Do-it-yourself projects

21 *Electronics Projects*

Alarms, touch sensors, and many more circuits

Projects

1) How to Build a Touch Sensor Circuit

2) How to Build a Sample-and-Hold Circuit

3) How to Build a Flashing LED Circuit

4) How to Build a Light-sensitive Alarm Circuit

5) How to Build a DC Motor Speed Controller

6) How to Build a Voltage Polarity Indicator

7) How to Build an Audio Mixer

8) How to Build a Moisture Meter

9) How to Build a Light Meter

10) How to Build an Amplifier

11) How to Build a Light-Dependent Amplifier

12) How to Build a Zener Diode Voltage Regulator

13) How to Build a Simple Voltmeter

14) How to Build an AC-DC Voltage Converter

15) How to Build a Simple Speaker Crossover Network

16) How to Build a Mode Light Status Indicator

17) How to Build a Latching Switch

18) How to Build a MOSFET Light Dimmer

19) How to Build a Solar Cell Battery Recharger

20) How to Build a Tone Generator

21) How to Build an Automatic Night Light

Summary of each project

Project – Touch Sensor Circuit

In this project, we will build a touch sensor circuit, which is a circuit that activates when a user touches the necessary part of it. Touch sensors have a wide variety of uses and applications. You've probably used or seen a lamp that when you touch the base of it, it turns on. We will be building a similar device in which when we touch wires, the LED in the circuit lights up. This project will teach you a lot about how touch sensor circuits work and how touching the circuit allows for activation.

Project – Sample-and-Hold Circuit

A Sample-and-hold circuit is a very important type of circuit. It is a circuit which takes a 'sample' of an analog signal and holds the signal, for any useful purpose, most of the time to be converted into a digital signal. In essence, a sample-and-hold circuit takes a snapshot of a signal and holds it, so that the signal can be used elsewhere. Think of the vast applications, such as for cameras, which take analog snapshots, which then need to be converted into digital signals. We will build such a circuit in this project.

Project – Flashing LED Circuit

In this project, we build a circuit with an LED that flashes at a rate which we set. We use an RC timing network to set the amount of flashes that occur in a certain time period. You will see how during this circuit setup, the LED can be made to flash any amount of times which we want it to. This circuit serves to teach you about RC networks and how an LED can be flashed in hardware. This is a very useful circuit as LED flashing is used in all types of devices, ranging from police sirens to fire alarms. You will see how this flashing can be achieved.

# Project – Light-sensitive Alarm Circuit
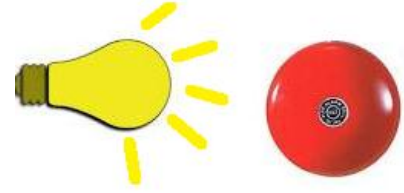
In this project, we build an alarm circuit which triggers when exposed to bright light and remains off when in darkness. This circuit has tremendous application to safeguard against items such as jewelry kept in a Chester drawer. If a burglar comes in at night and flashes the inside of the drawer with a flashlight or breaks in during the day exposing the circuit to light, the alarm goes off and the home keeper will hear its loud buzzing. This is a very practical circuit which you could even use in your very own home to safeguard your things.

Project - DC Motor Speed Controller

In this circuit, we build a device that efficiently sends out pulses of current, so that the speed of the DC motor which it powers can be adjusted. To increase motor speed, we send out higher-frequency pulses. To decrease motor speed, we send out lower-frequency pulses. In this project, we teach you how to build a device that can vary the pulse output to control the speed of a DC motor.

Project- Voltage Polarity Indicator
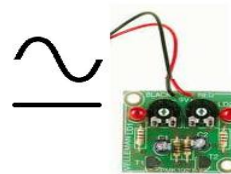
Using this device, a voltage polarity indicator, we can tell whether the signal input into the circuit is an AC signal or a DC signal. If it is DC, this circuit can tell whether it is DC in the forward direction or DC in the reverse direction. This circuit serves to teach about the properties of DC and AC current and how a device can be built to detect and indicate which type of signal is passing through. This is a very useful circuit if voltage type and polarity need to be known.

Project- Audio Mixer

In this project, we teach how to build an audio mixer, which is a device that can combine sounds together from multiple channels into one channel. So, for example, we can add together one sound input of a person singing a capella with another sound file of background music, and have both audio sounds on one channel. We teach how in this project, we can put together an audio mixer and how we can adjust the volume of each input entering the audio mixer to have desired output sound. This is a great project for learning about audio mixers and audio in general.

Project – Moisture Meter

In this project, we construct a meter which will indicate to us the amount of moisture which is in a medium, such as soil. In this project, you will learn how to build a moisture meter, how moisture meters work, and the properties that water exhibits that allow moisture meters to function as they do.

Project – Light Meter

In this project, we build a device that measures the intensity of light. This device serves to teach you specifically about a light-sensitive device, a LDR (or photoresistor), whose resistance changes dependent on the light striking it. You will learn about LDRs and how the complete circuit is constructed.

Project – Amplifier

In this project, we build an amplifier, which increases the input signal as many times as we design it to. An amplifier can be made from a transistor or an op amp. In this project, we show how to build an amplifier from an op amp chip, to create the gain that we want. This will teach you about op amp connections for amplification.

Project – Light-dependent Amplifier

Continuing with the last project, in this one, we now add to the previous amplifier. We now construct an amplifier that amplifies a signal based on the intensity of light that strikes the circuit. If the light intensity is strong, the amplifier amplifies the input signal to up to 25 times. If the light is low, the amplifier attenuates the signal and there is barely any output. In this circuit, you try this out.

Project – Zener Diode Voltage Regulator
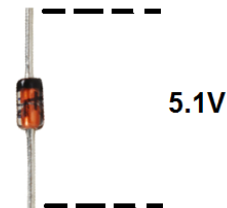
In this project, we will show how to connect a zener diode in a circuit to act as a voltage-regulating device, which is a device that outputs a constant DC voltage across its terminals. You will learn what zener diodes are, their characteristics, and how they can be connected in a circuit to provide a constant, stable voltage to any device we want to feed.

Project – Simple Voltmeter

In this project, you will learn how to build a simple voltmeter, which is a device that detects and measures the level of voltage in a circuit. Voltmeters are very useful and are present in all multimeters, an essential tool for any engineer or technician. In this project, we'll exploit the properties, specifically, of zener diodes to build a device that can measure the voltage entering a circuit.

Project – AC-DC Voltage Converter

In this project, we will build a device that takes an input AC signal and converts it into DC to power an electrical device. This is very common in practically all electrical devices which are plugged into AC outlets. The device takes the AC current from the wall outlet and converts it into DC, and then is powered by the DC current. You will learn in this project how to build your own AC-DC converter, so that if you want to design your own plug-in product one day, you'll know how the circuit construction is.

Project – Speaker Crossover Network

In this project, we build a speaker crossover network, which is a network of speakers. Woofers are speakers which are designed to play out low-frequency (bass) sounds well. Tweeters are speakers which are designed to play high-frequency (treble) sounds well. By building a filter to send bass to the woofer and treble to the tweeter, we can build a speaker system with the best possible sound, which is how the best speaker systems are built.

Project – Mode Light Status Indicator

We've all seen electronic devices that have LEDs which represent what mode or status they are in. For example, printers are devices that have these LEDs often on them. When the green LED is lit, that most likely means it's ready to print. When the red LED is lit, it means it is in a paused status, either waiting on a queue or there is a jam or some error. Cameras are the same, where a green LED means it is ready to take a picture, while a red LED means there is some pause. In this project, we will use a SPDT throw to enact a similar system to show how these mode light status indicators operate. Doing so, you can build a device that enacts the same type of operation.

Project – Latching Switch

Latching switches have widespread use in electrical circuits. A latching switch is a switch that once activated, remains on, until the whole system is disabled. Think of alarm circuits and how once they are triggered, remain on, until a special code has to disable it. In this project, we will build a latching switch that turns on an LED and only turns it off when the main power going to the LED is shut off. This project serves to teach how latching switches operate and how they are connected in a circuit to provide such operation.

Project - MOSFET Light Dimmer

Using a MOSFET circuit connected to a lamp, we will build a circuit in which the MOSFET allows for the changing of the intensity of the light by controlling the current output. In this project, you will learn how to connect MOSFETs for adjustable current output.

Project – Solar Cell Battery Recharger

In this project, we will build a battery recharger from solar cells. Solar cells, which harvest energy from light, convert the light energy into electrical current, which then charges a battery or batteries. In this project, you'll learn how solar cells can be connected to give enough power output to recharge a rechargeable battery.

Project – Tone Generator

In this project we will build a device that can play out varying tones, which we can change by varying the potentiometer of the RC module in the circuit. Thus, the speaker that is connected to our circuit will output different frequency tones according to the value we adjust the potentiometer to. This serves to teach you about RC circuits and how frequencies can be changed by changing either resistor or capacitor values in an RC circuit.

Project – Automatic Night Light

This is a circuit that turns on automatically when night time approaches. Say if you're a late night person and you love to stay up to watch TV or read during the night. This circuit can automatically turn on when it becomes dark, so you won't even have to move to turn on your lamp. This circuit will serve to teach you how photoresistors can be hooked up in a circuit to allow for such output.

How to Build a Touch Sensor Circuit

In this project, we are going to build a touch sensor circuit, which is a circuit that turns on when you touch the necessary part of the circuit. Specifically, for this circuit, when you touch the two necessary jumper wires, the LED which is in the circuit will light up.

Touch sensor circuits have tremendous application in electronic products. Maybe you've had a lamp where, when you touch the base of it, it turns on.

This circuit will serve to teach you how touch sensors work.

Parts for Touch Sensor Circuit

- **2N2222 Transistor**

- **220Ω Resistor**

- **270Ω Resistor**

- **LED**

- **2 Jumper Wires**

Circuit for Touch Sensor

Schematic Diagram
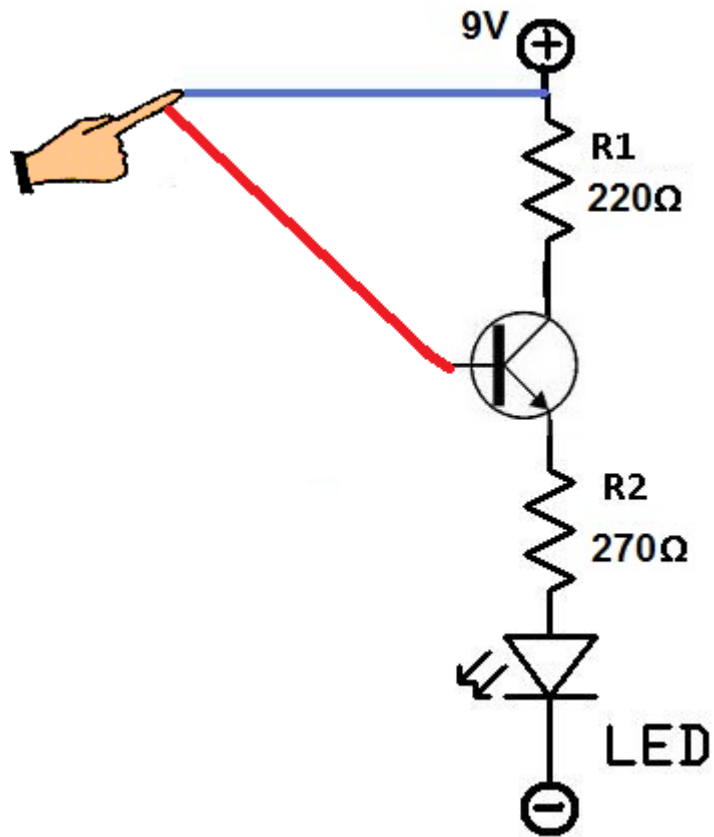
Operation of Circuit

In this project, the red and blue jumper wires, which extend out from the circuit, are the two touch sensors. When the wires are not touched, the circuit is open and no electricity can flow; thus, the LED does not light up. However, when a person touches both of the wires, the circuit is then closed and current runs through the circuit and lights the LED. The person's body acts to close the circuit. Meaning, the person's body completes the circuit so that current has a full path to flow. It uses the person's body as part of the path. Because a person offers so much resistance, the current is very small and doesn't cause any shock. So, don't worry about the fact that current is flowing through your body. It is negligible. In fact, this current alone would not be enough to power an LED. This is why a transistor is in the circuit. It amplifies this small current, making the current sufficient to light an LED.

Troubleshooting

Potential Problems that May Occur

Problems can arise while doing this circuit, so this is why it's recommended that you use the exact part values or extremely close when connecting this circuit up.

First of all, if you place too much voltage into the circuit, the LED will light up without you even touching the wires. This defeats the whole purpose of having touch sensor. Under too much bias voltage provided into the circuit, the collector of the transistor breaks down and conducts across into the emitter, even without any base current at all. This is called the transistor breakdown voltage. This is why the level of voltage and the values of components need to set up exactly, or else this circuit may not function.

In the same way, too little voltage will not make the circuit function as well. If too little voltage is used, even when you touch the wires, the LED will not light, because the current isn't sufficient to light the LED.

These two pitfalls need to be avoided to make sure that the circuit is successfully built and runs.

How to Build a Sample-and-Hold Circuit

In this project, we are going to build a sample-and-hold circuit. Sample-and-hold circuits are used to sample an analog signal and hold it so that it can be analyzed or converted into, a digital signal, if desired.

In a sample-and-hold circuit, we make a switch act as a sample/hold control. Sampling begins when the switch is closed and ends when the switch is open. This way, we can take a sample of any signal that we want to.

Sample-and-hold circuits are used extensively in analog-to-digital converters. The sample circuit takes a sample of the analog signal and then the chip converts the held analog signal to a digital signal. Sample-and-hold circuits also serve as analog memory devices in that the analog signal is held in the capacitor for a specified period of time. Just think also of other applications such as being used by a camera, which takes a picture. That analog signal is then held by a sample-and-hold circuit, to be converted into a digital signal by an ADC converter. The applications for sample-and-hold circuits are vast.

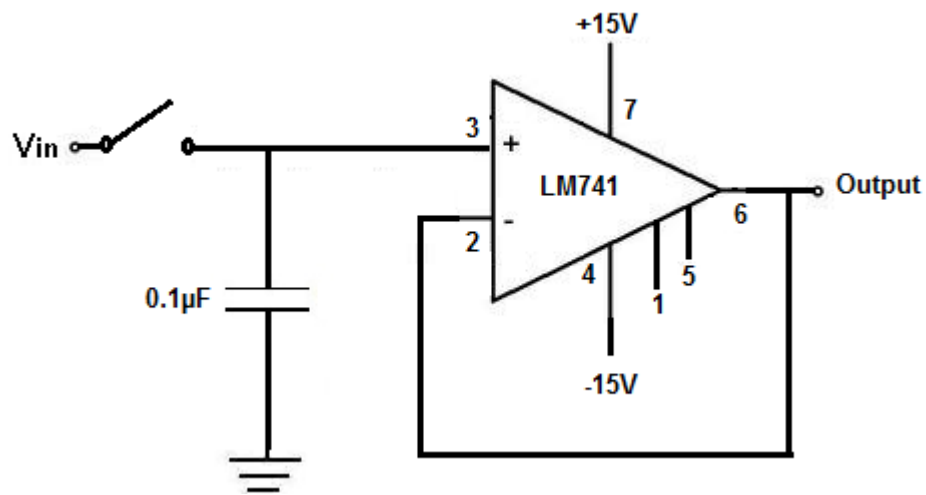# Parts for Sample-and-Hold Circuit

- **LM741 Op Amp**

- **0.1 µF Polyester Capacitor**

- **Normally Open Push Button**

Circuit for Sample-and-Hold

Operation of Circuit

In this circuit, the switch acts as a sample/hold control. Sampling begins when the switch is closed and ends when the switch is opened. When the switch is opened, the input voltage present at that exact moment will be stored in the capacitor.

The op amp acts as a buffer, meaning it doesn't amplify or attenuate the signal. It just keeps the full original signal intact and passes it out as output. The op amp also prevents the capacitor from discharging. Ideally, no current enters the input of the op amp, because of the high impedance of the op amp. This is also why it's a good idea to use a FET over a bipolar op amp. FETs have much higher impedances, which would make the capacitor current leak less, so the signal which is sampled can be held more efficiently.

The length of the sample voltage that can be held varies, depending on the capacitor's leakage. Capacitors with high ESR, low DA, low ESL, and high stability should be used. Polyester capacitors work well with sample-and-hold circuits because they have the matched characteristics. Electrolytic capacitors should not be used for sample-and-hold circuits, because their leakage levels are too high to hold a constant sample voltage.

How to Build a Flashing LED Circuit

In this project, we will configure a circuit that causes an LED to flash.

LED flashers have wide application to real-world life, including alarms that go off and on a repeated amount of times, sirens such as from a police car flashing on and off, and many other various units.

By knowing how to create a circuit that sends a predetermined amount of pulses to an LED to allow you to control the amount of times it flashes during a time period, you will learn a very useful skill that can be applied to real-life circuits.

In this project, we will use an RC circuit connected to a unijunction transistor to determine the flash rate and how to trigger it to an LED.

Parts for LED Flashing Circuit

- **2N4891 or Similar Unijunction Transistor**

- **100Ω Resistor**

- **10KΩ Resistor**

- **100KΩ Potentiometer**

- **22µF Electrolytic Capacitor**

- **LED**

Circuit for LED Flasher

Schematic Diagram
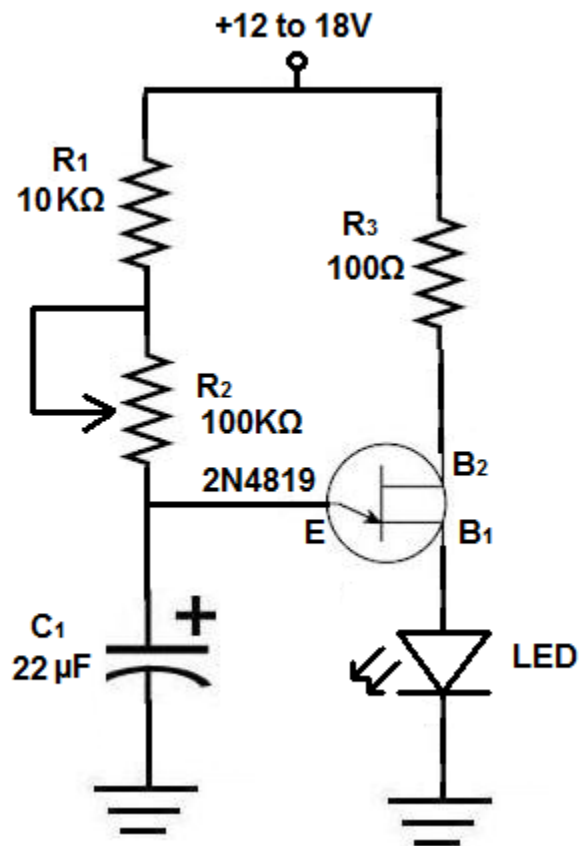
Operation of Circuit

The potentiometer R_2 and the C_1 capacitor control the rate of the flashes which go to the UJT and then to the LED. The flash rate is calculated by the product of R_2 x C_1. The larger the resistance value, the larger the RC time constant. Thus, the less pulses that are created, the less the LED flashes. In fact, when the potentiometer is at its maximum value, 1MΩ, the RC time constant is equal to RC= 100KΩ x 22μF= 2.2 seconds, the slowest it will flash for this capacitor-resistor pair value. As the resistance of the potentiometer is lowered, the RC time constant, the time required to charge the capacitor, becomes lower and lower. Thus, the LED flashes at a much faster rate. The R_2 potentiometer is what determines the rate of the flashes.

Each time the UJT turns on, the charge on the capacitor is dumped through the LED. The LED flashes in response. Between flashes, the LED glows dimly.

Note: The resistors R_1 and R_3 are just for biasing purposes, to limit the current going through the UJT circuit.

How to Build a Light-sensitive Alarm
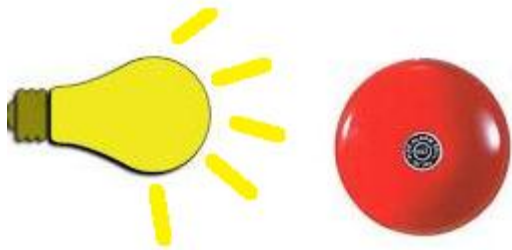
We will now build a circuit which functions as a light sensitive alarm, which can be used as a security device to safeguard personal items, such as jewelry and other valuables that may be kept in a drawer.

We will build a light-sensitive alarm circuit that when exposed to light will buzz loudly, alerting a person that the circuit is now exposed to light, meaning someone has opened the drawer (and allowed light in).

This can have very useful application to when a person stores something away in a drawer such a Chester drawer. In a drawer, the light-sensitive circuit, as well as all items in the drawer, will be completely shut, and all light outside is kept out. Thus, the device will be off. If someone now opens a drawer during the day or, at night, someone such a thief flashes a flashlight into the drawer, the alarm will turn on and the buzzer will go off. If the housekeeper is at home during the time of the burglary, he or she should be able to hear the buzzer and know that someone has gone into the drawer.

This project will serve to teach you how you can construct an alarm that goes on based on the intensity of light that reflects on it and how a light-dependent resistor (or photoresistor) works.
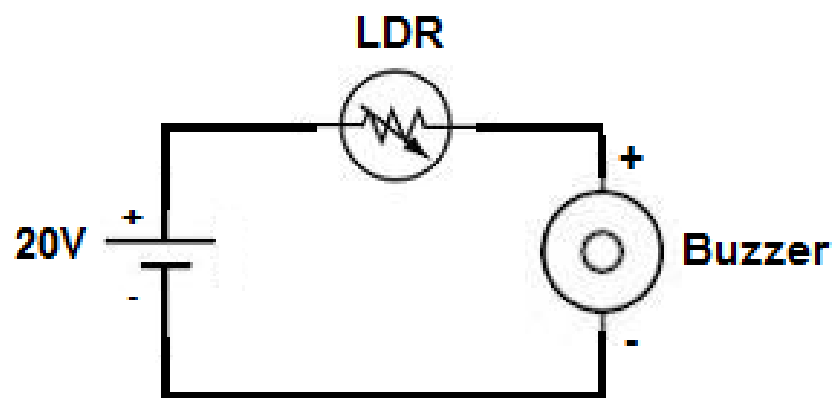
Parts for Light-sensitive Alarm

- **3-6 Volt Buzzer**

- **1MΩ Light Dependent Resistor (photoresistor)**

Circuit for Light-sensitive Alarm

Operation of Circuit

The main device controlling the circuit is the light-dependent resistor (also called photoresistor). The power source provides the power so that current can flow. But current only flows through the resistor depending on the amount of light which the photoresistor is exposed to.

When exposed to darkness or very dim light, the resistance of the photoresistor is very high. Depending on the rating of the photoresistor, the resistance can soar to several tens of thousands of ohms or even megohms (millions of ohms). We recommend for this project that you use one at least with a 1MΩ rating. This means that in complete dark, it will reach a resistance value near 1MΩ. When the resistance is this high, the current going through the circuit will be very small and will be insufficient power to sound the buzzer. Thus, the buzzer does not sound.

However, when exposed to bright light, such as a thief pointing a flashlight at the circuit to see the contents which are in the drawer, the photoresistor has much lower resistance, possibly as low as a few hundred ohms. The current flowing through the circuit is now large and is sufficient to turn on the buzzer. The buzzer now sounds, and the homeowner will know that someone has gone into the drawer.

How to Build a DC Motor Speed Controller

In this project, we are going to build a circuit that can control the speed of a DC Motor.

A Silicon-controlled rectifier (SCR) along with a few resistors, a capacitor, and a Unijunction transistors can be connected together to make a variable-speed control circuit to run a DC motor and adjust its speed.

Parts for DC Motor Speed Controller

- **2N4819 Unijunction Transistor**

- **6V DC Motor**

- **Silicon-controlled Rectifier**

- **100Ω Resistors (Qty: 2)**

- **-100KΩ Potentiometer**

- **4.7µF Capacitor**

Circuit for DC Motor Speed Controller

Schematic Diagram
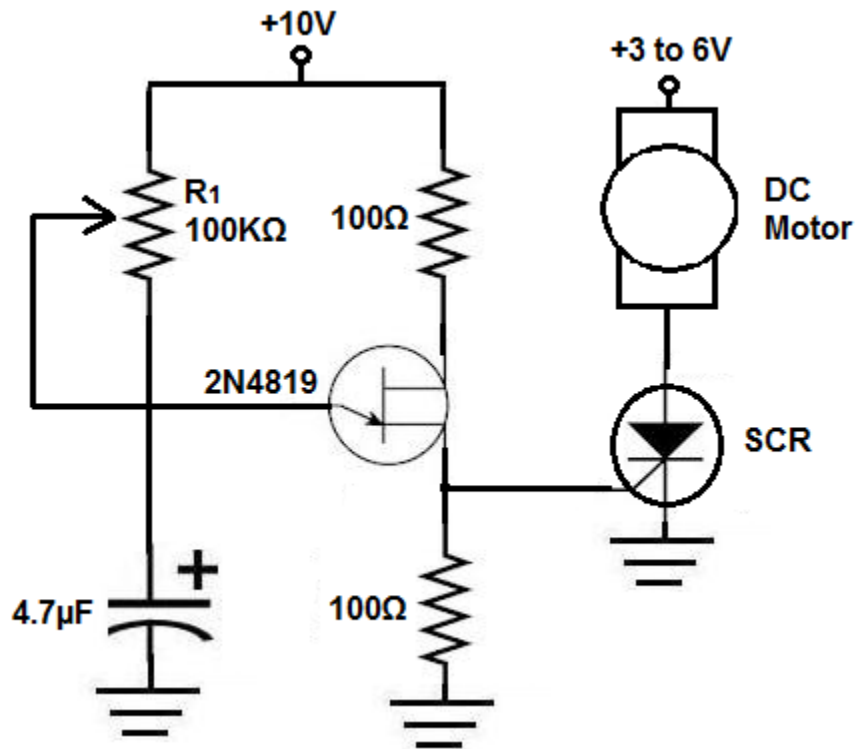
The 100Ω resistors are just to bias the UJT transistor. They are small resistors to help limit current passing through the base leads of the UJT.

The resistor, R_1, and the capacitor, C_1, form an RC network which controls the frequency of the waveform which goes to the UJT. This is discussed more below.

The UJT transistor provides some amplification to the RC output waveform and acts a switch that turns on the SCR's gate. The SCR then powers the DC motor.

Operation of Circuit

The Unijunction transistor (UJT), the R_1 resistor, and the C_1 capacitor make up an oscillator circuit that supplies an AC voltage to the SCR's gate. When the voltage at the gate exceeds the SCR's triggering voltage, the SCR turns on, thus allowing current to flow through the motor. Changing the resistance of R_1 changes the frequency of the oscillator and, hence, determines the number of times the SCR's gate is triggered over time, which controls the speed of the motor.

Note: The RC circuit really is the portion that controls the frequency of the oscillatory signals that then passes through the UJT.

Larger RC value produces longer waveforms, which oscillate much less frequently. Therefore, the speed of the motor will decrease with a larger R_1 value. Lower RC values produce shorter waveforms, which oscillate much more frequently. Therefore, the speed of the motor will increase. By controlling and adjusting the value of the R1 resistor, you can increase or decrease the speed of the DC motor.

Even though DC motors appear to turn continuously, they are actually triggered by on/off pulses. Feeding motors on/off pulses is more effective than feeding it a continuous voltage, because feeding it continuous power is a waste of power. This is why a circuit such as above is more effective (energy-efficient) than when using a simple series variable resistor to control the speed of a motor. Motors need only on-off pulses, not continuous. On/off pulses use less energy and are more the type of voltage that a motor should receive.

How to Build a Voltage Polarity Indicator
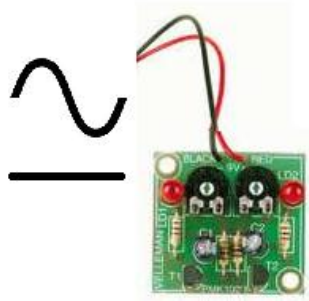
In this project, we will build a device that can indicate to us whether the voltage flowing through it is AC or DC, and for DC, it can tell whether the current is flowing forward or flowing in reverse.

This is a useful device for determining the type of input voltage that is coming into a circuit and its polarity (if DC).

This project will serve to teach you about how AC and DC signal operate and how it's possible to build an indicator system which can show what type of signal is input into the circuit.

Parts for Voltage Polarity Indicator

- **Red LED**

- **Green LED**

- **470Ω Resistor**

Circuit for Voltage Polarity Indicator
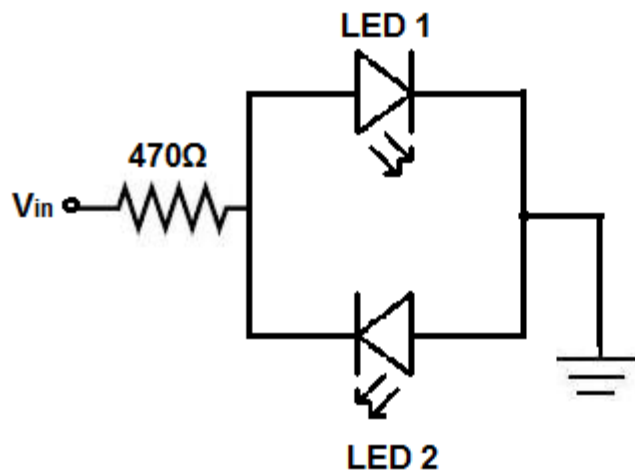
Vin	LED 1	2
DC +	On	Off
DC -	Off	On
AC	On	On

Operation of Circuit

V_{in} is where the input voltage comes in which is being input into the circuit. This is the signal which this circuit will test to see if it is forward DC, reverse DC, or AC.

The chart above shows all the combinations that can occur. If the signal is a forward DC signal, meaning it travels from V_{in} to ground, only LED 1 will turn on. This is because DC goes to only one direction, from V_{in} to ground. Being that only one LED is forward biased, the DC current can only light LED 1 and travel to ground. If the signal input is a reverse DC current, meaning it travels from ground to V_{in}, the other LED, LED 2, will light up in response. This is because the current travels in the opposite direction. Now, in reference to this current, LED 2 is forward biased and LED 1 is reverse biased. Therefore, a reverse DC current can only light LED 2. If you want to test both forward and reverse DC currents, hook up a forward DC current to V_{in} and then switch the polarities of the leads (making it reverse). You will observe that LED 1 lights up and then when switching the leads, LED 2 will then light up. If the signal input into the circuit is an AC signal, then both LEDs will light up. This is because AC signals switch polarities constantly. Therefore, both LEDs will be lit.

Again, this circuit is a good test for determining voltage polarities and whether the voltage input is DC or AC.

How to Build an Audio Mixer

In this project, you will build an audio mixer, which is a device that can add different audio signals together into one output. For example, say, one audio input is a cappella, with a person's lone voice singing and another audio signal is background music. With an audio mixer, you can combine the background music with the singing to have an output as if both audio signals were together, in other words, mixed, which is the function of a mixer.

This project will serve to teach you about mixers and how they can be constructed using a few resistors, potentiometers, capacitors, and a JFET transistor.

Parts for Audio Mixer

- **2N3819 JFET Transistor**

- **1µF Capacitor (Qty: 4)**

- **10KΩ Resistors (Qty: 3)**

- **100Ω Resistor**

- **100KΩ Potentiometers (Qty: 3)**
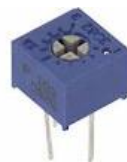
Circuit for Audio Mixer
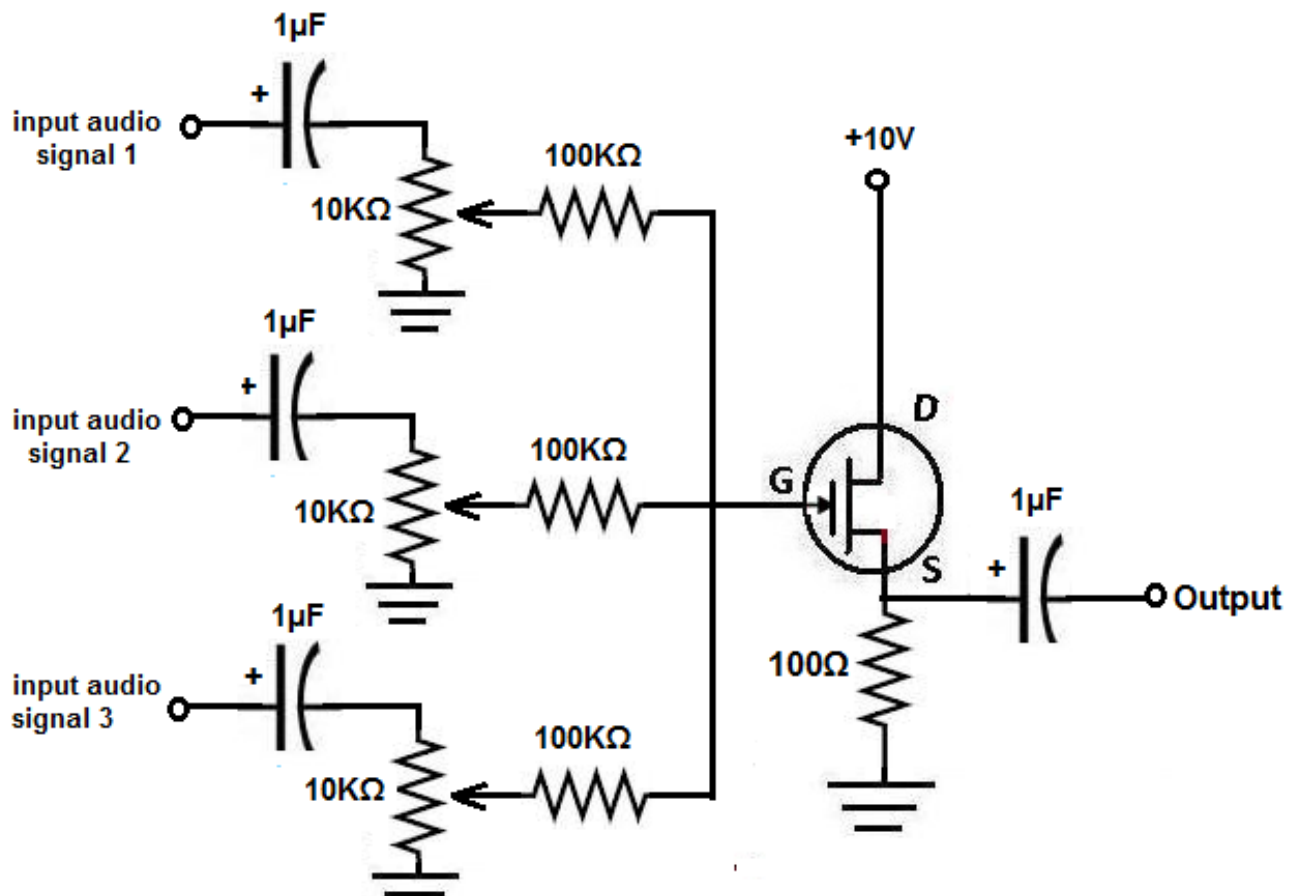
Operation of Circuit

A mixer is really nothing more than an adder. It adds all the different signals that are input into it and then combines the added signals into one.

The input audio signal inputs are where the audio signals which are to mixed enter the circuit. This could anything from microphone input, to a music audio file, to a speech file, any type of audio input which can be connected to this mixer.

The 1μF capacitors which are directly after the audio signal inputs are coupling capacitors. These capacitors serve to only let the AC aspect of the audio signal through and block the DC aspect. Only the AC aspect of the signal represents the audio of the signal. The DC is unwanted and would cause offset to the signal. Therefore, this is why we use these coupling capacitors. After these capacitors are 10KΩ resistors which are then hooked up to 100KΩ potentiometers. These 100KΩ potentiometers serve as volume controls. They control the volume of each individual audio signal which is input into the circuit. Turning the potentiometer so that its resistance increases lowers the volume of that audio signal. Conversely, turning the potentiometer so that its resistance decreases increases the volume of that audio signal.

The 6.8KΩ resistor is just for biasing purposes, to make sure the output signal drops across the source of the JFET.

The 1μF capacitor on the output again serves as a coupling capacitor, so that only the AC signal goes to output and not any DC to cause offset.

The output signal contains all the audio signals tied into one channel, which is exactly how a mixer works.

How to Build a Moisture Meter

Now, for this project, we will build a moisture meter. This is a meter that measures the moisture of a content, mostly used for reading the moisture of soil content. Meters like this have application when a gardener or a farmer wants to see how moist soil is a few feet down into the soil, to check his crops need watering.

This project serves to teach you about the construction of a moisture meter, how it operates, and the electrical properties of water which allows a moisture meter to work as it does.

Parts for Moisture Meter

- **100KΩ Potentiometer**

- **NPN 3904 Transistor**

- **1MΩ Potentiometer**

- **2.2KΩ Resistor**

- **Galvanometer**

- **Sensor probes**

Circuit for Moisture Meter
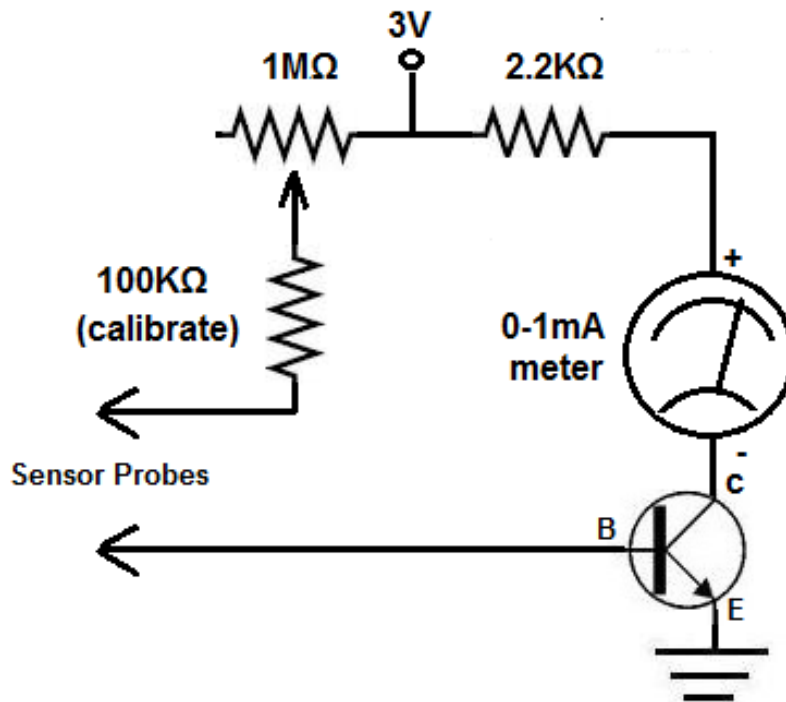
Operation of Circuit

Now it will be explained in detail how this circuit operates.

The circuit operates on the function of the electrical properties that water exhibits. When a medium, such as soil, has a high-moisture content, it conducts electricity better, because water is a conductor. When a medium has a high water content, its resistance lowers, and, thus, it is able to conduct electricity easier. When a medium has a lower water content (compared with a high one), its resistance is much greater; thus, it is not able to conduct electricity as easily. This shows up in the ammeter connected to this circuit. When a soil has high moisture, the meter (if calibrated correctly (explained below)) will swing far right, because the high-moisture soil has will conduct more current. When a soil has low moisture, the meter will not deflect much, because the low-moisture soil does not conduct current as well.

The bipolar transistor is in the circuit to allow for circuit amplification, because the signal, alone, would be weak.

The circuit needs 3V in order to function, because without a power source, there can be no current flowing through the circuit. The 3V is the power source, which can be from a power supply or 2 'AA' batteries in series. The current now, through the circuit, again, depends on the resistance of the soil (which depends on it moisture level).

The sensor probes used for this meter should be clean nail or wire, since impurities could alter the resistance reading.

Note: The calibrator will now be discussed as it is very important aspect of the circuit. The calibrator allows you to set the level that the meter deflects at different readings. When the moisture level is high, you want the ammeter to deflect far right, indicating a high moisture level. To do this, adjust the 1MΩ potentiometer so that it deflects far right when the probes are in high-moisture soil. Then when you put the probes in low-moisture soil, the ammeter will deflect less. This is a correct calibration.

To show you how calibration can go wrong, imagine putting the sensor probes in very high-moisture soil and the meter barely deflects at all. This could occur if the resistance of the potentiometer is set too high. This will make the meter a useless device. Calibration is key to get good readings.

How to Build a Light Meter

In this circuit, we will build a light meter, a device that detects and measures the intensity of light that falls upon it.

This project will serve to teach you how simple light level meters are constructed, how they operate, and how they are calibrated.

This is a very good project for learning how measuring devices, in general, operate.
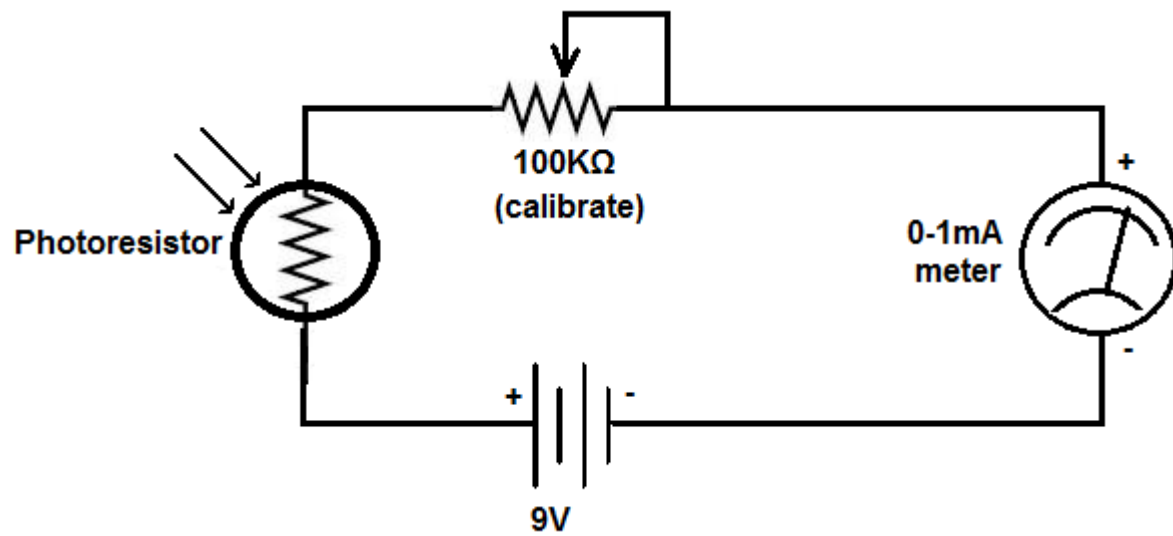
Parts for Light Meter

- **100KΩ Potentiometer**

- **Photoresistor**

- **Galvanometer**

Circuit for Light Meter

Operation of Circuit

Now it will be explained in detail how this circuit operates.

The meter to the far left of the circuit is the current meter. It has a needle that deflects left or right based on the current moving through the circuit. When the circuit has a high level of current moving through it, the needle will deflect more far right. When it has less current, it deflects less and moves only a little to the right.

The 9 volts is necessary to have power for the circuit. Without this voltage, there would be no current at all and the ammeter would never move.

The photoresistor's role in the circuit is that it varies the level of current in the circuit. When the photoresistor is exposed to total darkness, its resistance is very high, possibly as high as a few megohms (MΩ) (depending on the type of photoresistor). Thus, practically no current flows through the circuit. As the light that the photoresistor is exposed to begins increasing, its resistance lowers more and more. The current that flows through the circuit, then, is, thus, greater, and the needle deflects more. When exposed to full brightness, the resistance of the photoresistor is at its minimum, and the most current flows through the circuit. The needle of the ammeter deflects the greatest far right.

In this way, this circuit functions a light-measuring device.

Note: Now, for learning about how the calibration works for this light meter:

The 100KΩ potentiometer is the calibrator for this circuit. You will need to adjust in order for the ammeter to work well with this circuit. Say, for example, the photoresistor is only exposed to dim light, yet the needle swings far right, reading its highest value. This is not desired, as when the light increases, the ammeter will be no value. In a case like this, you will have to increase the resistance of the potentiometer so that in dim light, it swings only partially right and when exposed to very bright light, swings all the way right. The calibrator is just to make sure that the needle deflections make sense and show appropriate readings for the appropriate conditions.

How to Build an Amplifier

In this project, we will build an amplifier. An amplifier is a device that increases the level of a signal. It can increase the input signal by a factor of 5, 10, 20, etc. This means that the signal will be amplified by this amount.

Amplifiers can be built in many ways. They can be built using transistors or they can be built using op amps, which are integrated circuits that can allow for amplification.

In this project, we will build an amplifier using a LM741 Op Amp Chip. By connecting and using appropriate resistor values, we can connect it to provide the gain which we want.

Specifically, in this circuit, we will connect the LM741 op amp to provide a gain of 10, so it will increase our input signal by a factor of 10.

LM741 Op Amp

A LM741 Op Amp is a versatile chip that can be used as an amplifier.

Pinout

A LM741 is an 8-pin DIP chip, meaning it has 8 pins all having their different functions.

Below is the pinout diagram of a LM741 Op amp chip:

LM741 Pinout Diagram
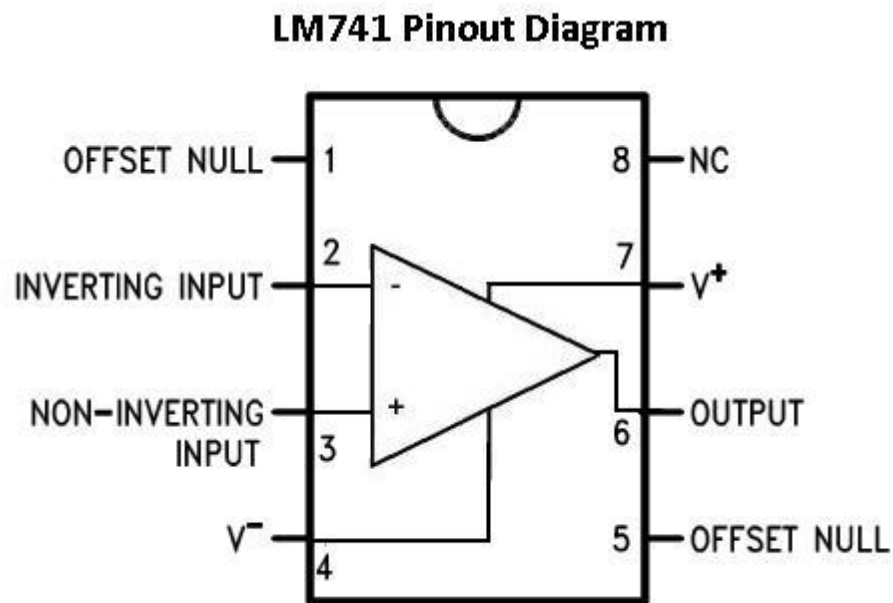
Pin 1: Offset Null- This pin will be left open.
Pin 2: Inverting Input- This is where the positive part of the input signal that we want to amplify goes if we want our amplified signal inverted. If we don't want it inverted, we place the positive part of the signal into the Non-inverting terminal and place the negative or ground part of our signal here.
Pin 3: Non-inverting Input- This is where the positive part of the input signal that we want amplified goes if we want our signal non-inverted.
Pin 4: V- The LM741 Op amp is a dual power supply op amp, meaning it must be supplied positive DC voltage and negative DC voltage. Pin 4 is where the op amp gets supplied with negative DC voltage. We will feed it -15V.
Pin 5: Offset Null- This pin will be left open.
Pin 6: Output- This is the terminal where the output, the amplified signal, comes out of. Whatever output the amplifier will drive gets connected to this terminal.
Pin 7: V+- This is the terminal which receives the positive DC voltage. We will feed it +15V.
Pin 8: NC- This pin will be left open.

Parts for Amplifier

- **LM741 Op Amp**

- **1KΩ Resistor**

- **10KΩ Resistor**

Circuit for Amplifier

Schematic Diagram
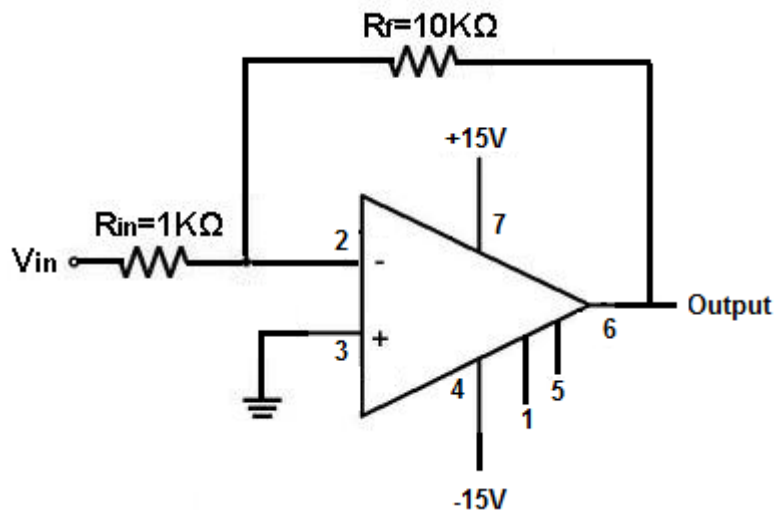
At V$_{in}$ of the schematic diagram above, we input the input AC signal which we want to amplify. We connect the positive part to the inverting pin of the op amp and the negative part into ground. And the output pin is where we retrieve our amplified output signal.

Operation of Circuit

The op amp just had 4 main pins that you use. The op amps uses V+ and V- to receive its DC voltage, needed to give it its DC bias voltage. We use the inverting input pin to input the signal which we want to amplify. And we use the output pin to retrieve the amplified output signal.

When you connect the signal input to the inverting side of an op amp, the voltage output is equal to:

$$\text{Voltage output} = -V_{in} \frac{(R_f)}{R_{in}}$$

The gain of the op amp is equal to:

$$\text{Gain} = \frac{(R_f)}{R_{in}}$$

If you want to increase the gain of the op amp, all you have to do is put a larger resistance for R$_f$. If you want to decrease the gain of the op amp, you just put a lower resistance value for R$_f$.

How to Build a Light-Dependent Amplifier

Now that we have built the above amplifier and you're now familiar with working with LM741 chips, we will now modify the circuit so that the amplifier will amplify not based on the output resistor value but will amplify according to the intensity of light that falls upon it, so that, when the amplifier is exposed to darkness or very dim light, it practically does not amplify at all. When it is exposed to bright light, it causes great amplification.

This project serves to teach you again about amplifiers and now how amplification can vary according to light intensity.

Parts for Light-Dependent Amplifier

- **LM741 Op Amp**

- **10KΩ Resistor**

- **Light Dependent Resistor**

Circuit for Light-Dependent Amplifier
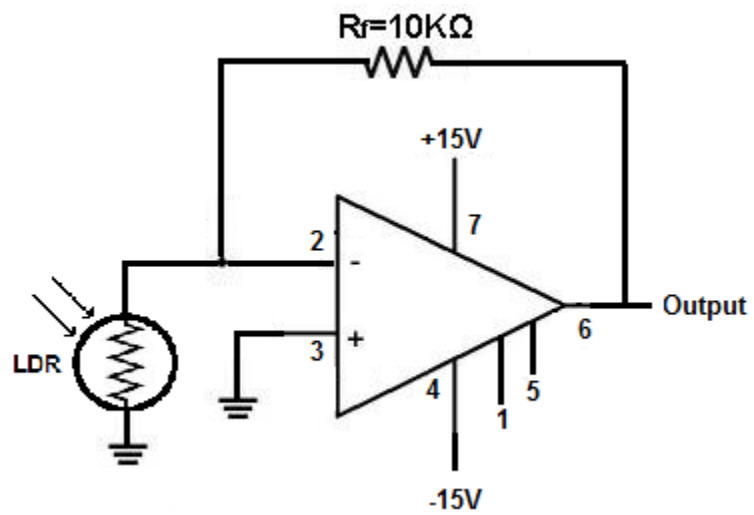
Operation of Circuit

When LDR is exposed to darkness, its resistance is very high, possibly 1MΩ or higher, depending on the specific LDR in use. In this circumstance, its resistance is so high that the amplifier does not function as an amplifier anymore but an attenuator. The resistance output to input is much less than 1, so attenuation occurs. When this happens, the amplifier decreases the input signal and we basically get no signal as output.

Now when the signal is exposed to bright light, its resistance largely decreases. Now the value of the output resistance to the input resistance can be over 20. Because the resistance of LDRs can fall to about as low as 400Ω, now the gain that would occur would be $R_f/R_I = 10KΩ/400Ω = 25$. So our output signal will now be 25 times the level as the input signal.

And this is how an amplifier can vary the amplification of signals depending on the input light into the circuit.

How to Build a Zener Diode Voltage Regulator
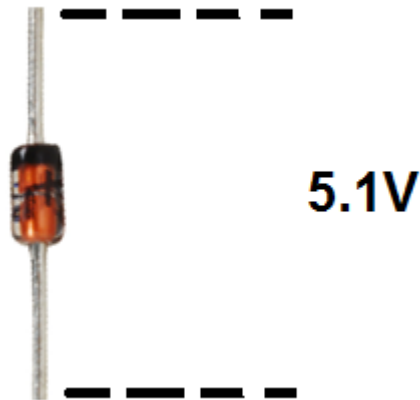
A Zener Diode is a special type of diode that is used most extensively as a voltage regulator; this is because once the reverse voltage supplied to a zener diode reaches its breakdown voltage, referred to as V_z, the voltage across the diode remains very constant at this voltage even if the current through the diode continues to increase or vary.

For this reason of steady voltage after its breakdown voltage is this diode able to function as a voltage-regulating device.

This project will serve to teach you the characteristics of zener diodes and how they can be connected in a circuit to act as voltage regulators.

Zener Diodes

The reverse voltage that a zener diode is fed is the voltage that a zener diode is fed when it is connected to reverse bias.

A diode connected in reverse biased is one in which the positive voltage of a voltage source is connected to the cathode of the diode and the negative or ground of the voltage source is connected to the anode of the diode.

Below you see both forward and reverse biased connections:
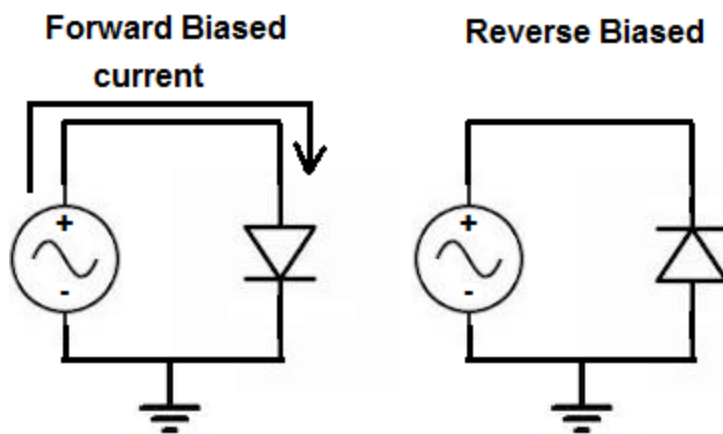
Zener diodes are connected reverse biased.

When a positive voltage is fed into it in reverse bias, this is called reverse voltage.

Looking at a characteristics curve, you can see at the far left the reverse voltage:

Zener Diode I-V Characteristics Curve
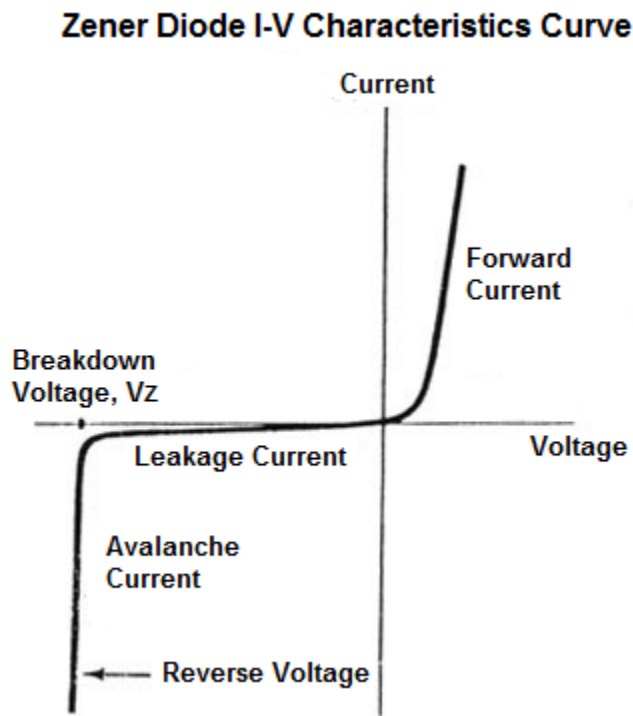
You can see in the above curve the how steady and constant the voltage across the zener diode is after it reaches the breakdown voltage, despite large changes in current.

This makes the zener diode very useful in circuits where steady voltages need to be supplied.

We will exploit this principle in this project to make a voltage regulator.

Parts for Zener Diode Voltage Regulator

- **5.1V Zener Diode**

- **1KΩ Resistor**

- **12V Power Source**

Circuit for Zener Diode Voltage Regulator

Schematic Diagram

Zener Diode Voltage Regulator Circuit
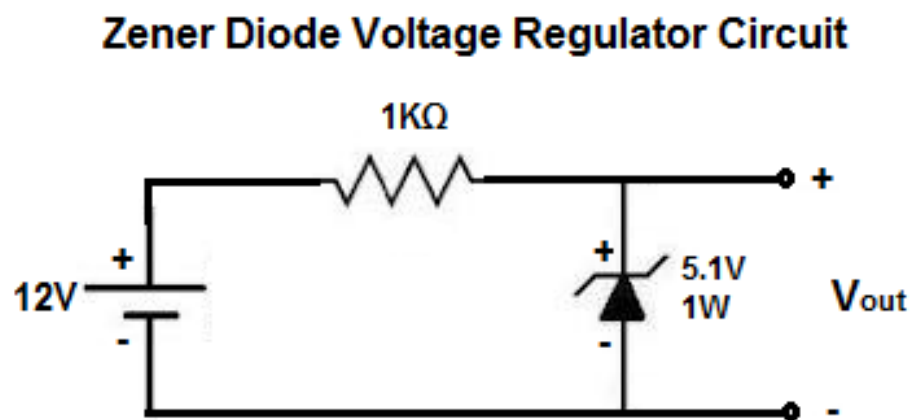
If you don't have access to a DC power supply to output 12V, you can use a 9-volt battery in place. The circuit will still work.

Operation of Circuit

This circuit outputs a pretty-near steady 5.1V to any load connected in parallel to the zener diode.

This is how the circuit works:

The circuit outputs 12V to start with, as you can see the 12V power supply. The 12V then goes into a voltage divider between the 1KΩ Resistor and the Zener Diode. Being that the zener diode is rated at a breakdown voltage of 5.1V, it drops a steady 5.1V across its terminals. The remaining of the voltage (6.9V) goes to the 1KΩ resistor. This is why the 1KΩ resistor is so important. It takes the remaining voltage and drops it across itself. Of course, again, the zener diode stays steady at its breakdown voltage, so it will not drop any more voltage across it than 5.1V, which is why it is good at regulating voltage.

Now, any device that you place in parallel to the zener diode will receive its steady 5.1V, because voltage in parallel is the same.

How to Build a Simple Voltmeter

In this circuit, we're going to build a circuit which keeps a check on the levels of voltage that enters the circuit. We're going to use a bar graph LED that lights up when the voltage reaches certain levels. The amount of rows lit on the LED will indicate to us the intensity of the voltage entering the circuit. This way, we know when the voltage has exceeded a certain level or not.

To build this circuit, we're going to use zener diodes and a LED bar graph as the two main components. Zener diodes are special diodes that conduct current across them when their breakdown voltage is reached. We will exploit this principle in this circuit to allow this current to light LEDs. Then we will know that that particular voltage has been exceeded. For example, one of the zener diodes we will use is a 5.1V zener diode. If the LED that is connected in series with the zener diode turns on, then we know that the 5.1 voltage level has been exceeded.

This project serves to teach you about zener diodes and how simple voltage level indicators can be built using them.

Parts for Simple Voltmeter

- **1N4728 3.3V Zener Diode**

- **1N4733 5.1V Zener Diode**
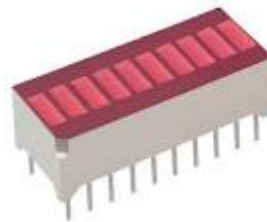
- **1N4739A 9.1V Zener Diode**

- **1N4742 12V Zener Diode**
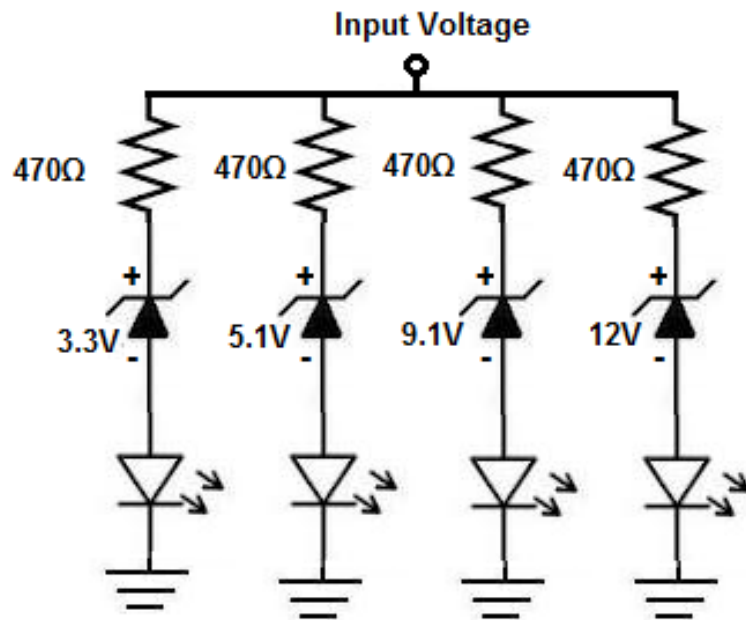
- **470Ω Resistors (Qty: 4)**

- **Bar graph LED**

Circuit for Simple Voltmeter

Schematic Diagram

Operation of Circuit

The voltage that is input into the circuit is shown in the schematic diagram as "Input Voltage." In each circuit, the LED(s) will glow when the input voltages reaches the breakdown voltage (V_z) of the zener diode. So, if the voltage, reaches, 3.3V (approximately, give or take), the first LED in the bar graph will turn on. If the voltage reaches 5.1V, the first and second LED will both turn on. If the voltage reaches 9.1V, the first three LEDS will turn on. And if the voltage reaches 12V, 4 LEDs of the bar graph LED will turn on.

The circuit again works on the fact that zener diodes will conduct current across them when their rated breakdown voltage is exceeded. This will then light the LED in series with the zener diode.

How to Build an AC-DC Voltage Converter

In this project, we will build a device that can convert an AC signal into a DC signal. This has tremendous application when we have an AC voltage, such as from a power plug, and we need to power a device that runs on DC voltage. AC-DC converters are very common. Knowing how to convert AC to DC can come in use if you're ever going to design an electronic device that gets plugged into an electrical outlet. An electrical outlet produces AC waves. Most devices operate on DC.

This circuit will serve to teach you about full wave rectifiers, smoothing capacitors, and voltage regulators, the three essential components for building AC-DC converters.

Full Wave Rectifiers

A full wave rectifier has valuable use when one wants DC output in exchange of an AC signal. A full wave rectifier converts the AC signal into DC, in the event that a device needs to be plugged into an AC outlet but needs to be powered and run on DC voltage.

**Pinout**

A full wave rectifier normally has 4 pins.
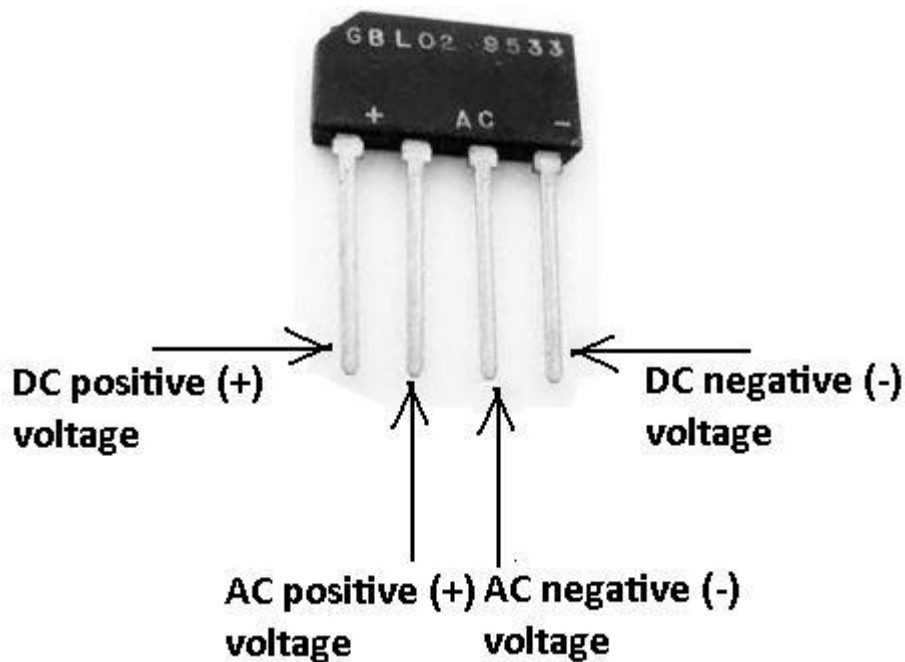
Pinout Terminals

The terminal that has the **+** symbol is the DC positive voltage. This is terminal that receives the converted positive DC voltage that is output by the rectifier.
The terminals that have the two AC signals (∿) are the two terminals where you place the incoming AC voltage, which you want to convert into DC. It doesn't matter the polarity. The positive and negative AC voltage can be switched.
The terminal that has the **-** symbol is the DC negative or ground voltage. This is the terminal that receives the converted negative DC voltage that is output by the rectifier.

This is a diagram to visualize how a full wave rectifier works:
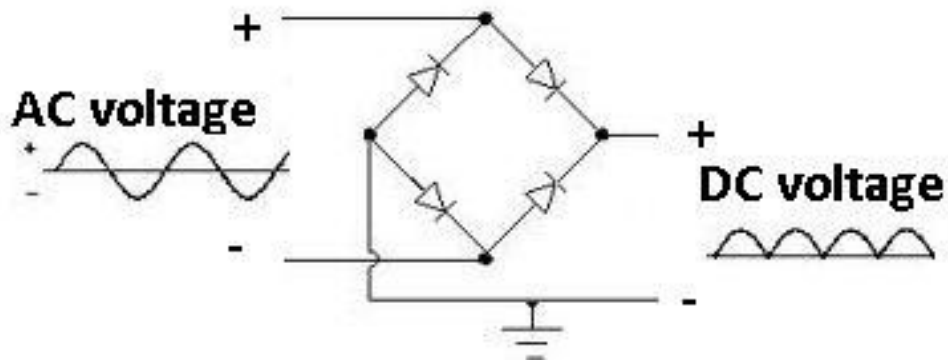
A full wave rectifier takes an AC signal input into it, such as the AC signal from a power outlet, and converts it into a DC signal. The AC is input. The DC is output.

To create a smoother and steadier DC signal, a smoothing capacitor is normally added in parallel to the output of the rectifier. A smoothing capacitor is a capacitor that acts to smooth or even out fluctuations in a signal. Below you can see the change that the smoothing capacitor brings.

And to make it even smoother and steady, a voltage regulator can then be added in parallel to that to produce a specific DC voltage that you want as output. In this circuit, we use a LM7805 voltage regulator, which outputs a regulated 5 volts DC.
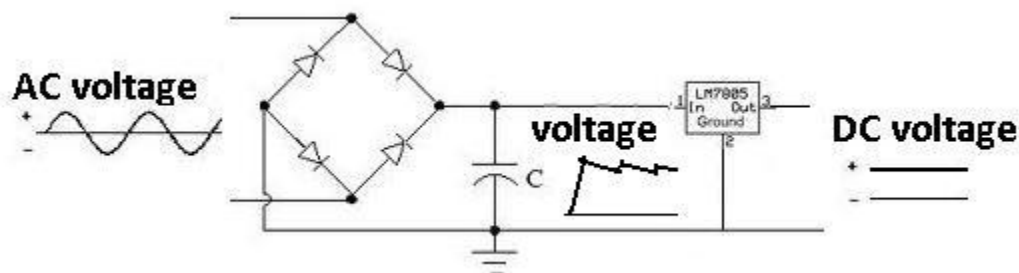
You can see after the smoothing capacitor that the voltage is pretty steady but not quite. After the voltage regulator, it should be nearly perfectly steady and even. This will produce the most precise DC voltage. A voltage regulator is a device that regulates a specific DC voltage output. It has 3 terminals, an input terminal, an output terminal, and ground. The input terminal receives the voltage that a user wants to regulate down. The
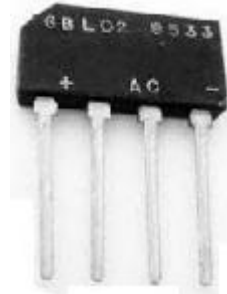
output terminal is where the regulated voltage is output from. And the ground terminal is for connection to ground.

In this circuit, we will put this altogether and you'll have a better understanding of how it all works.
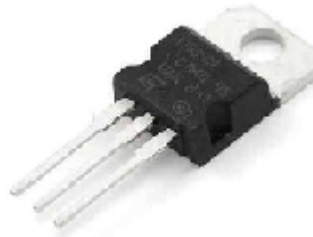
Parts for AC-DC Converter

-**Full Wave Rectifier**
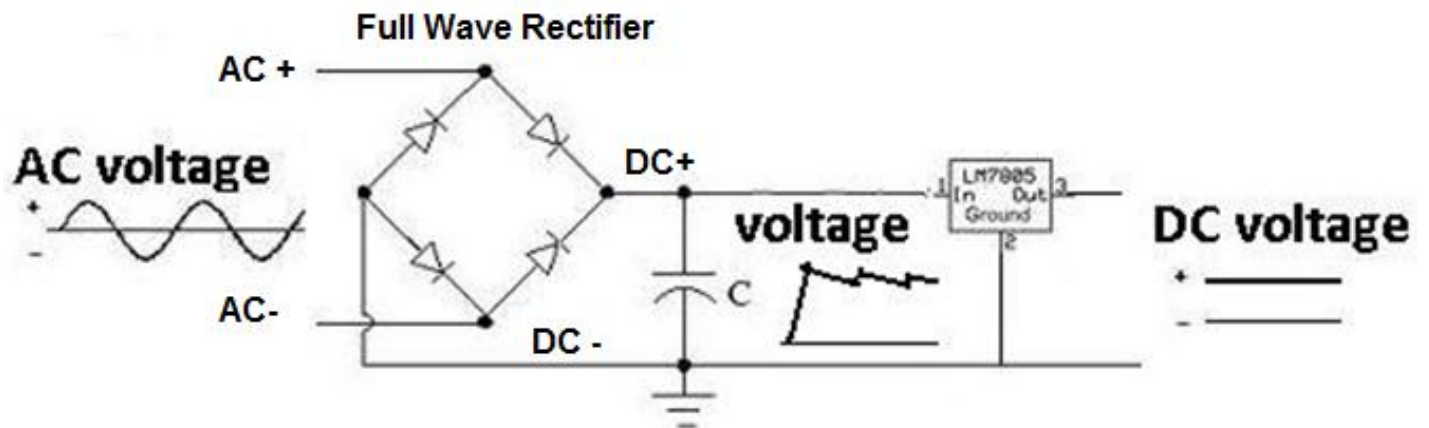
- **100µF Electrolytic Capacitor**

- **LM7805 Voltage Regulator**

Circuit for AC-DC Converter

Schematic Diagram

Full Wave Rectifier

AC voltage

AC +

AC-

DC+

DC -

voltage

C

LM7805
In   Out
Ground

DC voltage

Operation of Circuit

The circuit takes an AC signal and converts it into DC. The full wave rectifier takes the incoming AC signal and rectifies it so that all the voltage is above the 0V level. The smoothing capacitor then balances out fluctuations in this rectified signal so that it's steadier. The voltage regulator then perfectly flattens out the signal so that it's a (near) perfect DC signal. All of this is illustrated by the schematic diagram above.

Again, AC-DC converters are very popular. Most of the devices that you plug into an AC outlet convert the AC signals to DC, and then run off of DC power.

How to Build a Speaker Crossover Network

In this project, we build a speaker crossover network. A Speaker Crossover Network is a network that filters input audio sound signals by frequency, diverting low-frequency sounds to a woofer and high-frequency sounds to a tweeter.

The reason we form a network of these different speakers is because different speakers play different sounds better than others. We incorporate these 2 speakers to play the best possible quality audio we can. A woofer is a speaker that plays out low-frequency (bass) sounds with great audio quality. A tweeter would play out bass poorly. However, it plays out high frequency sounds with great quality. (Many times, too, a midrange speaker is added to play out midrange frequencies. However, for simplicity, we omit midrange speakers in this project.) To get a good speaker system, it's best to incorporate a woofer and tweeter together so that you get the best audio from the full audio spectrum, which is from 20 to 20,000Hz.

This project serves to teach you about woofers and tweeters, and how they can be incorporated into a network to get the best possible sound.

Parts for Speaker Crossover Network

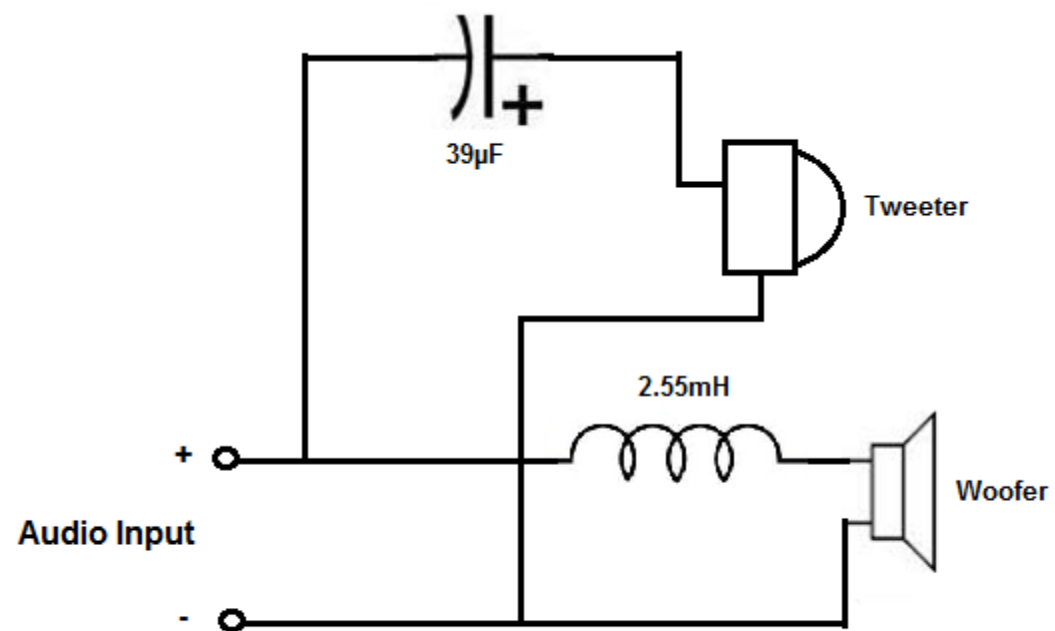- **Tweeter Speakers**

- **Woofer Speakers**

- **2.55mH Inductor**

- **39µF polypropylene, polyester, or metallized film Capacitor**

Circuit for Speaker Crossover Network

39μF

Tweeter

2.55mH

Woofer

+ o

Audio Input

- o

Operation of Circuit

The circuit operates on the principle of filtering systems. The input audio signals that enter the circuit are filtered by the inductor and capacitor network.

A capacitor is a device that has high reactance at low frequencies and low reactance at high frequencies. This means that a capacitor doesn't allow low-frequency signals to pass through it but allows high-frequency signals to do so. This is how a capacitor diverts the high-frequency sounds to the tweeter.

An inductor is a reactive device just like a capacitor but has the opposite reactive effect. An inductor has high reactance at high frequencies and low reactance at low frequencies. This means that an inductor doesn't allow high-frequency signals to pass through it but allows low-frequency signals to do so. This is how an inductor diverts the low-frequency sounds to the woofer.

When you have this type of network, it forms a speaker crossover network. Woofers can play out low-frequency sounds (bass) very well, and tweeters play out high-frequency sounds very well. Thus, you can incorporate a much better speaker system if you have these 2 speakers connected in a system rather than just a single general speaker.

How to Build a Mode Light Status Indicator

We've all seen electronic devices that have LEDs which represent what mode or status they are in. For example, printers are devices that have these LEDs often on them. When the green LED is lit, that most likely means it's ready to print. When the red LED is lit, it means it is in a paused status, either waiting on a queue or there is a jam or some error. Cameras are the same, where a green LED means it is ready to take a picture, while a red LED means there is some pause.

In this circuit, we will teach you a basic way how to create and light different LEDs to show the status of a device. This has tremendous application if you ever want to build a circuit where you want status LEDs to show whether it's in a ready state or a non-ready state, an ON state or an OFF state, or any various pairs that may exist.

In this project, we will exploit the principles of a Single Pole Double Throw relay. This is a relay which can be in any 1 of 2 states or modes. When wired in one way, it can light a green LED. When wired in the other way, it can light a red LED.

This project will serve to teach you how a SPDT relay works and how mode status indicators can be created.

Single Pole Double Throws (SPDTs)

Before we get into this project, we will go over extensively what SPDTs are and how they operate, so you will know how they work and how to connect them in a circuit.

First, a relay, plain and simple, is just a switch which is turned on when a voltage is fed to the coil. The relay is turned on when there is voltage and off when the voltage is removed.

A Single Pole Double Throw (SPDT) relay is a relay which has one input and can connect either to 1 of 2 outputs.

We will now extensively go over how to connect a SPDT so that you can understand the gist of this project:

Connecting a SPDT

In order to know how to connect a single pole double throw(SPDT) relay, you must know what each pin terminal represents and how the relay works.

Terminal Pins

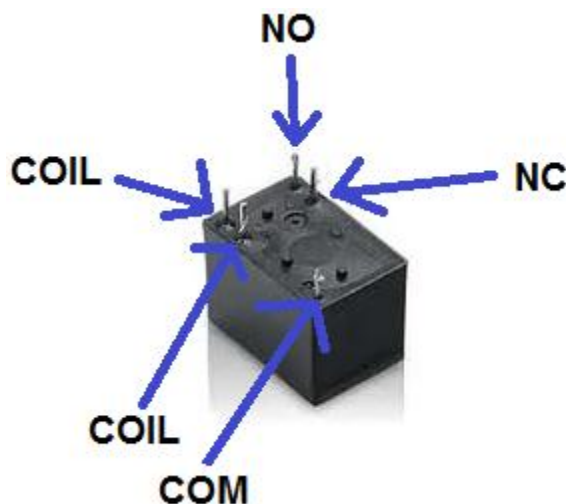A Single Pole Double Throw Relay comes with five terminal points.

The terminals are COIL, COIL, COM, and NO, and NC.

Single Pole Double Throw Relay Diagram
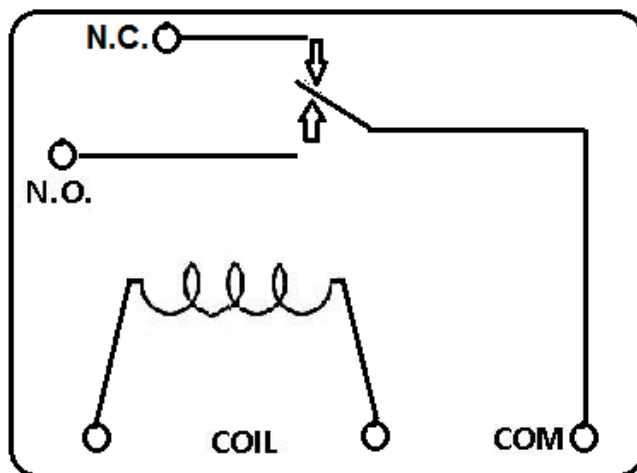
O represents the terminals of the relay

This correlates to this in the real-life relay:

Description of the Terminals

- **COIL**- This is one end of the coil.
- **COIL**- This is the other end of the coil. These are the terminals where you apply voltage to in order to give power to the coils (which then will close the switch). Polarity does not matter. One side gets positive voltage and the other side gets negative voltage. Polarity only matters if a diode is used.
- **NO**- This is Normally Open switch. This is the terminal where you connect the device that you want the relay to power when the relay is powered, meaning when the COIL receives sufficient voltage. The device connected to NO will be off when the relay has no power and will turn on when the relay receives power.
- **NC**- This is the Normally Closed Switch. This is the terminal where you connect the device that you want powered when the relay receives no power. The device connected to NC will be on when the relay has no power and will turn off when the relay receives power.
- **COM**- This is the common of the relay. If the relay is powered and the switch is closed, COM and NO have continuity. If the relay isn't powered and the switch is open, COM and NC have continuity. This is the terminal of the relay where you connect the first part of your circuit to.

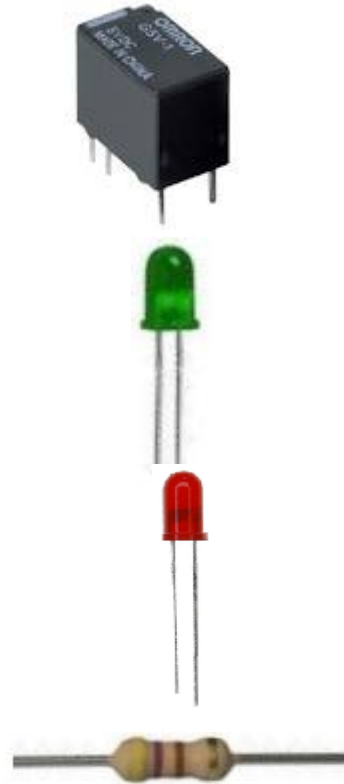Now that we know what each terminal pin represents, we now will wire a SPDT circuit.

Parts for Mode Light Status Indicators
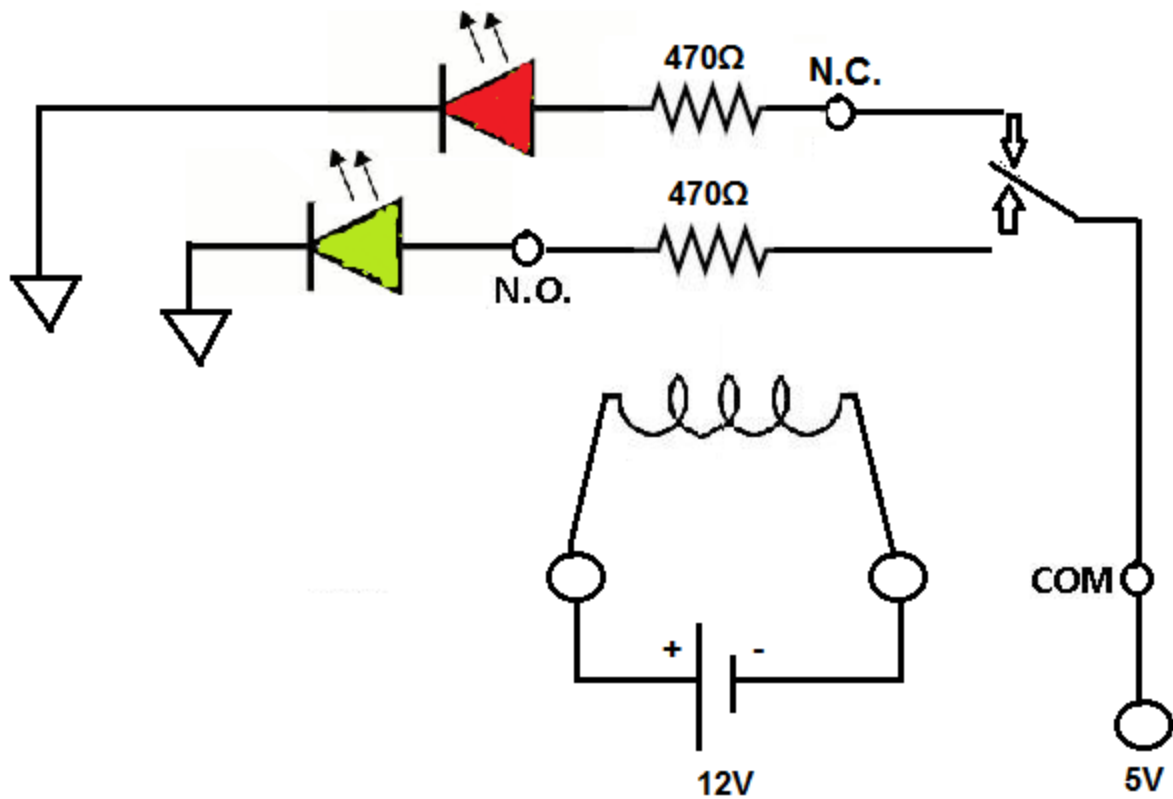
- **SPDT relay**

- **Green LED**

- **Red LED**

- **470Ω Resistors (Qty: 2)**

Circuit for Mode Light Status Indicator

Schematic Diagram

The SPDT relay that you get most likely will be one which operates on 12V. This means that you must feed the coil of the relay 12V in order for it to operate and function.

The 470Ω resistors are there as current-limiting resistors so that the LEDs won't get burnt out by excess current.

Operation of Circuit

For this circuit, you will need two separate voltages. You will need 12V and you will need 5V. The 12V is to operate the SPDT relay. This 12 volts is needed so that the relay has sufficient voltage to power on. Without this power, the relay cannot work. The 5 volts is for the main circuit. The main circuit (outside of the SPDT) needs 5 volts to operate. This is so that when the relay is not powered on, the other LED, the red LED, will turn on.

This is how the circuit works:

Normally when the relay is not receiving any power (without the 12V), the red LED of the circuit will be lit. This means that the relay is not powered.

When the relay is powered on and the coil is receiving 12V, the relay switch will close to the Normally Open (NO) terminal. This is the terminal which contains the green LED. So when the relay is powered, the green LED will light up, and the red LED will shut off.

You can see now how this circuit acts as a mode status indicator. When fully powered, the green LED will be lit, meaning it is powered. When not fully powered (the relay isn't receiving any power), the red LED will be lit, meaning it is not fully powered.

(The 5-volt source must always be on).

Again, this circuit has tremendous application, as cameras, printers, and many electronic devices have status LEDs indicating the mode of the device.

How to Build a Latching Switch

A Latching Switch is a very important circuit because it's used in many real-world circuits. It will be explained now what it is. A latching circuit is a circuit that once triggered on, stays on, until the whole system is disabled. Think about alarm circuits. When the circuit is triggered, the alarm goes off and stays on for as long of a period as time until it's disabled completely.

In other words, a latching circuit turns on and even when power is removed from the source that turned it on, will still remain on.

In this project, we use a silicon-controlled rectifier to acts as a latch in the circuit to allow the behavior which is described above.

We will build an SCR latch to turn on a buzzer.

Silicon-controlled Rectifier

A Silicon-Controlled Rectifier is a three-lead semiconductor device that acts as an electrically controlled switch.

When a specific positive trigger voltage/current is applied to the SCR's gate lead (G), a conductive channel forms between the anode (A) and the cathode (C) leads. Current flows in only one direction through the SCR, from anode to cathode, like a diode.
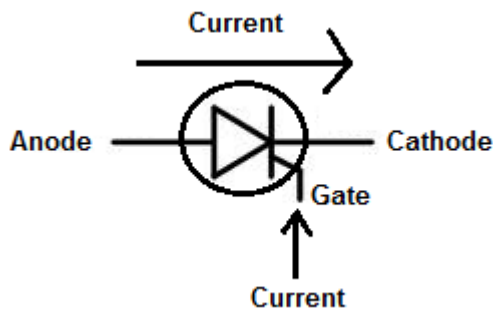
The unique characteristic of silicon-controlled rectifiers has to do with its conduction state after the gate current is removed. After an SCR is triggered into conduction (by allowing sufficient current to the gate lead), removing the gate current has no effect. The conduction from anode to cathode continues. That is, the SCR will remain on even when the gate current/voltage is removed. The only way, then, to turn the device off is to remove the anode-to-cathode current or to reverse the anode and cathodes polarities.

In terms of applications, SCRs are used in switching circuits, phase-control circuits, inverting circuits, clipper circuits, and relay-control circuits. One of the most important circuit applications is in alarm circuits. Think of how when an alarm is triggered. Once it is triggered, the alarm stays on, until an external source shuts it off. SCRs remain off until shut off, making them perfect in alarm circuit applications.

Switches

Normally Open Push Button

A Normally Open (NO) Push Button is a push button that, in its default state, makes no electrical contact with the circuit. Only when the button is pressed down does it make electrical contact with the circuit.

Normally Open Push Buttons don't make electrical contact with the circuit when not pressed down.
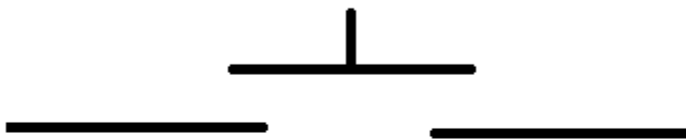
Once pressed down, normally open push buttons now make contact and the circuit is now closed and powers or turns on the respective part the button was made for.

When the button is pressed down, the switch makes electrical contact and the circuit is now closed. Therefore, electricity can now flow to the other part of the circuit connecting to the push button and make the device turn or power on the respective part.

Normally Closed Push Button

A Normally Closed (NC) Push Button is a push button that, in its default state, makes electrical contact with the circuit.

Normally Closed Push Buttons make electrical contact wtih the circuit in its default state, when not pressed.

Once pressed down, normally closed push buttons no longer make electrical contact with the circuit and now forms an open circuit. This means it will no longer power or turn on the device it was made for.
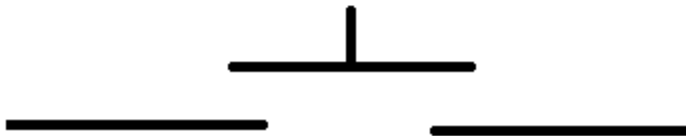
When the button is pressed down, the switch no longer makes electrical contact and the circuit is now open. Therefore, electricity can no longer flow to the other part of the circuit to turn or power on the respective part of the circuit the button was made to switch.

Parts for Latching Switch Circuit

- **Silicon-controlled Rectifier**

- **Normally Open Push Button**

- **Normally Closed Push Button**
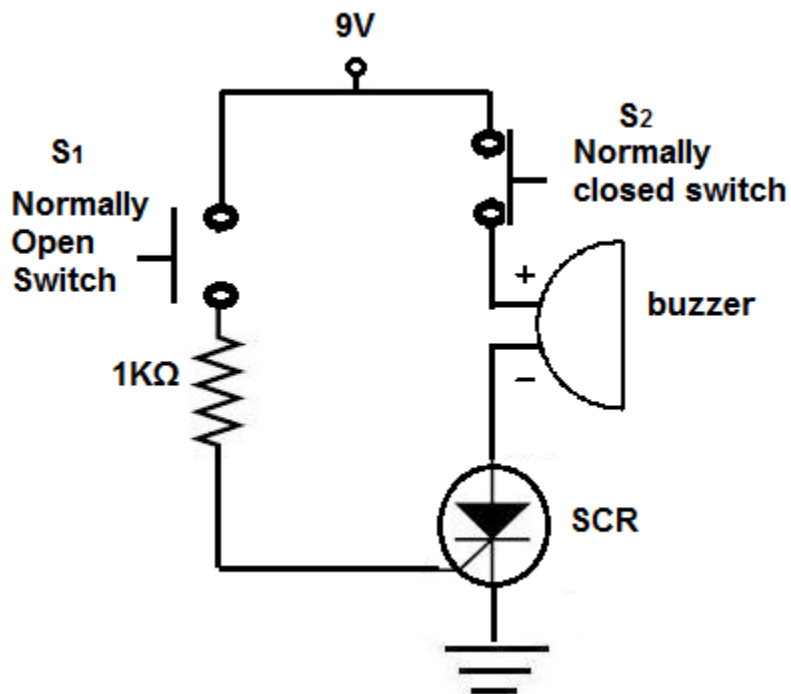

- **3-6 Volt Buzzer**

- **1KΩ Resistor**

Circuit for Latching Switch

Schematic Diagram

Operation of Circuit

When the S_1 switch is pushed in and released, a small pulse of current enters the gate of the SCR, turning it on. Current will then flow through the load, sounding the buzzer. The buzzer will remain on as long as S_2 is not pushed. Pushing S_1 as this point has no effect on the circuit. A latch stays on indefinitely regardless of what occurs to the trigger mechanism (S_1). The only way to dislodge the circuit (turn off the buzzer) is to stop current from flowing through the load. To do this, S_2 is pushed, opening the circuit.

Again, latching circuits are very important and form the basis of many circuits.

How to Build a MOSFET Light Dimmer

In this project, we are going to use an N-channel enhancement-type power MOSFET to control the current flow through a lamp.

This project serves to teach you how to connect a MOSFET and how you can connect it to a circuit so that the current that flows through it can be adjusted. This, in turn, will adjust the level of light that is emitted by the lamp. This is how the MOSFET circuit we will build will act as a light dimmer.

When the current flowing through the MOSFET is high, the lamp will be near its brightest level. When the current flowing through the MOSFET becomes lower, the lamp dims and emits less light.

Before we begin the circuit, we will thoroughly go over the MOSFET that we are going to use:

An N-Channel Enhancement-type Power MOSFET

Enhancement-type MOSFETs

We will go over first what an enhancement-type MOSFET is.

An Enhancement-type MOSFET is a MOSFET that is "normally off."

When you connect an enhancement-type MOSFET to a circuit, no current flows from source to drain when there isn't any voltage applied to its gate. This is why it is called a normally off device. There is no current flow without a gate voltage.
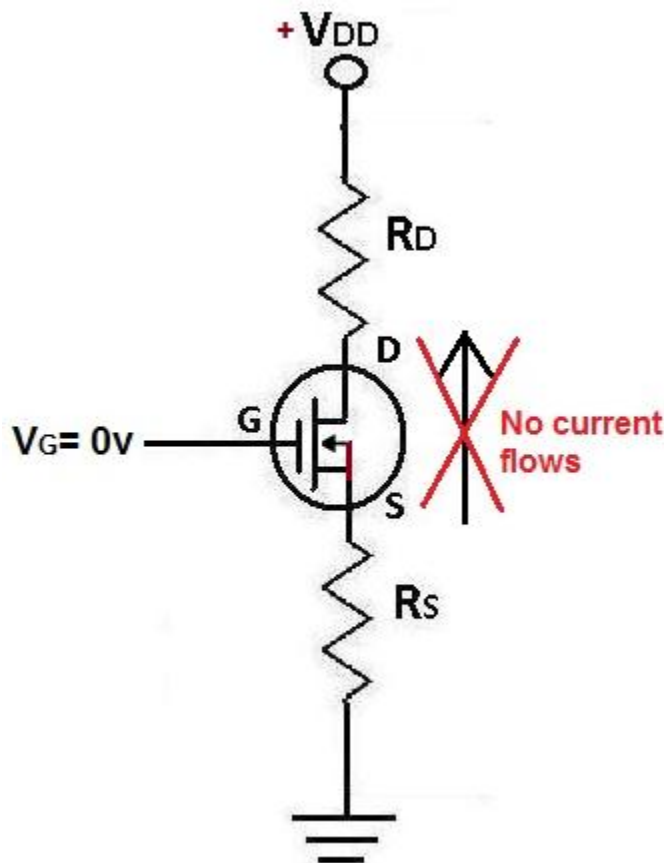
However, if a voltage is applied to the gate lead of the MOSFET, the drain-source channel becomes less resistive. As the gate voltage increases more and more, the current flowing from source to the drain increases more and more, until maximum current is flowing from source to drain.

An enhancement-type MOSFET is so named an enhancement device, because as the voltage to the gate increases, the current increases more and more, until at maximum level. The current enhances as the voltage increases.
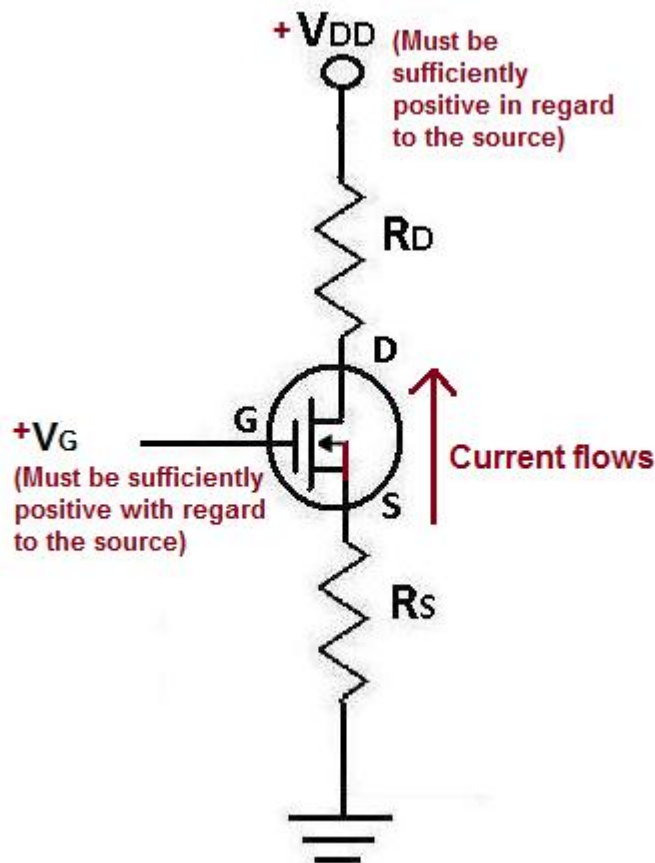
N-Channel MOSFETs

An N-Channel MOSFET is a MOSFET where the charge carriers in the channel are electrons. This means that when we are biasing an N-Channel MOSFET, we apply positive voltage to the drain and positive voltage to the gate in order to turn it on.

Operation of an N-Channel Enhancement MOSFET

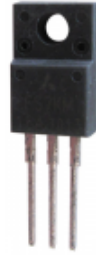So in order to operate an enhancement-type MOSFET, there are two requirements:

1) Sufficient positive voltage must be fed to the gate of the MOSFET.
2) Sufficient positive voltage must be fed to the drain of the MOSFET.

So that, the voltages connected to the MOSFET will be as such:

Parts for MOSFET Light Dimmer

- N-Channel Enhancement Power MOSFET

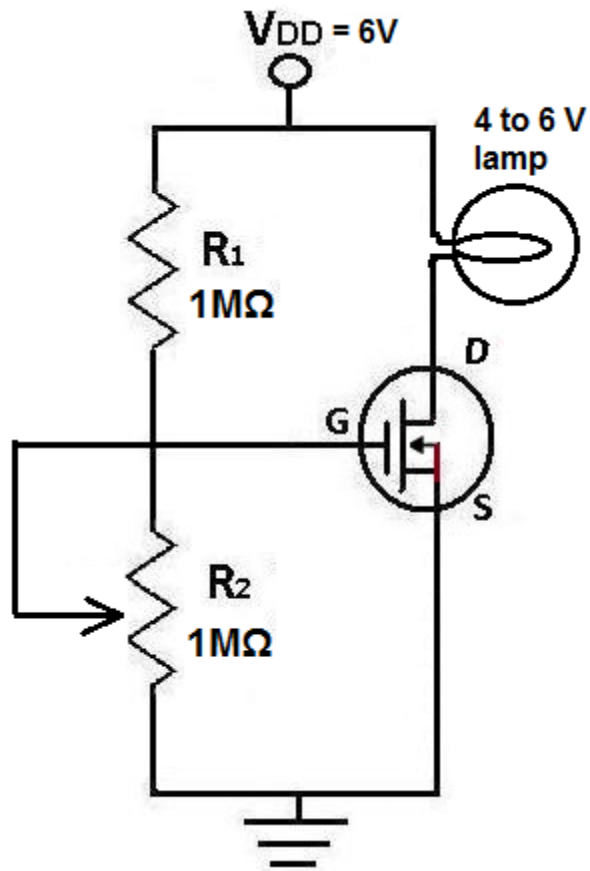- 4-6V Lamp

- 1MΩ Resistor

- 1 MΩ Potentiometer

The N-Channel Enhancement MOSFET can be any. For this project, the one which I used was the Mitsubishi# FS7KM.

Any lamp can be used as well, as long as it's 4-6V, it will function for this project. Many lamps have 12V requirements. Make sure that the lamp that you don't use these types.

Circuit for MOSFET Light Dimmer

Schematic Diagram

Operation of Circuit

The potentiometer R_2 is what allows this MOSFET circuit to act as a dimmer. When the potentiometer is at its full resistance, 1 MΩ, the light from the lamp is at its lowest. When the potentiometer is turned so that its resistance decreases, the lamp gets brighter and brighter, until near 0Ω, it reaches its full brightness level.

You may think you could have done this circuit without the MOSFET and just connected voltage and a potentiometer to the lamp. Why add a MOSFET? The reason is, the MOSFET, apart from acting as a switch, also amplifies the current, so it makes sure that the lamp receives sufficient current to power on. So it acts as a switch and amplifier.

How to Build a Solar Cell Battery Recharger

This project will teach how you can recharge batteries using solar cells. A solar cell is a device that takes energy from light and converts it into electricity. Solar cells, then, serve to harvest energy from light and convert it into electricity; the current they output then recharges the batteries. Usually one single solar cell unit (unless very large) will not be enough to recharge a battery. Multiple solar cells then must be tied together in order to output enough current to recharge a battery.

This project serves to teach you how to connect solar cells in a way that will be able to recharge batteries. For this, we must know how to wire solar cells

How to Connect Solar Cells

The importance of this project lies in knowing how to connect solar cells properly in a circuit. We will go over connecting them in series and in parallel.

How to Connect Solar Cells In Series

To connect solar cells in series, you tie the negative terminal of one solar cell to the positive terminal of the next cell and keep on doing this to tie all of the cells in series. This is shown below:
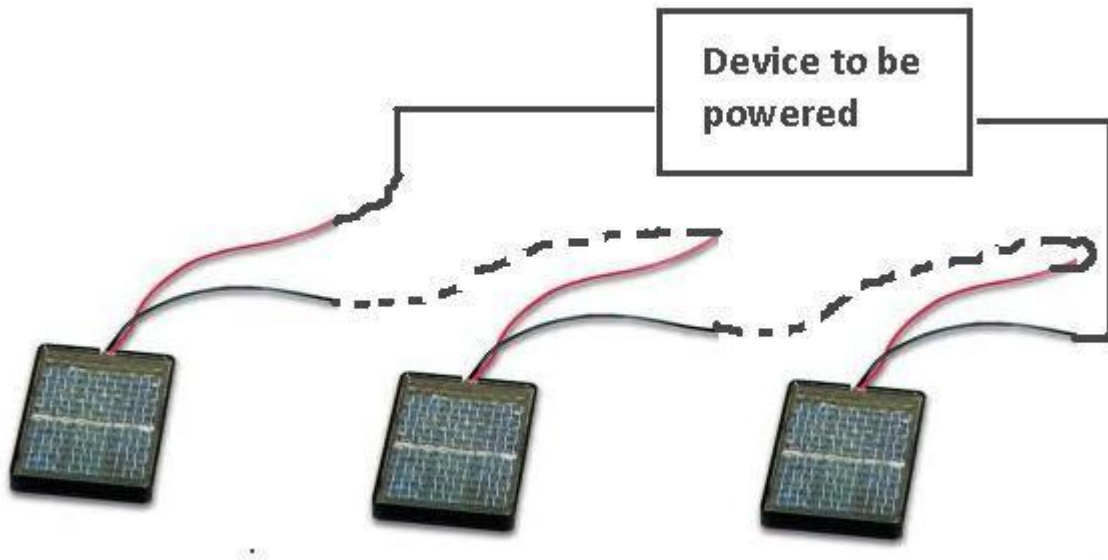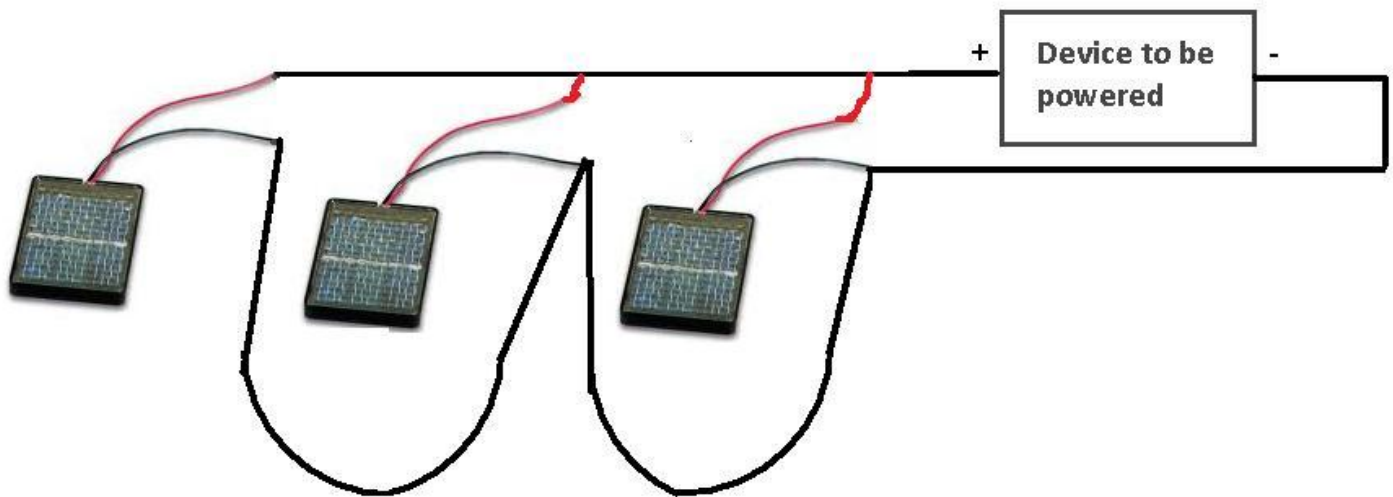

When you connect solar cells in series, the voltage of each cell adds up. You increase the net voltage of the circuit. For example, if you tie 3 solar cells together and each has a voltage rating of up to 0.5V, the net voltage will be 1.5V, since the 3 voltages add together.

In series, voltages add. Current stays the same. So if each solar cell had a current rating of up to 0.1A in bright light, the circuit will output 0.1A in series.

How to Connect Solar Cells In Parallel

To connect solar cells in parallel, you tie all the positive terminals of the cells together to form a common positive connection, and you tie all the negative terminals of the cells together to form a common negative connection. This way, all the cells are in parallel.

This is shown below:

When you connect solar cells in parallel, the current of each cell adds up. You increase the net current of the circuit. For example, if you tie 3 solar cells together and each has a current rating of up to 0.1A in bright light, the net current will be 0.3A, or 300mA, since the 3 currents add together. In parallel, currents add. Voltage stays the same. So if each solar cell had a voltage rating of 0.5V, the circuit will output 0.5V.

Parts for Solar Cell Battery Recharger

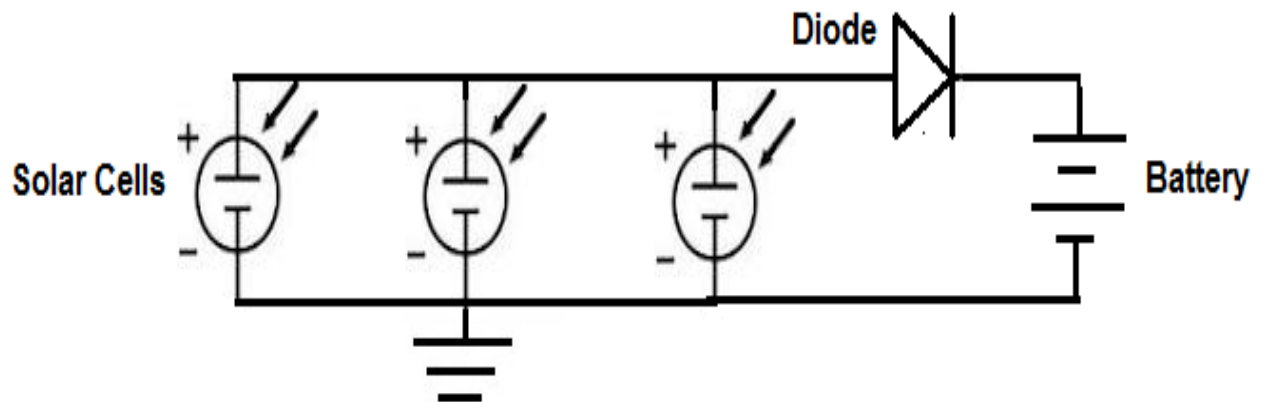- **3V 0.5A Solar Cell (Qty:3)**

- **1N400X Diode**

- **Rechargeable Battery**
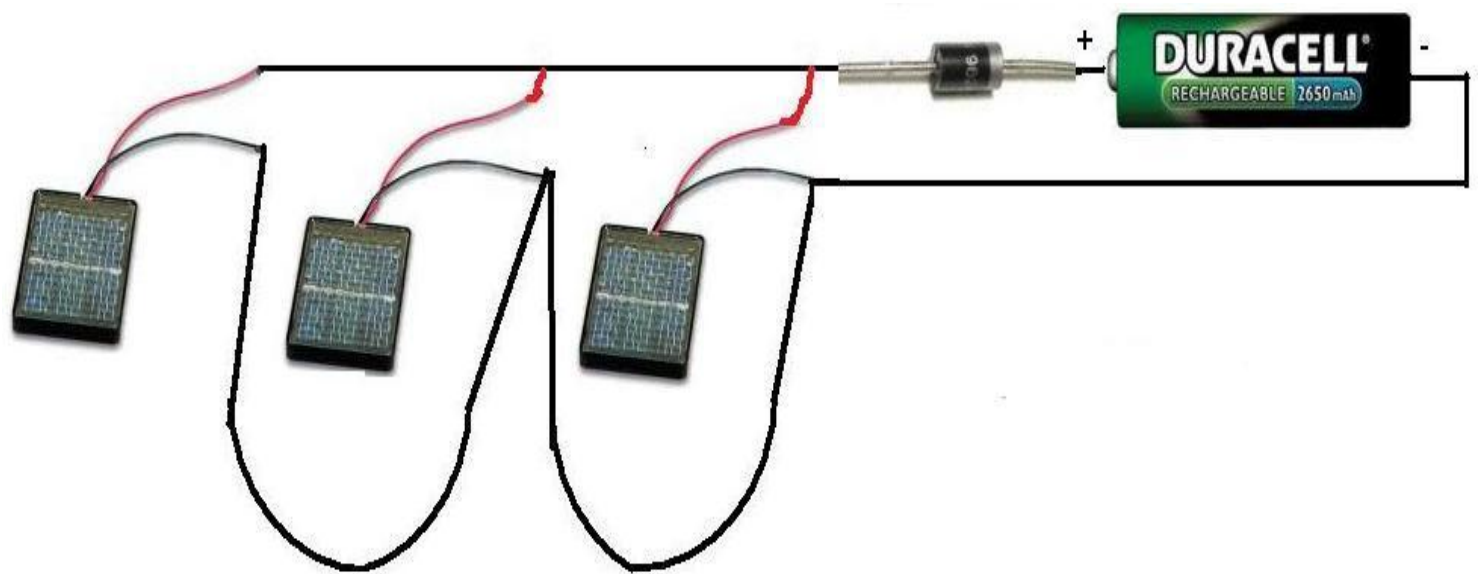
- **AA Battery Holder**

- **Alligator Clips**

Circuit for Solar Cell Battery Recharger

Real Life Representation of Circuit

Operation of Circuit

To recharge a battery with a solar cell, all you need is a solar cell that outputs the current necessary to recharge the battery.

When recharging a rechargeable battery, the battery comes with a specification to the amount of current that is needed to recharge it. This is the current that you have to make sure to feed the battery in order for it to charge properly to full strength.

In our example, we will use a solar panel that can output a maximum of 3 volts and a maximum of 0.1A (or 100mA) when exposed to bright light. The battery we are going to charge, a 'AA' Duracell 2650mA, needs 270mA for 16h in order to charge to its full potential.

Since each solar cell can only output 100mA and the battery needs 270mA to charge, we need to take 3 solar cells and place them in parallel in order to increase the current output to acceptable charging levels for the battery.

Once we have the 3 solar panels in parallel, the current output will be about 300mA. The reason we connect a diode in the circuit is to keep the charged battery from leaking back current into the solar cells (when exposed to dark). We have omitted the battery holder and alligator clips in this above illustration, but you will need these for connecting the circuit.

All one must do is keep this circuit exposed to bright light for about 1 day or 2 days and the battery will be fully charged. This is how solar panels can charge batteries.

How to Build a Tone Generator

In this project, we will build a device that is able to generate different audio tones. Just like a function generator is capable of outputting different frequency signals by the varying of the frequency knob, in this circuit, we will have a potentiometer control the audio signal which is to be played out.

If you're ever interested in building your own function generator, this could be a basic method of how to achieve this aspect of it.

This project will serve to teach you about RC circuits and UJTs.

Parts for Tone Generator Circuit

- **2N4891 or Similar Unijunction Transistor**

- **1KΩ Resistor**

- **100Ω Resistor**
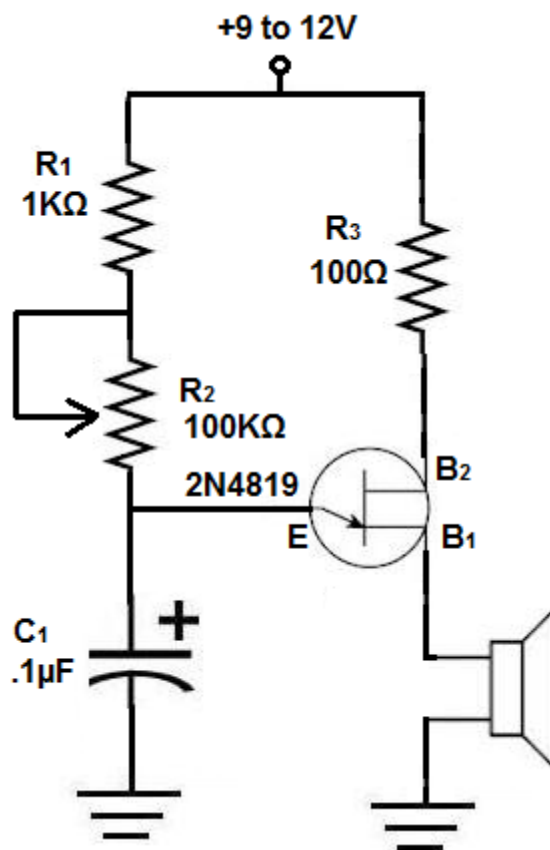
- **100KΩ Potentiometer**

- **0.1 µF Capacitor**

- **8-ohm Speaker**

Circuit for Tone Generator

Schematic Diagram

Operation of Circuit

The potentiometer R_2 and the capacitor C_1 determine the frequency of the audio tone which is generated. Adjusting the R_2 potentiometer changes the tone that the speakers play out. Turning the potentiometer so that resistance is greater creates a lower frequency. Turning the potentiometer so that the resistance decreases creates a higher frequency.

The audio signal is then passed through the UJT, which then emits the signal through the speakers, which plays the signal.

Note: The R_1 and R_3 resistors are just for biasing purposes, to limit the amount of current passing through the UJT circuit.

How to Build an Automatic Night Light

In this circuit, we build an automatic night light. This is a light that comes on automatically when it gets dark. For example, if you're a person who stays up late and reads or does some other activity, you would find this very useful, because as soon as your room gets dark, this light will turn on.

This project serves to teach you about Light Dependent Resistors (LDRs) (also called photoresistors). These are resistors that have very high resistance when exposed to dark but very low resistance when exposed to light. They change according to light intensity they are exposed to. We use this principle that LDRs have to create a circuit that turns on an LED in the dark.

Parts for Automatic Night Light Circuit

- **1N400X Diode**

- **100KΩ Potentiometer**

- **Light Dependent Resistor**

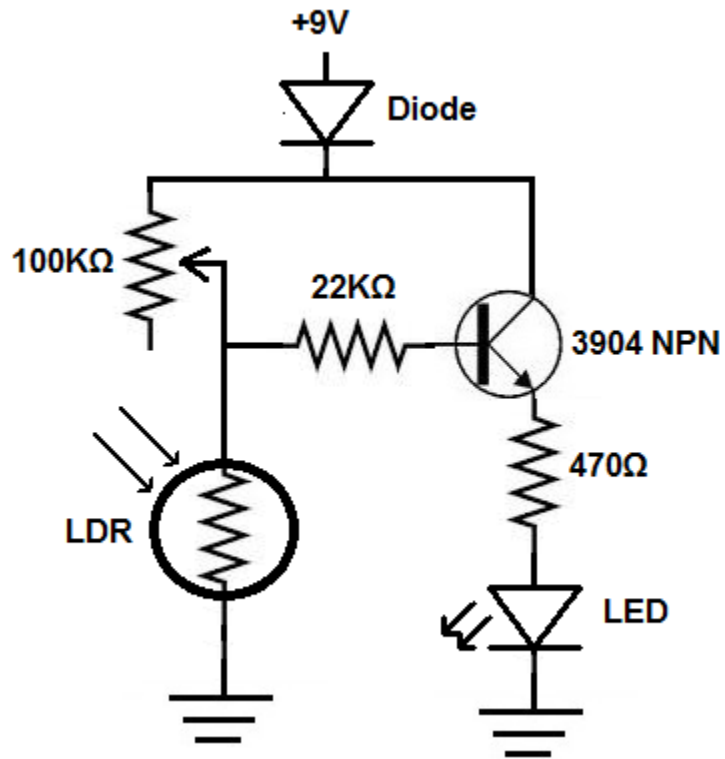- **22KΩ Resistor**

- **470Ω Resistor**

- **NPN 3904 Transistor**

- **5mm LED**

Circuit for the Automatic Night Light

Schematic Diagram

The potentiometer should be set normally at 50KΩ. This will be discussed more in detail below.

Operation of Circuit

When the circuit is exposed to darkness, the resistance of the LDR (or photoresistor) is extremely high, possibly several megohms, depending on the rating of the photoresistor. (Try to use one with a rating of at least 1MΩ. This means that the resistance of the photoresistor will be 1MΩ when exposed to complete darkness.) Current takes the path of least resistance. So practically all of the current goes through the base of the transistor, turning on the transistor and the LED.

When the circuit is exposed to bright light, the resistance of the LDR falls dramatically, possibly to a few hundred ohms. Current again takes the path of least resistance. Now practically all the current travels through the LDR to ground, since now that is the path of least resistance. Almost none travels through the base of the transistor and the current is not sufficient to turn it on. Thus, the LED is off whenever the circuit is in a lighted area.

This is how this circuit functions as an automatic night light.

Note: This circuit contains a 100KΩ potentiometer which acts as a calibrator. Normally, this should be set to about 50KΩ. However, this may need some adjustment. Adjust the potentiometer as needed so that in darkness, the light turns on, and when exposed to light, dims out. It should be about 50KΩ but different light intensities could have an effect, so experiment with this.

Thank you for reading this eBook. I hope you enjoyed it and learned a lot at the same time.