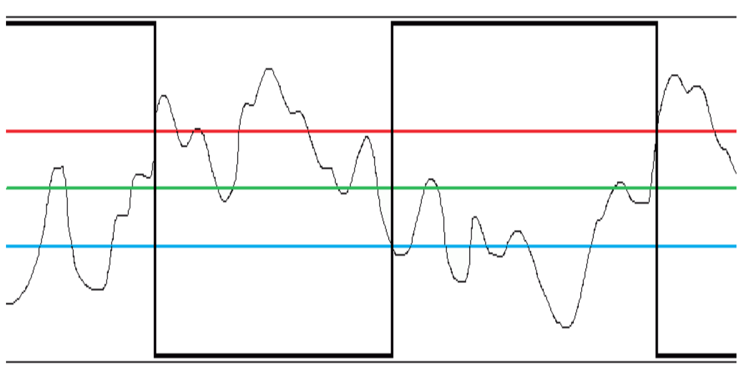


Figure 7-26: Effect of Schmitt trigger to clean up switching noise

Here, in figure 7-26, you can see a very noisy input that would produce errors in different places with a simple comparator using the green line reference voltage. The Schmitt trigger has two reference voltages. The input must go above the red line for a low output and below the blue line for a high output. This removes all noise unless the noise signal is so large that it exceeds the difference between the reference voltages. Although the amplitude noise is removed, there is still some timing noise called Jitter. The pulses might switch slightly too soon or too late. This can be removed with other digital techniques beyond a level scope.

Experiment 7-6: Light level sensor and dark detector with Schmitt trigger

Design a light level sensor (light and dark switch) based on Schmitt trigger so that the circuit switches on for a specific light level without light interference.

Part list

No	Items	Quantity
1	Light dependent resistor (LDR) ORP12	1
2	9 VDC supply (battery)	1
3	Op-amp IC (LM741)	1
4	Breadboard	1
5	Connecting wire, 22-gauge (0.33mm^2) x50cm and 1.5 mm^{2}x150cm solid wire	200cm
6	Buzzer	1
7	5.6KΩ, 4.7KΩ, 2.7KΩ, 680Ω, 120Ω ; ½ watt resistors	5
8	Transistor BSR13	1
9	Potentiometer 100KΩ, potentiometer 10KΩ	2
10	10Vdc/ 220V, 5A SPDT relay	1
11	220V/50Hz Bulb lamp	1

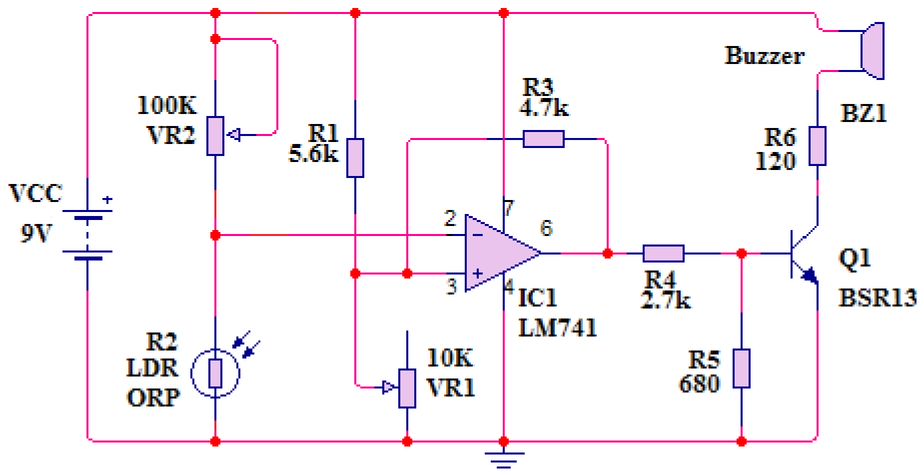


Figure 7-27: Light-level sensing with Schmitt trigger

In this light-level sensing circuit, figure 7-27, the light dependent resistor LDR (R_2) and the potentiometer VR_2 form one voltage divider across the supply. The potential at inverting terminal of op-amp depends on the light level. As it gets lighter, the potential decreases. Resistor R_1 and potentiometer VR_1 form other potential divider across the supply VCC. Potentiometer VR_1 makes the switch over light level adjustable. Potentiometer VR_2 determines light sensitivity level of LDR. As the op-amp is configured as inverting the low input voltage at inverting, compared to the non-inverting input determined by resistor R_1 and VR_1 all with effect of feedback resistor R_3, will make a high output voltage which is required to drive the transistor Q_1 into saturation. As in practice the output of op-amp is not exactly zero, resistor R_4 and R_5 are used to determine the voltage at the base and R_5 is kept low to make sure that a low output voltage of op-amp does not trigger the base of transistor. Resistor R_6 limits the current across the buzzer.

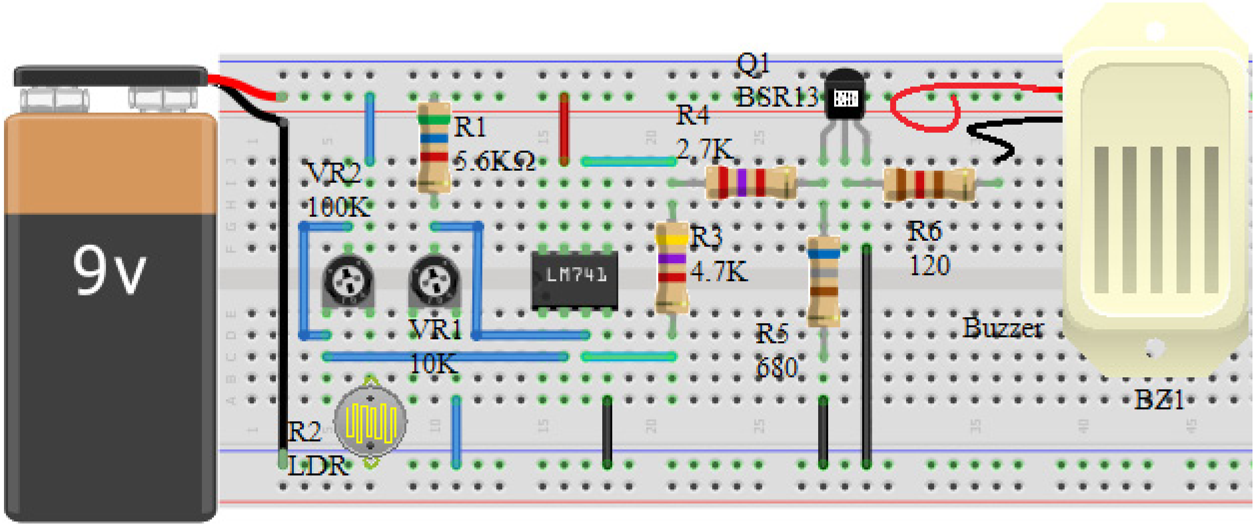
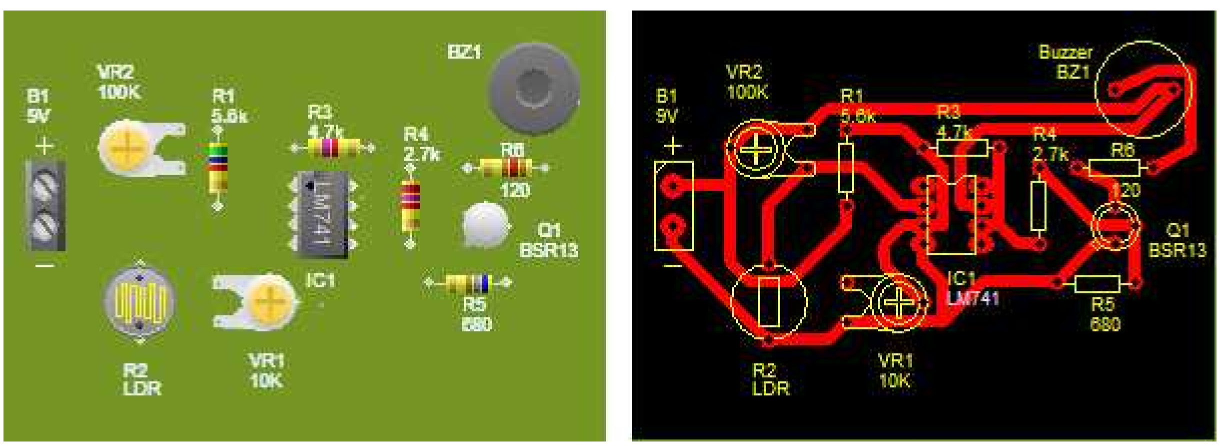


Figure 7-28: Light-level sensing with Schmitt trigger circuit connection



(a) (b)

Figure 7-29: Light-level sensing with Schmitt trigger, (a) Real world circuit and (b) PCB designer artwork

The previous circuit, figure 7-27, has been modified to respond to the darkness by interchanging the position of LDR and potentiometer VR_2. In this dark detector circuit, the voltage on inverting of op-amp is determined by the darkness sensed by the light dependent resistor LDR (R_2) and the pre-set value of potentiometer VR_2. This forms a voltage divider across the supply. As it gets darker, the potential on pin 3 decreases. Resistor R_1 and potentiometer VR_1 form other potential divider across dc supply and are taken as reference adjustable via VR_1. Potentiometer VR_2 determines dark sensibility level. Potentiometer VR_2 determines light sensitivity level of LDR. As the op-amp is configured as inverting the low input voltage at inverting make a high output voltage which for driving the transistor Q_1 into saturation. This leads the relay to be energized and closes the switch to turn on the lamp. Diode D_1 protects the transistor from back excessive current from the coil of relay.

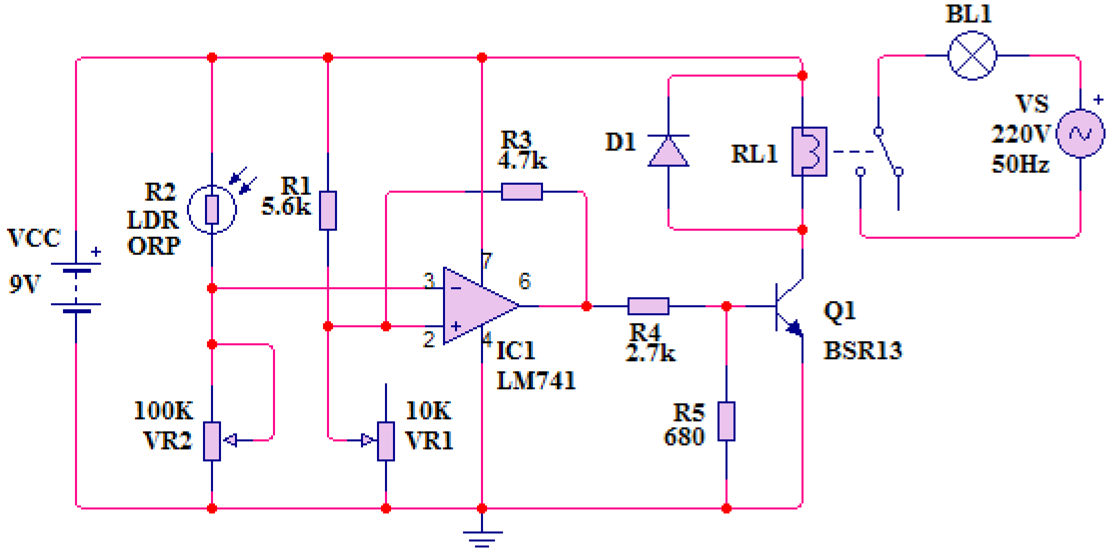


Figure 7-30: Circuit for dark detector with Schmitt trigger

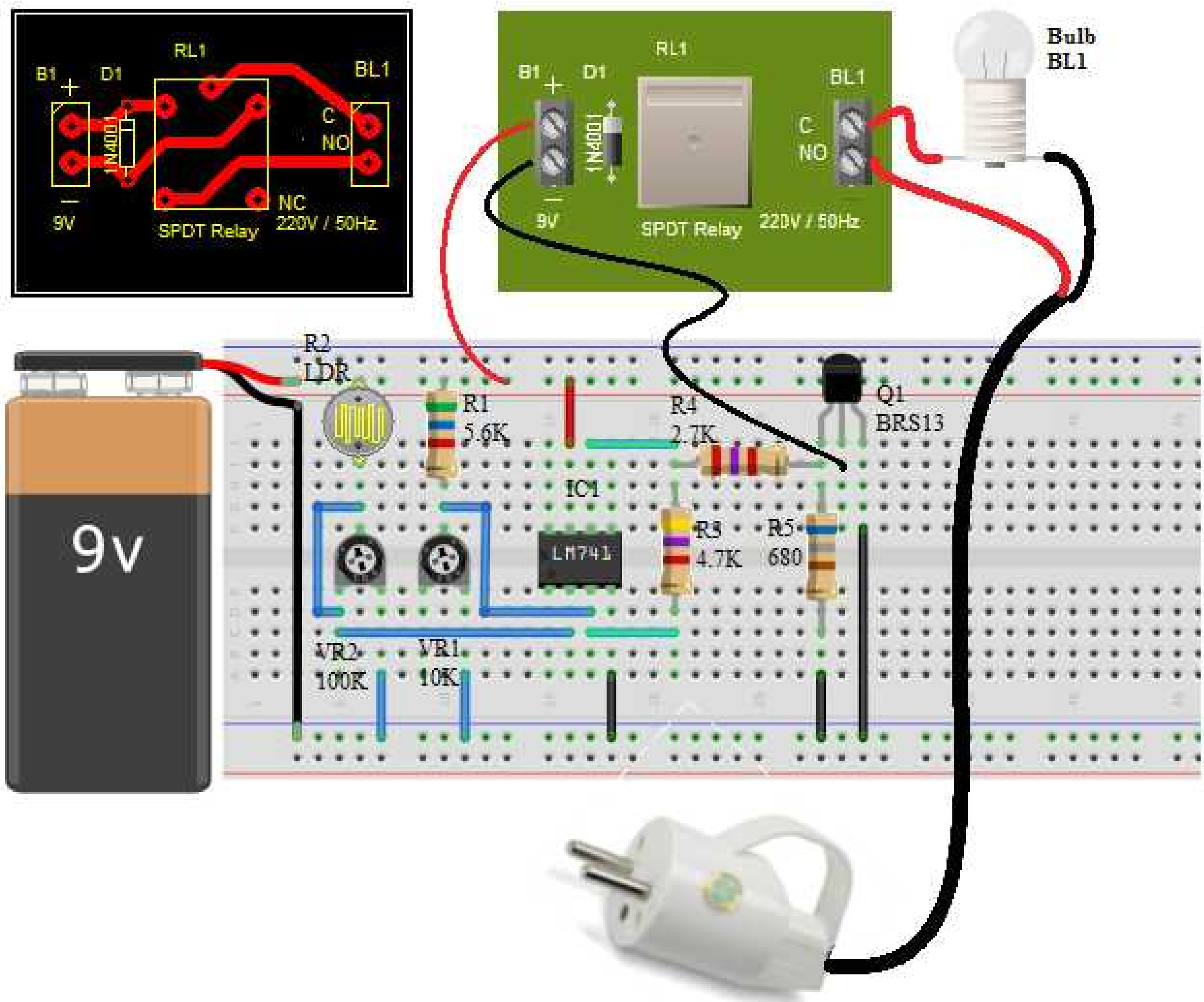
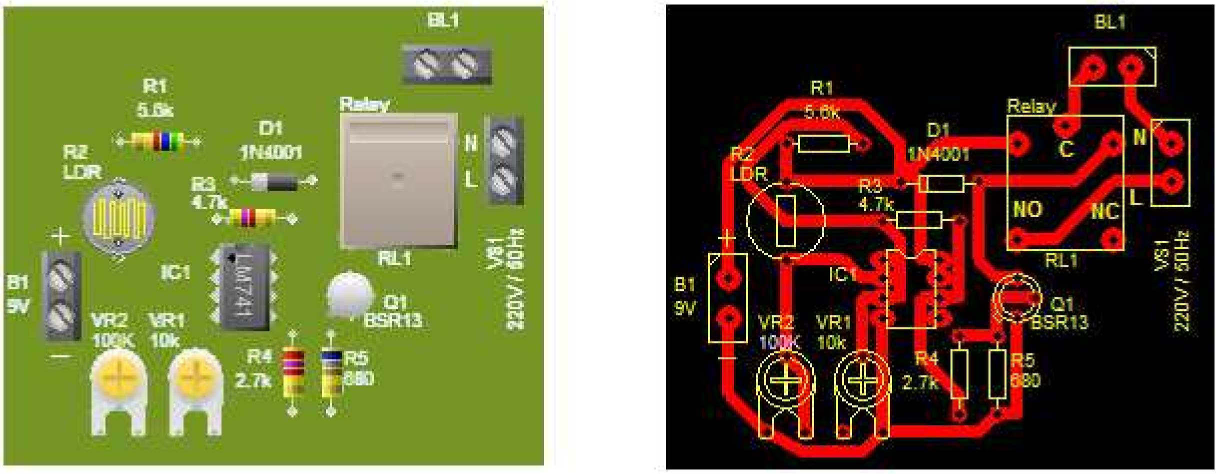


Figure 7-31: Dark detector with Schmitt trigger circuit connection



(a) (b)

Figure 7-32: Dark detector with Schmitt trigger, (a) Real world circuit and (b) PCB designer artwork

7. 5. Temperature Activated Switch

Temperature activated switch work the same way the light activated switch expect the use of temperature sensor. Thermistor is the most temperature sensor used in electronic circuits.

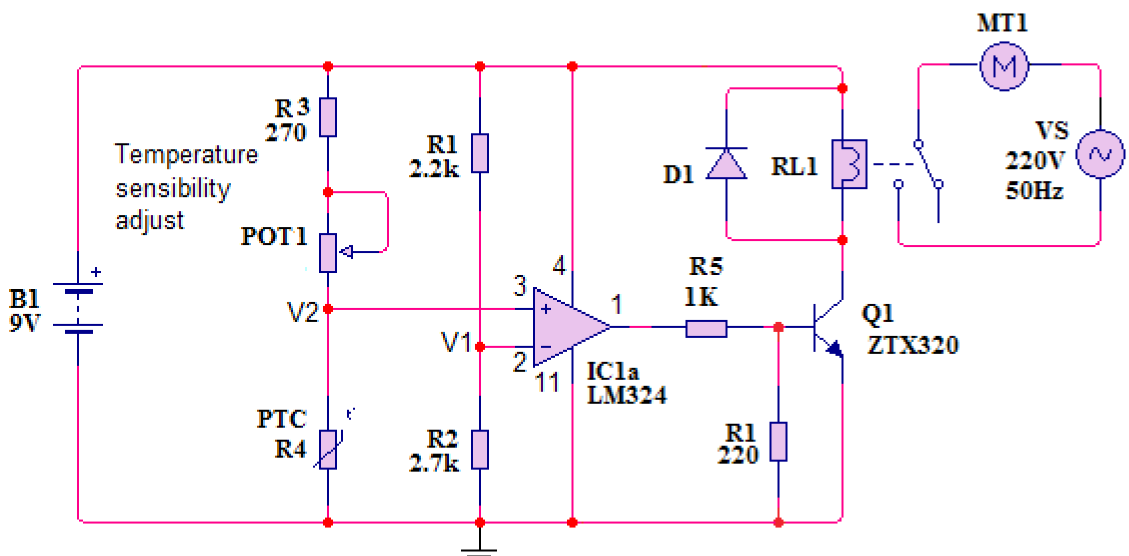


Figure 7-33: Temperature activated switch

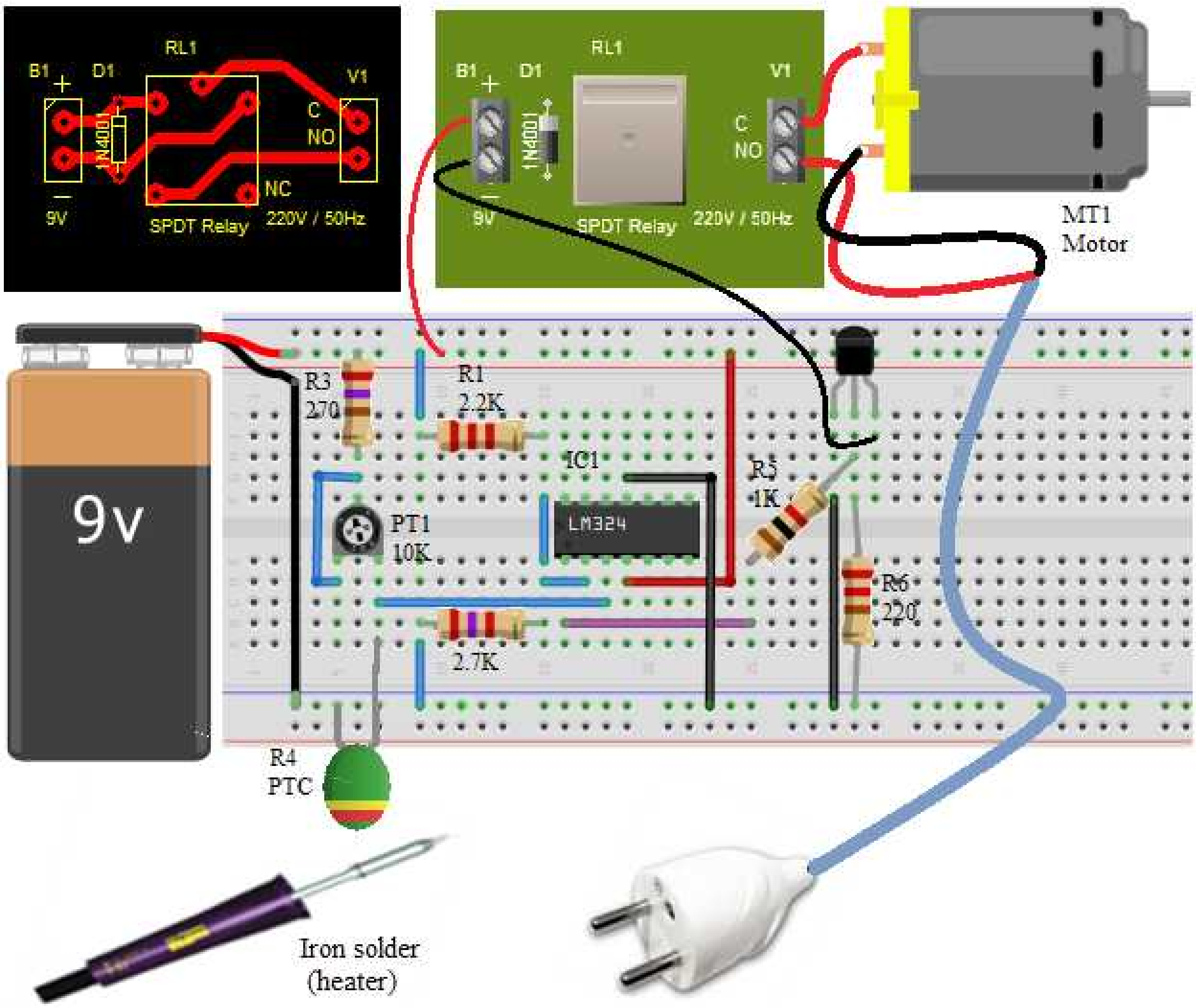


Figure 7-34: Temperature activated switch circuit connection

7. 6. Bargraph Driver

A bargraph driver is one of the applications of op-amp comparator. Bargraph displays are noted for their ability to very quickly convey a relative sense of how much of something is present. They are particularly useful in process monitoring applications where quick communication of a relative value is more important than providing specific data. If we connect several op-amps as comparator, each with its own reference voltage connected to the inverting input, but each one monitoring the same voltage signal on their non-inverting inputs, we could a bargraph-style meter such as what is commonly seen on the face of stereo tuners and graphic equalizers. As the signal voltage (representing radio signal strength or audio sound level) increased, each comparator would “turn on” in sequence and send power to its respective LED. With each comparator switching “on” at different level of audio sound, the number of LEDs illuminated would indicate how strong the signal was.

Experiment 7-7: Bargraph driver

Design a bargraph driver that turns on four consecutive LEDs according to the input signal level.

Parts list

No	Item	Quantity
1	5V DC supply	1
2	Variable ac source (Audio amplifier or function generator)	1
3	5x1KΩ, 4x270Ω and 47Ω ¼ watt Resistors	10
4	Breadboard	1
5	LED	4
6	Connecting wires 22-gauge solid	50cm
7	Op-amp IC (4-in 1 op-amp IC) LM324	1

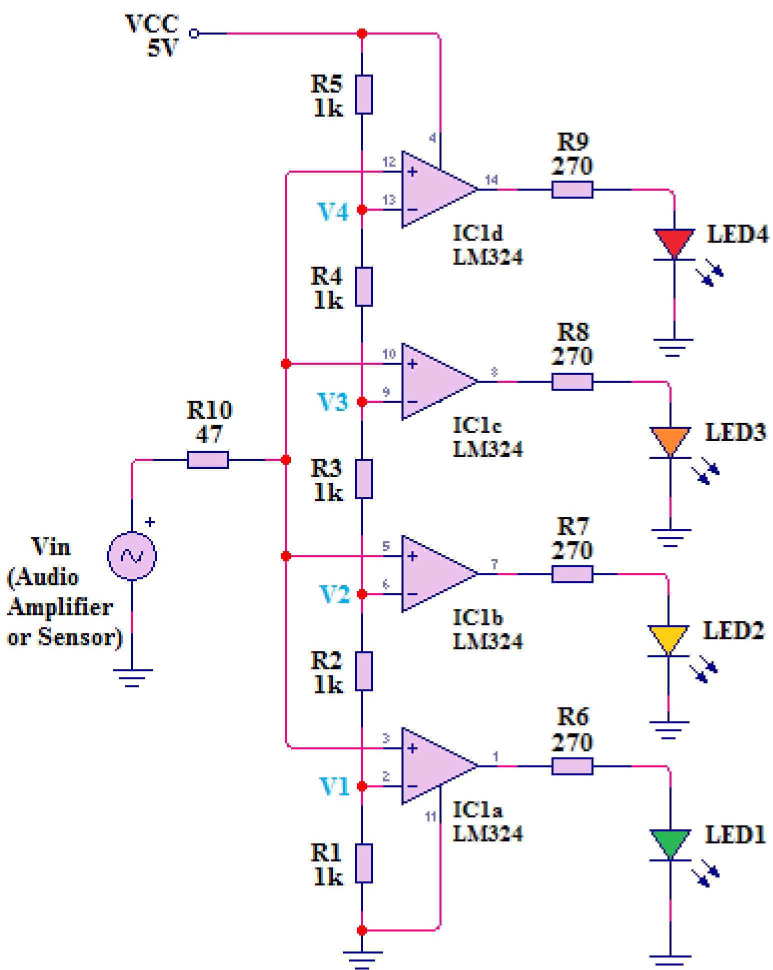


Figure 7-35: Bargraph driver using op-amp comparator

In the circuit shown in figure 7-35, LED_1 would be the first to be light up as the input voltage increased in positive direction. As the input voltage continued to increase, the other LEDs would illuminate in succession, until all were lit.

For VCC = 5V, the reference voltage (V_1 up to V_4) on each op-amp inverting input is calculated by using voltage divider formula.

V_1 = VCC*$R_1/R_1+R_2+R_3+R_4+R_5$
= 5V*1KΩ/1KΩ+1KΩ+1KΩ+1KΩ+1KΩ = 1V

V_2 = VCC*$(R_1 +R_2)/R_1+R_2+R_3+R_4+R_5$
= 5V*(1KΩ+1KΩ)/1KΩ+1KΩ+1KΩ+1KΩ+1KΩ = 2V

V_3 = VCC*$(R_1 +R_2$ +R3)/$R_1+R_2+R_3+R_4+R_5$
= 5V*(1KΩ+1KΩ+R_3)/1KΩ+1KΩ+1KΩ+1KΩ +1KΩ= 3V

V_4 = VCC*$(R_1 +R_2 +R_3+R_4)/R_1+R_2+R_3+R_4+R_5$
= 5V*(1KΩ+1KΩ+1KΩ+1KΩ)/1KΩ+1KΩ+1KΩ+1KΩ+1KΩ = 4V

Since $R_1=R_2=R_3=R_4=R_4= R_5$=1KΩ and are in series connection, the reference voltages can simply be calculated as:

$$V_k = VCC*k/n,$$

Where *k* is the order of the reference voltage and *n* number of series connected resistor that form voltage divider.

Thus,

V_1=VCC*1/5=5V*1/5=1V

V_2=VCC*2/5=5V*2/5=2V

V_3=VCC*3/5=5V*3/5=3V

V_4=VCC*2/5=5V*4/5=4V

chap 7

If V_{in} (voltage applied on noninverting of comparator IC1a) is greater than V_1 (reference voltage applied on inverting of IC1a), but less than V_2, V_3 and V_4, the output of IC1a is driven in saturation and outputs voltage close to VCC to make LED_1 on. In this case LED_2, LED_3 and LED_4 are stay off.

If voltage applied to the input (V_{in}) is greater than V_2, both op-amp IC1a and IC1b are driven in saturation to make LED_1 and LED_2 on while LED_3 and LED_4 are off.

All LEDs will be on if the input voltage (V_{in}) is greater than V_4, i.e. V_{in}>4V.

This very same technology is used in some analog-to-digital signal converters, namely the *flash converter*, to translate an analog signal quantity into a series of on/off voltages representing a digital number.

Note: LM3915 is one of the IC bargraph driver with 10 outputs.

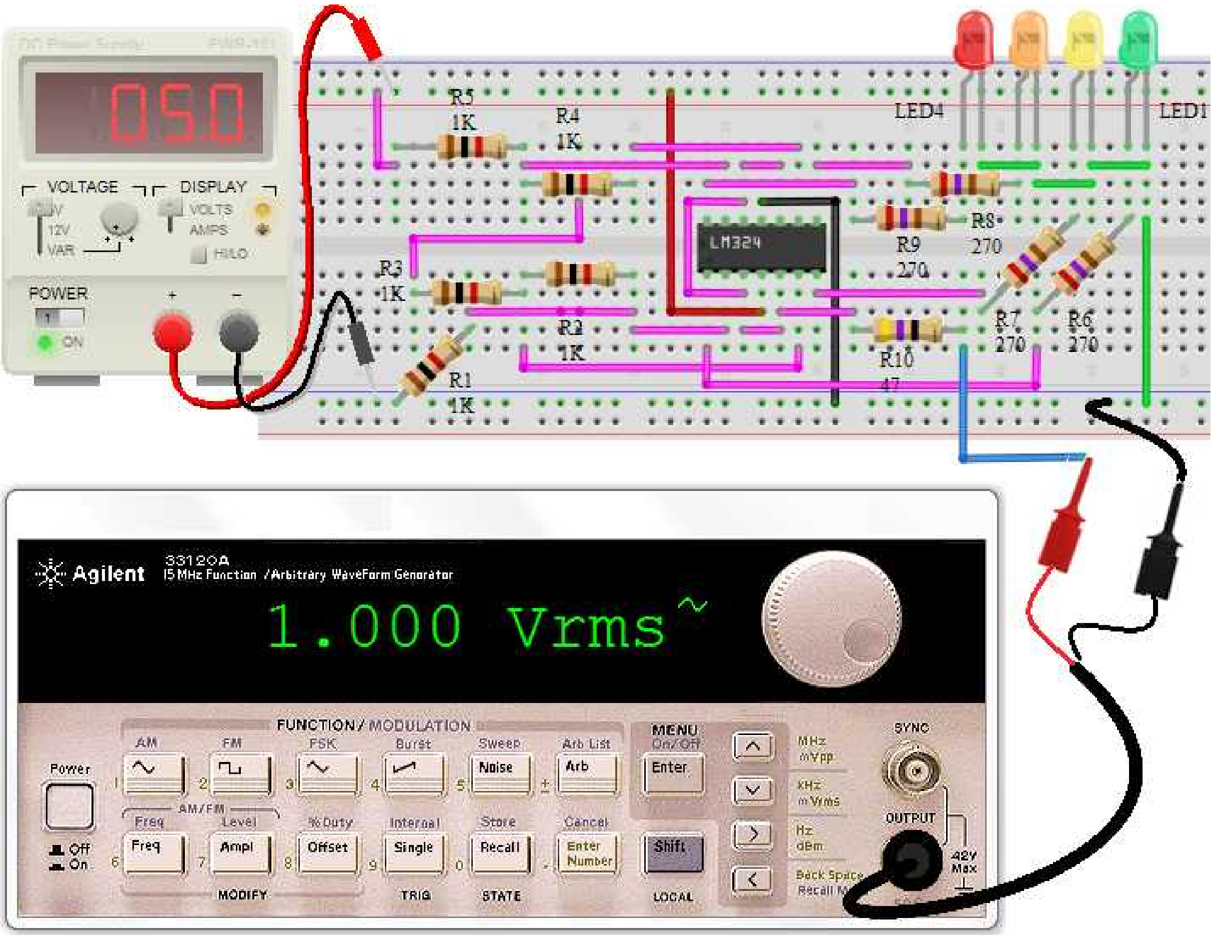


Figure 7-36: Bargraph driver using op-amp comparator circuit connection

Procedure

1. Set up the circuit as illustrate in figure 7-36 for the 4 segments bargraph display. Use short wiring to make connection on breadboard. Avoid arching wires connection. Make all wiring to be neatly routed and flush to the breadboard surface. Also avoid having wires or components passing over the IC in case it has to be removed. Make sure to identify correctly the pins of IC. VCC is at pin 4 and ground is at pin 11 of 324 IC. Make sure to tie together the negative of the dc supply, input ac source (function generator) with that of the circuit.
2. Power on the circuit and adjust the function generator to output nearly zero volts. Adjust the knob of the function generator to increase the input voltage until the green LED (LED_1) emits the light. Measure and record the differential voltage at each op-amp input for the green LED to just light. To get the correct polarity, place the red lead (+) at the plus input of the op-amp and the black lead (−) to the minus of the op-amp.
3. Increase V_{in} for nearly 2.2V, 3.2V and 4.2V to light up other remaining LEDs. Repeat step 2.
4. Decrease V_{in} to turn all LEDs off. Replace function generator with an audio amplifier and play a music or just use your voice with microphone and speak by increasing you voice also having a look on LEDs illumination.

7. 7. Noninverting Amplifier

An op-amp connected in a closed-loop configuration as a nonimverting amplifier is shown in figure 7-37. The input signal is applied to the noninverting (+) input. A portion of the output is applied back to the inverting (–) input through the feedback network. This constitutes negative feedback. The feedback fraction, *B*, is determined by R_f and *Ri*, which form a voltage divider. The attenuation of the feedback network is the portion of the output returned to the inverting input and determines the gain of the amplifier. This smaller feedback voltage, V_f, can be written:

$$V_f = V_{out}(Ri/Ri+R_f) = BV_{out}$$

The fraction of the output voltage, V_{out}, that is returned to the inverting input is found by applying the voltage-divider rule to the feedback network.

$$V_{in} \approx BV_{out} \approx V_{out}(Ri/Ri+R_f)$$

$$V_{out}/V_{in} \approx 1/B$$

Rearranging,

$$V_{out}/V_{in} = (Ri/Ri+R_f)$$

$$Av = 1 + Ri/R_f \qquad \text{[Equation 7-3]}$$

chap 7

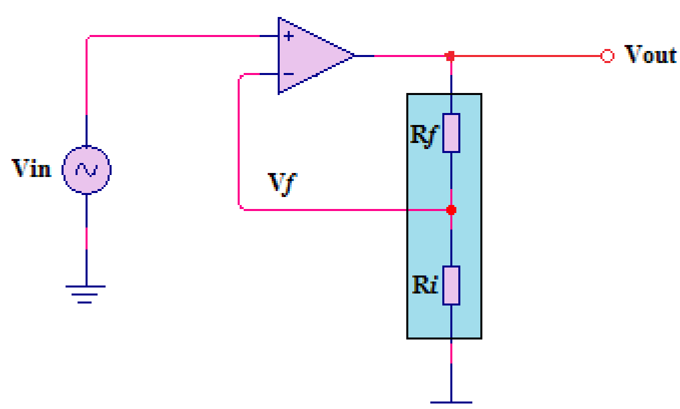


Figure 7-37: Noninverting amplifier base circuit

The equation 7-3 shows that the closed-loop voltage gain, A, of the noninverting amplifier is not dependent on the op-amp's open-loop gain but can be set by selecting R*i* and R_f. This equation is based on the assumption that the open-loop gain is very high compared to the ratio of the feedback resistors, causing the input differential voltage to be nearly very small. In nearly all practical circuits, this is an excellent assumption.

Voltage-Follower

The *voltage-follower* configuration is a special case of the noninverting amplifier where all the output is fed back to the inverting (–) input by a straight connection as shown in figure 7-38.

As you can see, the straight feedback connection has a voltage gain of approximately one (1). The closed-loop voltage gain of noninverting amplifier is 1/*B* as previously derived.

Since $B = 1$, the closed-loop gain of the voltage-follower is:

$$Av = 1 \qquad \text{[Equation 7-4]}$$

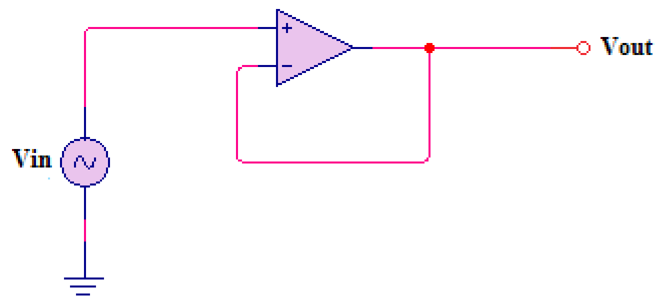


Figure 7-38: Op-amp voltage-follower base circuit

The most important features of the voltage-follower configuration are its very high input impedance and its very low output impedance. These features make it a nearly ideal buffer amplifier for interfacing high-impedance sources and low-impedance loads.

Experiment 7-8: Noninverting amplifier

Although the standard op-amp configuration is as an inverting amplifier, there are some applications where such inversion is not wanted. Since the op-amp itself is actually a differential amplifier, there is no reason why it cannot be configured to operate in a non-inverting mode.

However, we can't just switch the inverting and non-inverting inputs to the amplifier itself. We will still need negative feedback to control the working gain of the circuit. Therefore, we will need to leave the resistor structure around the op-amp intact, and swap the input and ground connections to the overall circuit.

Of course, in doing so, we will change the characteristics of the overall circuit, so that its behavior will necessarily be different in some way. In this experiment, we will construct and test such a circuit, and determine experimentally just how this circuit behaves.

Parts and materials

No	Item	Specification	Quantity
1	Op-amp IC	µA741	1
2	Feedback, Load, series and input resistors	100K, 10KΩ, 100Ω and 4.7KΩ	3
3	0.1V-2V/1KHz sinusoidal voltage source	Function generator	1
4	Oscilloscope	Digital oscilloscope	1
5	Prototyping board	RSR 03MB102 breadboard	1
6	Connecting wire	22-gauge solid wire	40cm
7	±9V to ±15V dual polarity dc supply	9V battery	2

The circuit of figure 7-39 shows the implementation of noninverting amplifier. The series resistor, Rs, still affects only the input offset voltage, so we can ignore it for our calculations of circuit gain. Therefore, we must look at the voltage divider formed by R_f and R_{in}. V_{out} must be such that the output of the voltage divider is the same as V_{in}.

Thus,

$V_{in} = V_{out} * R_{in} / R_{in} + R_f$

$V_{in} / V_{out} = R_{in} / R_{in} + R_f$

$V_{out} / V_{in} = (R_{in} + R_f) / R_{in}$

$V_{out} / V_{in} = 1 + R_f / R_{in}$

$A_v = 1 + 100K\Omega / 4.7K\Omega = 22.3$

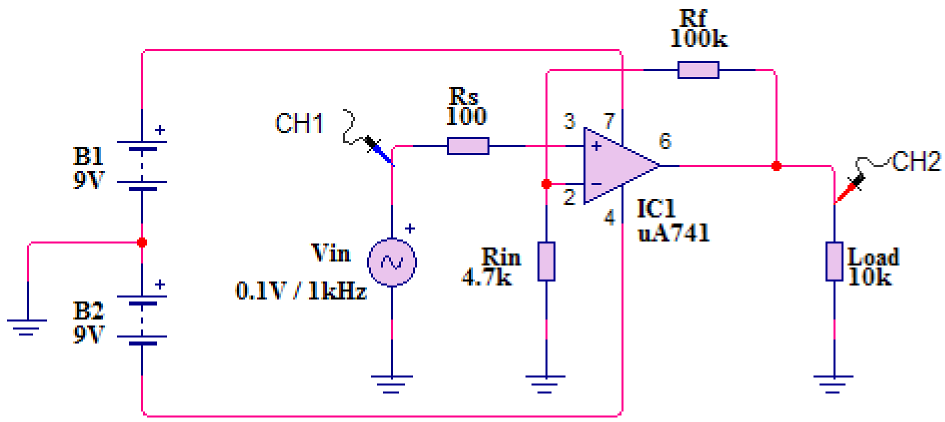


Figure 7-39: Noninverting amplifier

chap 7

Procedure

1. Wire the bus strips on your breadboard to provide positive power, negative power and ground buses. Connect pin 4 (VCC–) to the negative power supply bus (–9V). Connect Pin 7 (VCC+) to the positive power supply bus (+9 V). For this experiment connect pin 4 to the positive of B1 and pin 7 to the negative of B2. Take the ground at the interconnection of the two batteries.
2. Locate the pins of IC. The dot is located next to pin 1 and the notch is located between pins 1 and 8. Plug an op-amp into the breadboard so that it straddles the gap between the top and bottom sections of the socket strip. Pin 1 should be to the left.
3. Set the function generator to output 100mV rms (282mV peak-to-peak) at 50Hz sine wave. Connect the signal lead to pin 3, through 100Ω R_s, of µA741 IC op-amp and ground lead to common ground of the circuit.
4. Connect CH1 of the scope to V_{in}, just on the terminal of the function generator, and CH2 to V_{out} at pin 6, on the terminal of the R_{Load}. Set CH1 Volts/Div to 100mV and CH2 Volts/Div to 1V and Time/Div to 500µS.
5. Press measure button to measure V_{in}, V_{out} and frequency.
6. As practical results, V_{out} peak value is 3.18V thus the amplification gain Av is:

   $A_V = V_{out}/V_{in} = 3.18V/0.14142V = 22.48$; compared to the theoretical gain 22.3, the small difference is tolerable and this is normal, practical result differ in most of the case to the theoretical results.

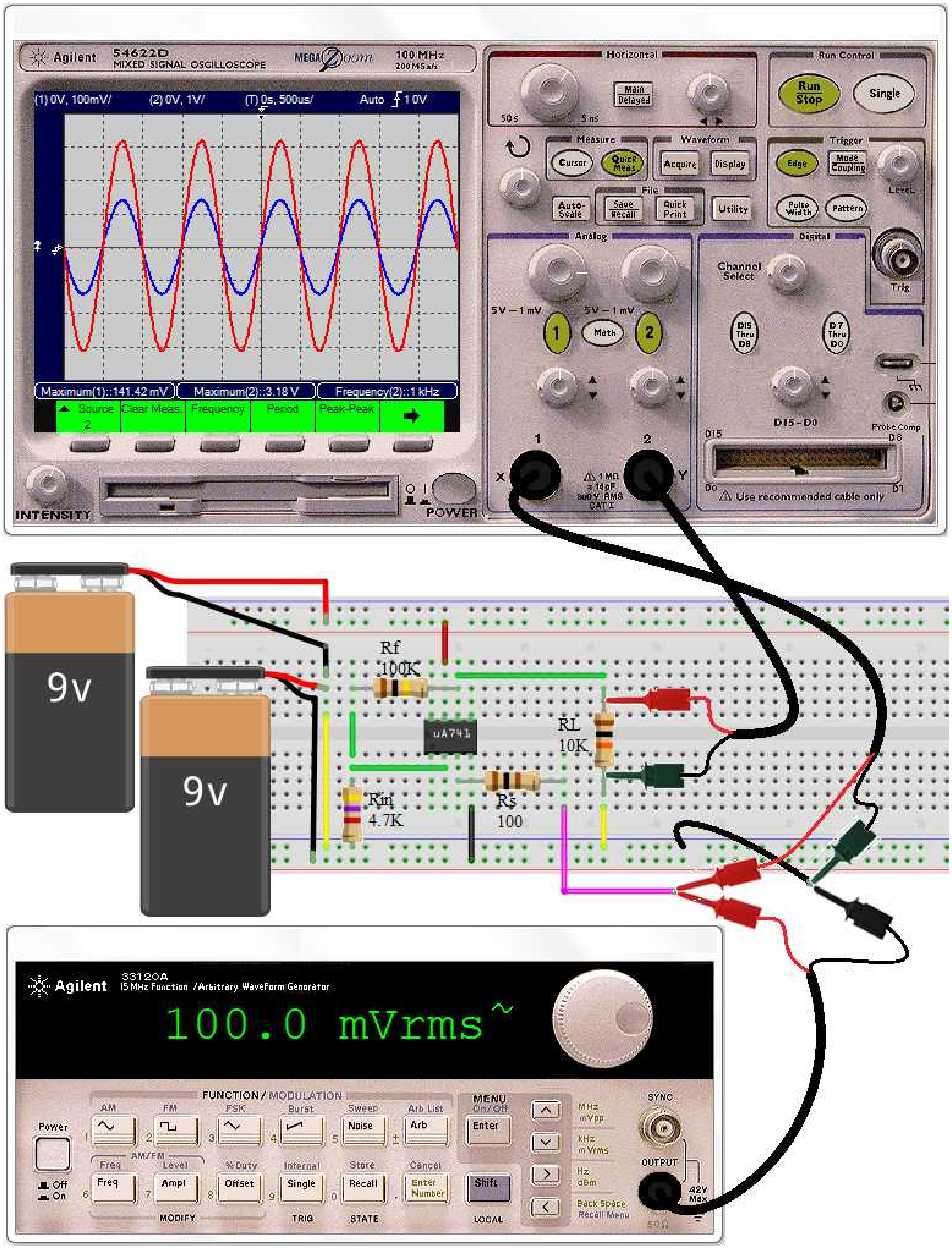


Figure 7-40: Noninverting amplifier circuit connection

Voltage-follower (non-inverting)

As mentioned above the voltage-follower is constructed from a noninverting amplifier by feeding back all the output voltage to the inverting terminal. No feedback and input resistors required therefore the gain is unit (1).

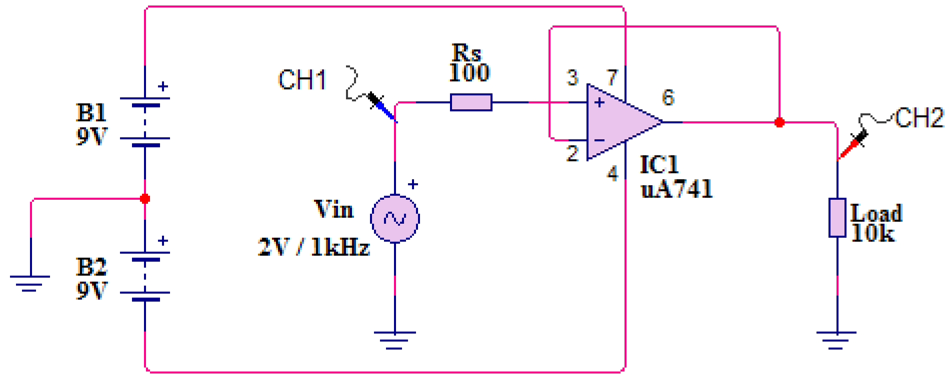


Figure 7-41: Voltage-follower circuit

Procedure

1. Connect the circuit as illustrated in figure 7-42.
2. Set the function generator to output 2V rms (2.83V peak) at 50Hz sine wave. Connect the signal lead to pin 3, through 100Ω (R_s), of µA741 IC op-amp and ground lead to common ground of the circuit.
3. Connect CH1 of the oscilloscope to V_{in}, just on the terminal of the function generator, and CH2 to V_{out} at pin 6, on the terminal of the R_{Load}. Set CH1 Volts/Div to 1V and CH2 Volts/Div to 2V and Time/Div to 500µS.
4. Press measure button to measure V_{in}, V_{out} and frequency.
5. As practical result, V_{out} peak value is 2.83V thus the amplification gain Av is:

   $A_V = V_{out}/V_{in}$ = 2.83V/2.83V = 1; the same as theoretical.

A key point on a noninverting amplifier is that the output voltage polarity is the same as the input polarity. Thus, this amplifier circuit is indeed non-inverting. It is still bipolar, in that a negative input voltage produces a negative output voltage at twice the voltage. But the output polarity is always the same as the input polarity. Replacing R_{in} and R_f with equal resistors, i.e. 100KΩ, you calculatethe gain of the non-inverting amplifier, since they are equal in value, their ratio is 1, and the total gain of the circuit is 1 + 1 = 2. Replace both R_{in} and R_f in experimental circuit with the equal value, 100KΩ just on the top of µA741 IC at pin 6-2 and pin 2 and ground. Does your result gain V_{out}/V_{in} reflect to 2? Of course, with an input of 0.00 volts, the output should also be 0.00 since the function generator is adjustable. You may have seen a very small offset voltage at the output, but not much. Then, with 1.00 volt input, V_{out} measured 2.00 volts (between 1.96 and 2.04 volts) at the output. You should have obtained similar results for each value of V_{in}, obtaining a V_{out} that was just double the input voltage. Now use R_f less than R_{in}, by swapping R_f and R_{in} values in experimental circuit, now R_f=4.7KΩ and R_{in} =100KΩ.

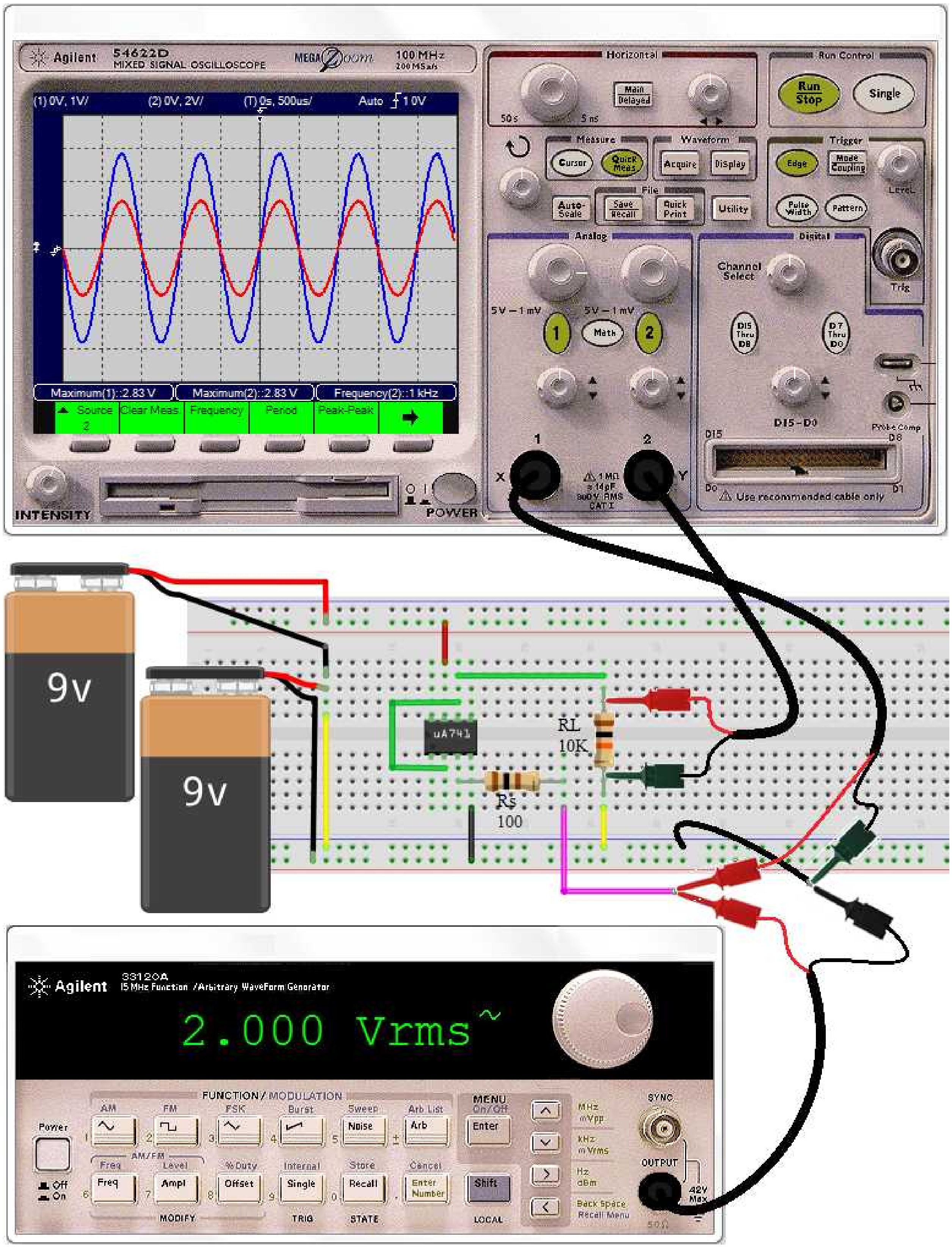


Figure 7-42: Voltage-follower circuit connection

Measure the output voltage and calculate the gain. One key point here is that, unlike the standard inverting amplifier configuration, the non-inverting amplifier configuration using op-amps cannot produce a voltage gain less than 1. This does not make this configuration less useful than the inverting op-amp configuration, but you need to be aware of this characteristic whenever you design or modify such a circuit. You can reduce the gain by using a voltage divider at the non-inverting input. This reduces the input voltage applied to the op-amp, and hence reduces the effective gain of the overall circuit.

7. 8. Inverting Amplifier

An op-amp connected as an *inverting amplifier* with a controlled amount of voltage gain is shown in figure 7-43. The input signal is applied through a series input resistor (Ri) to the inverting (–) input. Also the output is fed back through R_f to the inverting input. The noninverting (+) input is grounded.

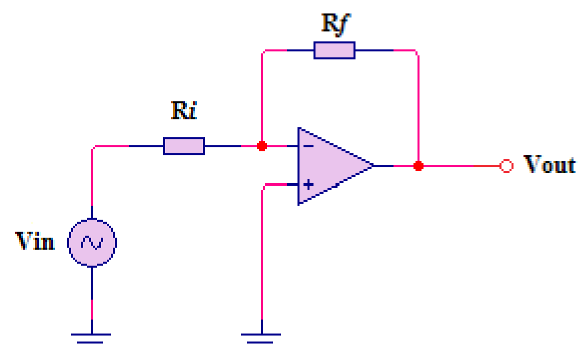


Figure 7-43: Inverting op-amp base circuit

The ideal op-amp parameters, in particular, the infinite input impedance, are useful for analyzing the inverting amplifier. Infinite input impedance implies that there is no currents out of the inverting input. If there is no current through the input impedance, then there must be no voltage drop between the inverting and noninverting. This means that the voltage at the inverting (–) input is zero because the noninverting (+) is grounded. This zero voltage at the inverting input terminal is referred to as *virtual ground*. This condition is illustrated in figure 7-44.

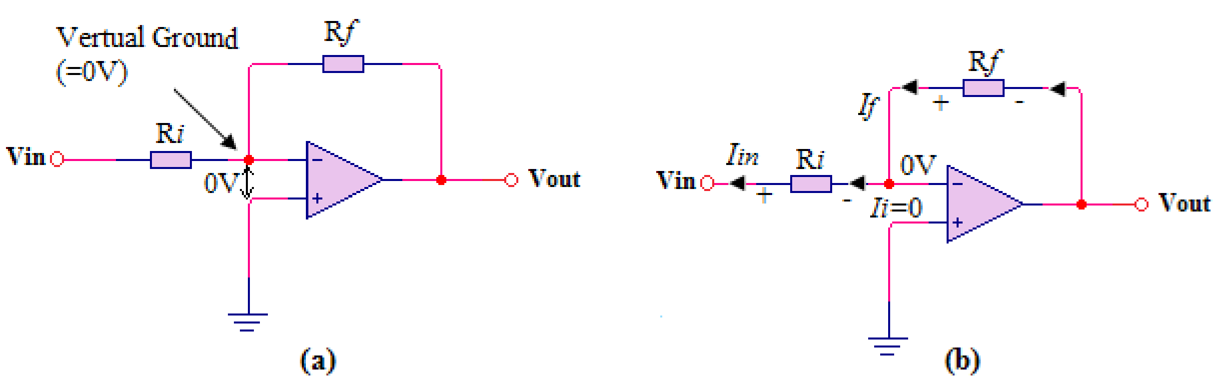


Figure 7-44: Virtual ground concept and closed-loop voltage gain development for the inverting amplifier; (a) Virtual ground, (b) $I_{in} = I_f$ and current at the inverting input, $Ii = 0$

chap 7

Thus,

$$I_{in} = I_f$$

The voltage across Ri equals V_{in} because of virtual ground on the other side of the resistor. Therefore,

$$I_{in} = V_{in}/Ri$$

Also, the voltage across R_f equals $-V_{ou}t$ because of virtual ground, and therefore,

$$I_f = -V_{out} / R_f$$

Since $I_f = I_{in}$,

$$-V_{out} / R_f = V_{in} / Ri$$

Rearranging the terms,

$$-V_{out} / V_{in} = R_f / Ri$$

Of course, V_{out} / V_{in} is the overall gain of the inverting amplifier

$$Av = R_f / Ri \qquad \text{[Equation 7-5]}$$

The equation 7-5 shows that the closed-loop voltage gain, Av, of the inverting amplifier is the ratio of the feedback resistance, R_f, to the input resistance, Ri. The *closed-loop gain is independent of the op-amp's internal open-loop gain*. Thus, the negative feedback stabilizes the voltage gain. The negative sign indicates inversion.

chap 7

Experiment 7-9: Inverting amplifier

A standard op-amp amplifier is an inverting amplifier. The circuit is configured so that the non-inverting input is grounded directly or through a resistor, and the input signal and feedback are applied to the inverting input. The gain of this circuit configuration is set entirely by the ratio of R_f/R_{in}. The output must assume whatever voltage will hold the inverting input at ground potential, which is where the non-inverting input is held directly. This experiment will discuss on inverting amplifier and give an overview to the other circuits based on inverting amplifier.

Parts and materials

No	Item	Specification	Quantity
1	Op-amp IC	µA741	1
2	Feedback, Load, ground and input resistors	100K, 10KΩ, 100Ω and 8.2KΩ	3
3	0.0V-5V/1KHz sinusoidal voltage source	Function generator	1
4	Oscilloscope	Digital oscilloscope	1
5	Prototyping board	RSR 03MB102 breadboard	1
6	Connecting wire	22-gauge solid wire	40cm
7	±9V to ±15V dual polarity dc supply	9V battery	2

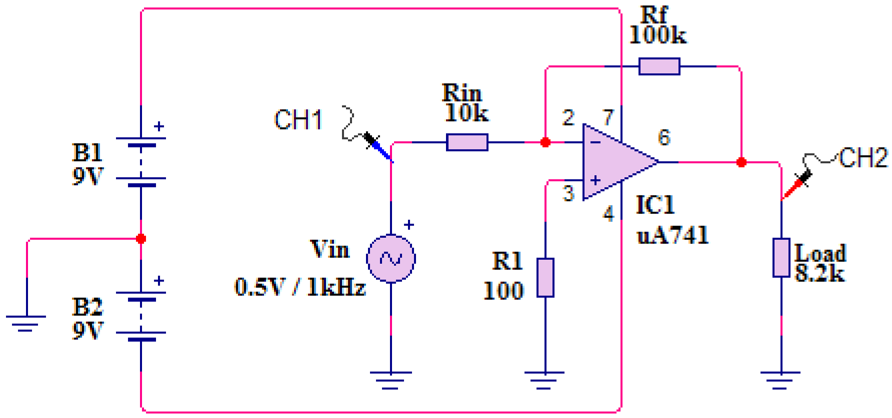


Figure 7-45: Inverting amplifier

Figure 7-45 shows an inverting amplifier. The output voltage is taken across the load resistor R_{load} just on pin 6 of µA741 IC. The input signal is applied to the inverting input terminal of the amplifier (pin 2) through the R_{in} and the feedback resistor R_f which is also connected between the output terminal (pin 6) and directly to inverting terminal (pin 2). The noninverting terminal is connected to ground through R_1.

Gain $Av = R_f/R_{in} = 100K\Omega/10K\Omega = 10$

$V_{out} = -Av*V_{in} = -10*0.5V = -5V$ (negative sign means inversion)

chap 7

Procedure

1. Connect the circuit as illustrated in figure 7-46.
2. Set oscilloscope CH1 volt/div to 500mV and CH2 Volt/div to 2V. Set function generator to output 0.5V rms (0.707V peak) at 1KHz. Power the circuit with symmetric dc voltage 9-15V.
3. Measure using oscilloscope V_{in} and V_{out} and calculate the gain $Av= V_{out}/V_{in}$. Allowing for the inevitable errors that you have experienced, does the gain of this circuit closely match the gain you calculated for it?
4. Swap R_f and R_{in}, now $R_f = 10K\Omega$ and $R_{in} = 100K\Omega$, thus the gain $Av = 10K\Omega/100K\Omega = 0.1$. Repeat step 3.

The output Voltage V_{out} is of the opposite polarity as the input voltage V_{in}. The Gain of the inverting amplifier can be set below unity i.e. Av<1. For this reason the inverting amplifier configuration with feedbacks lends itself to a majority of applications as against those of the noninverting amplifier.

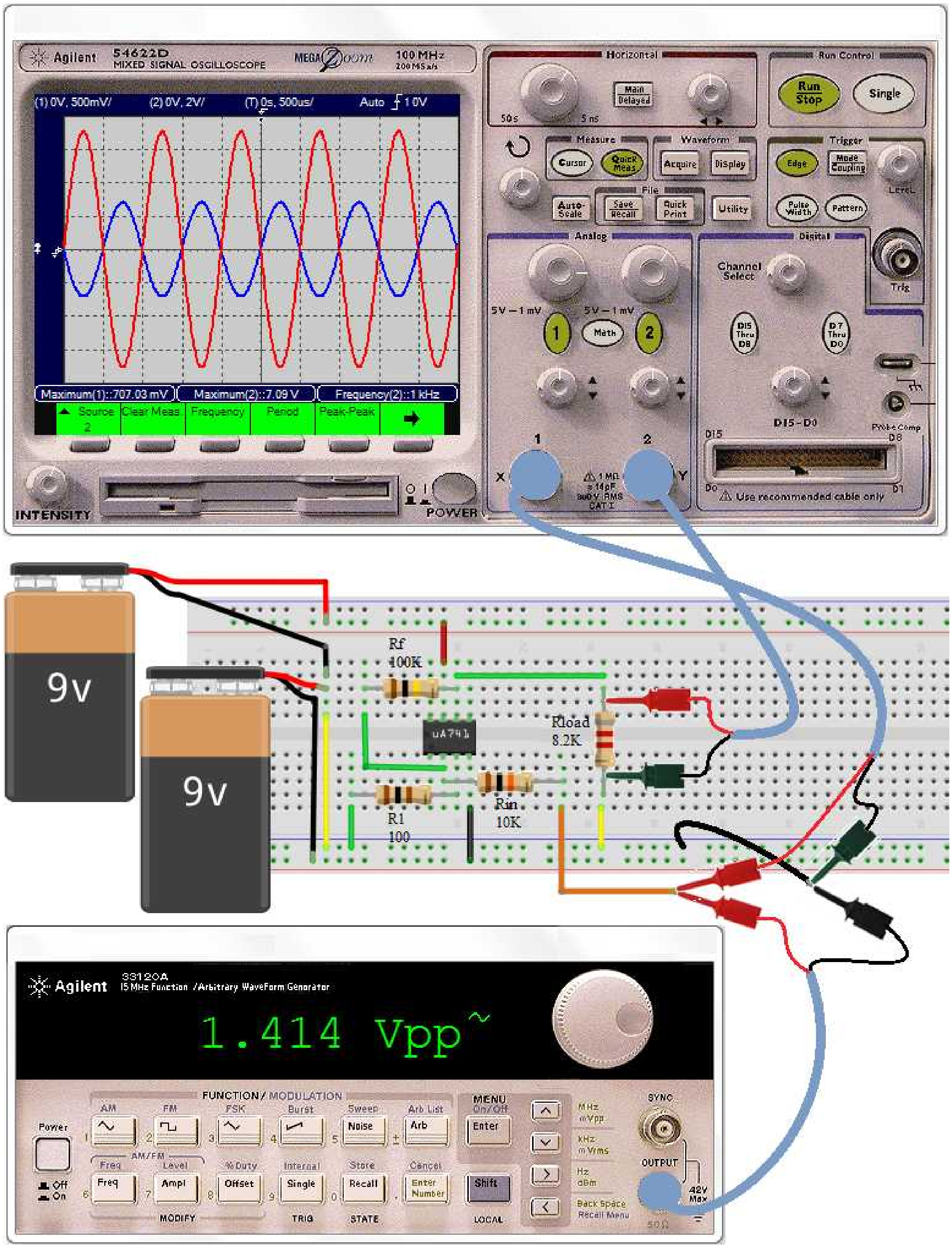


Figure 7-46: Inverting amplifier circuit connection

7. 9. Subtractor (differential) Amplifier

A subtractor is a circuit that gives the difference of the two inputs, $V_{out} = V_{in2} - V_{in1}$, Where V_{in1} and V_{in2} are the inputs. By connecting one input voltage V_{in1} to inverting terminal and another input voltage V_{in2} to the non-inverting terminal, we get the resulting circuit as the Subtractor. This is also called as differential or difference amplifier using op-amps. Output of a differential amplifier (subtractor) is given as:

$$V_{out} = (-R_f/R_{in1})\,(V_{in1} - V_{in2})$$

If all external resistors are equal in value, then the gain of the amplifier is equal to 1.

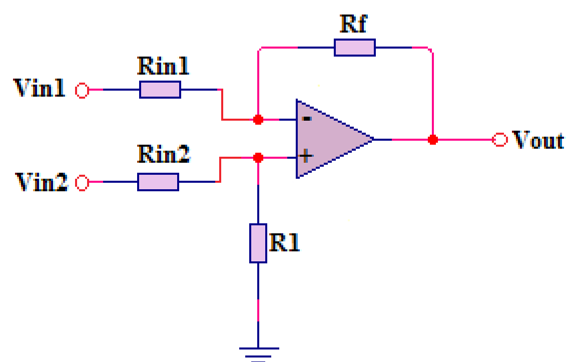


Figure 7-47: Subtractor amplifier base circuit

The output voltage of the differential amplifier with a gain of 1 is: $V_{out} = (V_2 - V_1)$ Thus the output voltage V_{out} is equal to the voltage V_{in2} applied to the non-inverting terminal minus the voltage V_{in1} applied to the inverting terminal. Hence the circuit is called a Subtractor.

Experiment 7-10: Subtractor (differential) amplifier

Design a subtractor amplifier based on op-amp which the output voltage is the difference of the two input voltages applied on inverting and non-inverting of the op-amp. Start with voltage on inverting less than that on non-inverting and vice versa and make a record on each result.

Parts and materials

No	Item	Quantity
1	Op-amp IC LM741	1
2	Feedback, Load and input resistors 47Kx2, 4.7KΩx4, 10KΩx2,	8
3	0V-5V/1KHz sinusoidal voltage source (Function generator)	2
4	Oscilloscope, (Digital oscilloscope Tektronix TDS2024)	1
5	Prototyping board, (RSR 03MB102 breadboard)	1
6	Connecting wire, 22-gauge solid wire	60cm
7	±9V to ±15V dual polarity dc supply	1

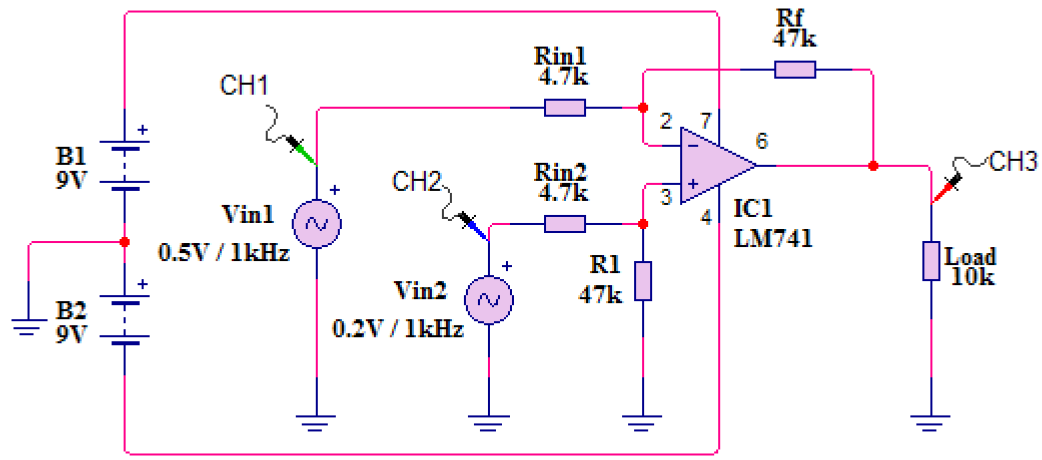


Figure 7-48: Subtractor circuit

Performing experiment

1st case: Inverting input greater than non-inverting input

With R_f=47KΩ, R_{in1} and R_{in2}= 4.7KΩ the gain *Av* of the amplifier is:

$Av = R_f/R_{in1}$ = 47KΩ/4.7KΩ =10

V_{in1} peak value = 0.5V*√2 = 0.707V

V_{in2} peak value = 0.2V*√2 = 0.283

Therefore voltage on load resistor V_{out} is:

$V_{out} = Av\ (V_{in2}-V_{in1})$ =10(0.283V–0.707V) = –4.24V (negative sign means inversion)

With R_f=4.7KΩ, R_{in1} and R_{in2}= 4.7KΩ the gain *Av* of the amplifier is:

$Av = R_f/R_{in1}$ = 4.7KΩ/4.7KΩ =1

Therefore voltage on load resistor V_{out} is:

$V_{out} = V_{in2}-V_{in1}$ = 0.283V–0.707V = –0.424V (negative sign means inversion)

This implies that when the input signal applied on inverting of the op-amp is greater than that on non-inverting, the resulting output voltage is differential of inputs and inversed with respect to both the inputs.

2nd case: Non-inverting input greater than inverting input

With R_f = 47KΩ, R_{in1} and R_{in2} = 4.7KΩ the gain *Av* of the amplifier is:

$Av = R_f/R_{in1}$ = 47KΩ/4.7KΩ =10

V_{in1} peak value = 0.2V*√2 = 0.283

V_{in2} peak value = 0.5V*√2 = 0.707V

Therefore voltage on load resistor V_{out} is:

$$V_{out} = Av\ (V_{in2} - V_{in1}) = 10(0.707V – 0.283V) = 4.24V$$

With R_f = 4.7KΩ, R_{in1} and R_{in2} = 4.7KΩ the gain *Av* of the amplifier is:

$$Av = R_f / R_{in1} = 4.7K\Omega / 4.7K\Omega = 1$$

Therefore voltage on load resistor V_{out} is:

$$V_{out} = V_{in2} - V_{in1} = 0.707V – 0.283V = 0.424V$$

This implies that when the input signal applied on non-inverting of the op-amp is greater than that on inverting, the resulting output voltage is differential of inputs and with the same polarities with respect to both the inputs.

Note that in practical the value of resistor divider R1 on non-inverting must be equal to the value of feedback resistor R*f*. Both the input voltage sources have to be on the same frequency for fine expected result.

Procedure

1. Connect the circuit as illustrated in figure 7-49.
2. Before powering on the experimental circuit, calculate the gain for resistance values R_f=47 Ω, R_{in1}= R_{in2}=4.7KΩ and expected output voltage for given input voltages. Enter you calculated results for pair's resistor R_f and R_{in1} in the first text box rebelled by "gain" in the table. Set all function generators to 1KHz. Set both CH1 and CH2 volt/Div to 500mV, CH3 volt/Div to 2V all at 500µS time/Div. If your oscilloscope has only two channels you will need to use one channel for two measurement sides one another and record the result.
3. Now turn on power to your experimental circuit, and adjust both function generators to produce 0.00 volts. Measure the output voltage, V_{out}, at pin 6 of the 741 IC. Enter this value in the appropriate text box in the table 7-3.
4. Adjust the function generator named V_{in1} to produce 0.2 volts. Measure the resulting value of V_{out}, and record this value in the corresponding text box in the table just under your entry for step 3. Increase its output voltage up to 0.5V and record measured output voltage of the op-amp just on pin 6.
5. Keep the first function generator V_{in1} to 0.5V and now increase the second function generator named V_{in2} to produce 0.2V. Measure the output voltage V_{out} of the op-amp at pin 6 just on the terminal of the load resistor and record it in table 7-3. Set the first function generator to 0.0V and measure V_{out}. Keep the first function generator to 0.0V and increase the second function generator up to 0.5V and measure the output voltage V_{out} and record it.

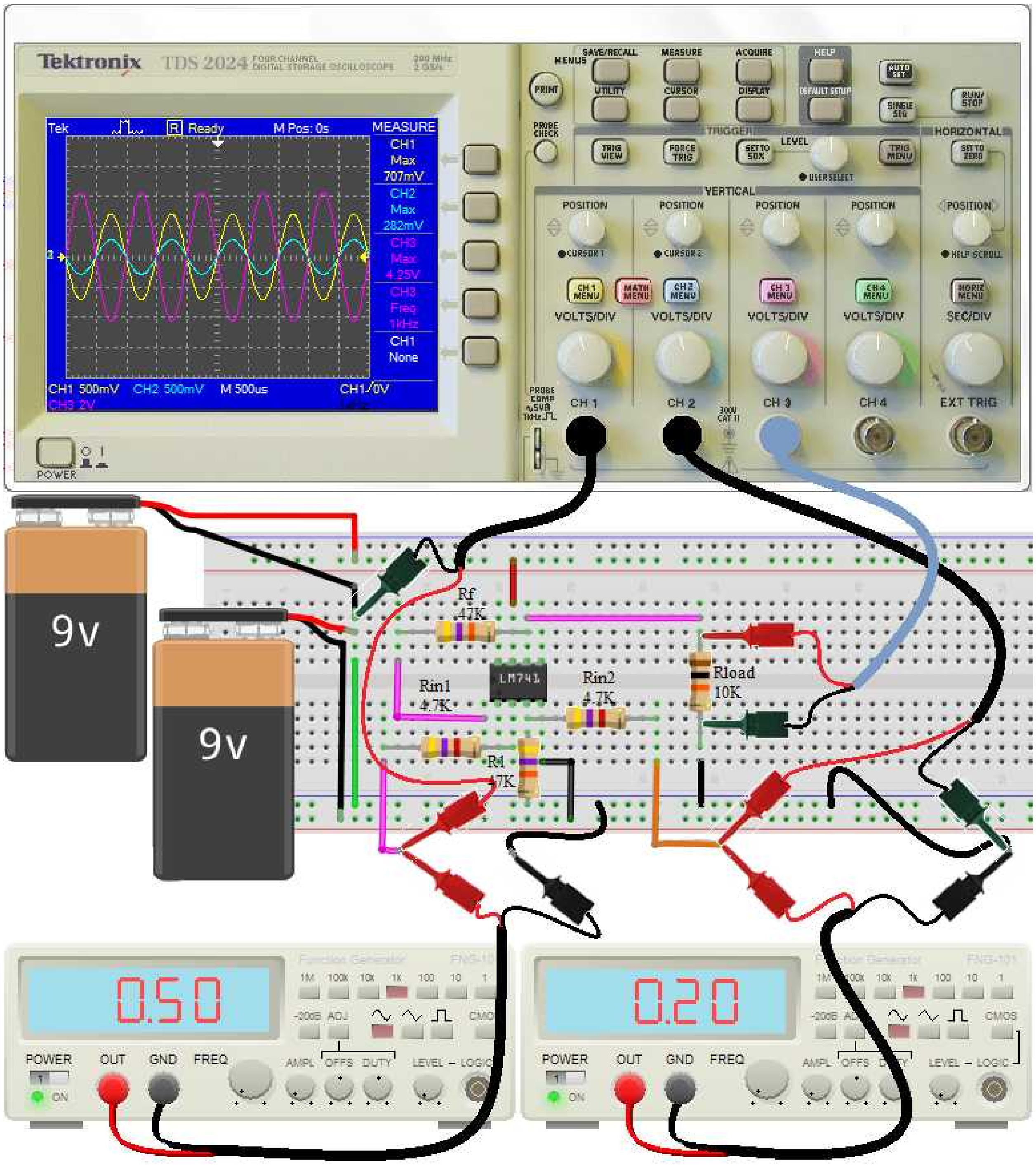


Figure 7-49: Subtractor circuit connection

6. Turn off the power, remove R_f and R_1 (47 KΩ) and replace with 4.7 KΩ to make all resistors to be 4.7 KΩ except load resistor (10K). Repeat step 3 up to step 5.
7. Pause in your measurements and look over your results. Allowing for the inevitable errors that you have experienced, does the gain of this circuit ($Av = V_{out} / \| V_{in2} - V_{in1} \|$) closely match the gain you calculated for it in step 1? If not, how do you account for the discrepancy?

Steps		1 - 5	3 - 5
R_f		47K	4.7K
R_{in1} & R_{in2}		4.7K	4.7K
Gain			
V_{in1}	V_{in2}	Measured V_{out}	
0.0	0.0		
0.2	0.0		
0.5	0.0		
0.5	0.2		
0.0	0.2		
0.0	0.5		
0.2	0.5		

Table 7-3: Measured values for different subtractor's input voltages

chap 7

7. 10. Summing Amplifier

The summing amplifier is variation of the inverting op-amp configuration. The *summing amplifier* has two or more inputs, and its output voltage is proportional to the negative of the algebraic sum of its input voltages. The averaging amplifier and the scaling amplifier are variations of the basic summing amplifier.

The figure 7-50 shows a two-input summing amplifier. Two voltages, V_{in1} and V_{in2}, are applied to inputs and produce currents I_1 and I_2 as shown. Using the concepts of infinite input impedance and virtual ground, you can see that the inverting input of the op-amp is approximately 0V, and there is no current from the input. Therefore, the total current, which is the sum of I_1 and I_2, is through R_f.

$$It = I_1 + I_2$$

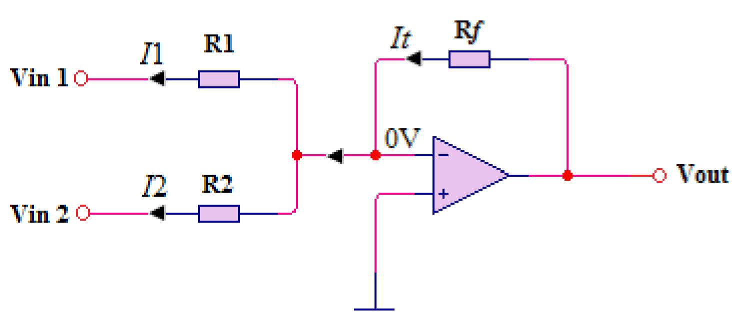


Figure 7-50: Two-input inverting summing amplifier

Since $V_{out} = -It*R_f$,

$$V_{out} = -(I_1 + I_2)R_f = -(V_{in1}/R_1 + V_{in2}/R_2)R_f$$

If all three of the resistors are equal ($R_1 = R_2 = R_f = R$), then

$$V_{out} = -(V_{in1}/R + V_{in2}/R)R$$

$$= -(V_{in1} + V_{in2})$$

This shows that the output voltage is the sum of the two input voltages. A general expression is given in equation 7-6 for a summing amplifier with *n* inputs, as shown in figure 7-51, where all the resistors are equal in value.

$$V_{out} = -(V_{in1} + V_{in2} + V_{in3} + \dots + V_{inn}) \qquad \text{[Equation 7-6]}$$

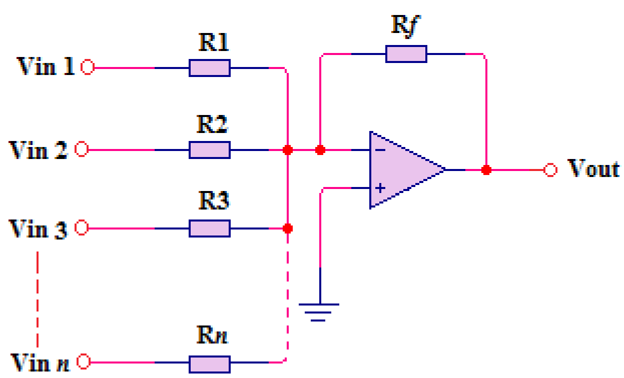


Figure 7-51: Summing amplifier with *n* inputs

7. 10. 1. Summing Amplifier with Gain Greater Than Unit

When R_f is larger than the input resistors, the amplifier has a gain of R_f / R, where *R* is the value of each input resistor. The general expression for the output is

$$V_{out} = R_f/R\,(V_{in1} + V_{in2} + V_{in3} + \dots + V_{inn}) \qquad \text{[Equation 7-7]}$$

As you can see, the output is the sum of all the input voltages multiplied by a constant determined by a ratio R_f/R.

7. 10. 2. Averaging Amplifier

An *averaging amplifier*, which is a variation of a summing amplifier, can produce the mathematical average of the input voltages. This is done by setting the ration R_f / R_1 equal to the reciprocal of the number of inputs. You know that the average of several numbers is obtained by first adding the numbers and then dividing by the quantity of numbers you have. Examination of equation 7-7 and a little thought will convince you that a summing amplifier will do the same. Circuit example is illustrated in figure 7-52.

Example: Show that the amplifier in figure 7-52 produces an output whose magnitude is the average of the input voltages.

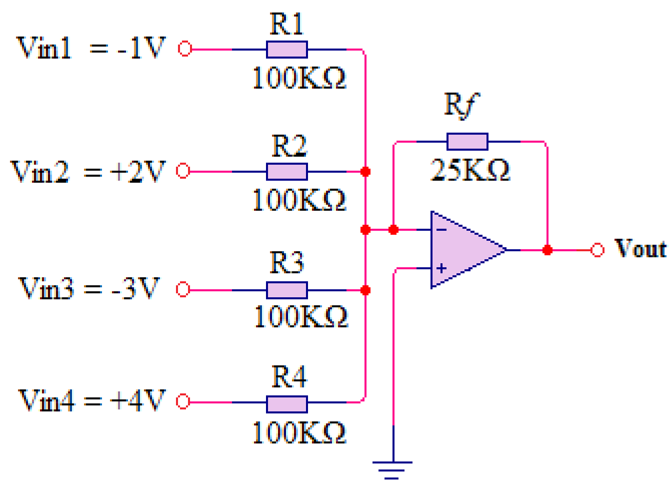


Figure 7-52: Average adder op-amp

Solution:

Since the input resistors are equal, R =100KΩ. The output voltage is

$$V_{out} = R_f/R\ (V_{in1} + V_{in2} + V_{in3} + V_{in4})$$

$$= 25K\Omega/100K\Omega\ (-1V + 2V - 3V + 4V) = ¼(2V) = 0.5V$$

A simple calculation shows that the average of the four input values is the same as V_{out} of the opposite sign.

$$V_{in(average)} = (-1V + 2V - 3V + 4\ V)/4 = 2V/4 = 0.5V$$

7. 10. 3. Scaling Adder

A different weight can be assigned to each input of a summing amplifier, forming a *scaling adder* by simply adjusting the values of the input resistors. As you have seen, the output voltage can be expressed as

$$V_{out} = -\ (R_f/R_1{*}V_{in1} + R_f/R_2{*}V_{in2} + \ldots + R_f/R_n{*}V_{inn}) \qquad \text{[Equation 7-8]}$$

The weight of a particular input is set by the ratio of R_f to the resistance for that input. For example, if an input voltage is to have a weight of 1, then $R = R_f$. Or, if a weight of 0.5 is required, $R = 2R_f$. The smaller the value of the input resistance R, the greater the weight, and vice versa.

Example: for the scaling adder in figure 7-53, determine the weight of each input voltage and find the output voltage.

Solution:

Weight of input 1: $R_f/R_1 = 10K\Omega\ /10K\Omega = 1$

Weight of input 2: $R_f/R_1 = 10K\Omega\ /100K\Omega = 0.1$

Weight of input 3: $R_f/R_1 = 10K\Omega\ /47K\Omega = 0.213$

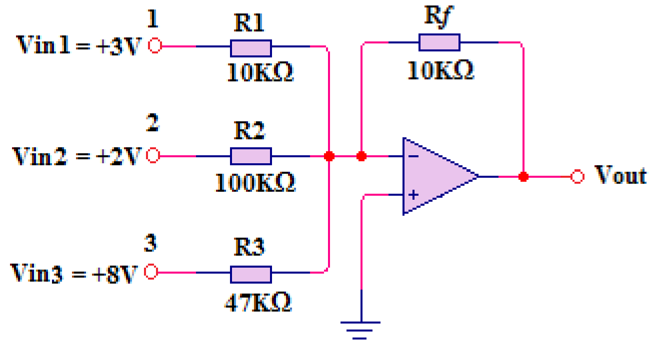


Figure 7-53: Scaling adder circuit example

The output voltage is

$$V_{out} = -(R_f/R_1*V_{in1} + R_f/R_2*V_{in2} + R_f/R_3*V_{in3})$$
$$= -[1(3) + 0.1(2V) + 0.213(8V)]$$
$$= -(3V + 0.2V + 1.7V)$$
$$= -4.9V$$

Experiment 7-11: Summing amplifier

One of the most common applications for an op-amp is to algebraically add two (or more) signals or voltages to form the sum of those signals. Such a circuit is known as a *summing amplifier*, or just as a *summer*.

The point of using an op-amp to add multiple input signals is to avoid interaction between them, so that any change in one input voltage will not have any effect on the other input. In this experiment, we will determine whether or not this is the case, and whether the op-amp will correctly generate an output voltage that reflects the arithmetic sum of the input voltages. In this experiment we will examine also the scaling adder and averaging adder.

chap 7

Parts and materials

No	Item	Specification	Quantity
1	Op-amp IC	LM741	1
2	resistors	100K, 10KΩ, 100Ω and 8.2KΩ	3
3	0.0V-5V/1KHz sinusoidal voltage source	Function generator	1
4	Oscilloscope	Digital oscilloscope	1
5	Prototyping board	RSR 03MB102 breadboard	1
6	Connecting wire	22-gauge solid wire	40cm
7	±9V to ±15V dual polarity dc supply	9V battery	2

Performing experiment

Output voltage (V_{out}) on load resistor at unit gain: $R_{in1} = R_{in2} = R_{in3} = R_{in} = 5.6K\Omega$ and $R_f = 5.6K\Omega$

Gain $Av = R_f/R_{in} = 5.6K\Omega/5.6K\Omega = 1$

$V_{out} = -Av\,(V_{in1} + V_{in2} + V_{in3})$

$= -(0.1V\sqrt{2} + 0.2V\sqrt{2} + 0.8V\sqrt{2})$

$= -(0.1414V + 0.2828V + 1.313V)$

$= -1.556V$ (negative sign means inversion)

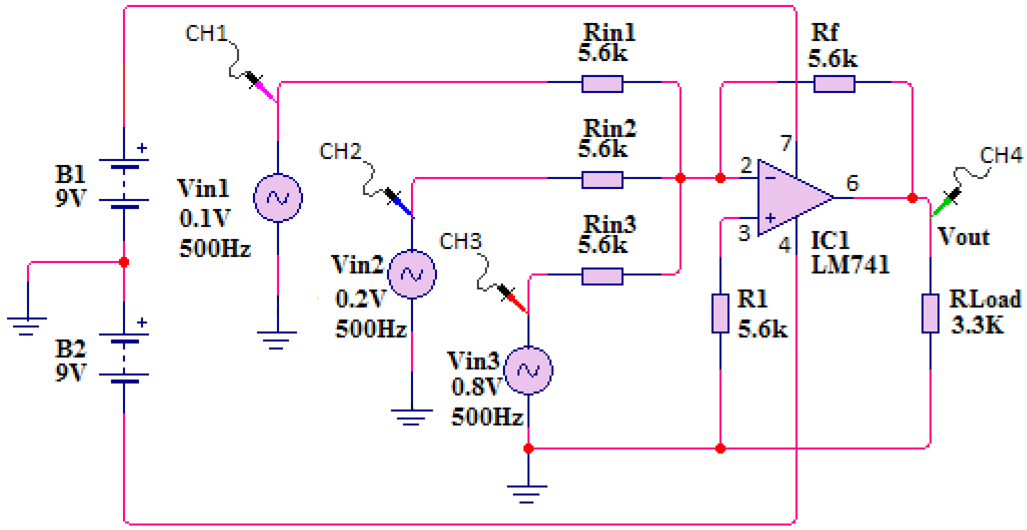


Figure 7-54: Adder (summing) amplifier

The gain of the circuit can be increased by increasing the value of feedback resistor R_f but not larger otherwise the op-amp can be driven in saturation unless the input voltages are reduced. As for this experiment we are using function generators, we consider that there are some function generators that cannot output such lower value (10 times less than the voltage V_{in1}, V_{in2} and V_{in3} represented in the circuit), we will increase the gain by increasing the value of R_f up to 27KΩ. Now the gain becomes:

Gain $Av = R_f/R_{in} = 27K\Omega/5.6K\Omega = 4.82$

Thus the output voltage $V_{out} = -Av\,(V_{in1} + V_{in2} + V_{in3})$

$= -4.82(0.1V\sqrt{2} + 0.2V\sqrt{2} + 0.8V\sqrt{2})$

$= -4.82(0.1414V + 0.2828V + 1.313V)$

$= -7.5V$ (negative sign means inversion)

The above demonstration is also applicable for averaged adder.

Consider $R_{in1} = 5.6K\Omega$, $R_{in2} = 10K\Omega$ and $R_{in3} = 56K\Omega$ with $R_f = 27K$

Weight of input 1: $R_f/R_{in1} = 27K\Omega / 5.6K\Omega = 4.82$

Weight of input 2: $R_f/R_{in2} = 27K\Omega / 10K\Omega = 2.7$

Weight of input 3: $R_f/R_{in3} = 27K\Omega / 56K\Omega = 0.482$

The output voltage V_{out} is:

$V_{out} = -(R_f/R_{in1}*V_{in1} + R_f/R_{in2}*V_{in2} + R_f/R_{in3}*V_{in3})$

$= -[4.82(0.1V\sqrt{2}) + 2.7(0.2V\sqrt{2}) + 0.482(0.8V\sqrt{2})]$

$= -(0.682V + 0.764V + 0.545V)$

$= -1.991V$ (negative sign means inversion)

This is scaling adder amplifier.

Procedure

1. Connect the circuit as illustrated in figure 7-55. Before turning on the power calculate the gain and the expected result for each value of resistors you are using so that you will be able to compare the theoretical and practical result.
2. Start by using equal resistors (5.6KΩ) for both input resistors (R_{in1}, R_{in2} and R_{in3}) and feedback resistor R_f.
3. Power on the experimental circuit and switch on the function generators. For this experiment, you will not really need to set precise voltages, although you will need to accurately measure the input and output voltages you use. Set function generators to output the voltages so that the sum of the three input voltages will not exceed the output range of our op-amp.
4. Let initially set all the function generators to output 0.0V and measure output voltage V_{out} just at the pin 6 of the LM741 IC and ground of the circuit. Start by increasing the voltage of the first function generator up to 0.141mV Peak and measure V_{out}. Again increase the voltage of the second function generator up to 0.283V peak and measure V_{out}. At last increase the voltage of the third function generator to 1.313V peak and measure V_{out}.
5. Keep the input resistors at the same value, remove and replace feedback resistor R_f, just on the top of IC at pin 6 and pin 2, with 27KΩ and repeat step 3. Is there any increase of the output voltage? Does this increase equal to the calculated gain Av?
6. Remove and replace R_{in2} with 10K, R_{in3} with 56K by keeping R_{in1} to 5.6KΩ and R_f to 27KΩ. Repeat step 3 to 4.
7. Adjust the oscilloscope volt/division according to the input voltages in such way not to overlap the screen. Set time/division to 500uS as the all function generators are set to produce voltages at 500Hz. If you are using two channels oscilloscope you will need to keep one channel to measure the output and use other channel to measure one input after other.

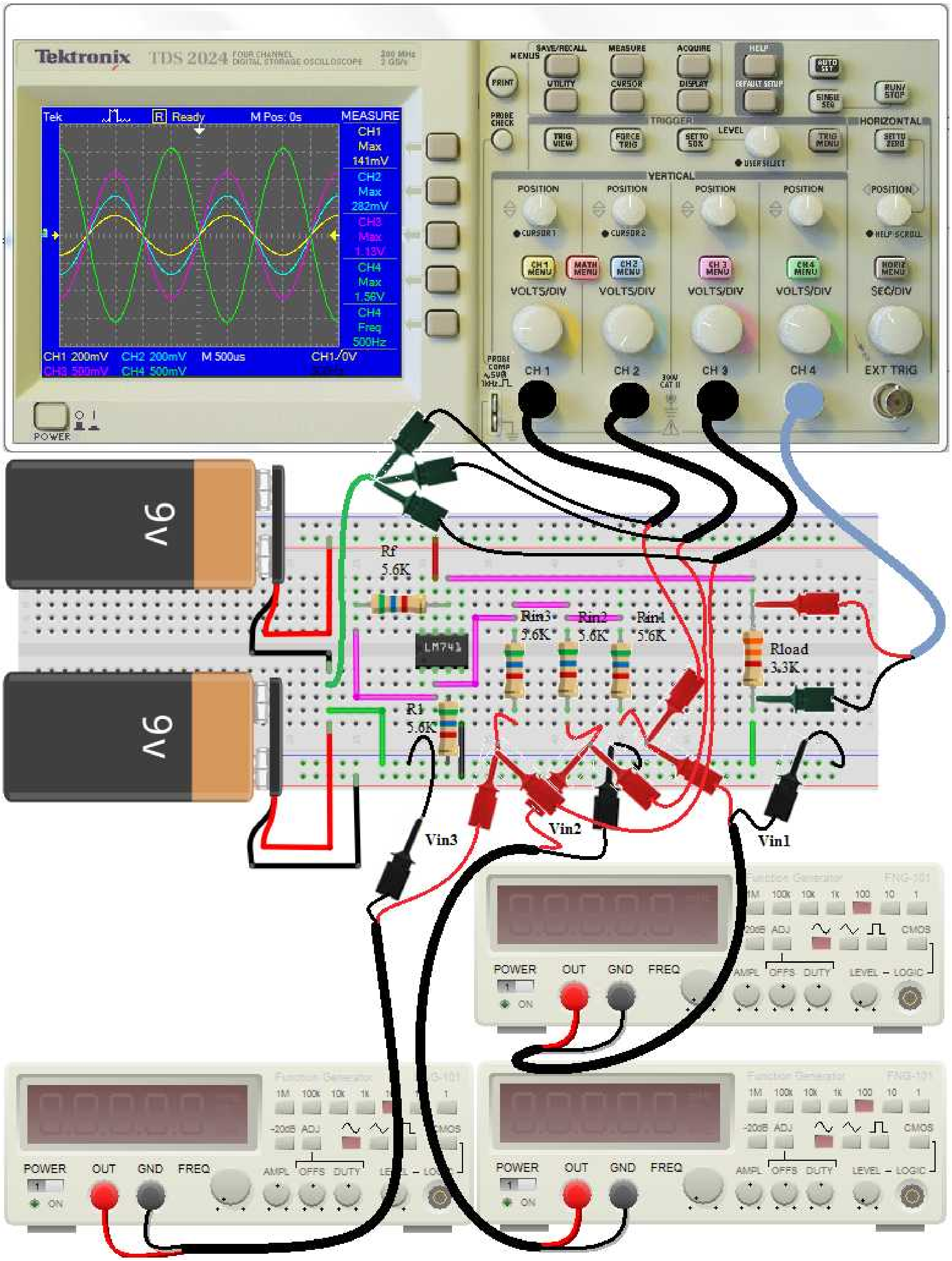


Figure 7-55: Adder (summing) amplifier circuit connection

Applications of Adder (summing) Amplifier

Adder (summing) amplifier finds its main application in audio mixers but it can also be used in other application such as waveform vertical level control.

7. 10. 4. Waveform Vertical-level Control

When you apply an AC signal to one of the summing amplifier inputs, and a DC voltage to the other, the DC voltage is added to the AC signal, changing the DC level of the AC wave. An example application of this is found on the Y shift control on an analog oscilloscope changing the vertical position of the waveform. The typical circuit which can perform such action is shown in figure 7-56.

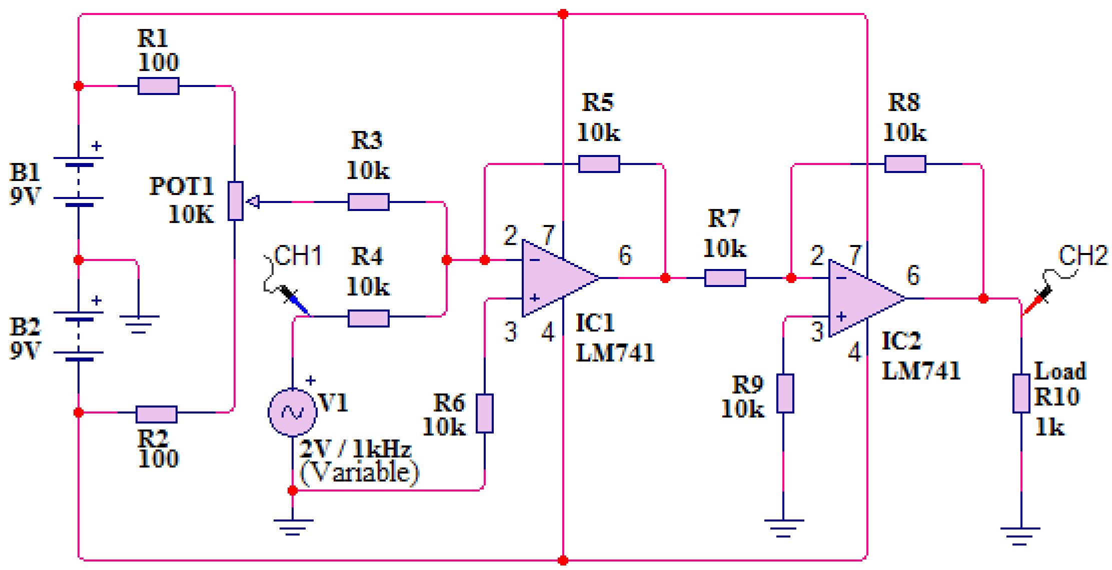


Figure 7-56: Waveform vertical-level control

As the summing amplifier used in stage one (IC1) is based on an inverting amplifier, the signal at the output of stage one will be in anti-phase to the input signal (V_1), so to restore the signal to its original phase a second inverting amplifier (IC2) is used. With R_3 to R_8 all of equal value (precision resistors), the gain of each stage is the unit, and therefore the overall gain, will be 1. This means that no change of the amplitude of the original signal. You can move up and down by using potentiometer POT_1 at which, the output waveform is at X-axis when the potentiometer POT_1 is at 50% of its value. Resistor R_1 and R_2 (precision resistors) limit the max and min value of potentiometer.

Procedure

1. Connect the circuit as illustrated in figure 7-57. Check the connection for error correction. Power on the circuit with 9V dc source.
2. Set the function generator to output 2V rms (2.83x2 peak-to-peak) at any frequency, at 1KHz for this experimental circuit.
3. Set both channel one (CH1) and channel two (CH2) of oscilloscope to 2V volt/div and to 500uS. Set CH1 connected to the input to be zeroed to x-axis. Adjust the potentiometer in clockwise and anticlockwise.

chap 7

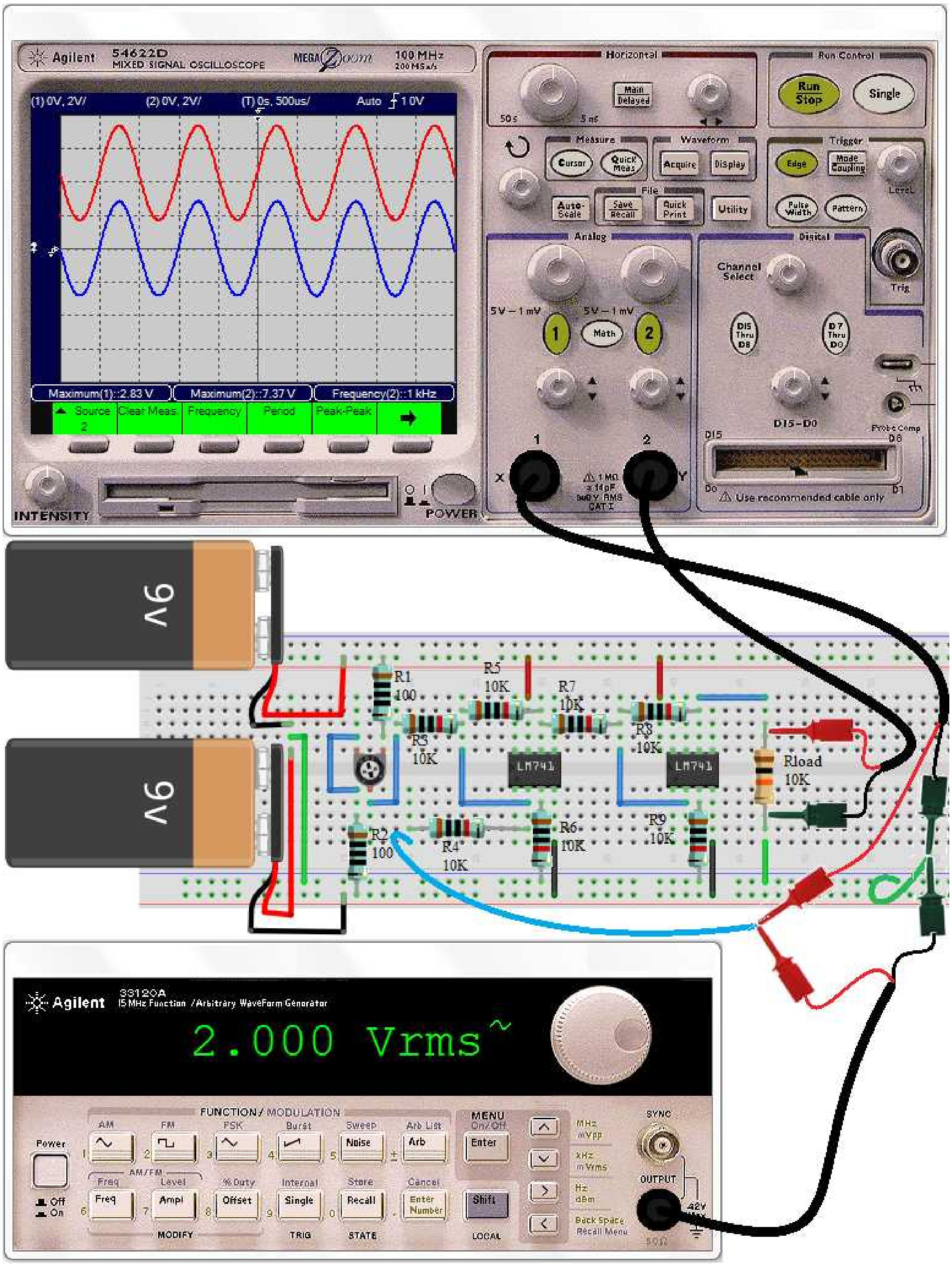


Figure 7-57: Waveform vertical-level control circuit connection

4. As result, as you adjust the potentiometer POT_1 in clockwise the output waveform (CH2) displayed in red shifts upward. Turning the POT_1 in anticlockwise the output waveform shifts downward. Setting the POT_1 in its half value, the output waveform will oscillate at x-axis.

The application example of this circuit is vertical position found on oscilloscope.

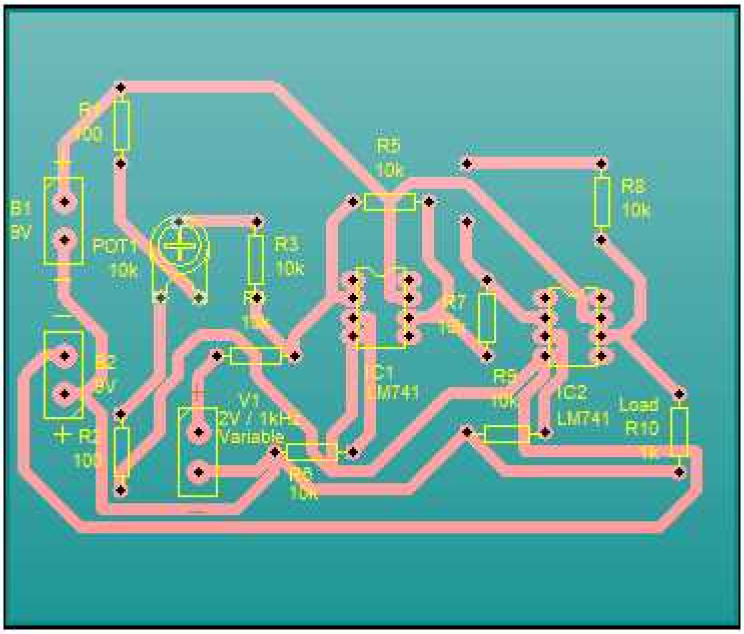


Figure 7-58: Waveform vertical-level control circuit PCB designer artwork

7. 10. 5. Audio mixer

The other practical application of summing operational amplifier is to produce an audio mixer for adding or mixing together individual waveforms (sounds) from different source channels (vocals, piano, drums, etc.) before sending them combined to an audio amplifier.

The circuit above uses a summing amplifier made from an inverting op-amp with input resistors (R_1, R_2 and R_3), which together with the feedback resistor R_4, add the individual signal input voltages at the inverting input of the op-amp. Most often in audio mixers R_1, R_2 and R_3 will usually be the same value. The individual input signals from MIC_1, MIC_2 and MIC_3 can be mixed together by varying their amounts though the potentiometers named Vol_1, Vol_2 and Vol_3 respectively

Since the summing amplifier used in stage one is based on an inverting amplifier, the signal at the output of stage one will be in anti-phase to the input signal, so to restore the signal to its original phase a second inverting amplifier is used. As R_1 to R_6 are all of equal value, the gain of each stage is the unit, and therefore the overall gain, will be 1. The Load resistor R_7 stand for the place of audio amplifier which can be used to increase the resulting signal to a convenient level.

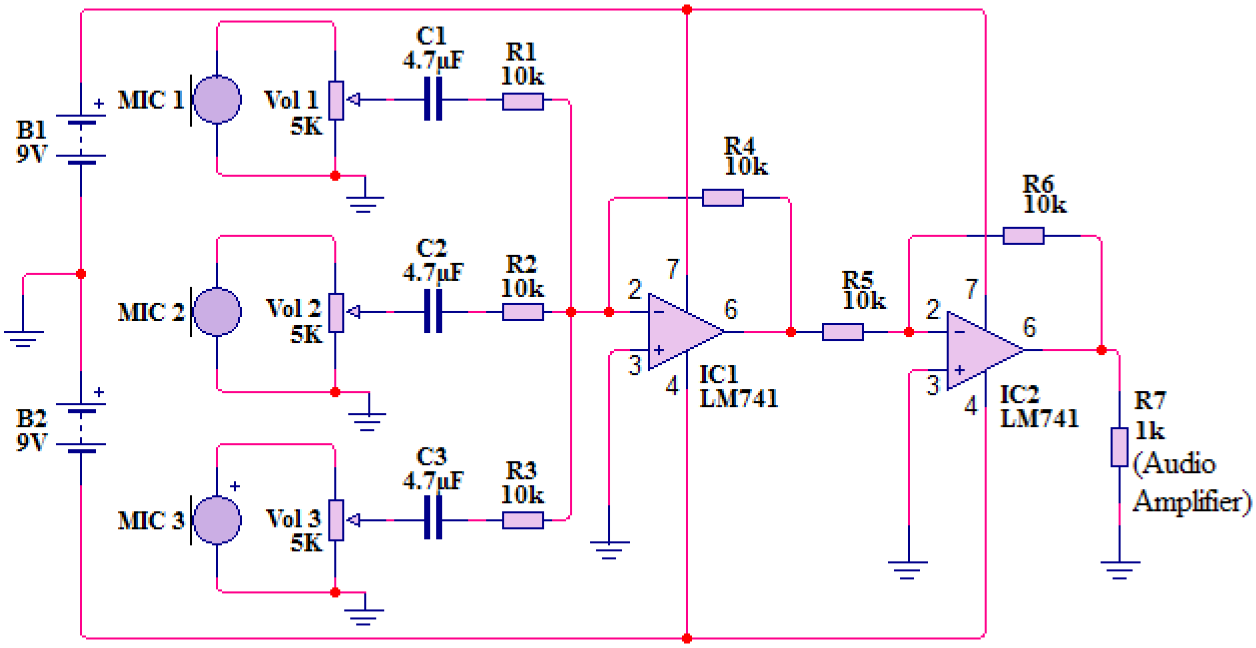


Figure 7-59: Audio mixer summing op-amp

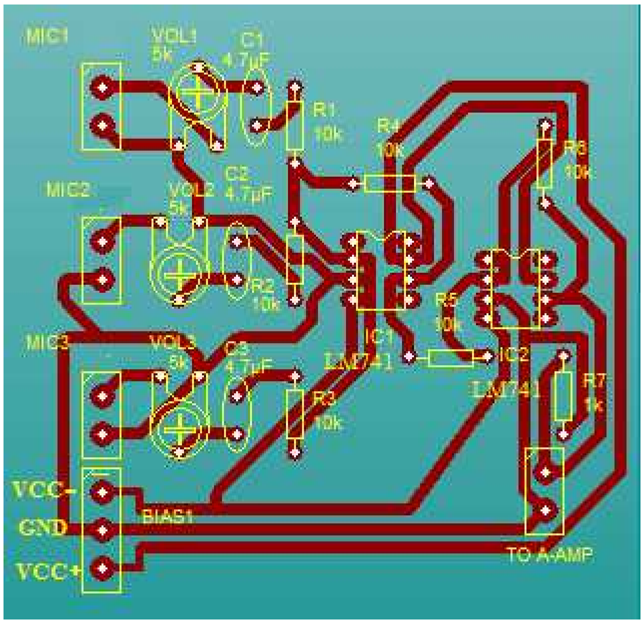


Figure 7-60: Audio mixer summing op-amp artwork, circuit connection

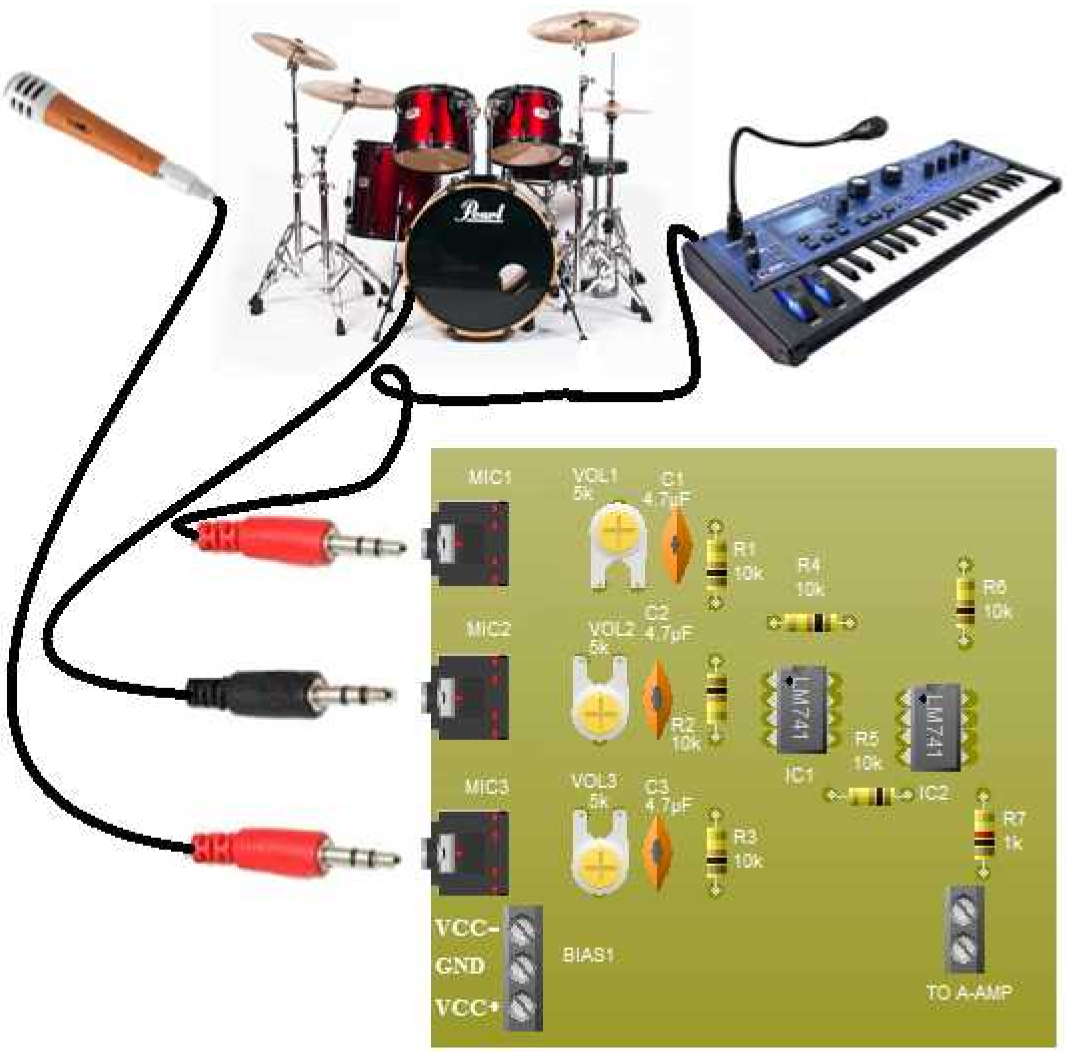


Figure 7-61: Audio mixer summing op-amp, real world circuit connection

Integrator

An op-amp integrator simulates mathematical integration, which is basically a summing process that determines the total area under the curve of a function. Integrator shown in this section is idealized to show basic principles. Practical integrators often have an additional resistor or other circuitry in parallel with the feedback capacitor to prevent saturation.

7. 11. The Op-Amp Integrator

An ideal integrator is shown in figure 7-62. Notice that the feedback element is a capacitor that forms an *RC* circuit with the input resistor. Although a large-value resistor is normally used in parallel with the capacitor to limit the gain at low frequencies, it does not affect the basic operation and is not shown for purpose of this analysis.

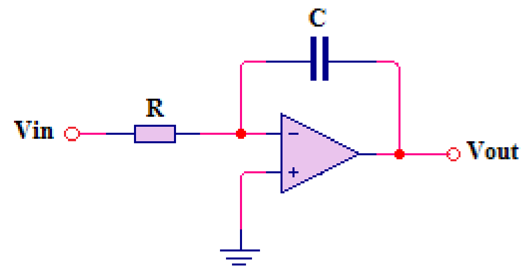


Figure 7-62: An ideal op-amp integrator

How a Capacitor Charges

To understand how the integrator works, it is important to review how a capacitor charges. Recall that the charge Q on a capacitor is proportional to the charging current and the time.

$$Q = I_C t$$

Also, in terms of the voltage, the charge on a capacitor is

$$Q = CV_C$$

From these two relationships, the capacitor voltage can be expressed as

$$V_C = (I_C/C)t$$

This equation has the form of an equation for straight line beginning at zero with a constant slop of I_C/C. Remember from algebra that the general formula for a straight line is $y = mx + b$. In this case, $y = V_C$, $m = I_C/C$, $x = t$, and $b = 0$.

Recall that the capacitor voltage in simple *RC* circuit is not linear but is exponential. This is because the charging current continuously decrease as the capacitor charges causes the rate of change of voltage to continuously decrease. The key thing about using an op-amp with an *RC* circuit to form an integrator is that the capacitor's charging current is made constant, producing a straight line (linear) voltage rather than an exponential voltage. Now let's see why this is true.
In figure 7-63, the inverting input of the op-amp is virtual ground (0), so the voltage across Ri equals V_{in}. Therefore, the input current is

$$I_{in} = V_{in}/R_i$$

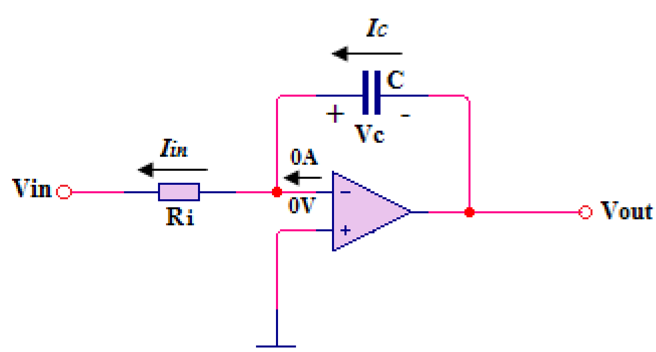


Figure 7-63: Currents in an integrator

If V_{in} is a constant voltage, then I_{in} is also a constant because the inverting input always remain at 0V, keeping a constant voltage across R_i. Because of the very high input impedance of the op-amp, there is negligible current from the inverting input. All of the input currents is through the capacitor, as indicated in figure 7-63.

$$I_C = I_{in}$$

The Capacitor Voltage

Since I_{in} is constant, so is I_C. The constant I_C charges the capacitor linearly and produces a linear voltage across C. The positive side of the capacitor is held at 0V by virtual ground of the op-amp. The voltage on the negative side of the capacitor decreases linearly from zero as the capacitor charges, as shown in figure 7-64. This voltage is called a *negative ramp*.

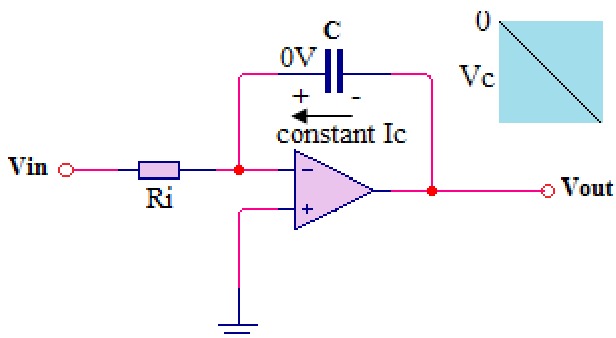


Figure 7-64: A linear ramp voltage is produced across C by the constant charging current

chap 7

The Output Voltage

V_{out} is the same as the voltage on the negative side of the capacitor. When a constant input voltage in the form of a step or pulse (a pulse has a constant amplitude when high) is applied, the output ramp decreases negatively until the op-amp saturates at its maximum negative level. This is indicated in figure 7-65.

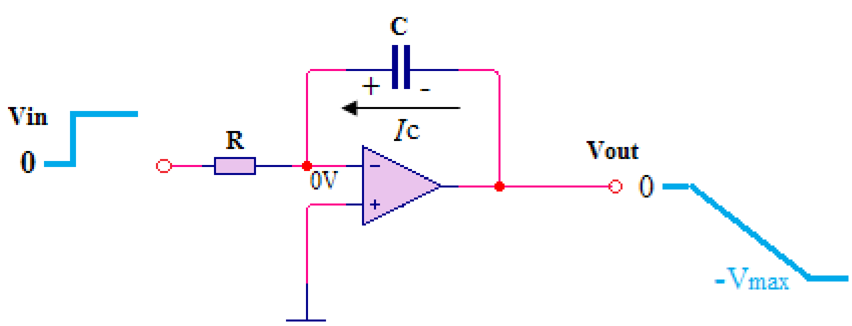


Figure 7-65: A constant input voltage produces a ramp on the output

Rate of Charge of the Output

The rate at which the capacitor charges, and therefore the slope of the output ramp, is set by the ratio I_C/C, as you have seen. Since $I_C = V_{in}/R_{in}$, the rate of change or slope of the integrator's output voltage is $\Delta V_{out}/\Delta t$.

$$\Delta V_{out}/\Delta t = -V_{in}/R_{in}C$$

Experiment 7-12: Op-amp integrator

Determine the rate of change of the output voltage in response to the first input pulse in pulse waveform of input. The output voltage is initially high (+2.5V).

Part list

No	Items	Specifications	Quantity
1	Oscilloscope	Agile oscilloscope 54622D	1
2	Function generator	Agilent function generator 33120A	1
3	Resistors	10kΩ, 100kΩ, 820Ω	3
4	Breadboard	RSR03MB102 breadboard	1
5	Connecting wire	22-gauge solid wire	40cm
6	Op-amp IC	LM741	1
7	Capacitor	0.01µF	1
8	±9V to ±15V dc supply	9V battery	2

chap 7

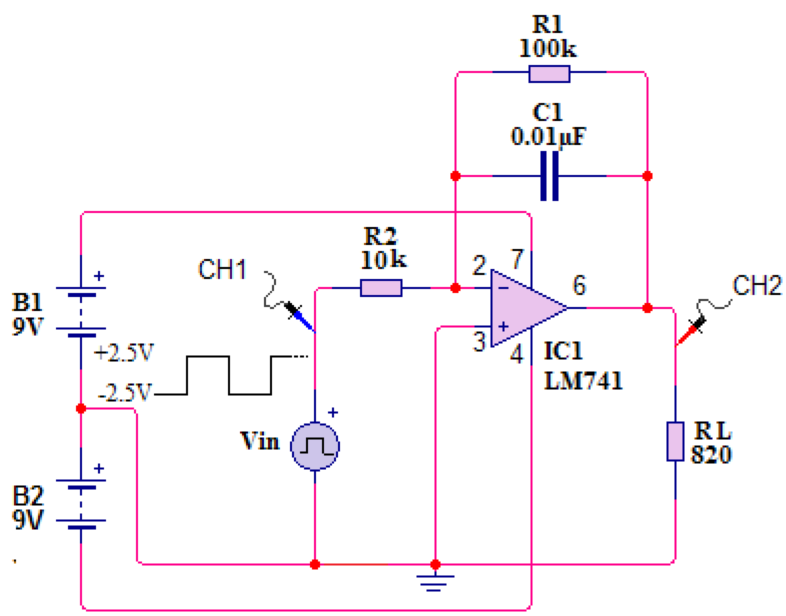


Figure 7-66: Op-amp integrator

- The rate of change of the output voltage during the time that the input is positive (capacitor charging) is

$$\Delta V_{out}/\Delta t = -V_{in}/R_{in}C = 2.5V/(10K\Omega)(0.01uF) = -2.5kV/s = -25mV/\mu S$$

The rate of change of the output during the time that the input is negative (capacitor discharging) is the same as during charging except it is positive.

$$\Delta V_{out}/\Delta t = +V_{in}/R_{in}C = +25mV/\mu S$$

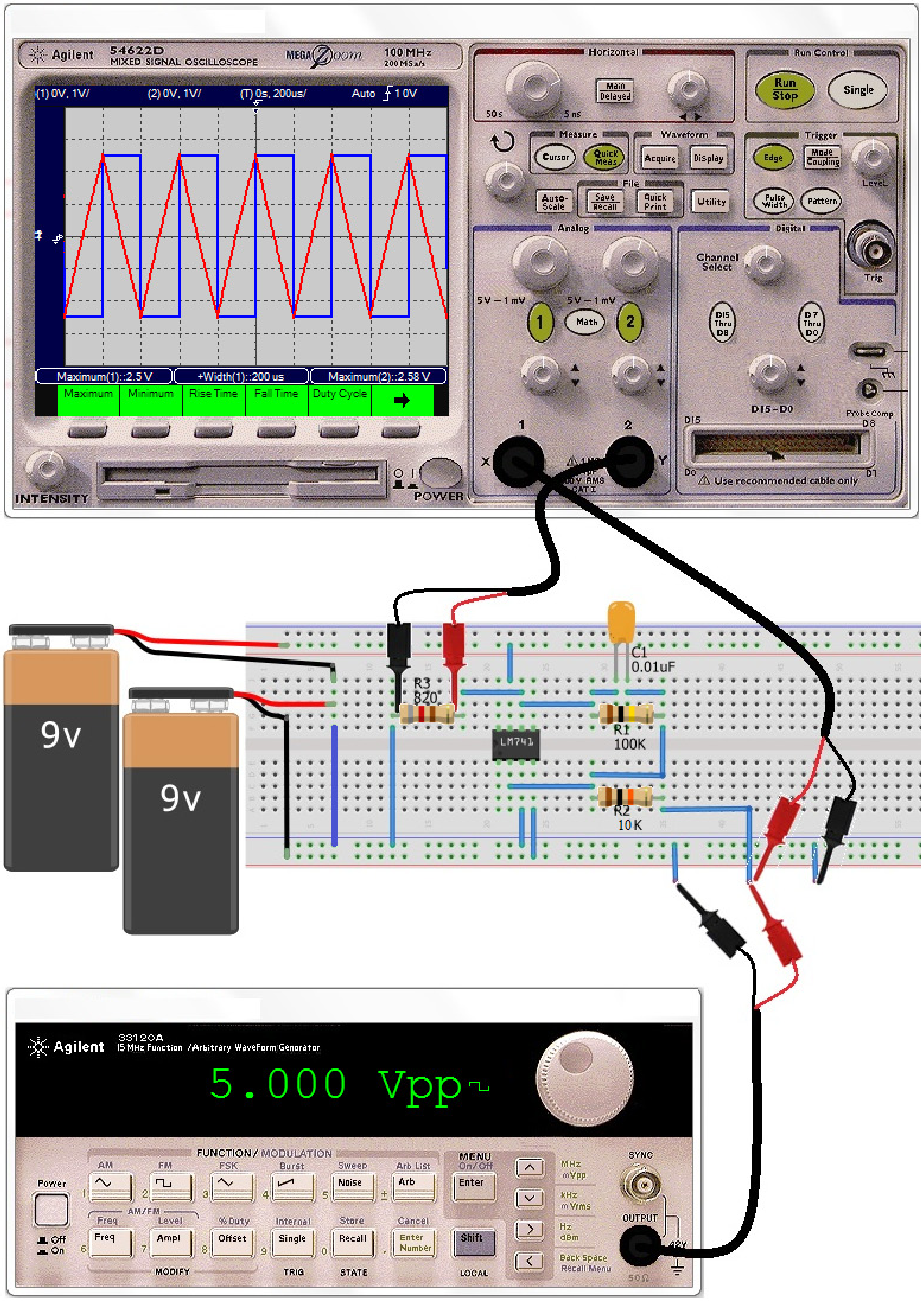


Figure 7-67: Op-amp integrator circuit connection

- When the input is at +2.5V, the output is negative-going ramp. When the input is at –2.5V, the output is a positive-going ramp.

$$\Delta V_{out} = (25mV/\mu S)(200\mu S) = 5V$$

During the time the input is at +2.5V, the output will go from +2.5V to –2.5V. During the time the input voltage is at –2.5V, the output will go from –2.5V to +2.5V. Therefore, the output voltage is a triangular wave with peaks at +2.5V and –2.5V as illustrated in figure 7-67.

Differentiator

An op-amp differentiator simulates mathematical differentiation, which is a process of determining the instantaneous rate of change of a function. Differentiator shown in this section is idealized to show basic principles. Practical differentiator may include a series resistor to reduce high frequency noise.

7. 12. The Op-amp Differentiator

An ideal differentiator is shown in figure 7-68 part (a). Notice how placement of the capacitor and resistor differs from their placement in integrator. The capacitor is now the input element. Differentiator produces an inverted output that is proportional to the rate of change of the input voltage. Although a small-value resistor is normally used in series with the capacitor to limit the gain, it does not affect the basic operation and is not shown for purpose of this analysis.

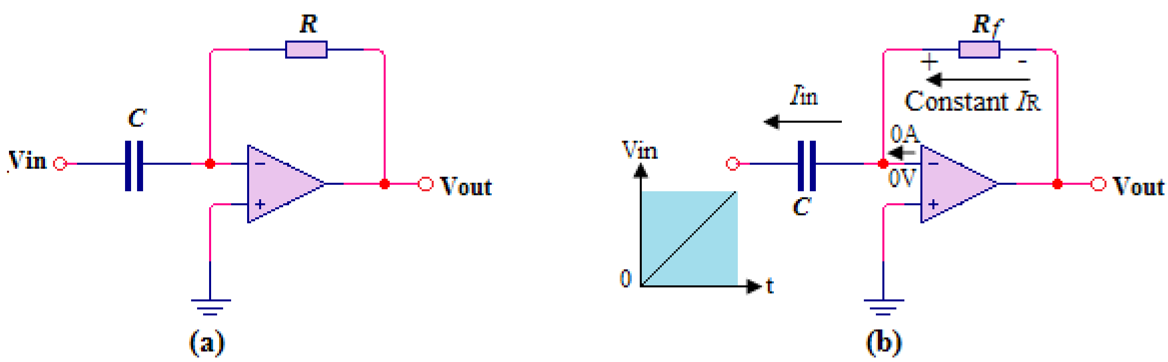


Figure 7-68: Op-amp differentiator; (a) An ideal op-amp differentiator, (b) A differentiator with a ramp input.

To show how the differentiator works, let's apply a positive-going ramp voltage to the input as indicated in figure 7-68 part (b). In this case, $I_C = I_{in}$ and the voltage across the capacitor is equal to V_{in} at all times ($V_C = V_{in}$) because of virtual ground on the inverting input. From the basic formula, $V_C = (I_C/C)t$, the capacitor current is:

$$I_C = (V_C/t)C$$

Since the current at the inverting input is negligible, $I_R = I_C$. Both currents are constant because the slope of the capacitor voltage (V_C/t) is constant. The output voltage is also constant and equal to the voltage across R_f because one side of the feedback resistor is always 0V (virtual ground).

$$V_{out} = I_{R*}R_f = I_{C*}R_f$$

$$V_{out} = -(V_C/t)\, R_{f*}C$$

The output is negative when the input is positive-going ramp and positive when the input is a negative-going ramp. During this positive going slope of the input, the capacitor is charging from the input source and the constant current through the feedback resistor is in the direction toward virtual ground. During the negative slope of the input, the current is in the opposite direction because the capacitor is discharging.

Notice that the term V_C/t is the slope of the input. If the slope increases, V_{out} increases. If the slope decreases, V_{out} decreases. So, the output voltage is proportional to the slope (rate of change) of the input. The constant of the proportionality is the time constant, R_fC.

Experiment 7-13: Op-amp differentiator

Determine the output voltage of the op-amp differentiator for the triangular-wave input. Input voltage is 10V peak-to-peak and at 5KHz.

Part list

No	Items	Specifications	Quantity
1	Oscilloscope	Agile oscilloscope 54622D	1
2	Function generator	Agilent function generator 33120A	1
3	Resistors	2.2kΩ, 1.8kΩ, 100Ω	3
4	Breadboard	RSR03MB102 breadboard	1
5	Connecting wire	22-gauge solid wire	40cm
6	Op-amp IC	LM741	1
7	Capacitor	0.001µF	1
8	±9V to ±15V DC supply	9V battery	2

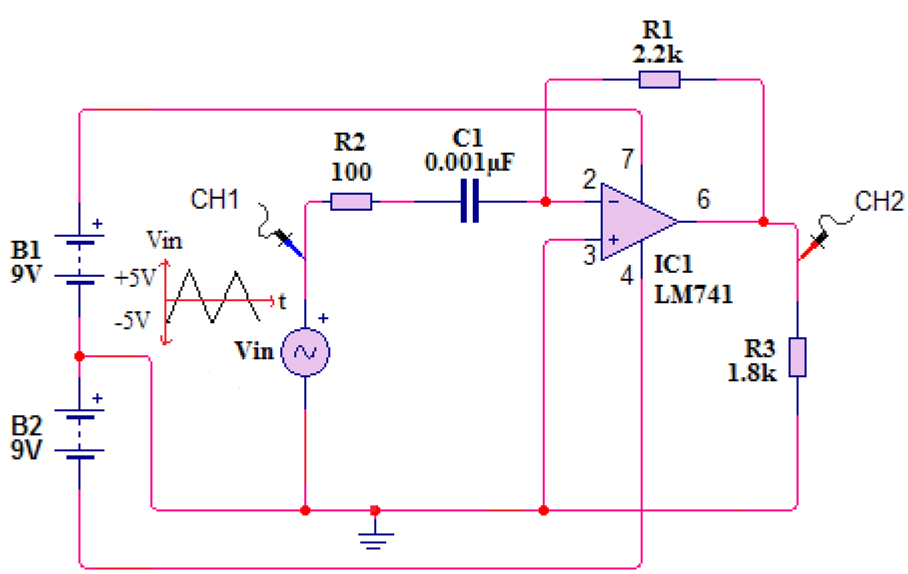


Figure 7-69: Op-amp differentiator

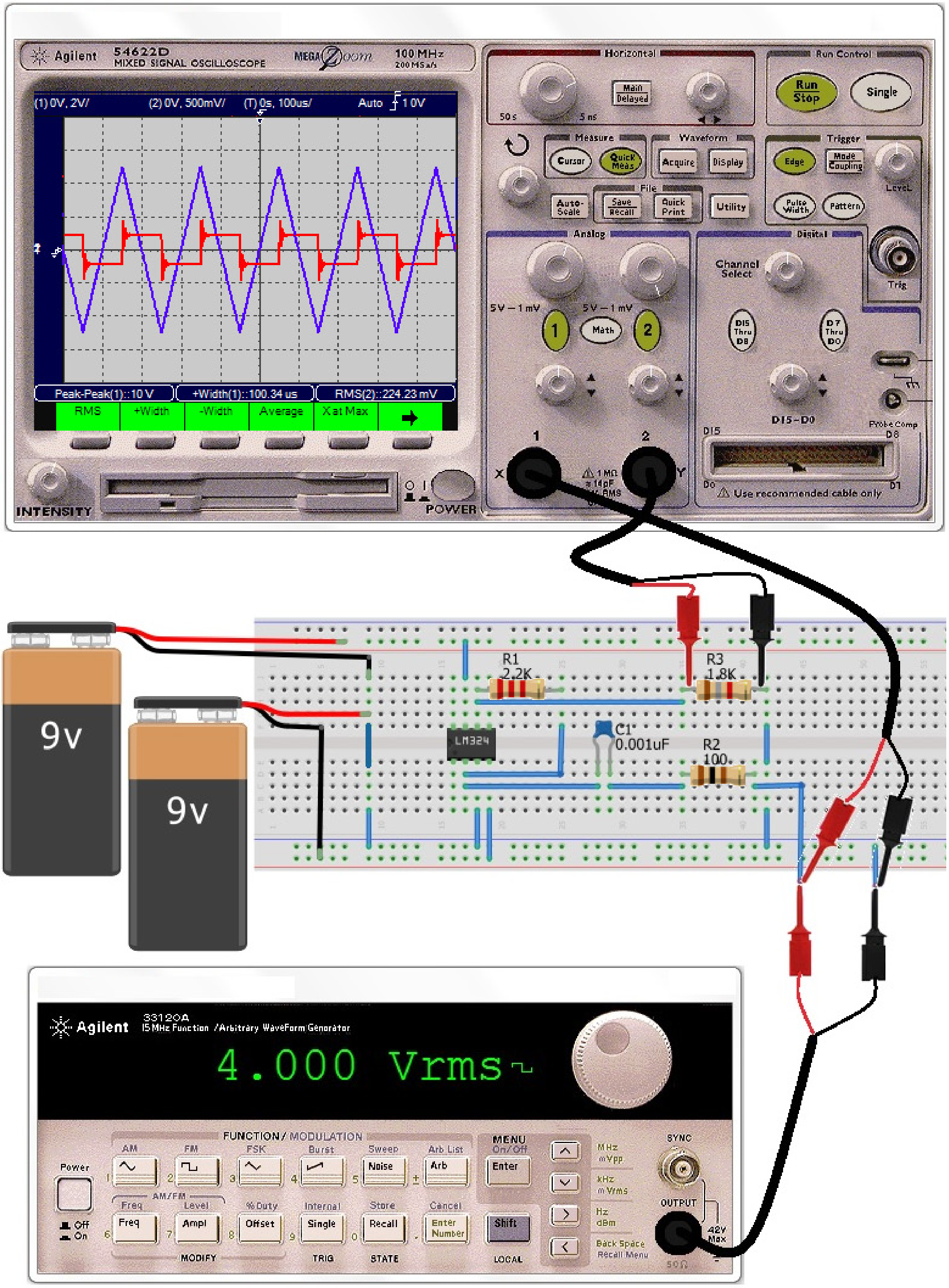


Figure 7-70: Op-amp differentiator circuit connection

- At starting, the input voltage is a positive-going ramp from –5V to +5V (a +10V change) in 100µS. Then it changes to a negative-going ramp ranging from +5V to –5V (a –10V change) in 100µS.

  The time constant is

$$R_f C = (2.2K\Omega)(0.001\mu F) = 2.2\mu S$$

- Determine the slope or rate of change (V_C/t) of the positive-going ramp and calculate the output voltage as follows:

$$V_C/t = 10V/100\mu S = 0.1V/\mu S$$

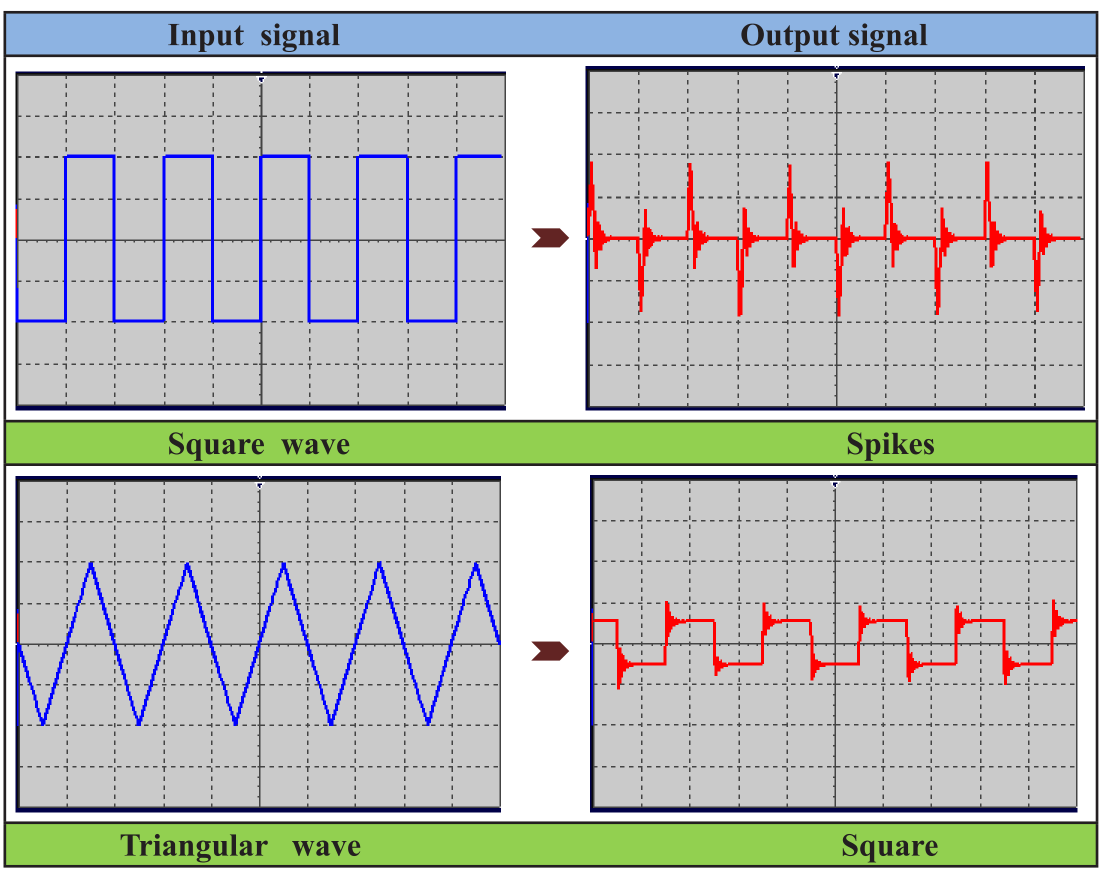
$$V_{out} = -(V_C/t)*R_f C = -(0.1V/\mu S)*2.2\mu S = -0.22V$$

Likewise, the slope of the negative-going ramp is –0.1V/µS. Calculate the output voltage.

$$V_{out} = -(V_C/t)*R_f C = -(-0.1V/\mu S)*2.2\mu S = 0.22V$$

Op-amp Differentiator Waveforms

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



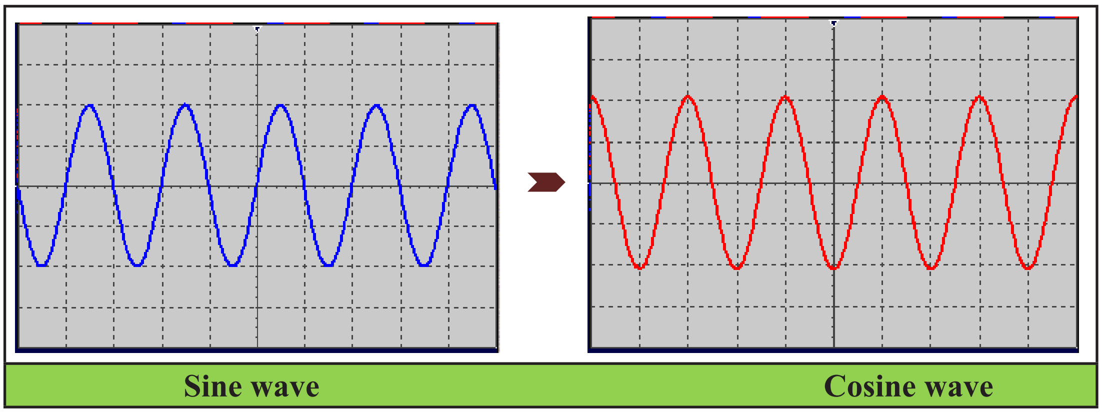


Table 7-4: Op-amp differentiator waveforms

7. 13. A Square Wave Relaxation Oscillator

The basic square-wave relaxation oscillator shown in figure 7-71 is a type of *relaxation oscillator* because its operation is based on the charging and discharging of a capacitor. Notice that the op-amp's inverting input is the capacitor voltage and the noninverting input is a portion of the output fed back through resistor R_2 and R_3. When the circuit is first turned on, the capacitor is uncharged, and thus the inverting input is at 0V. This makes the output a positive maximum, and the capacitor begins to charge toward V_{out} through R_1. When the capacitor voltage reaches a value equal to the feedback voltage on the noninverting input, the op-amp switches to the maximum negative state. At this point, the capacitor begins to discharge from $+V_f$ toward $-V_f$. When the capacitor voltage reaches $-V_f$, the op-amp switches back to the maximum positive state. This action continues to repeat and the square wave is obtained.

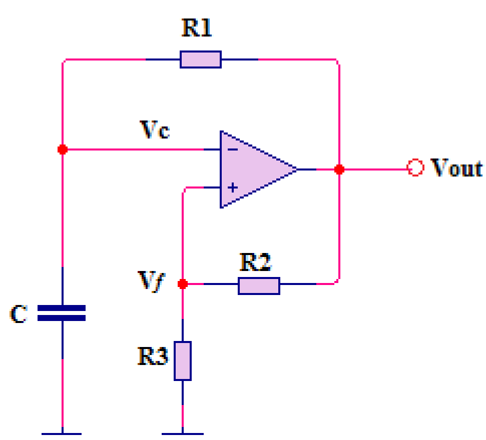


Figure 7-71: A square wave relaxation oscillator

The output period (T) is approximated to: $T=2.2R_1C_1$,
Thus, the frequency (F) is given by: $F=1/2.2R_1C_1$

Experiment 7-14: A square wave relaxation oscillator

Build a relaxation oscillator that outputs a square wave of 45Hz. With oscilloscope measure the frequency due to charging and discharging of capacitor and max and min voltage across the capacitor. Measure also the output frequency just on the load resistor, the maximum and minimum peak voltages. Take R_1 to be 100KΩ and calculate the required capacitor and set feedback resistors to 10KΩ.

From the frequency formula; $F=1/2.2R_1C_1$

$$C_1=1/2.2R_1F=1/2.2*100*10^3*45= 1.01*10^{-7}=0.1\mu F$$

Parts list

No	Items	Specifications	Quantity
1	Oscilloscope	Agile oscilloscope 54622D	1
2	Resistors	100KΩ, 2x10KΩ, 1.5KΩ	3
3	Breadboard	RSR03MB102 breadboard	1
4	Connecting wire	22-gauge solid wire	40cm
5	Op-amp IC	LM741	1
6	Capacitor	0.1μF	1
7	±9V to ±15V DC supply	9V battery or 1xdual polarity dc supply	2

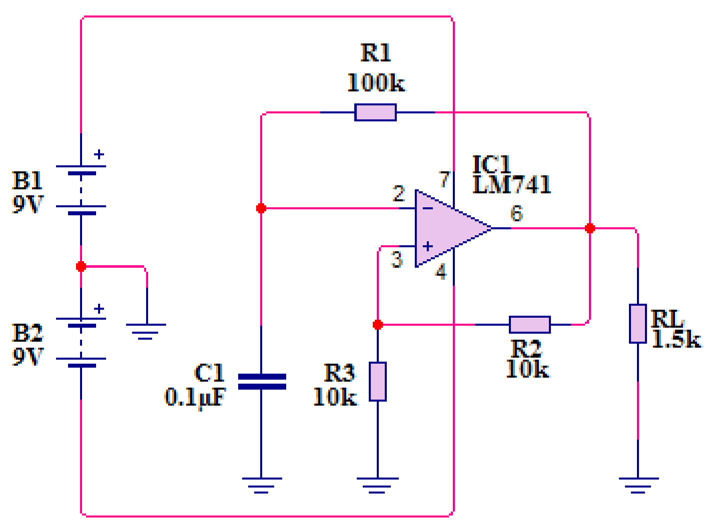


Figure 7-72: A square wave relaxation oscillator experimental circuit

If we take output of a Schmitt trigger back to its inverting input through a RC low-pass filter, we get a circuit whose output switches back and forth between the op-amp's two saturation limits. This is a relaxation oscillator. The operation of the circuit of the figure 7-72 is that the op-amp's output charges the capacitor C_1 via the resistor R_1. Because the capacitor's voltage is monitored by the op-amp's inverting input, every time it charges up to a trigger threshold, the op-amp output changes sign, and the capacitor voltage then begins to "relax" toward the opposite output saturation limit. The trigger threshold voltage at the op-amp's noninverting input has also changed sign, however, so that the op-amp output again changes state as the capacitor voltage reaches this opposite threshold; the process is then repeated.

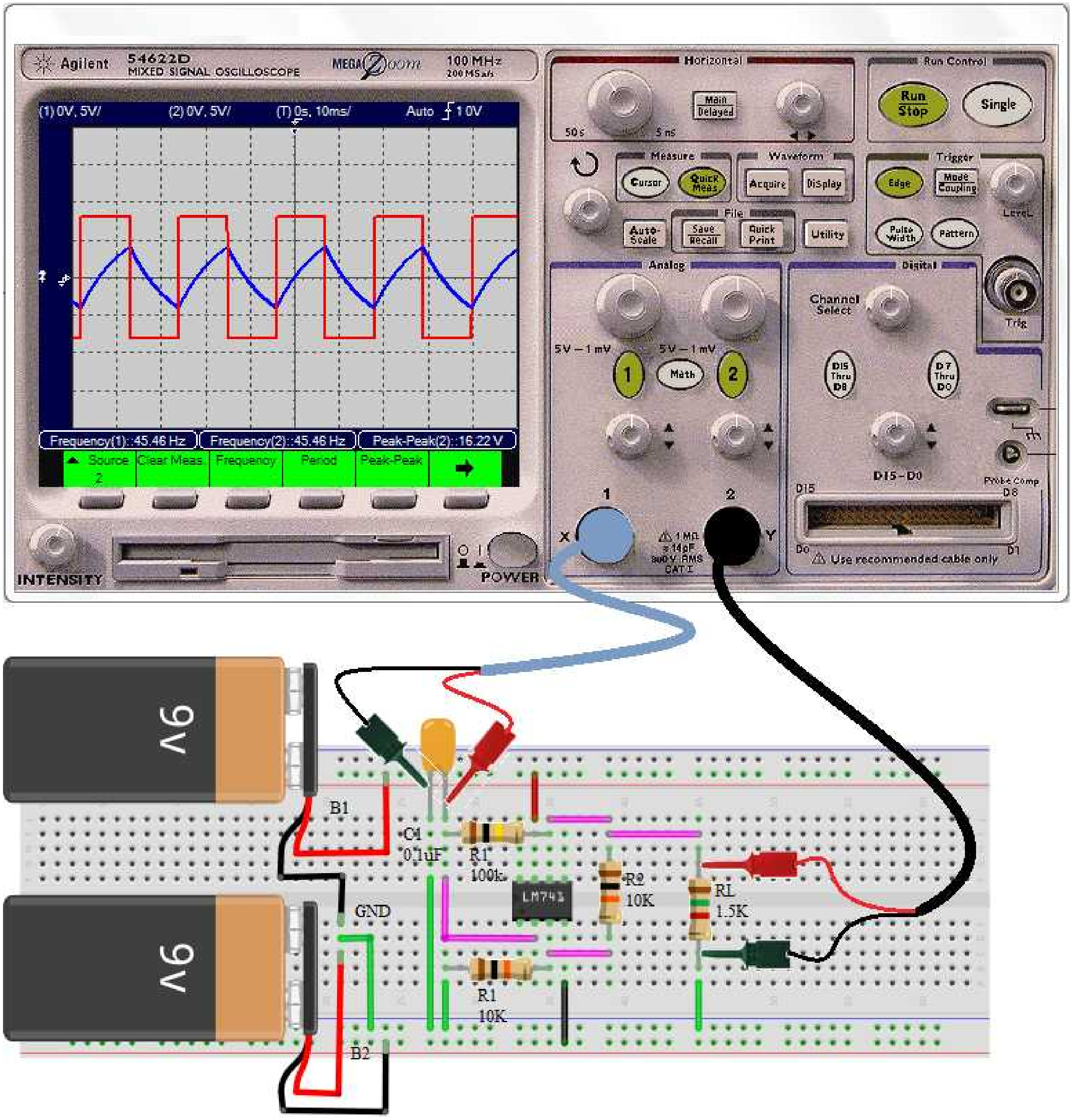


Figure 7-73: A square wave relaxation oscillator experimental circuit connection

Procedure

1. Connect the circuit as illustrated in figure 7-73. Locate the pins of the op-amp IC and make sure that pin 7 is connected to +VCC and pin 4 to –VCC of dc supply.
2. Power on the circuit and set both channel one and channel two to 5V volt/div and to 10mSec time/division.
3. Connect channel one (CH1) terminal of the capacitor C_1 and channel two (CH2) on the terminal of load resistor R_L.
4. Measure, using oscilloscope, the maximum ($+V_{th}$) and the minimum ($-V_{th}$) voltages and

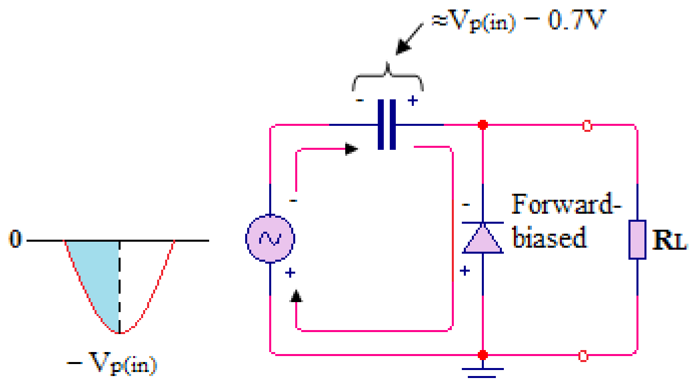
the frequency just on terminal of capacitor C_1. Measure also the maximum and minimum output peak voltages and the frequency on the load resistor R_L. Compare measured result to already calculated frequency.

Active Diode Circuits

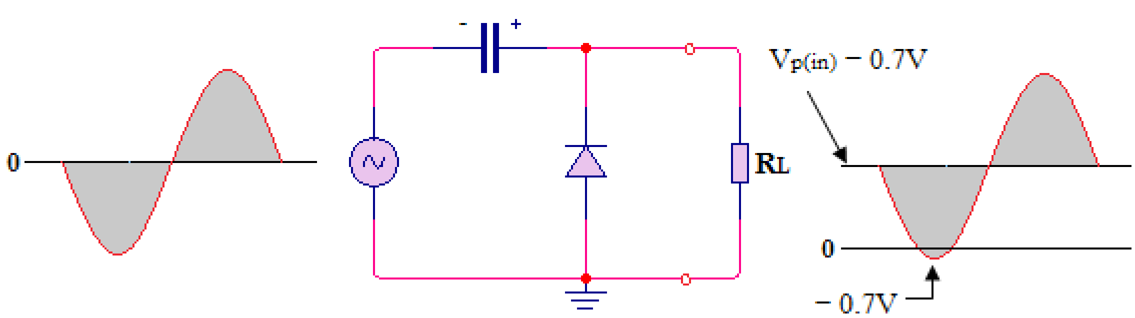
The term *active* in relation to a circuit indicates that a gain element is used, in this case, an op-amp. Circuits using both op-amp and diodes are introduced, including the clamper and the limiter.

The Basic Diode Clamper:

The clamping circuit or *clamper* is used to add a dc level to a signal voltage. Clampers are often referred to as *dc restores* because they are used to restore a dc level to a signal that has been processed through a capacitive coupled amplifiers. To illustrate the basic principle of clamper operation, figure 7-74 shows a simple passive diode clamping circuit that adds a positive dc level to the input signal. To understand the operation of this circuit, start with the first negative half-cycle of the input voltage. When the input initially goes negative, the diode is forward-biased, allowing the capacitor to charge to *near* the peak of the input, as shown in figure 7-74 part (b). Just past the negative peak, the diode becomes reverse-biased because the cathode is held to $V_{p(in)}-0.7V$ by the charge on the capacitor.



(a)



(b)

Figure 7-74: Positive clamping operation with passive clamper

The capacitor can discharge only through R_L. Thus, from the peak of one negative half-cycle to the next, the capacitor discharges very little. The amount that is discharged depends on the value of R_L and the period of the input signal. For good clamping action, the *RC* time constant should be at least ten times the period of the input. The net effect of the clamping action is that the capacitor retains a charge approximately equal to the peak value of the input less the diode drop. The dc voltage of the capacitor adds to the input voltage by superposition, as shown in figure 7-74 part (b). If the diode is turned around, a negative dc voltage is added to the input signal, as shown in figure 7-75.

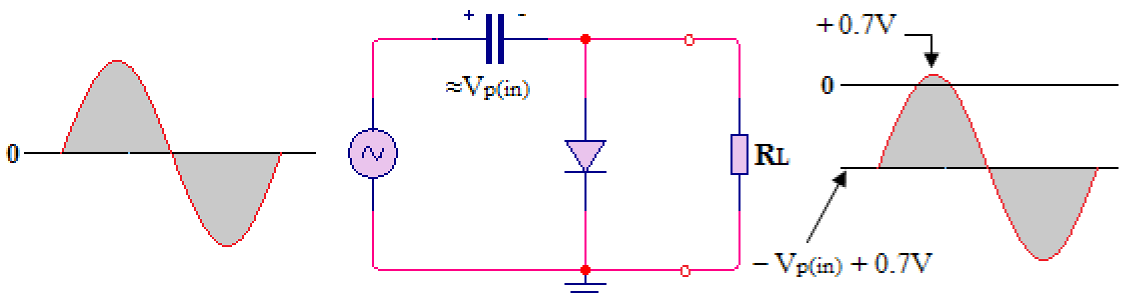


Figure 7-75: Negative clamper

7. 14. An Active Clamping Circuit

A positive clamping with an op-amp and a diode is shown in figure 7-76. This circuit overcomes a couple of disadvantage of the passive clamper. The use of the op-amp eliminates the 0.7V peak found in the passive clamper output, and it prevents loading the input source when the diode is forward-biased.

chap 7

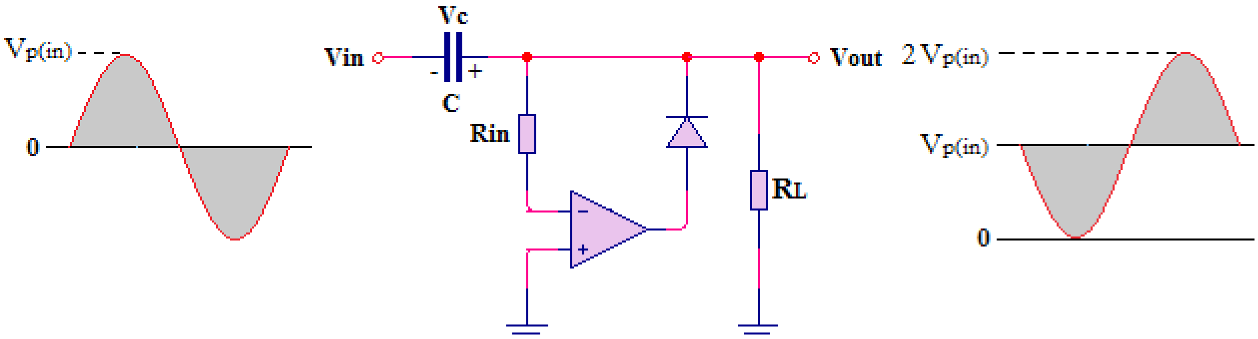


Figure 7-76: An active clamping circuit and its operation

The operation is as follow. On the first negative half-cycle of the input voltage, V_{in}, the differential input is positive, which produces a positive output voltage. Because of the feedback loop, the positive op-amp output voltage forward-biases the diode, allowing the capacitor to quickly charge. The maximum voltage across the capacitor occurs at the negative peak of the input with the polarity shown in figure 7-76. This capacitor voltage adds to the input voltage so that the minimum peak of the output voltage, V_{out}, is at 0V as indicated. During the time between the minimum output peaks of V_{out} and after the capacitor is charged. The differential input voltage to the op-amp becomes negative (inverting input more positive than non-inverting input). As a result, the output of the op-amp becomes negative and reverse-biases the diode, thus breaking the feedback path. The only change in the capacitor voltage during this time is due to a very small discharge through R_L. At each minimum peak of the signal, the diode is forward-biased for a very short time to replenish the voltage across the capacitor.

The positive clamper can be converted to a negative clamper by reversing the diode. In this case, the output waveform would occur below 0V with its maximum peaks at zero, as illustrated in figure 7-77 part (a). Also, the clamping level can be changed to a value other than 0V by connecting a reference voltage source at the + input (non-inverting) of the op-amp, as shown in figure 7-77 part (b).

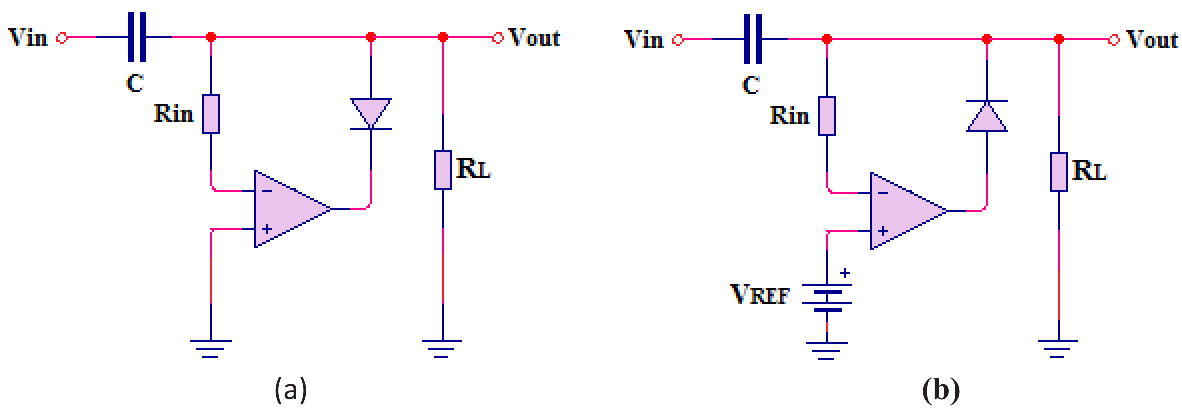


Figure 7-77: Other active clamper configurations. (a) Negative clamper, (b) Nonzero reference clamper

Experiment 7-15: Active Clamper

Design a circuit based on an active clamper which can shift the input AC signal up (positive clamping) and down (negative clamping) of the x-axis or ground level. Use a dc voltage source to shift the signal at 1.5V from the ground level.

Parts and materials

No	Items	Specifications	Quantity
1	Oscilloscope	Agile oscilloscope 54622D	1
2	2V, 500Hz voltage source	Agilent function generator 33120A	1
3	2.7KΩ and 10KΩ resistors	2.7kΩ (R_{in}), 10kΩ (R_{Load}) 100Ω	2
4	Breadboard	RSR 03MB102 breadboard	1
5	Connecting wire	22-gauge solid wire	40cm
6	Op-amp IC	LM741	1
7	4.7uF Capacitor	4.7µF capacitor	1
8	±9V to ±15V DC supply	9V battery	2
9	1.5V dc source	1.5V cell battery	1
10	Small signal diode	1N4148 small signal diode	1

Procedure

1. Connect the circuit as illustrated in figure 7-79.
2. Set oscilloscope's CH1 and CH2 volt/Div to 2V and time/Div to 1mS. Set the function generator to output 2V rms (2.82V peak) at 500Hz. Make sure that both CH1 and Ch2 of the oscilloscope are zeroed just to x-axis.
3. Now turn on power to your experimental circuit and measure the input voltage (just directly on the terminal of the output of function generator), V_{in}, peak-to-peak, max and min. Measure also the output voltage, V_{out}, peak-to-peak, max and min on the terminal of load

resistor R_L and enter these values in the appropriate text box in the table 7-5.

4. Switch the power off and insert the cell battery 1.5V in between non-inverting terminal of the op-amp and ground. Connect the positive of the battery at non-inverting of the op-amp and negative of the battery at ground. Repeat step 3.

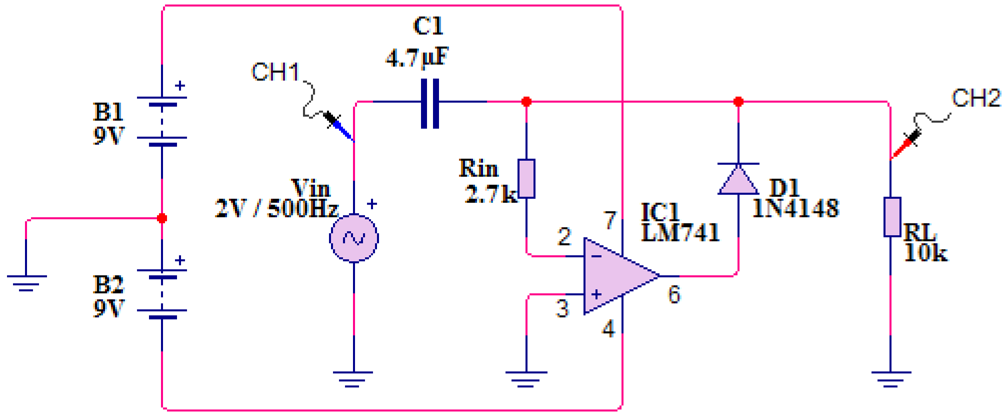


Figure 7-78: An active clamper circuit

5. Switch the power off and change the position of diode (cathode connected at pin 6). Repeat step 3.
6. Switch the power off and change the position of the cell battery (negative of the cell connected to non-inverting of op-amp and the negative of the cell to ground). Repeat step 3.
7. Switch the power off and remove the cell battery and replace it with a jumper wire by connecting non-inverting terminal of the op-amp directly to the ground. Repeat step 3.
8. With the results from step 3 to step 7 name the circuit for each connection. Does the result reflect to the expected theoretical that for unbiased positive clamping the minimum peak is at 0V while for the negative clamping the positive peak is at 0V? This type of active clamping are said to be precise compared to passive clamping; are the above results in the same agreement?

Steps		1 - 3	4-3	5-3	6-3	7-3
Input voltage (V_{in})	V_{p-p}					
	V_{max}					
	V_{min}					
Output voltage (V_{out})	V_{p-p}					
	V_{max}					
	V_{min}					

Table 7-5: Measured values for experimental circuit of an active clamper

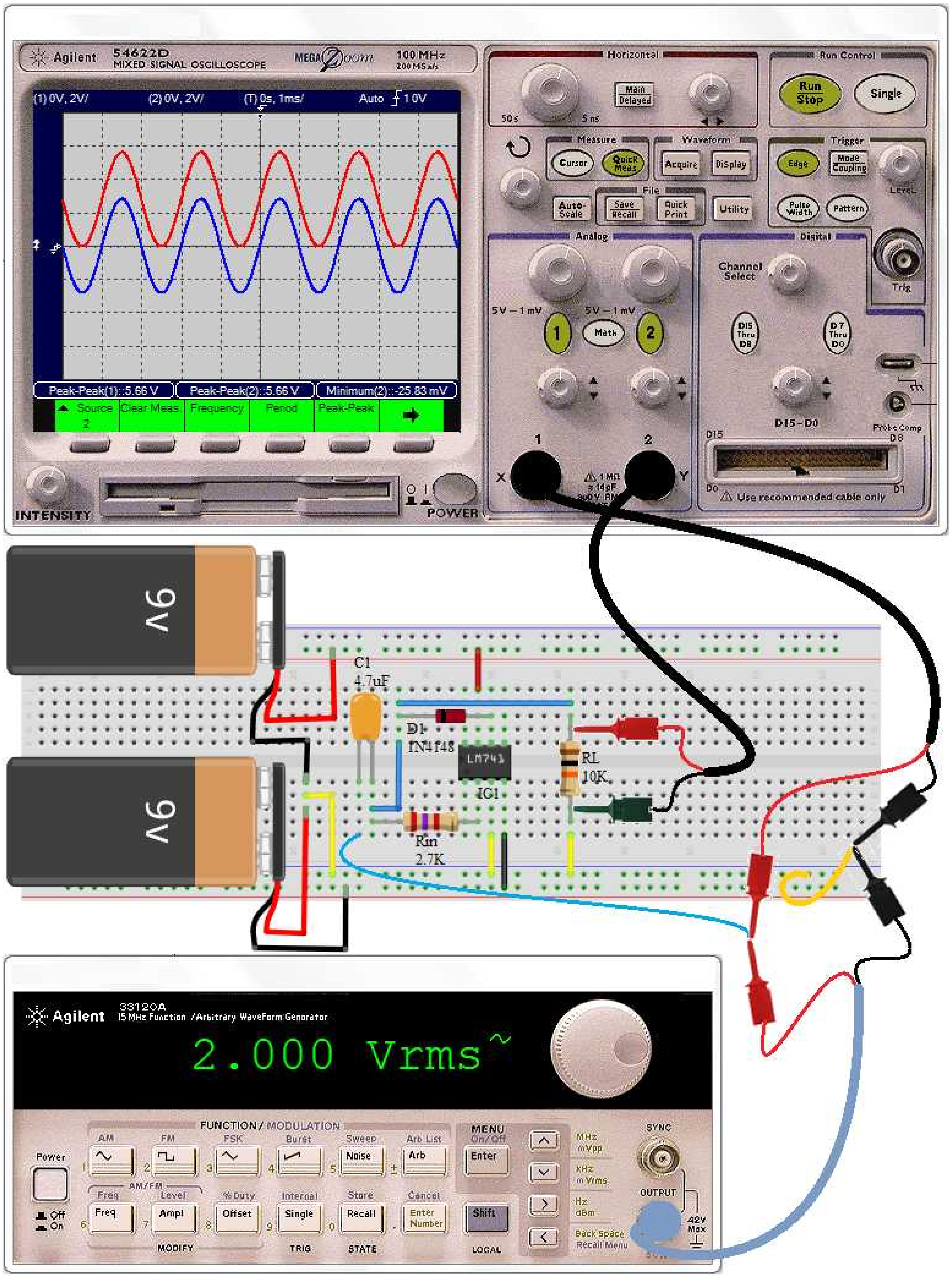


Figure 7-79: An active clamper circuit connection

chap 7

A Clamper Application

A clamping circuit is often used in television receivers as a dc restorer. The incoming composite video signal is normally processed through capacitively coupled amplifiers that eliminate the dc component in the signal. This results in loss of the black and white reference levels and the blanking level. Before being applied to the picture tube, these reference levels and the blanking level. Before being applied to the picture tube, these reference levels must be restored.

Limiting Circuits

The basic diode limiter: Diode limiters (also called clippers) cut off or limit voltage above or below a specified voltage levels. To understand how a limiter works, let's first look at a simple passive positive limiter such as the one shown in figure 7-80 part (a). When the input signal is positive, the diode is reverse-biased and the output voltage looks like the input voltage. When the input signal is negative, the diode is forward-biased and the output is limited to −0.7V, which is diode drop. If you turn the diode around, you get a negative limiter, as shown in figure 7-80 part (b). To change the limiter level, a reference voltage source can be used in series with the diode or a zener diode can be used in place of the rectifier diode.

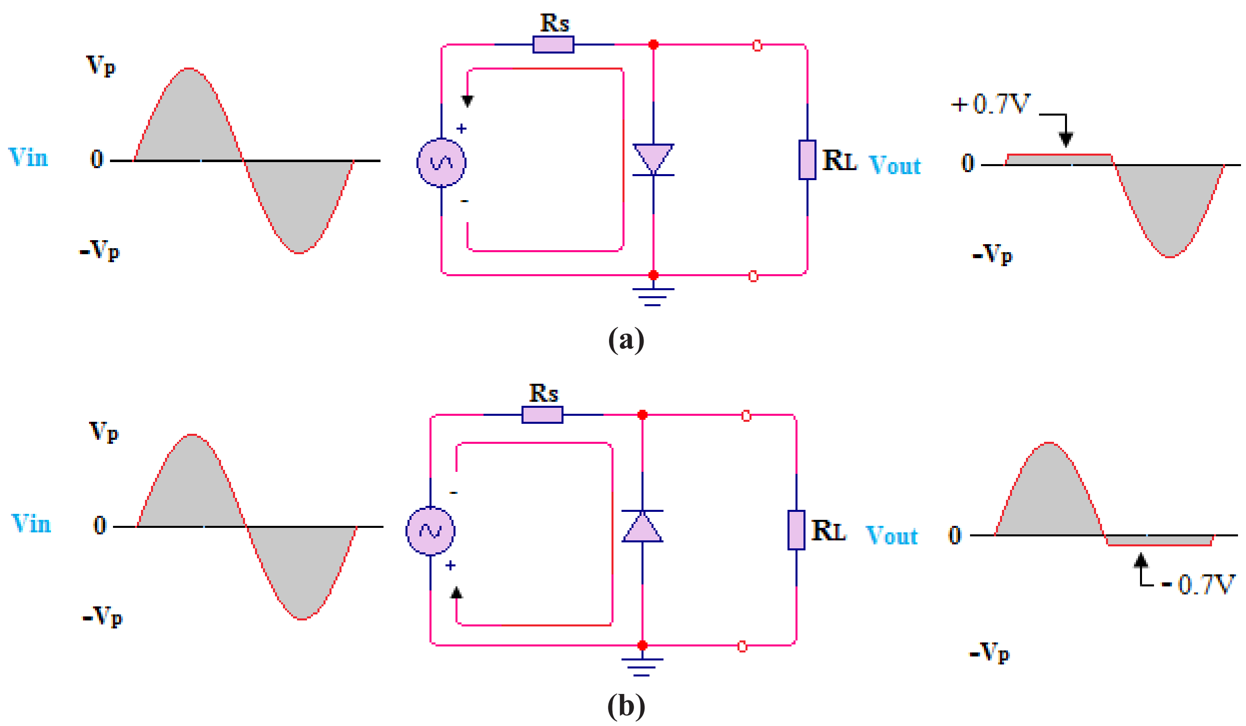


Figure 7-80: Diode limiters. (a) Limiting of positive alternation, (b) Limiting of negative alternation

7. 15. Active Limiting Circuit

One type of op-amp limiting circuit that uses an op-amp and diode is shown in figure 7-81. The operation is as follows, assuming negligible output loading. When the input voltage, V_{in}, is less than the reference voltage, V_{REF}, the op-amp differential input voltage is positive. This produces a positive voltage at the op-amp output that forward-biases the diode. When the diode is forward-

chap 7

biased, the op-amp operates as a voltage follower and the output voltage, V_{out}, is limited to V_{REF} so $V_{out} = V_{REF}$. When the input voltage, V_{in}, is greater than V_{REF}, the op-amp differential voltage is negative. This produces a negative voltage at the op-amp that reverse-biases the diode. With the diode effectively open, the input voltage is coupled directly to the output through R_{in} so that $V_{out} = V_{in}$.

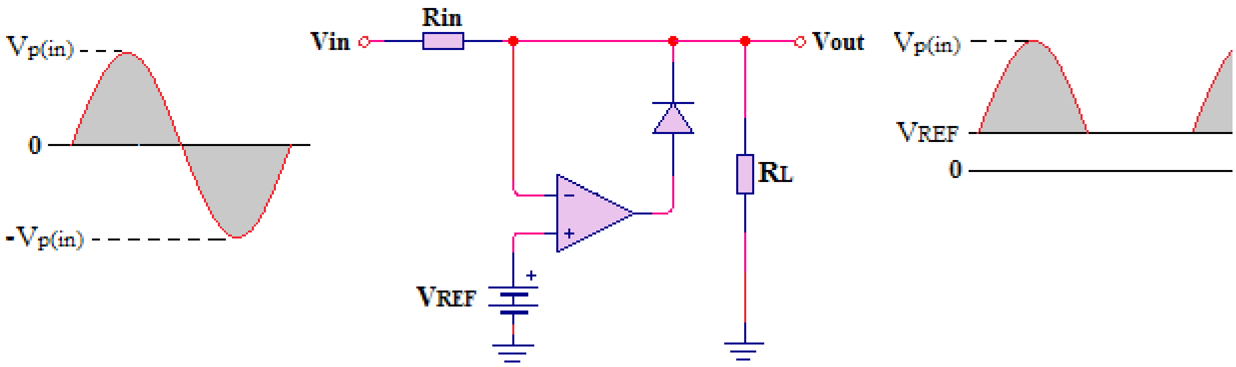


Figure 7-81: An active limiting circuit

Another example of an active limiter uses two diodes to limit the output voltage both positively and negatively as shown in figure 7-82. The limiting voltage is set by the zener diodes connected in feedback loop of an inverting amplifier to a value $\pm(V_Z+0.7V)$. Of course, when the input voltage is so small that of the limiting voltage is not reached, one of the zener diodes is reverse-biased and acts as an open, so the output of the op-amp is linear and equal to $V_{out} = (R_f/R_{in})V_{in}$. When the output reaches $\pm(V_Z+0.7V)$, one of the zeners goes into reverse breakdown and the other is forward-biased.

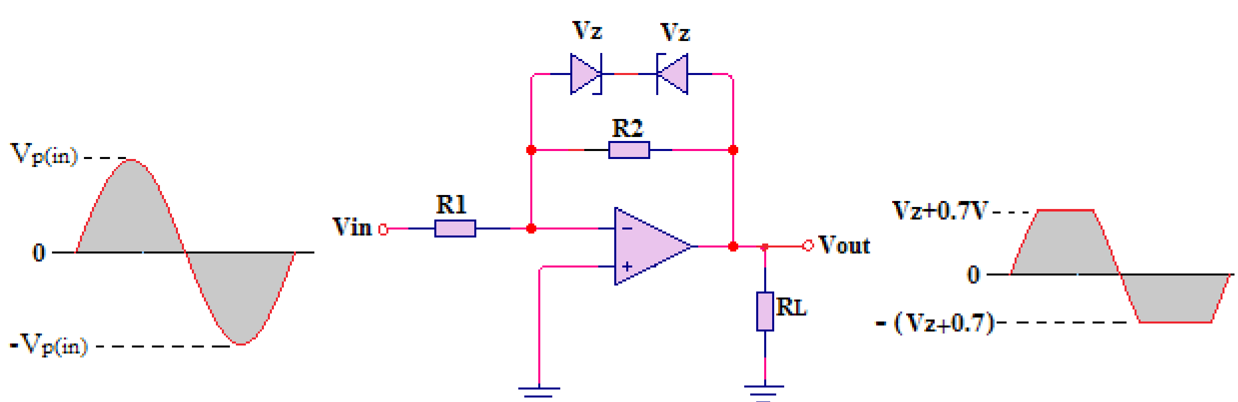


Figure 7-82: Limiting circuit using zener diodes in an inverting op-amp configuration

Experiment 7-16: Limiter

Design an active limiting circuit which limits the positive half-cycle to 0V and to 1.5V, negative half-cycle limited to 0V and to −1.5V. Use 2.7V and 3.3V zener diodes to limit both positive and negative half-cycles to form a dual polarity.

Parts and materials

No	Items	Specifications	Quantity
1	Oscilloscope	Agile oscilloscope 54622D	1

2	2.5V, 50Hz voltage source	Agilent function generator 33120A	1
3	560Ω and 10KΩ resistors	560Ω (R_{in}), 10kΩ (R_{Load}) 100Ω	2
4	Breadboard	RSR 03MB102 breadboard	1
5	Connecting wire	22-gauge solid wire	40cm
6	IC op-amp	LM741	1
7	±9V to ±15V DC supply	9V battery	2
8	1.5V dc source	1.5V cell battery	1
9	2.7V and 3.3V zener diodes	BZX55C, 2v7 and BZX55C, 3V3	2

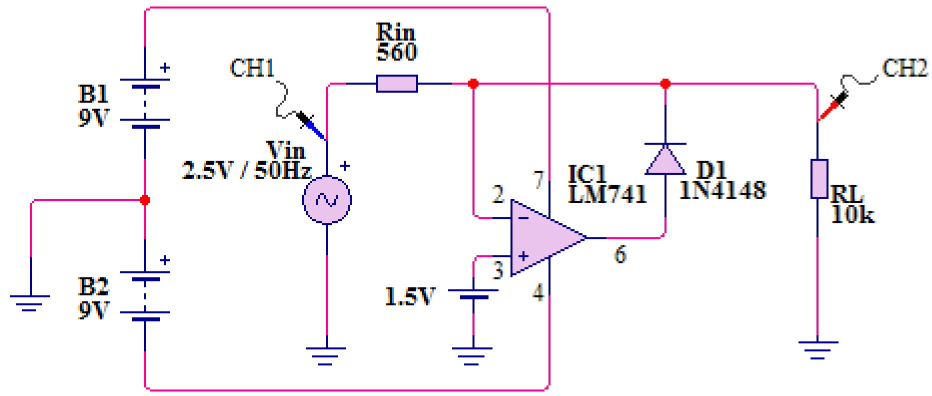


Figure 7-83: Active limiting experimental circuit

Procedure

1. Connect the circuit as illustrated in figure 7-84.
2. Set oscilloscope's CH1 volt/Div to 2V, CH2 volt/Div to 1V and time/Div to 10mS. Set the function generator to output 2.5V rms (3.53V peak) at 50Hz. Make sure that both CH1 and Ch2 of the oscilloscope are zeroed to x-axis.
3. Now turn on power to your experimental circuit and measure the input voltage (just directly on the terminal of the output of function generator), V_{in}, peak-to-peak, max and min. Measure also the output voltage, V_{out}, peak-to-peak, max and min on the terminal of load resistor R_L and enter these values in the appropriate text box in the table 7-6.
4. Switch the power off and change the position of the cell battery (negative of the cell connected to non-inverting pin 3 of op-amp and the negative of the cell to ground). Repeat step 3.
5. Switch the power off and change the position of diode (cathode connected at pin 6). Repeat step 3.
6. Switch the power off and change the position of the cell battery (negative of the cell connected to non-inverting pin 3 of op-amp and the negative of the cell to ground). Repeat step 3.
7. Switch the power off and remove the cell battery and replace it with a jumper wire by connecting non-inverting terminal pin 3 of the op-amp directly to the ground. Repeat step 3.

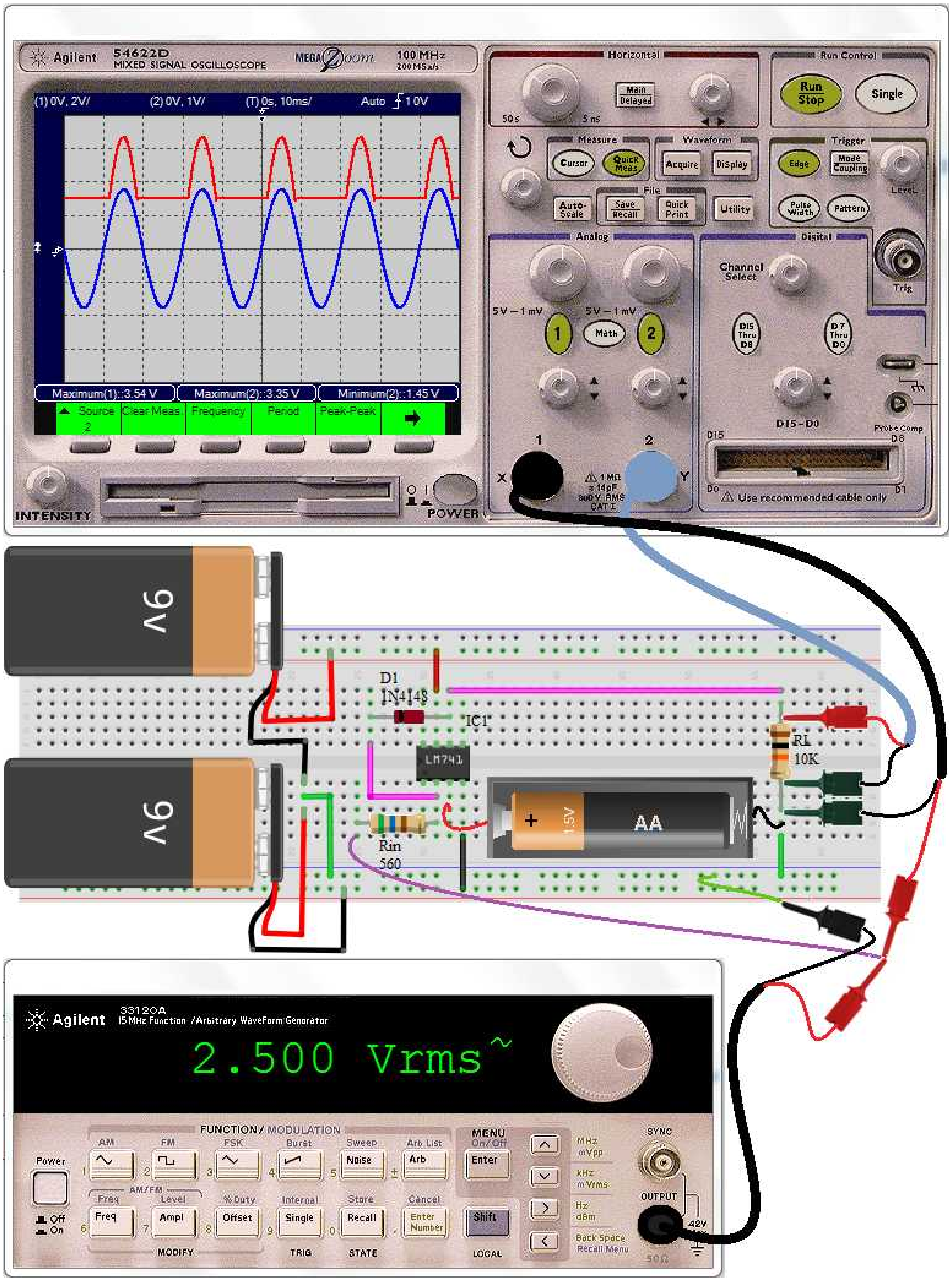


Figure 7-84: Active limiting experimental circuit connection

8. With the results from step 3 to step 7 name the circuit for each connection. Does the result reflect to the expected theoretical that for unbiased positive clipping the maximum peak is at 0V while for the negative clipping the positive peak is at 0V? This type of active clamping are said to be precise compared to passive clippers; are the above results in the same agreement?

Steps		1 - 3	4-3	5-3	6-3	7-3
Input voltage (V_{in})	V_{max}					
	V_{min}					
	V_{p-p}					
Output voltage (V_{out})	V_{p-p}					
	V_{max}					
	V_{min}					

Table 7-6: Measured result for experimental circuit clipping

7. 16. DC Motor Control

Small DC motors have many applications in electronics such as in robotics, wheels, fans, pumps etc. and their speed and direction can be controlled using electronics system.

7. 16. 1. DC Motor Speed Control using PWM

Small DC motors are efficiently controlled using pulse-width modulation (PWM) method. The circuit described here is built using an LM324 low-power quad-operational amplifier. Of the four op-amps (operational amplifiers) available in this IC, two are used for triangular wave generator and one for comparator.

Op-amp IC1a generates a 200Hz square wave, while op-amp IC1c is configured as an integrator. The square wave output of IC1a at its pin 1 is fed to the integrator input IC1b (pin 6) through resistor R_1. As IC1b is configured as an integrator, it outputs a triangular wave of the same frequency as the square wave. The output of integrator is fed to inverting amplifier IC1c at pin 9 through resistor R_6 to increase the amplitude of triangular to about 2 times. The triangular wave is fed to pin 12 of op-amp IC1d, which is configured as a comparator.

The reference voltage at pin 13 of the comparator is fixed through the potential divider arrangement formed by potentiometer POT_1 and resistor R_8. It can be set from 0V (lowermost position of POT_1) to about VCC +9V (uppermost position of POT_1 minus voltage drop of R_8). R_8 must be low value to keep the uppermost of POT to around VCC.

The triangular wave applied at pin 12 of IC1d is compared with the reference voltage at its pin 13. The output at pin 14 is about VCC (9V) when the voltage at pin 12 is greater than the voltage at pin 13. Similarly, the output at pin 14 is about 0V when the voltage at pin 12 is lower than the voltage at pin 13.

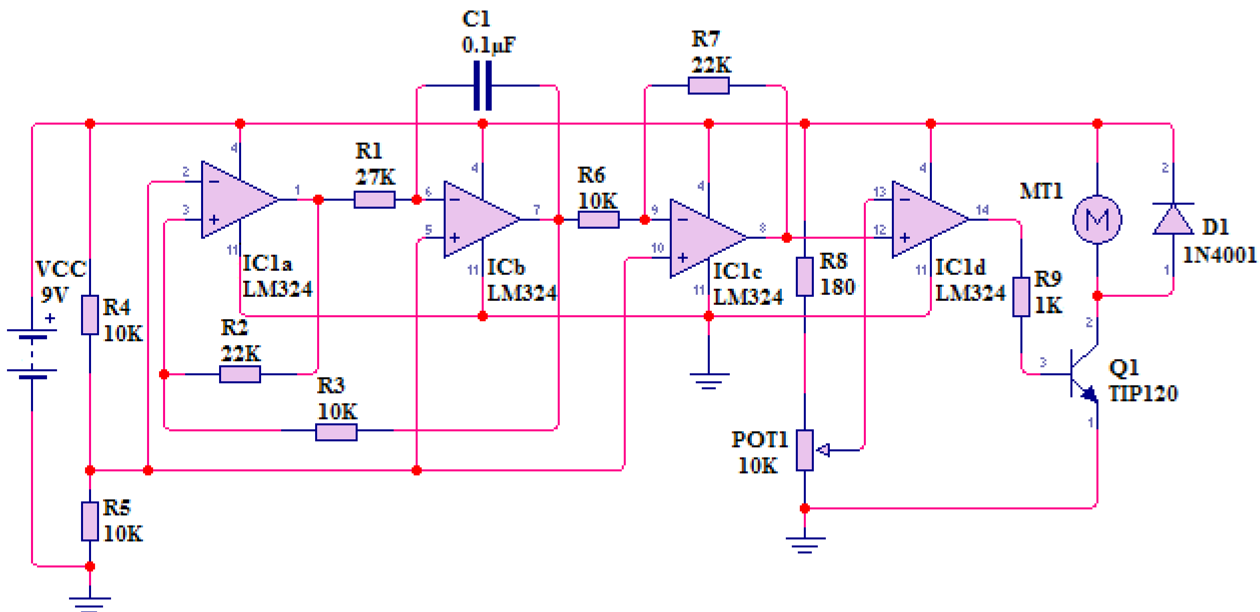


Figure 7-85: PWM DC motor speed control

The output from comparator IC1d is the base (gate) voltage for NPN transistor (Q_1) or n-channel MOSFET. Q_1 switches on when the base (gate) voltage is positive and switches off when the base (gate) voltage is negative. Setting of the reference voltage therefore controls the pulse-width of the motor. The diode D_1 prevents back-emf from inductive loads such as brushed motors from damaging the switching transistor. With "brushless" computer fan motors it is not necessary to fit this diode across the load, as they have any needed protection already in-fan. A heat-sink is not necessary at moderate loads. For load currents up to about 600mA a 2N2222A NPN can be used. For higher loads go for a Darlington power transistor such as the TIP120, 121 or 122, rated to 5A, or a power MOSFET such as the IRF530 which can carry up to 14A.

When Q_1 is switched on for a longer period, the pulse width will be wider, which means more average DC component and faster speed of the motor. Speed will be low when the pulse width is small. Thus potentiometer POT_1 controls the speed of the motor MT_1. The system can be made automatic by replacing potentiometer by an NTC (negative temperature coefficient) resistor.

Procedure

1. Assemble the circuit on a breadboard as illustrated in figure 7-86. The circuit requires 9VDC to 15VDc power supply for its working with a margin limit of the DC voltage. Remember to locate the pin 1 of the LM324 IC. The circuit has advantage to control the motor from full-off to full-on.
2. The square wave from the first op-amp IC1a is displayed in yellow, just on the pin 1, the triangular ware is displayed in right-blue at the output of inverting op-amp IC1c on pin 8, a reference voltage at inverting of comparator is displayed in violet at pin 13 of IC1d and PWM signal is displayed in green at pin 14 of IC1d.
3. The frequency of the square wave is calculated by using formula: $F = R_2/4*R_3*R_1*C$. For this experimental circuit the frequency is: F=22K/4*27K*10K*0.1µF=203Hz. If you want to try different frequencies you can alter R_1 and/or C_1 by keeping R_2 or R_3.

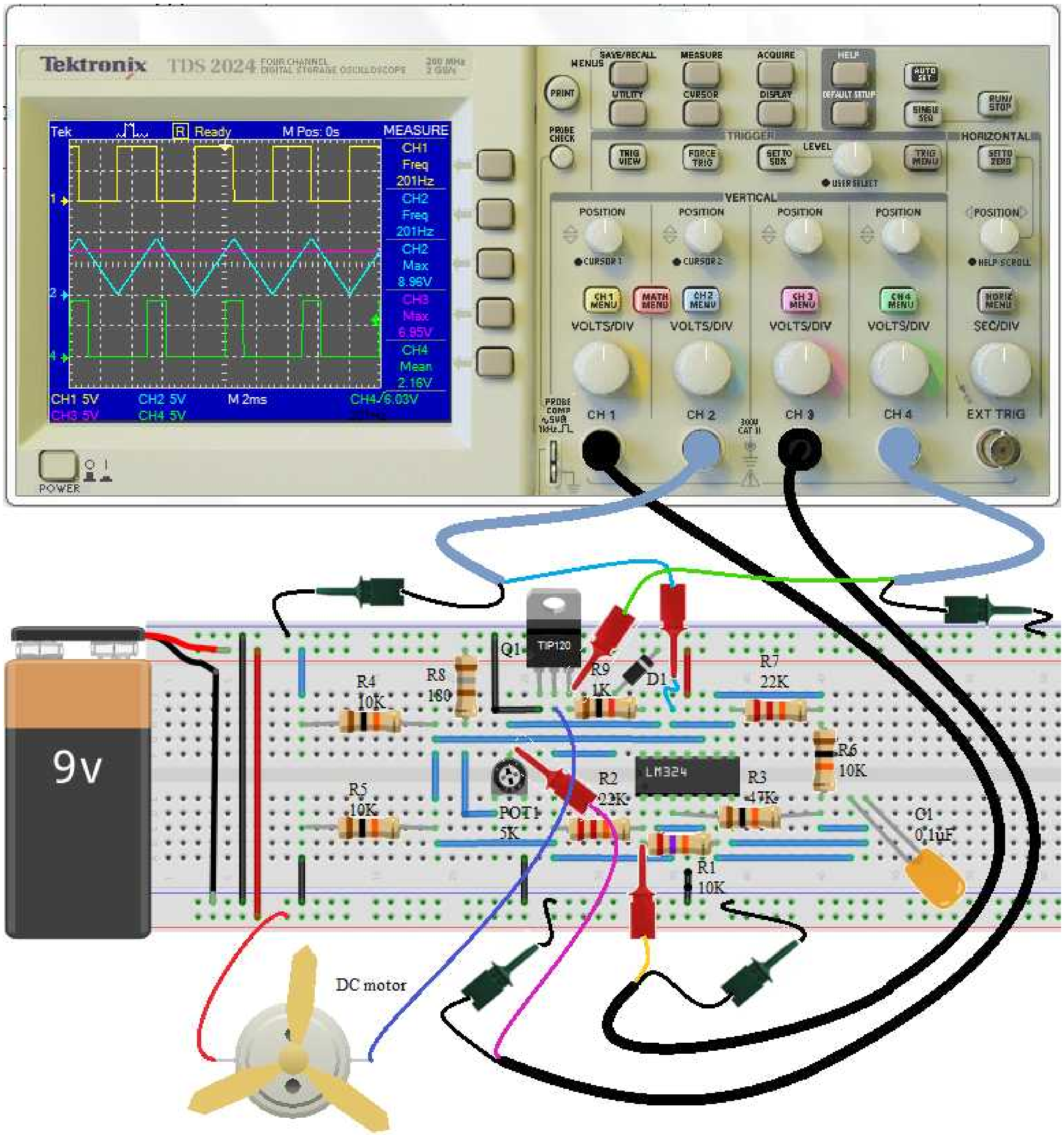


Figure 7-86: PWM DC motor speed control circuit connection

4. With the potentiometer turned fully clock-wise the resistance of wiper from ground is increased and the wiper voltage is around VCC, making to be greater than the peak of the triangular wave. This makes the load to be full-on (100% ON of the time). Decreasing the wiper voltage by turning the potentiometer anticlockwise reduces the duty cycle to full-off. It is easy to set a minimum speed just by changing the value of R_8.

7. 16. 2. DC Motor Direction Control

In robotics and other many applications we need to operate the motor in both directions forward and backward (Bi-directional). To control the direction of a DC motor, the polarity of the DC power applied to the motor's connections must be reversed allowing its shaft to rotate in the opposite direction. One very simple and cheap way to control the rotational direction of a DC motor is to use switching devices configured to form an H-bridge. H-bridge is an electronic circuit that enables a voltage to be applied across a load (motor) in either direction. Let's consider the controlling circuit that can be used to the sun position (solar tracker) for solar panel.

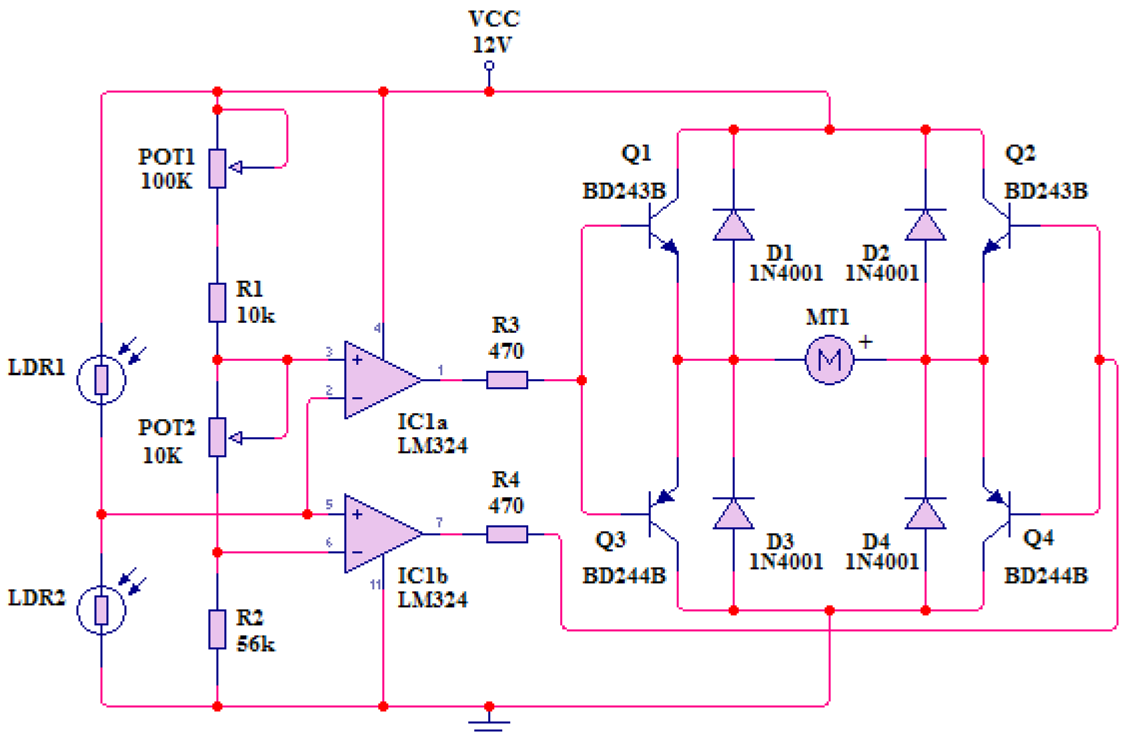


Figure 7-87: DC motor direction control using H-bridge and LDRs for solar tracker

The control circuit is based on LM324 quad-opamp configured as comparator. Op-amps IC1a and IC1b are configured as voltage comparators. The reference voltage that each triggers at is derived from the resistor voltage divider of POT_1, R_1, POT_2 and R_2. The reference voltage for IC1a is connected to the "+" noninverting input (pin 3) while for IC1b it is connected to the "–" inverting input (pin 6). Thus, IC1b is triggered by a voltage greater than its reference whereas IC1a is triggered by a voltage less than its reference. Inverting input (pin 2) of Ica is interconnected with noninverting input (pin 5) of IC1b to sense voltage level from LDR_1 and LDR_2 to determine which Op-amp is to be triggered. The solar panels are operating at optimal parameters when they are at the perfect right angle to the sun. Unfortunately this is accomplished only if solar panels are rotated by the sun. This is the purpose of solar tracker. In case of solar tracker, one LDR must be oriented in East while other in West. This is to direct the panel attached to the shaft of dc motor according to the sun light. This increases the efficiency of the solar panel. Let's take LED_1 in East and LED_2 in West. In morning LDR_1 will receive more sun light intensity than LDR_2; therefore, as both LDRs are configured as voltage divider, the voltage on LDR_2 will be greater than voltage on LDR_1 and comparator IC1b will be triggered to output a high voltage and IC1a outputs low (0V) voltage, transistors Q_2 and Q_3 conduct. At this time the motor will rotate in one direction, forward for example. At evening the sun is now in West, thus more light intensity received by LDR_2 causes a low voltage to be developed on it and IC1a to be triggered, transistors Q_1 and Q_4 conduct.

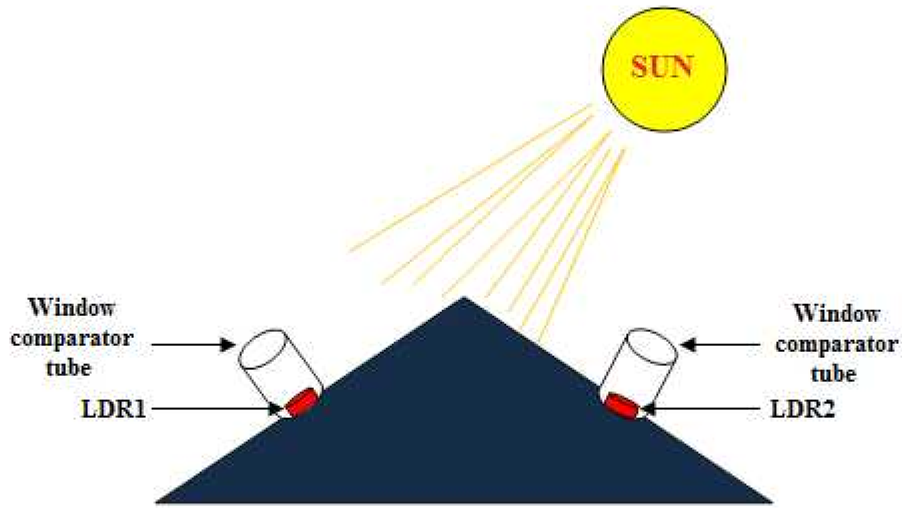


Figure 7-88: Position of LDRs on the roof for solar tracker pannel

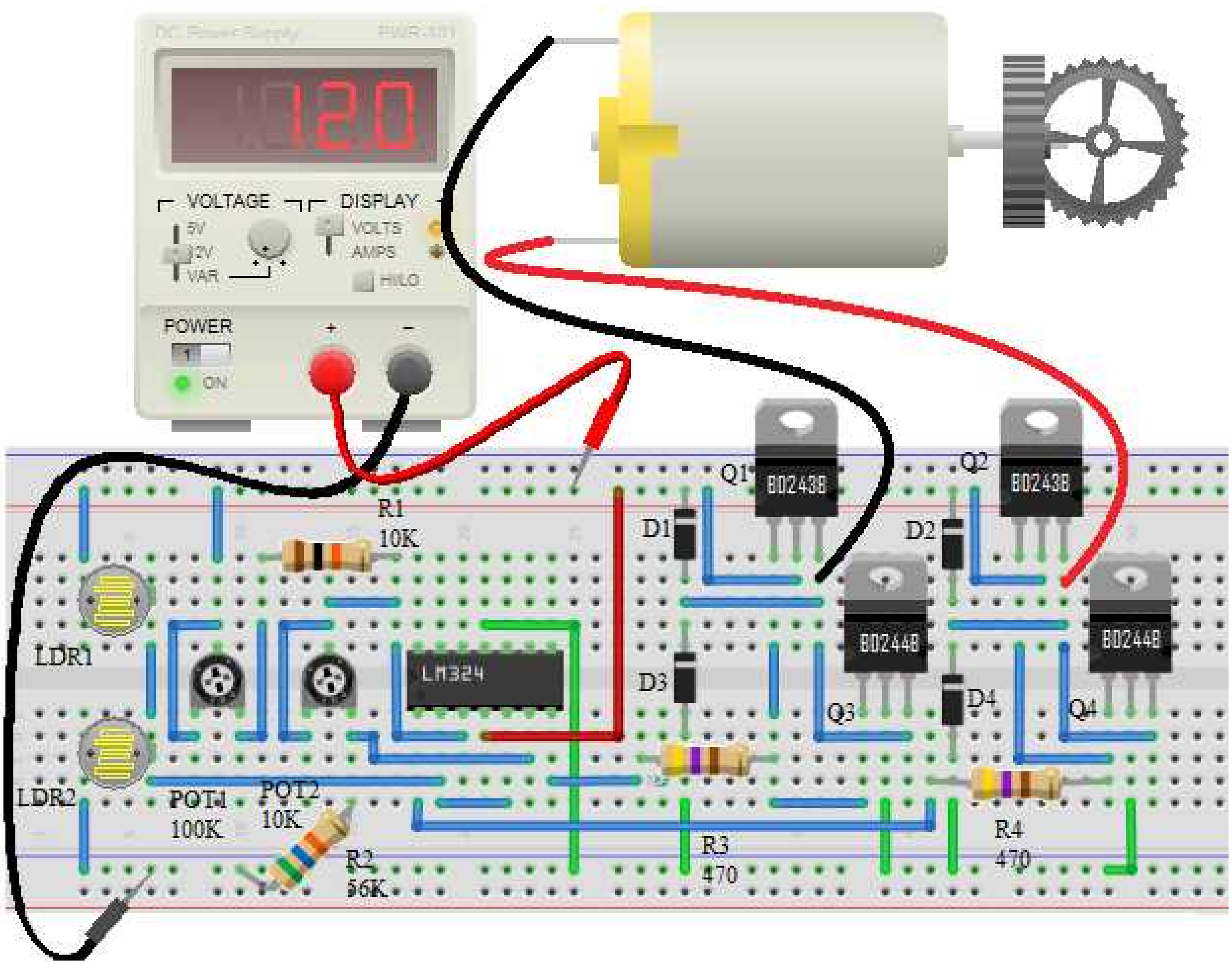


Figure 7-89: DC motor direction control using H-bridge and LDRs for solar tracker

As result the motor MT_1 rotate in other direction, reverse. The solar tracker circuit uses a window comparator to maintain the motor in an idle state as long as the two LDRs are under the same illumination level.

In this case, half the voltage is applied to the noninverting input of IC1a and to the inverting input of IC1b. Potentiometers POT_1 and POT_2 are adjusted in such way that the motor stands still when the LDRs get the same amount of solar light. When the sun position is changing so does the illumination level on the LDRs and the input voltage for the window comparator is no longer half of the supply voltage thereby the output of the comparators generate information for the motor that rotates the panels for tracking the sun. This solar tracker is used to control the panel in horizontal plane.

This circuit uses complementally transistor, BD243 (NPN) and BD244 (PNP) which can handle 6A. You can use any other source of light to test this experimental circuit.

7. 16. 3. DC Motor Speed and Direction Control

The schematic circuit of figure 7-90 shows PWM speed control and direction control for dc motor and essentially consists of an LM324 quad-op amp and four MOSFETs in a bridge configuration to drive the motor.

Both the control circuit and the motor use the same power supply. The maximum operating voltage of the circuit is limited by LM324 to 18V DC, any more than this means that; there is very little safety margin. The IRFZ44 MOSFET can handle 49A and the IRF4905 can handle 74A. An H-bridge is composed by MOSFETs Q_3 up to Q_6. Only two of these MOSFETs are on at any one time. When Q_3 and Q_6 are on then current flows through the motor MT_1 and it turn in one direction, say forward. When Q_4 and Q_5 are on the current flow is reversed and the motor spins in the opposite direction. IC1c and IC1d control which MOSFETs are turned on. Zener diodes Z_1 up to Z_4 limit the gate-source voltage of the MOSFETs to 8.2 volts. This protects the MOSFETs when powering the control circuit from higher voltages i.e. 24V battery. Op-amps IC1c and IC1d are configured as voltage comparators. The reference voltage that each triggers at is derived from the resistor voltage divider of R_6, R_7 and R_8. Note that the reference voltage for IC1d is connected to the "+" (noninverting) input but for IC1c it is connected to the "–" (inverting) input. Therefore, IC1d is triggered by a voltage greater than its reference whereas IC1c is triggered by a voltage less than its reference. Op-amp IC1b is set up as a triangle wave generator and provides the trigger signal for the voltage comparators. The frequency is approximately the inverse of the time constant of R_5 and C_1; 270Hz for the values used. Reducing R_5 or C_1 will increase the frequency; increasing either will decrease the frequency. The peak-to-peak output level of the triangle wave is less than the difference between the two voltage references. Therefore, it is impossible for both comparators to be triggered simultaneously. Otherwise all four MOSFETs would conduct, causing a short circuit that would destroy them. The triangle waveform is centred on a DC offset voltage. Raising or lowering the offset voltage changes the DC position of the triangle wave accordingly. Shifting the triangle wave up causes comparator IC1d to trigger; lowering it causes comparator IC1c to trigger. When the voltage level of the triangle wave is between the two voltage references then neither comparator is triggered.

Potentiometer POT is used to vary the motor speed and rotation; an input to the operational amplifier LM324 which acts as a comparator circuit. The DC offset voltage is controlled by the potentiometer POT via IC1a, which is configured as a voltage follower. This provides a low output impedance voltage source, making the DC offset voltage less susceptible to the loading effect of IC1b. As the POT is turned the DC offset voltage changes, either up or down depending on the direction the POT is turned. Winding the POT up increases the DC bias on the triangle wave and pushes the peaks further above the DC reference, resulting in the motor being powered for a greater portion of each cycle.

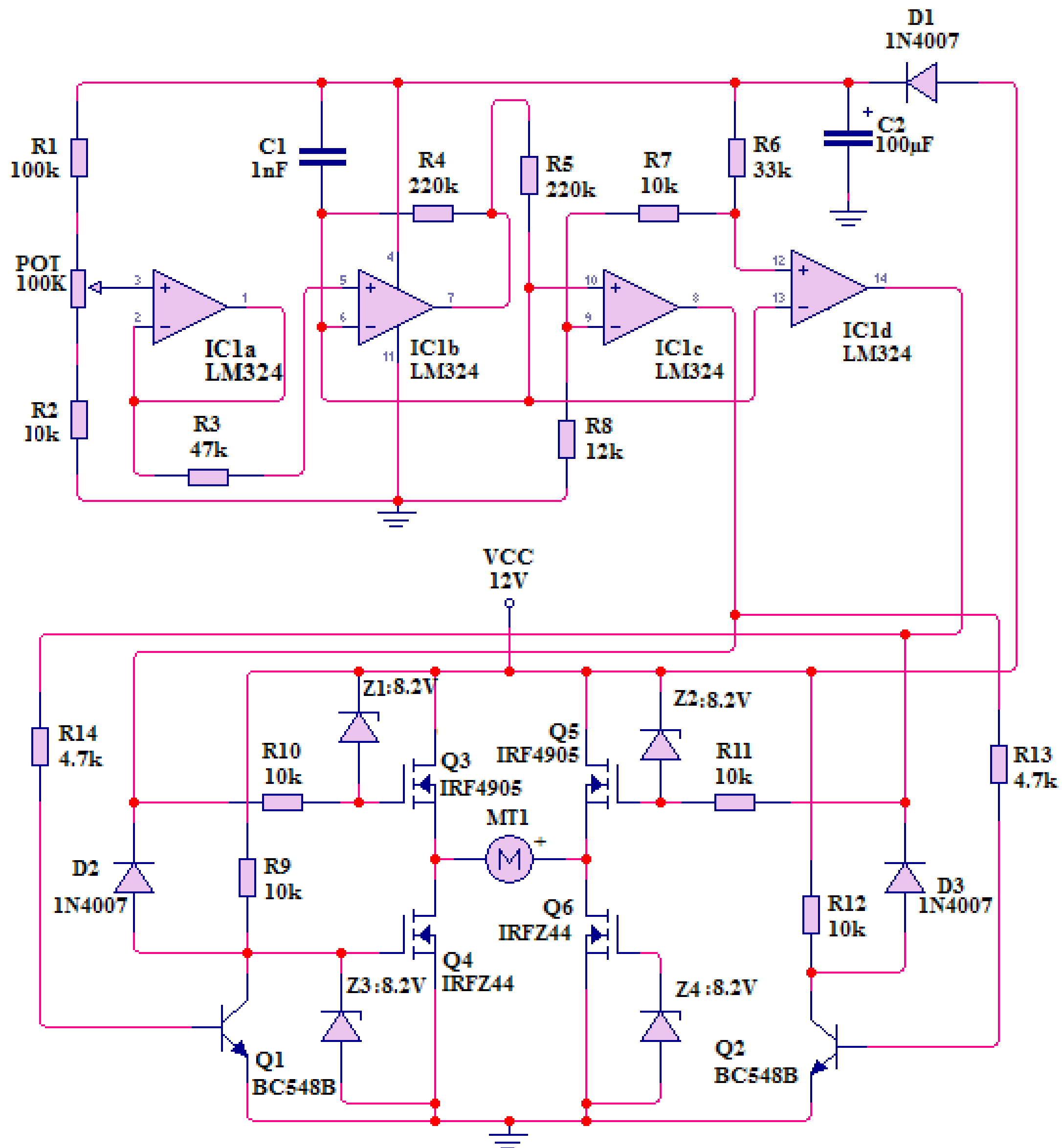


Figure 7-90: DC motor speed and direction control

Note that an unavoidable side effect of decreasing DC bias is a decrease in the oscillation frequency of the triangle generator. With the pot set for minimum DC bias (full speed); the frequency will be about 150Hz. Diode D_1 provides reverse polarity protection for the controller circuit. Capacitor C_2 is designed to filter out any voltage spikes caused by the MOSFETs as they switch to supply power to the motor. Motor direction can only be changed when the analog input is close to 0V because reversing motor direction while the motor is turning could have damaging effects on the motor and drive circuit. This circuit can run a DC motor in clockwise or anti-clockwise direction and stop it while increasing or decreasing its speed using a single potentiometer POT.

Chapter eight

The 555 timer and Applications

Objectives

After completing this chapter, you should

- Have an understanding on the working principle of 555 timer
- Be able to design astable oscillator based on 555 timer
- Be able to design monostable multivibrator based on 555 time
- Be able to design bistable multivibrator based on 555 timer
- Be able to design different circuit projects around 555 timer

chap 8

Further reading

Study aids for this chapter are available at

- *555 Timer Applications Sourcebook Experiments*; H. Berlin; BPB Publications; 218 pages; 2008.
- *IC 555 Projects*; E.A. Parr; Bernard Babani Publishing; 144 pages; 1978.
- http://www.555-timer-circuits.com/operating-modes.html

Early operational amplifier (op-amps) were used primarily to perform mathematical operations such as addition, subtraction, integration, and differentiation; hence the term *operational*. These early devices were constructed with vacuum tubes and worked with high voltages. Today's op-amps are linear integrated circuits (ICs) that are relatively low dc supply voltages and are reliable and inexpensive.

In this chapter, linear integrated circuits (ICs), in which many transistors, diodes, resistors, and capacitors are fabricated on a single silicon chip and packaged in a single case to form an operational amplifier (Op-amp), are introduced. The manufacturing process for ICs is complex and beyond the scope of this coverage.

In our study of ICs, we will treat the entire circuit as a single device. That is, you will be more concerned with what the circuit does from external point of view and will be less concerned about the internal, component-level operation.

8. 1. The 555 timer IC overview

The 555 Integrated Circuit (IC) is a popular timer that is widely used in electronic and has many applications. Its dual version is called 556 in which includes two independent 555 ICs in one package. The following illustration, figure 8-1, shows both the 555 (8-pin) and the 556 (14-pin).

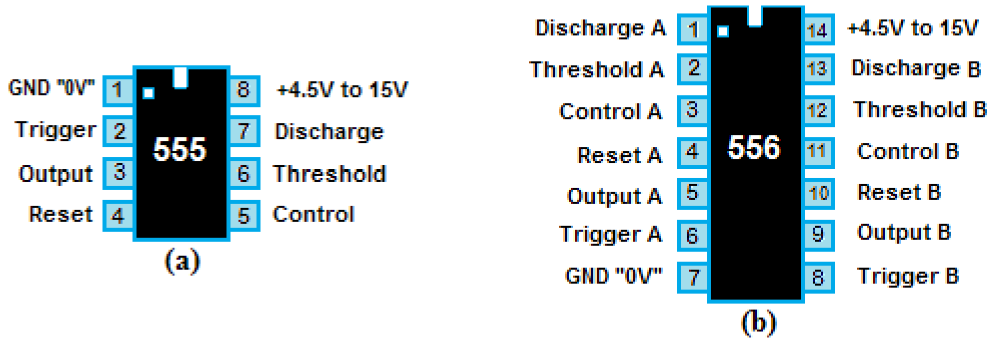


Figure 8-1: Timer ICs; (a) 555 (8-pin) and (b) the 556 (14-pin)

In a circuit diagram the 555 timer chip is often drawn as a building block in a standard layout illustrated in figure 8-2. Yet the pins are not in the same order as the actual chip, but this facilitates to recognize the function of each pin, and makes drawing circuit diagrams much easier.

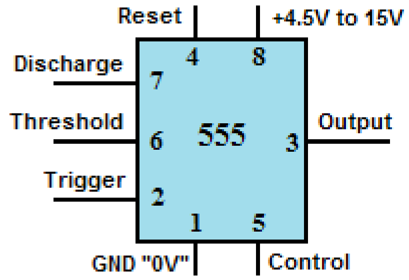


Figure 8-2: Standard schematic drawing of 555 timer

Pin Configuration of the 555 Timer

- **Pin 1 (Ground pin):** The ground (or common) pin is the most negative supply potential of the device, which is normally connected to circuit common (ground) when operated from positive supply voltages.
- **Pin 2 (Trigger):** The action of the trigger input is level-sensitive, allowing slow rate of change waveforms, as well as pulses, to be used as trigger sources. Triggering is accomplished by taking the pin from above to below a voltage level of 1/3 of rail voltage to make output HIGH. Pin 2 has control over pin 6. If pin 2 is LOW, and pin 6 is LOW, output goes and stays HIGH. If pin 6 is HIGH, and pin 2 goes LOW, output goes LOW while pin 2 is LOW.
- **Pin 3 (Output):** This is the output of the 555 IC timer. Pin 3 and pin 7 are in phase. Pin 3 goes HIGH to about 1.7V less than rail voltage and goes low to about 0.25V from ground level and can deliver up to 200mA. Exact output saturation levels vary markedly with supply voltage for both high and low states. At a V+ of 5 volts, for instance, the low state is typically 0.25 volts at 5mA. Operating at 15 volts, however, it can sink 200mA if an output

low voltage level of 2 volts is allowable (power dissipation should be considered in such case, of course), high state level is typically 3.3 volts at V+ = 5 volts; 13.3 volts at V+ = 15 volts.

- **Pin 4 (Reset):** This pin once held at voltage less than 0.8V force the output to go at low state no matter what state the other inputs are in. The reset voltage threshold level is 0.7 volt, and a sink current of 0.1mA from this pin is required to reset the device. When not used, it is recommended that the reset input be tied to V+ to avoid any possibility of false resetting.
- **Pin 5 (Control):** By applying a voltage to this pin, it is possible to vary the timing of the device independently of the RC network. In the event the control voltage pin is not used, it is recommended that it is bypassed, to ground, with a capacitor of about 0.01uF (10nF) for immunity to noise. Yet this fact is not obvious in many 555 circuits. In many circuit this pin is not connected.
- **Pin 6 (Threshold):** The action of the threshold pin is level sensitive, allowing slow rate of change waveforms. This pin detects 2/3 of rail voltage to make output LOW only if pin 2 is HIGH. Resetting via this pin is accomplished by taking the terminal from below to above a voltage level of 2/3 V+.
- **Pin 7 (Discharge):** Usually the timing capacitor is connected between pin 7 and ground. Pin 7 goes LOW when pin 6 detects 2/3 rail voltage but pin 2 must be HIGH. If pin 2 is HIGH, pin 6 can be HIGH or LOW and pin 7 remains LOW. Goes OPEN (HIGH) and stays HIGH when pin 2 detects 1/3 rail voltage (even as a LOW pulse) when pin 6 is LOW. Pins 7 and 3 are in phase but pin 7 does not go high. It goes open and low and will sink about 200mA.

chap 8

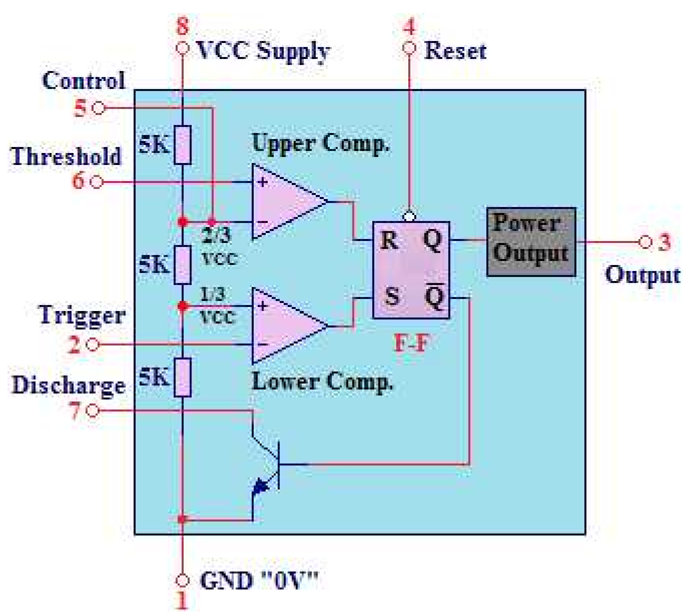


Figure 8-3: NE555 timer internal structure

- **Pin 8 (Positive supply):** This pin, also referred to as VCC, is the positive supply voltage terminal of the 555 timer IC. Supply voltage operating range for the 555 is +4.5 volts (minimum) to +16 volts (maximum), and it is specified for operation between +5 volts and

+15 volts. The device will operate essentially the same over this range of voltages without change in timing period. Actually, the most significant operational difference is the output drive capability, which increases for both current and voltage range as the supply voltage is increased.

Basic Circuits with 555 Timer IC

If we consider the output of the 555 timer we can classify it as a digital device but by looking to its function it relies on both analog and digital electronic techniques. The output of the 555 timer takes one of the two logic states at any time. The output can either be in "low-state" which is 0V or be in "high-state" which is the supply voltage VCC. The state and behavior of the output determines one of the three main circuits with 555 timer. All IC timers rely upon an external capacitor to determine the off/on time intervals of the output pulses. As you recall, it takes a finite period of time for a capacitor (*C*) to charge or discharge through a resistor (*R*). Those times are clearly defined and can be calculated given the values of resistance and capacitance.

8. 2. Monostable 555 Timer

555 timer configured as monostable functions as a "one-shot". A Monostable circuit produces one pulse of a set length in response to a trigger input such as a push button. The output of the circuit stays in the low state until there is a trigger input, hence the name "monostable" meaning "one stable state".

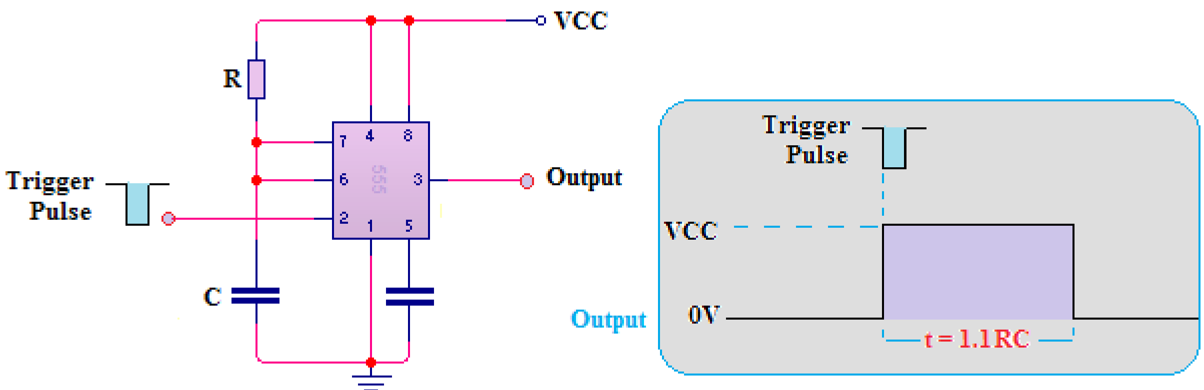


Figure 8-4: Monostable with 555 timer

An external *RC* network is connected between the supply voltage and ground. The junction of the resistor and capacitor is connected to the threshold input which is the input to the upper comparator. The internal discharge transistor is also connected to the junction of the resistor and the capacitor. An input trigger pulse is applied to the trigger input which is the input to the lower comparator. With this configuration, the circuit is initially reset. Therefore, the output voltage is near zero volts. The signal from the control flip-flop causes transistor to conduct and act as a short circuit across the external capacitor. For that reason, the capacitor cannot charge. During that time, the input to the upper comparator is near zero volts causing the comparator output to keep the control flip-flop reset.

The trigger input is initially high (about 1/3 of VCC). When a negative-going trigger pulse is applied to the trigger input the threshold on the lower comparator is exceeded. The lower comparator, therefore, sets the flip-flop. That causes transistor to cut off, acting as an open circuit. The setting of the flip-flop also causes a positive-going output level which is the beginning of the output timing pulse.

The capacitor now begins to charge through the external resistor. As soon as the charge on the capacitor equal 2/3 of the supply voltage, the upper comparator triggers and resets the control flip-flop. That terminates the output pulse which switches back to zero. At this time, transistor again conducts thereby discharging the capacitor. If a negative-going pulse is applied to the reset input while the output pulse is high, it will be terminated immediately as that pulse will reset the flip-flop.

Whenever a trigger pulse is applied to the input, the 555 will generate its single-duration output pulse. Depending upon the values of external resistance *R* and capacitance *C* used, the output timing pulse may be adjusted from approximately one millisecond to as high as on hundred seconds. For time intervals less than approximately 1-millisecond, it is recommended that standard logic one-shots designed for narrow pulses be used instead of a 555 timer. IC timers are normally used where long output pulses are required. In this circuit configuration, the duration of the output pulse in seconds is approximately equal to:

$$T = 1.1RC \qquad \text{[Equation 8: 1]}$$

Monostable is said to have a single stable state, that is the off state. Whenever it is triggered by an input pulse, the monostable switches to its temporary state. It remains in that state for a period of time determined by an *RC* network. It then returns to its stable state. In other words, the monostable circuit generates a single pulse of fixed time duration each time it receives an input trigger pulse, thus the name one-shot.

Use of Monostable Timer

Monostable timers are used for turning some circuit or external component on or off for a specific length of time. It is also used to generate delays. When multiple monostables are cascaded, a variety of sequential timing pulses can be generated. Those pulses will allow you to time and sequence a number of related operations.

Experiment 8-1: Monostable using 555 timer

Design a monostable based on 555 timer that turns on an LED for 11 seconds once a push button is pressed-released. Modify the circuit to drive a solenoid which has to be used to unlock the gate for 11 seconds delay and back in its locked state.

Parts and materials

No	Item	Quantity
1	12V dc supply	1
2	LED (Light Emitting Diode)	1
3	100µF/50V electrolytic capacitor, 0.1nF ceramic capacitor	2
4	100KΩ ½ watt resistor, 3x1KΩ ¼ watt resistors	3
5	12V Solenoid	1
6	12V SPDT Relay	1
7	2N2222a transistor	1

8	1N4001 diode	1
9	Breadboard	1
10	22-gauge connecting solid wire	50cm

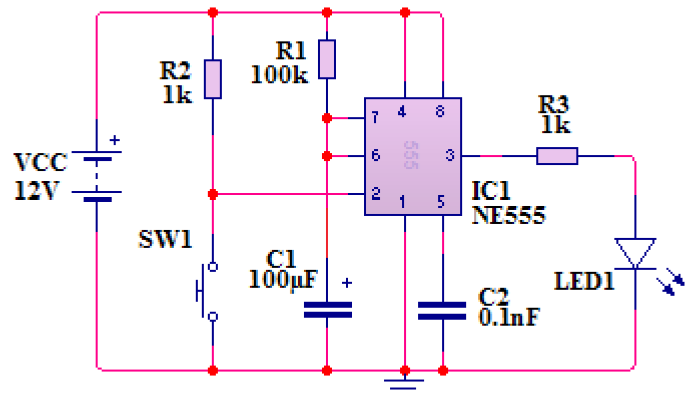


Figure 8-5: Monostable circuit using 555 timer

The circuit of figure 8-5 shows a 555 timer configured as monostable multivibrator. A 555 timer in monostable mode will output a high pulse (of voltage close to VCC) when the trigger pin is pulsed low. The duration of this output pulse is dependent on the values of R_1 and C_1. From the equation 8: 1 the duration of the pulse output from the 555 in monostable mode with R =100KΩ and C =100µF is:

$$t = 1.1*R*C \text{ seconds}$$
$$t = 1.1*100000*0.0001$$
$$t = 11 \text{ sec}$$

This means that with a 100KΩ resistor and 100µF capacitor, a pulse low to the 555's trigger pin (pin 2) will cause the output to go high for 11 seconds. A load (LED) connected to the output of monostable will be on for 11 seconds once a low pulse is applied to pin 2 of 555 timer. Resistor R_3 is used to limit the current for the safe of LED_1.

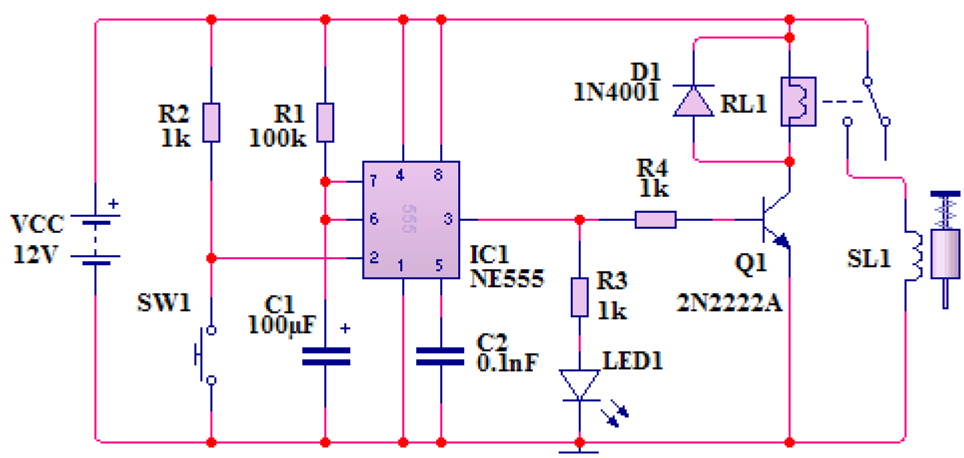


Figure 8-6: Monostable using 555 timer for driving a solenoid

There are hundreds of circuits based on 555 monostable, but the only difference relies on connected loads and the type of trigger connection. Now the load of the circuit the figure 8-6 is modified to drive a solenoid. Resistor R_4 limits the current on the base of transistor Q_1. This resistor has to be well chosen so that at the stable state of monostable the transistor stays in cutoff. A small limiting resistor will pass a small current which can drive a connected load since in practice at stable state of monostable the output is not exactly 0V. When the output of IC1 is high the transistor Q_1 is saturated and make the relay RL_1 to be energized. Whenever the relay is energized will close the contact and the solenoid SL_1 will be activated (pull to unlock). After a certain time, determined by R_1 and C_1, the solenoid will be denergized and the spring will push the sponger to go back in its normal state (locked).

Procedure

1. Connect the circuit as illustrated in figure 8-7. Connect power (positive) and ground (negative) to pins 8 and 1 of the 555 timer (red and black).

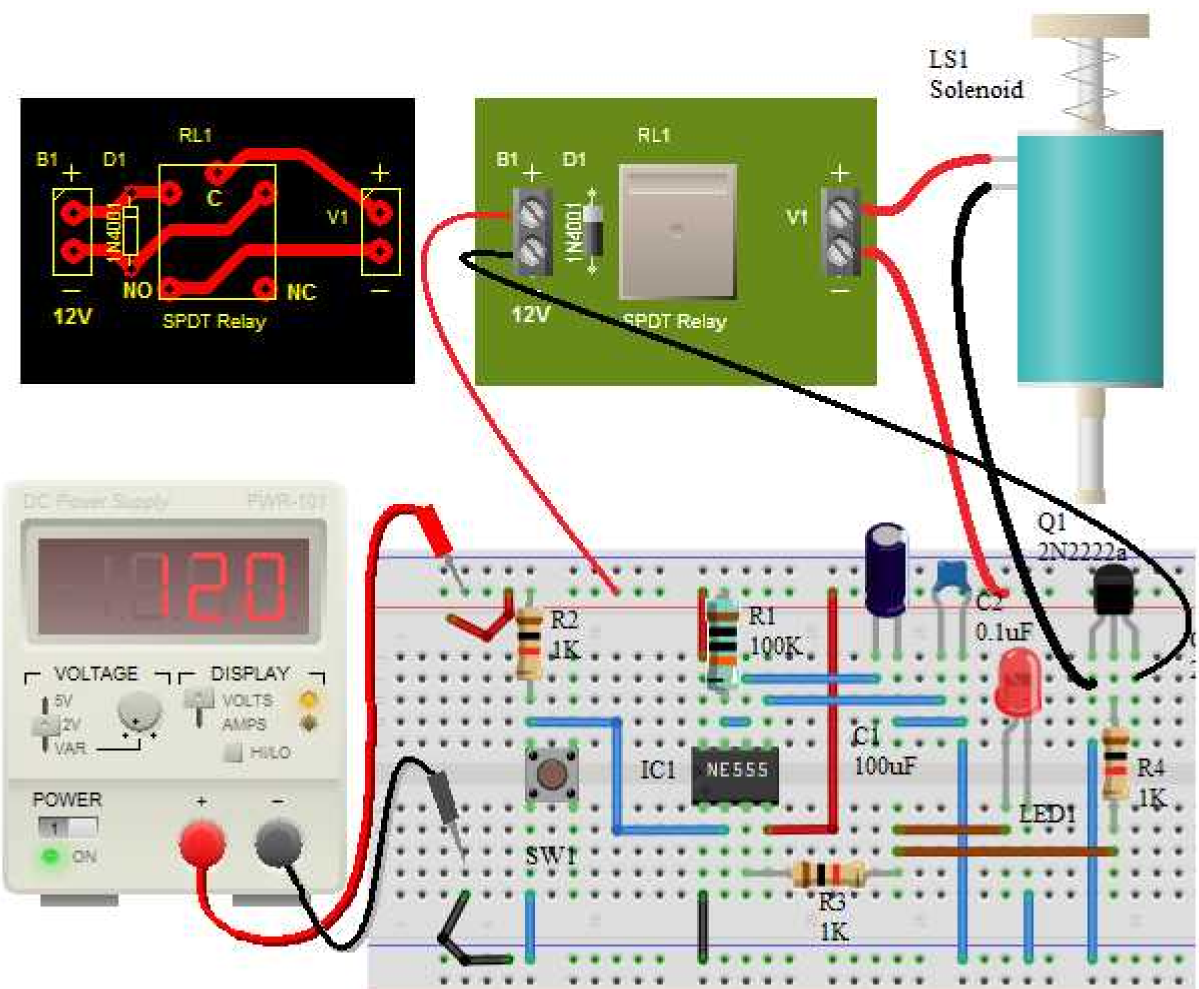
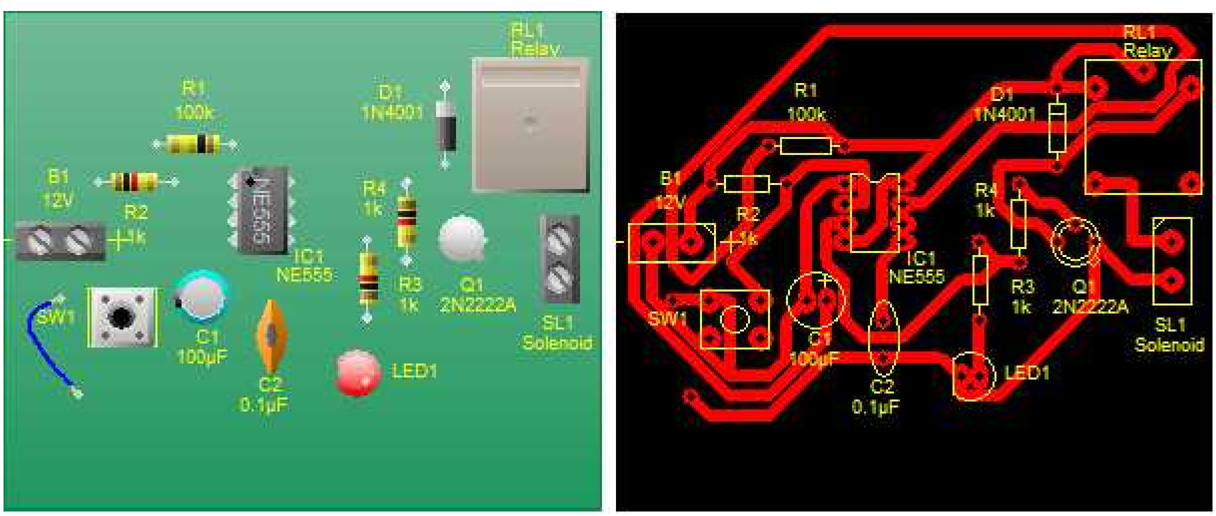


Figure 8-7: Monostable using 555 timer for driving a solenoid circuit connection

2. Connect a 0.1µF capacitor between pin 5 and ground. Connect a 100µF capacitor between pin 6 and ground; make sure the negative led of electrolytic capacitor is at ground. Use

a jumper wire to connect pins 7 and 6. Connect 100KΩ precision resistor (brown, black, black, orange, brown) between pin 7 and VCC. Connect pin 4 to VCC. Connect an LED and 1KΩ current limiting resistor in series from the output of the 555 to ground. Remember to identify anode and cathode of LED. Cathode must be connected to ground. Connect the momentary push button switch in series with a 1KΩ resistor between VCC and ground. Connect a wire from the junction between the switch and resistor to the trigger pin so that when the switch is not pressed the trigger pin is held high. When the switch (push button) is pressed the trigger pin is dropped to low (ground).

3. Power on the circuit. Press the button SW_1. The LED should light up for a time and turn off. If you time the LED, you will find that it lights up for exactly 11 seconds just as we calculated above.
4. Add transistor and relay to drive solenoid. Connect 1KΩ current limiting resistor in series from the output of the 555 to the base of switching transistor Q_1. Connect the emitter of transistor to ground and collector to negative marked terminal of the relay. Connect positive marked terminal of the relay to VCC. Make sure the relay is designed with freewheeling diode, just in parallel with relay inductor, for protection of transistor. Connect common of the relay to VCC and NO (or vice versa) to positive terminal of the solenoid. Connect negative terminal of the solenoid to ground. In some cases the solenoid can be supplied with a separate battery.
5. Power on the circuit. Press a momentary button SW_1. The LED should light up for a time and turn off. At the same time, the solenoid should also be pulled for a time and back (pushed by spring). Use timing watch to calculate the time at which the solenoid is activated. This should be 11 seconds.



(a) (b)

Figure 8-8: Monostable using 555 timer for driving a solenoid; (a) Real world circuit, (b) Artwork circuit

Applications of Monostable

This type of circuit is ideal for use in a "push to operate" system for a model displayed at exhibitions. A visitor can push a button to start a model's mechanism moving, and the mechanism will automatically switch off after a set time. Other applications include timers, missing pulse detection, bouncefree switches, touch switches, frequency divider, etc..

8. 3. Astable 555 Timer

An astable 555 timer circuit has no stable state, hence the name "astable". The output, called a square wave, continually switches between high and low states without any involvement of the user. The basic circuit is illustrated in figure 8-9.

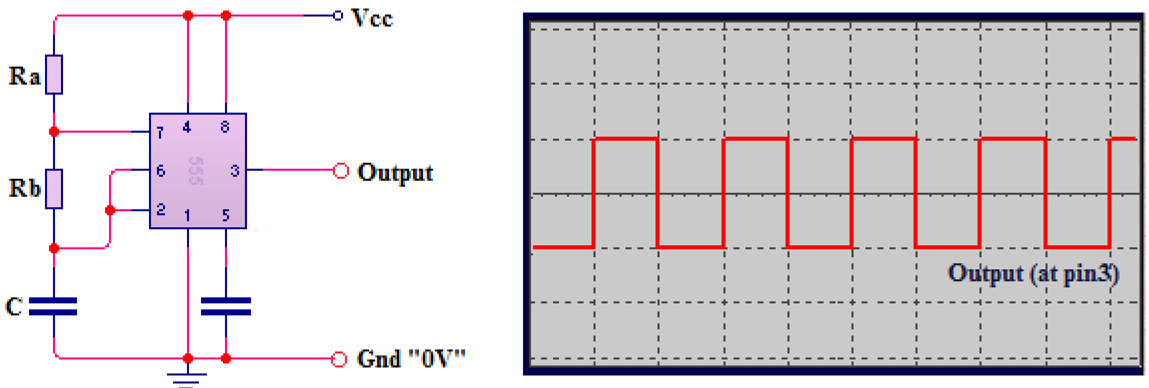


Figure 8-9: Astable 555 timer base circuit

As shown in figure 8-9, both the trigger and threshold inputs (pins 2 and 6) to the two comparators are connected together and to the external capacitor. The capacitor charges toward the supply voltage through the two resistors, R_a and R_b. The discharge pin (7) connected to the internal transistor is connected to the junction of those two resistors. When power is first applied to the circuit, the capacitor will be uncharged; therefore, both the trigger and threshold inputs will be near zero volts. The lower comparator sets the control flip-flop causing the output to switch high. That also turns off internal transistor. That allows the capacitor to begin charging through R_a and R_b. As soon as the charge on the capacitor reaches 2/3 of the supply voltage VCC, the upper comparator will trigger causing the flip-flop to reset. That causes the output to switch low. Transistor also conducts. The effect of transistor conducting causes resistor R_b to be connected across the external capacitor. Resistor R_b is effectively connected to ground through internal transistor. The result of this is that the capacitor now begins to discharge through R_b.

As soon as the voltage across the capacitor C reaches 1/3 of the supply voltage, the lower comparator is triggered. That again causes the control flip-flop to set and the output to go high. Internal transistor cuts off and again the capacitor C begins to charge. That cycle continues to repeat with the capacitor alternately charging and discharging, as the comparators cause the flip-flop to be repeatedly set and reset. The resulting output is a continuous stream of rectangular pulses.

The frequency of operation of the astable circuit is dependent upon the values of R_a, R_b, and C. The frequency is calculated with the use of formula:

$$f = 1/[0.693 * C * (R_a + 2 * R_b)] \qquad \text{[Equation 8: 2]}$$

Where: - f is frequency in Hz

- R_a and R_b are resistors in ohm and
- C is capacitance if Farads

The time duration between pulses is known as the "period", and usually designated with a "T".

The pulse is on for T_{ON} seconds, then off for T_{OFF} seconds. The total period T is:

$$T = T_{ON} + T_{OFF} \quad \text{[Equation 8: 3]}$$

This time interval is related to the frequency by the familiar relationship:

$$f = 1/T \text{ or } T = 1/f \quad \text{[Equation 8: 4]}$$

The time intervals for the on and off portions of the output depends upon the values of R_a and R_b. The ratio of the time duration when the output pulse is high to the total period is known as the duty-cycle and is expressed in percentage. The duty-cycle can be calculated with the formula:

$$D = T_{ON} / T*100 = (R_a + R_b) / (R_a + 2R_b)*100 \quad \text{[Equation 8: 5]}$$

If an astable has a duty cycle of 60%, this means that the output pulse is on or high for 60% of the total period. The duty-cycle can be adjusted by varying the values of R_a and R_b.

You can calculate T_{ON} and T_{OFF} times with the formulas below:

$$T_{ON} = 0.693*(R_a+R_b)*C \quad \text{[Equation 8: 6]}$$

$$T_{OFF} = 0.693 * R_b * C \quad \text{[Equation 8: 7]}$$

A 50% duty cycle is gotten by inserting diode in parallel with R_b as shown in figure 8-10 in such way that R_a equal to R_b.

chap 8

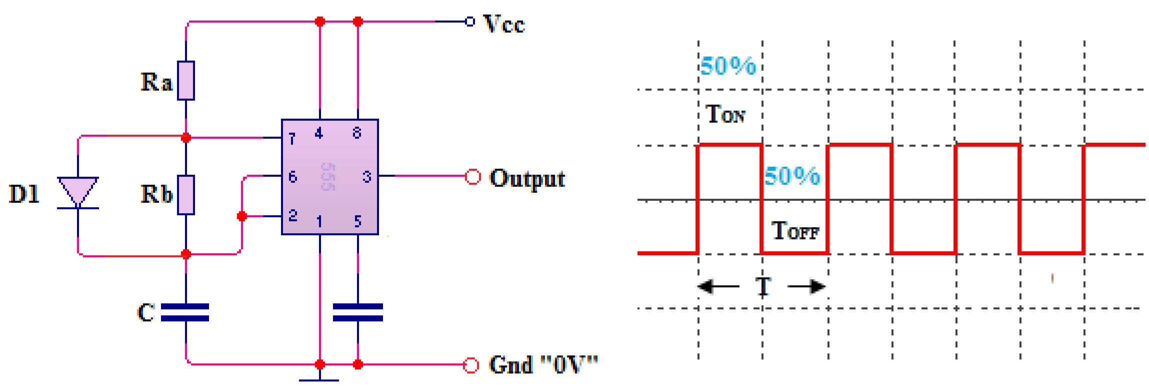


Figure 8-10: Astable configured to 50% duty cycle

During the charging time the diode *D1* is forward biased and short R_b, thus, the capacitor *C* is charged through R_a. When capacitor is full charged the diode becomes reverse biased and capacitor *C* discharges through R_b. As $R_a = R_b = R$, T_{ON} also equals to T_{OFF} and are calculated as follow:

$$T_{ON} = T_{OFF} = 0.693 * R* C \quad \text{[Equation 8: 8]}$$

Experiment 8-2: Astable multivibrator with 555 timer

Design an astable oscillator based on 555 timer that turns on/off two LEDs (red and green LEDs) alternatively. The red LED turns on when the output of time is low (0V) while the green LED is off and the green LED turns on when the output of time is high (VCC) while the red LED is off.

Parts and materials

No	Item	Quantity
1	9V dc supply	1
2	LED (Light Emitting Diode) Green and red	2
3	0.1nF ceramic capacitor	1
4	2x27KΩ ½ watt Ra and Rb Resistors, 2x560Ω ¼ watt resistors	4
5	68uF/50V, electrolytic capacitor	1
6	Breadboard	1
7	Connecting wires (22-gauge solid wire)	30Cm
8	1N4148 small signal diode	1
9	NE555 timer	1
10	Oscilloscope	1

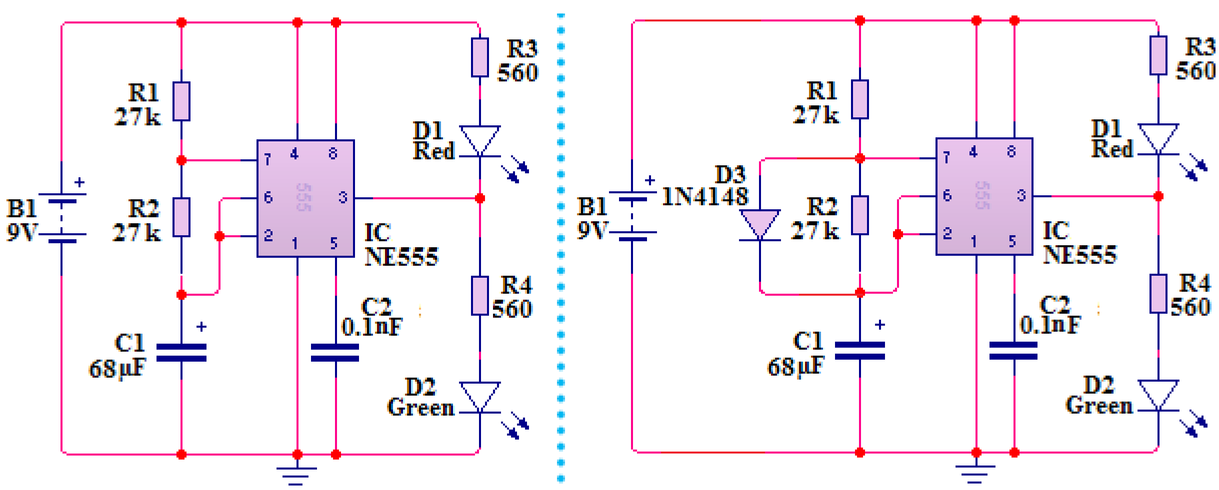


Figure 8-11: Astable multivibrator with 555 timer

Calculation of period (T), T_{ON}, T_{OFF}, frequency (f) and duty cycle (left circuit)

- Period T = $0.693 * C_1 * (R_1 + 2 * R_2) = 0.693 * 68*10^{-6} (27000\Omega + 2*27000\Omega)$
  $= 3.817044$ Sec

T_{ON} and T_{OFF} are calculated using equation 8:6-7:

- $T_{ON} = 0.693*(R_1+R_2)*C_1 = 0.693*(27000\Omega + 27000\Omega)*68*10^{-6}$
  $= 2.544696$Sec

This implies that the green LED (D_2) will light on for 2.544696Sec of the total period

- $T_{OFF} = 0.693 * R_2 * C_1 = 0.693*27000\Omega*68*10^{-6}$
  $= 1.272348$Sec

chap 8

This implies that the red LED (D_1) will light on for 1.272348Sec of the total period

Equation 8:3 can be verified by adding T_{ON} and T_{OFF}:

$T = T_{ON} + T_{OFF} = 2.544696+1.272348 = 3.817044$ Sec

- From equation8:2, $f = 1/[0.693 * C_1 * (R_1 + 2 * R_2)] = 1/[0.693 * 68*10^{-6} (27+ 2 * 27)1000]$ $= 0.262$Hz

Equation 8:4 is also verified by taking inverse of period:

Frequency f =1/T = 1/3.817044Sec =0.26198Hz

- Duty cycle is calculated by using equation 8:5:

$D\% = T_{ON}/T = 2.544696/3.817044 *100 = 66.6\%$

Calculation of period (T), T_{ON}, T_{OFF}, frequency (f) and duty cycle (Right circuit)

The use of D_3 makes capacitor to be charged through R_1 and discharged through R_2. Since R_1 equal to R_2, T_{ON} also equal to T_{OFF} and are calculated using equation 8: 8.

- $T_{ON} = T_{OFF} = 0.693 * R * C_1$
  $= 0.693 * 27000\Omega * 68*10^{-6} = 1.272348$Sec

This signifies that the time the red LED (D_1) and green LED (D_2) take to light up or off is equal to 1.272348Sec alternatively.

Period $T = T_{ON} + T_{OFF}$ = 1.272348Sec +1.272348Sec = 5.544696Sec

Frequency f = 1/T = 1/5.544696Sec = 0.3929Hz

Duty cycle $D = T_{ON}/T$ = 1.272348Sec/5.544696Sec *100 =50%

Procedure

1. Connect the circuit as illustrated in figure 8-12. Connect the power (positive) and ground (negative) to pins 8 and 1 respectively. You can use power supply from 5V to 15V. Connect 0.1nF capacitor C_2 between pin 5 and ground. Connect 68µF electrolytic capacitor C_1 between pin 6 and ground and make sure that the negative lead of capacitor is at ground. Connect pin 2 to pin 6 with a jumper wire. Connect 27KΩ resistor R_1 between VCC and pin 7. Connect 27KΩ resistor R_2 between pin 7 and pin 6. Connect pin 4 to VCC.
2. Make series connection of 560Ω resistor R_3 and D_1 (red LED) from VCC to pin 3. Make series connection of 560Ω resistor R_4 and D_2 (green LED) from pin 3 to ground.
3. Use the oscilloscope to measure the frequency, period, V_{max}, +Width (T_{ON}), −Width (T_{OFF}), and duty cycle. Use the stop-watch to calculate the time at which the red LED and green LED are on and off. Compare practical results to the theoretical results as calculated earlier.
4. Turn off the power and connect the small signal diode D_3 (1N4148) in parallel with R_2 in such way that the anode is at pin 7 and cathode at pin 6. Now switch on the power and repeat step3.

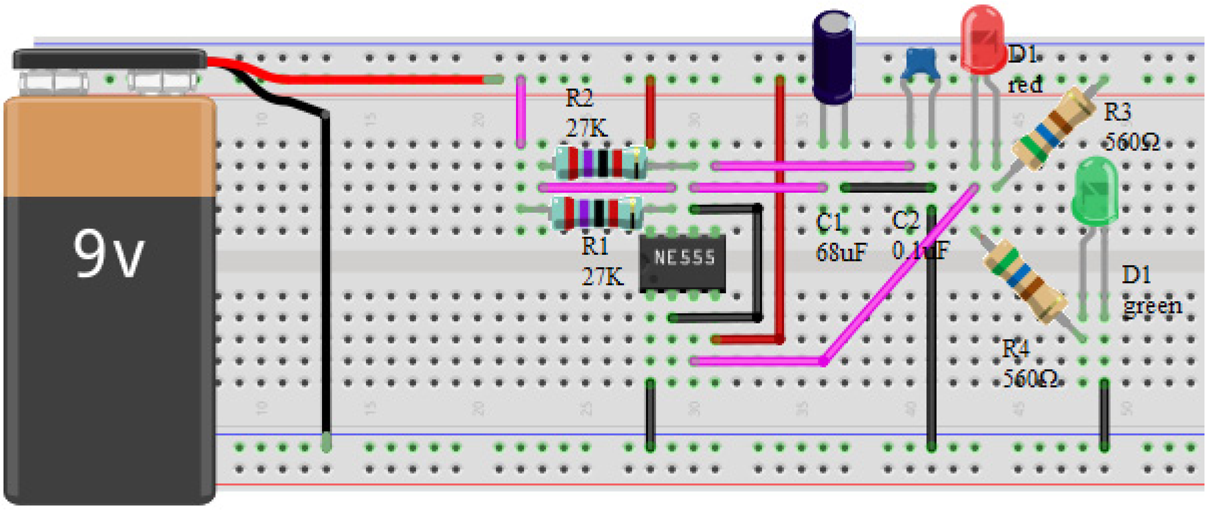


Figure 8-12: Astable multivibrator with 555 timer circuit connection

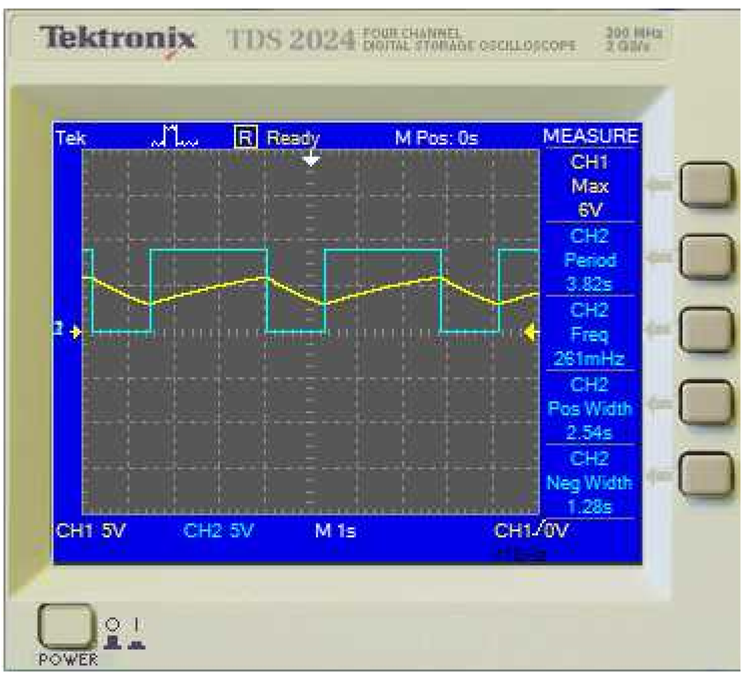
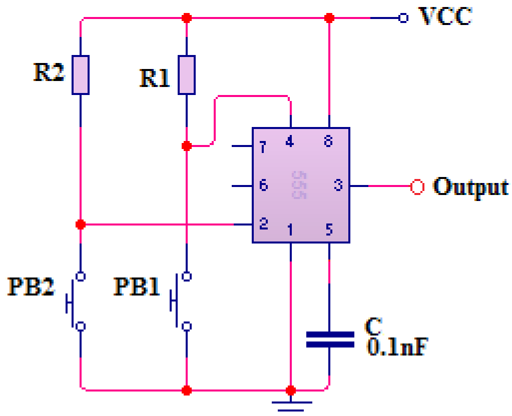


Figure 8-13: Astable 555 timer waveform. Waveform on capacitor is displayed in yellow and output waveform on pin 3 is displayed in blue

From the graph displayed on oscilloscope, the peak voltage on capacitor is 6V (the voltage at which the output goes low, the min voltage on capacitor is 3V (the voltage at which the output goes high). This is because the threshold (pin 6) and trigger pins (pin 2) are connected together.

8. 4. Bistable 555 Timer

Bistable 555 timer has two stable states. The working of bistable 555 timer is based on flip-flop principle. A flip-flop is a circuit which alternates between two stable states depending on its inputs. For the case of timer the two inputs are the trigger (pin 2) and reset (pin 4). Initially, both are kept high via a pull up resistors in bistable mode. When the trigger pin is pulsed low, it causes the output to go high (VCC). The output will remain high even if the trigger is high again. When a reset pin is pulsed low, the output goes low. The output will remain in this state even if the reset pin goes high again. In astable and monostable, the output pin would remain high until the voltage at the threshold (pin 6) reaches 2/3 of the supply. However, because pin 6 is not connected no voltage is ever present on pin 6. Thus, the threshold is never reached, so the output remains high indefinitely until a low pulse is applied to pin 4.



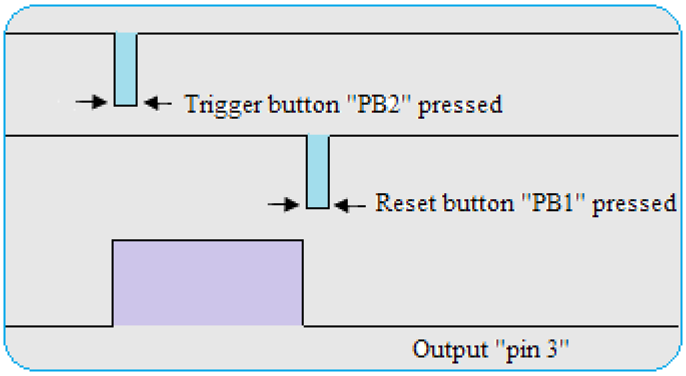


Figure 8-14: Bistable 555 timer

The momentary buttons are often used to pulse the trigger and reset pin but in other circuits the buttons can be replaced by any other electronic switching device. The reset input pin (pin 4) is connected to VCC in the same manner as the trigger input. As a result, when the reset switch is pressed, pin 4 is short circuited to ground, creating a low pulse, which resets 555 timer and brings the output back to high.

Experiment 8-3: Bistable 555timer

Design a bistable based on 555 timer that turn on an LED when a push button (one) PB_1 is pressed and turn off an LED when a push button (two) PB_2 is pressed.

Parts and materials

No	Item	Quantity
1	9V dc supply	1
2	LED (Light Emitting Diode)	1
3	0.1nF ceramic capacitor	1
4	6.8KΩ ½ watt Resistors and 1.2KΩ resistor	3
5	NO push button	2
6	Breadboard	1
7	Connecting wires (22-gauge solid wire)	20Cm
8	NE555 timer IC	1

Press the button attached to pin 2 (trigger). The LED should light up indicating that the output is at high state. Release the button, the LED will remain lit. Now press the button attached to pin 4 (reset). This will cause the output to go low and turn the LED off. The LED will remain off until you press the trigger button again. As a result, the circuit toggles between two states depending on the button pressed.

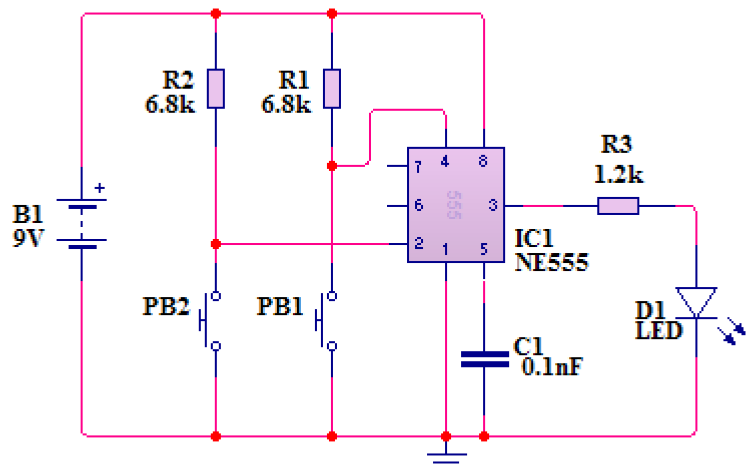


Figure 8-15: Bistable multivibrator with 555 timer

Procedure

1. Connect the circuit as illustrated in figure 8-16. Connect power (positive) and ground (negative) to pin 8 and pin 1 respectively. Leave pin 7 floating. Connect a 0.1nF ceramic capacitor C_1 between pin 5 and ground. Connect 6.8KΩ resistor between pin 2 and VCC and between pin 4 and VCC. These are pull-up resistors which maintain pin 2 and pin 4 high by default. Connect the momentary switches PB_1 and PB_2 between ground and the rest and trigger pins. Use jumper wire to connect pin 2 and pin 4 to two momentary switches (PB_1 and PB_2) connected to ground. When each of the push button is pressed it will cause its corresponding pin to go low (ground) for a short time.
2. Make series connection from pin 3, 1.2KΩ resistor R_3, and LED D_1 to ground. Remember to identify anode and cathode of LED. Cathode has to be connected to ground.
3. Press PB_1. LED will light up and remains lit until you press PB_2 to turn it off.

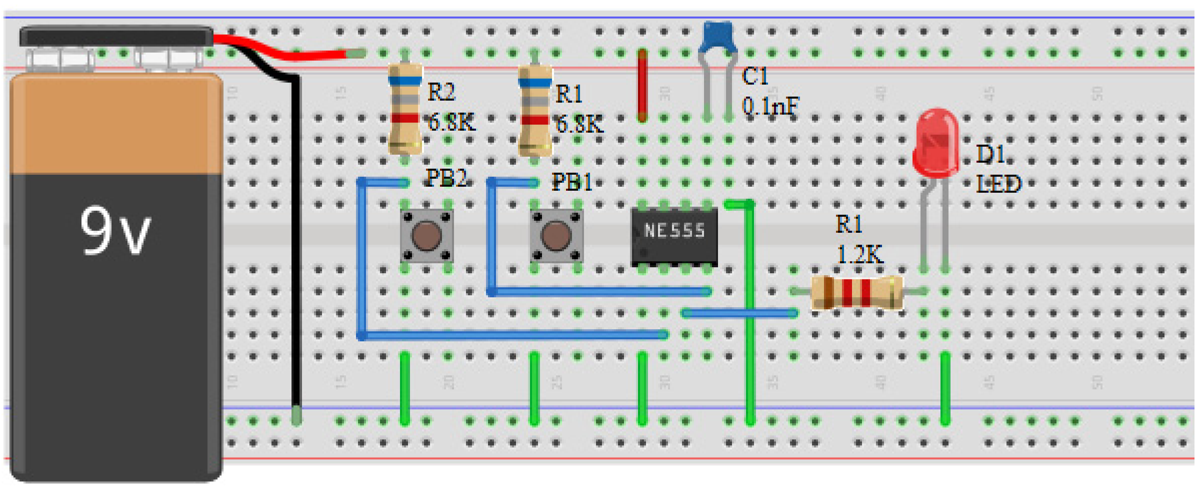


Figure 8-16: Bistable 555 timer circuit connection

chap 8

8. 5. Common Circuits with 555 Timer

There are hundreds of circuits realized using 555 timer but all of them are based on astable, bistable and monostable configurations. Some of them are described below.

8. 5. 1. 555 Timer as Schmitt Trigger

555 timer can be used as Schmitt trigger. Here two internal comparators are tied together and externally biased at VCC/2 through R_1 and R_2. Since the upper comparator will operate at 2/3VCC and the lower comparator at 1/3VCC the bias provided by R_1 and R_2 is centered within these two thresholds. Hence a sine wave of sufficient amplitude ($V_{in(pp)}$> VCC /3) to exceed the reference levels causes the internal flip–flop to alternately set and reset providing a square wave output.

Experiment 8-4: 555 timer Schmitt trigger

Design a Schmitt trigger based on 555 timer that outputs square wave (clean wave) from a sinusoidal wave (noisy).

Parts and materials

No	Item	Quantity
1	9V dc supply	1
2	Oscilloscope	1
3	0.01nF and 0.1nF ceramic capacitors	2
4	2x10KΩ ½ watt resistors and 5.6KΩ load resistor	3
5	0-9V/50Hz AC source (function generator)	1
6	Breadboard	1
7	Connecting wires (22-gauge solid wire)	20Cm
8	NE555 timer IC	1

chap 8

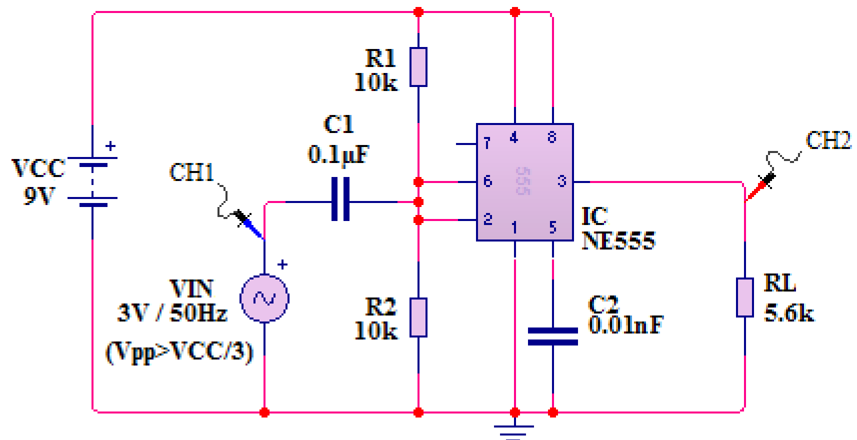


Figure 8-17: 555 timer Schmitt trigger

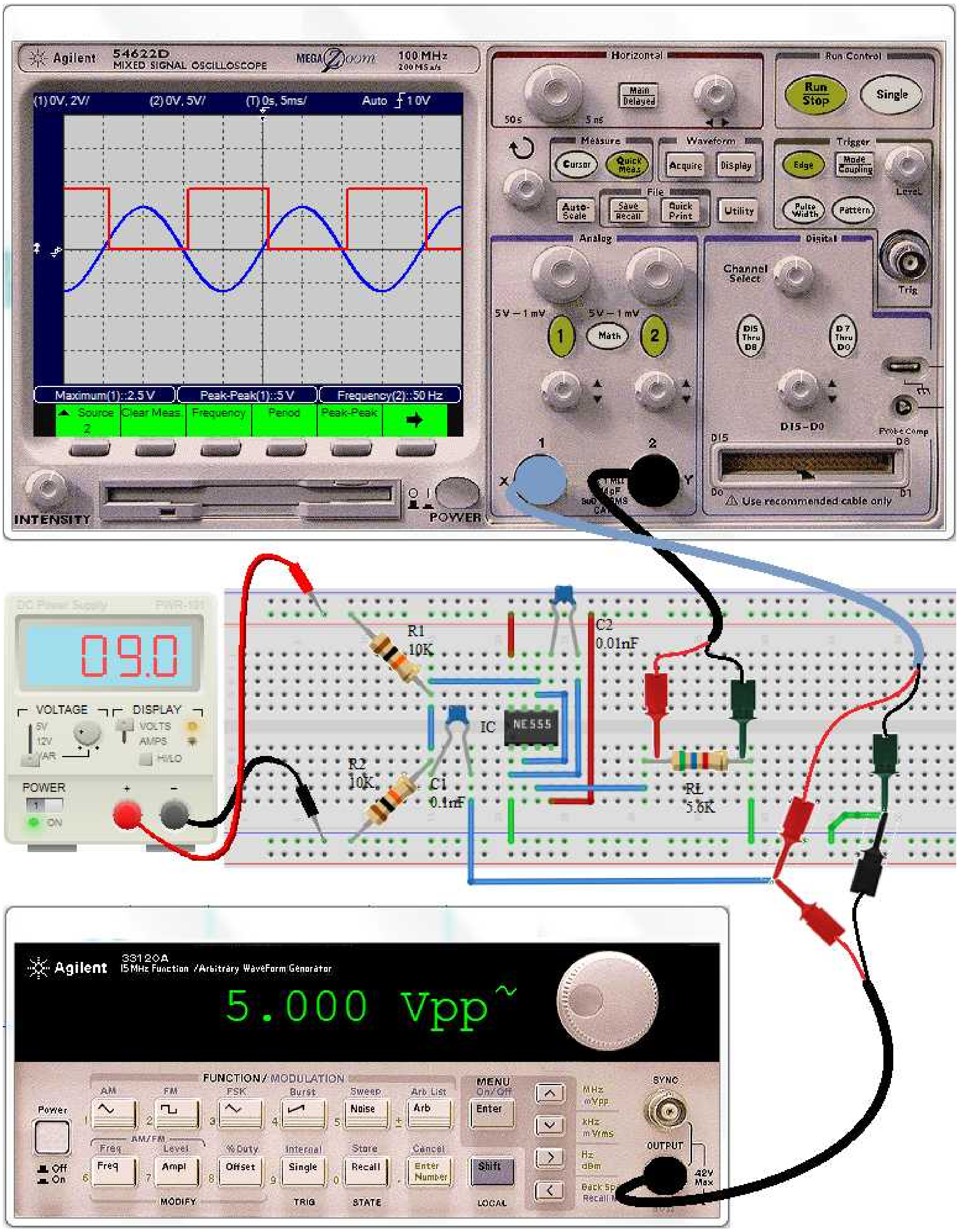


Figure 8-18: 555 timer Schmitt trigger connection circuit

chap 8

Procedure

1. Connect the components/equipment as shown in the circuit connection figure 8-18.
2. Switch ON the DC power supply.
3. Apply the input sine wave using function generator.
4. Connect the channel 1 of oscilloscope at the input terminals (on terminals function generator) and channel 2 at the output terminals (at pin 3 and ground) just on terminals of R_L.
5. Adjust the output voltage from function generator from $0V_{pp}$ up to V_{pp} equal to VCC. Observe the output square waveform corresponding to input sinusoidal signal. Overlap both the input and output waves and note down positions on sine wave where output changes its state. These positions denote the upper threshold voltage and the lower threshold voltage.
6. Verify that these practical threshold voltages are almost same as the theoretical threshold voltages calculated using formulas given in the theory section above. For this practical experiment, VCC = 9V, initially the output voltage is 0V. The output will remain 0V in case the input voltage $V_{in(peak\text{-}to\text{-}peak)}$ is less than VCC/3, i.e. $3V_{pp}$. By increasing the input voltage peak-to-peak at voltage greater than $3V_{pp}$ the output will toggle between two states (high and low) with respect to the alternation of the input voltage. By decreasing the voltage from function generator to output voltage less than $3V_{pp}$, the output will remain high in case the input goes below the threshold voltage while the output was at high state, otherwise will stay low.

chap 8

8. 5. 2. Metronome

A metronome is a piece of equipment used in the music production and it indicates the rhythm by a “toc-toc” sound which speed can be adjusted with potentiometer. This helps the singer and music player to keep the correct rhythm up. In case of high volume a transistor can be added. In most of the case a LED is also added for visual as well as audio output.

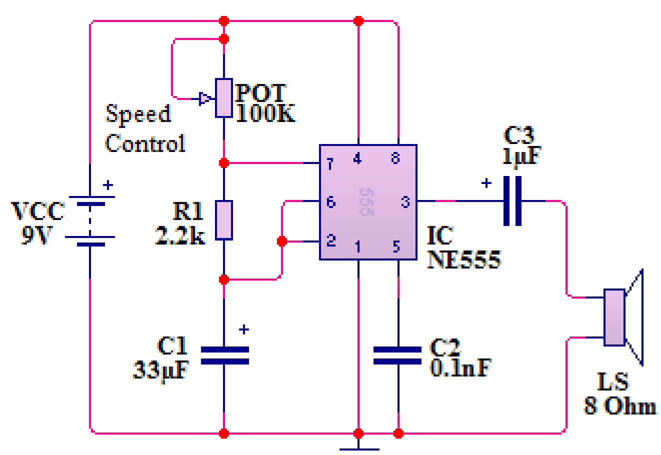


Figure 8-19: Metronome based on 555 timer

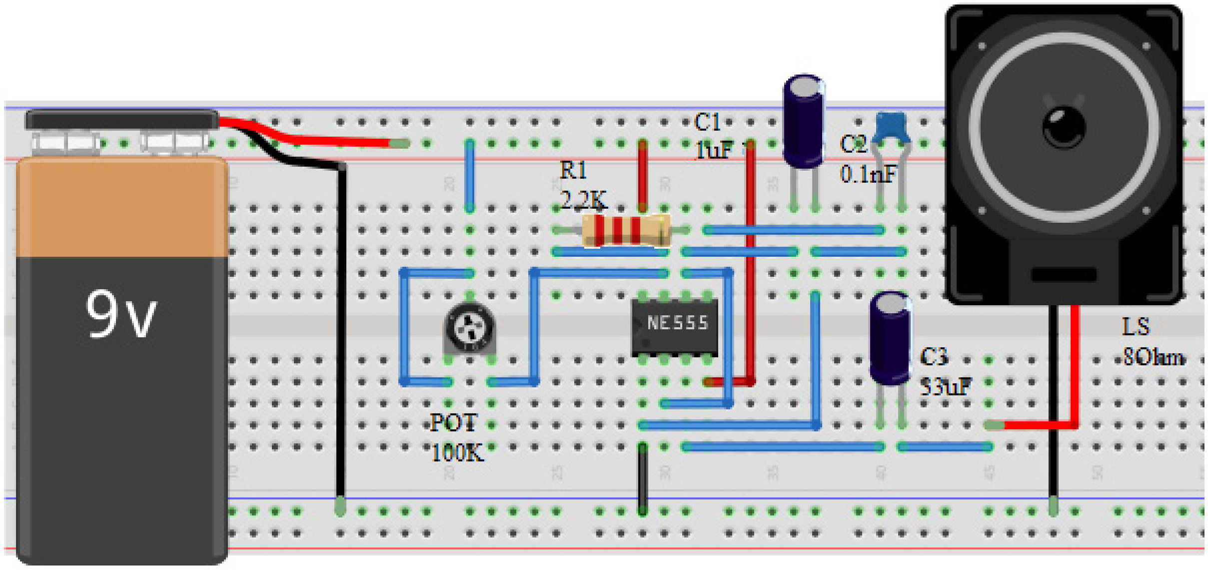
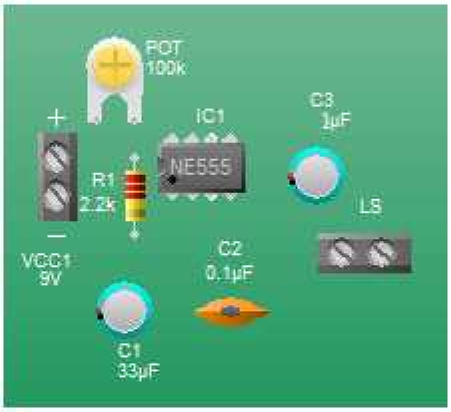
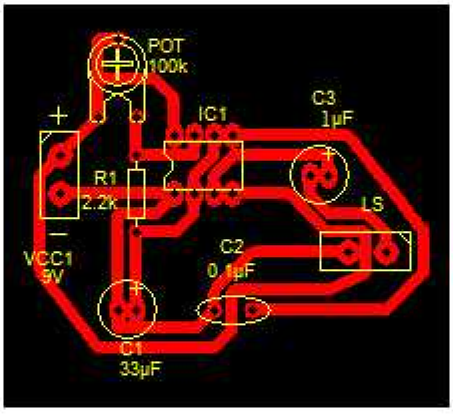


Figure 8-20: Metronome based on 555 timer, circuit connection



(a)



(b)

Figure 8-21: Metronome based on 555 timer; (a) Real world circuit, (b) Art work circuit

Note: For the safe of the potentiometer, never turn the POT to the lowest value.

8. 5. 3. Missing Pulse Detector

A basic missing pulse detector that uses 555 timer is configured as one-shot (monostable) that is continually retriggered by incoming pulses. Incoming pulses are fed on trigger input of 555 timer (pin 2) and generates a predetermined recycling one-shot output. A missing pulse that prevents retriggering before a timing cycle is complete, causes output of timer (pin 3) to go low until a new input pulse arrives. The response time is controlled by R_1C_1 combination. This is a basic model.

Working Principle of the circuit

A 555 timer IC is wired as a monostable and can detect a missing pulse or abnormally long period between two consecutive pulses in a train of pulses. The signal from the pick up transducer is shaped to form a negative going pulse and is applied to pin 2 of the IC. As long as the spacing between the pulses is less than the timing interval, the timing cycle is continuously reset by the input pulses and the capacitor is discharged via Q_1.

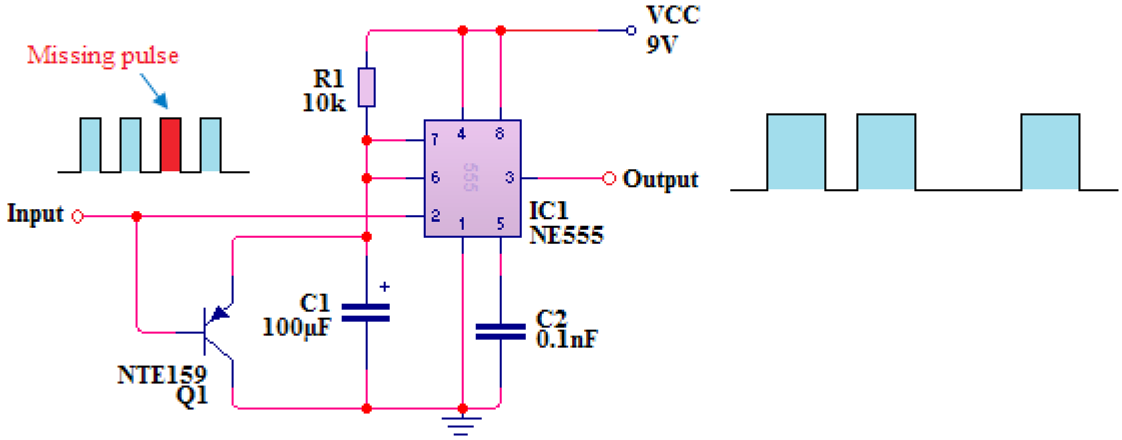


Figure 8-22: Missing pulse detector

A decrease in pulse frequency or a missing pulse permits completion of time interval which causes a change in the output level. Whenever the input goes low, it turns on transistor Q_1 which quickly drains the capacitor. When the input goes high, the capacitor recharges, but does not quite make it to the threshold voltage before the input goes low again. If the input stays high, then the capacitor is allowed to charge fully, and the 555 timing interval ends, bringing the output low.

Application of Missing Pulse Detector

Missing pulse detector is used in alarms, continuity tester and a typical example would be the “Crashed Aircraft Locator” beacon used in radio control. If there is no signal it sees this will be a missing pulse and sounds an alarm (buzzer). Such circuits can also be used to detect the intermittent firing of the spark plug of an automobile or to monitor the heart beat of a sick patient.

8. 5. 4. Burglar Alarm (dark detector)

This circuit is based on astable multivibrator that uses light sensor to activate and deactivate 555 timer. It will sound an alarm if it gets too dark all over sudden. In domestic installation there may be a place where the presence of light is a must. As the thieves always tend to still in dark they can destroy the lamps, thus this circuit could be used to notify when a lamp (or bulb) burns out or damaged. The sensing module is used to activate oscillator. The sensing module used can be a photoresistor or phototransistor to sense the absence of light and to operate a buzzer or small speaker. The light sensing enables the alarm when light falls below a certain level.

Working Principle of the Circuit

This burglar alarm (smoke detector) is designed using 555 timer circuit and other electronic components. The phototransistor is used as the light sensor while timer 555 is wired in astable configuration as an AF oscillator for sounding alarm via a buzzer. The phototransistor must be installed on position of light-way. In case of smoke detector an LED and phototransistor must be housed inside an enclosure with holes to allow entry of smoke. When the bulb is lighten up (In the absence of any smoke), the phototransistor will receive the light.

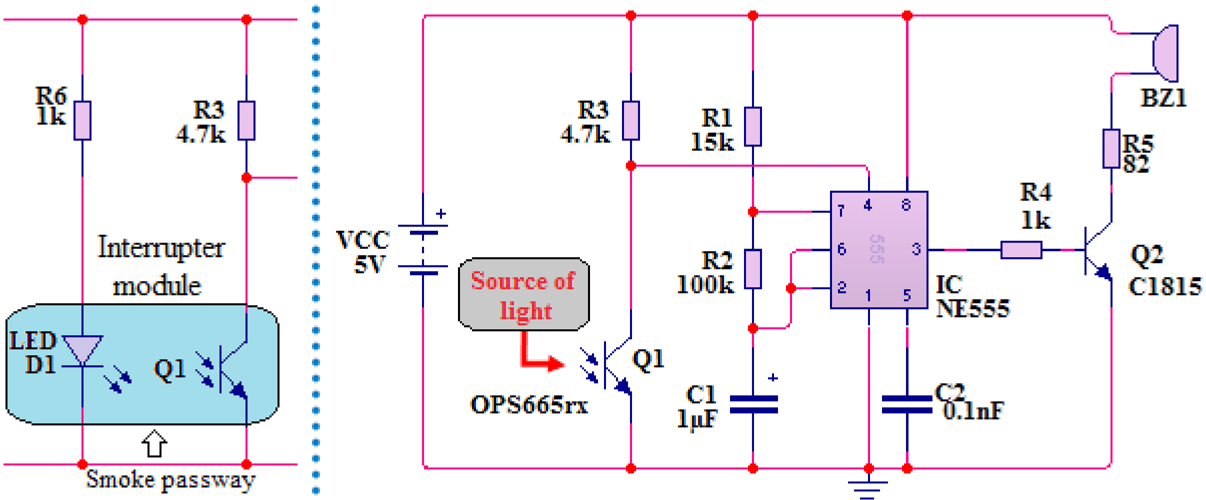


Figure 8-23: Burglar alarm (smoke detector)

For the case of smoke detector, the gap of photo interrupter module (LED and phototransistor) is clear and the light from LED falls on the phototransistor through the slot. As a result, the collector of phototransistor is pulled towards ground.

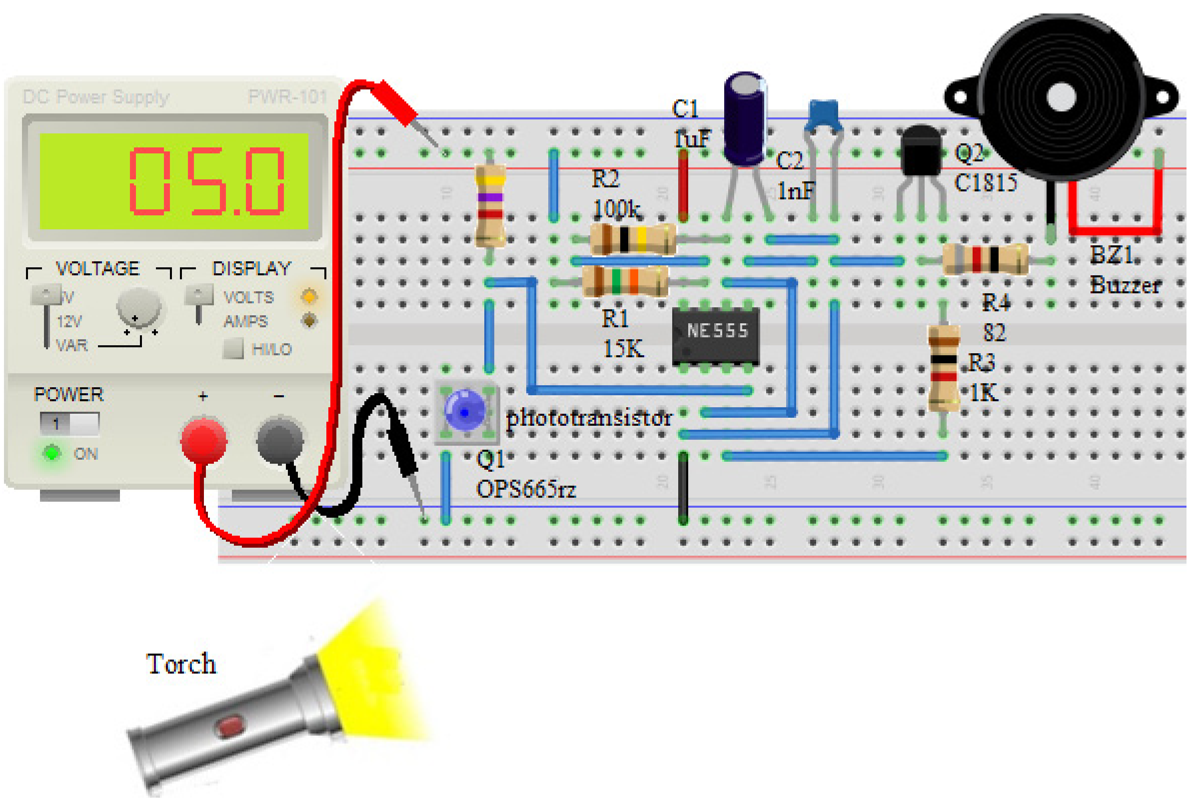


Figure 8-24: Burglar alarm (smoke detector), Circuit connection

This causes reset pin 4 of IC 555 to go low. Accordingly, the timer is reset and hence the alarm does not sound. When the bulb is turned off or the smoke is present in the gap of the photo interrupter module, the light beam from LED to the phototransistor is obstructed. As a result, the phototransistor stops conducting and pin 4 (reset) of IC 555 goes high to activate the alarm. A high voltage at this pin 4 enables 555 IC and it produces square waves continuously turning on/off the buzzer. The transistor is used for high volume.

8. 5. 5. Infrared Beam Barrier for Object Counter

The infrared beam barrier consists of infrared *transmitter* and infrared *receiver* and both circuits are installed in such way when an object passes though it cuts a beam of infrared light falling on infrared receiver. The infrared receiver is designed so that when it is receiving the infrared light, its output is activated and deactivated when an infrared beam is obstructed. One of the commonly used infrared receiver module is TSOP1738. TSOP17xx series are sensitive to different centre frequencies of the *IR* spectrum. For example TSOP1738 is sensitive to 38 kHz whereas TSOP1740 to 40 kHz centre frequency. The TSOP 1738 is a member of *IR* remote control receiver series. This *IR* sensor module consists of a PIN diode and a pre-amplifier which are embedded into a single package. The output of TSOP is active low and it gives +5V in off state. When *IR* waves, from a source, with a centre frequency of 38 kHz incident on it, its output goes low. Lights coming from sunlight, fluorescent lamps etc. may cause disturbance to it and result in undesirable output even when the source is not transmitting *IR* signals. A band-pass filter, an integrator stage and an automatic gain control are used to suppress such disturbances.

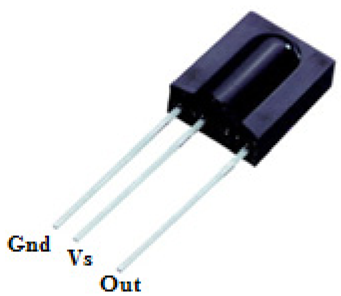


Figure 8-25: Typical TSOP1738 infrared receiver module

chap 8

The infrared transmitter uses NE555 integrated circuit configured as astable oscillator and must be configured in such way that it falls in frequency range of the infrared receiver. The receiver requires the frequency ranging in 37 KHz to 39 KHz.

When an object passes through the infrared light, just between infrared transmitter and infrared receiver, it cuts infrared light beam and the output of TSOP1738 goes low. Thus, a negative pulse is obtained at pin 2 of NE555. Due to this NE555 wired as a monostable multivibrator produces high signal at its output which makes the transistor Q_1 to go in saturation and makes the clock pin of counter IC 74192 to go at low state. A negative going received on clock pin of counter make the counter to increment by one on 7-segment display. Capacitor C_5 prevents false triggering. The time interval between two consecutive objects has to be greater than time constant determined by R_4 and C_4.

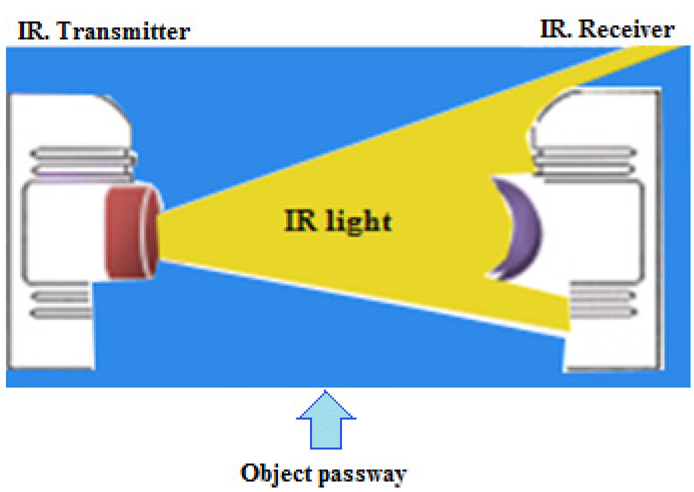


Figure 8-26: Infrared transmitter and infrared receiver arrangement for beam barrier

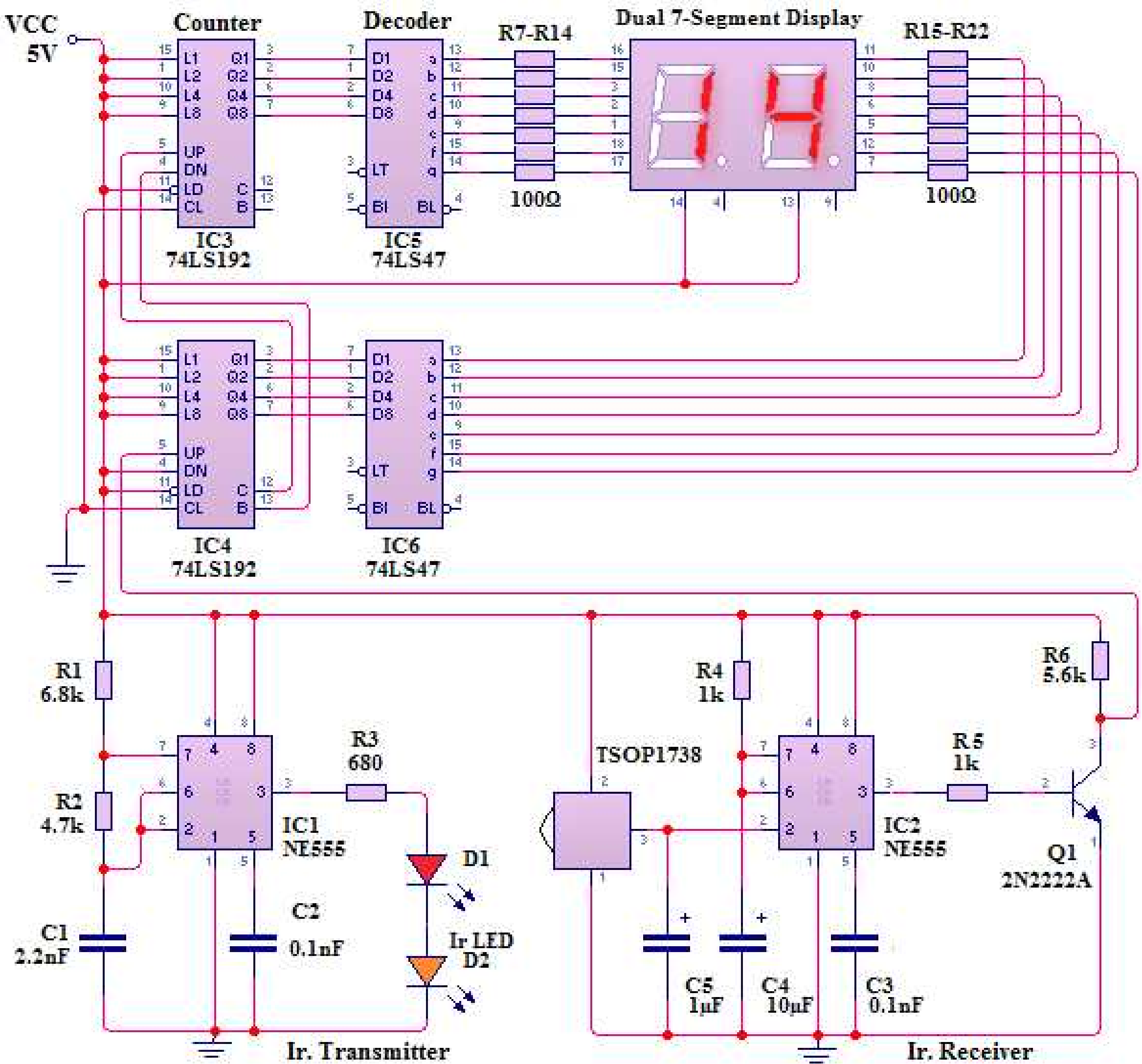


Figure 8-27: Beam barrier circuits for object counter

8. 5. 6. Pulse Width Modulation (PWM) with 555 Timer

Pulse width modulation is the process of switching the power to a device on and off at a given frequency, with varying on and off times. The overall power supplied to a device can be varied by controlling the ratio that device is on to the time it is off. For example, if a motor was powered from a 9V power supply and 50% of the time the power supplied was off and 50% was on then the average voltage the motor receives is 4.5V making it run half as fast. The ratio of the time a device is on to off can be expressed as a percentage as in the example above and the percentage a device is on compared to off is called the "*duty cycle*". This percentage corresponds to actual times for example a 50% on to 50% off can correspond to 500ms on to 500ms off and this would create a frequency of 1 hertz.

The circuit figure 8-28 is based on astable multivibrator configuration. The 555 timer astable arrangement creates a square wave with time on and time off. The ratio of these times can be varied by changing R_1 and/or POT_1 or C_1. Note the configuration of POT_1, D_1, and D_2. Capacitor C_1

charges through one side of POT_1 and D_1 and discharges through D_1 and the other side of POT_1. The sum of the charge and discharge resistance is always the same; therefore the wavelength and frequency of the output signal are constant. Only the duty cycle varies with POT_1.

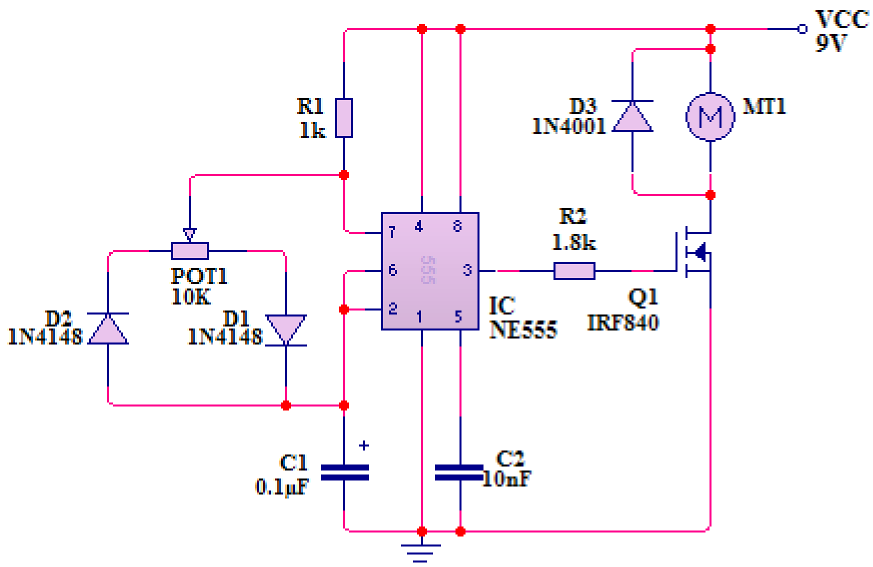


Figure 8-28: Pulse Width Modulation using 555 timer

So the potentiometer allows you to vary the width of the pulses and changing the capacitor C_1 from 100uF to 0.01uF allowing you to change the frequency of the output from around 1Hz to 10Khz giving it a range of possible uses from LED's to motors.

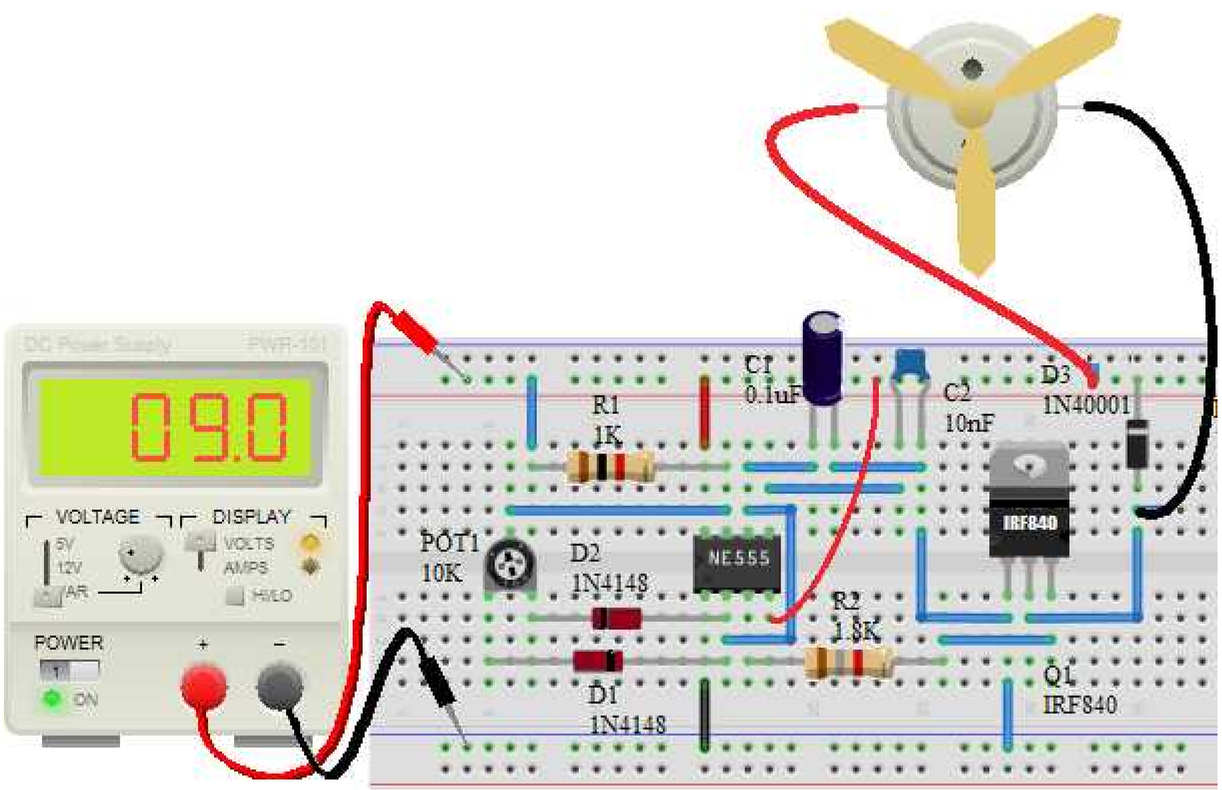
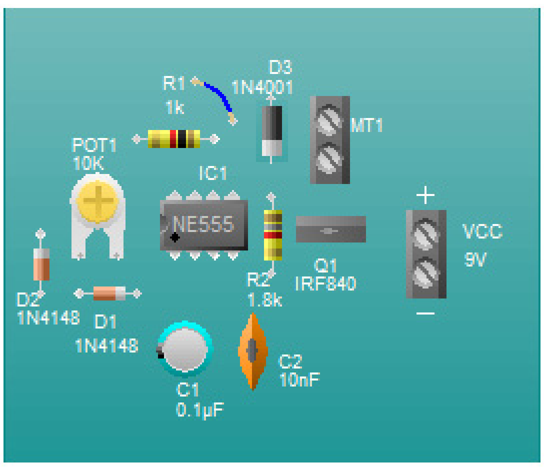


Figure 8-29: Pulse Width Modulation using 555 timer circuit connection

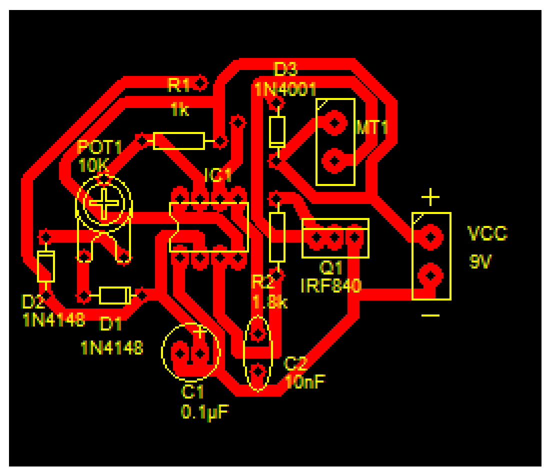
When using PWM the frequency supplied to a component needs to be high enough to where it is not noticed. For example a motor is typically over a few thousands hertz as anything below and the motor would not rotate smoothly. This would mean the percentages of 50% on to 50% off would correspond to say 0.25ms on to 0.25ms off which would supply 4.5V from 9V and a frequency of 2000 hertz which is high enough for the motor to run smoothly. For example a frequency of 10Hz would be noticed as a flicker.

That is why when controlling LEDs a few hundred hertz is suitable. When controlling fans and motors you can sometimes hear a whistling sound which is the coils inside vibrating at a frequency we can hear, it causes no damage to the motor however can be annoying; so an increase in frequency would remove the noise.

For this experimental circuit you can use oscilloscope to view graphic oscillation and pulse width change as you change the resistance of potentiometer. Just connect the oscilloscope on pin 3 of 555 IC. Measure the frequency and duty cycle of the resulting pulses. You will find out that the frequency is kept constant all along the change of time on and time off of the pulse. For duty cycle nearly to 5% the motor MT_1 rotates at low speed while at duty cycle close to 98% the motor rotate at nearly to its nominal rpm.



(a)



(b)

Figure 8-30: Pulse Width Modulation using 555 timer. (a) Real world circuit on PCB, (b) Artwork circuit on PCB

8. 5. 7. Touch Switch with 555 Timer

The touch switch depicts the concept of touching a plate (or two plates separated by a small gap) and turning a circuit ON or OFF. A touch plate is considered as a high impedance device (or high impedance circuit) as the effect of a finger will be detected by the circuit connected to the plate. The modern mechanic switches are improved concerning of old technology. We however need to check current greater than the durability of certain switches or simply we need something with modern appearance. The electronics switch have advantages over the mechanical switches such as high speed switching (over 1000 times per second), high efficiency, no movable parts, no sparks, small size and they are light weight. The touch switch described here is built around a 555 timer configured in such a way that latch is on one state and some action is required to change that state.

The input impedance of the trigger is very high and the touch sensor switch can be triggered by the voltage induced in the human body. When you touch the plates on either side of the gap you connect the circuit through your skin.

A very small current, imperceptible to a human, flows through across your skin when you bridge the circuit over the plates. Don't worry about being electrocuted, the current is so low you will not notice it.

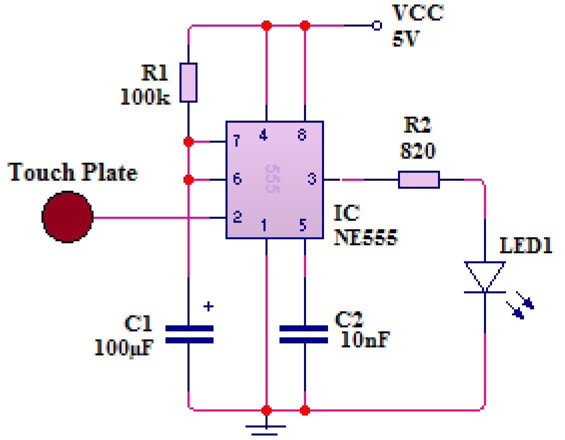


Figure 8-31: Timed on touch switch

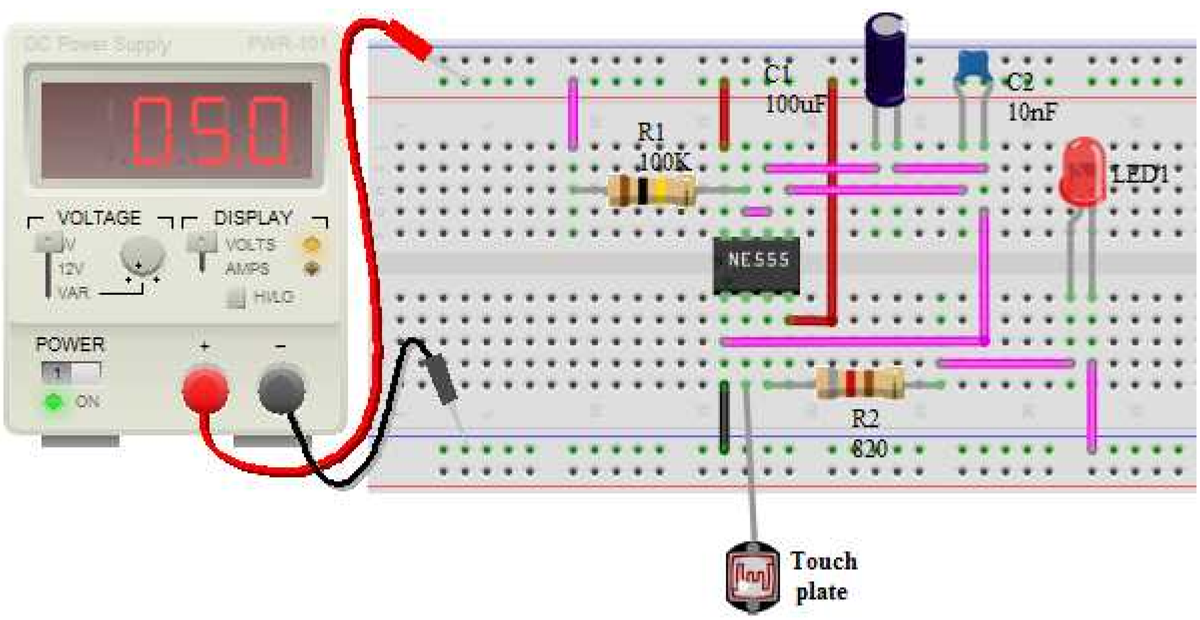


Figure 8-32: Timed on touch switch, circuit connection

The circuit figure 8-33 is configured to flip in two states. A pull up resistor R_1 keeps pin 2 at high while pull down resistor R_2 keeps low voltage (ground) at pin 6. The plates are made with conductive material installed in side by side of each other so that when the finger touches the gap between the plates it connects the circuit by current passing through the skin. Remember that human being is a good conductor of electricity. The 555 timer senses voltage change at pin 6 and pin 2, depending on which plate you touched and flips the output state at pin 3 turning the LED/lamp BL_1 ON or OFF. The ON touch plates is used to turn on the LED_1/lamp BL_1 while the OFF touch plates is used to turn off the LED_1/lamp BL_1. When your finger touches the on touch plates you connect the trigger pin (pin 2) to the ground through your skin. A low voltage receives on pin 2 of 555 timer changes the output (pin 3) state to be high (about to VCC voltage). This makes the LED_1 on and transistor to be driven in saturation and relay RL_1 changes the state to switch on the lamp BL_1.

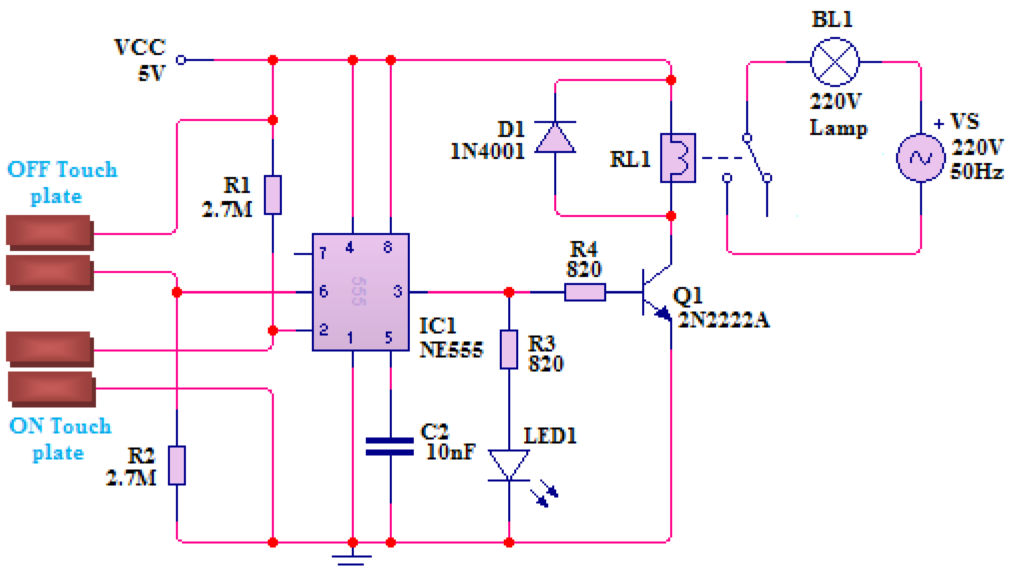


Figure 8-33: ON/OFF touch switch

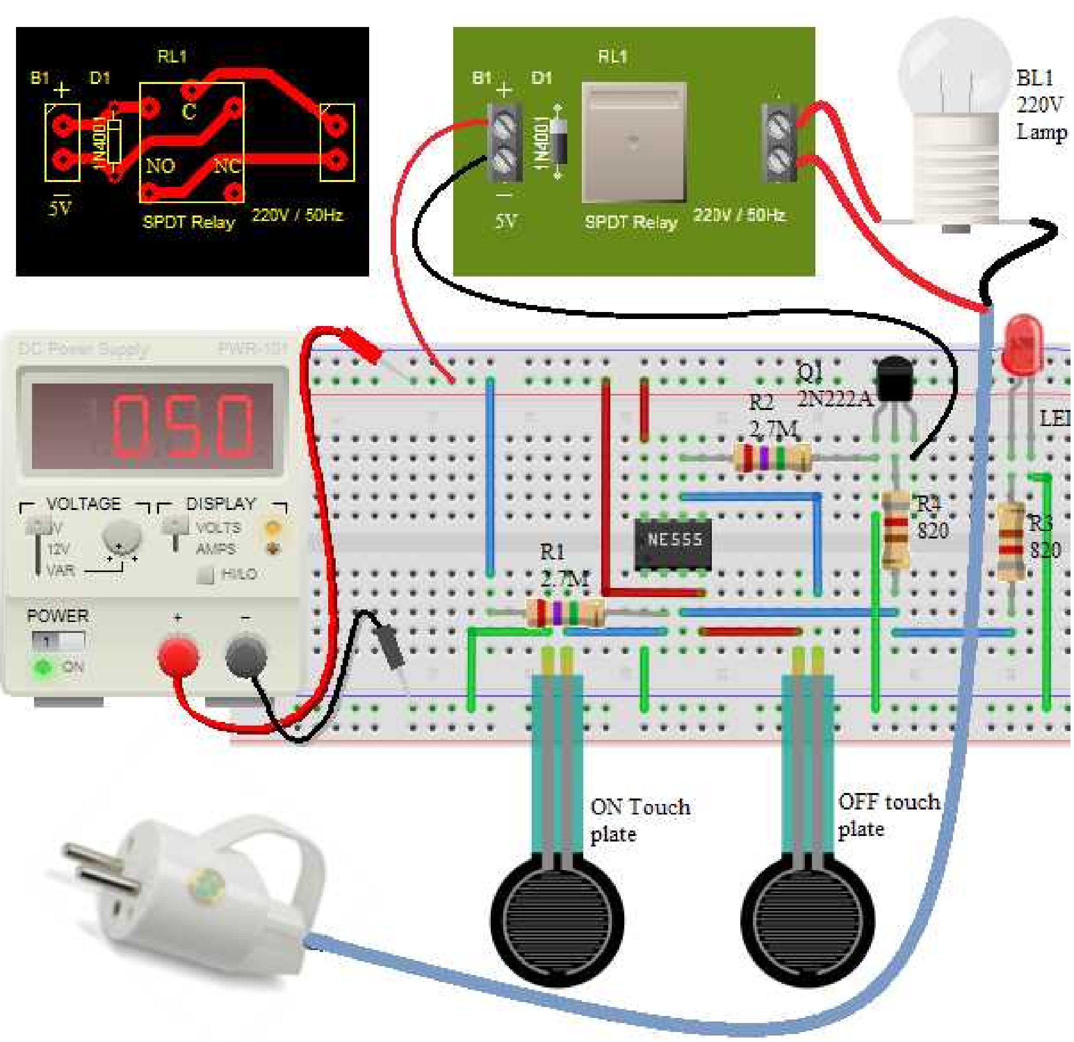
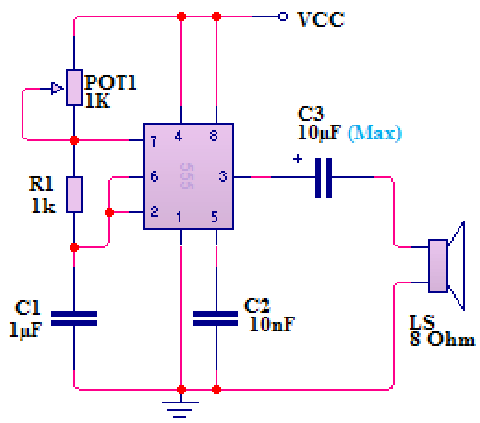
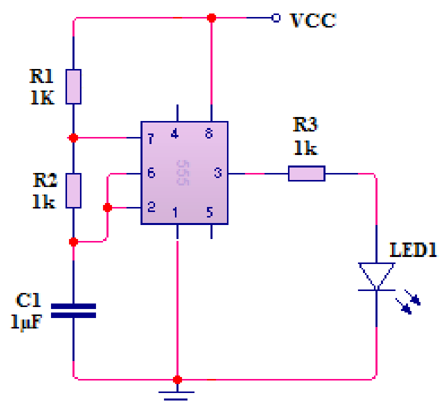


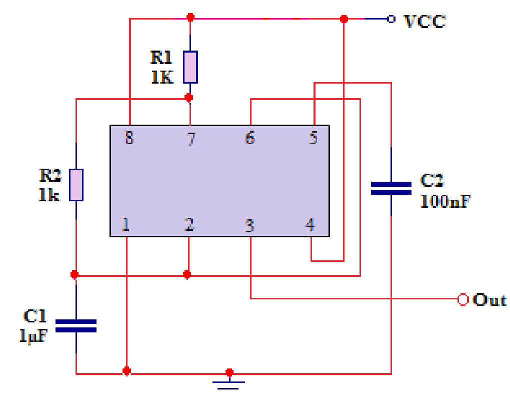
Figure 8-34: ON/OFF touch switch circuit connection

8. 6. 555 Timer Circuits' Practical Considerations



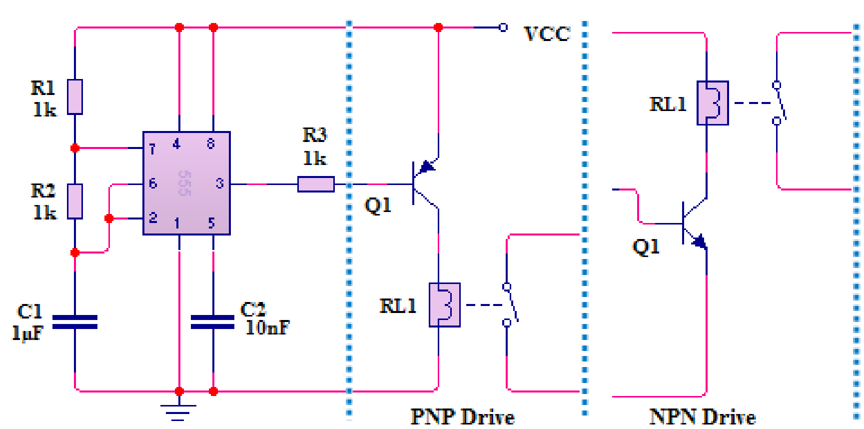
- During part of operation pin 7 gets connected to ground (0V) via internal transistor of 555 timer chip. So, avoid turning the POT_1 from the following circuit to very low resistance because a high current will flow through the POT_1 and it will be damaged.
- The impedance of the high electrolytic capacitor connected to the output of 555 timer will allow a very high current to flow and the chip will get very hot. Use 10μF maximum when using 8Ω speaker.
- Inside 555 timer chip the reset pin (pin 4) is tied high via about 100KΩ but it should not be left floating as stray pulses may reset the IC.





chap 8

- Keep away from drawing 555 circuits as in any layout you want like that shown above on right. It takes a lot of time to know what the circuit is standing for. The main purpose is to lay out a circuit so that it shows instantly what is happening. That is why everything must be in recognized locations. From this diagram it is obvious the circuit is an oscillator. So keep to a standard layout so the circuit is easy to follow.
- Do not use high value electrolytic capacitor and high resistance to produce long delays. Use time delay less than 5 minutes to avoid time-out before calculated time.
- Do not connect a PNP to the output of a 555 as shown in the following circuit diagram. Pin 3 does not raise high enough to turn off the transistor and the current taken by the circuit will be excessive. Use an NPN driver instead.



Apandix

Devices Datasheet

TEXAS INSTRUMENT

LM741 Operational Amplifier

FEATURES

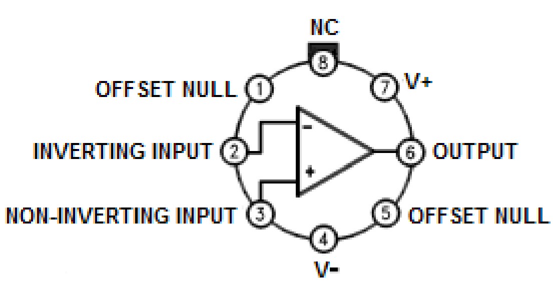
- Overload Protection on the Input and Output
- No Latch-Up When the Common Mode Range is exceeded

DESCRIPTION

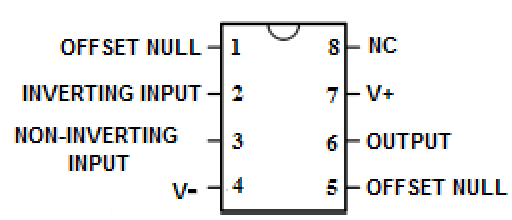
The LM741 series are general purpose operational amplifiers which feature improved performance over industry standards like the LM709. They are direct, plug-in replacements for the 709C, LM201, MC1439 and 748 in most applications. The amplifiers offer many features which make their application nearly foolproof: overload protection on the input and output, no latch-up when the common mode range is exceeded, as well as freedom from oscillations.

The LM741C is identical to the LM741/LM741A except that the LM741C has their performance ensured over a 0°C to +70°C temperature range, instead of −55°C to +125°C.

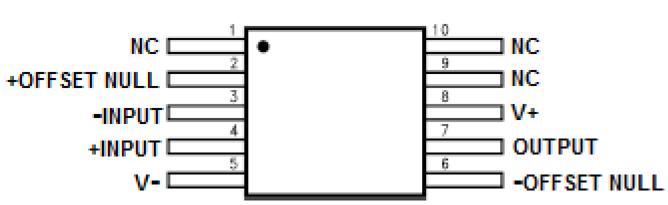
Connection diagrams



TO-99 PACKAGE



CDIP or PDIP PACKAGE



CLGA PACKAGE

Absolute Maximum Ratings (1)(2)(3)

	LM741A	LM741	LM741C
Supply Voltage	±22V	±22V	±18V
Power Dissipation (4)	500 mW	500 mW	500 mW
Differential Input Voltage	±30V	±30V	±30V
Input Voltage (5)	±15V	±15V	±15V
Output Short Circuit Duration	Continuous	Continuous	Continuous
Operating Temperature Range	−55°C to +125°C	−55°C to +125°C	0°C to +70°C
Storage Temperature Range	−65°C to +150°C	−65°C to +150°C	−65°C to +150°C
Junction Temperature	150°C	150°C	100°C
Soldering Information			
P0008E-Package (10 seconds)	260°C	260°C	260°C
NAB0008A- or LMC0008C-Package (10 seconds)	300°C	300°C	300°C
M-Package			
Vapor Phase (60 seconds)	215°C	215°C	215°C
Infrared (15 seconds)	215°C	215°C	215°C
ESD Tolerance (6)	400V	400V	400V

(1) "Absolute Maximum Ratings" indicate limits beyond which damage to the device may occur. Operating Ratings indicate conditions for which the device is functional, but do not ensure specific performance limits.
(2) For military specifications see RETS741X for LM741 and RETS741AX for LM741A.
(3) If Military/Aerospace specified devices are required, please contact the TI Sales Office/Distributors for availability and specifications.
(4) For operation at elevated temperatures, these devices must be derated based on thermal resistance, and T_j max. (listed under "Absolute Maximum Ratings"). $T_j = T_A + (\theta_{jA} P_D)$.
(5) For supply voltages less than ±15V, the absolute maximum input voltage is equal to the supply voltage.
(6) Human body model, 1.5 kΩ in series with 100 pF

Electrical Characteristics (1)

Parameter	Test Conditions	LM741A			LM741			LM741C			Units
		Min	Typ	Max	Min	Typ	Max	Min	Typ	Max	
Input Offset Voltage	T_A = 25°C R_S ≤ 10 kΩ R_S ≤ 50Ω		0.8	3.0		1.0	5.0		2.0	6.0	mV
	T_{AMIN} ≤ T_A ≤ T_{AMAX} R_S ≤ 50Ω R_S ≤ 10 kΩ			4.0			6.0			7.5	mV
Average Input Offset Voltage Drift				15							μV/°C
Input Offset Voltage Adjustment Range	T_A = 25°C, V_S = ±20V	±10				±15			±15		mV
Input Offset Current	T_A = 25°C		3.0	30		20	200		20	200	nA
	T_{AMIN} ≤ T_A ≤ T_{AMAX}			70		85	500			300	
Average Input Offset Current Drift				0.5							nA/°C
Input Bias Current	T_A = 25°C		30	80		80	500		80	500	nA
	T_{AMIN} ≤ T_A ≤ T_{AMAX}			0.210			1.5			0.8	μA

Input Resistance	T_A = 25°C, V_S = ±20V	1.0	6.0		0.3	2.0		0.3	2.0		MΩ
	$T_{AMIN} \le T_A \le T_{AMAX}$, V_S = ±20V	0.5									
Input Voltage Range	T_A = 25°C							±12	±13		V
	$T_{AMIN} \le T_A \le T_{AMAX}$				±12	±13					
Large Signal Voltage Gain	T_A = 25°C, $R_L \ge$ 2 kΩ										V/mV
	V_S = ±20V, V_O = ±15V	50									
	V_S = ±15V, V_O = ±10V				50	200		20	200		
	$T_{AMIN} \le T_A \le T_{AMAX}$, $R_L \ge$ 2 kΩ,										V/mV
	V_S = ±20V, V_O = ±15V	32									
	V_S = ±15V, V_O = ±10V				25			15			
	V_S = ±5V, V_O = ±2V	10									
Output Voltage Swing	V_S = ±20V $R_L \ge$ 10 kΩ	±16									V
	$R_L \ge$ 2 kΩ	±15									
	V_S = ±15V $R_L \ge$ 10 kΩ				±12	±14		±12	±14		V
	$R_L \ge$ 2 kΩ				±10	±13		±10	±13		
Output Short Circuit Current	T_A = 25°C	10	25	35		25			25		mA
	$T_{AMIN} \le T_A \le T_{AMAX}$	10		40							
Common-Mode Rejection Ratio	$T_{AMIN} \le T_A \le T_{AMAX}$										dB
	$R_S \le$ 10 kΩ, V_{CM} = ±12V				70	90		70	90		
	$R_S \le$ 50Ω, V_{CM} = ±12V	80	95								
Supply Voltage Rejection Ratio	$T_{AMIN} \le T_A \le T_{AMAX}$,										dB
	V_S = ±20V to V_S = ±5V										
	$R_S \le$ 50Ω	86	96								
	$R_S \le$ 10 kΩ				77	96		77	96		
Transient Response	T_A = 25°C, Unity Gain										
Rise Time			0.25	0.8		0.3			0.3		µs
Overshoot			6.0	20		5			5		%
Bandwidth (2)	T_A = 25°C	0.437	1.5								MHz
Slew Rate	T_A = 25°C, Unity Gain	0.3	0.7			0.5			0.5		V/µs
Supply Current	T_A = 25°C					1.7	2.8		1.7	2.8	mA
Power Consumption	T_A = 25°C										mW
	V_S = ±20V		80	150							
	V_S = ±15V					50	85		50	85	
LM741A	V_S = ±20V										mW
	T_A = T_{AMIN}			165							
	T_A = T_{AMAX}			135							
LM741	V_S = ±15V										mW
	T_A = T_{AMIN}					60	100				
	T_A = T_{AMAX}					45	75				

(1) Unless otherwise specified, these specifications apply for V_S = ±15V, −55°C ≤ T_A ≤ +125°C (LM741/LM741A). For the LM741C/LM741E, these specifications are limited to 0°C ≤ T_A ≤ +70°C.

(2) Calculated value from: BW (MHz) = 0.35/Rise Time (µs).

Thermal Resistance	CDIP (NAB0008A)	PDIP (P0008E)	TO-99 (LMC0008C)	SO-8 (M)
θ_{jA} (Junction to Ambient)	100°C/W	100°C/W	170°C/W	195°C/W
θ_{jC} (Junction to Case)	N/A	N/A	25°C/W	N/A

TEXAS INSTRUMENT

LM224K, LM224KA, LM324, LM324A, LM324K, LM324KA, LM2902, LM124, LM124A, LM224, LM224A, LM2902V, LM2902K, LM2902KV, LM2902KAV

LMx24, LMx24x, LMx24xx, LM2902, LM2902x, LM2902xx, LM2902xxx Quadruple Operational Amplifiers

Features

- 2-kV ESD Protection for:
  - LM224K, LM224KA
  - LM324K, LM324KA
  - LM2902K, LM2902KV, LM2902KAV
- Wide Supply Ranges
  - Single Supply: 3 V to 32 V (26 V for LM2902)
  - Dual Supplies: ±1.5 V to ±16 V (±13 V for LM2902)
- Low Supply-Current Drain Independent of Supply Voltage: 0.8 mA Typical
- Common-Mode Input Voltage Range Includes Ground, Allowing Direct Sensing Near Ground
- Low Input Bias and Offset Parameters
  - Input Offset Voltage: 3 mV Typical
    A Versions: 2 mV Typical
  - Input Offset Current: 2 nA Typical
  - Input Bias Current: 20 nA Typical
    A Versions: 15 nA Typical
- Differential Input Voltage Range Equal to Maximum rated supply voltage: 32V (26V for LM2902)
- Open-Loop Differential Voltage Amplification: 100 V/mV Typical
- Internal Frequency Compensation
- On Products Compliant to MIL-PRF-38535, All Parameters are Tested Unless Otherwise Noted. On All Other Products, Production Processing Does Not Necessarily Include Testing of All Parameters.

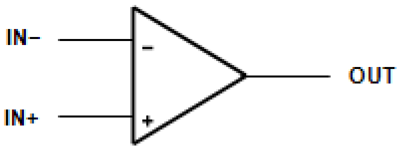
Applications

- Blu-ray Players and Home Theaters
- Chemical and Gas Sensors
- Chemical and Gas Sensors
- Digital Multimeter: Bench and Systems
- Digital Multimeter: Handhelds
- Field Transmitter: Temperature Sensors
- Motor Control: AC Induction, Brushed DC, Brushless DC, High-Voltage, Low-Voltage, Permanent Magnet, and Stepper Motor
- Oscilloscopes
- TV: LCD and Digital
- Temperature Sensors or Controllers Using Modbus
- Weigh Scales

Description

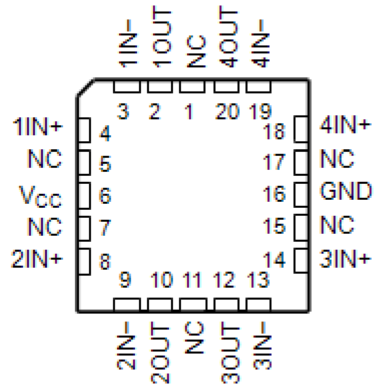
These devices consist of four independent high-gain frequency-compensated operational amplifiers that are designed specifically to operate from a single supply or split supply over a wide range of voltages.

Symbol (each amplifier)

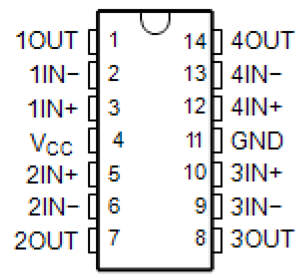


Pin configulation and function

FK Package
20-Pin LCCC
(Top View)



D, DB, J, N, NS, PW, W
14-Pin SOIC, SSOP, CDIP,
PDIP, SO, TSSOP, CFP
(Top View)



Function

PIN			I/O	DESCRIPTION
NAME	LCCC NO.	SOIC, SSOP, CDIP, PDIP, SO, TSSOP, CFP NO.		
1IN–	3	2	I	Negative input
1IN+	4	3	I	Positive input
1OUT	2	1	O	Output
2IN–	9	6	I	Negative input
2IN+	8	5	I	Positive input
2OUT	10	7	O	Output
3IN–	13	9	I	Negative input
3IN+	14	10	I	Positive input
3OUT	12	8	O	Output
4IN–	19	13	I	Negative input
4IN+	18	12	I	Positive input
4OUT	20	14	O	Output
GND	16	11	—	Ground
NC	1 5 7 11 15 17	—	—	Do not connect
VCC	6	4	—	Power supply

Absolute Maximum Ratings

Over operating free-air temperature range (unless otherwise noted)[1]

		LM2902		LMx24, LMx24x, LMx24xx, LM2902x, LM2902xx, LM2902xxx		UNIT
		MIN	MAX	MIN	MAX	
Supply voltage, VCC[2]		±13	26	±16	32	V
Differential input voltage, VID[3]			±26		±32	V
Input voltage, VI (either input)		–0.3	26	–0.3	to 32	V
Duration of output short circuit (one amplifier) to ground at (or below) TA = 25°C, VCC ≤ 15 V[4]		Unlimited		Unlimited		
Operating virtual junction temperature, TJ			150		150	°C
Case temperature for 60 seconds	FK package				260	°C
Lead temperature 1.6 mm (1/16 inch) from case for 60 seconds	J or W package		300		300	°C
Storage temperature, Tstg		–65	150	–65	150	°C

(1) Stresses beyond those listed under *Absolute Maximum Ratings* may cause permanent damage to the device. These are stress ratings only, and functional operation of the device at these or any other conditions beyond those indicated under *Recommended Operating Conditions* is not implied. Exposure to absolute-maximum-rated conditions for extended periods may affect device reliability.
(2) All voltage values (except differential voltages and V_{CC} specified for the measurement of I_{OS}) are with respect to the network GND. (3) Differential voltages are at IN+, with respect to IN−.
(4) Short circuits from outputs to VCC can cause excessive heating and eventual destruction.

ESD Ratings

			VALUE	UNIT
LM224K, LM224KA, LM324K, LM324KA, LM2902K, LM2902KV, LM2902KAV				
V(ESD)	Electrostatic discharge	Human-body model (HBM), per ANSI/ESDA/JEDEC JS-001[1]	±2000	V
		Charged-device model (CDM), per JEDEC specification JESD22-C101	±1000	
LM124, LM124A, LM224, LM224A, LM324, LM324A, LM2902, LM2902V				
V(ESD)	Electrostatic discharge	Human-body model (HBM), per ANSI/ESDA/JEDEC JS-001[1]	±500	V
		Charged-device model (CDM), per JEDEC specification JESD22-C101	±1000	

(1) JEDEC document JEP155 states that 500-V HBM allows safe manufacturing with a standard ESD control process.

Recommended Operating Conditions

Over operating free-air temperature range (unless otherwise noted)

MIN	LM2902		LMx24, LMx24x, LMx24xx, LM2902x, LM2902xx, LM2902xxx		UNIT
	MAX	MIN	MAX		
V_{CC} Supply voltage	3	26	3	30	V
V_{CM} Common-mode voltage	0	VCC – 2	0	VCC – 2	V

T_A Operating free air temperature	LM124			–55	125	°C
	LM2904	–40	125			
	LM324			0	70	
	LM224			–25	85	

Thermal Information

THERMAL METRIC(1)		LMx24, LM2902					LMx24			UNIT
		D (SOIC)	DB (SSOP)	N (PDIP)	NS (SO)	PW (TSSOP)	FK (LCCC)	J (CDIP)	W (CFP)	
		14 PINS	14 PINS	14 PINS	14 PINS	14 PINS	20 PINS	14 PINS	14 PINS	
$R_{\theta JA}$ (2)(3)	Junction-to-ambient thermal resistance	86	86	80	76	113	—	—	—	°C/W
$R_{\theta JC}$ (4)	Junction-to-case (top) thermal resistance	—	—	—	—	—	5.61	15.05	14.65	°C/W

(1) For more information about traditional and new thermal metrics, see the *IC Package Thermal Metrics* application report, SPRA953. (2) Short circuits from outputs to VCC can cause excessive heating and eventual destruction.

(3) Maximum power dissipation is a function of $T_{J(max)}$, $R_{\theta JA}$, and T_A. The maximum allowable power dissipation at any allowable ambient temperature is $P_D = (T_{J(max)} – T_A)/R_{\theta JA}$. Operating at the absolute maximum T_J of 150°C can affect reliability.

(4) Maximum power dissipation is a function of $T_{J(max)}$, $R_{\theta JA}$, and T_C. The maximum allowable power dissipation at any allowable case temperature is $P_D = (T_{J(max)} – T_C)/R_{\theta JC}$. Operating at the absolute maximum T_J of 150°C can affect reliability.

Electrical Characteristics for LMx24 and LM324K

at specified free-air temperature, V_{CC} = 5 V (unless otherwise noted

PARAMETER		TEST CONDITION(1)	T_A(2) MIN	LM124, LM224 TYP(3)	MAX	MIN	LM324, LM324K TYP(3)	MAX		UNIT
V_{IO}	Input offset voltage	V_{CC} = 5 V to MAX, V_{IC} = V_{ICR}min, V_O = 1.4 V	25°C		3	5		3	7	mV
			Full range		7			9		
I_{IO}	Input offset current	V_O = 1.4 V	25°C		2	30		2	50	nA
			Full range		100			150		
I_{IB}	Input bias current	V_O = 1.4 V	25°C		-20	-150		-20	-250	nA
			Full range		-300			-500		
V_{ICR}	Common-mode input voltage range	V_{CC} = 5 V to MAX	25°C	0 to V_{CC} – 1.5			0 to V_{CC} – 1.5			V
			Full range	0 to V_{CC} – 2		0 to V_{CC} – 2				
V_{OH}	High-level output voltage	R_L = 2 kΩ	25°C	V_{CC} – 1.5			V_{CC} – 1.5			V
		R_L = 10 kΩ	25°C							
		V_{CC} = MAX, R_L = 2 kΩ	Full range	26			26			
		R_L ≥ 10 kΩ	Full range	27	28		27	28		

V_{OL} Low-level output voltage	$R_L \leq 10$ kΩ		Full range		5	20		5	20	mV
A_{VD} Large-signal differential voltage amplification	$V_{CC} = 15$ V, $V_O = 1$ V to 11 V, $R_L \geq 2$ kΩ		25°C	50	100		25	100		V/mV
			Full range	25		15				dB
CMRR Common-mode rejection ratio	$V_{IC} = V_{ICR}$min		25°C	70	80		65	80		dB
k_{SVR} Supply-voltage rejection ratio ($\Delta V_{CC}/\Delta V_{IO}$)			25°C	65	100		65	100		dB
V_{O1}/V_{O2} Crosstalk attenuation	f = 1 kHz to 20 kHz		25°C		120			120		dB
I_O Output current	$V_{CC} = 15$ V, $V_{ID} = 1$ V, $V_O = 0$,	Source	25°C	-20	-30	-60	-20	-30	-60	mA
			Full range	-10			-10			
	$V_{CC} = 15$ V, $V_{ID} = -1$ V, $V_O = 15$ V,	Sink	25°C	10	20		10	20		
			Full range	5			5			
	$V_{ID} = -1$ V, $V_O = 200$ mV		25°C	12	30		12	30		μA
I_{OS} Short-circuit output current	V_{CC} at 5 V, $V_O = 0$, GND at –5 V		25°C		±40	±60		±40	±60	mA
I_{CC} Supply current (four amplifiers)	$V_O = 2.5$ V, no load		Full range		0.7	1.2		0.7	1.2	mA
	V_{CC} = MAX, $V_O = 0.5\ V_{CC}$, no load		Full range		1.4	3		1.4	3	

(1) All characteristics are measured under open-loop conditions, with zero common-mode input voltage, unless otherwise specified. MAX V_{CC} for testing purposes is 26 V for LM2902 and 30 V for the others.
(2) Full range is –55°C to 125°C for LM124, –25°C to 85°C for LM224, and 0°C to 70°C for LM324.
(3) All typical values are at T_A = 25°C

Electrical Characteristics for LM2902 and LM2902V

at specified free-air temperature, V_{CC} = 5 V (unless otherwise noted)

PARAMETER	TEST CONDITIONS(1)		TA(2)	LM2902			LM2902V			UNIT
				MIN	TYP(3)	MAX	MIN	TYP(3)	MAX	
V_{IO} Input offset voltage	V_{CC} = 5 V to MAX, $V_{IC} = V_{ICR}$min, V_O = 1.4 V	Non-A-suffix devices	25°C		3	7		3	7	mV
			Full range			10			10	
		A-suffix devices	25°C					1	2	
			Full range						4	
$\Delta V_{IO}/\Delta T$ Input offset voltage temperature drift	$R_S = 0$ Ω		Full range					7		μV/°C
I_{IO} Input offset current	V_O = 1.4 V		25°C		2	50		2	50	nA
			Full range		300			150		
$\Delta I_{IO}/\Delta T$ Input offset voltage temperature drift			Full range					10		pA/°C
I_{IB} Input bias current	V_O = 1.4 V		25°C		-20	-250		-20	-250	nA
			Full range		-500			-500		
V_{ICR} Common-mode input voltage range	V_{CC} = 5 V to MAX		25°C	0 to V_{CC} – 1.5			0 to V_{CC} – 1.5			V
			Full range	0 to V_{CC} – 2		0 to V_{CC} – 2				

V_{OH} High-level output voltage	R_L = 2 kΩ		25°C							V
	R_L = 10 kΩ		25°C	V_{CC} – 1.5			V_{CC} – 1.5			
	V_{CC} = MAX	R_L = 2 kΩ	Full range	22			26			
		R_L ≥ 10 kΩ	Full range	23	24		27			
V_{OL} Low-level output voltage	R_L ≤ 10 kΩ		Full range		5	20		5	20	mV
A_{VD} Large-signal differential voltage amplification	V_{CC} = 15 V, V_O = 1 V to 11 V, R_L ≥ 2 kΩ		25°C	25	100		25	100		V/mV
			Full range	15		15			dB	
CMRR Common-mode rejection ratio	V_{IC} = V_{ICR}min		25°C	50	80		60	80		dB
k_{SVR} Supply-voltage rejection ratio ($\Delta V_{CC}/\Delta VIO$)			25°C	50	100			100		dB
V_{O1}/V_{O2} Crosstalk attenuation	f = 1 kHz to 20 kHz		25°C		120		-20	120		dB
I_O Output current	V_{CC} = 15 V, V_{ID} = 1 V, V_O = 0	Source	25°C	-20	-30	-60	-10	-30	-60	mA
			Full range	-10			10			
	V_{CC} =15 V, V_{ID} = 1 V, V_O = 15V	Sink	25°C	10	20		5	20		
			Full range	5						
	V_{ID} = –1 V, V_O = 200 mV		25°C		30		12	40		μA
I_{OS} Short-circuit output current	V_{CC} at 5 V, V_O = 0, GND at –5 V		25°C		±40	±60		±40	±60	mA
	V_O = 2.5 V, no load		Full range		0.7	1.2		0.7	1.2	mA
	V_{CC} = MAX, V_O = 0.5 V_{CC}, no load		Full range		1.4	3		1.4	3	

(1) All characteristics are measured under open-loop conditions, with zero common-mode input voltage, unless otherwise specified. MAX V_{CC} for testing purposes is 26 V for LM2902 and 32 V for LM2902V.
(2) Full range is –40°C to 125°C for LM2902.
(3) All typical values are at T_A = 25°C.

Electrical Characteristics for LMx24A and LM324KA

at specified free-air temperature, V_{CC} = 5 V (unless otherwise noted)

PARAMETER	TEST CONDITIONS(1)	T_A(2)	LM124A			LM224A			LM324A, LM324KA			UNIT
			MIN	TYP(3)	MAX	MIN	TYP(3)	MAX	MIN	TYP(3)	MAX	
V_{IO} Input offset voltage	V_{CC} = 5 V to 30 V, V_{IC} = V_{ICR}min, V_O = 1.4 V	25°C			2		2	3		2	3	mV
		Full range		4			4			5		
I_{IO} Input offset current	V_O = 1.4 V	25°C			10		2	15			30	nA
		Full range		30			30			75		
I_{IB} Input bias current	V_O = 1.4 V	25°C			-50		-15	-80		-15	-100	nA
		Full range		-100			-100			-200		

V_{ICR} Common-mode input voltage range	V_{CC} = 30 V		25°C	0 to V_{CC} − 1.5			0 to V_{CC} − 1.5						V
	Full range		0 to V_{CC} − 2			0 to V_{CC} − 2							
V_{OH} High-level output voltage	R_L = 2 kΩ		25°C	V_{CC} − 1.5			V_{CC} − 1.5						V
	V_{CC} = 30 V	R_L = 2 kΩ	Full range	26			26			26			
		R_L ≥ 10 kΩ	Full range	27			27	28		27	28		
V_{OL} Low-level output voltage	R_L ≤ 10 kΩ		Full range			20		5			5	20	mV
A_{VD} Large-signal differential voltage amplification	V_{CC} = 15 V, V_O = 1 V to 11 V, R_L ≥ 2 kΩ		25°C	50	100		50	100		25	100		V/mV
			Full range	25		25			12				
CMRR Common-mode rejection retio	V_{IC} = V_{ICR}min		25°C	70			70	80		65	80		dB
k_{SVR} Supply-voltage rejection ratio ($\Delta V_{CC}/\Delta V_{IO}$)			25°C	65			65	100		65	100		dB
V_{O1}/V_{O2} Crosstalk	f = 1 kHz to 20 kHz		25°C		120			120			120		dB
I_O Output current	V_{CC} = 15 V, V_{ID} = 1 V, V_O = 0	source	25°C	-20			-20	-30	-60	-20	-30	-60	mA
			Full range	-10			-10			-10			
	V_{CC} = 15 V, V_{ID} = −1 V, V_O = 15 V	Sink	25°C	10			10	20		1	20		
			Full range	5			5			5			
	V_{ID} = −1 V, V_O = 200 mV		25°C	12			12	30		12	30		μA
I_{OS} Short-circuit output current	V_{CC} at 5 V, GND at −5 V, V_O = 0		25°C		±40	±60		±40	±60		±40	±60	mA
I_{CC} Supply current	V_O = 2.5 V, no load		Full range		0.7	1.2		0.7	1.2		0.7	1.2	mA
	V_{CC} = 30 V, V_O = 15 V, no load		Full range		1.4	3		1.4	3		1.4	3	

(1) All characteristics are measured under open-loop conditions, with zero common-mode input voltage, unless otherwise specified.
(2) Full range is –55°C to 125°C for LM124A, –25°C to 85°C for LM224A, and 0°C to 70°C for LM324A.
(3) All typical values are at T_A = 25°C.

Operating Conditions

V_{CC} = ±15 V, T_A = 25°C

PARAMETER	TEST CONDITIONS	TYP	UNIT
SR Slew rate at unity gain	R_L = 1 MΩ, C_L = 30 pF, V_I = ±10 V	0.5	V/μs
B_1 Unity-gain bandwidth	R_L = 1 MΩ, C_L = 20 pF	1.2	MHz
V_n Equivalent input noise voltage	R_S = 100 Ω, V_I = 0 V, f = 1 k	35	nV/√Hz

TEXAS INSTRUMENT

xx555 Precision Timers

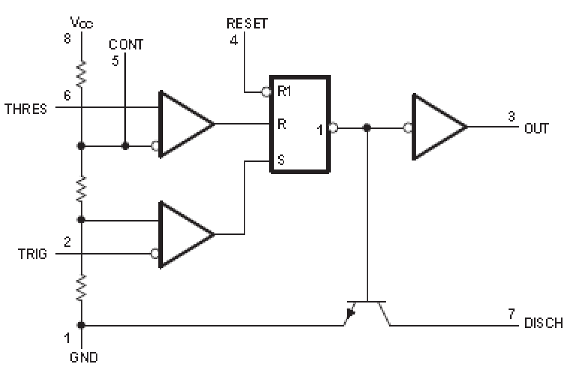
1 Features

- Timing From Microseconds to Hours
- Astable or Monostable Operation
- Adjustable Duty Cycle
- TTL-Compatible Output Can Sink or Source Up to 200 mA
- On Products Compliant to MIL-PRF-38535, All Parameters Are Tested Unless Otherwise Noted. On All Other Products, Production Processing Does Not Necessarily Include Testing of All Parameters.

2 Applications

- Fingerprint Biometrics
- Iris Biometrics
- RFID Reader

4 Simplified Schematic



3 Description

These devices are precision timing circuits capable of producing accurate time delays or oscillation. In the time-delay or mono-stable mode of operation, the timed interval is controlled by a single external resistor and capacitor network. In the a-stable mode of operation, the frequency and duty cycle can be controlled independently with two external resistors and a single external capacitor.

The threshold and trigger levels normally are two-thirds and one-third, respectively, of VCC. These levels can be altered by use of the control-voltage terminal. When the trigger input falls below the trigger level, the flip-flop is set, and the output goes high. If the trigger input is above the trigger level and the threshold input is above the threshold level, the flip-flop is reset and the output is low. The reset (RESET) input can override all other inputs and can be used to initiate a new timing cycle. When RESET goes low, the flip-flop is reset, and the output goes low. When the output is low, a low-impedance path is provided between discharge (DISCH) and ground.

The output circuit is capable of sinking or sourcing current up to 200 mA. Operation is specified for supplies of 5 V to 15 V. With a 5-V supply, output levels are compatible with TTL inputs.

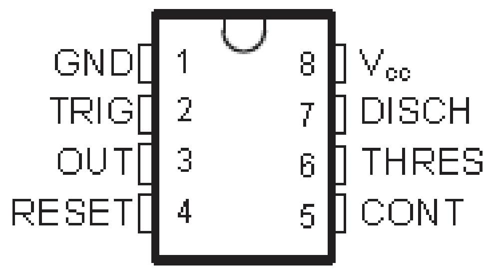
Device Information(1)

PART NUMBER	PACKAGE	BODY SIZE (NOM)
xx555	PDIP (8)	9.81 mm × 6.35 mm
	SOP (8)	6.20 mm × 5.30 mm
	TSSOP (8)	3.00 mm × 4.40 mm
	SOIC (8)	4.90 mm × 3.91 mm

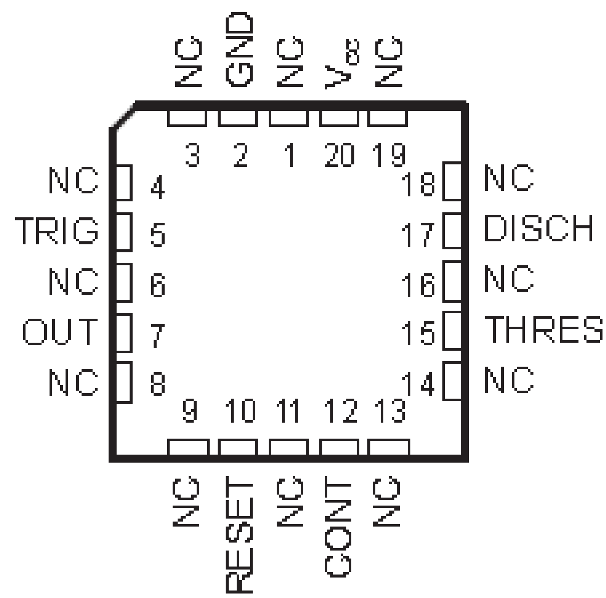
(1) For all available packages, see the orderable addendum at the end of the datasheet.

5 Pin Configuration and Functions

NA555...D OR P PACKAGE
NE555...D, P, PS, OR PW PACKAGE
SA555...D OR P PACKAGE
SE555...D, JG, OR P PACKAGE
(TOP VIEW)



SE555...FK PACKAGE
(TOP VIEW)



NC – No internal connection

Pin Functions

PIN			I/O	DESCRIPTION
NAME	D, P, PS, PW, JG	FK		
	NO.			
CONT	5	12	I/O	Controls comparator thresholds, Outputs 2/3 VCC, allows bypass capacitor connection
DISCH	7	17	O	Open collector output to discharge timing capacitor
GND	1	2	–	Ground
NC		1, 3, 4, 6, 8, 9, 11, 13, 14, 16, 18, 19	–	No internal connection
OUT	3	7	O	High current timer output signal
RESET	4	10	I	Active low reset input forces output and discharge low.
THRES	6	15	I	End of timing input. THRES > CONT sets output low and discharge low
TRIG	2	5	I	Start of timing input. TRIG < ½ CONT sets output high and discharge open
V_{CC}	8	20	–	Input supply voltage, 4.5 V to 16 V. (SE555 maximum is 18 V)

6 Specifications

6.1 Absolute Maximum Ratings[1]

over operating free-air temperature range (unless otherwise noted)

			MIN	MAX	UNIT
V_{CC}	Supply voltage[2]			18	V
V_I	Input voltage	CONT, RESET, THRES, TRIG		V_{CC}	V
I_O	Output current			±225	mA
θ_{JA}	Package thermal impedance[3][4]	D package		97	°C/W
		P package		85	
		PS package		95	
		PW package		149	
θ_{JC}	Package thermal impedance[5][6]	FK package		5.61	°C/W
		JG package		14.5	
T_J	Operating virtual junction temperature			150	°C
Case temperature for 60 s		FK package		260	°C
Lead temperature 1,6 mm (1/16 in) from case for 60 s		JG package		300	°C

(1) Stresses beyond those listed under *Absolute Maximum Ratings* may cause permanent damage to the device. These are stress ratings only, and functional operation of the device at these or any other conditions beyond those indicated under *Recommended Operating Conditions* is not implied. Exposure to absolute-maximum-rated conditions for extended periods may affect device reliability.

(2) All voltage values are with respect to GND.

(3) Maximum power dissipation is a function of T_J(max), θ_{JA}, and T_A. The maximum allowable power dissipation at any allowable ambient temperature is $P_D = (T_J(max) - T_A) / \theta_{JA}$. Operating at the absolute maximum T_J of 150°C can affect reliability.

(4) The package thermal impedance is calculated in accordance with JESD 51-7.

(5) Maximum power dissipation is a function of T_J(max), θ_{JC}, and T_C. The maximum allowable power dissipation at any allowable case temperature is $P_D = (T_J(max) - T_C) / \theta_{JC}$. Operating at the absolute maximum T_J of 150°C can affect reliability.

(6) The package thermal impedance is calculated in accordance with MIL-STD-883.

6.2 Handling Ratings

PARAMETER	DEFINITION	MIN	MAX	UNIT
T_{stg}	Storage temperature range	–65	150	°C

6.3 Recommended Operating Conditions

over operating free-air temperature range (unless otherwise noted)

			MIN	MAX	UNIT
V_{CC}	Supply voltage	NA555, NE555, SA555	4.5	16	V
		SE555	4.5	18	
V_I	Input voltage	CONT, RESET, THRES, and TRIG		V_{CC}	V
I_O	Output current			±200	mA
T_A	Operating free-air temperature	NA555	–40	105	°C
		NE555	0	70	
		SA555	–40	85	
		SE555	–55	125	

6.4 Electrical Characteristics

V_{CC} = 5 V to 15 V, T_A = 25°C (unless otherwise noted)

PARAMETER	TEST CONDITIONS		SE555 MIN	SE555 TYP	SE555 MAX	NA555 NE555 SA555 MIN	NA555 NE555 SA555 TYP	NA555 NE555 SA555 MAX	UNIT
THRES voltage level	V_{CC} = 15 V		9.4	10	10.6	8.8	10	11.2	V
	V_{CC} = 5 V		2.7	3.3	4	2.4	3.3	4.2	
THRES current[(1)]				30	250		30	250	nA
TRIG voltage level	V_{CC} = 15 V		4.8	5	5.2	4.5	5	5.6	V
		T_A = –55°C to 125°C	3		6				
	V_{CC} = 5 V		1.45	1.67	1.9	1.1	1.67	2.2	
		T_A = –55°C to 125°C			1.9				
TRIG current	TRIG at 0 V			0.5	0.9		0.5	2	μA
RESET voltage level			0.3	0.7	1	0.3	0.7	1	V
	T_A = –55°C to 125°C				1.1				
RESET current	RESET at V_{CC}			0.1	0.4		0.1	0.4	mA
	RESET at 0 V			–0.4	–1		–0.4	–1.5	
DISCH switch off-state current				20	100		20	100	nA
DISCH switch on-state voltage	V_{CC} = 5 V, I_O = 8 mA						0.15	0.4	V
CONT voltage (open circuit)	V_{CC} = 15 V		9.6	10	10.4	9	10	11	V
		T_A = –55°C to 125°C	9.6		10.4				
	V_{CC} = 5 V		2.9	3.3	3.8	2.6	3.3	4	
		T_A = –55°C to 125°C	2.9		3.8				
Low-level output voltage	V_{CC} = 15 V, I_{OL} = 10 mA			0.1	0.15		0.1	0.25	V
		T_A = –55°C to 125°C			0.2				
	V_{CC} = 15 V, I_{OL} = 50 mA			0.4	0.5		0.4	0.75	
		T_A = –55°C to 125°C			1				
	V_{CC} = 15 V, I_{OL} = 100 mA			2	2.2		2	2.5	
		T_A = –55°C to 125°C			2.7				
	V_{CC} = 15 V, I_{OL} = 200 mA			2.5			2.5		
	V_{CC} = 5 V, I_{OL} = 3.5 mA	T_A = –55°C to 125°C			0.35				
	V_{CC} = 5 V, I_{OL} = 5 mA			0.1	0.2		0.1	0.35	
		T_A = –55°C to 125°C			0.8				
	V_{CC} = 5 V, I_{OL} = 8 mA			0.15	0.25		0.15	0.4	
High-level output voltage	V_{CC} = 15 V, I_{OH} = –100 mA		13	13.3		12.75	13.3		V
		T_A = –55°C to 125°C	12						
	V_{CC} = 15 V, I_{OH} = –200 mA			12.5			12.5		
	V_{CC} = 5 V, I_{OH} = –100 mA		3	3.3		2.75	3.3		
		T_A = –55°C to 125°C	2						

Supply current	Output low, No load	V_{CC} = 15 V		10	12		10	15	mA
		V_{CC} = 5 V		3	5		3	6	
	Output high, No load	V_{CC} = 15 V		9	10		9	13	
		V_{CC} = 5 V		2	4		2	5	

(1) This parameter influences the maximum value of the timing resistors RA and RB in the circuit of astable multivibrator. For example, when VCC = 5 V, the maximum value is R = RA + RB ⍰3.4 MΩ, and for VCC = 15 V, the maximum value is 10 MΩ.

6.5 Operating Characteristics

V_{CC} = 5 V to 15 V, T_A = 25°C (unless otherwise noted)

PARAMETER		TEST CONDITIONS[1]	SE555			NA555 NE555 SA555			UNIT
			MIN	TYP	MAX	MIN	TYP	MAX	
Initial error of timing interval[2]	Each timer, monostable[3]	T_A = 25°C		0.5	1.5[4]		1	3	%
	Each timer, astable[5]			1.5			2.25		
Temperature coefficient of timing interval	Each timer, monostable[3]	T_A = MIN to MAX		30	100[4]		50		ppm/°C
	Each timer, astable[5]			90			150		
Supply-voltage sensitivity of timing interval	Each timer, monostable[3]	T_A = 25°C		0.05	0.2[4]		0.1	0.5	%/V
	Each timer, astable[5]			0.15			0.3		
Output-pulse rise time		C_L = 15 pF, T_A = 25°C		100	200[4]		100	300	ns
Output-pulse fall time		C_L = 15 pF, T_A = 25°C		100	200[4]		100	300	ns

(1) For conditions shown as MIN or MAX, use the appropriate value specified under recommended operating conditions.

(2) Timing interval error is defined as the difference between the measured value and the average value of a random sample from each process run.

(3) Values specified are for a device in a monostable circuit with the following component values: R_A = 2 kΩ to 100 kΩ, C = 0.1 μF.

(4) On products compliant to MIL-PRF-38535, this parameter is not production tested.

(5) Values specified are for a device in an astable circuit with the following component values: R_A = 1 kΩ to 100 kΩ, C = 0.1 μF.

7. Device Functional Modes

Function Table

RESET	TRIGGER VOLTAGE[1]	THRESHOLD VOLTAGE[1]	OUTPUT	DISCHARGE SWITCH
Low	Irrelevant	Irrelevant	Low	On
High	<1/3 VCC	Irrelevant	High	Off
High	>1/3 VCC	>2/3 VCC	Low	On
High	>1/3 VCC	<2/3 VCC	As previously established	

(1) Voltage levels shown are nominal.

Your notes

Your notes