

2. Design

The ATmega328 AVR microcontroller (IC1) is pre-programmed with firmware to control the LCD, buttons, and GPS interface. The design is similar to the Ultimate3 QRSS/WSPR transmitter kit. This design re-uses the same circuit board as the U3 kit. There are some surplus components in the bag that would be used in the U3 kit. You can ignore these and re-use them in your own projects!

3. Parts List

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

3.2 Capacitors

C2, 3	22pF (ceramic, marking 22, or 220)
C6, 7	10nF (ceramic, marking 103)

3.3 Semiconductors

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
Socket for IC1
20MHz quartz crystal
16-pin female header socket
16-pin male header plug
Four 12mm nylon hex PCB spacers
Eight 6mm M3 screws

4. Construction

4.1 General construction tips

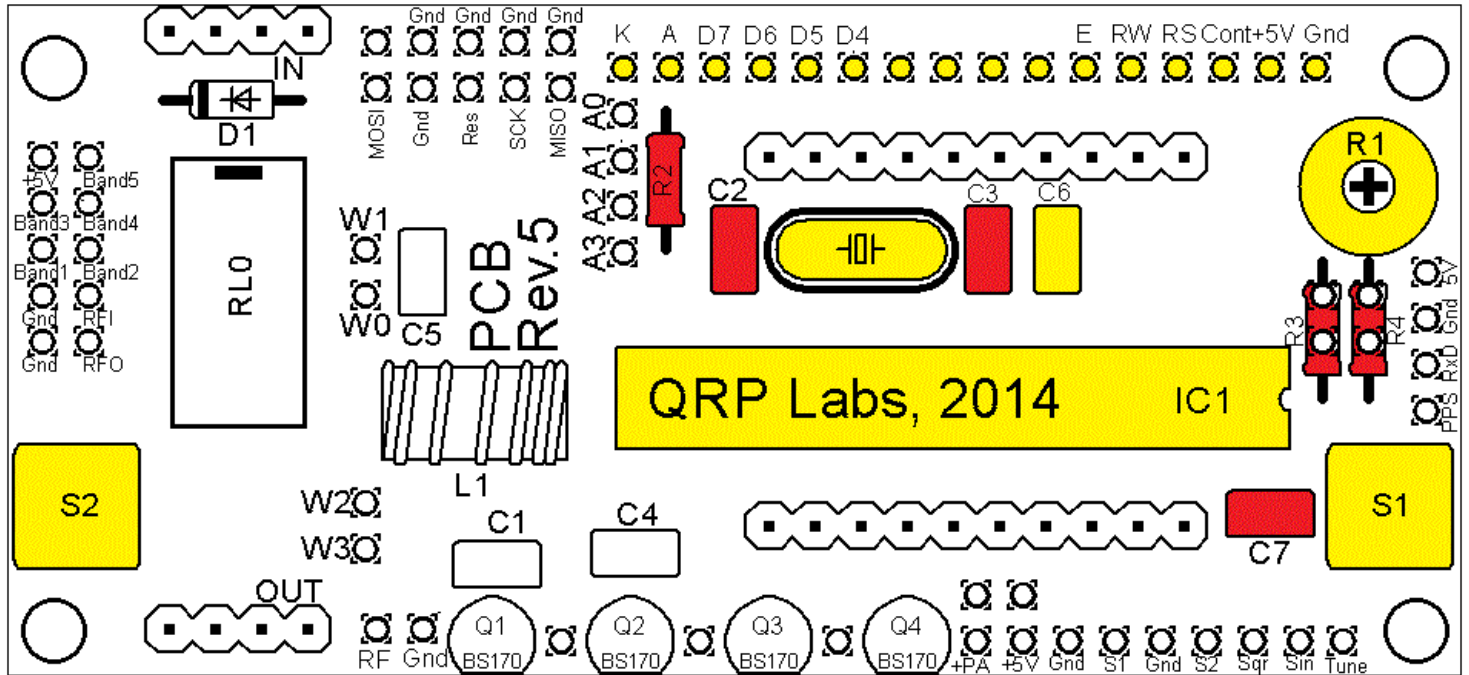
Parts placement is defined by the silkscreen printed legend on the PCB, so please observe it carefully, paying particular attention to the correct orientation of IC1.

Please note again, that a number of additional components are included in the bag. These are used in the Ultimate3 QRSS/WSPR TX Kit. They are not to be fitted to this PCB when it is built as a shack clock kit.

Note that components R2, R3 and R4 are not required or 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. Components that WILL be fitted are coloured in Yellow. OPTIONAL components are red. All the others, are not used in the clock (they are for the U3 kit).



Pay special attention to the orientation of the semiconductor 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.

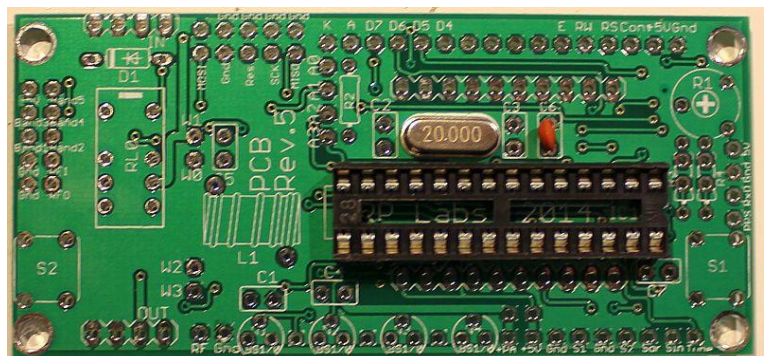
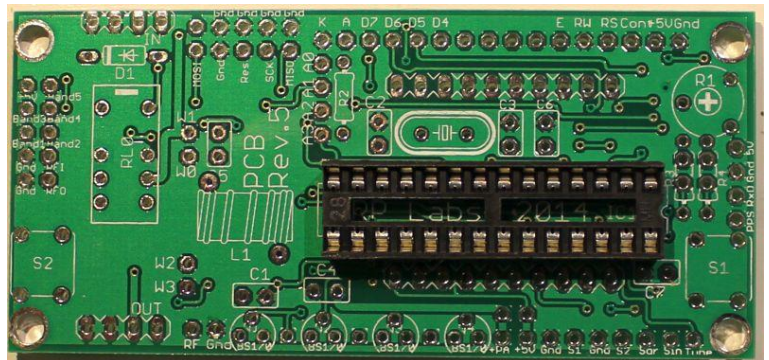
1) Solder in the socket for IC1.

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 in capacitor C6 and the quartz crystal.

Solder in the 20MHz crystal and 10nF capacitor C6. (Be careful not to mix up the 22pF and 10nF capacitors).

The optional two 22pF capacitors C2 and C3 are supplied and shown in the circuit diagram. However I recommend NOT installing them. Some crystals do not oscillate reliably with 22pF capacitors, but

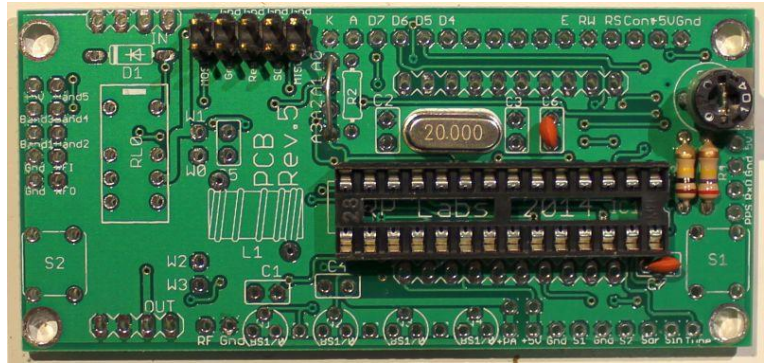


apparently they will work fine without any capacitors; the board and lead parasitic capacitances are sufficient for reliable operation. It does not matter that the oscillation frequency will not be exactly 20MHz, this is compensated in software.

3) Fit and solder R1, C7 and jumpers

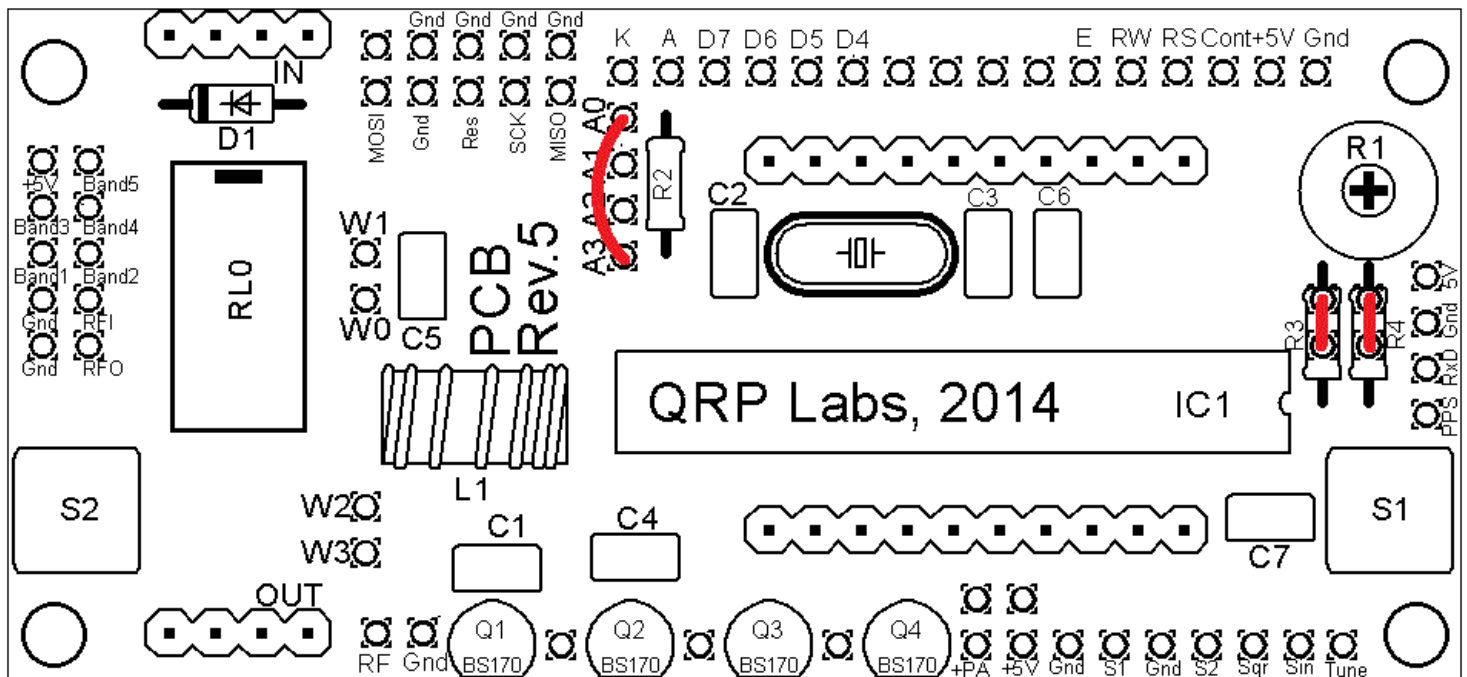
C7 only needs to be fitted only if you intend to measure analogue temperature sensors (it improves the Analogue to Digital converter noise performance).

Fit the contrast trimmer potentiometer R1 (code "104"). 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.



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 semicircular arch, perhaps 5mm high off the board, as pictured. 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.

Refer to the following diagram in the following description of optional components R3, R4 and jumpers:



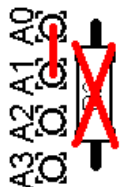
Optional resistor R3: This can be fitted to enable in-circuit-programming for those wishing to update their own firmware (and having the necessary equipment). Use a 100K resistor to pull IC1's "reset" input high, so that the in-circuit programmer can control the signal during programming.

If R3 is not fitted, you must fit a wire jumper in place of the resistor, as shown in the red line in the diagram.

Optional resistor R4: This resistor 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.

If R4 is not fitted, you must fit a wire jumper in place of the resistor, as shown in the red line in the diagram.

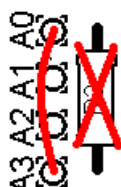
LCD backlight brightness configuration: Finally you must choose the backlight brightness method.



a) 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.



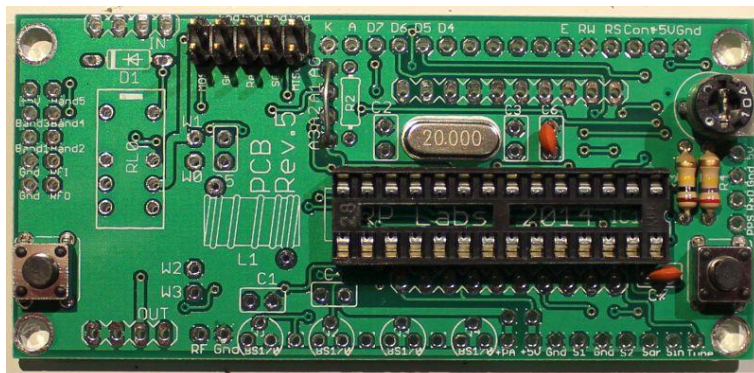
c) Software variable brightness: The Clock firmware contains a "Backlight" configuration setting which can be set from 0 (backlight OFF) to 9 (maximum brightness). This is achieved using pulse width modulation. To enable this setting, you must connect A0 to A3 using a wire jumper (do not fit any resistor at the R2 position).

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 between A0 and A1, 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.

In the photograph on the previous page, optional resistors R3 and R4 (not supplied) have been fitted; the LCD backlight configuration is set to the Software variable brightness option. At the top of the PCB you can see a 5 x 2-pin PCB header has been installed, for in-circuit programming of the AVR. This option is described in a later section 5.5 of this assembly manual.

4) Fit push button switches S1 and S2

The switches are in a convenient location to press with your left and right index fingers, when holding the module in your hands, with the LCD facing you. However, it is certainly nicer to fit off-board switches as well, on a front panel (see later section 5.3 of this assembly manual).



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!

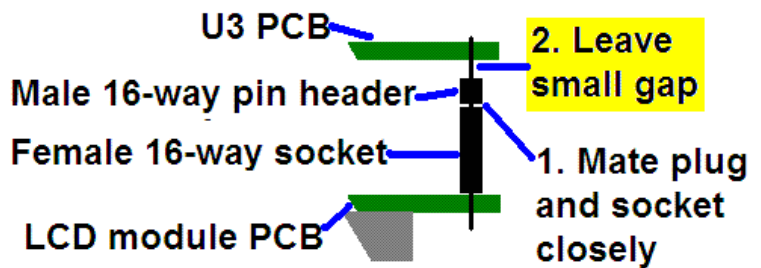
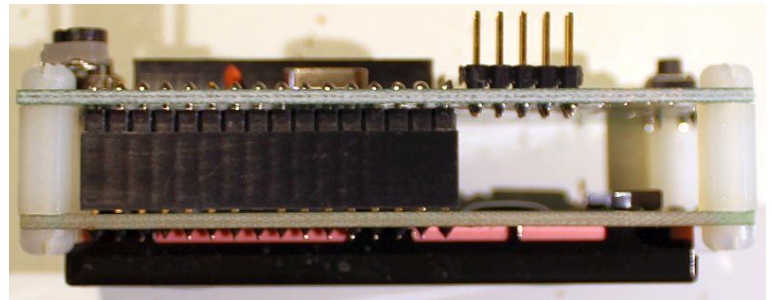
Bolt together the LCD module and the PCB, with the 16-way connectors in place but not soldered.

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.

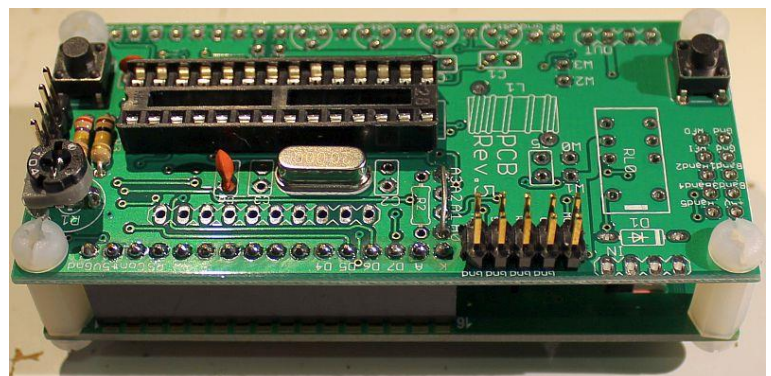
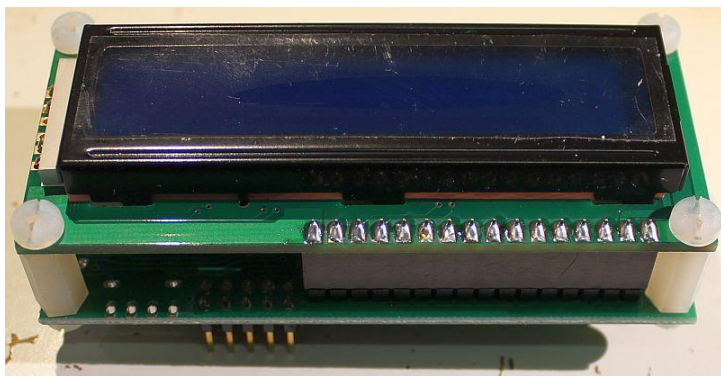
My suggested method is illustrated to the right here. Closely mate the plug and socket. Leave the gap between the male header (plug) and the PCB. I have found this method works successfully.



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.

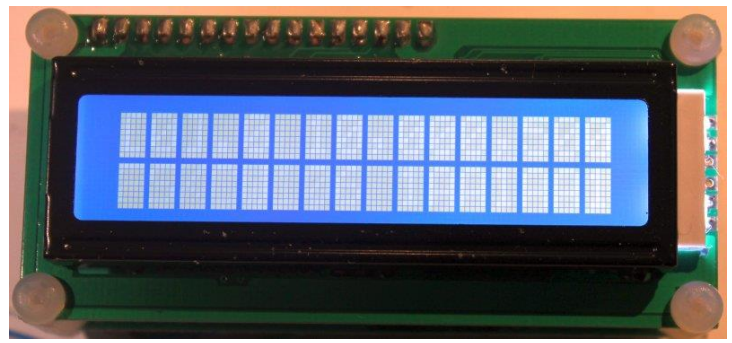
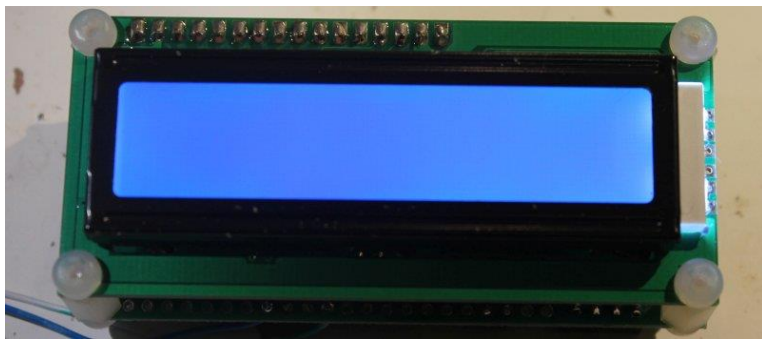
At this stage, also solder the pins of the 16-way socket to the LCD module.

Now 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.



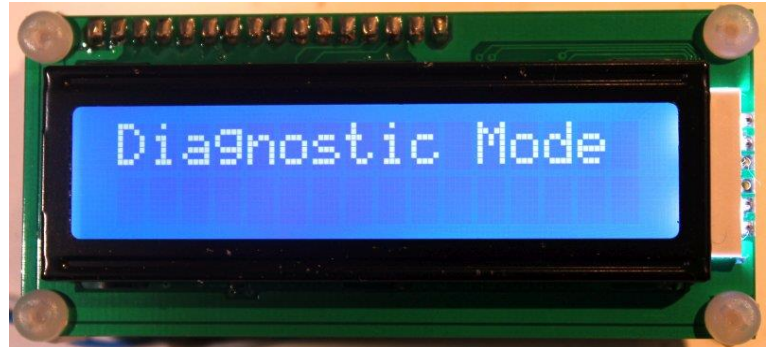
Important! Upon power-up, you will need to adjust the contrast potentiometer R1 to view the LCD properly. Turn it fully clockwise to start with (before applying power). Then turn it gradually anticlockwise until the displayed text looks correct.

The series of photographs below show the display with R1 fully clockwise (left); fully anti-clockwise (right); and 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.

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 U3S website (see resources section below).



5. Hardware Options

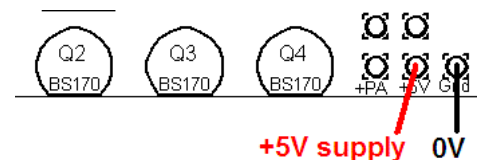
Label	Group	Description
5V	GPS	+5V supply to GPS module
Gnd	GPS	Ground connection to GPS module
RxD	GPS	Serial data input from GPS module
Pps	GPS	1 pulse per second input from GPS module
R	Future	NOT USED IN CLOCK KIT
S2	Buttons	Optional external button S2, switch to Gnd
Gnd	Buttons	Ground for optional external buttons
S1	Buttons	Optional external button S1, switch to Gnd
Clk2	Signal	NOT USED IN CLOCK KIT
Clk1	Signal	NOT USED IN CLOCK KIT
Clk0	Signal	NOT USED IN CLOCK KIT
Gnd	Power	Ground connection to Power supply
Heat	Power	NOT USED IN CLOCK KIT
+5V	Power	+5V power supply
+PA	Power	NOT USED IN CLOCK KIT
Key	Control	NOT USED IN CLOCK KIT
Gnd	RF	NOT USED IN CLOCK KIT
RF	RF	NOT USED IN CLOCK KIT

5.1 Explanation of connections

The table above details the purpose of the connection points around the right-hand and bottom edges of the PCB. The pin-spacing is 0.1-inches and a suitable connector could be used if desired. These are further described in the following sections.

5.2 Connection of power supply

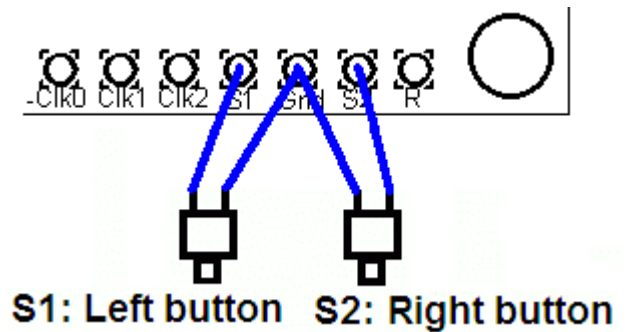
Connect a well-regulated, well-smoothed 5V power supply. It may be necessary to add an additional electrolytic smoothing capacitor e.g.



1000uF, at the +5V/0V connections to the PCB.

5.3 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. Suitable buttons are supplied in the official QRP Labs U3 Case and accessories kit.

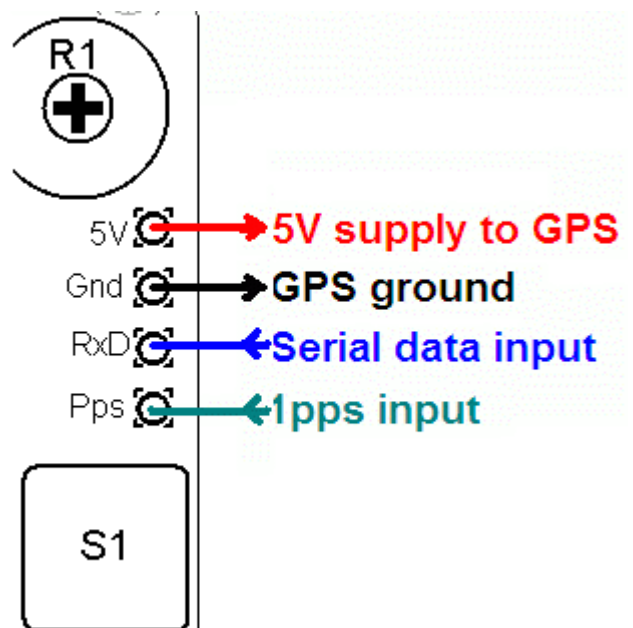


5.4 Optional connection of GPS module

A GPS module may optionally be connected to the kit, to allow the clock to display accurate time, date, latitude, longitude and other GPS information.

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. QRP Labs supply a suitable GPS module that is powered from 5V.

See operation instructions for details on how to configure the GPS interface.



5.5 In-circuit programming of AVR

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

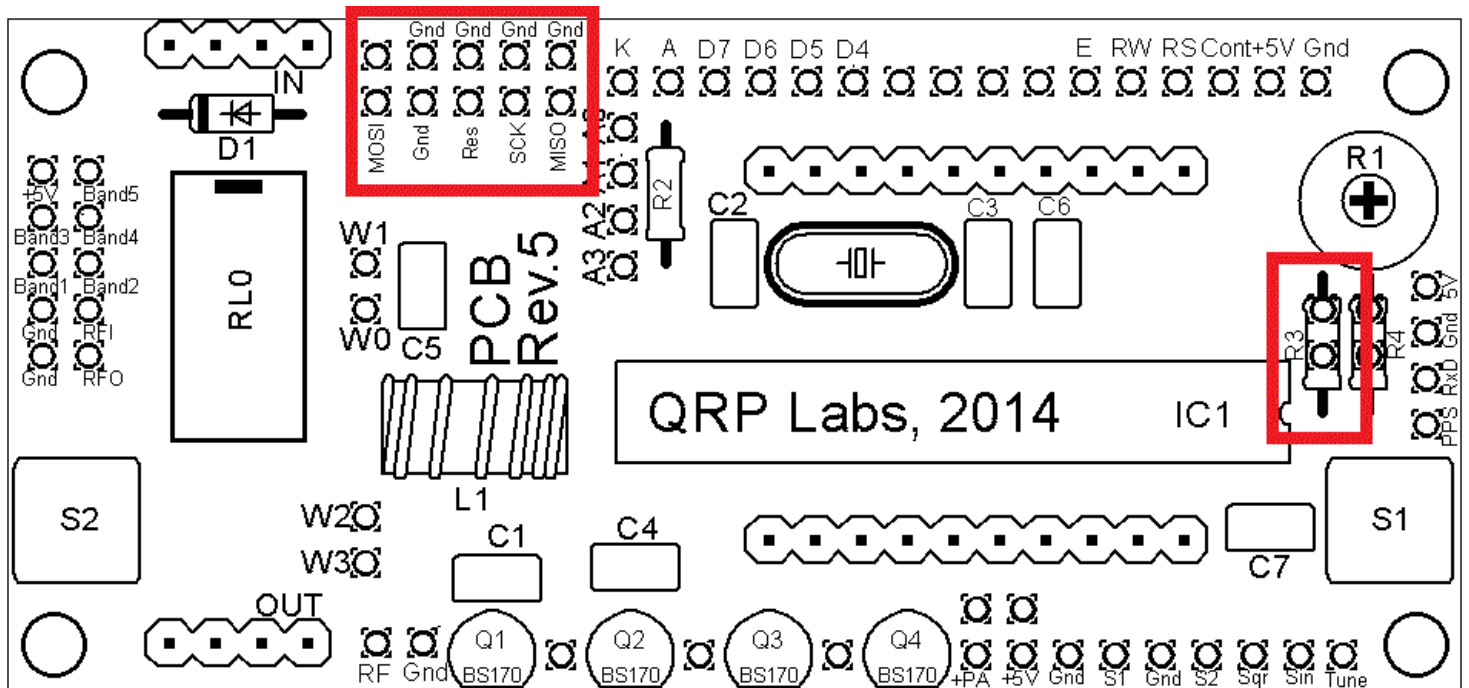
In this case, you should fit a 100K resistor as R3, not a jumper link (see step 3 of section 4.2 above). This is shown enclosed in RED on the right side of the below diagram.

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 5-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. In some programmers the Gnd connection between MOSI and Res is “NC” (no connection). These are also suitable for connecting to the Clock.

Notes:

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

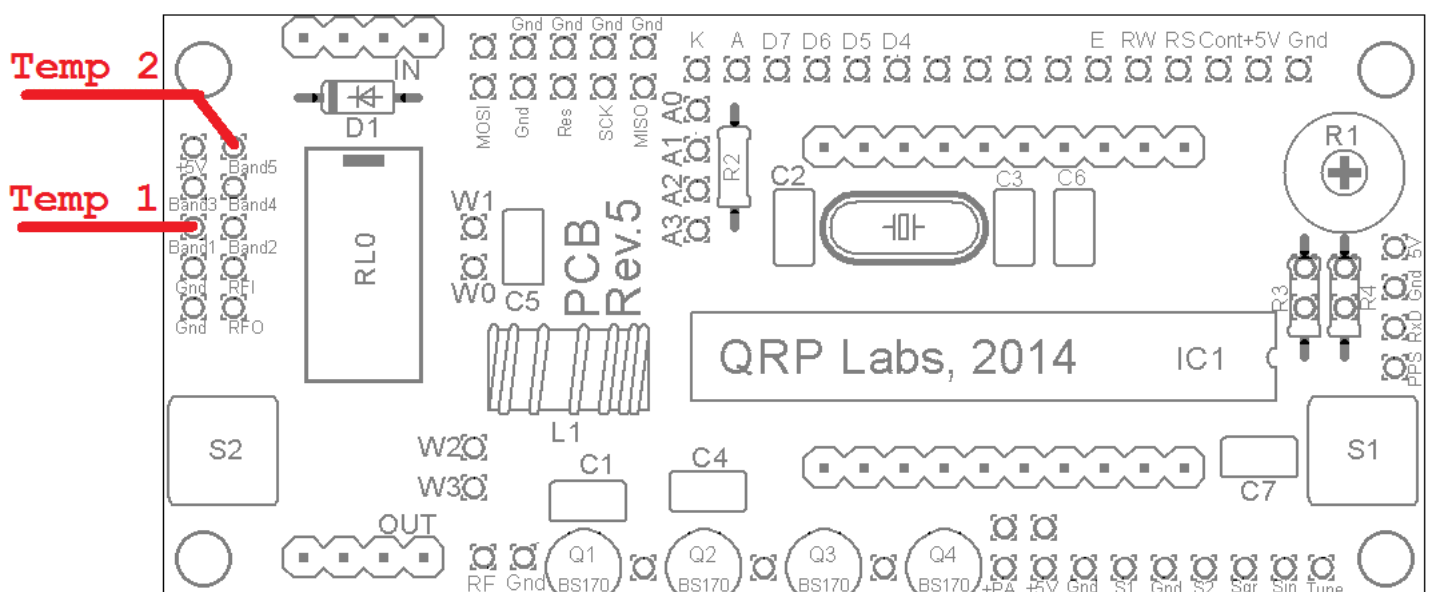
2) The physical space at the 2 x 5-pin header is limited. You may not be able to fit your 2 x 5-pin connector. Please check before trying to install it.



5.6 Connecting temperature sensors

Two temperature sensors can optionally be connected to this 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 page <http://www.hanssummers.com/clockkit> for any information on latest updates and issues.

For troubleshooting please refer to <http://www.hanssummers.com/ultimate3/u3trouble> - this is based on the Ultimate3 QRSS/WSPR TX kit but the sections on the 20MHz oscillator, connections, and Diagnostic mode apply equally to this Clock kit.

Further references are listed in the Operation manual.

7. Version History

0 **06-Apr-2015**

- First version

1 **15-Sep-2015**

- Added note about power supply filtering.