Ultimate3S: QRP Labs Multi-mode QRSS Beacon Kit

PCB Revision 2

1. Introduction

Thank you for purchasing the QRP Labs third generation "Ultimate3S" Multi-mode QRSS beacon kit. This kit is capable of automated transmission of a range of weak signal modes that are capable of worldwide HF propagation using a fraction of a watt of RF output power. The Si5351A synthesiser module permits accurate, stable operation anywhere on LF/MF/HF/VHF whilst plug-in low-pass-filter modules allow easy band changing.

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. Use a receiver connected to your PC, with a slow-signal decoding program such as Argo. Monitor your signal, experiment and understand the various features before connecting an antenna! **Good performance depends on proper set up: see the calibration section.**

This assembly manual must be read in conjunction with the operation manual!

The U3S kit supports the following modes:

- QRSS mode (plain on/off keyed slow CW)
- FSK/CW mode (frequency shift keyed slow CW)
- DFCW mode (dual frequency CW, dit's and dah's on different frequencies)
- WSPR and WSPR-15 modes (Weak Signal Propagation Reporter)
- Slow-Hellschreiber (frequency shifted slow Hellschreiber)
- Hellshreiber (full-speed standard Hellschreiber, and half-speed Hellshreiber)
- CW (plain CW) and fast FSK mode
- Transmitter mode (manual keyed, on/off CW or FSK)
- Customisable FSK patterns
- Opera
- PI4
- JT9, including data telemetry

Other features:

- Si5351A synthesiser controlled output frequency (kit included with the U3S)
- Plug-in low pass filter boards (for all LF/MF/HF bands from 2200m to 10m, plus 6m band)
- 16-column x 2-row LCD with backlight, and two-button user interface
- User-programmable (callsign, message, speed, FSK, mode, etc.), settings stored in EEPROM
- GPS interface, for locking the frequency, timing and location information
- On-chip generation of WSPR encoded message (no PC required), JT9, Opera, PI4 etc.
- WSPR maidenhead locator can be generated from GPS-derived latitude/longitude
- Selectable "frame" size, for stacked QRSS reception
- Plain CW callsign identifier at selectable interval
- Produces approximately 250mW RF output on 30m (lower output on higher frequency bands)
- Higher output power by fitting additional PA transistors and/or higher PA supply voltage

Ultimate 3S Multimode QRSS / WSPR TX Kit

ATmega328

Copyright © QRP Labs 2014

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

The transmitter is designed to be powered with 5V DC. This could come from a mobile phone charger, wall wart, or even four 1.5V batteries connected in series with a suitable low dropout regulator. However not all chargers and wall warts are clean enough for powering the microcontroller. Do not use more than 6V: this may kill the microcontroller; the LCD may not work properly with more than 5V. Best results will be obtained with a well-regulated and well-smoothed 5V supply.

2. Design

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

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
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 – space is provided for future enhancements

3.3 Inductors

L1	25 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
Si5351A	Synthesiser module, including 27MHz reference oscillator

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
16-pin male header plug
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

The kit comes as a main board, plug-in Si5351A synthesiser module, and a plug-in low-pass-filter module for the desired band. **The Si5351A synthesiser module and low-pass filter module are separate kits. The synthesiser and one LPF are included with the U3S. Please refer to the separate instructions for constructing these kits.**

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, C1, C2 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.

4.2 Construction steps

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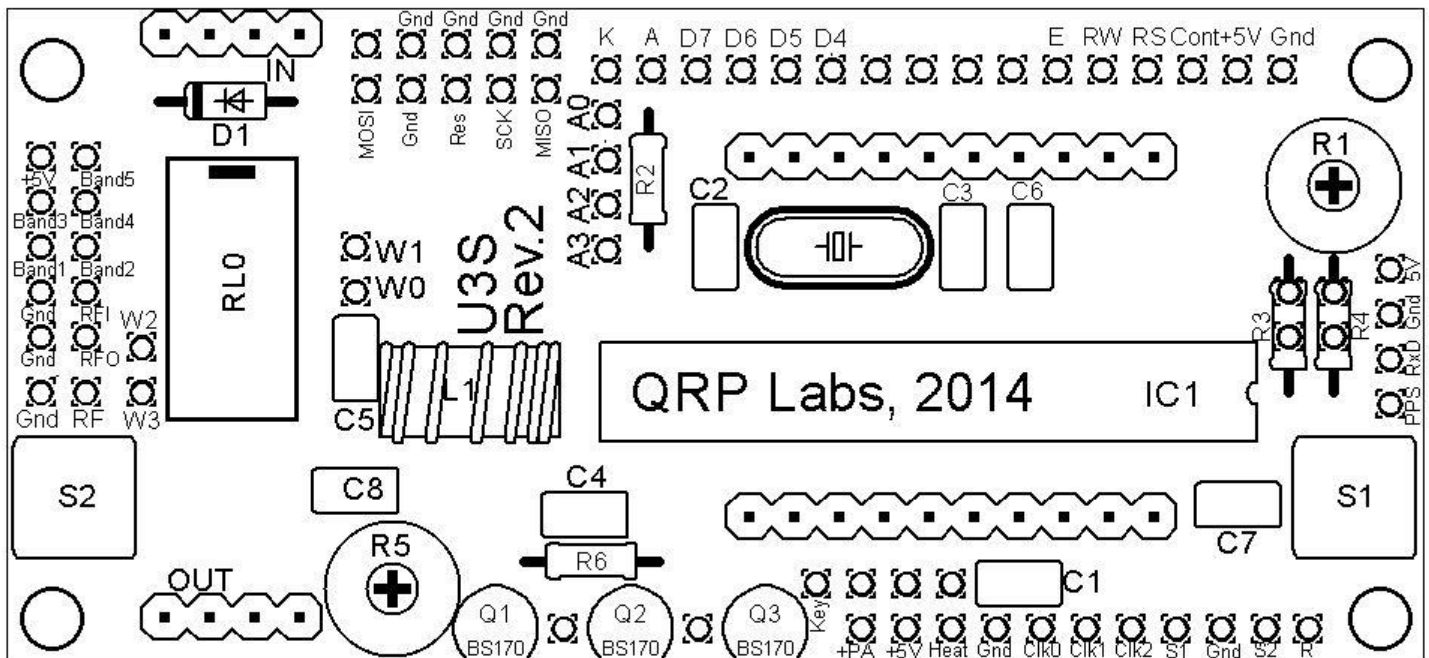
Pay special attention to the orientation of the semiconductors. For IC1, the dimple in the PCB silkscreen must be aligned with the dimple at the top of the IC socket and the IC.

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. C7 is also an optional capacitor, not supplied.

3) Wind and install the toroid

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. 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.

4) Solder the sockets for the Si5351A synthesiser module and the low-pass-filter board.

These are the two 4-way sockets, and two 10-way sockets. They are installed as shown in the above photograph.

Some care needs to be taken with the alignment, to ensure that there is a good fit when the plug-in boards are added. One method is to build the plug-in modules first; then use the Si5351A and LPF boards plugged into the sockets. You can then solder the pins, ensuring correct alignment.
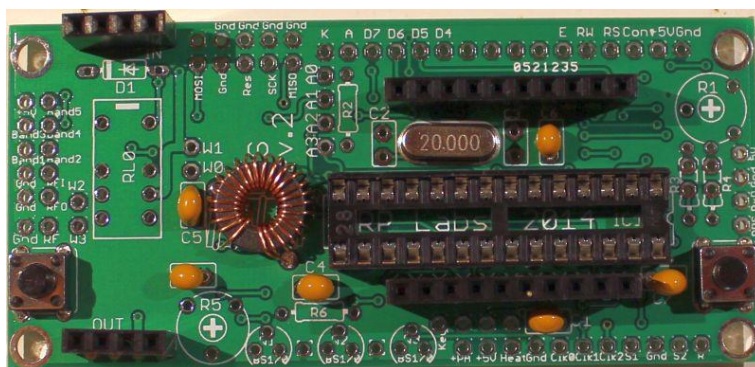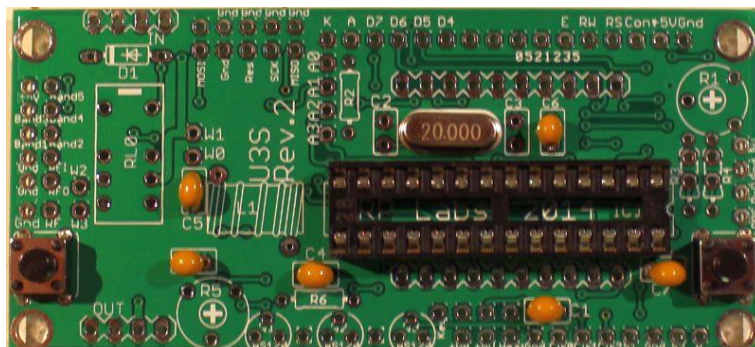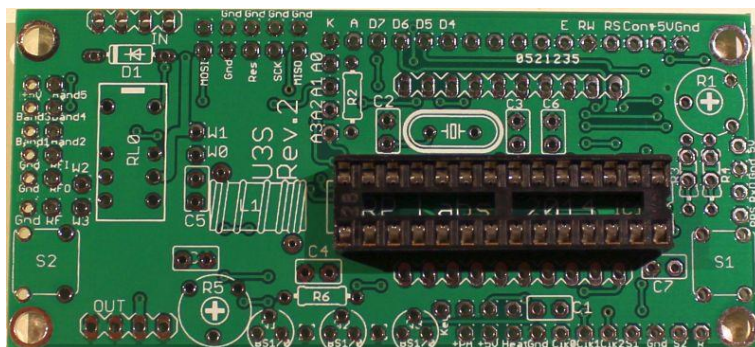
5) **Fit and solder R1, the preset potentiometer that sets the LCD contrast.**

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.
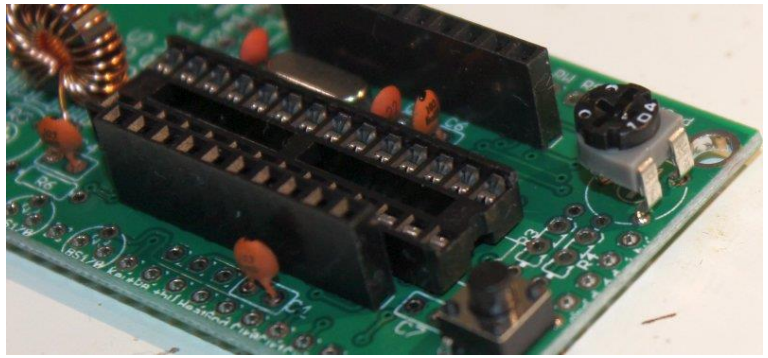
Adjust this potentiometer to the fully clockwise position initially.

6) **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 U3S board. Please see section "PA bias set-up" below. The photograph (right) shows R5 in the correct ANTI-clockwise position.
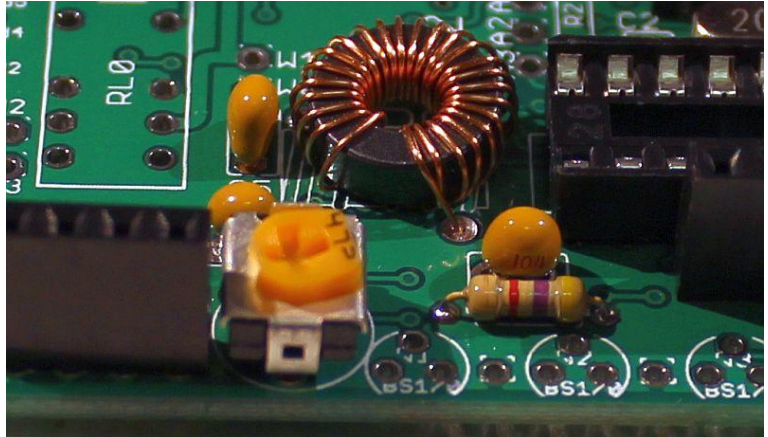
7) **Solder wire jumpers as shown in the photograph and diagram below, depending on your requirements.**

Typical jumper connections are shown in the photograph circled in yellow, and by red lines on the parts placement diagram below.
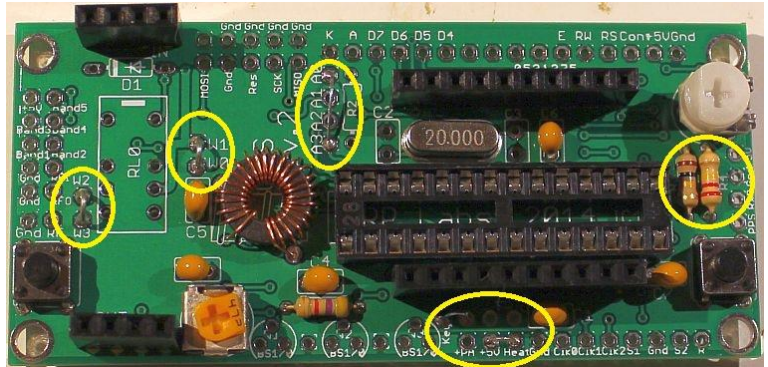
Please read the "Hardware options" sections below, to decide which jumpers to fit.

Note that the photo contains an error: the "key" connector is incorrectly jumpered to "+PA", at the bottom of the picture. It's an easy mistake to make, the silkscreen lettering on the PCB is small and the pads are next to each other. Make sure you don't fall into the same trap I did, here!

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.

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, that expands the capability of the U3S 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.

6

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 200K resistor for R4, then you must fit a wire jumper here as shown.
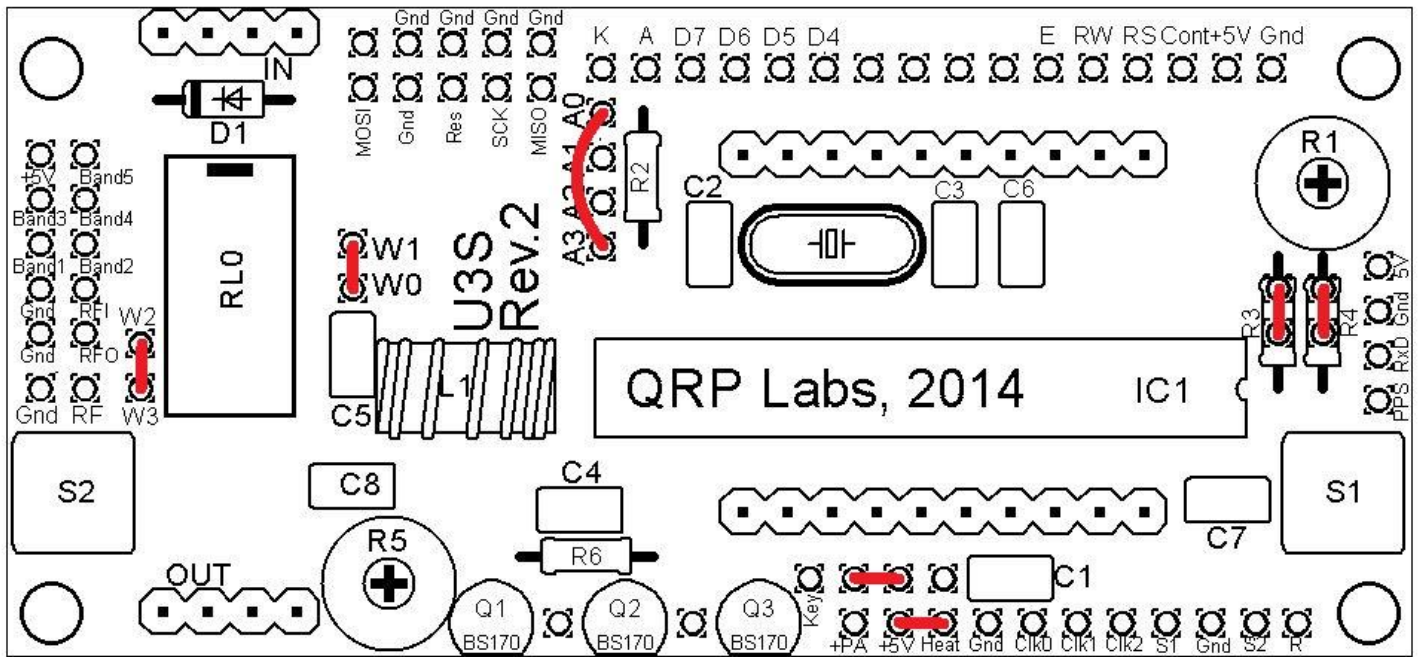
8) **Solder in transistor Q1.**

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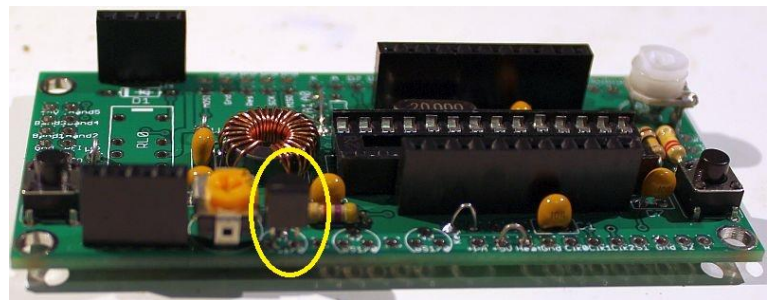
The transistors are located at the edge of the PCB with their flat side outwards. This is to allow easy attachment of a heatsink if desired (for higher voltage and hence higher power operation).

9) **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 U3S 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 U3S 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.

**(Photos show Rev 1 PCB, there are slight differences only).**
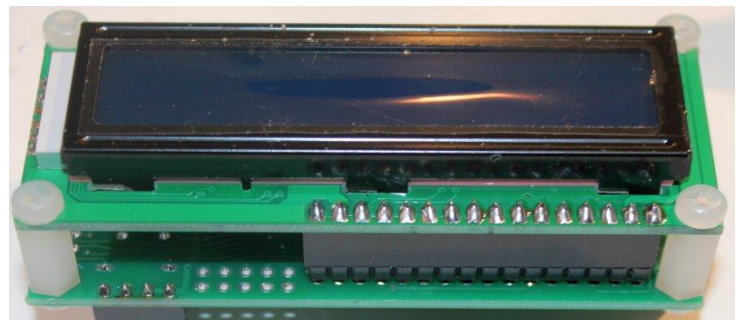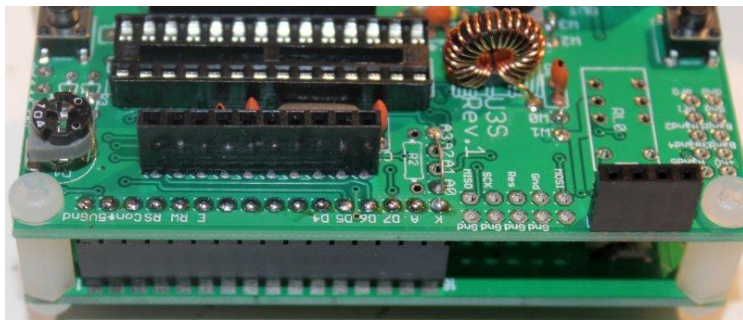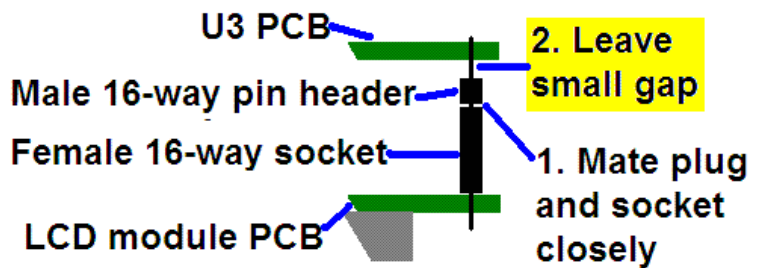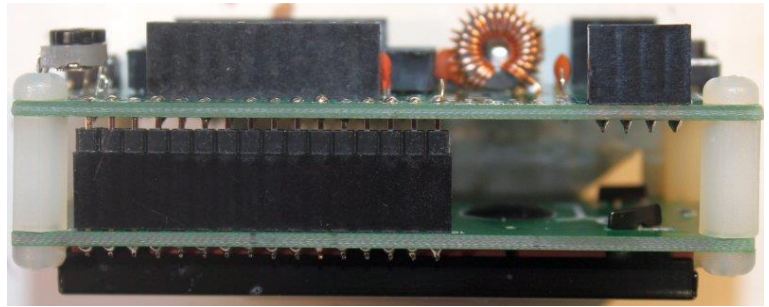
4.3 Module assembly

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 the three modules as shown in the following photographs.

These pictures show the Revision 1 PCB, but the module assembly is identical.
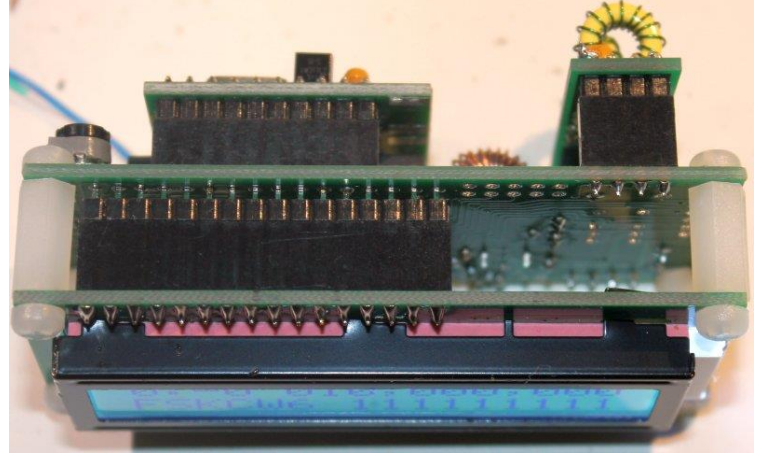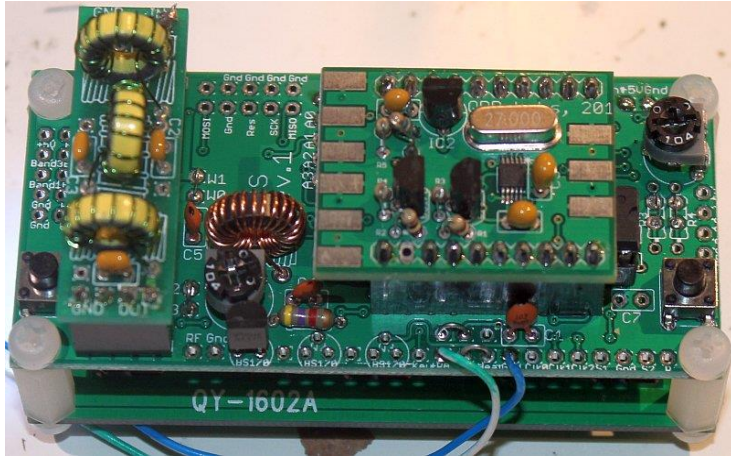
Ensure that the Si5351A synthesiser module is inserted the correct way round as shown in the photograph. The Low Pass Filter kit module should be plugged in with the "Out" legend aligned next to the RF Output connector of the main PCB (see photo).

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.
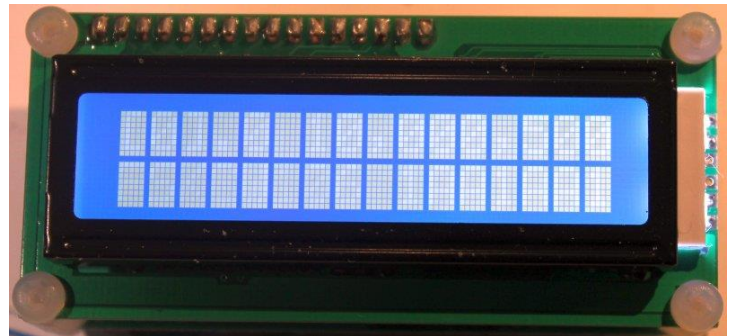
When you see "Diagnostic Mode" it means all is well with the processor and the LCD communication. When you press the left button, diagnostic mode will be cleared. Then you should see the flash screen. If the Si5351A module is correctly built and inserted, the firmware will recognise it. The text in the splash screen will say "Ultimate3S". The "S" is important, it signifies that the Si5351A has been correctly detected.
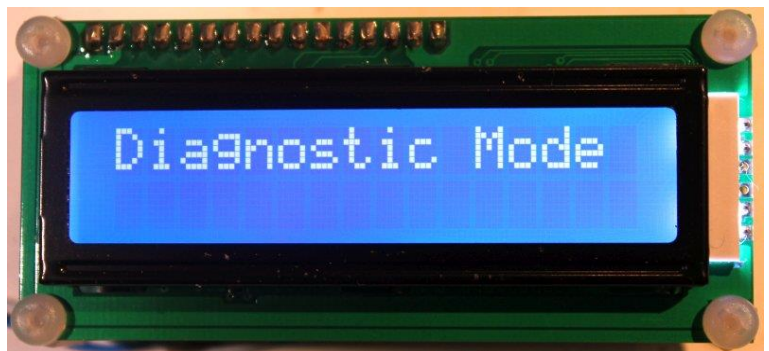
If you cannot see any text on the screen even after adjusting the contrast setting, you have a problem. Please refer to the "Troubleshooting" page on the U3S website (see resources section below).

4. Hardware Options

4.1 Explanation of connections

The table to the right 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.

Note that the cluster of connections at the left board edge are for fitting the connector to the relay-switched LPF board kit. They can be ignored for now.

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	No connection – for future enhancements
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	Clk2 Si5351A output, 3.3V p-p squarewave
Clk1	Signal	Clk1 Si5351A output, 3.3V p-p squarewave
Clk0	Signal	Clk0 Si5351A output, 3.3V p-p squarewave
Gnd	Power	Ground connection to Power supply
Heat	Power	+5V heater for OCXO/Si5351A module option
+5V	Power	+5V power supply
+PA	Power	PA power supply – may be connected to +5V
Key	Control	"Keying" output: high during transmit
Gnd	RF	Ground for RF output
RF	RF	RF output

Connections A0, A1, A2 and A4 allow options concerning the LCD backlight, as described below. Typically you should connect A0 and A3, this enables the software controlled backlight control.

## 4.2 Connection for basic operation
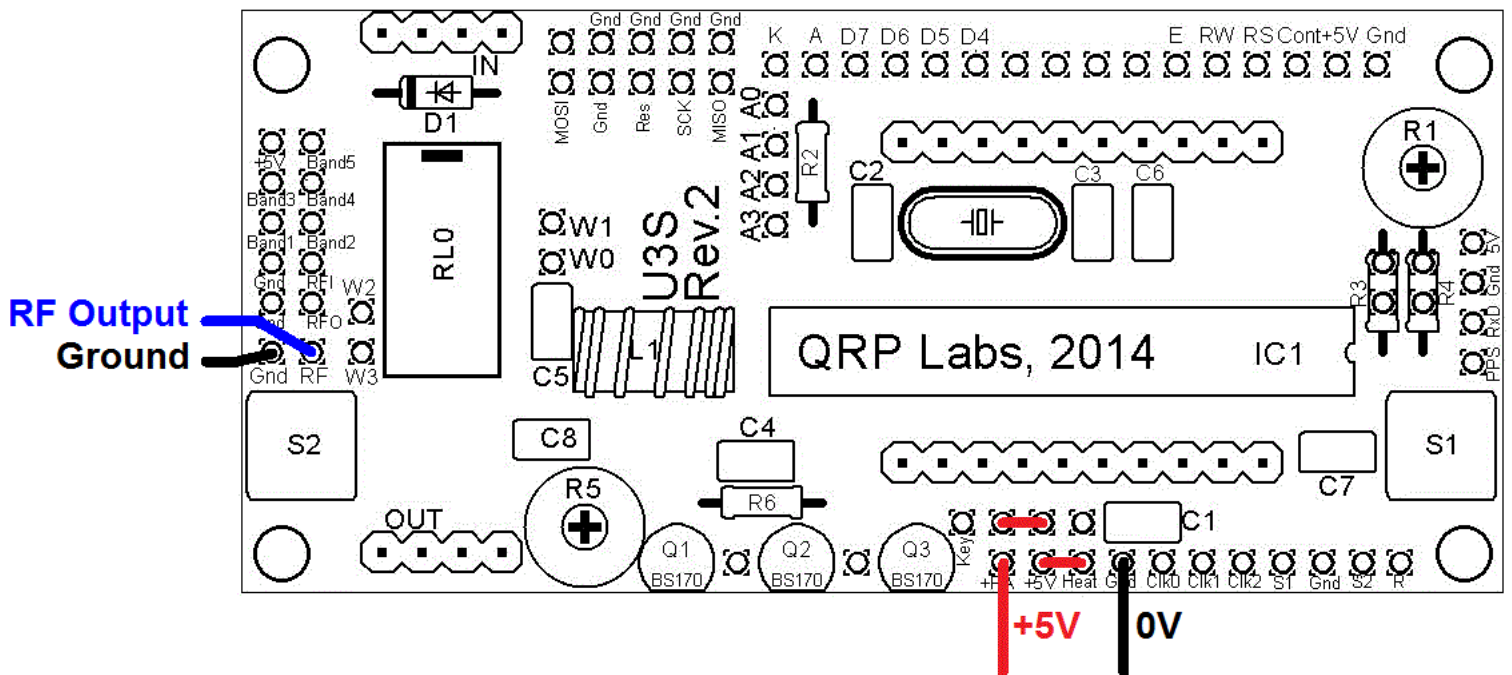
For the most basic operation of this kit as QRSS/WSPR transmitter using 5V supply, it needs only a power supply and RF output connection.

To allow use of a higher voltage supply for the PA, there are separate connections for the PA voltage and +5V supply. To run the transmitter using the 5V supply, be sure to connect a wire between the +5V and PA pins. This is shown by the short red horizontal line in the diagram (right).

The "Heat" connection is only required if you are using the OCXO version of the Si5351A synthesiser module. You must connect it to +5V to power the oven heater.

4.3 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.
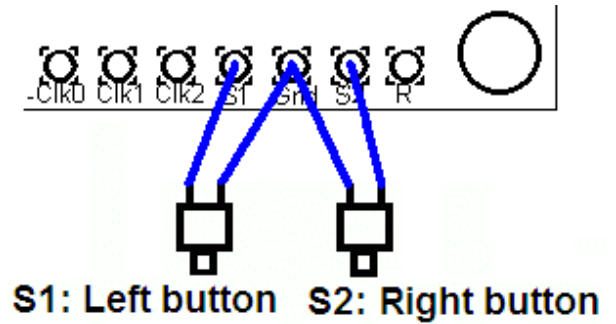
4.4 Optional connection of GPS module

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. 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.
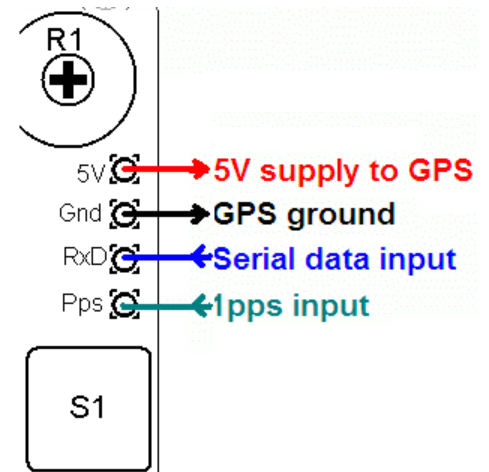
In cases where the location information is not required (modes other than WSPR) or you wish to enter it manually, the serial data input can be left unconnected. The 1 pps input is enough for the frequency lock function. A 1pps signal could be supplied from a non-GPS source, such as a laboratory frequency 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.

4.5 Higher power output

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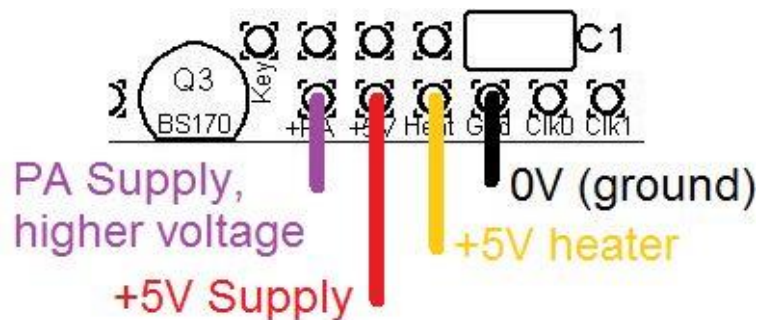
At higher supply voltages and power outputs, the heat dissipation in the BS170 will increase and its temperature will be higher. Provision is made on the PCB for two more transistors in parallel with the first. This will slightly increase the power output even at 5V supply, but more importantly at higher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

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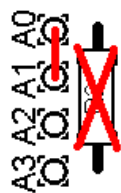
a) Full brightness: For basic operation connect A0 to A1 using a wire jumper (made from a capacitor lead offcut, for example). The blue LED backlight of the LCD module will be at full brightness. This is shown as the Red line (left). In this case, do not fit any resistor at the R2 position.

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.

4.7 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.

4.8 In-circuit programming of AVR

If you own an AVR programmer, you can make connections to the U3S 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 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 U3S.

Notes:

1) Power needs to be applied to the U3S during programming! When you start programming, whatever the U3S 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) 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.
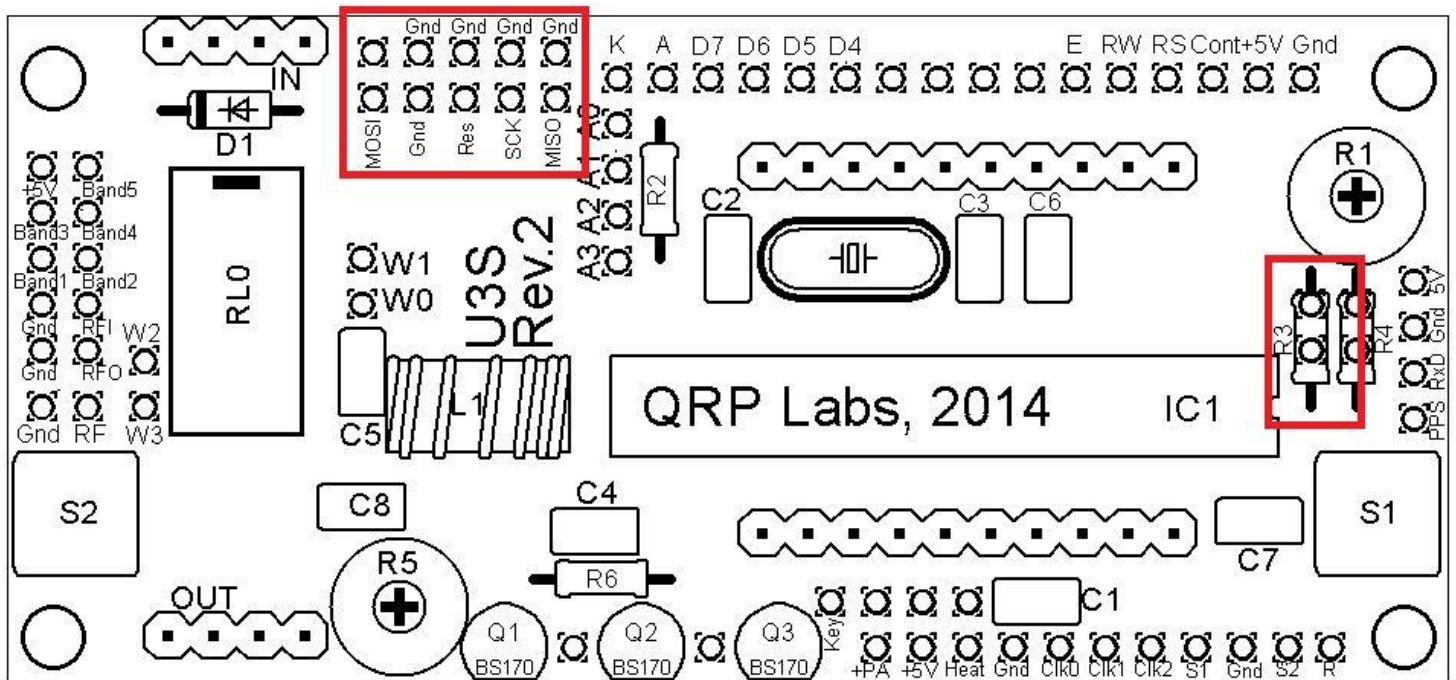
4.9 Higher sensitivity contrast potentiometer adjustment

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 220K resistor at R4 instead of the wire jumper.

5. Calibration

This section is REALLY IMPORTANT! 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. 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.

6. 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 PCB Rev 1 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. Here 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.

7 Resources

Please see the kit page http://www.qrp-labs.com/ultimate3 for any 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.

8. Version History

0   14-Jan-2015

- First version

1   26-Jan-2015

- Correction to section 2, Design. Removed incorrect reference to DDS module and keying transistor Q1

2   03-Feb-2015

- Correction to R5 value, the supplied trimmer is 4.7K (not 5K as previously stated)

3   25-May-2015

- Warning about pin 2 of programmer header, in U3S PCB Rev 1.

- Clarification regarding Si5351A output p-p magnitude

4   16-Jun-2015

- Updates for PCB Rev 2: different PCB layout; 22pF C1 & C2 are omitted; new C8 capacitor, etc.

5   06-Jul-2015

- Correct typo, wrong capacitors mentioned in construction step on page 5

6   09-Nov-2015

- Add note to horizontally mount L1 if you intend to use the OCXO/Si5351A kit