GPS Locator for Amateur Radio Stations

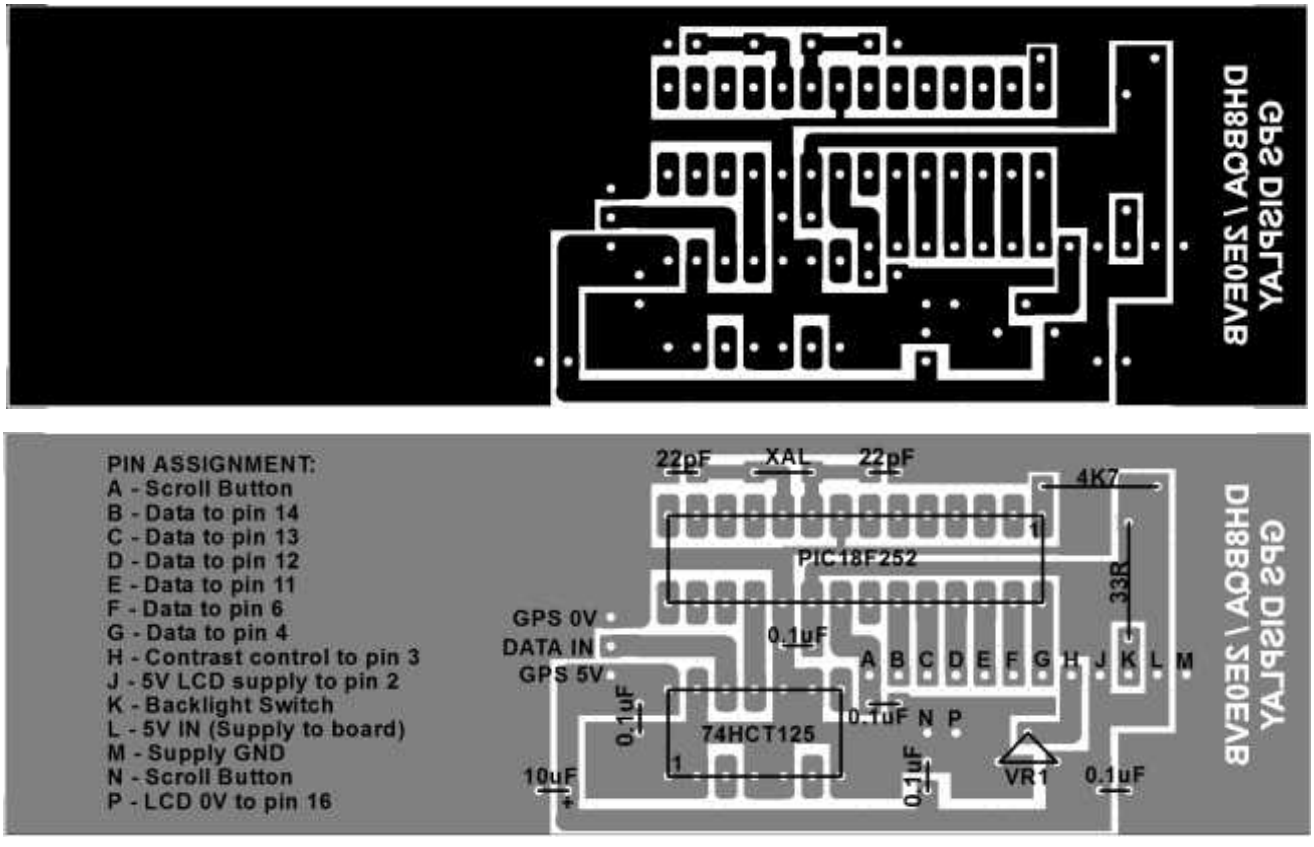
This is a small project I put together for a club construction project for the Stafford and Districts Amateur Radio Society (SADARS). The original concept and the program on the chip was produced by Olli Dröse, DH8BQA. Please head over to his website to find out more: www.dh8bqa.de/gps-display. You can also find the **hex code** for the PIC, his **circuit diagram** and **board layout** free for download on that page.

The unit displays your current location (IARU Locator and Latitude/Longitude), elevation, speed, direction and the date and time. See the cluster of photos of the screen for more details. It is intended for use by radio amateurs operating on field days, SOTA activations, in the car or simply as a shack clock. The possibilities are endless! Wherever you choose to operate from, if you have this in your backpack or pocket you will always know exactly where you are. The project was put together for around £25.

The unit is compact and battery powered, simple to use and can be charged from any USB power source (e.g. mobile phone charger, computer, car charger etc.). The unit should last a whole day's operation on one charge, and the display is easily readable by the day, and by night thanks to the backlight. The position is determined using a GPS receiver built in to the unit. In large buildings it may need to be near a window to get a GPS fix.

There is only one circuit board in the project which has only a handful of components on it, none of which are surface-mounted. As a result, it took no more than a few hours to build completely.

My PCB design is pictured below and a full scale PDF copy is available from this link. Remember to ***print it at 100%*** too. The image is the copper side viewed THROUGH the board from the component side.



I give all credit to Olli, this was his idea & PIC program, and I followed his circuit diagram to design my own PCB layout (so that the PCB fitted nicely into the case I was using).

Initially I built the project on breadboard, following Olli's circuit diagram. The first version of my PCB proved a little unstable and required extra decoupling capacitors to work reliably. The revised board, above, uses a much larger area of grounded copper, and includes the necessary decoupling capacitors. As a result, this revision has proven entirely reliable.

The differences between Olli's version of the project and my version are:

- **Battery Operated** – I incorporated a "USB emergency power" battery pack to provide a 5V supply to allow the unit to be used on SOTA expeditions etc.
- **Different Case** – Olli used an ABS case to house his project. I used a translucent blue polycarbonate case. I also laser engraved my callsign and button labels onto the inside of the front panel.
- **No Bluetooth** – Olli incorporated a bluetooth transmitter into his design to allow the GPS data to be used by a mobile phone or computer. I felt this was unnecessary for my purposes.
- **Different GPS Reciever** – Due to availability and price. The one I used ran from a 5V supply removing the need for 3.3V regulators as in Olli's circuit.
  Below is a list of the items used:

- 1 PIC18F252
- 1 Standard 2×16 LCD Display with backlight (Black-on-white/yellow/green give best visibility without the backlight)
- 1 Latching Switch, Black (For backlight) (ProPower PPW01076)
- 1 Latching Switch, Red (For power) (ProPower PPW01075)
- 1 Momentary Switch, Black (For screen scroll) (ProPower PPW01081)
- 1 Case (120 x 50 x 65 mm) (Hammond No. 1591C)
- 3 DIL Sockets, 14 pin
- 1 Buffer Chip (74HCT125, 14 pin DIL)
- 1 Crystal (12MHz Low Profile)
- 1 Battery Pack (**2600mAh USB PowerBank**)
- 1 10k Trimmer (Bourns 3362P-1-103LF)
- 1 GPS Receiver (VK16E, SIRF III Chipset, others will work as long the default baud rate is 9600)
- 1 Header Strip (12 pins)
- 1 33Ω Resistor
- 1 4k7 Resistor
- 5 0.1uF Capacitors
- 2 22pF Capacitors
  1 10uF Capacitor

Download the build manual from THIS LINK. *Last updated June 2016.*

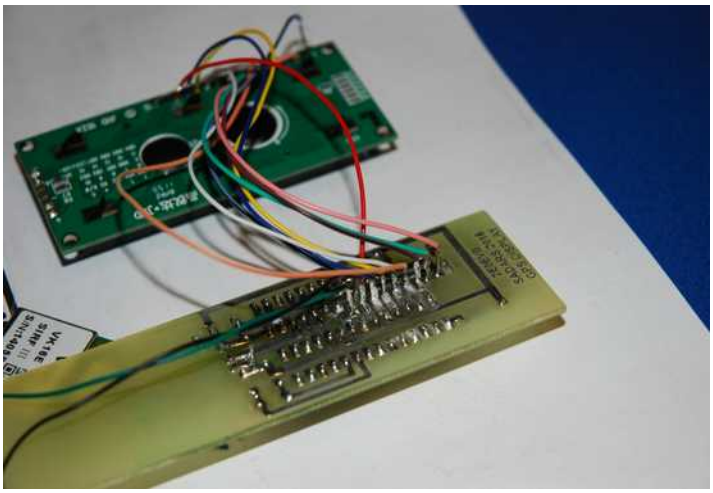
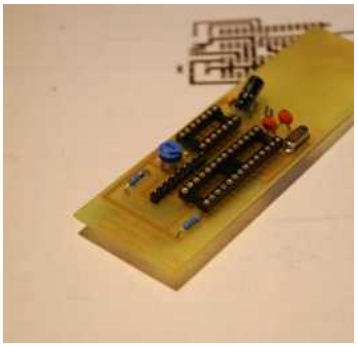
There are 3 templates for the front panel; a DXF FILE, a PDF WITH DIMENSIONS (in mm), and a PDF WITHOUT DIMENSIONS.

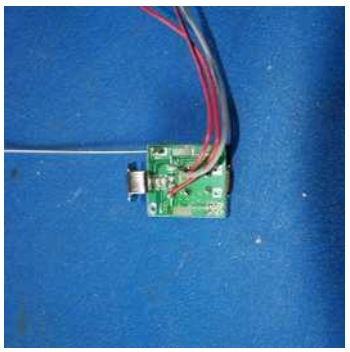
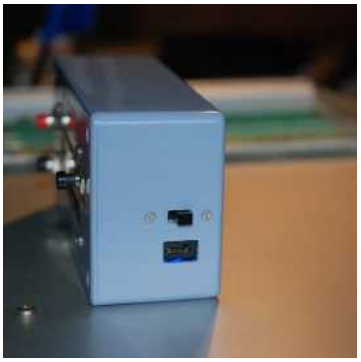
There is also a STICKER TEMPLATE to aid location of the holes in the side of the case. I suggest printing this at 100% and sticking it to the side of the case. Bear in mind, this is designed for the cases we used in the club, in the list above. The mark on

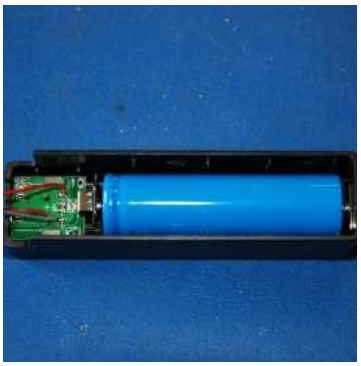
the edge of the sticker should be in the centre of the gap between the 5th and 6th ridges on the inside of the case.

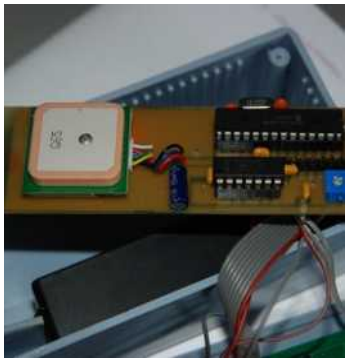
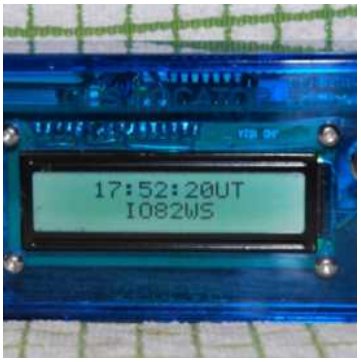
Here are some more photos of the prototype build:

SADARS have begun building the definitive version of this project using the final version of the circuit board as available above. One such unit completed by Steve 2E0HHK proved itself on a SOTA activation, as you can see below:

SADARS have begun building the definitive version of this project using the final version of the circuit board as available above. One such unit completed by Steve 2E0HHK proved itself on a SOTA activation, as you can see below:

There are a few slight differences between the prototype above, and the refined version which uses the PCB above and is described in the build manual above, and which SADARS will be making as a club project:

- **Power Options** – the prototype will not power on whilst on charge. This is due to the circuitry in the battery pack. The final version incorporates a slide switch on the side to choose between external and internal power so that it can be powered on even whilst the battery is charging.
- **Cases** – the prototype used a blue case with the lettering laser engraved onto the inside of the front panel, the final version uses a black/grey ABS case, laser etched on the front and with paint rubbed into the lettering to enhance the readability.
- **Screen** – As mentioned above, the final version used a black-on-yellow display due to better readability without the backlight and availability.
- **Capacitors –** The capacitors that are tacked onto the switch and underside of the PCB in the prototype have been incorporated into the revised PCB available from this page.

Finally, many thanks again to Olli Dröse, DH8BQA for allowing me to publish this article. Please pay his website a visit: www.dh8bqa.de

73,

Joe 2E0EVB