

# Ultimate3S: Multi-mode QRSS/WSPR transmitter kit

## PCB Revision: "QCU Rev 3"

### 1. Introduction

Thank you for purchasing this QRP Labs kit. You will also plug in other modules. You need to download assembly instructions for those kits also. You will need the following manuals:

- This Ultimate3S assembly manual
- The Ultimate3S operating manual appropriate to your firmware version
- Low Pass Filter kit (bands 2200m to 6m)
- Si5351A Synthesiser kit
- OCXO/Si5351A Synthesiser kit (if you will use this option)
- Relay-switched LPF kit (if you will use this option)
- QLG1 GPS receiver kit (if you will use this option)

**[Recommended approach to building the kit:](#)** This is a simple design but there are a large number of features which provide a great deal of flexibility. **Read this [WHOLE manual and understand it!](#)** Follow the construction section to build the kit.

**[This assembly manual must be read in conjunction with the operation manual!](#)**

### 2. Design

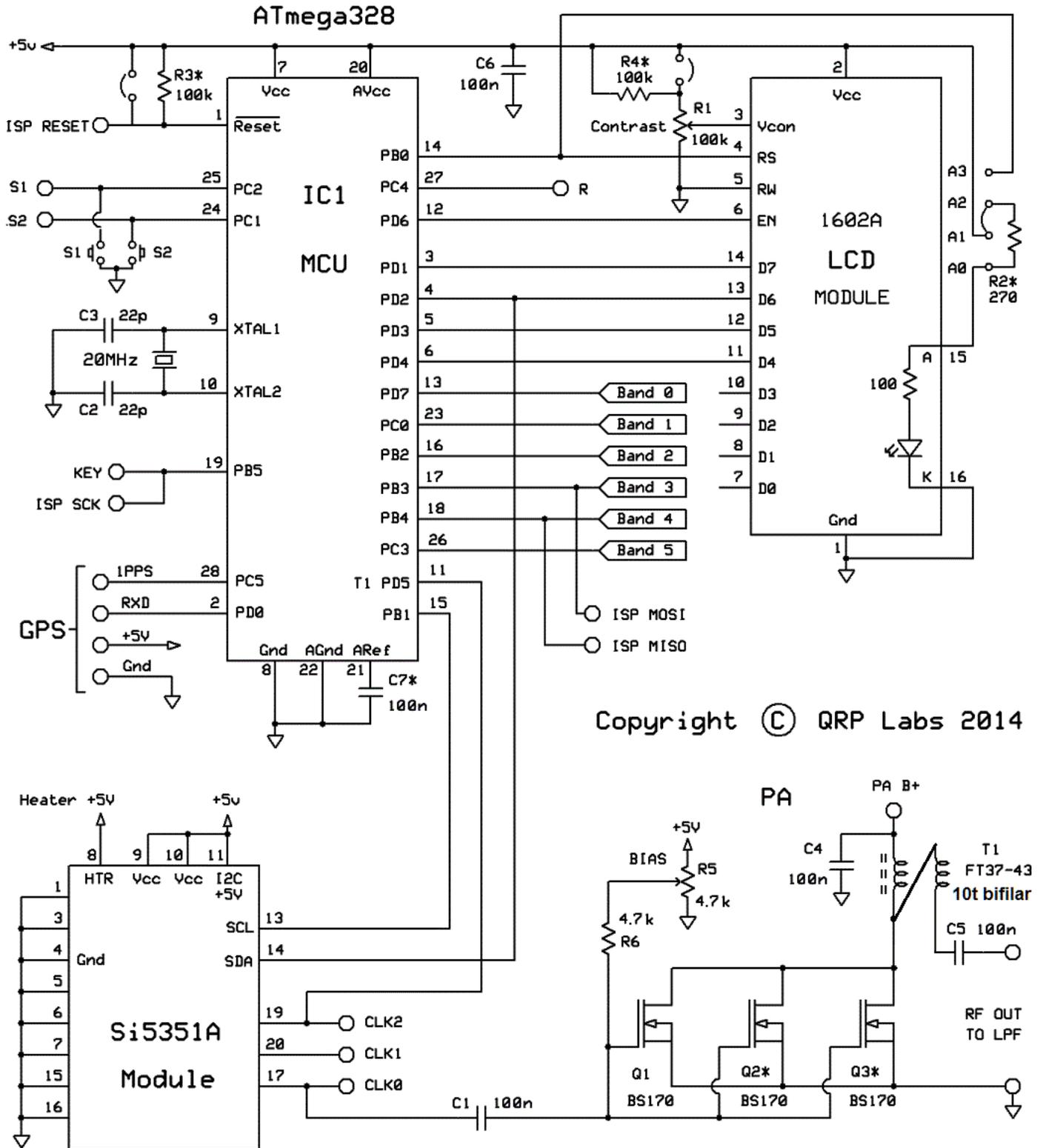
The ATmega328 AVR microcontroller (IC1) is pre-programmed with firmware to control the LCD, buttons, and GPS interface.

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.

The PCB contains the microcontroller, Power Amplifier (PA) and connections to other boards and interfaces. The LCD module, Si5351A Synth module, Low Pass Filter module, Relay-switched LPF module all plug into this main PCB. The separate Si5351A Synthesiser module kit includes its own on-board 27MHz crystal reference oscillator.

On/off keying is handled in software by switching on/off the output stage of the Si5351A synthesiser IC. The Q1 BS170 MOSFET is the power amplifier, producing about 250mW from a 5V supply on 30m (power output varies on different bands). Finally standard 7-element low pass filter plug-in modules remove harmonics of the transmission frequency. The schematic diagram on the following page shows the Ultimate3S QRSS/WSPR transmitter.

# Ultimate 3S Multimode QRSS / WSPR TX Kit



Copyright © QRP Labs 2014

(\*) optional parts not included by default in the kit.

## **3. Parts List**

In the following parts list, components which are not supplied are shown in grey. They are optional, or supplied with other optional kits such as the relay-switched LPF kit, if you are going to use it.

### **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, for setting the PA bias (code "472")
R6	4.7K resistor (colour coding yellow-purple-red-gold)

### **3.2 Capacitors**

C2, 3	No capacitor – space is provided for optional 22pF capacitors if required
C1,4,5,6,8	100nF (ceramic, marking 104)
C7	No capacitor – optional, reduces noise when using analogue to digital conversion

### **3.3 Inductors**

T1	10 bifilar turns, FT37-43 core (black)
----	--

### **3.4 Semiconductors**

Q1, 2, 3	BS170 (note: only one BS170 is provided in the kit, to be installed at the Q1 position. For use with higher PA supply voltages, more transistors may be added, i.e. Q2 and Q3).
D1	No diode - provided for use with relay-switched LPF board kit
IC1	Pre-programmed ATmega328 microcontroller

### **3.5 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
Socket for IC1
20MHz quartz crystal
Two 10-pin female header sockets
Two 4-pin female header sockets
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, provided for use with relay-switched LPF board 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.

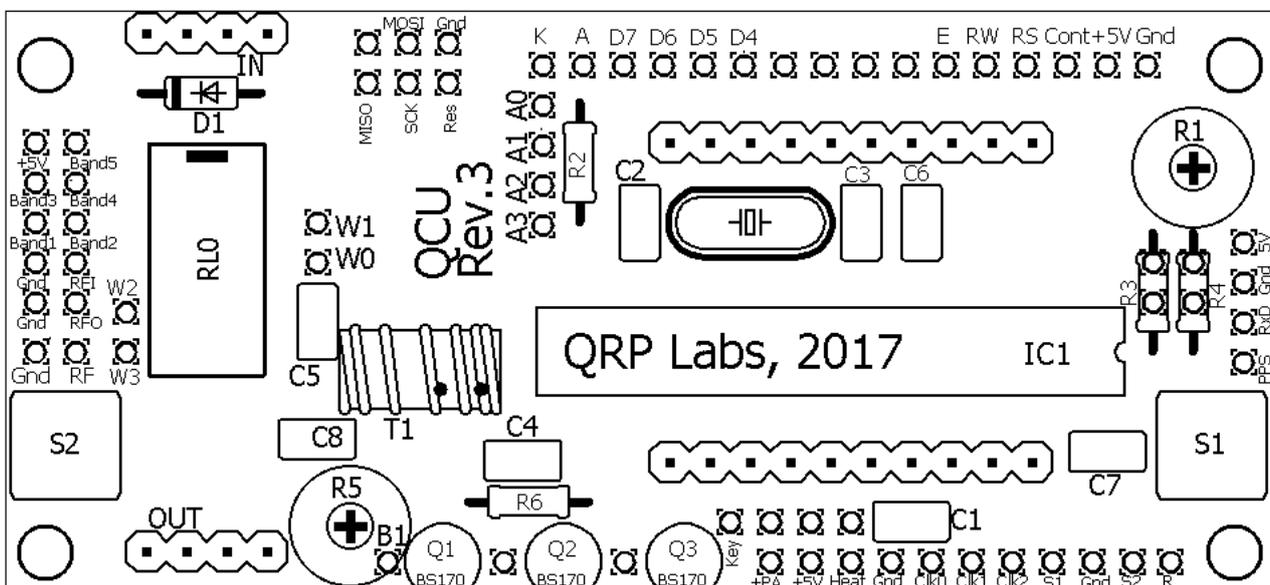
The PCB is quite small and the parts are close together. You are recommended to use a low wattage iron with a fine tip, and fine solder e.g. 1mm diameter or less. Take care not to overheat the PCB and risk damaging it. A well-lit area and magnifying glass are useful. Be careful not to bridge solder across 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 D1, Q2, Q3, R2, R3, R4, C2, C3 and C7 are not required or supplied in the kit (R2 may be added by the builder if lower LCD backlight brightness is desired). Q2 and Q3 are not supplied, additional BS170 transistors may be installed here to increase the output power (see below).

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. Sockets are provided for the synthesiser module and the low-pass-filter module, if applicable to your kit.

### 4.2 Construction steps

Please refer to the parts placement diagram below.

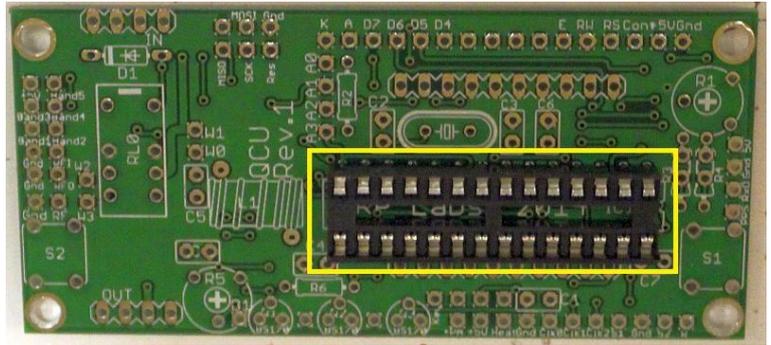


Pay special attention to the orientation of the semiconductors. For IC1, the dimple in the PCB silkscreen must be aligned with the dimple at the top of the IC socket and the IC.

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.

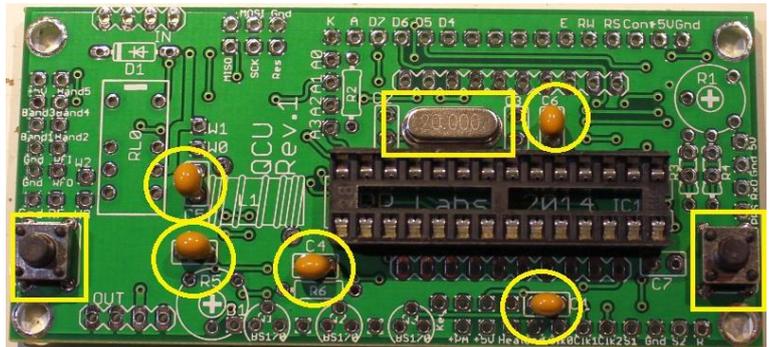
To avoid confusion or mistakes later, align the dimple at one end of the socket, with the dimple illustrated on the PCB. The dimple should be at the end nearest the right-hand edge of the PCB.



### 2) Solder all capacitors, quartz crystal, and buttons S1/S2

There are five 100nF (0.1uF) capacitors. These are fitted in locations C1, C4, C5, C6, C8.

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) Wind and install the toroid

There are two options here.

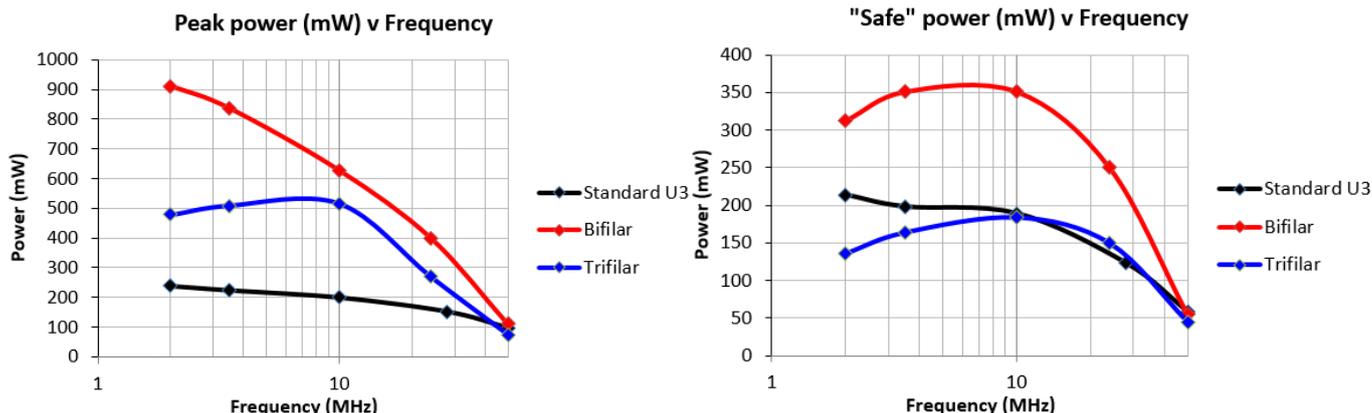
- 25 turns inductor. This option was used on all Ultimate, Ultimate2, Ultimate3 and Ultimate3S kits prior to PCB revision “QCU Rev 3”. If you wish, you can also use this option on the Rev 3 board.
- 10 turns bifilar transformer. This option is supported from PCB revision “QCU Rev 3” onwards.

The bifilar transformer option provides higher power output but is a little trickier to wind. There is an article on using a bifilar and trifilar transformer in the U3S PA section here: <http://grp-labs.com/ultimate3/u3info/u3sbifilar.html>

From the article, the charts below show the power output using the 25-turn single-winding toroid that was then standard in the U3 (BLACK line), the power output using a 10-turn bifilar transformer (RED line) and using a 10-turn trifilar transformer (BLUE line).

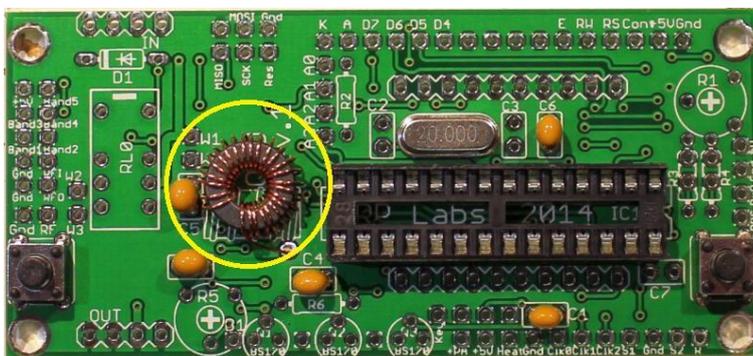
The left graph (Peak power) shows the power output when the bias potentiometer trimmer is adjusted for maximum power output; the right graph (“Safe” power) shows the power output when the bias potentiometer is adjusted more conservatively using the “zero idle current” method (more

on this topic later). In both cases, the power output is considerably higher using the bifilar transformer option.



**The bifilar option is recommended. Note, however that if you are planning to use the 5W HF PA kit with your Ultimate3S – then you should use the 25 turns single winding option, which works better with the 5W HF PA kit.**

The silkscreen printing on the PCB shows vertical mounting of the toroid. If you prefer horizontal mounting, there is enough space for it. **Please mount horizontally if you intend to use the OCXO/Si5351A Synth kit!** When winding the toroid, remember that each time the wire goes through the centre of the toroid counts as one turn. 35cm of wire should be enough for 25 turns, or for the 10 turns bifilar option.

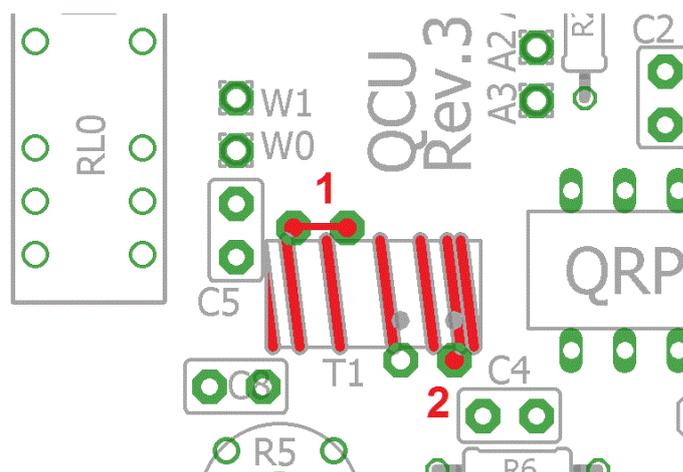


### **25-turn single-winding option**

Wind 25 turns on the toroid. Each time the wire passes through the hole counts as ONE turn.

Install the toroid on the PCB as shown in RED in the diagram. The two ends of the wire are connected to the holes marked 1 & 2.

You must connect a bridge wire across both holes at the “1” end as shown. At the “2” end be sure to put the wire in the right-side hole next to the C4 capacitor. Leave the left of the two holes empty!

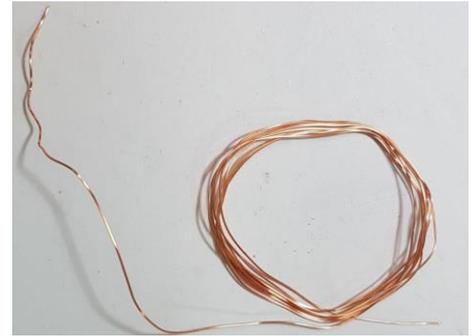


Trim the ends of the wire, scrape the enamel off and tin them with solder. As an alternative to scraping the enamel off, the wire ends may be held in a hot blob of solder on the iron tip for a few seconds, and the enamel will bubble away. Check continuity on the board with a DVM.

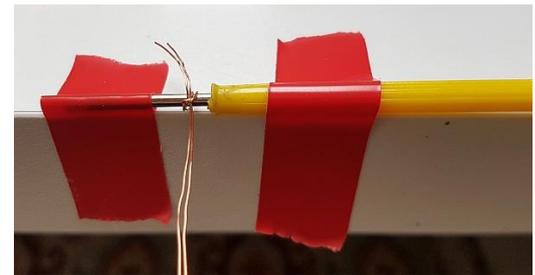
## 10-turn bifilar-winding option

The 10-turn bifilar transformer can be tricky if you haven't wound a bifilar transformer before. It needs care so please follow these instructions very carefully.

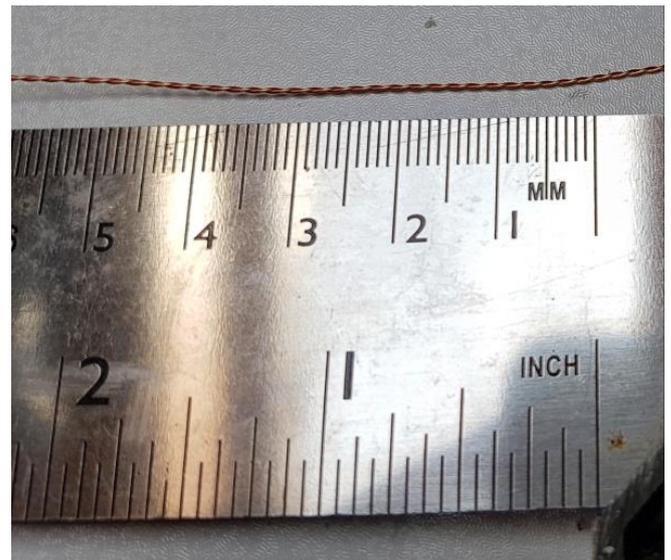
Firstly, the wire. The best way to un-wind it, without tangling it up, is to think of what the kit-packing person that wound it up did. Then reverse his steps. So, first unwind the tightly wrapped part in the middle where the end of the wire has been secured. Then, open out the spool of wire so that it is a circle. Then unwind the spool, around your fingers, reversing the process of winding it in the first place.



When you have unwound the wire and straightened it, then fold it in half (do not cut it yet). The two halves now need to be tightly twisted together to make the bifilar wire. My method for this is to tie one end in a knot around a small screwdriver shaft. Now clamp it to something solid. You could use a vice, if you have one. If you don't, then you have to get creative and think of something. Here I taped it to the edge of the desk. At the other end of the wire you have a loop (where you folded it in half). Put another screwdriver through this loop. Now you can twist the screwdriver at the free end (with the loop), repeatedly turning until you have twisted the two wires together thoroughly. You need to keep the wire under a little tension to keep the twists evenly spaced.

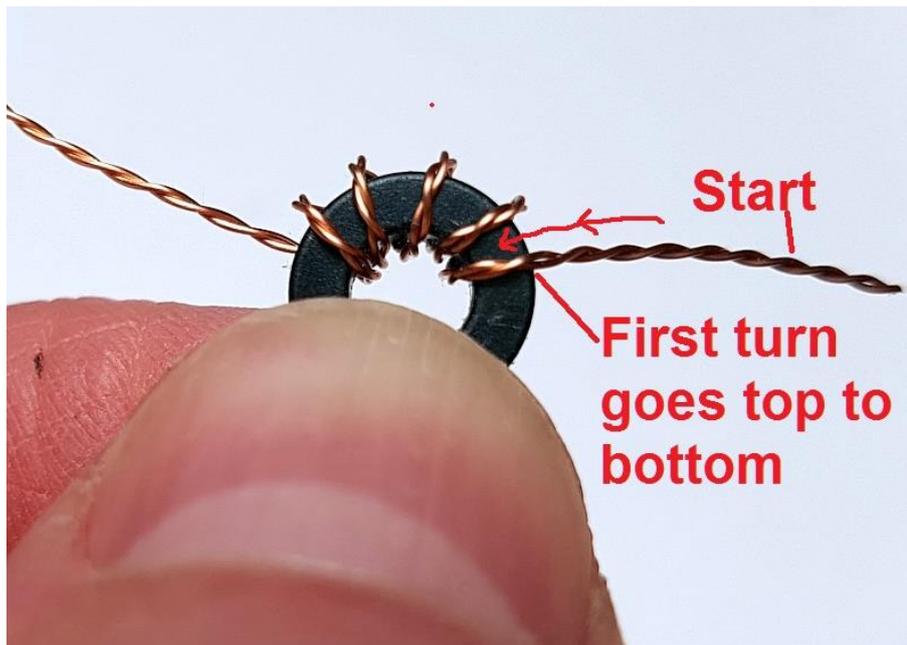
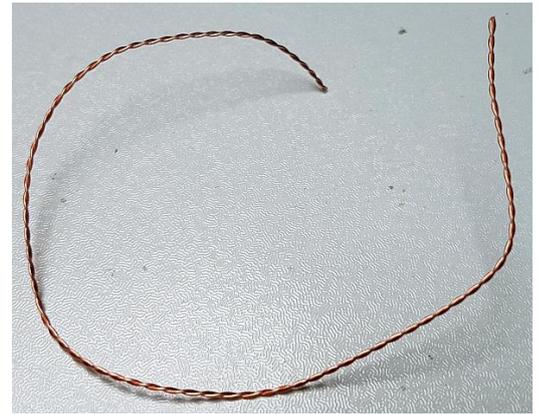


I put about 40-50 twists into the 15cm length of wire. The end result is something like in these photos.



Now cut off the untidy pieces at one end, and the loop from the other: and this is the piece of wire that will be used to wind the FT37-43 toroidal core as a bifilar transformer.

Hold the core between thumb and finger. Pass the wire first from above, to below. Then take the wire from below, and bring it around to pass through the toroid again to form the second turn. After each turn, ensure the wire is fitting snugly around the toroidal core. Wind 10 turns on the core. Each time through the toroid's central hole counts as one turn. Cut off the excess wire, leaving about 2.5cm remaining.

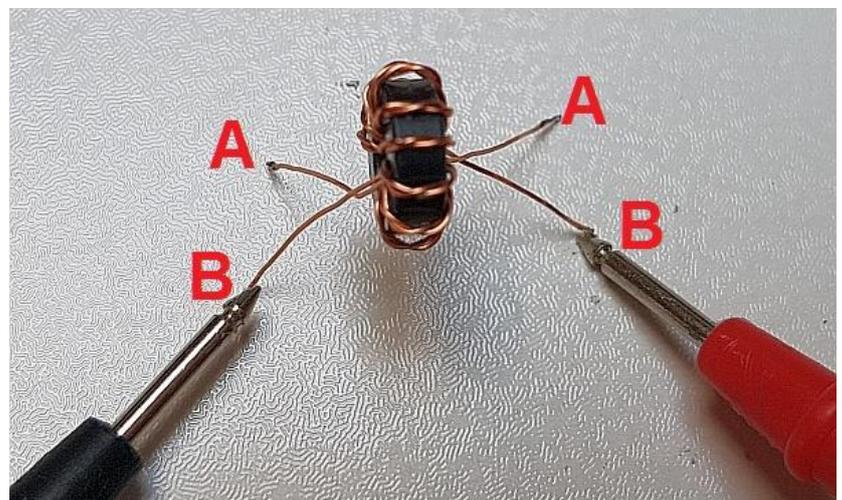


Now it's necessary to identify which wire belongs to which winding. The only way to do this is with a DVM as continuity tester. First, untwist and straighten the wire ends that are not wound around the toroidal core (above right).

Now tin the last few mm at the ends of each wire. You can do this by scraping off the enamel then tinning with the soldering iron; or, hold the wire end in a blob of molten solder for a few (maybe 10) seconds, until the enamel burns off.

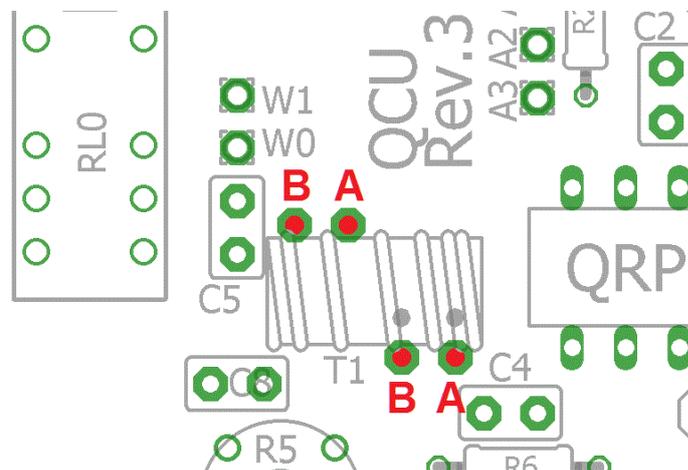
Now use a DVM to test for continuity. Re-arrange the wires so that there is continuity from A to A, and B to B.

Carefully keep this orientation of wires



and insert the transformer this way into the PCB. You can carefully cut off those few mm of tinned section of wire, if it won't fit through the PCB holes. But BE CAREFUL not to lose the orientation of the wires! The correct wires must be in the correct holes, so that the windings are connected correctly in the circuit!

If you are planning have the toroid lay horizontally on the board, make sure you put it in the intended position now, before cutting/soldering the wires.

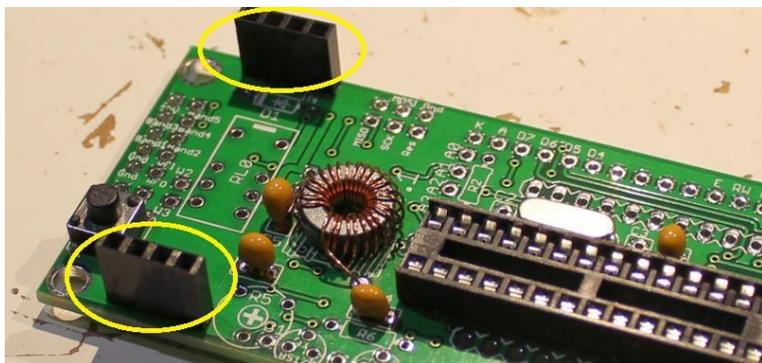


Now trim the wires underneath the board, and tin the ends again. The EASIEST way to do this is simply apply solder so that it sits in the hole and surrounds the 1mm of wire ending, and hold the soldering iron in that position for a few seconds until the enamel is burnt away.

Finally you can check that the wires are properly soldered, and properly orientated, by checking with a DVM for continuity between all four pads labelled A, A, B, B on the diagram (above right). All four toroid wire pads should have continuity to the “PA” pin at the board edge. If you do not see this continuity, then either the windings are not soldered in the correct holes, or the enamel is not completely scraped/burnt off the wire so there is no electrical connection.

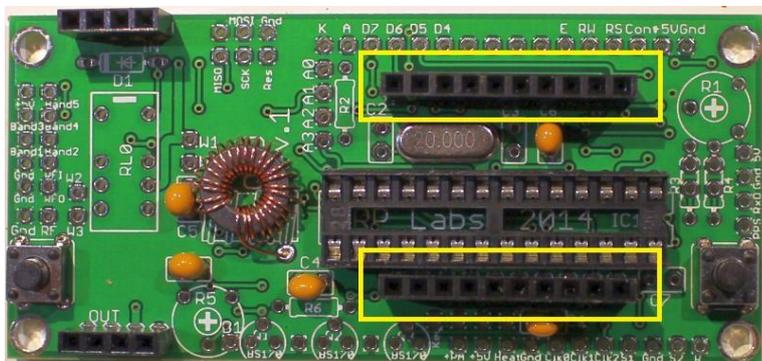
#### 4) Solder the sockets for the low-pass-filter (LPF) module.

These are the two 4-way sockets. Some care needs to be taken with the alignment, to ensure that there is a good fit when the plug-in board is added. One method is to build the plug-in LPF module first; then plug it into the sockets. You can then solder the pins, ensuring correct alignment.



#### 5) Solder the sockets for the Si5351A synthesiser module

These are the two 10-way sockets. Again take care of the alignment of the sockets, to ensure that there is a good fit when the plug-in board is added.

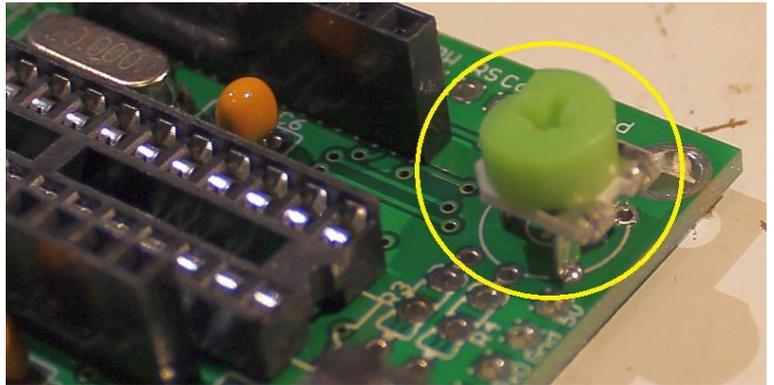


6) **Fit and solder R1, the preset potentiometer that sets the LCD contrast.**

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

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

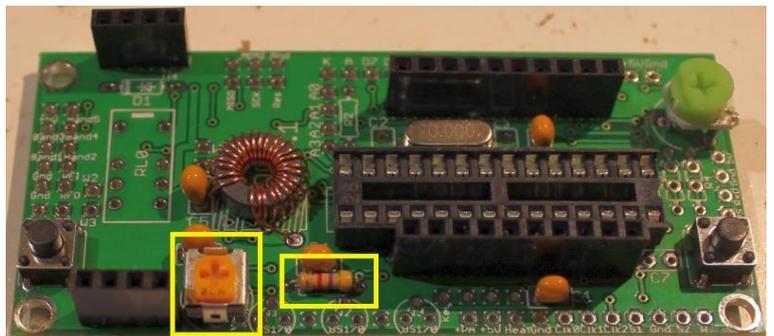
Adjust this potentiometer to the fully clockwise position initially.



7) **Fit and solder R5, the preset potentiometer that sets the PA bias.**

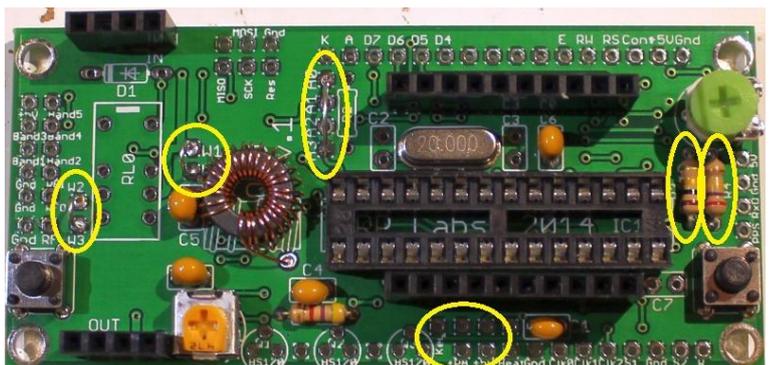
Also fit and solder the fixed 4.7K resistor R6. Be careful not to mix up R1 and R5!

**IMPORTANT: Turn the trimmer potentiometer all the way ANTI-clockwise, BEFORE applying power to the board. Please see section "PA bias set-up" below. The photograph (right) shows R5 in the mid position. In the correct completely ANTI-clockwise position, the "472" text will be to the right side, next to the Q1 position.**



8) **Solder wire jumpers as shown in the photograph and diagram below, depending on your requirements.**

Typical jumper connections are shown in the photograph circled in yellow, and by red lines on the parts placement diagram below.



**Please read the "Hardware options" sections below, to decide which jumpers to fit.**

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 unsoldering them.

The W0-W1 and W2-W3 jumpers hard-wire the LPF into the circuit. Note that when a relay is fitted at RL0, you do NOT connect these two jumper wires. The relay is part of another kit, the relay-switched LPF board, which expands the capability of the Ultimate3S kit to sequence through up to 6 different bands.

If R2 is installed, in order to reduce LCD brightness, then connect A1-A2 instead of A0-A1 (see "hardware options" section below). You may decide to connect A0-A3 to use the software-implemented brightness control.

You may also install a jumper at the +PA and +5V connections at the bottom of the picture below, in order to power the power amplifier (PA) with the 5V supply. You should leave out this jumper if you intend to run the PA at a higher voltage. See more details in the "Hardware options" section below.

The "Heat" connection should be connected to +5V also, if you are using the OCXO version of the Si5351A module. It provides +5V to the oven heater, using a thick track on the PCB. This avoids voltage drop on narrow traces. If you are using the non-OCXO version of the Si5351A synthesiser module, then you do not need to connect the "Heat" connection.

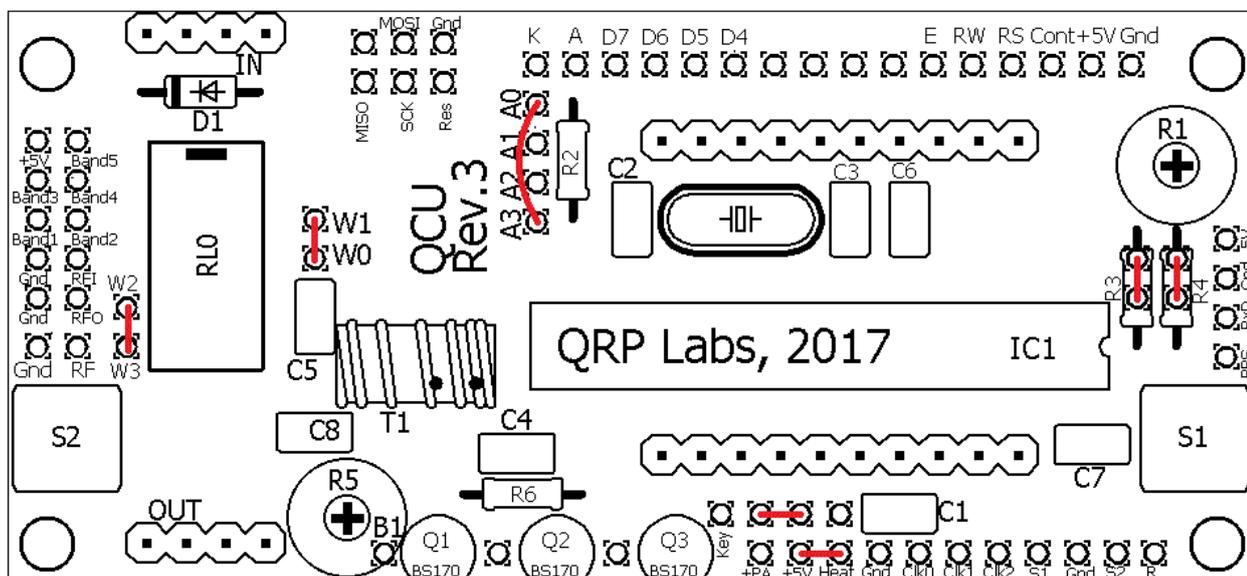
Also pay attention to the jumpers at R3 and R4 on the right-hand side of the diagram. R3 is an optional resistor, to be fitted to enable in-circuit-programming for those wishing to update their own firmware and having the necessary equipment.

**If you do not fit a 100K resistor for R3, then you must fit a wire jumper here as shown.**

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.

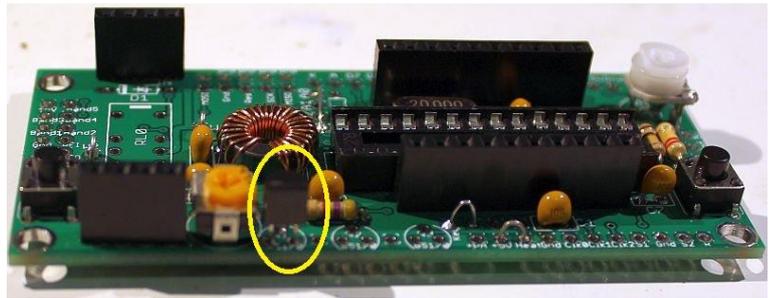
**If you do not fit a 220K resistor for R4, then you must fit a wire jumper here as shown.**

An example of typical jumper wire installations for basic operation of a single-band Ultimate3S QRSS/WSPR kit is shown in the following diagram.



### 9) Solder in transistor Q1.

If you have purchased additional transistors for Q2 and Q3 to increase the power output, I still recommend starting with just the basic, single transistor, 5V PA supply setup. Get things working basically first, then later install the additional transistors. Walk before you try to run!



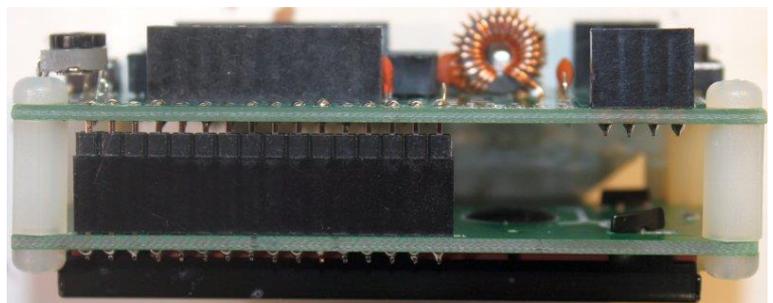
The transistors are located at the edge of the PCB with their flat side outwards. This is to allow easy attachment of a heatsink if desired (for higher voltage and hence higher power operation).

### 10) 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 U3S 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.

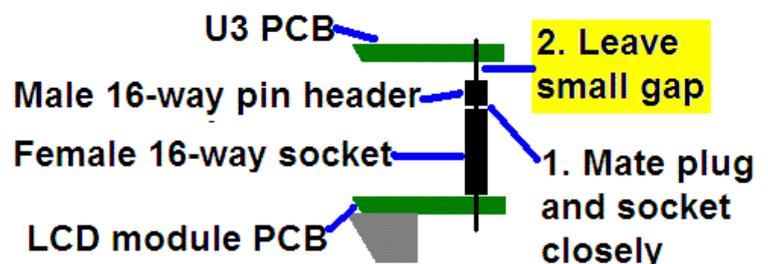


The combined 16-way plug and socket when mated together, have a height of a little under 12mm. Therefore it is necessary to have a gap somewhere.

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.

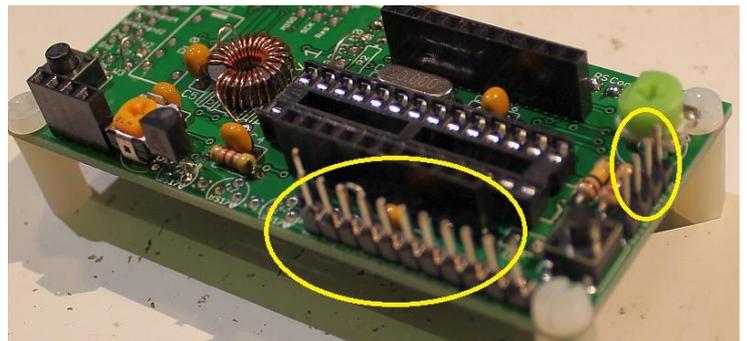


At this stage, also solder the pins of the 16-way socket to the LCD module.

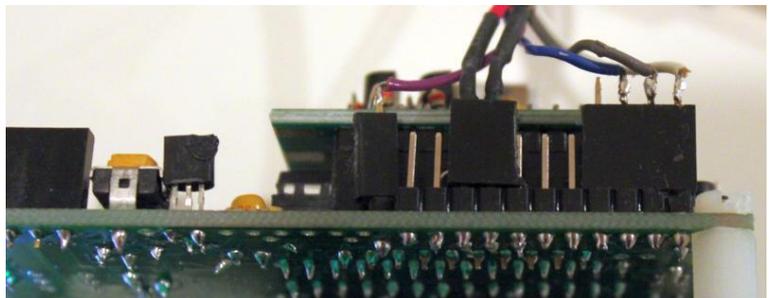


### 11) Optional header pins

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.

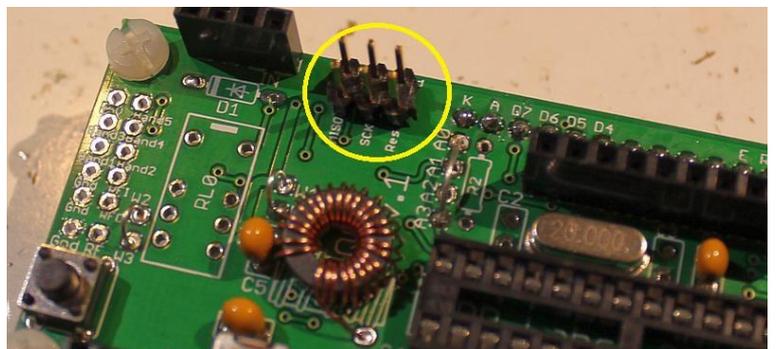


The photograph (right) shows an example of making connections to this header.

### 12) Optional In-circuit programming header

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

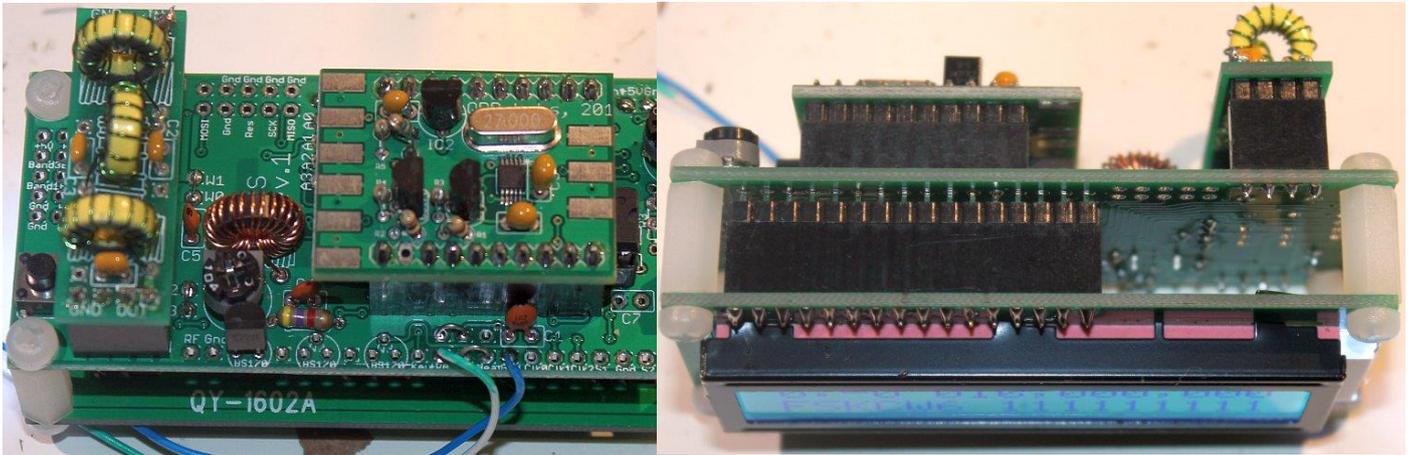
This option is useful if you want to use one of the many inexpensive AVR programmers to upgrade the kit firmware, when updates are published by QRP Labs.



### 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.

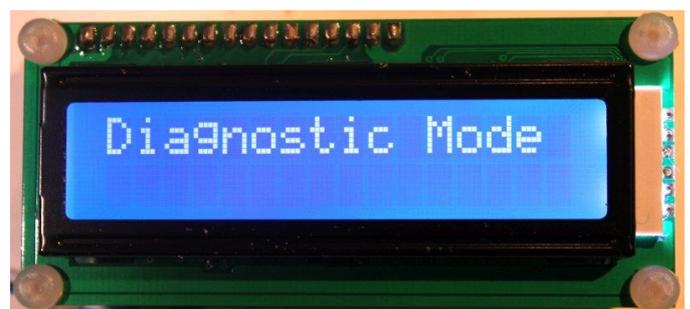
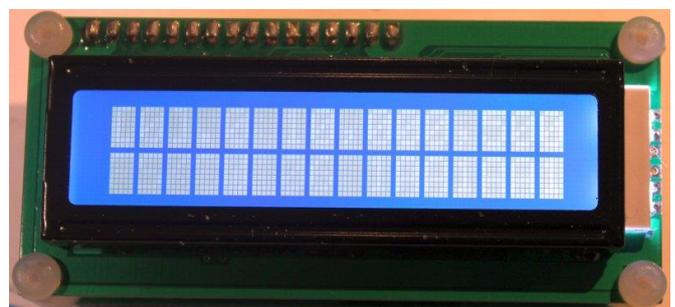
Next, plug together in the other modules (LPF, Si5351A Synth). **These pictures show the U3S PCB Revision 1, but the module assembly is the same.**



Ensure that the Si5351A synthesiser module is inserted the correct way round as shown in the photograph. The Low Pass Filter kit module should be plugged in with the “Out” legend aligned next to the RF Output connector of the main PCB (see photo).

**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.

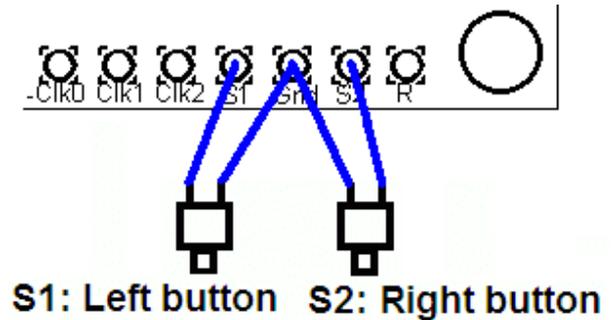


When you see "Diagnostic Mode" it means all is well with the processor and the LCD communication. When you press the left button, diagnostic mode will be cleared. Then you should



### 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

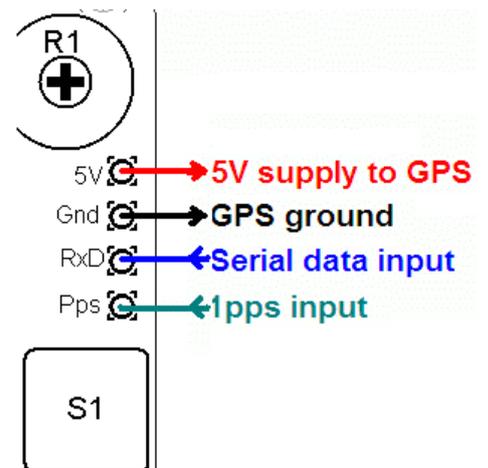
A GPS module may be connected to the kit, to provide frequency stability, accurate time, and latitude and longitude which can be converted to Maidenhead locator format for encoding in the WSPR message transmission.

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. Some GPS modules may need a pull-up resistor however.

The QRP Labs QLG1 GPS receiver module kit is powered from 5V and does not require pull-up resistors. It is suitable for direct connection to the Ultimate3S kit.

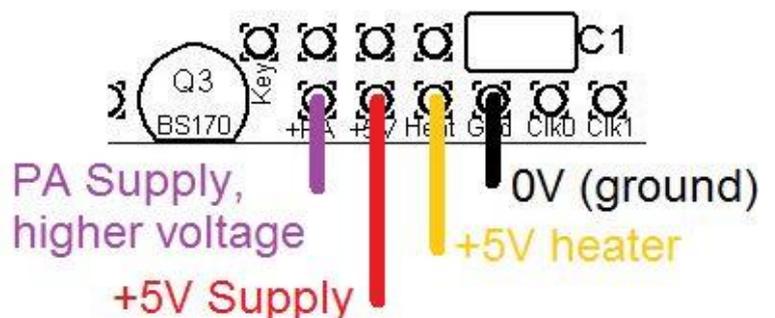
In cases where the location information is not required (modes other than WSPR) or you wish to enter it manually, the serial data input can be left unconnected. The 1 pps input is enough for the frequency lock function. A 1 pps signal could be supplied from a non-GPS source, such as a laboratory frequency standard. Note that the frequency locking function can only operate if you have selected a wait period between message transmission (i.e. a non-zero "Frame" parameter).

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



### 5.5 Higher power output

The PA in the kit provides around 250mW of output power using a 5V supply and a single BS170 transistor (depending on the band). The transistor gets warm to the touch. If higher output powers are desired, this can be achieved by using a higher supply voltage for the Power Amplifier (PA). The microcontroller must still be run at a voltage not exceeding its 5.5V rating



(5V is recommended). To facilitate running the PA at higher voltages to provide more power, the PA supply connection is a separate pin.

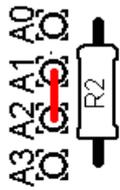
At higher supply voltages and power outputs, the heat dissipation in the BS170 will increase and its temperature will be higher. Provision is made on the PCB for two more transistors in parallel with the first. This will slightly increase the power output even at 5V supply, but more importantly at higher voltages it should share the heat dissipation between the devices.

It may be necessary to fix a heat sink to the transistors if the temperature rise is excessive. The transistors are located near the edge to the board to allow a heatsink. Experiment is needed in this area, with PA voltages up to perhaps 12V.

### 5.6 LCD backlight options



**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 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).

**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.

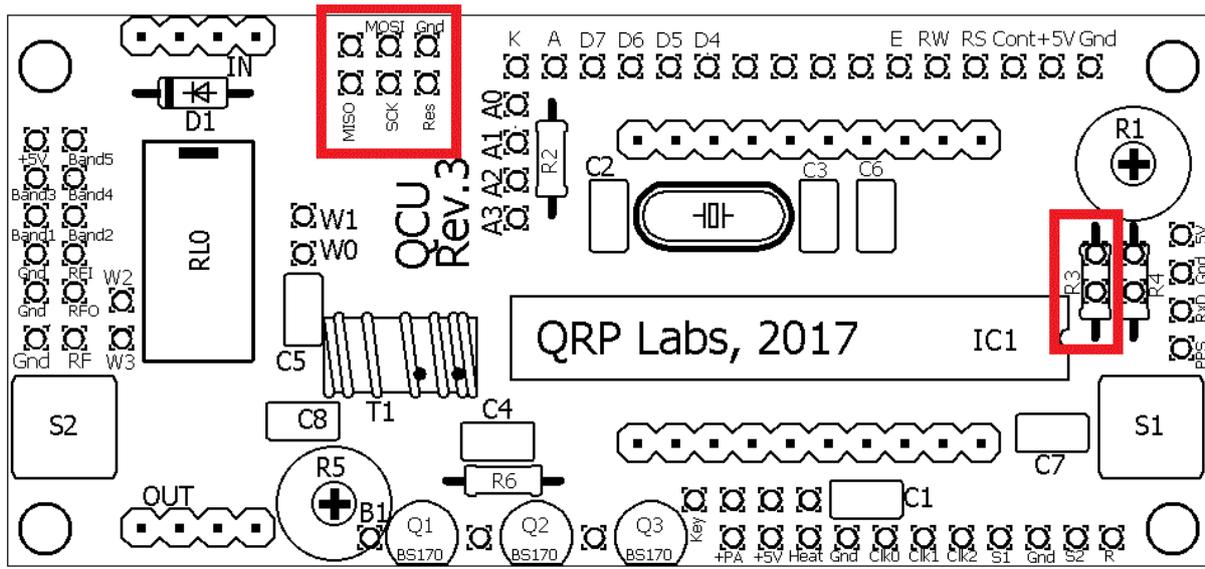
### 5.7 Si5351A outputs

For possible use in other projects, the connection points labelled Clk0, Clk1 and Clk2 provide direct connection to the Si5351A module's outputs. They are 50-ohm impedance and provide up to a 3.3V peak-peak squarewave. Actual peak-peak amplitude may be less than this (worst case 2.1V p-p), particularly at higher frequencies – please refer to the Si5351A datasheet. Do not draw excessive loads from these connections.

## 5.8 In-circuit programming of AVR

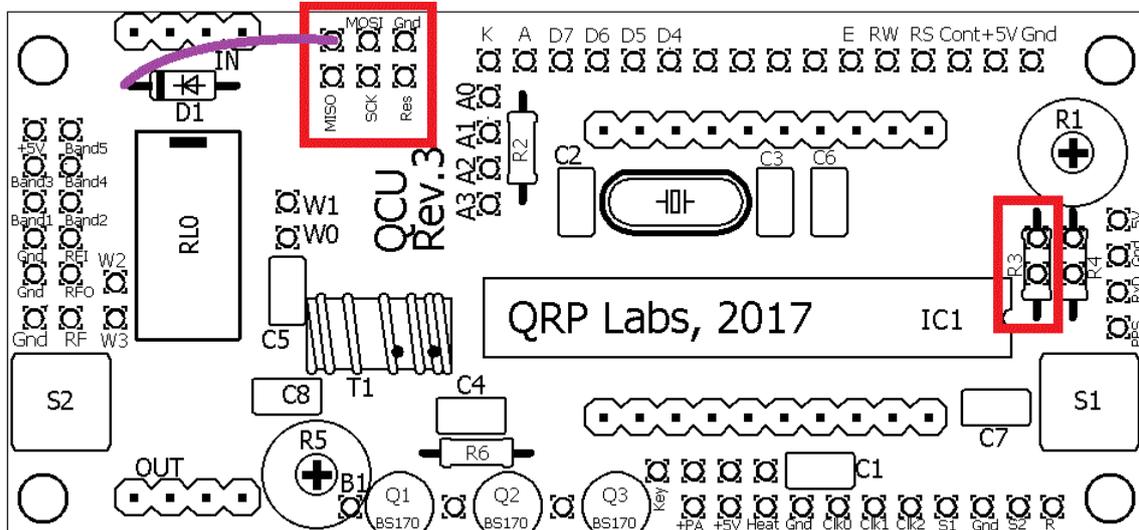
If you own an AVR programmer, you can make connections to the QCU PCB to allow updating the firmware without removing the AVR chip.

In this case, you should fit a 100K resistor as R3, not a jumper link. 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 kit during programming! When you start programming, whatever the processor is doing at that moment will be interrupted.
- 2) Some constructors have reported that if the relay-switched LPF board is fitted, the relays chatter because the programming lines are shared with relays 3 and 4, and programming errors. If this happens with YOUR programmer, you need to either remove the relay board during programming, or cut the wires to relays 3 and 4 and put a DPST switch in, to disconnect them when required.
- 3) Some programmers require a connection to sense the voltage level of the target processor. They do this on the 6<sup>th</sup> 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. This connection is shown by the purple wire in the below diagram. The wire should be soldered on the underside of the PCB.



## 5.9 Higher sensitivity contrast potentiometer adjustment

The contrast voltage required by the LCD is typically under one volt, but the potentiometer R1 is connected between +5V and Gnd. Therefore most of the available travel of the potentiometer adjustment is redundant, and the contrast adjustment is rather sensitive. If you wish to improve this, you may fit a resistor at R4 instead of the wire jumper. A suitable resistor value is 220K or 270K, for example.

## 6. Calibration

It is very important to realise that by far the hardest part of any successful QRSS beacon operation, is tuning the oscillator to the correct frequency. For example, most 30m QRSS stations monitor a narrow 100Hz-wide band from 10,140,000 to 10,140,100. The WSPR sub-bands are 200Hz wide. If you are much outside this, the chances are that nobody will see your signal. It is therefore essential either to adjust your output frequency using an accurately calibrated frequency counter, or an accurately calibrated receiver.

If you are going to use a GPS receiver module to provide a 1pps signal to the kit, no calibration is required: the kit will self-calibrate from the 1pps signal.

In the case that you will not be using a GPS signal, you need to calibrate the kit by entering the correct value of the 27MHz reference oscillator frequency into the "Ref. Frq." configuration setting.

It is possible that you have an accurate means to measure the 27MHz reference oscillator frequency itself by probing the correct point on the Si5351A module. However most kit builders will not have the necessary equipment, and such a measurement is not easy to make without itself risking disturbing the frequency.

The easiest method is to set the output frequency to something convenient such as 13.500000 MHz, and measure it. Measurement can either be by using an accurately calibrated frequency counter, or by setting up an accurately calibrated receiver with Argo and monitoring the output signal frequency that way.

Once you have measured the actual output frequency, you can calculate the required correction to the 27MHz reference frequency and enter it in the “Ref. Frq.” configuration setting. For example, suppose you set the output frequency to 13.500000 MHz but you actually measure 13.500075. Your output frequency is 75 Hz too high. Since your output frequency setting is 2 times less than the 27MHz reference oscillator frequency, this means that the actual reference oscillator value is 150Hz too high. So enter 27,000,150 in the “Ref. Frq.” setting.

## **7. PA bias set-up**

If you are using the OCXO/Si5351A Synth you will find that the R5 trimmer potentiometer is only just accessible under the OCXO when plugged in. This is a great improvement over earlier PCB revisions in which the trimmer potentiometer was completely covered by the OCXO! There is only so far it could be moved, before falling off the board edge. An alternative method of setting up described below, can be used – and does not require the OCXO to be present.

Initially the bias trimmer potentiometer R5 should be set all the way anti-clockwise. With the Si5351A module plugged in, and when you have set up the software to configure your U3S for some transmission, connect a 50-ohm dummy load (two 100-ohm resistors in parallel is fine, or a 47-ohm resistor is fine too). Monitor the power output. As you turn R5 gradually clockwise, the power output will increase. At some point beyond half way, the power output will peak, and as you continue to turn clockwise, power will decrease. This is too far! Now your BS170 will start getting hot. Maybe very hot. So back-off a little. My recommendation is to set the power output a little below the peak setting. A little ANTI-clockwise, from the peak power output setting.

An alternative adjustment procedure also works very nicely. I recommend this method, if you have access to a means to measure the current consumption of the kit with reasonable accuracy. You should measure the overall current consumption of the U3S kit in “idle” mode (key up, not transmitting). You do not even need to plug in the Si5351A module for this. It is a useful method for those using the OCXO, who find accessing R5 to be difficult. Monitor the current consumption of the U3S in “idle” mode and adjust R5 until you just about see a slight increase. This will be a fairly reasonable bias point for good operation of the PA, when you plug in the Si5351A module.

## **8. Resources**

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

For troubleshooting please refer to <http://www.qrp-labs.com/ultimate3/u3trouble>

Further references are listed in the Operation manual.

## **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**                    **25-Oct-2016**

- Simplified and clarified the assembly manual

**4**                    **04-Apr-2017**

- Updated for “QCU Rev 3” PCB version

**5**                    **24-Apr-2017**

- Corrected mis-named component in section 4.1 – it is C2 and C3 which are not supplied

**6**                    **14-Jun-2017**

- Corrected circuit diagram, capacitors were labelled 10nF but are now supplied as 100nF