# eWoggle Instructor Guide

Please read these instructions fully, and take particular note on ordering the PCB and checking the order. This kit has been made with EASYEDA Pro, and is using a Multicolour Silkscreen board that you order from JLCPCB. So please follow the ordering instructions carefully, (page 6 and 7) otherwise you will end up with a board with no Scouts Logo.


NOTE: This kit has been built with AAA Batteries, it only weighs about 42g. It is not heavy when attached to a standard rolled Scout Scarf.

Please do not redesign it with a button cell Lithium battery, as these are not safe to use with youth members.

# Design principles

The eWoggle is designed to be built by young Scouting youth members with limited to no soldering experience. It has many design principles that should ensure lower assembly failure rate, and a longer lasting robust kit.

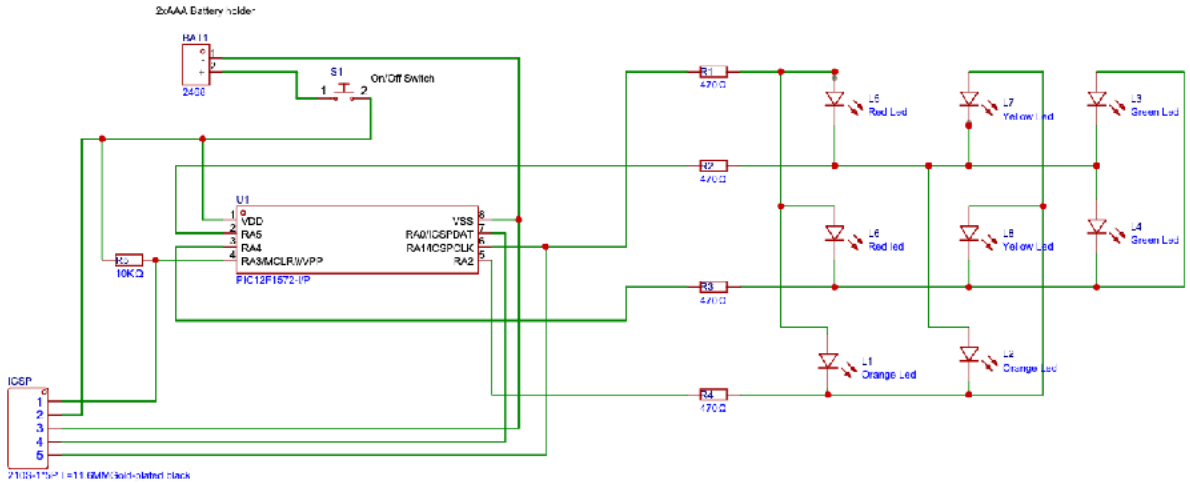
Design features

- Low profile on/off switch, partially shielded by the battery holder - to reduce the ability to be accidentally turned on, when squashed up against other things in a bag.
- PCB Mount battery holder, that wont break off.
- Larger solder pads 2.5-3mm, to make soldering easier for Scouts
- Wide spacing for radial component leads to prevent solder bridges, and components laid down to prevent breakage.
- Wider tracks so that if through holes are delaminated, then the component lead can be soldered to the track - saves replacing the board and starting again,
- No more than 1 track connected to a through hole to reduce the number of tracks to be reconnected if a pad is delaminated.
- Step by step assembly process, with stops to ensure faults are captured early enough to prevent issues that require rework.

## Design notes

- The eWoggle has been designed to be used as a woggle on a Scout Scarf. It has double sided Velcro that is fed between the battery holder and PCB, that holds it securely onto a Scout scarf.
- AAA batteries have been used in this project. A lithium ion battery was not used for safety purposes. By law, any lithium ion battery powered project requires a case with a captive screw to stop young children ingesting them. There are no PCB mount battery holders on the market with captive screws, available, so for safety reasons AAA batteries were used. Please do not modify this project to use a lithium ion battery.
- When the board is attached to a Scarf with Velcro, the Velcro pulls the scarf round the side of the battery holder, and effectively hides it. So the size of the AAA batteries are not an issue..
- The combined thickness of the battery holder and PCB is less than the thickness of the average woggle.
- The weight of the eWoggle with batteries is about 42 grams ( 3 times the weight of my Gillwell woggle), and the weight is not noticeable when attached to the average rolled Scout scarf worn round the neck.


# Circuit details




This project is based on a PIC12F1572 microcontroller. It generates 7 different patterns on the 8 LED's, in a roughly 2 minute cycle.

Only 1 LED is actually on at a time, as the LEDS's are arranged in a "Charlieplexing arrangement. The circuit draws about 2-3ma, and  works down to about 2.6v, so it should (but not tested) run in excess of 100 hours if you put in a good quality AAA batteries.

Its has a set of ICSP pads on the board, but as it does not have a blocking diode(to reduce component count) between MCLR and VDD,  its recommended that you do not use ICSP if the batteries are installed, so take them out first, or program the PIC's before putting them in.

# Pic programming options

You will need a PIC programmer

The following programmer and ZIF socket does the job nicely, just get the PicKit 3.5 with ICD2

https://www.aliexpress.com/item/4001283158851.html?spm=a2g0o.order_list.order_list_main.25.10f01802ENwKsh

Note MPLAB version 6.2 supports this programmer, higher versions no longer support PicKit 2,3 or 3.5

## Source Code

The code is available for download on the same page you found this documentation.
It is available in 2 formats -

1) Complied Hex file ready to program the Pic using MPLAB IPE.
2) C file in HTML form
3) Source main.c file, in case you want to play with the program and make it better. If you do play with it and add extra patterns etc, then please send me a copy and I will publish it on the Cobargo Website with your name on it. Please send both the main.c file and Hex File

Email to michael.Gross@nsw.scouts.com.au or venturers@cobargoscouts.org.au

Disclaimer - I am not an expert programmer - i am self taught using Google searches, and much hair pulling (only kidding), so don't groan too much at my code - it works but its not pretty !!!!

## Programming methods

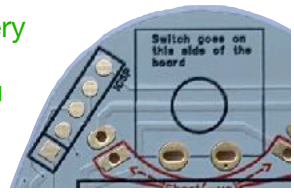
There are 2 methods you can use to program the PIC

1) Program them beforehand in a pic programmer, So that they just need to be put into the IC socket, once the Scouts have soldered it together - This is by far the easiest Option.

2) Program the Pic using ICSP, once the Scouts have soldered it together.

WARNING - This option is a lot more involved and I have only included it for those that like to do ICSP, and just because I could add it in - call it over the top design - i just could not help myself !!! If you have designed PIC boards before you will understand/sympathise - LOL !!!!

The Board has ICSP contacts on it. You would need to ensure that the Battery holder is not soldered in until the PIC has been programmed and Board tested using this method as a blank PIC will not illuminate any LED's, so you will not know if there are any assembly issues, and the battery holder is hard to remove once soldered on as the double sided tape is rather strong.



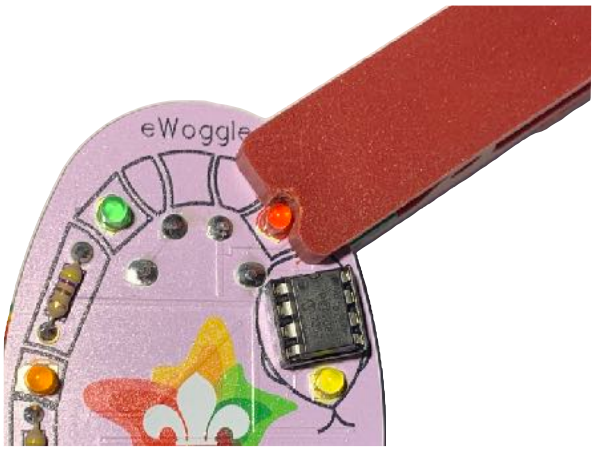

Also its important not to have batteries installed when you program the PIC as there is no blocking diode - to reduce component count and cost.

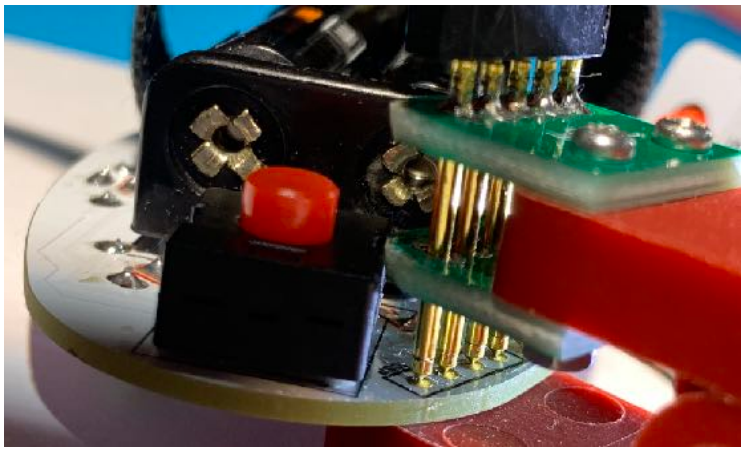
To program the PIC using ICSP, you will need one of the following - get a 5 pin Single row 2.54mm spacing

https://www.aliexpress.com/item/1005006358713448.html?spm=a2g0o.order_list.order_list_main.57.50371802Un3mEP

# Connecting to the ICSP Pins

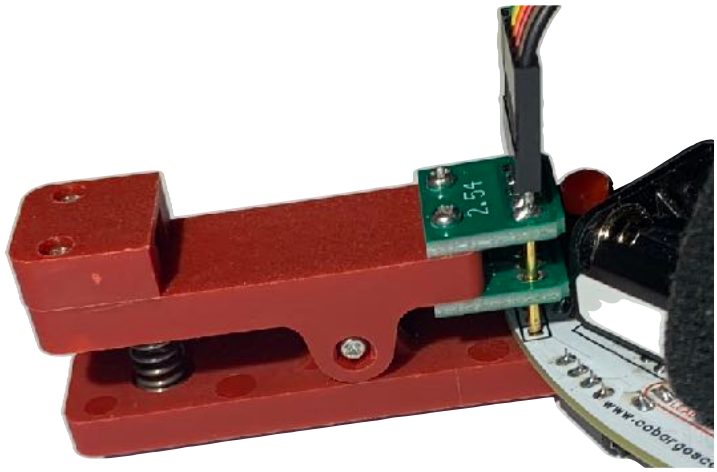
You will need to cut a slot, so that the PCB clip goes round the red LED and the pogo pins sit nicely over the ICSP pads. It will take a bit of trial and error to get this right.



Careful alignment when putting the clip on, is required as the clip will rotate off the pins. Clever people with 3D printers will come up with a solution for this !!!

The Square Pin on the ICSP pads closest to the IC socket is connected to the pin with the triangle on the PicKit (pin 1 MCLR)

# Ordering the PCB

The PCB has a multicolour Silkscreen on both Sides. The top side has the Scout Logo in colour, and the bottom side had a red outline denoting where the “-ve” LED leads go.
It has been designed with EASYEDA PRO. https://pro.easyeda.com/ The Pro version is free, and is the only version that supports the muticolour Silkscreen, so you must use it otherwise the Scout logo etc will be missing on the boards.

Also you will need to get the boards printed at JLCPCB, as the Gerber export with a multicolour silkscreen will work with JLCPCB, but i am not sure if it will work with other PCB manufacturers.

As usual I have provided the original design files, so that you can change logos and wording to suit your particular state/group

*** Attribution ***
Please note: these designs are provide free for Scouts to use, the only thing we ask is that you leave all references to Cobargo Scouts on the PCB’s. Small price to pay for the free designs.

The 2 references are
1- "Designed by Cobargo Scouts", and
2- Our web address “www.cobargoscouts.org.au"

READ CAREFULLY - Very important otherwise you will not get the logo on the board.

1) Once you have created an account on EASYEDA Pro, and uploaded the design files you will need to configure EASYEDA Pro to tell it that you are using Multicolour Silkscreens. Look at the following document. https://prodocs.easyeda.com/en/pcb/other-draw-colorful-silkscreen/

2) It describes how to configure EASYEDA pro to turn on Multicolour Silkscreen, and pay particular attention to the section on “**Exporting manufacturing files**”, so that the extra 2 silkscreen files are exported with the Gerber to JLCPCB, otherwise you will lose the Silkscreen.

3) Also carefully read the section on “**Order Colorful Silkscreen PCB**”, as you have to select the correct Base material PCB colour, Surface finish, gold thickness, advanced options etc etc.

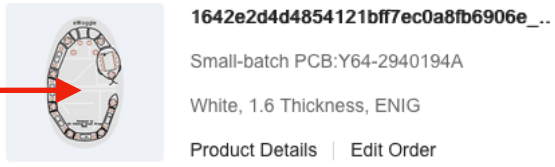
NOTE: In Small quantities these boards are more expensive as they are gold plated and use ENIG (google it), in large quantities, they are quite cheap.

**Please also read the next page on Checking the Order on JLCPCB**

# Checking the order on JLCPCB before paying

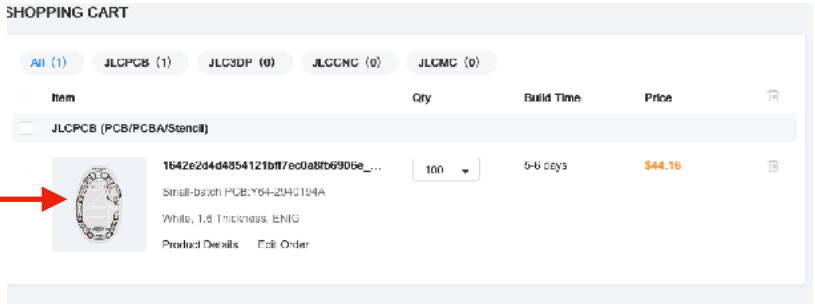
When you upload the Multicolour Silkscreen Gerber to JLCPCB, you will not see the colour silkscreen on any of the order/cart/checkout pages, don't panic, if you clicked all the correct options it will be there. Its only visible in a Gerber Viewer window.

Missing the Scout Logo




**Shopping Cart View,**
Once you have put it in the shopping cart, click on the board image and it will open in a Gerber Viewer. If you have done everything correctly, this will have the colour Scout logo on the front. If you look at the rear of the board, the red outline of the "-ve" LED pads only vaguely shows at this stage. This is ok, if the Scout logo appears on the front then the back will turn out OK

Click the board image in the shopping cart and it will open a Gerber Viewer window




**Shipping cost**
Note: there are multiple shipping options when you check out on JLCPCB. Global Standard Direct Line is the cheapest option. The available shipping options are determined by weight. However as a test I added 100 eWoggle boards to a shopping cart, and Global Standard Direct option was available, for $15US.

# Recommended LED's

Only use, 3mm diffused, red, orange, yellow and green LED's in this project. They all have a forward voltage in the range of 1.8 - 2V. The circuit will not power, Blue, White, purple etc LED's that require more voltage. These generally require at least 3.2 V and this circuit will not provide enough power to run them.

# Battery Holder

The board is designed to do take a PCB mounted AAA battery holder. You will need to put a pice of 2mm thick VHB tape on the end of the battery holder. This is to ensure that it is held securely against the board not only by the pins, but also by the tape. The Velcro used to hold it against the scarf goes between the board and the battery holder, so its important that the battery holder is securely attached to the board by the pins and the tape.

**Tape required**

3MVHB 4991 (2.3mm Thick)  10-12 mm wide - you will need 1*20mm lengths per battery holder.
can get 3M lengths from aliexpress

(Or any equivalent double sided tape approx 2mm thick)

## Name label

As the board is small, there is no room to put a white space to write the youth members name and group on. So order  the following Avery label and stick it on the side of the battery holder so that they can write their name and group name on it.
https://www.averyproducts.com.au/product/white-rectangle-stickers-932009

# Switches

The main ones I have designed the board for, are made by  HENGQI https://www.hqdz123.com/en/ and are available on Aliexpress and Amazon, they are all 12mm switches with 90 degree pins.

**Option  1 -  1212-112DB**

Search for 1212-112DB on Aliexpress
2 pin on/off switch
This is the preferred switch as the pins are longer and more robust than the other switch
They need 0.8mm tape, see below

## Tape required for the Switch

These switches only have pins on one side, so if they get knocked they may lift up. Ideally a piece of 3M VHB tape under them will help prevent this. Note 3M VHB tapes can take up to 72 hours to achieve full bonding strength, so it is advisable to stick this to the bottom of the switch a few days before they are due to be assembled by the Scouts.

- Use  0.8mm tape
  - 3M VHB 5608 tape (0.8mm thick) 10-12mm wide

https://www.aliexpress.com/item/1005007389297919.html?spm=a2g0o.order_list.order_list_main.5.488b18026RcnLF

# To Do list before the Day

1) Order Boards
2) Order Components
3) Source and buy the 2- Double sided tapes,
4) Put a piece of double sided tape on the battery holder

5) Put the name label on the side of the battery holder.

6) Put the double sided tape onto the back of the switch

7) **Solder requirements** - Cut 30cm solder (approx 1.3g) for each kit
(Can do it with about 20cm, if you are good at soldering)

8) Download the PIC eWoggle code from the Cobargo website and program and test the IC's so that they are ready to be put into the boards

9) Print one copy of the Component Sorter per Soldering station, and laminate them. These are used by the Scouts to sort the components, do not let them start soldering until you have checked that they have sorted them correctly

10) Print one copy of the Assembly guide per Soldering station per - it is recommended that you laminate them.

11) Package only the following components

- 5 * Resistors (4 *470 ohm, and 1* 10k ohm)
- 1-  8- pin IC Socket
- 1 * switch
- 8 * Leds  - (2 of each colour)
- PCB
- Solder
- Velcro strip

- Do not give out the battery holder, batteries or IC. When the youth members are assembling them, get the Youth to assemble all the components listed above that they have been given 1st (see assembly guide).
- Check the board for soldering issues, dry joints, joined pins, insufficient solder etc.
- Then if all is ok, put in the IC and test the board by putting the batteries in the battery holder and just putting it into the board and twisting is slightly so that it contacts the battery socket pads, and check that it works. Also check that the switch turns off and on alternately. Once you have checked it then let them solder in the battery holder, feed through the Velcro Strip. and then get them to put their name on the label on the side of the battery holder.