Shack Clock kit

PCB Revision: "QCU Rev 1" or "QCU Rev 3"

1. Introduction

Thank you for purchasing this QRP Labs Shack Clock kit. The kit uses the same PCB and bag of components as some other QRP Labs kits. So there are several components in the bag which are not necessary for the clock kit. You will have surplus components after constructing this clock kit!
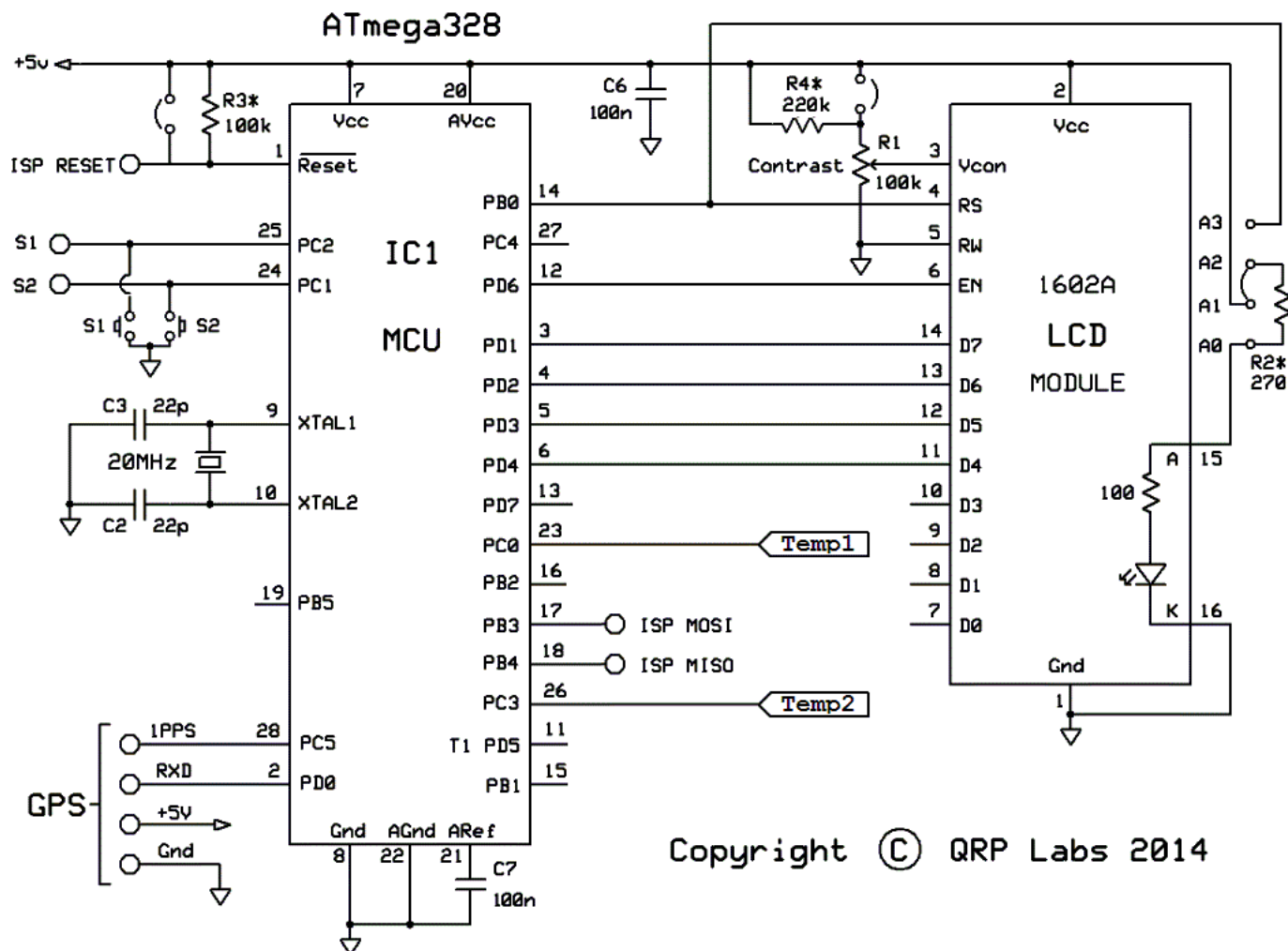
This assembly manual must be read in conjunction with the clock operation manual!

2. Design

The ATmega328 AVR microcontroller (IC1) is pre-programmed with firmware to control the LCD, buttons, and GPS interface.

For best results, please ensure the use of a well-regulated, clean and solid 5V power supply. The majority of problems with the kits arise due to poor power supplies! Not all wall-wart type or switched mode supplies are clean enough to power the kit.

Shack Clock Kit

Copyright © QRP Labs 2014

3. Parts List

In the following parts list, components which do not apply to this Clock kit are greyed out.

3.1 Resistors

R1   100K preset potentiometer, for LCD contrast adjustment (code "104")
R2   No resistor – space is provided on the PCB only for reducing LCD backlight brightness
R3   No resistor – space is provided for a 100K resistor, if you want to do AVR firmware upgrades
R4   No resistor – space is provided for a 220K resistor, for smoother contrast adjustment
R5   4.7K preset potentiometer, NOT INSTALLED in the Clock kit
R6   4.7K resistor, NOT INSTALLED in the Clock kit

3.2 Capacitors

C2, 3   No capacitor – space is provided for optional 22pF capacitors if required
C6   100nF (ceramic, marking 104)
C1,4,5,8   100nF, NOT INSTALLED in the Clock kit
C7   100nF (ceramic, marking 104)

3.3 Semiconductors

Q1, 2, 3   BS170, NOT INSTALLED in the Clock kit
D1   NOT provided in the Clock kit
IC1   Pre-programmed ATmega328 microcontroller

3.4 Miscellaneous

Two push buttons, S1 and S2
16-column x 2-row LCD (HD44780-compatible) with blue LED backlight
Printed circuit board, 80 x 37mm size
Wire for winding toroid, NOT USED in the Clock kit
Socket for IC1
20MHz quartz crystal
Two 10-pin female header sockets, NOT INSTALLED in the Clock kit
Two 4-pin female header sockets, NOT INSTALLED In the Clock kit
16-pin female header socket
Two 16-pin male header plugs
Four 12mm nylon hex PCB spacers
Eight 6mm M3 screws
Relay RL0 – not supplied in the Clock kit

4. Construction
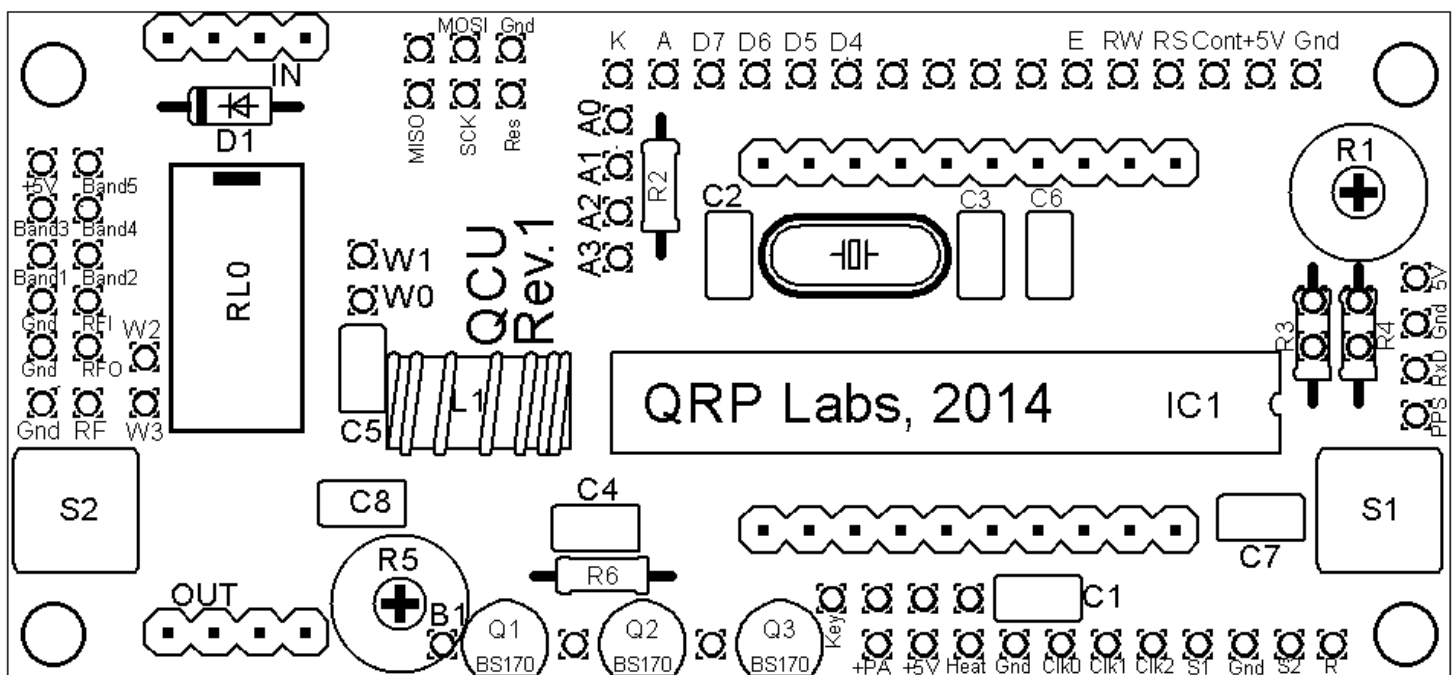
4.1 General construction tips

Parts placement is defined by the silkscreen printed legend on the PCB, so please observe it carefully, paying particular attention to the correct orientation of the semiconductors.

The PCB is quite small and the parts are close together. You are recommended to use a low wattage iron with a fine tip, and fine solder e.g. 1mm diameter or less. Take care not to overheat the PCB and risk damaging it. A well-lit area and magnifying glass are useful. Be careful not to bridge solder across closely packed connections. Some of the joints are very close to each other. I recommend checking with a DVM to make sure no solder bridges have been created.

Note that components R2, R3, R4, C2 and C3 are not supplied in the kit (R2 may be added by the builder if lower LCD backlight brightness is desired). IC1 (the microcontroller) has an IC socket, in case in future you wish to change the microcontroller e.g. for a firmware upgrade for new features, etc., or in case you wish to program it yourself.

4.2 Construction steps

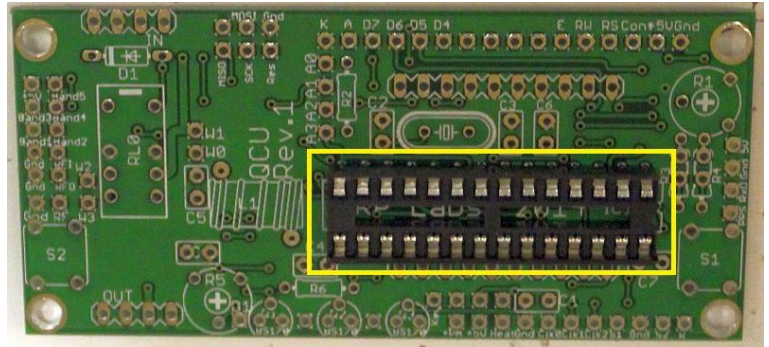Please refer to the parts placement diagram below.

Pay special attention to the orientation of the semiconductors. For IC1, the dimple in the PCB silkscreen must be aligned with the dimple at the top of the IC socket and the IC.

The order of construction is not important. However, a good principle to follow is to install the smaller components first, so that the larger ones do not prevent easy access. One suggested order of construction is described below. I recommend following it carefully.

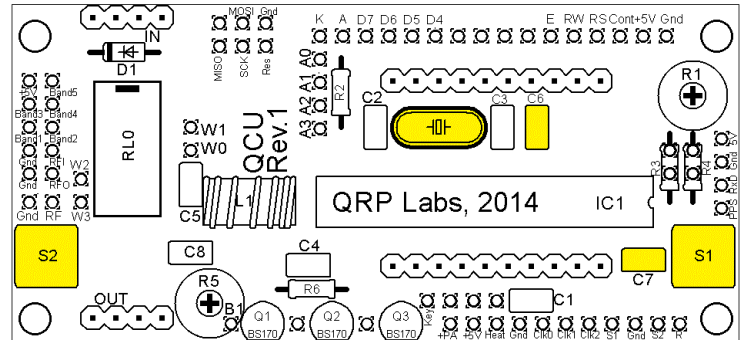1) **Solder in the socket for IC1.**

To avoid confusion or mistakes later, align the dimple at one end of the socket, with the dimple illustrated on the PCB. The dimple should be at the end nearest the right-hand edge of the PCB.

2) **Solder capacitors C6, C7 quartz crystal, and buttons S1/S2**

There are five 100nF (0.1uF) capacitors supplied but only two (C6 and C7) are installed in the Clock kit.

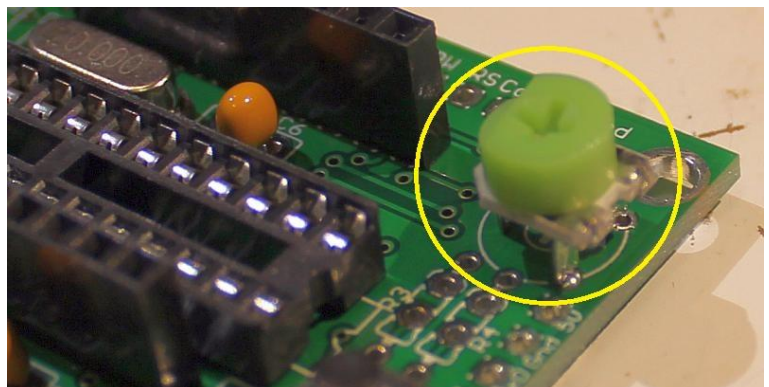The other 100nF capacitors C1, C4, C5 and C8 are NOT installed in the Clock kit.

Note that the 22pF capacitors C2 and C3 either side of the crystal are NOT fitted – the 20MHz crystal has been found to oscillate reliably without them (there were some occasional issues when the 22pF capacitors were fitted). Therefore 22pF capacitors C2 and C3 are not supplied.

3) **Fit and solder R1, the preset potentiometer that sets the LCD contrast.**

Be careful not to mix up the presets R1 (100K) and R5 (4.7K)! The code on R1 is "104". R5 is not used in the Clock kit.

It is a slightly tight fit but apply pressure carefully and evenly, and the pins of the potentiometer will fit into the PCB perfectly.
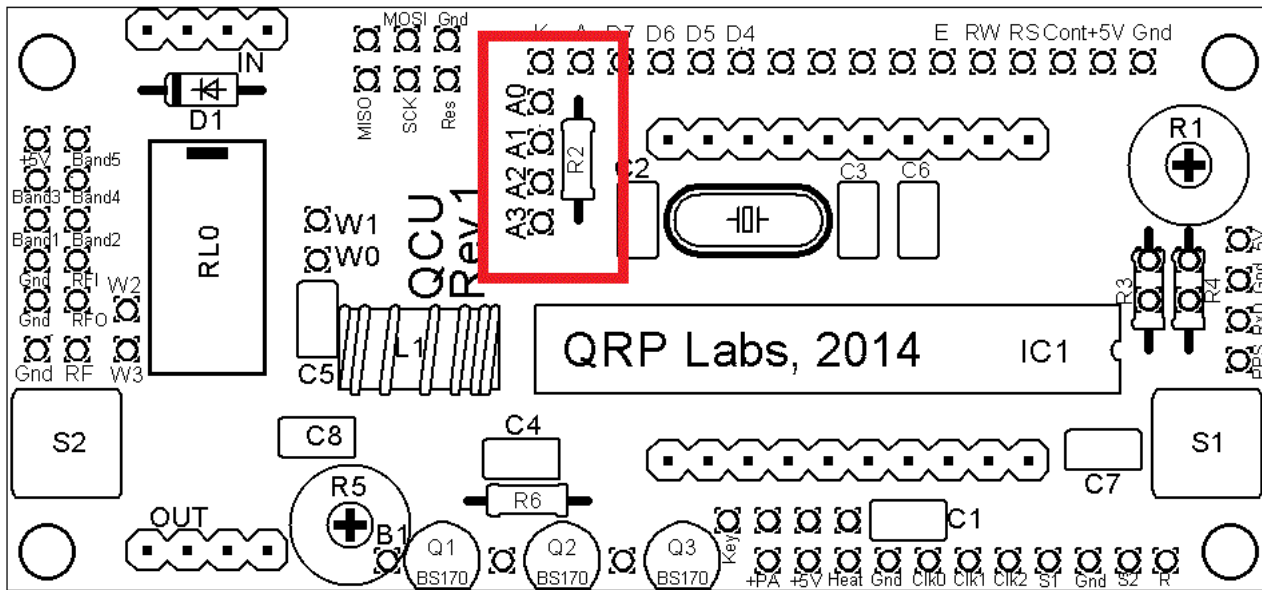

Adjust this potentiometer to the fully clockwise position initially.

4) **Solder wire jumpers depending on your requirements.**

The wire jumpers can be made from offcuts from the other components e.g. capacitor leads. When installing these wire jumpers, I find it convenient to install them as a small semi-circular arch, perhaps 5mm high off the board. This is so that later, if they need to be removed (for different hardware options configuration), you can easily cut the links using wire cutters. It is easier than un-soldering them.

Three wire jumpers are needed depending on your preferences, as described in the following three sections.

LCD brightness control

The LCD module has a blue LED backlight. The brightness of the LED can be set up using a jumper wire at the locations A0..A3, or optionally a resistor R2 which you may select for your preference. Choose one of the four options for the backlight as listed below.

 a) Always full brightness: For basic operation connect A0 to A1 using a wire jumper (made from a capacitor lead offcut, for example). The blue LED backlight of the LCD module will be at full brightness. This is shown as the Red line (left). In this case, do not fit any resistor at the R2 position.

 b) Reduced brightness: You may find the LED backlight of the LCD module too bright for your liking, or if you wish to operate from battery power, perhaps you consider that it consumes too much current. In that case, fit a wire jumper between A1 and A2, and use a suitable valued resistor at the R2 position: experiment, e.g. starting at 100-ohms. 180-ohms has been reported to work well. 220-ohms or 270-ohms may also suit your taste.
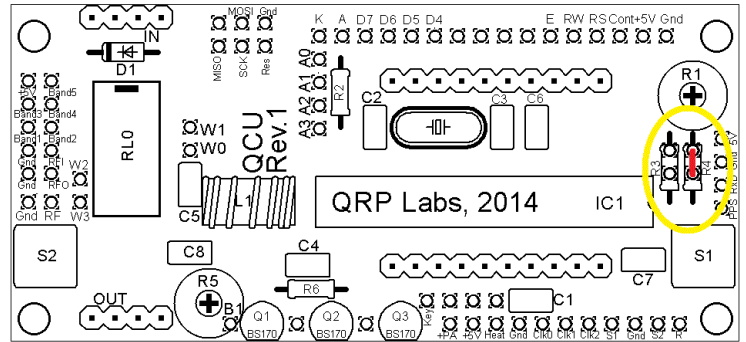
 c) Software variable brightness: The U3 firmware contains a "Backlight" configuration setting which can be set from 0 (backlight OFF) to 9 (maximum brightness). This is achieved using pulse width modulation. To enable this setting, you must connect A0 to A3 using a wire jumper (do not fit any resistor at the R2 position). Note that option c) using Pulse width modulation may cause a whine sound interference on a radio receiver, due to the on/off switching of the LED current.

d) Switched backlight: If you do not wish the LCD module's LED backlight to be on all the time, then instead of the wire jumper, you could connect wires to an on/off switch. This could be useful for battery powered operation, to reduce current consumption. For example, you could switch the display on to set up the kit, then leave it switched off during operation. The official QRP Labs U3 Case and accessories kit includes a spare switch which could be used for this purpose.

LCD contrast adjustment

Resistor R4 is an optional resistor, which is in series with the top of the contrast adjustment potentiometer R1 and +5V. Typically the contrast voltage required is less than 1V and the adjustment is rather sensitive. If you wish, you can fit a 220K resistor in this position, which will make the contrast potentiometer less sensitive to adjust. Install one of the two choices:
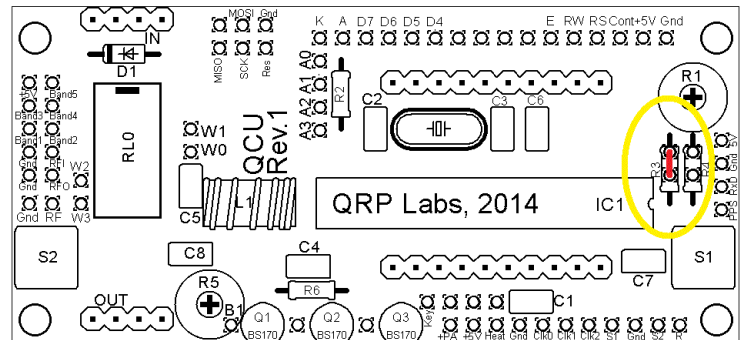
a) A 220K resistor at R4 (this resistor is not supplied in the kit, OR

b) A jumper wire in the position shown, and NO R4.

Microcontroller reset

Resistor R3 is an optional resistor, to be fitted to enable in-circuit-programming for those wishing to update their own firmware and having the necessary equipment. Install one of the two choices:

a) A 100K resistor at R3 (this resistor is not supplied in the kit, OR
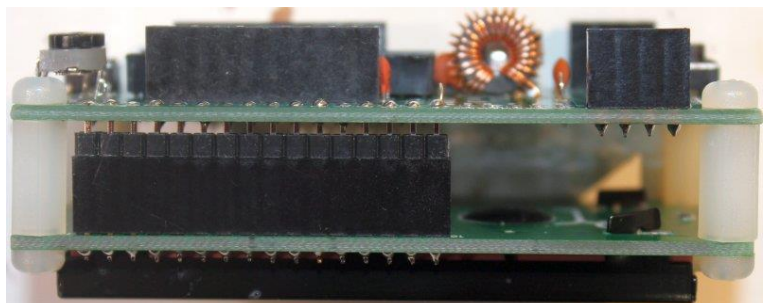
b) A jumper wire in the position shown, and NO R3.

5) **Install two 16-way connectors on the main PCB and LCD.**

Install two 16-way connectors on the main PCB and LCD. Either way is fine, but my recommendation for installing the LCD connectors, is to use the female (socket) connector on the LCD side and the male (plug) on the PCB side.

Be sure to solder the socket to the reverse of the main PCB!! Not the top side! It is very hard to fix later, if you get this wrong!

Bolt together the LCD module and the PCB, with the 16-way connectors in place but not soldered. Then when you solder them, you will ensure perfect alignment. Use the eight 6mm M3 screws, and four 12mm nylon hex spacers, to bolt the PCB to the LCD.
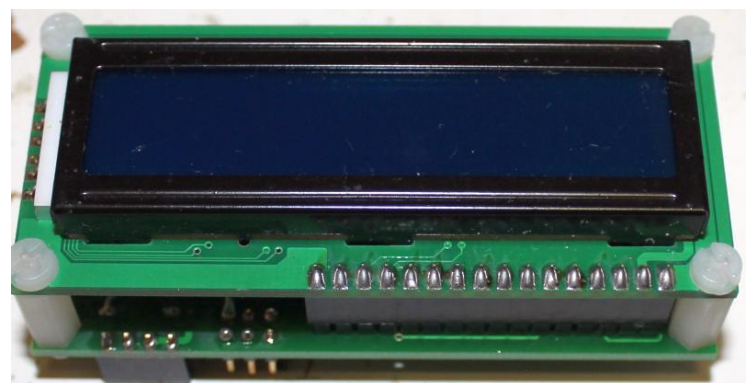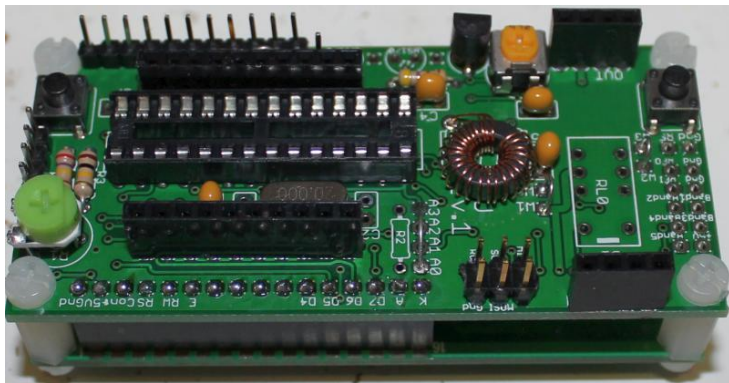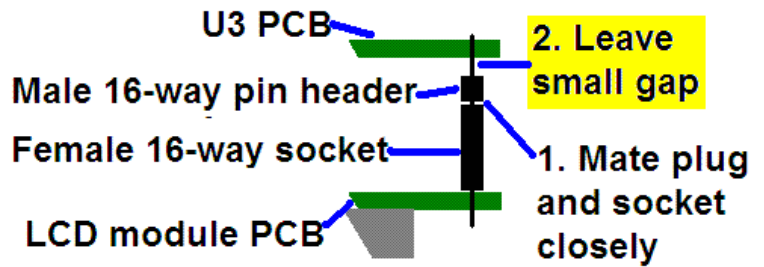
The combined 16-way plug and socket when mated together, have a height of a little under 12mm. Therefore it is necessary to have a gap somewhere.

My suggested method is illustrated to the right here. Closely mate the plug and socket. Leave the gap between the male header (plug) and the PCB. I have found this method works successfully.

As you can see in the photo, the remaining pin length poking through to the component side of the PCB is short. However, the pins are still present in the holes and solder will fill the holes and connect to the pins.

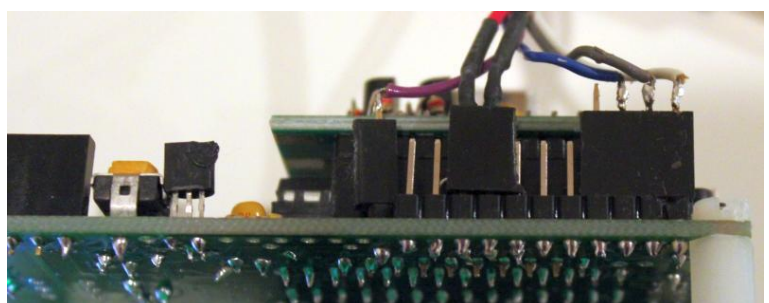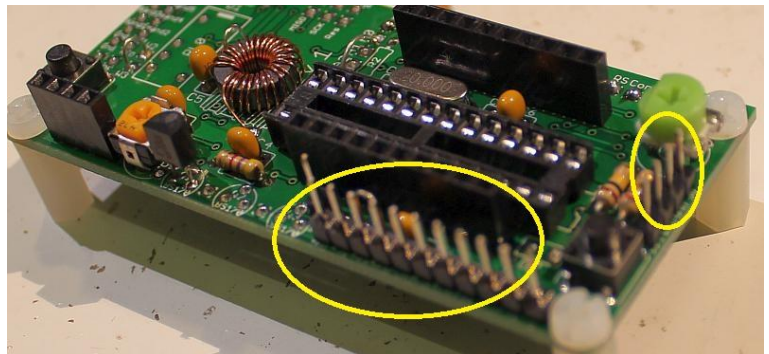At this stage, also solder the pins of the 16-way socket to the LCD module.

6) Optional header pins

The kit is supplied with TWO of the 16-pin headers. Only one is needed for the connection to the LCD module. The other is supplied so that if you wish, you can solder it to the PCB and use it for more convenient connections to the kit. You can either solder directly to the pins, or use the appropriate connector.

The 16-pin header strip can easily be snapped into smaller pieces. In the photograph (right) I have broken off a 4-pin section to use for the GPS connector, and installed an 11-pin section along the board edge. The final pin was installed in the "Key" connection pad.
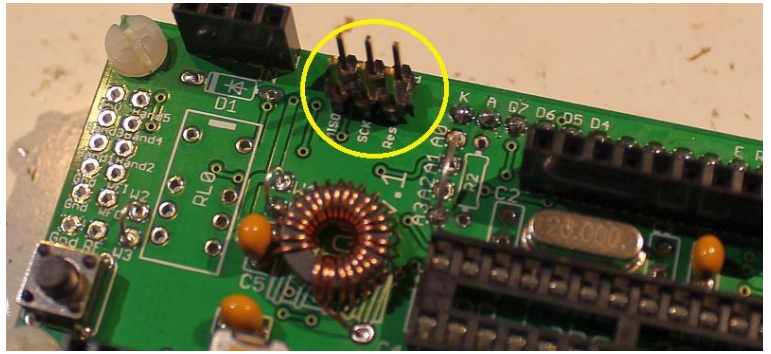
The photograph (right) shows an example of making connections to this header, using female sockets cut to size.

7) Optional In-circuit programming header

The PCB has provision for installing a standard Atmel 3 x 2 header (not supplied), for in-circuit programming of IC1 the ATmega328 processor.

This option is useful if you want to use one of the many inexpensive AVR programmers to upgrade the kit firmware, when updates are published by QRP Labs.
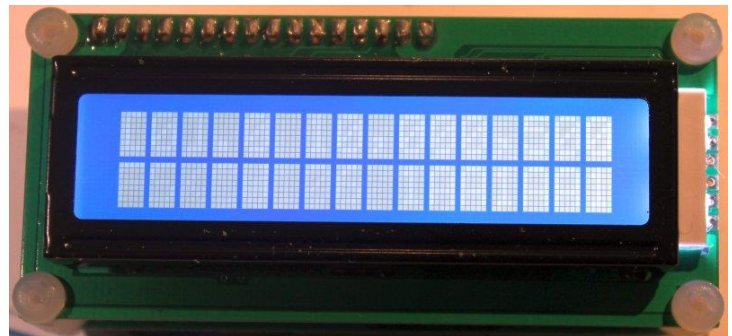
4.3 Module assembly

First the microcontroller IC1 should be inserted in its socket. Be sure to align the microcontroller chip correctly. The dimple on the chip must be at the same end as the dimple on the socket, and the dimple on the silkscreen legend on the PCB.

Important! Upon power-up, you will need to adjust the contrast potentiometer R1 to view the LCD properly. Turn it fully clockwise to start with (before applying power). Then turn it gradually anticlockwise until the displayed text looks correct.

The series of photographs below show the display with a) fully clockwise; b) fully anti-clockwise; c) adjusted perfectly. When adjusted perfectly, you will see "Diagnostic Mode" message on the screen.

When you see "Diagnostic Mode" it means all is well with the processor and the LCD communication. When you press the left button, diagnostic mode will be cleared. Then you should see the splash screen, which shows the firmware type and version number.
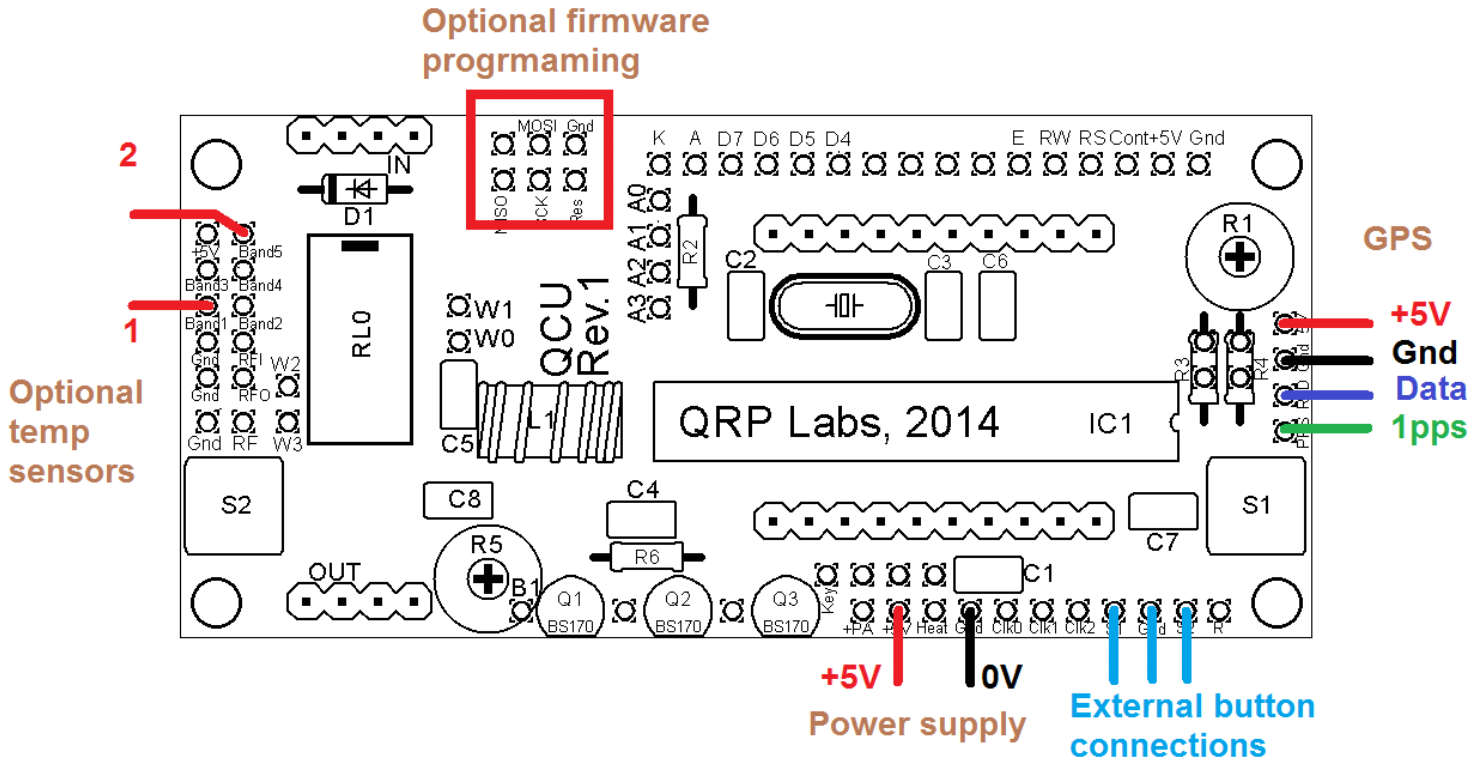

If you cannot see any text on the screen even after adjusting the contrast setting, you have a problem. Please refer to the "Troubleshooting" page on the QRP Labs website (see resources section below).
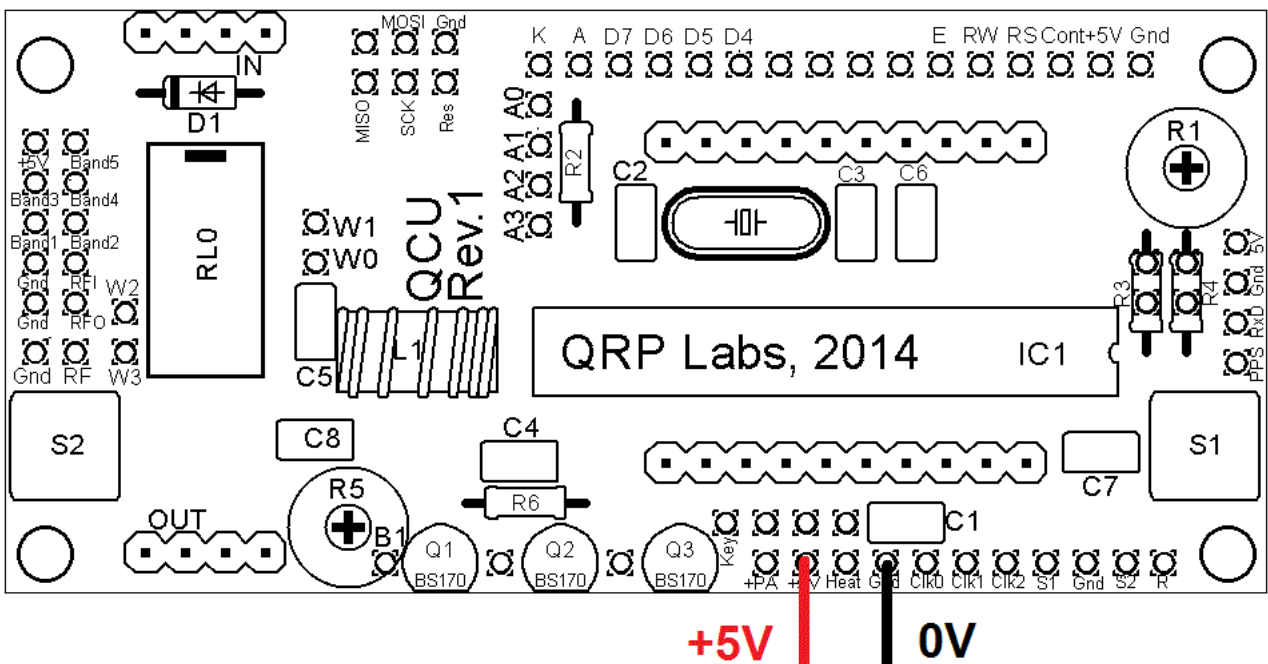
5. Connections

The pin-spacing is 0.1-inches and a suitable connector could be used (see section above).

The diagram below gives an overview of the connections to the PCB. Each part will be described in the subsequent sections.
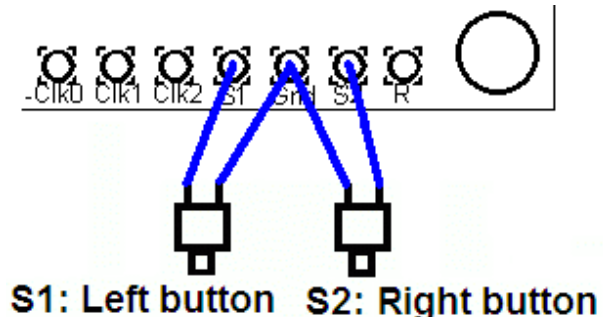

5.1 Power connection

The kit requires a clean well-regulated 5V supply.

5.2 Alternative button mounting option

You may use an external pair of buttons to control the kit. The button input signals are available at the edge of the board, labelled S1 (left button) and S2 (right button). The buttons should be of the push-to-make variety. The left button should be connected between the S1 signal and ground; similarly the right button should be connected between the S2 signal and ground.

S1: Left button   S2: Right button

Suitable buttons are supplied in the official QRP Labs U3 Case and accessories kit, this case kit is suitable for this clock kit.
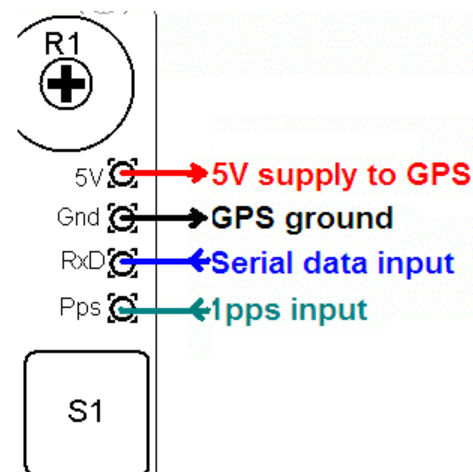
5.3 Optional connection of GPS module

A GPS module may be connected to the kit, to accurate time, and latitude and longitude which can be converted to Maidenhead locator format for encoding in the WSPR message transmission (depending on kit firmware type). All of these (and more) may be displayed on the Clock screen – please refer to the operating manual for details of the configuration.

Check that your module is powered from 5V. Many modules specify a 3.3V supply - in this case you will need to provide an external 3.3V voltage regulator. Where a 3.3V GPS module is used, the serial data and 1 pulse-per-second (pps) inputs are not a problem for the 5V microcontroller on the kit PCB - no voltage level conversion is required. Some GPS modules may need a pull-up resistor however.

If you are using the QRP Labs QLG1 GPS receiver, it is suitable for direct connection to these 4 pins (see right) without any pullups etc.
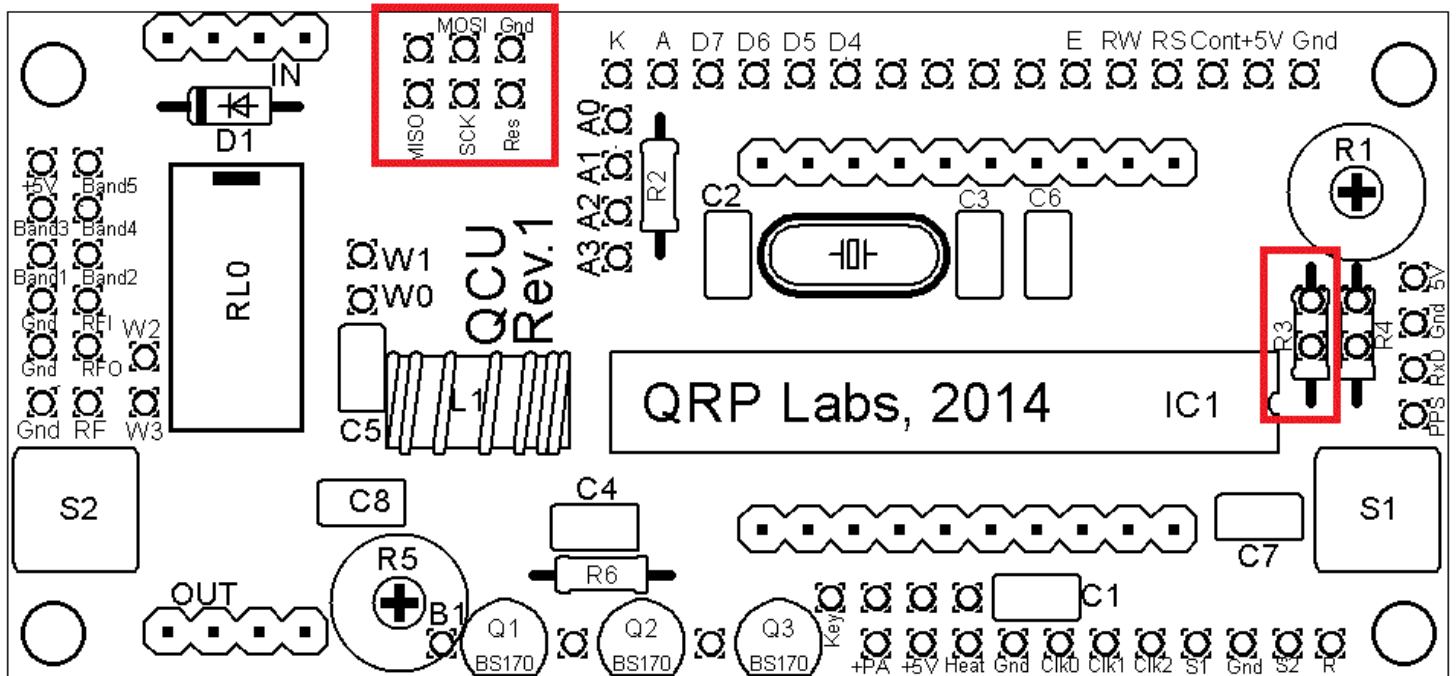
See operation instructions for details on how to configure the GPS interface.

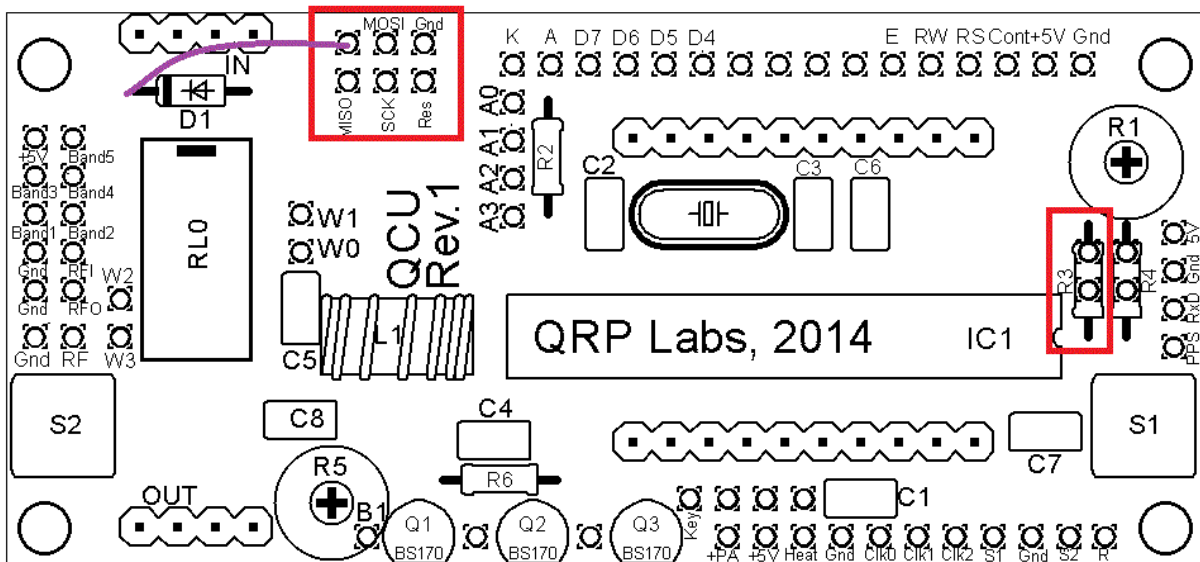## 5.4 In-circuit programming of AVR

If you own an AVR programmer, you can make connections to the PCB to allow updating the firmware without removing the AVR chip.

In this case, you should fit a 100K resistor as R3, not a jumper link (as described previously). This is shown enclosed in RED on the right side of the below diagram. The other connections required to your programmer, are MISO, MOSI, SCK and Res (reset), these are indicated in the RED square on the diagram. The connection layout is designed to fit a 2 x 3-pin header, to match common AVR programmer cables. Before use, you should check that the connections match your programmer, because there are several standards in use. This board is designed to be as flexible as possible.
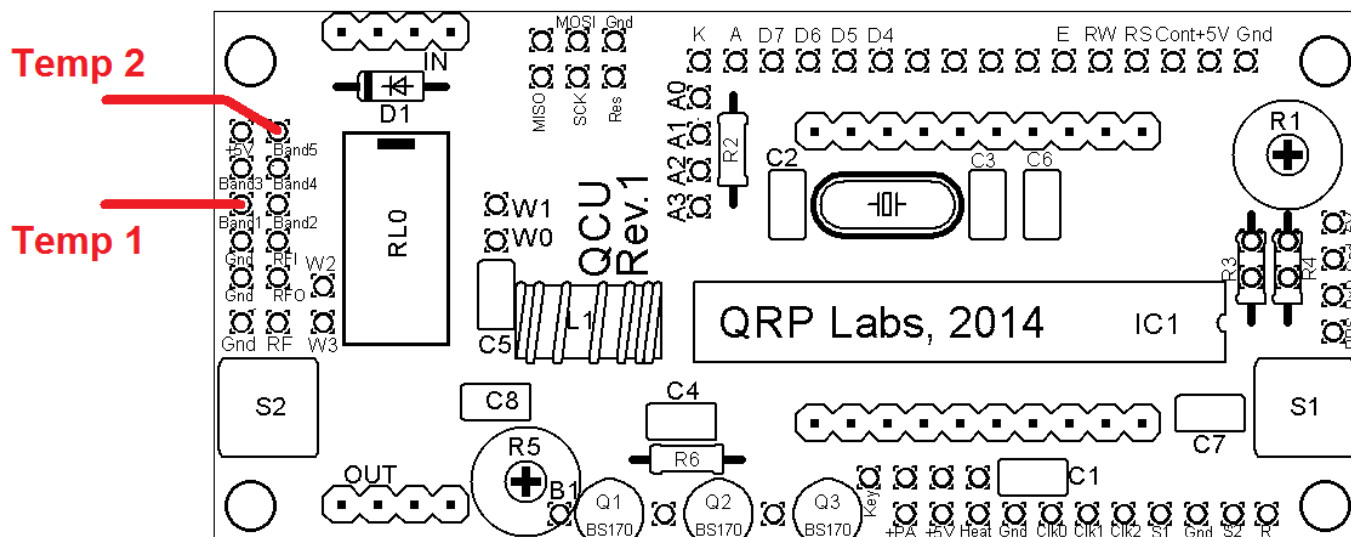
Notes:

1) Power needs to be applied to the QCU during programming! When you start programming, whatever the processor is doing at that moment will be interrupted.

2) Some programmers require a connection to sense the voltage level of the target processor. They do this on the 6th header pin (the top right of the 3 x 2 matrix in the picture above). On this QCU PCB, this pin is not connected. If your controller needs to sense the board voltage on this pin, then you need to wire a connection from this pin to +5V. A convenient (and nearby) place to do this is the cathode of the D1 diode (D1 is not installed in this clock kit). This connection is shown by the purple wire in the below diagram. The wire should be soldered on the underside of the PCB.

5.5 Connecting temperature sensors

Two temperature sensors can optionally be connected to the kit. These could, for example, measure indoor and outdoor temperature. Any temperature sensor which produces an analogue voltage output whose voltage varies linearly with temperature, can be used. The output voltage must always be kept within the range 0 to +5V. The examples given, use the TMP36 temperature sensor (see Operating manual).

The temperature sensor voltage inputs should be connected to the pads labelled "Band1" (Temp 1) and "Band5" (which is Temp 2), on the 5 x 2 connector at the left side of the board. Please refer to the following diagram:

6. Resources

Please see the kit pages at http://qrp-labs.com/clockkit for any information on latest updates and issues.

For troubleshooting please refer to http://www.qrp-labs.com/ultimate3/u3trouble for Ultimate3 kit troubleshooting, which in general applies also to this clock kit.

Further references are listed in the Operation manual appropriate to your firmware version.

9. Version History

0   18-Jan-2016

- First version, based initially on Ultimate3S kit assembly manual for U3S Rev2 PCB

1   04-Apr-2016

- Correction to diode connection mentioned on page 16, should be cathode (was anode)

- Clarification of the "anticlockwise" bias potentiometer photograph

2   06-Oct-2016

- Correction to circuit diagram, R4 shown as 100K should be 220K; also on page 7 R4 was referred to as 200K, should be 220K.

3   23-Oct-2016

- Simplified and clarified of the manual

4   24-Apr-2017

- Corrected mis-named component in section 4.1; it is C2 and C3 which are not supplied with the kit