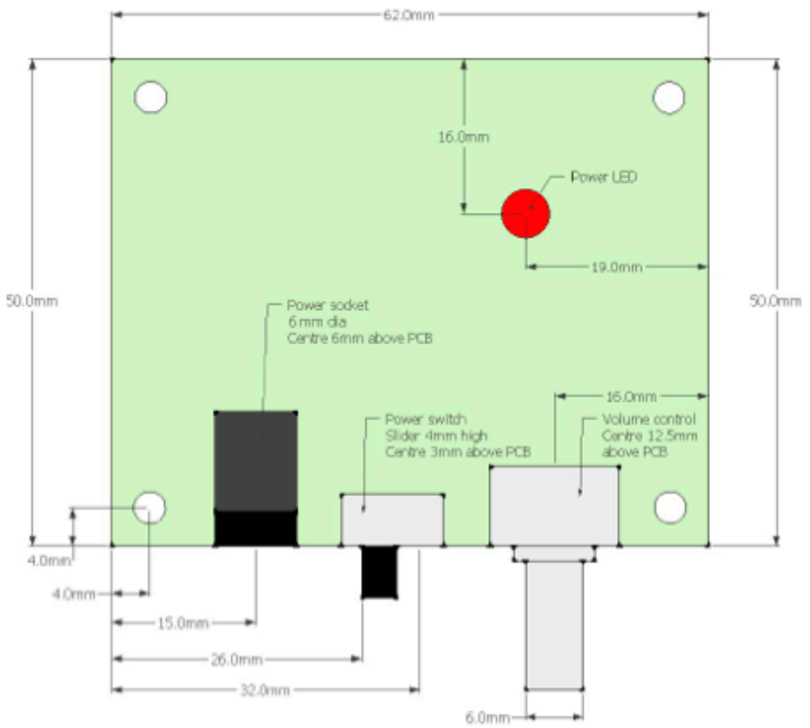
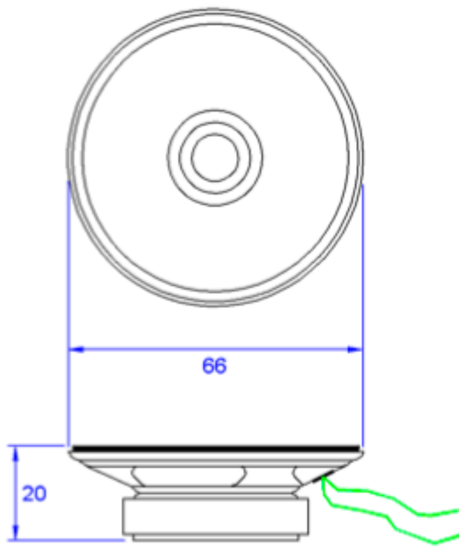
Designing the Enclosure

When you design the enclosure, you will need to consider:

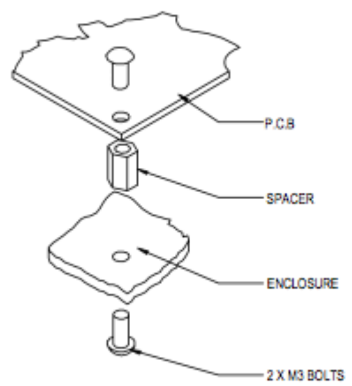
- The size of the PCB (below left, height including components = 15mm)
- How big the batteries are (if used).
- How to mount the two speakers (below right).
- How to allow the audio cable out of the box.
- Are you making the amplifier for a particular MP3 player, if so should the MP3 player go in the box?
- Position of the volume control, DC power jack, switch and LED.

These technical drawings of the amplifier PCB and speaker should help you plan this.





All dimensions in mm



Mounting the PCB to the enclosure

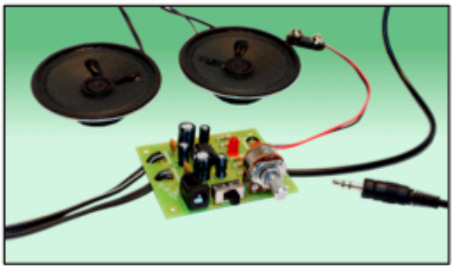
The drawing to the left shows how a hex spacer can be used with two bolts to fix the PCB to the enclosure.

Your PCB has four mounting holes designed to take M3 bolts

Enclosure Prototype

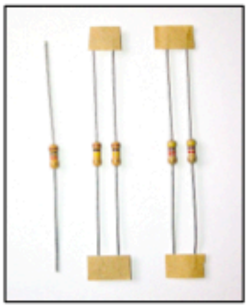
Using card board, foam or anything else that is suitable, make proto type of your enclosure design. This will give you the chance of changing any aspects of the design that do not work as well as expected.

Build Instructions



Before you put any components in the board or pick up the soldering iron, just take a look at the Printed Circuit Board (PCB). The components go in the side with the writing on and the solder goes on the side with the tracks and silver pads.

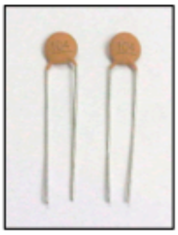
You will find it easiest to start with the small components and work up to the taller larger ones. If you've not soldered before get your soldering checked after you have done the first few joints.



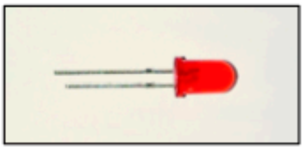
Step 1
Start with the five resistors (shown right):
The text on the PCB shows where R1, R3 etc go. Make sure that you put the resistors in the right place.

R1 is 1K (brown, black, red coloured bands)
R3 and R4 are 4.7Ω (yellow, purple, gold coloured bands)
R5 and R6 are 10K (brown, black, orange coloured bands)

Step 2
Solder the Integrated Circuit (IC) holder in to IC1. When putting this into the board, be sure to get it the right way around. The notch on the IC holder should line up with the notch on the lines marked on the PCB. Once this has been done insert the 8 pin IC into this socket making sure that the notch on the device matches the notch on the IC holder.



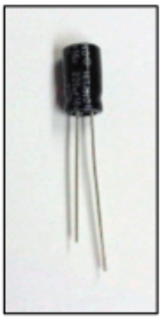
Step 3
There are two ceramic disc capacitors (as shown right). These should be soldered into C6 and C7. It does not matter which way around they go.



Step 4
Solder the LED (as shown left) into the PCB where it is labeled LED1. When putting it into the board make sure the flat edge on the LED matches the outline on the PCB.

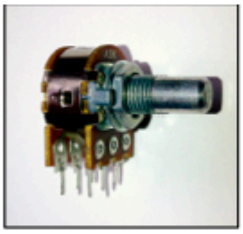
Step 5
Solder the PCB mount right angled on / off switch (shown right) into SW1. The row of three pins that exit the back of the switch must be soldered, but it doesn't matter if you can't solder the other two pins.

Step 6
Solder the DC power socket (shown left) into the PCB where it is labeled CONN1.



Step 7
Now solder in the five electrolytic capacitors (an example is shown right). The capacitors have text printed on the side that indicates their value. The capacitors are placed as:
C1 and C2 = 100μF. C3 = 10μF. C4 and C5 = 470μF.
Make sure the capacitors are the correct way around. The capacitors have a '-' sign marked on them which should match the same sign on the PCB.

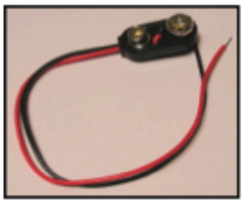
Step 8
Solder the dual potentiometer (shown right) into the PCB where it is labeled R2. Make sure the volume knob is facing away from the PCB.

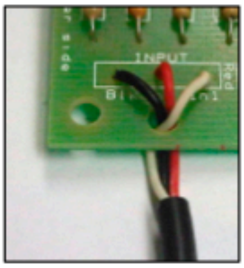


Step 9
The kit is supplied with a metre of twin cable. This cable will be used to connect the two speakers. You will need to cut this to the required length for each speaker in your enclosure design.

Take each piece of wire that you have cut off and strip the ends of the wire. Connect one end of each to the two terminals on the speaker (shown above left), and the other end of each to the terminals on the PCB marked SPEAKER1 and SPEAKER2, after feeding it through the strain relief hole. It does not matter which way around these connections go.

Step 10
The PP3 battery clip (shown right) should be attached to the terminals labeled POWER. Connect the red wire to '+' and the black wire to '-' after feeding it through the strain relief hole.





Step 11
The stereo jack / iPod lead (see picture left) should be connected to the 'INPUT' terminal. First feed the wires through the strain relief hole. The black wire should be connected to the terminal labeled 'BLK'. The other two can go the either of the two remaining inputs.

Checking Your Amplifier PCB

Carefully check the following before you insert the batteries:
Audio equipment may become damaged if connected to an incorrectly built amplifier.

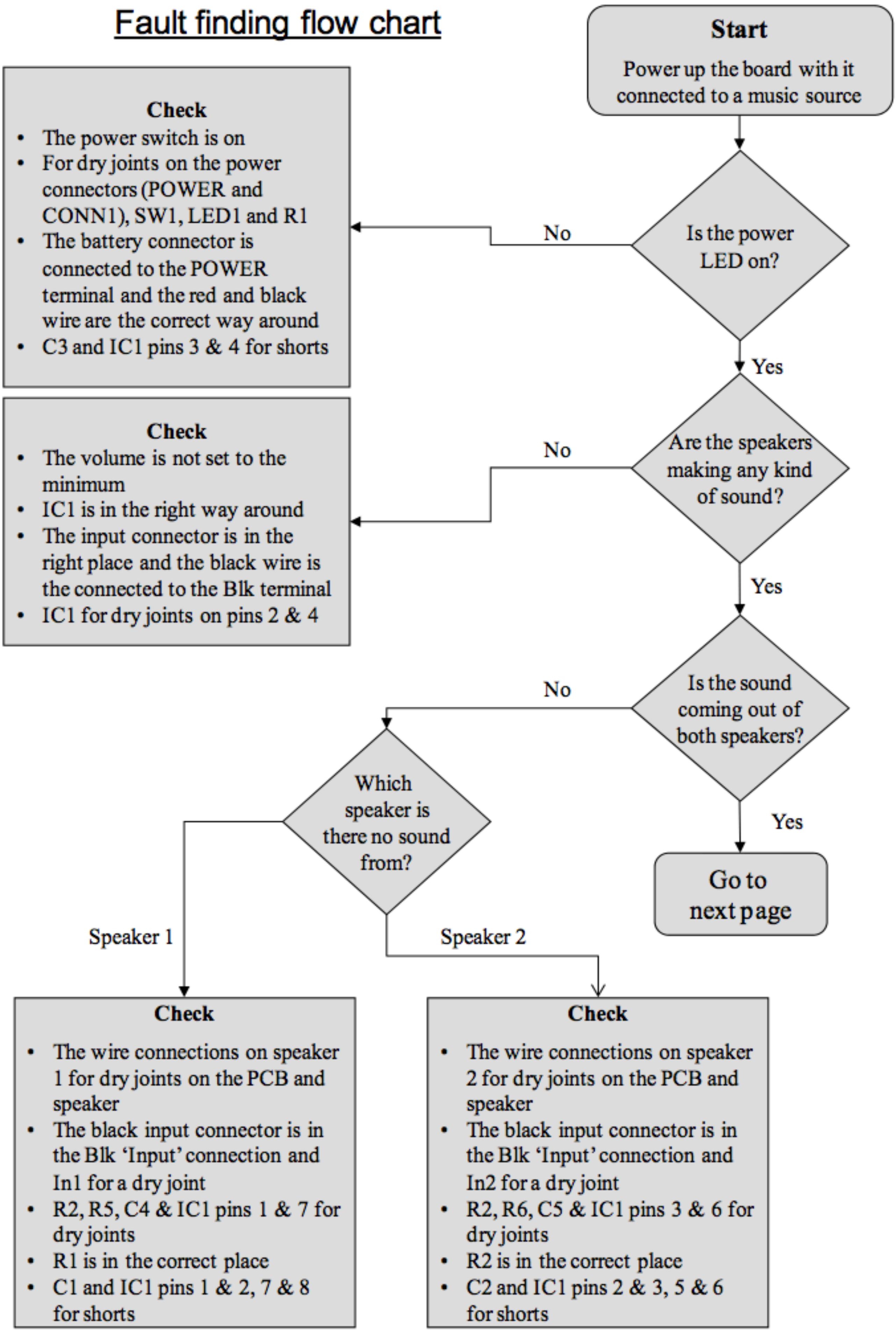
Check the bottom of the board to ensure that:

- All holes (except the 4 large (3 mm) holes in the corners) are filled with the lead of a component.
- All these leads are soldered.
- Pins next to each other are not soldered together.

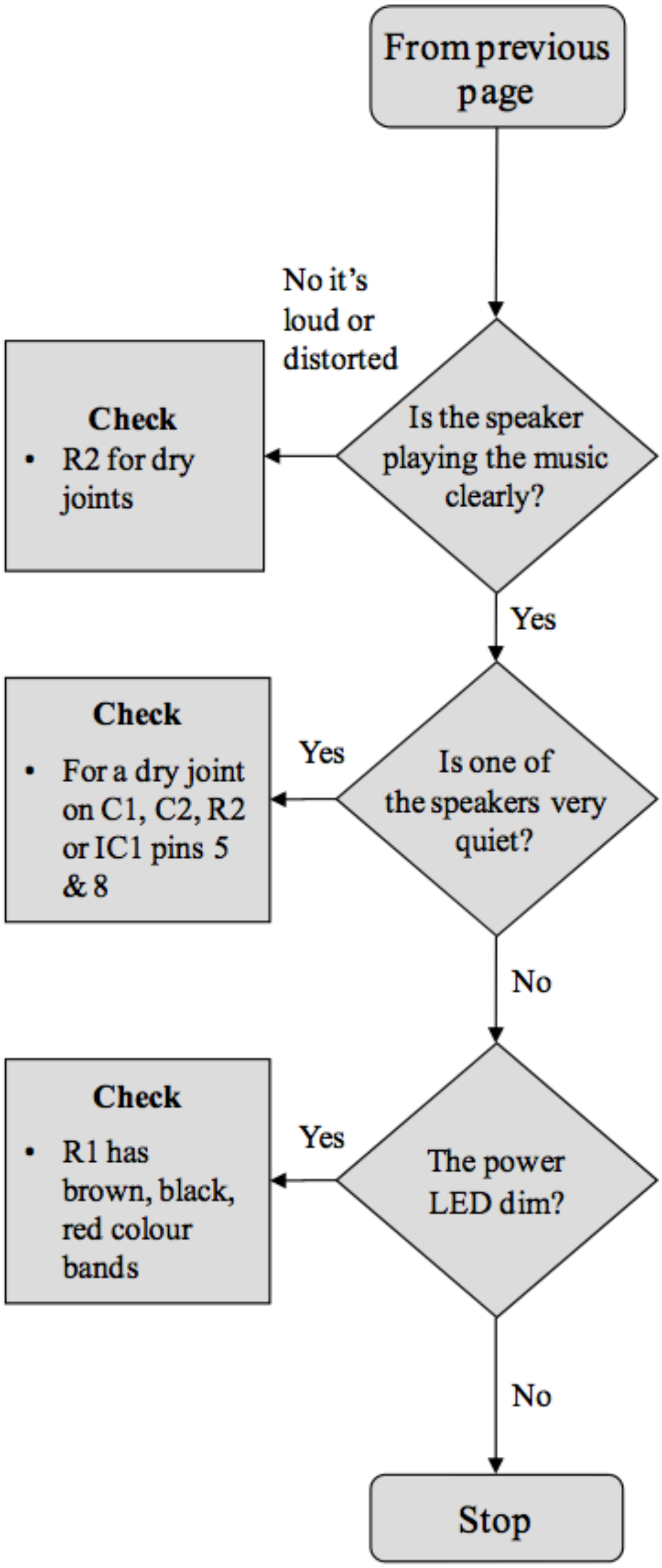
Check the top of the board to ensure that:

- The speakers, power lead etc are connected to the right place.
- The '-' on the capacitors match the same marks on the PCB.
- The colour bands on R1 are brown, black, red.
- The colour bands on R3 & R4 are yellow, purple, gold.
- The colour bands on R5 & R6 are brown, black, orange.
- C3 is a 10μF capacitor.
- The red and black wires on the battery clip match the red & black text on the PCB.
- The notch on the IC is next to C1 & C4.
- The flat edge on the LED matches the outline on the PCB.

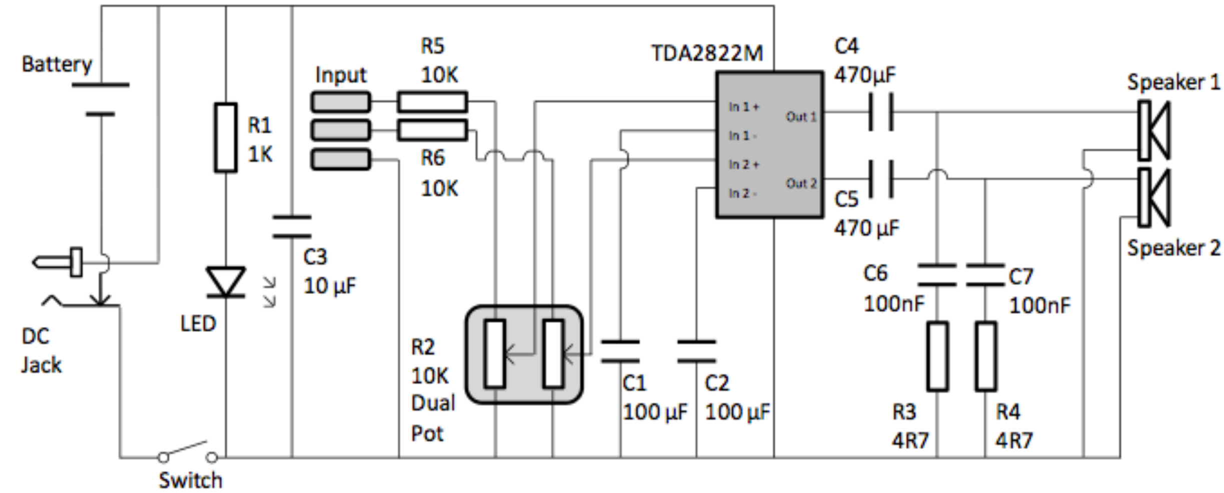
Fault finding flow chart



Fault finding flow chart continued

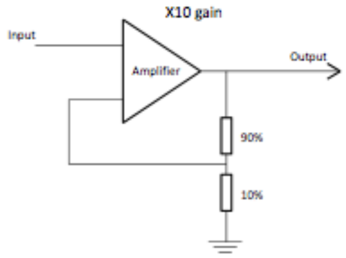


How the Amplifier Works



At the centre of the circuit is an audio amplifier Integrated Circuit or IC. Inside the IC are lots of transistors, which are connected together to allow the small input signal to be amplified into a more powerful output that can drive a speaker.

All amplifiers need to use feedback to ensure the amount of gain stays the same. This allows the output to be an exact copy of the input just bigger. The gain is the number of times bigger the output is compared to the input, so if an amplifier has a gain of 10 and there is 1 volt on the input there will be 10 volts on the output. An operational amplifier has two inputs, these are called the inverting (-) and non-inverting (+) inputs. The output of the operational amplifier is the voltage on the non-inverting input less the voltage on the inverting input multiplied by the amplifiers gain. In theory an operational amplifier has unlimited gain so if the non-inverting input is a fraction higher than the inverting input (there is more + than -) the output will go up to the supply voltage. Change the inputs around and the output will go to zero volts. In this format the operational amplifier is acting as a comparator, it compares the two inputs and changes the output accordingly.



With an infinite gain the amplifier is no good to amplify audio, which is where the feedback comes in. By making one of the inputs a percentage of the output the gain can be fixed, which allows the output to be a copy of the input but bigger. Now when the two inputs are compared and the output is adjusted, instead of it going up or down until it reaches 0 volts or V+, it stops at the point when the two inputs match and the output is at the required voltage.

Looking at the circuit diagram for the audio amplifier it's not obvious where the feedback is, this is because it is inside the IC. The TDA2822M chip has fixed the gain so the output is about 90 times bigger than the input. To make the gain useful in our application there is a potential divider on each channel that is fed into the IC (R2+R5 and R2+R6). Each of these reduces the input signal to a percentage of the original signal. As R2 is a variable potentiometer, it can be used to vary this percentage, which in turn varies the output volume. C3 is connected across the supply to make sure it remains stable. The other capacitors have a filtering role, either to cut out high frequency noise or get the best out of the speaker.

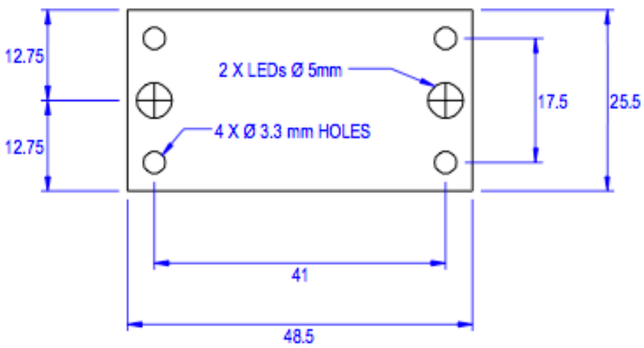
A power switch is inserted in the ground (0V) power line, which is used to turn the amplifier on and off. There is also a power LED that lights up when the power is switch is on. R1 is used to limit the current flowing into this LED, which stops it drawing too much power, which over time will damage the LED. Power is supplied to the circuit by either a battery, or from a DC power socket. If a power supply is plugged into the DC power socket the battery is automatically disconnected.

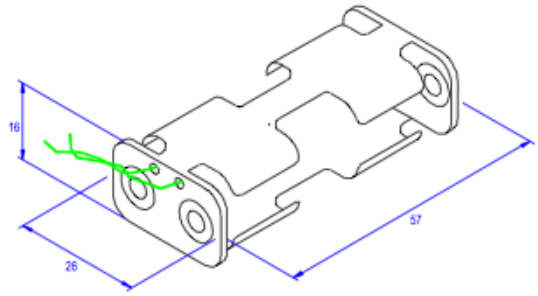
Designing the Enclosure

When you design the enclosure, you will need to consider:

- The size of the PCB (below left)
- Where the LEDs are mounted & how big they are
- Where the batteries will be housed (below right)

These technical drawings of the bike light PCB and battery holder should help you plan this.

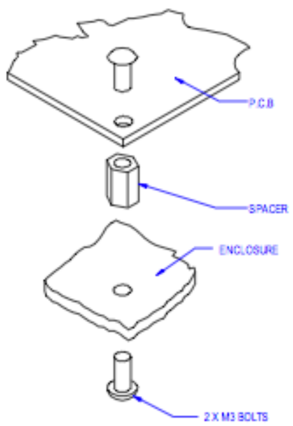




All dimensions in mm
x4 holes 3.3 mm diameter
x2 LEDs 5mm diameter

Enclosure Prototype

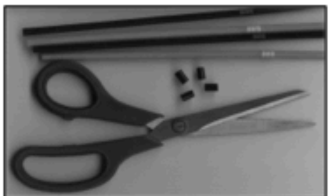
Using card board, foam or anything else that is suitable, make proto type of your enclosure design. This will give you the chance of changing any aspects of the design that do not work as well as expected.



Mounting the PCB to the enclosure

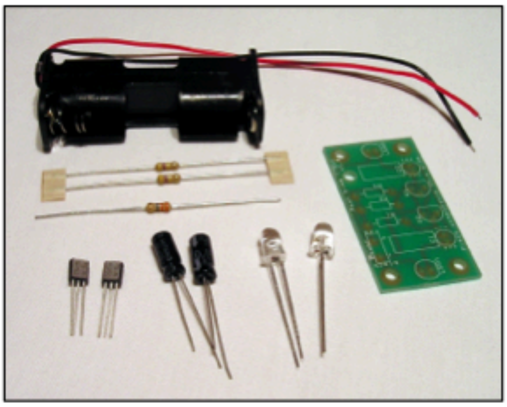
The drawing to the left shows how a hex spacer can be used with two bolts to fix the PCB to the enclosure.

Your PCB has four mounting holes designed to take M3 bolts



Top tip
If your design of enclosure needs the LEDs to be higher, you can get LED spacers. Or why not make your own by simply cutting a drinking straw to size.

Build Instructions

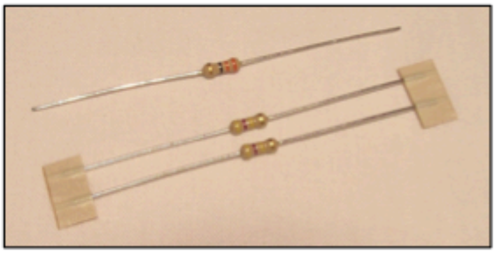


The complete set of bike light parts are shown in the picture on the left.

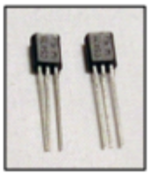
Before you put any components in the board or pick up the soldering iron, just take a look at the Printed Circuit Board (PCB). The components go in the side with the writing on and the solder goes on the side with the tracks and silver pads.

You will find it easiest to start with the small components and work up to the taller larger ones. If you've not soldered before get your soldering checked after you have done the first few joints.

Step 1
Start with the three resistors (shown right):
R1 and R2 are 470KΩ (yellow, purple, yellow coloured bands)
R3 is a 33Ω (orange, orange, black coloured bands)



The text on the PCB shows where R1, R2, etc go. Make sure that you put the resistors in the right place.

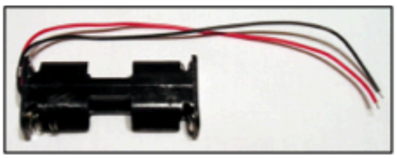


Step 2
Place the two transistors (shown left) in to the board where it is labeled Q1 and Q2. Make sure the device is the correct way around. The shape of the device should match the outline on the PCB.

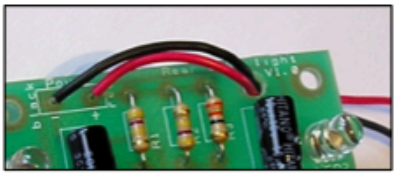
Step 3
Place the two capacitors (shown right) in to the board where it is labeled C1 and C2. Make sure the device is the correct way around. The capacitors have a '-' sign marked on them which should match the same sign on the PCB. Once the legs have been pushed through the board the capacitor should be folded flat against the PCB before it is soldered into place.

Step 4
Place the two Light Emitting Diodes (LEDs - shown left) in to LED1 and LED2. It does not matter which goes where, but the light won't work if they don't go in the right way around. If you look carefully one side of the LED has a flat edge, which must line up with the flat edge on the lines on the PCB. You may want to solder them in at a specific height depending upon how you have designed your enclosure (if you are making one). Once you are happy solder them into place.

Step 5
Finally you must attach the battery holder (shown right). Start by feeding the leads through the strain relief hole near R3. The wire should be fed in from the rear of the board (see below & right).



The red lead should be soldered to the '+' terminal (also marked with the text 'red') and the black lead should be soldered to the '-' terminal (also marked with the text 'black').



Adding an on / off switch

To add a power switch, don't solder both ends of the battery cage directly into the board, instead:

- Solder one end of the battery cage to the PCB, either black to '-' or red to '+'.
- Solder the other end of the battery cage to the on / off switch.
- Using a piece of wire, solder the remaining terminal on the on / off switch to the remaining power connection on the PCB.

Checking Your Bike PCB

Check the following before you insert the batteries:

Check the bottom of the board to ensure that:

- All holes (except the 4 large (3 mm) holes in the corners) are filled with the lead of a component.
- All these leads are soldered.
- Pins next to each other are not soldered together.

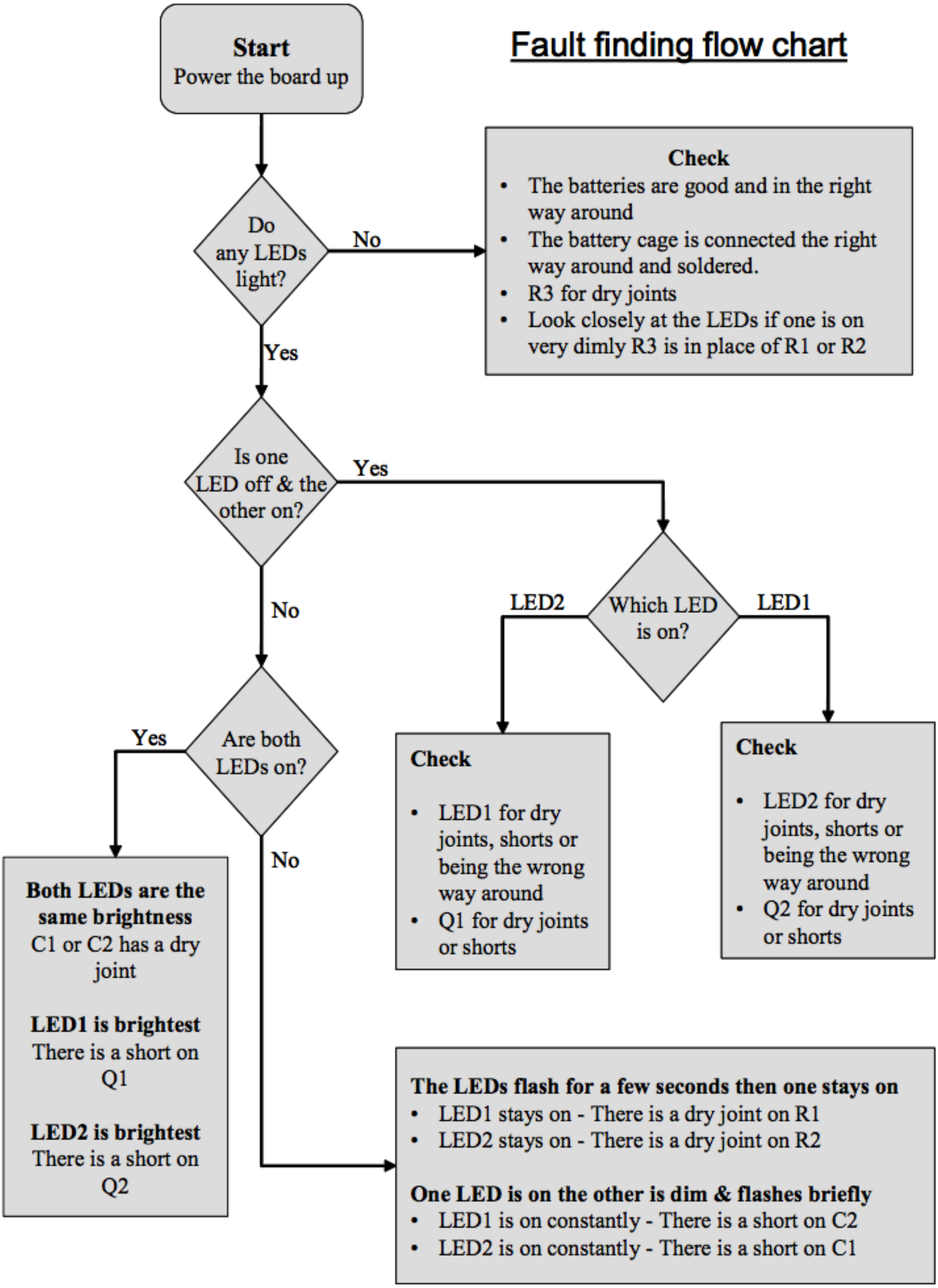
Check the top of the board to ensure that:

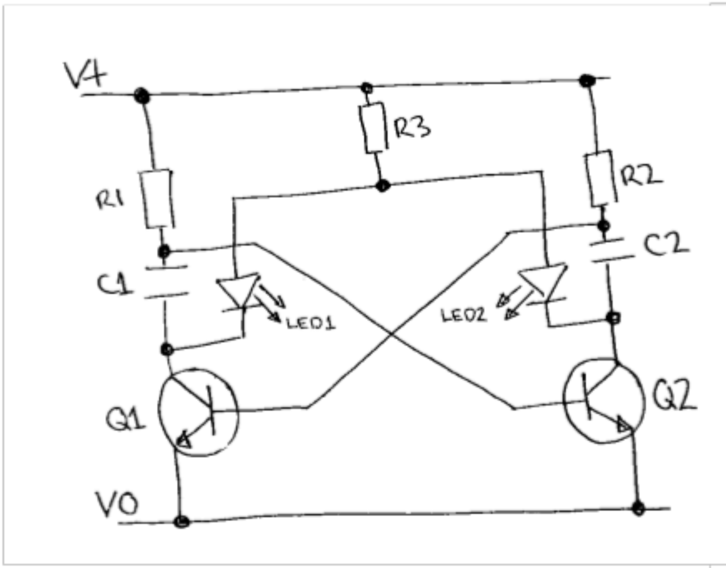
- The shape of the transistors match the outline on the PCB.
- The flat edge of each of the LEDs matches the outline on the PCB.
- The '-' on the capacitors match the same marks on the PCB.
- The colour bands on R3 are orange, orange and black.
- The battery cage red and black wires match the red & black text on the PCB.

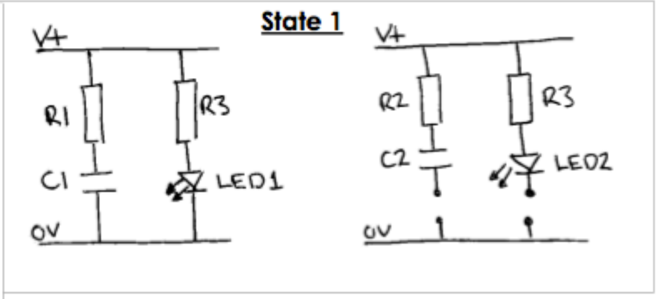
Power Up

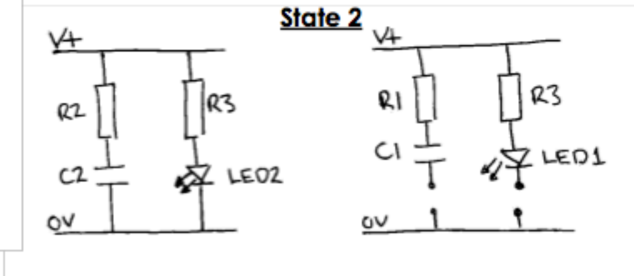
On inserting the batteries the LED's should start flashing in an alternating pattern. If this does not happen use the fault finding sheet to find the fault.

Fault finding flow chart









The circuit has two states which it alternates between. In each of the states one of the LEDs is on while the other is off.

State 1 (see picture above right):

- Q1 is turned on which connects LED1 and C1 to 0V. This turns LED1 on and C1 starts to charge through the resistor R1 causing the voltage across it to increase (it starts at less than 0.7V). The voltage at the base of Q2 starts to rise as C1 charges as they are both connected to each other.
- As C1 has less than 0.7V across it Q2 is turned off. This means LED2 is not connected to 0V and is therefore turned off. C2 (which has more than 0.7V across it) is gradually discharging into the base of Q1.

This continues until the C1 has sufficient charge to produce a voltage >0.7V on the base of Q2, which causes it to turn on.

State 2 (see picture above right):

- Q2 is now turned on which connects LED2 and C2 to 0V. This turns LED2 on. This connection of C2 to 0V causes the voltage across it to drop below 0.7V turning off Q1. Now C2 starts to charge through the resistor R2 causing the voltage across it to increase. The voltage at the base of Q1 starts to rise as C2 charges as they are both connected to each other.
- As C2 has less than 0.7V across it Q1 is turned off. This means LED1 is not connected to 0V and is therefore turned off. C1 (which has more than 0.7V across it) is gradually discharging into the base of Q2.

The right hand side of the circuit is in the same state that the left hand side started in Stage 1, but with C2 charging instead of C1. When the charge gets high enough the circuit flips back to Stage 1.

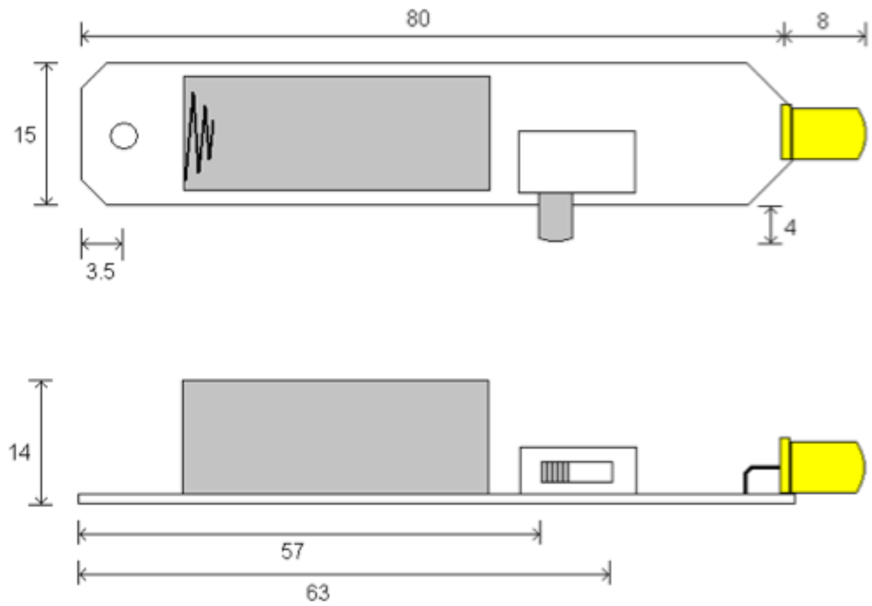
R3 is needed to limit the amount of current flowing through the LED. The transistors aren't turned fully on so also contribute to the limiting of current flowing through the LED. This means the current limit resistor is smaller than it would otherwise be.

Designing the Enclosure

When you design the enclosure, you will need to consider:

- The size of the PCB.
- Where the LED is mounted.
- Where the on off switch is mounted.
- There is a 3mm hole at the back of the torch to secure the PCB to your enclosure, depending on your design you may be able to hold the board in place with a key ring.

This technical drawing of the built torch PCB should help you design your enclosure.

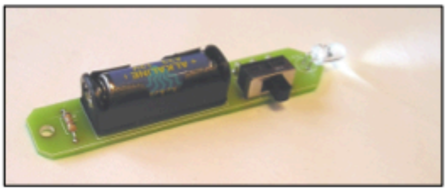


All dimensions in mm.
x1 mounting hole 3.3 mm diameter.
The diameter of the LED is 5 mm.

Enclosure Prototype

Using card board, foam or anything else that is suitable, make proto type of your enclosure design. This will give you the chance of changing any aspects of the design that do not work as well as expected.

Build Instructions



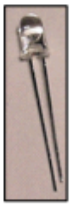
The finished torch is shown left.

Before you put any components in the board or pick up the soldering iron, just take a look at the Printed Circuit Board (PCB). The components go in the side with the writing on and the solder goes on the side with the tracks and silver pads.

You will find it easiest to start with the small components and work up to the taller larger ones. If you've not soldered before get your soldering checked after you have done the first few joints.

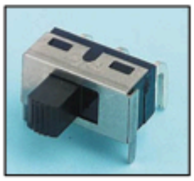
Step 1
Start with the resistor (shown right):
R1 is a 680Ω (it will be marked with blue, grey, brown coloured bands)

The text on the PCB shows where R1 should go. It doesn't matter which way around the resistor goes into the board.



Step 2
Place the Light Emitting Diode (LED - shown left) in to LED1. The light won't work if it doesn't go in the right way around. If you look carefully one side of the LED has a flat edge, which must line up with the flat edge on the outline on the PCB. **You will need to put a 90º bend into the LED legs**, just make sure you bend it so the flat edge on the LED is next to the flat edge on the board. Once you are happy solder into place.

Step 3
Solder the PCB mount right angled on / off switch (shown right) in to SW1. The row of three pins that exit the back of the switch must be soldered, but it won't matter too much if you can't solder the other two pins.



Step 4
Finally the battery holder needs to be soldered into the board where it is marked CONN1. The battery holder outline on your PCB shows the spring. Make sure when you put the holder into the board that the spring on the holder lines up with the spring marked on the board.

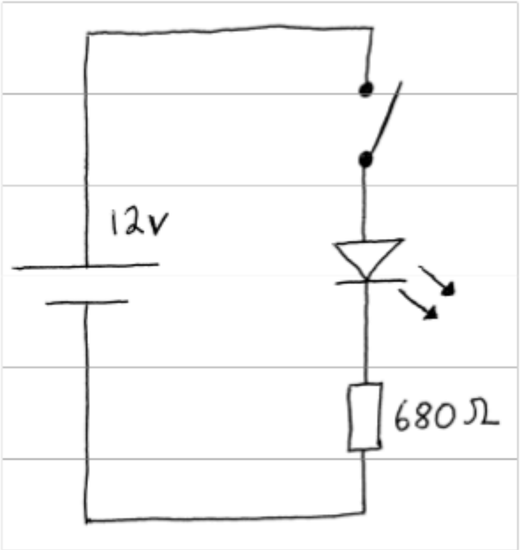
Checking Your Torch

Check the following before you insert the battery:

Check the bottom of the board to ensure that:
- All holes (except the large mounting hole) are filled with the lead of a component.
- All these leads are soldered.
- Pins next to each other are not soldered together.

Check the top of the board to ensure that:
- The flat edge on the LED matches the outline on the PCB.
- The spring on the battery holder is next to the resistor.

How the Torch Works



The circuit diagram for the torch is shown on the left. It is a very simple circuit, powered by a 12 volt battery.

The LED would be damaged if the current through it was not limited. A 680Ω resistor has been selected to limit the current through the LED. This allows approximately 10mA to flow through the LED so that it is at a good brightness.

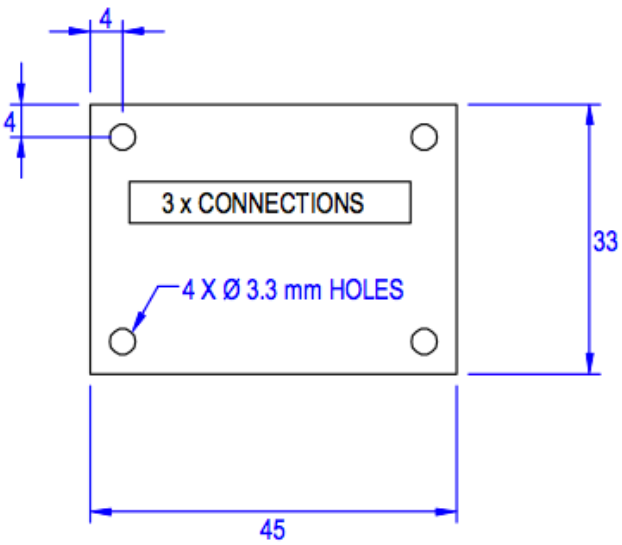
Finally the on off switch allows the circuit to be opened when the LED will be off or completed when the LED will be on.

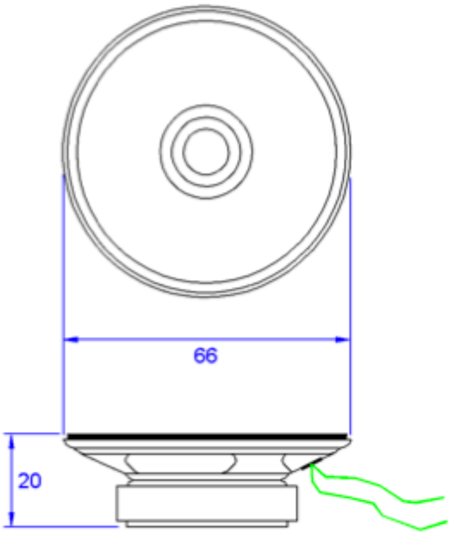
Designing the Enclosure

When you design the enclosure, you will need to consider:

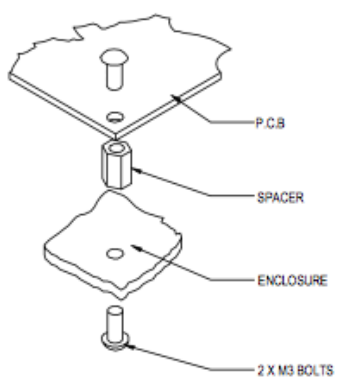
- The size of the PCB (below left, height including components = 15mm)
- How big the batteries are.
- How to mount the speaker (below right).
- How to allow the audio cable out of the box.
- Are you making the amplifier for a particular MP3 player, if so should the MP3 player go in the box?

These technical drawings of the amplifier PCB and speaker should help you plan this.





All dimensions in mm
x4 holes 3.3 mm diameter



Mounting the PCB to the enclosure

The drawing to the left shows how a hex spacer can be used with two bolts to fix the PCB to the enclosure.

Your PCB has four mounting holes designed to take M3 bolts

Enclosure Prototype

Using card board, foam or anything else that is suitable, make proto type of your enclosure design. This will give you the chance of changing any aspects of the design that do not work as well as expected.

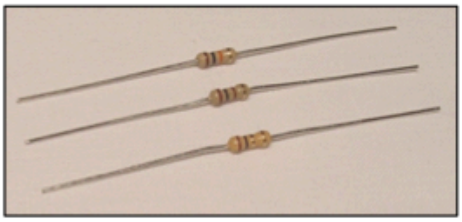
Build Instructions

Before you put any components in the board or pick up the soldering iron, just take a look at the Printed Circuit Board (PCB). The components go in the side with the writing on and the solder goes on the side with the tracks and silver pads.

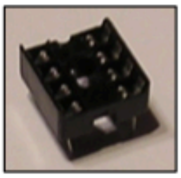
You will find it easiest to start with the small components and work up to the taller larger ones. If you've not soldered before get your soldering checked after you have done the first few joints.

Step 1

Start with the three resistors (shown right):
R1 is a 10K (brown, black, orange coloured bands)
R2 is a 100Ω (brown, black, brown coloured bands)
R3 is a 1Ω (brown, black, gold coloured bands)

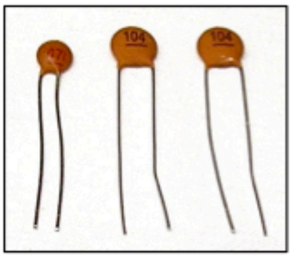


The text on the PCB shows where R1, R2 & R3 go. Make sure that you put the resistors in the right place.



Step 2

Solder the Integrated Circuit (IC) holder in to IC1. When putting this into the board, be sure to get it the right way around. The notch on the IC holder should line up with the notch on the lines marked on the PCB.

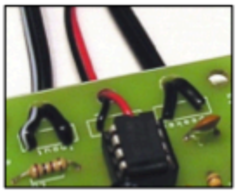


Step 3

There are three ceramic disc capacitors, the smaller one is a 470pF capacitor and is printed with the number 471. This should be soldered into C1. The other two capacitors are 100nF capacitors and are printed with 104. These need soldering in to C2 and C3.

Step 4

The other three capacitors are electrolytic capacitors, the two smaller capacitors are marked 100µF. Place these two capacitors in to the board where it is labeled C4 and C5. Make sure the device is the correct way around. The capacitors have a '-' sign marked on them which should match the same sign on the PCB. The bigger capacitor is a 220µF, which should be soldered in to C6.

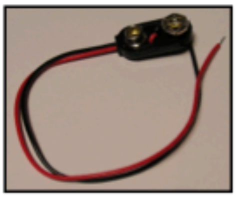


Step 5

The three connections to your amplifier PCB need to go through the strain relief holes as shown in the picture on the right.

Start with the connection labeled Speaker. The kit is supplied with ½ a meter of twin cable with a 3.5mm Jack connector on one end. This cable will be used to connect both the speaker and the MP3 player. You will need to cut a length from the end that does not have the jack connector on which will be used to connect the speaker. Make sure you leave enough cable so that you have a long enough lead to connect your MP3 player!

Take the piece of wire that you have cut off and strip the ends of the wire. Connect one end to the two terminals on the speaker and the other end to the board marked speaker. It does not matter which way around these connections go.



The middle connection is for the power. The PP3 battery clip (shown left) should be attached to the power connection. Connect the red wire to '+' and the black wire to '-'.

Build Instructions (continued)

The final connection is the audio input. Strip the insulation off the other end of the remaining cable that has the jack plug on. Run some solder in to the wire and trim the wire so only 2 or 3 mm of bare wire is left. Solder these wires into the board where it is labeled 'input'. It doesn't matter which of the pair of wires goes each of the two pads.

Step 6
The IC can now be put into the holder ensuring the notch on the chip lines up with the notch on the holder. Your amplifier is ready for use. You can use the volume control on your MP3 player to control how loud the amplifier is. Just make sure it's mid volume when you test the amplifier.

Adding an on / off switch

If you wish to add a power switch, don't solder both ends of the battery clip directly into the board, instead:

- Solder one end of the battery clip to the PCB, either black to '-' or red to '+'.
- Solder the other end of the battery clip to the on / off switch.
- Using a piece of wire, solder the remaining terminal on the on / off switch to the remaining power connection on the PCB.

Checking Your Amplifier PCB

Carefully check the following before you insert the batteries:
Audio equipment may become damaged if connected to an incorrectly built amplifier.

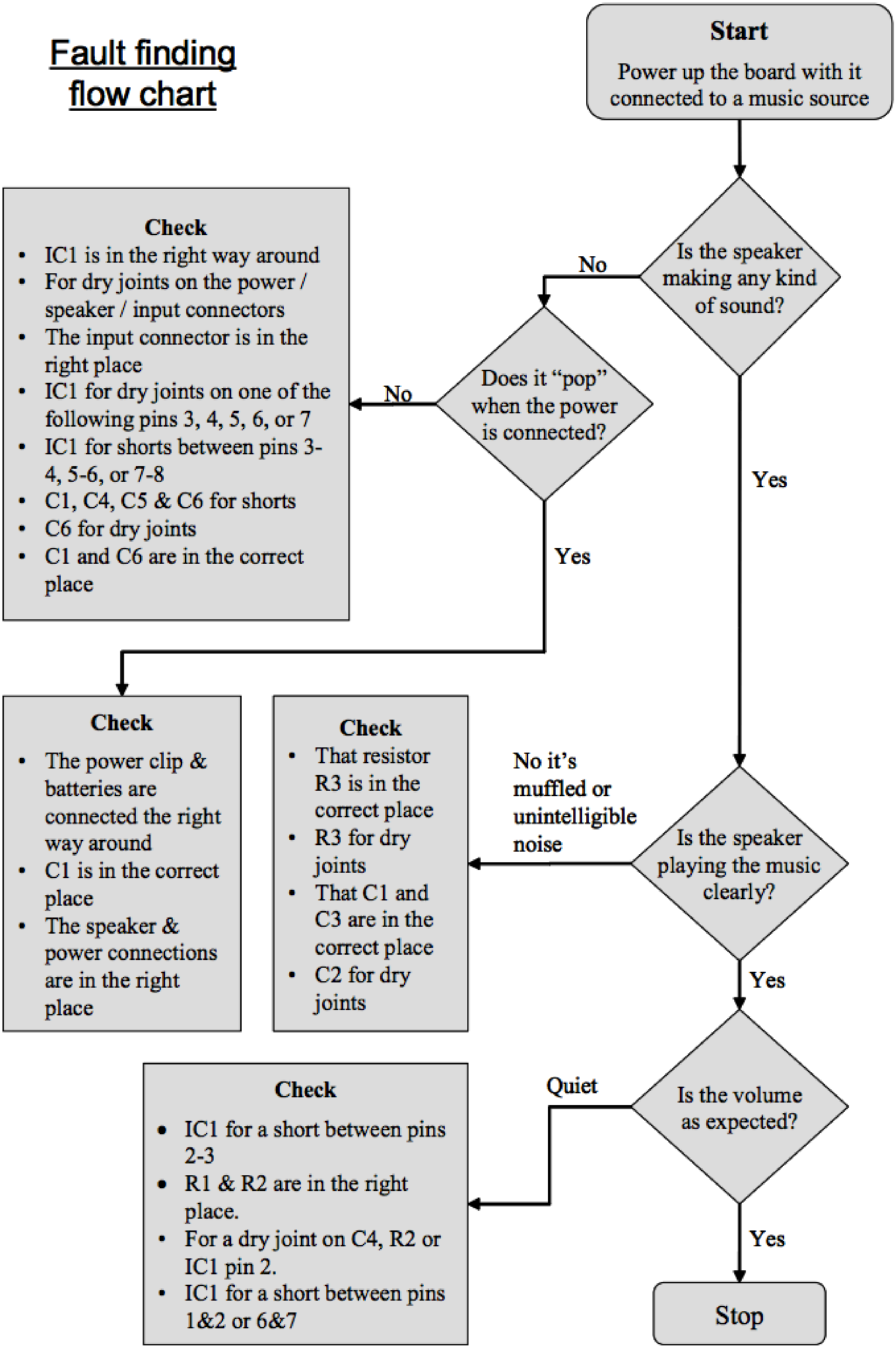
Check the bottom of the board to ensure that:

- All holes (except the 4 large (3 mm) holes in the corners) are filled with the lead of a component.
- All these leads are soldered.
- Pins next to each other are not soldered together.

Check the top of the board to ensure that:

- The three wires are connected to the right place.
- The '-' on the capacitors match the same marks on the PCB.
- The colour bands on R1 are brown, black, orange & R2 are brown, black, brown.
- The battery clip red and black wires match the red & black text on the PCB.
- The notch on the IC is next to the power connection.

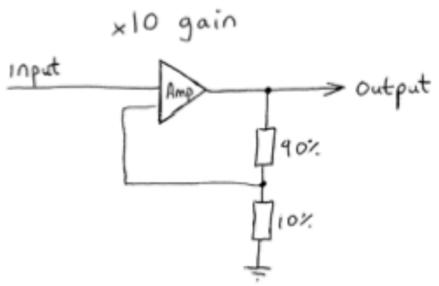
Fault finding flow chart



How the Amplifier Works

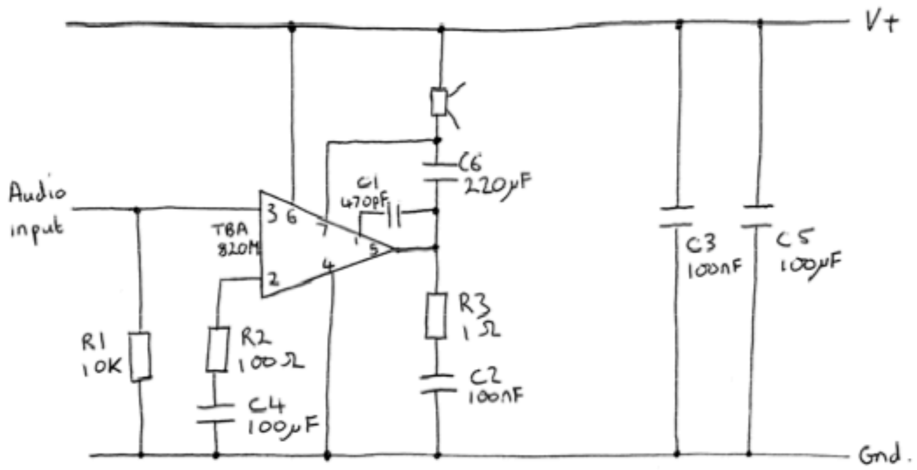
At the centre of the circuit is an audio amplifier Integrated Circuit or IC. Inside the IC are lots of transistors, which are connected together to allow the small input signal to be amplified into a more powerful output that can drive a speaker.

All amplifiers need to use feedback to ensure the amount of gain stays the same. This allows the output to be an exact copy of the input just bigger. The gain is the number of times bigger the output is compared to the input, so if an amplifier has a gain of 10 and there is 1 volt on the input there will be 10 volts on the output. Before looking at how the feedback works, we first need to understand how a standard amplifier works. An operational amplifier has two inputs these are called the inverting (-) and non-inverting (+) inputs. The output of the operational amplifier is the voltage on the non-inverting input less the voltage on the inverting input multiplied by the amplifiers gain. In theory an operational amplifier has unlimited gain so if the non-inverting input is a fraction higher than the inverting input (there is more + than -) the output will go up to the supply voltage. Change the inputs around and the output will go to zero volts. In this format the operational amplifier is acting as a comparator, it compares the two inputs and changes the output accordingly.



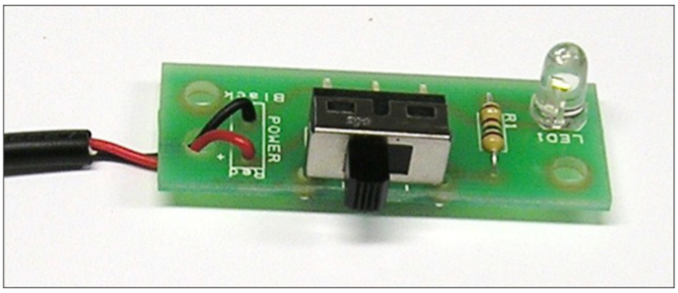
With an infinite gain the amplifier is no good to amplify audio, which is where the feedback comes in. By making one of the input a percentage of the output the gain can be fixed, which allows the output to be a copy of the input but bigger. Now when the two inputs are compared and the output is adjusted, instead of it going up or down until it reaches 0 volts or V+, it stops at the point when the two inputs match and the output is at the required voltage.

Looking at the circuit diagram for the audio amplifier it's not obvious where the feedback is, this is because inside the IC is a 6K resistor between the output (pin 5) and the gain setting input (pin 2). The internal 6K resistor and the 100Ω resistor (R2) on the gain setting pin make up a potential divider that feeds back approximately a sixtieth of the output. This fixes the gain so the output is about 60 times bigger than the input.



The rest of the components are needed as follows:
C3 & C5 are connected across the supply to make sure it remains stable.
The other capacitors have a filtering role, either to cut out high frequency noise or get the best out of the speaker.

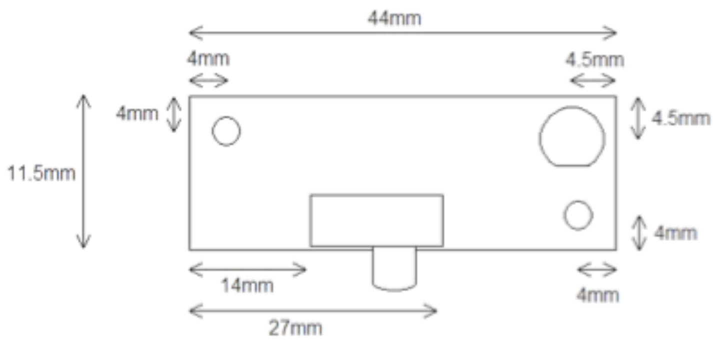
USB Lamp



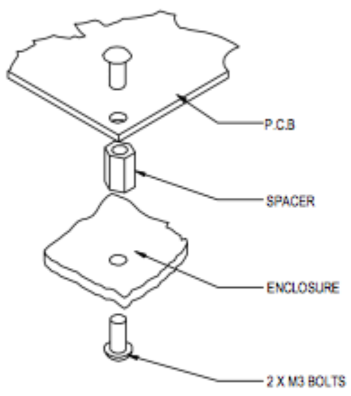
When you design the enclosure, you will need to consider:

- The size of the PCB.
- Where the LED is mounted (shown in the top right corner of the PCB).
- Where the on / off switch is mounted.
- There are two 3.3mm holes in the corners of the PCB to secure the PCB in the enclosure.

This technical drawing of the built USB lamp PCB should help you design your enclosure.



The 2 mounting holes are 3.3mm diameter.
The diameter of the LED is 5 mm and the total height of the unit approximately 11mm.



Mounting the PCB to the enclosure

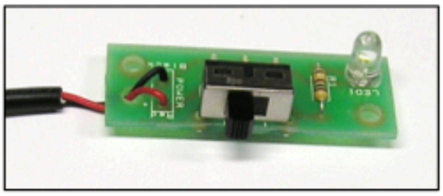
The drawing to the left shows how a hex spacer can be used with two bolts to fix the PCB to the enclosure.

Your PCB has two mounting holes designed to take M3 bolts

Enclosure Prototype

Using card board, foam or anything else that is suitable, make a proto-type of your enclosure design. This will give you the chance of changing any aspects of the design that do not work as well as expected.

Build Instructions

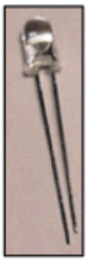


The finished USB lamp PCB is shown left. Before you put any components in the board or pick up the soldering iron, just take a look at the Printed Circuit Board (PCB). The components go in the side with the writing on and the solder goes on the side with the tracks and silver pads. You will find it easiest to start with the small components and work up to the taller larger ones. If you've not soldered before get your soldering checked after you have done the first few joints.

Step 1
Start with the resistor R1 (example shown right). The text on the PCB shows where R1 should go. It doesn't matter which way around the resistor goes into the board.

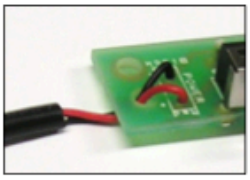
If you are building the white light version of the kit:
R1 is a 150Ω resistor (it will be marked with brown, green, brown coloured bands).

Or if you are building the colour changing version of the kit:
R1 is a 0Ω (it will be marked with a single black coloured band).



Step 2
Place the Light Emitting Diode (LED - shown left) in to LED1. The LED won't work if it doesn't go in the right way around. If you look carefully one side of the LED has a flat edge, which must line up with the flat edge on the outline on the PCB. You can mount this facing up from the board, or if you prefer you can mount it at 90° angle to the PCB. To do this you will need to put a 90° bend into the LED legs, just make sure you bend it so the flat edge on the LED is next to the flat edge on the board. Once you are happy solder it into place.

Step 3
Solder the PCB mount right angled on / off switch (shown right) into SW1. The row of three pins that exit the back of the switch must be soldered, but it won't matter too much if you can't solder the other two pins.



Step 4
Finally the USB power lead needs to be connected. Feed the red and black wire of the lead through the strain relief hole (see left). The red wire of the USB power cable is soldered to the power connector labeled 'Red' and the black wire of the USB power cable is soldered to the power connector labeled 'Black'.

Checking Your Circuit

Check the following before you plug your lamp into a USB port.

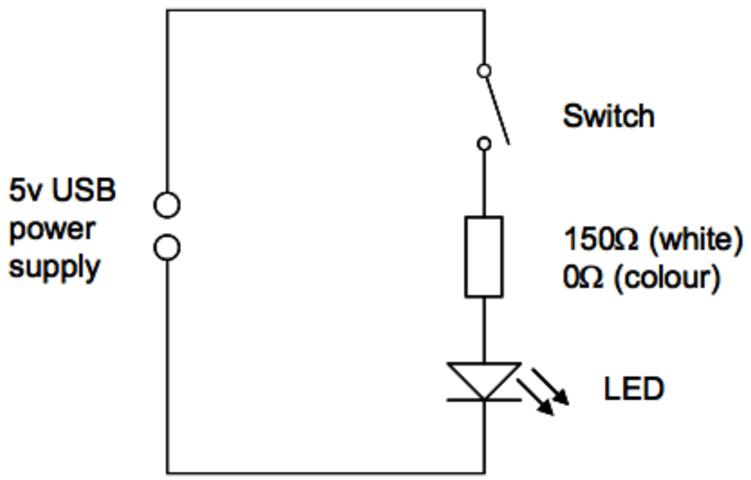
Check the bottom of the board to ensure that:

- All holes (except the two large mounting holes) are filled with the lead of a component.
- All these leads are soldered.
- Pins next to each other are not soldered together.

Check the top of the board to ensure that:

- The flat edge on the LED matches the outline on the PCB.
- The red wire on the USB power cable is connected to the power connector labeled 'Red' and the black wire on the USB power cable is connected to the power connector labeled 'Black'.

How the USB Lamp Works



The circuit diagram for the USB lamp is shown on the left. It is a very simple circuit. The 5V that powers the circuit is supplied from the USB connector.

LED's can be damaged if the current through them is not limited.

If you are using a white LED a 150Ω resistor has been selected to limit the current through the LED. This allows 10mA to flow through the LED so that it is at a good brightness.

If you are using a colour changing LED a 0Ω resistor is used instead. This is because the current limit resistor is built into the LED itself therefore we simply want to connect this LED directly to the 5V supply.

Finally the on off switch allows the circuit to be opened when the LED will be off or closed when the LED will be on.

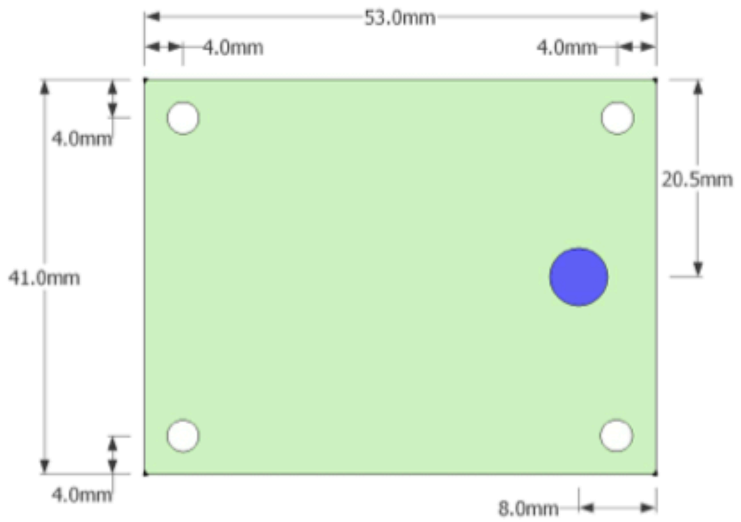
Designing the Enclosure

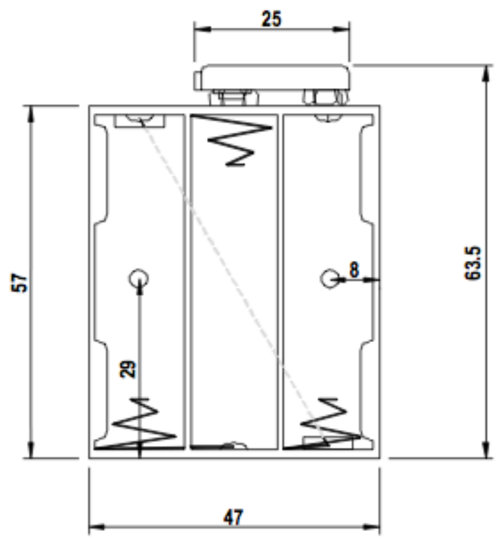
When you design the enclosure, you will need to consider:

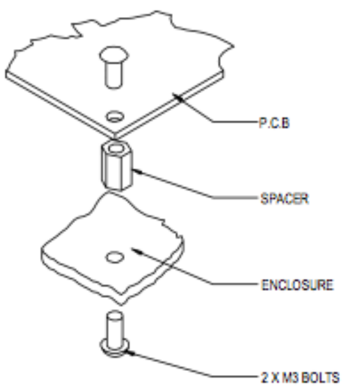
- The size of the PCB (below left).
- How big the batteries are (below right).

These technical drawings of the PCB and battery holder should help you plan this.

All dimensions in mm.
Four PCB mounting holes are 3.3 mm in diameter.







Mounting the PCB to the enclosure

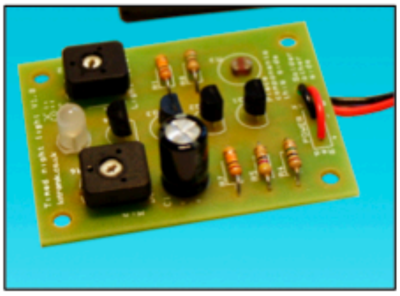
The drawing to the left shows how a hex spacer can be used with two bolts to fix the PCB to the enclosure.

Your PCB has four mounting holes designed to take M3 bolts

Enclosure Prototype

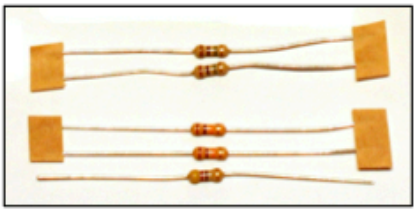
Using card board, foam or anything else that is suitable, make proto type of your enclosure design. This will give you the chance of changing any aspects of the design that do not work as well as expected.

Build Instructions



Before you put any components in the board or pick up the soldering iron, just take a look at the Printed Circuit Board (PCB). The components go in the side with the writing on and the solder goes on the side with the tracks and silver pads.

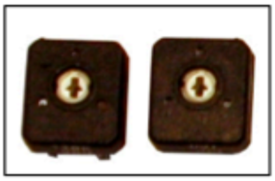
You will find it easiest to start with the small components and work up to the taller larger ones. If you've not soldered before get your soldering checked after you have done the first few joints.



Step 1
Start with the resistors (shown left). The board is labeled with R1, R2, R3 and so on, the resistors must go in the right place as follows:

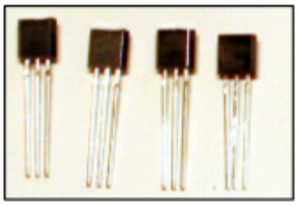
R1 & R7 are 10K resistors (brown, black, orange colour bands)
R4 & R8 are 1M resistors (brown, black, green colour bands)
R5 is a 47Ω resistor (yellow, purple, black colour bands)

Step 2
The LDR (shown right) needs to be soldered in to the board where it is marked R3.



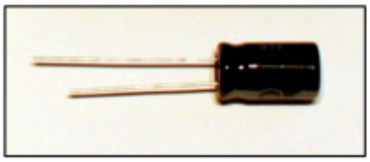
Step 3
Solder the variable resistors (shown left) into R2 & R6. They will only fit in the holes in the board when they are the correct way around. The two resistors are different values, R2 is a 220K and R6 is 10M, make sure you put each in the right place.

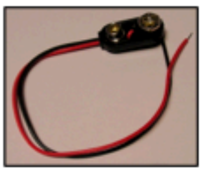
Step 4
The four FETs (shown right) should be placed into Q1 to Q4. All four are the same type, but it is important that they are inserted in the correct orientation. Ensure the shape of the device matches the outline printed on the PCB. Once you are happy solder the devices into place.



Step 5
Solder the Light Emitting Diode (LED - shown left) in to LED1. The LED won't work if it doesn't go in the right way around. If you look carefully one side of the LED has a flat edge, which must line up with the flat edge on the lines on the PCB.

Step 6
Solder the capacitor (shown right) into the board where it is marked C1. It is important that the '-' on the capacitor matches the '----' markings on the PCB.





Step 7
Now you must attach the battery clip (shown left). It needs to be connected to the terminals marked 'Power'. The red lead should be soldered to the '+' terminal also marked 'red' and the black lead should be soldered to the '-' terminal also marked 'black'.

Checking Your Circuit

Check the following before you connect power to the board:

Check the bottom of the board to ensure that:

- All the leads are soldered.
- Pins next to each other are not soldered together.

Check the top of the board to ensure that:

- The body of the four FETs matches the outline on the PCB.
- The flat edge on the LED lines matches the outline on the PCB.
- The red wire on the power clip goes to the connection marked 'red' and the black wire to the connection marked 'black'.
- R1 & R7 are 10K resistors (brown, black, orange colour bands).
- R5 is a 47Ω resistor (yellow, purple, black colour bands).
- R2 has 220K printed on the side.

Testing the PCB

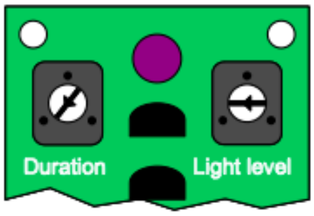
Set the duration to min – fully anti-clockwise.
Set the light level so it points at the other trimmer.

Power the board up and cover the LDR, the LED should turn on for about five seconds. Uncover the LDR, then cover it again, the LED should turn on again for about five seconds. The duration and light level can be adjusted.

If the circuit doesn't function as expected use the fault finding flow chart to locate the problem.

Fault finding flow chart

Start
Set the two trimmers as shown in the diagram.
Then power the board up



Is the LED off?

No →

The LED is constantly on
All 3 pins on Q2 are shorted.

The LED is on dim & goes brighter when the LDR is covered
Check Q2 & Q3 for shorts.

Yes ↓

Cover the LDR (and keep it covered)

Does the LED light?

No →

Check

- The batteries are good and in the right way around.
- Check the power clip is connected the right way around and soldered correctly.
- The LED is the right way around, for a short or dry joint.
- Q1 is the right way around, for shorts or dry joints.
- Q2 for shorts or dry joints.
- Q3 is the right way around or for shorts.
- Q4 for shorts or dry joints.
- R3 for a short or dry joint.
- R1, R2, R5 & R8 for dry joints.
- R5 is a 47Ω resistor (yellow, purple, black).
- R1 is a 10K resistor (brown, black, orange).
- C1 for a short.

The LED flashes briefly ←

Check

- Q2 for dry joints.
- R5 is in the right place, 47Ω resistor (yellow, purple, black).
- C1 for dry joints.

Yes *Even if dimly* ↓

After about 5 seconds does it go off?

No →

Check

- Q2 is the right way around.
- Q3 for dry joints.
- Q4 is the right way around and for shorts.
- R6 for dry joints.
- R7 for dry joints.
- R7 is a 10K resistor (brown, black, orange).

Yes ↓

Can you adjust the light level & duration OK?

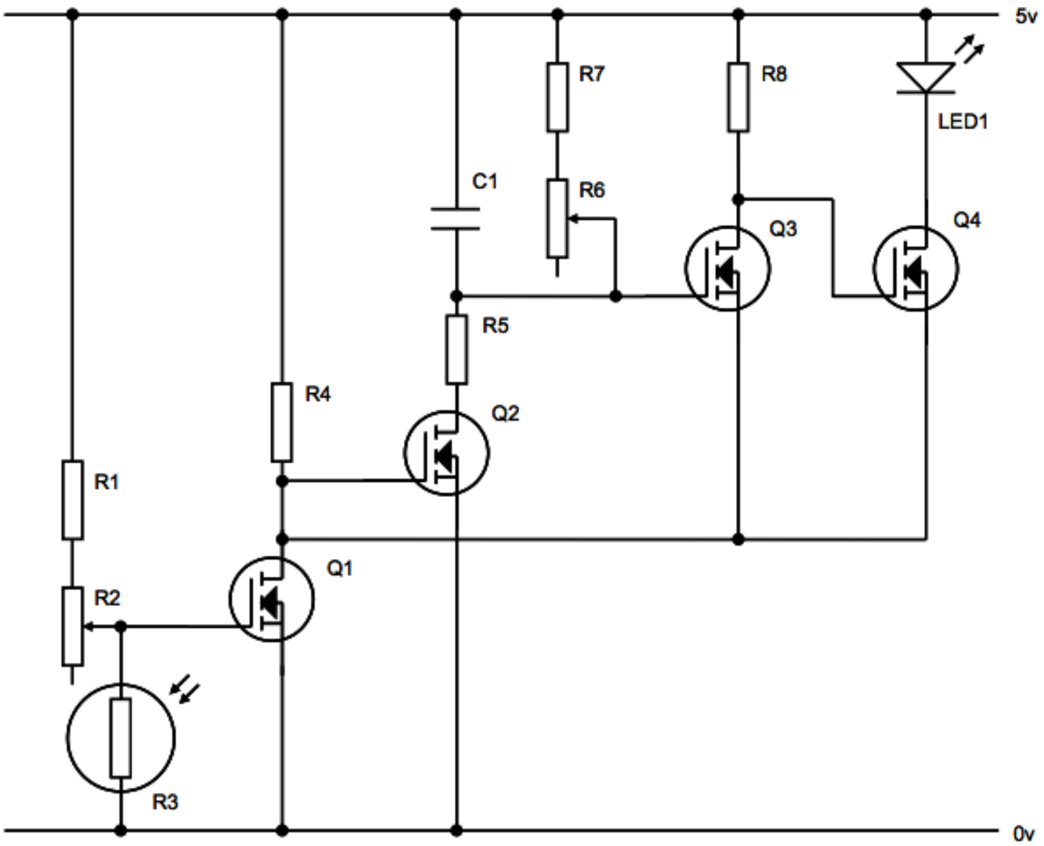
Yes ← Stop

No →

If the setting of the light level is close to min & the maximum duration is about a minute

The two trimmer pots are in the wrong place.
(R2 should be the 220K)

How the Timed Night Light Circuit Works



When the Light Dependant Resistor or LDR for short (R3) is in day light the resistance will be low. R1, R2 & R3 are connected together to form a potential divider, where R2 sets the amount of light needed to switch the FET. With the LDR having a low resistance, the voltage on the gate of the FET Q1 will be low and the FET will be off. In this case the two FETs Q3 and Q4, which drive the LED, don't have a 0V connection and as a result the LED won't light. Also whilst Q1 is off the resistor R4 pulls the gate of FET Q2 into a high state, and as a result current flows through the drain source of Q2 and charges capacitor C1 through R5.

When the LDR (R3) is dark the resistance is much higher, and consequently the voltage on the gate of the FET Q1 is high and the FET is turned on. In this case the FETs Q3 and Q4 now have a 0V connection and the LED can operate. Initially the capacitor C1 is charged and as a result has 5V across it. This means that the gate of FET Q3 is low, and therefore the FET is turned off. As a result the gate of Q4 is held in a high state by resistor R8, and the LED is turned on. Over time the Capacitor (C1) is discharged through R6 & R7 and gradually the voltage across it drops. As the capacitor voltage reduces, the voltage on the gate of Q3 starts to rise. After a period of time there will be sufficient voltage on the gate of FET Q3 and the FET will turn on. When this happens the gate of FET Q4 will be pulled low and the FET Q4 will switch off, as will the LED.

Resistors R1 & R7 are present so that the circuit can't be damaged if the trimmer potentiometers are set to zero.