Sensor Alarm

Project: **Sensor Alarm**

Author: A.J.Booker (www.ajbooker.co.uk)

Description: This worksheet shows you how to construct a simple alarm in PCB Wizard.

When viewed in PCB Wizard this worksheet can be edited, copied and switched into different views by using the Style toolbar on the left. You can also produce your own PCB masks directly from artwork in this document.

This worksheet assumes a basic working knowledge of PCB Wizard. Those new to the software should refer to the User Guide (press **F1** now if in PCB Wizard) and Tutorials for assistance.

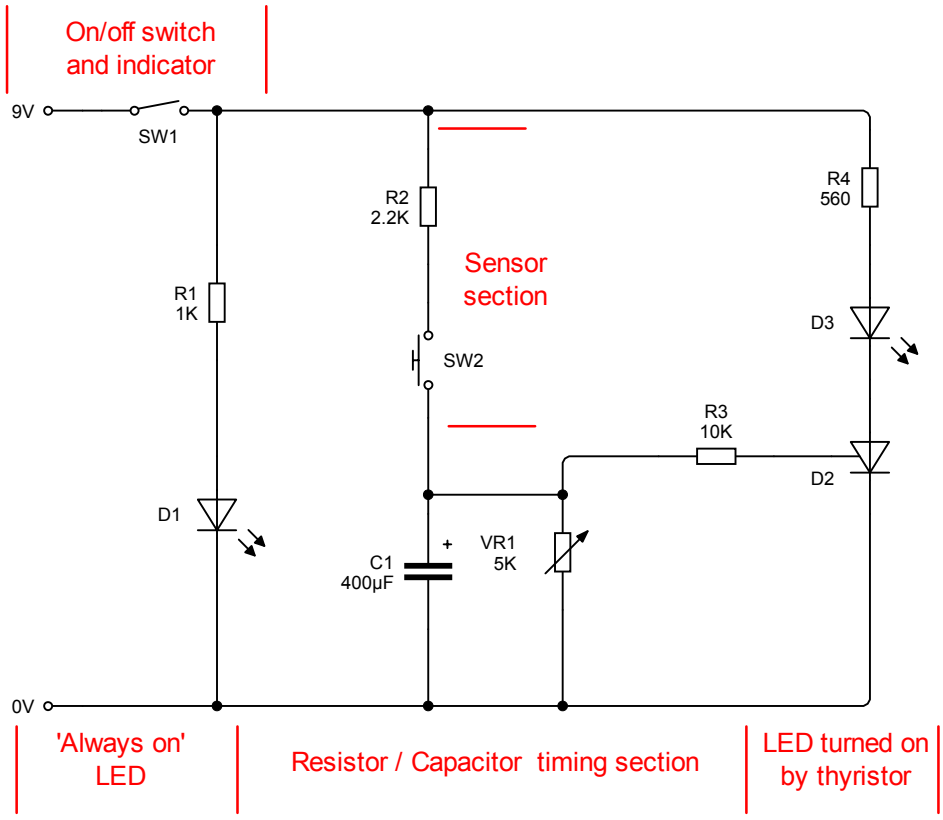
1 Tilt switch alarm

Simple alarms are normally triggered as soon as they receive an input from a sensor, which is fine if they are guarding something only ever touched when it is stolen. In real life a bicycle in a bike rack might be blown by the wind or nudged by someone removing the bike next to it.

This project improves on the simple alarm circuit by sensing the charge in a capacitor. When the capacitor is fully charged it will trigger a thyristor which in turn will drive either an LED or a buzzer.

2 The circuit diagram

If you have Livewire, open the file named **Sensor Alarm Circuit.lvw** which is available from our website at www.new-wave-concepts.com/courseware.html. The circuit has five main parts:



3 Hand layout

Very simple circuits like this one can be laid out using the tools within PCB Wizard. There are three different ways in which this can be done.

1. Import and automatically route the Livewire circuit.

2. Construct the circuit diagram using the Circuit Symbols Gallery within PCB Wizard and then automatically route it.

3. Construct the PCB layout directly using components from the PCB Components Gallery and then hand route the circuit by placing tracks and pads.

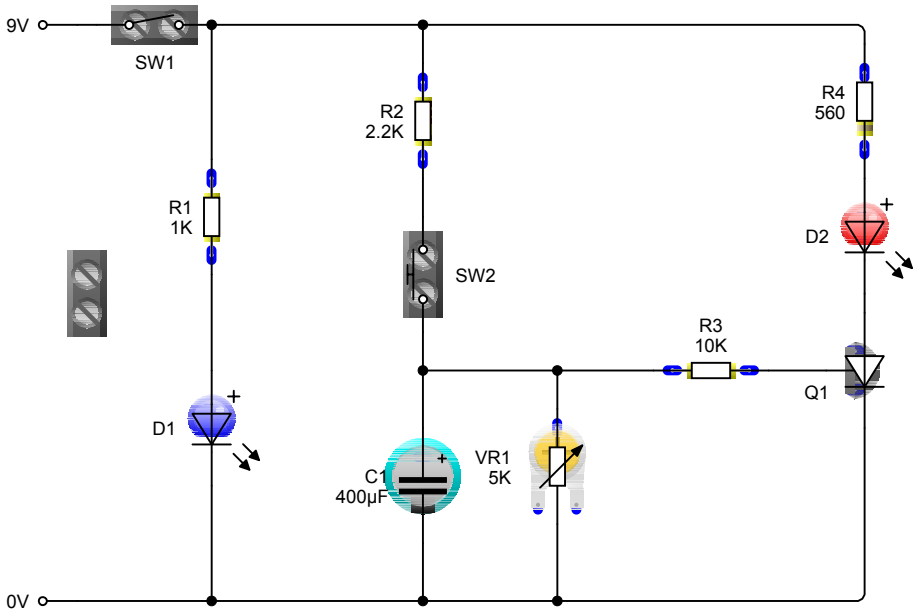
We will be using option 3 for this circuit.

First the components from the PCB Components Gallery are placed in position.

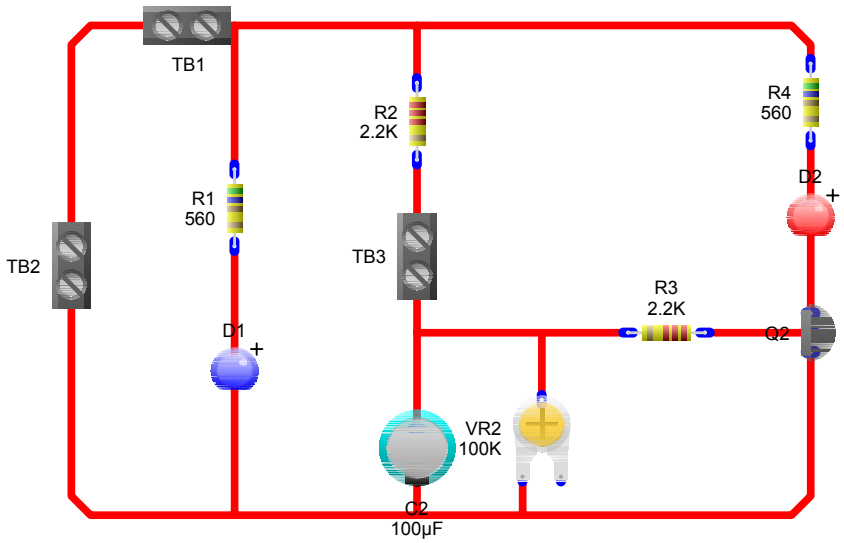
2 pin screw terminal blocks will be used to connect some components that are not soldered to the circuit board.

These components are:

- The battery clip
- The on/off switch
- The tilt switch



4 Joining the components together



With the components in place, the circuit diagram can be deleted and the components linked together using the Track tool.

This layout will work perfectly well but is much bigger than needed.

We are going to fit the final alarm into a case. Once we know the dimensions of the case we can plan the PCB layout more accurately, leaving adequate room for a battery and the switches.

5 Choosing a case

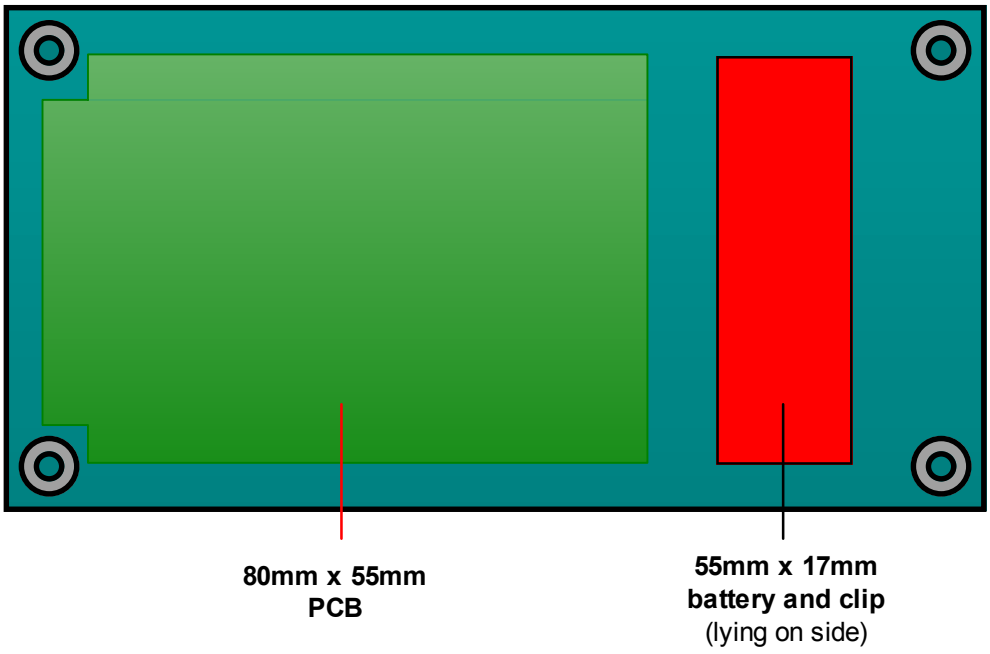
A case now needs to be chosen. Cases are available in many materials but the plain black ABS plastic ones are usually cheapest. The case we will use here has a number of features that will have to be taken into account when the PCB is designed. These are:

- Corners of box are used to secure lid on.
- Width of box reduced by slots used to hold a PCB in.
- Because of the method of manufacture (injection moulding) the sides all taper and therefore the base is narrower than the lid.
- The size of the power source and the connector. We will be using a PP3 battery but its connector adds another 5 mm to its length.
- The stated sizes for the box, 130 mm x 68 mm x 45 mm, are likely to be external sizes and therefore extra space will be required.

6 Measure twice... cut once

With the case chosen, we have decided that the total internal size will be less than the size stated. Normally you would lose about 8 mm of the width and 5 mm of the height though this varies between case designs.

The amount of room available in the base of the case will be about 120 mm x 60 mm. The battery and battery clip are about 55 mm x 30 mm, so we are actually left with only 80 mm x 55 mm for the PCB if we want to make sure it will all fit in.

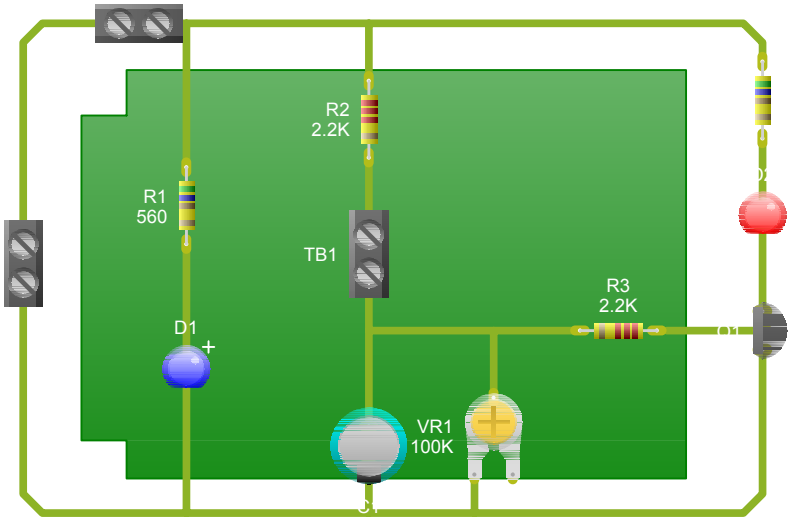


To make room for the case pillars, the PCB shape has been changed to a polygon (by clicking the right mouse button on the board and choosing **Polygon** from the **Shape** menu). Extra nodes are added to change the shape of the PCB.

Here the corners have been brought in by 8 mm on the left to give a bit more PCB space.

7 Shrink fit

With the dimensions of the PCB calculated, we must reposition the components from our circuit so that they fit inside. You must take care not to change or lose connections at this point.

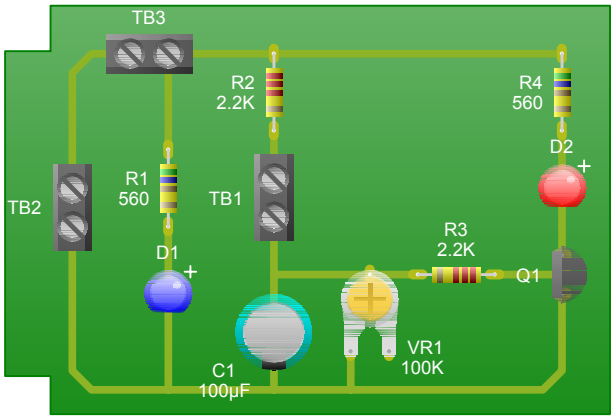


8 Sticky problem

By moving all of the components closer together, the circuit can be made to fit inside the PCB as shown..

We still need to adapt the PCB, however, so that it can be fixed to the case.

PCB pillars will be used to attach the PCB to the case and a steel spring battery holder will be used to keep the battery in place. Both are described below.



This is a PCB pillar. It fits through and locks into a 4 mm diameter hole drilled in the PCB and the base has a very strong self adhesive pad. Holes carefully sited in the PCB will allow four of the pillars to hold it securely a few mm off the base of the case.

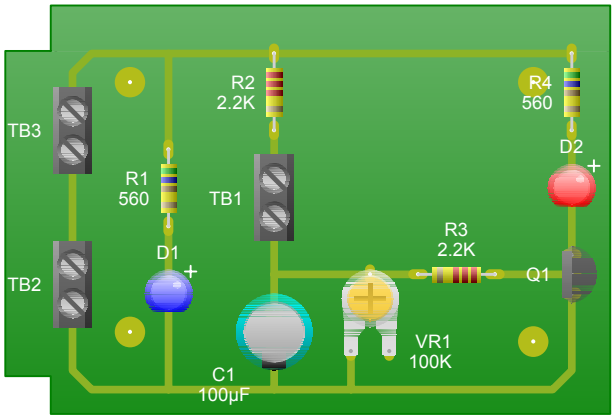
This is a spring steel battery holder. It is best fixed to the base of the case with a self adhesive foam pad, otherwise the clip flexes when the battery is inserted causing standard glues to become ineffective.

9 The hole story

Four PCB pillars will be used to fix the PCB in place. Each PCB pillar is self adhesive with a 20 mm square base. They will have to be fitted so that the centre of the 4 mm mounting hole is at least 10 mm from the edge of the PCB.

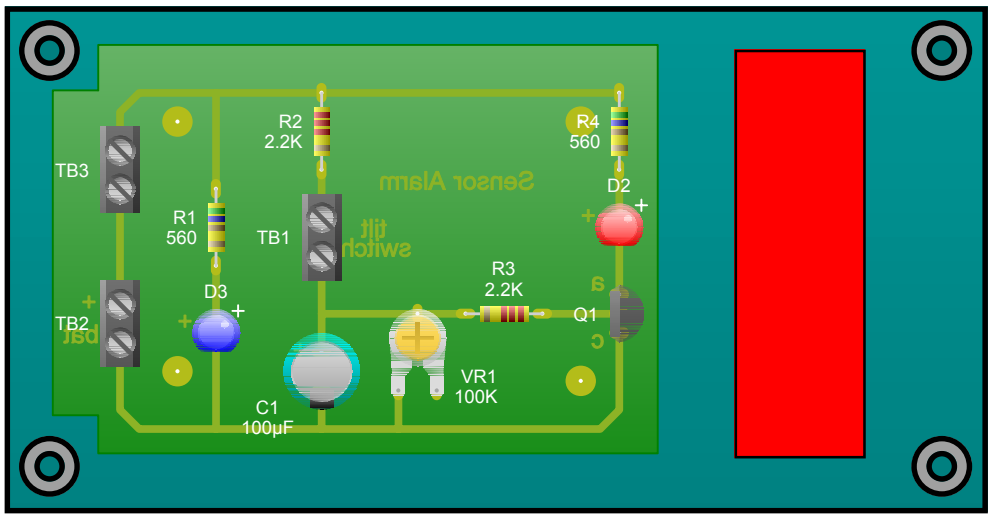
Pads will be added to the PCB to indicate where the holes for the pillars need to be drilled (if you switch to the **Normal** style view you will be able to see where the bases for the pillars will be placed).

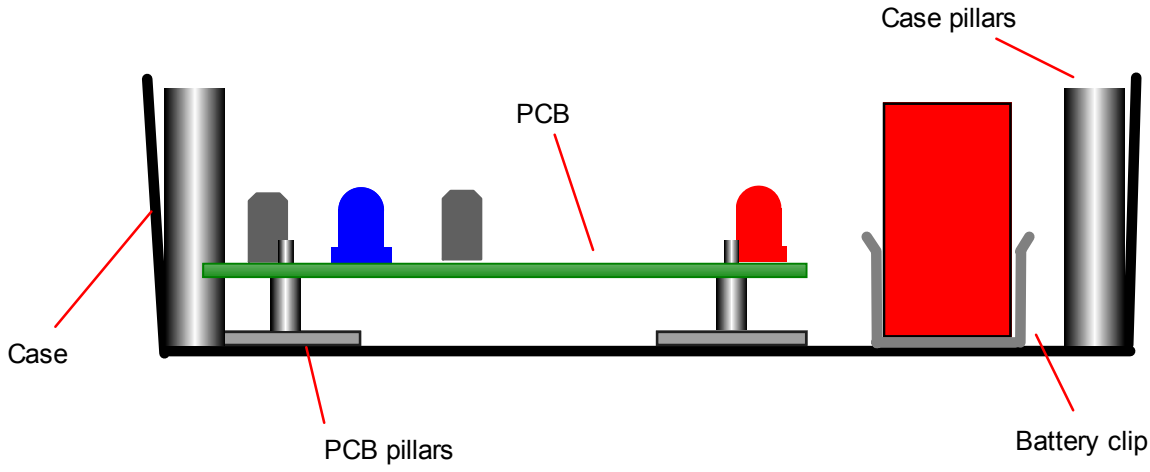
Note that the switch contacts (**TB3**) have been moved to fit around the 4 mm holes.



10 Adding the artwork

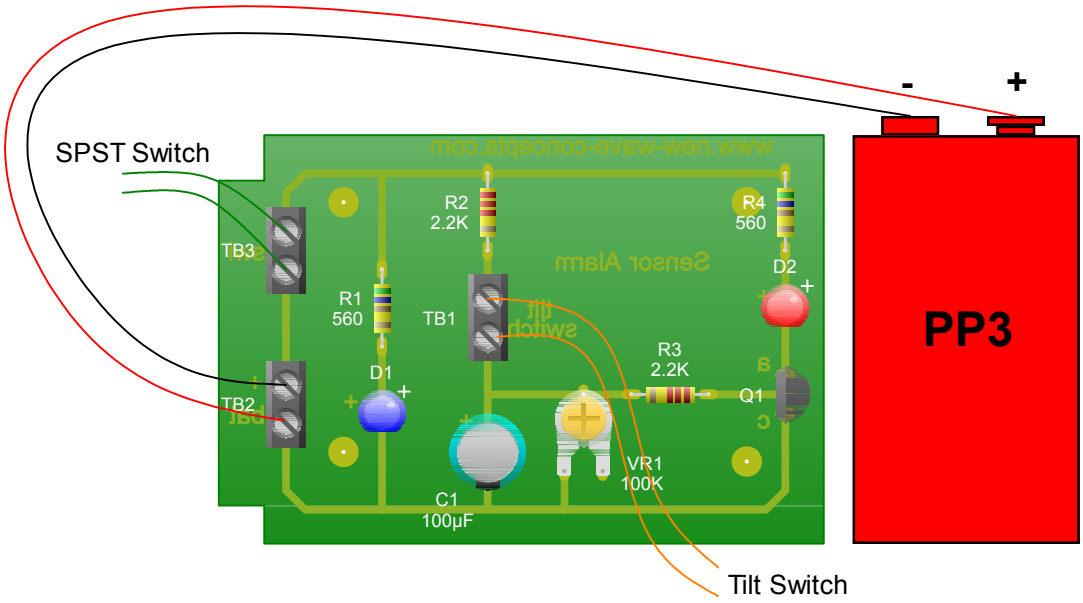
Finally, copper labels have been added to the PCB to make it easier to solder and to help identify the board once it has been manufactured. The diagram below shows the finished PCB inside its case.





11 Wiring it up

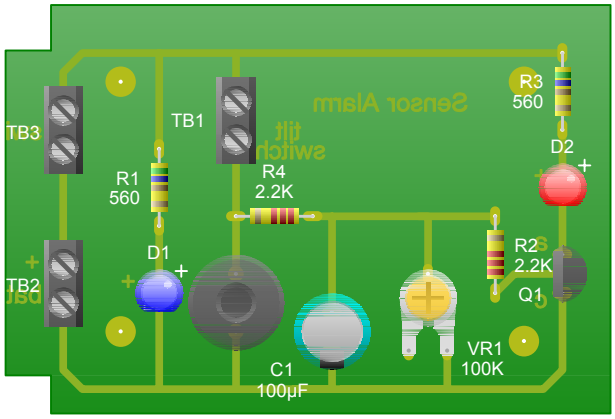
All that is left to do is to wire up the battery and switches. It is best to join the external components with flexible (multicore) wires. Rigid wires will work but fracture when bent a few times. The tilt switch can be stuck to the case sides with Blu-Tack and then adjusted to the right angle. Choose a tilt switch with two flexible wires attached, rather than the pin type. The SPST switch should ideally be a key switch.



12 Buzz, buzz, flash...

The project is now completed. However, you may wish to consider a variation on the original circuit design. This is shown below and has the tilt switch operating a buzzer in addition to the capacitor. This modified circuit has the benefit of providing an audible alarm when it is triggered.

To fit, a few things have to be moved around and also the tilt switch contacts need to come before the 2.2K resistor, otherwise the buzzer will be powered through it and will be silent.



Ideally the buzzer would also be on a 2 pin screw terminal but then most of the circuit would not be on the PCB.

13 Circuit diagram for buzzer version

The circuit diagram for the buzzer version of the alarm is shown below. You should note that the values of the resistor by the tilt switch (**R4**) and the variable resisitor (**VR1**) may need adjusting due to power drain by the buzzer.