

Ultimate3: QRSS Labs Multi-mode QRSS Beacon Kit

PCB Revisions 3 and 4

1. Introduction

Thank you for purchasing my third generation “Ultimate3” 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 DDS module permits accurate, stable operation anywhere on HF 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, to monitor your signal, experiment and understand the various features before connecting an antenna! **Good performance depends on proper set up:, see the calibration section.**

Please read the DDS module stability section 7! Do not miss this section, or your output signal may be unstable and look horrible!

This assembly manual is to be read in conjunction with the operation manual!

The kit supports the following modes:

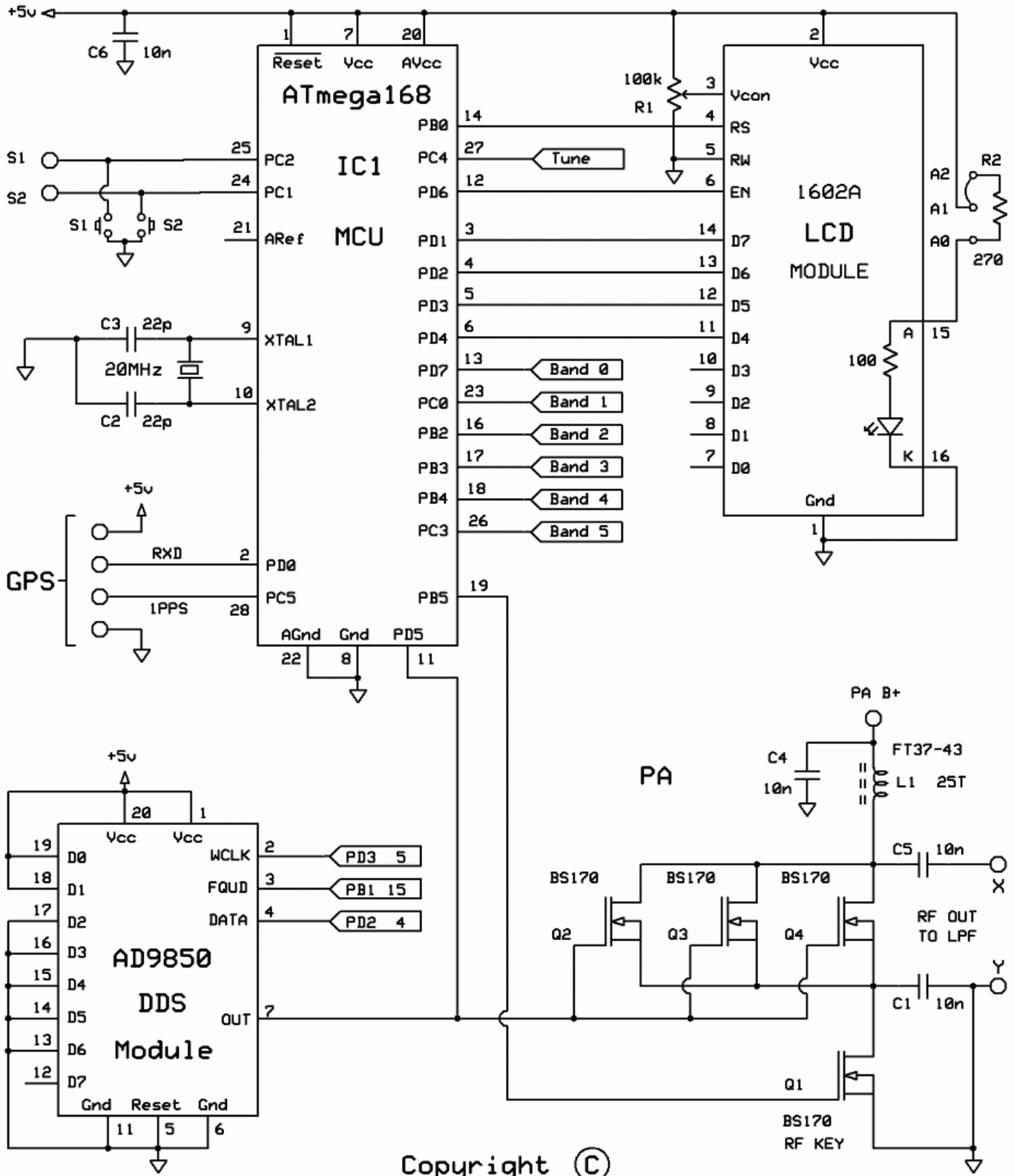
- QRSS mode (plain on/off keyed slow CW)
- FSK/CW mode (frequency shift keyed slow CW)
- DFCW mode (dual frequency CW, dit's and dah's on different frequencies)
- WSPR and WSPR-15 modes (Weak Signal Propagation Reporter)
- Slow-Hellschreiber (frequency shifted slow Hellschreiber)
- Hellshreiber (full-speed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

Other features:

- DDS-controlled output frequency (through-pin DDS module, no SMD soldering required)
- Plug-in low pass filter boards (available for 10 HF and 2 LF bands)
- 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)
- 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 150mW RF output on 30m (lower output on higher frequency bands)
- Higher output power by fitting additional PA transistors and/or higher PA supply voltage

The transmitter is designed to be powered with 5V DC, which could come from a mobile phone charger, wall wart, or even four 1.5V batteries connected in series with a suitable low dropout regulator. 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



Ultimate-3 Multimode QRSS Kit by Hans Summers G0UPL SEP 2013

The ATmega328 AVR microcontroller (IC1) is pre-programmed with firmware to control the LCD, buttons, and GPS interface. The AD9850 Direct Digital Synthesis (DDS) module includes its own on-board 125MHz crystal reference oscillator, and reconstruction filter components ready-mounted. On/off keying is provided by the BS170 transistor Q1, and power amplification by another BS170 transistor, Q2, producing over over 150mW from a 5V supply on 30m. 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
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)
C1, 4, 5, 6	10nF (ceramic, marking 103)
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, 4	BS170 (note: only two are provided in the kit. The other two may be added for more power).
D1	No diode – provided for use with relay-switched LPF board kit
IC1	Pre-programmed ATmega328 microcontroller
DDS	AD9850 DDS module, including 125MHz reference oscillator and reconstruction filter

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, with pre-assembled DDS module, and a plug-in low-pass-filter module for the desired band. **Since the low-pass filter module is a separate kit in its own right, please refer to the separate instructions for constructing that kit.**

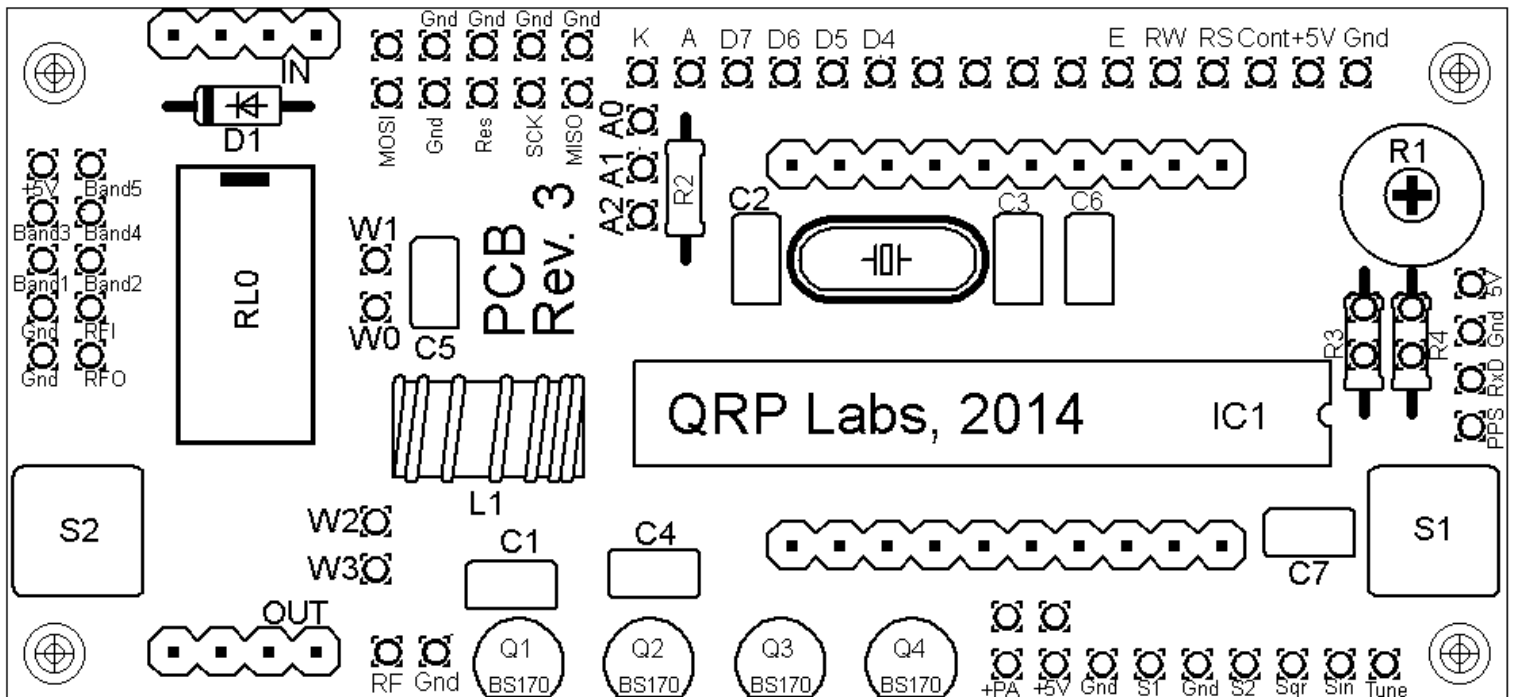
Parts placement is defined by the printed legend on the PCB, so please observe it carefully, paying particular attention to the correct orientation of the semiconductors.

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 can assist. 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 inadvertently created.

Note that components R2 and C7 are not required or supplied in the kit (R2 may be added by the builder if lower LCD backlight brightness is desired). Q3 and Q4 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 DDS module and the low-pass-filter module.

4.2 Construction steps

Please refer to the parts placement diagram below.



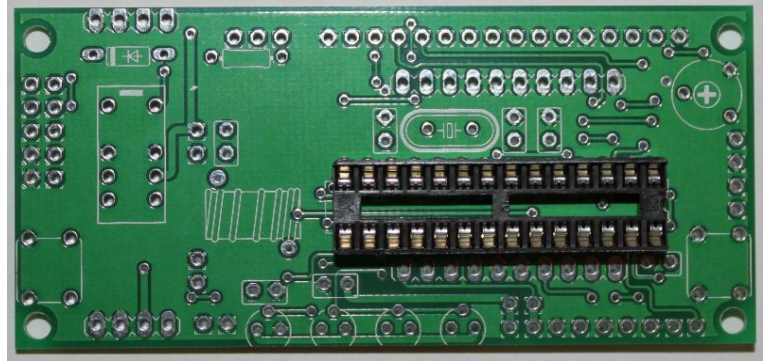
Pay special attention to the orientation of the semiconductors. For IC1, the dimple in the PCB silkscreen must be aligned with the dimple at the top of the IC socket and the IC.

The order of construction is not important but 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 and I'd recommend following it carefully.

NOTE: Photos below show PCB revision 1, but they are very similar!

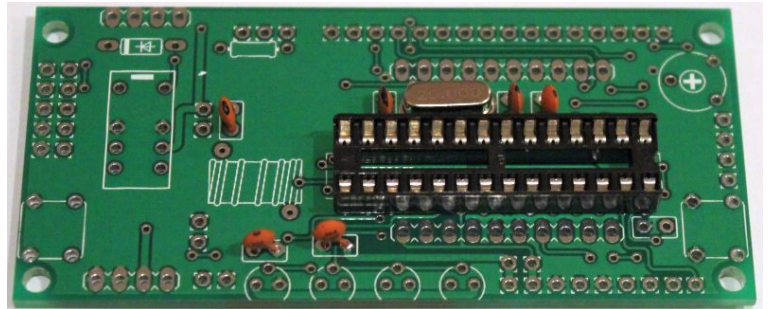
1) Solder in the socket for IC1.

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.



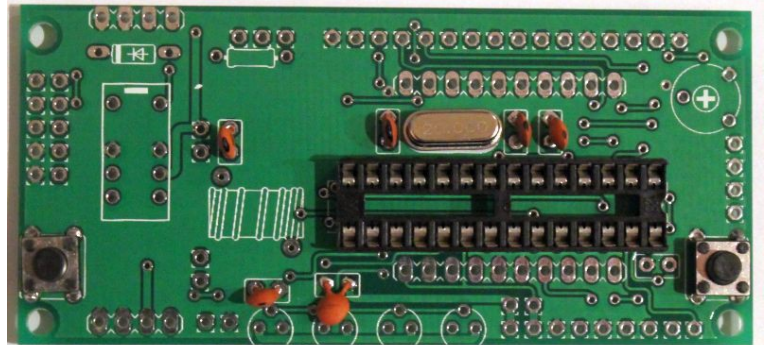
2) Solder in all capacitors and the quartz crystal (note there is no C7).

Be careful not to mix up the 22pF and 10nF capacitors. The two 22pF capacitors are the ones on each side of the crystal.



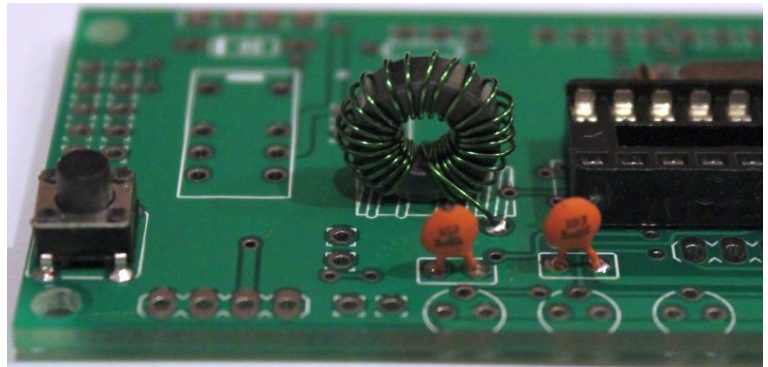
3) Fit and solder switches S1 and S2, if not using off-board switches.

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 “hardware options” section).



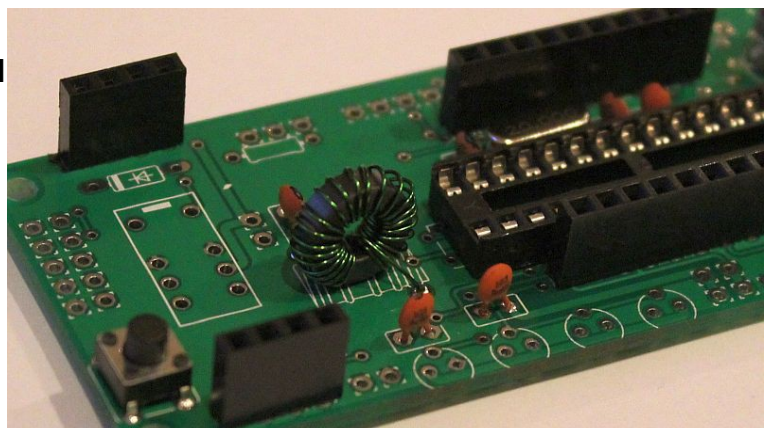
4) Wind and install the toroid

The toroid can be mounted horizontally or vertically. I prefer the horizontal method as there is enough space for it, and it keeps the toroid away from the LPF module which is plugged in above. 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.



5) Solder the sockets for the DDS module and the low-pass-filter board.

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 LPF kit board first, and

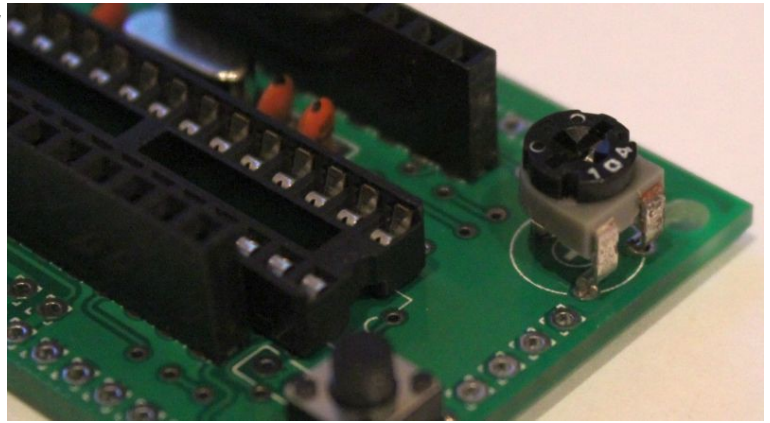


use the DDS and LPF boards plugged into the sockets, then solder the pins, to ensure correct alignment.

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

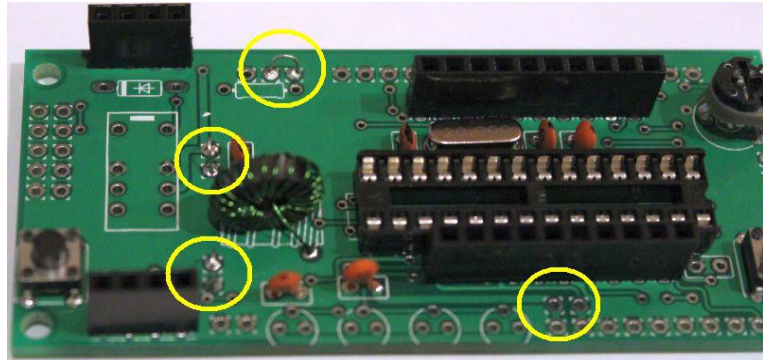
It is a slightly tight fit but apply pressure carefully and evenly, and the pins of the potentiometer will fit into the PCB perfectly.

Adjust this potentiometer to the fully clockwise position initially.



7) Solder wire jumpers as shown in the diagram below (red wires circled in yellow), depending on your requirements.

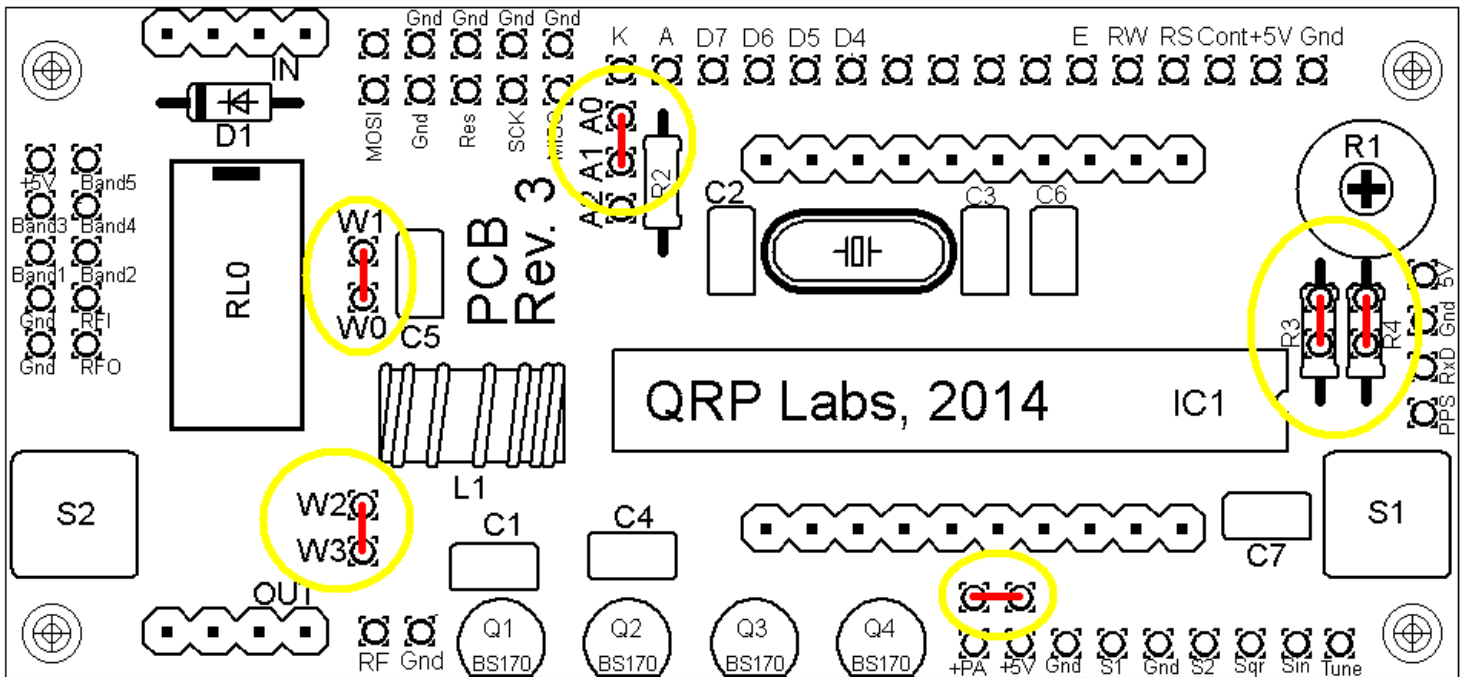
These can be made from offcuts from the other components e.g. capacitor leads. When installing these wire jumpers, I find it convenient to install them as a small semi-circular arch, perhaps 5mm high off the board, as pictured. This is so that later, if they need to be removed (for different hardware options configuration), a convenient alternative is to just cut them using wire cutters. **PCB Rev 3/4: please follow diagram below, not photo!**



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 U3 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 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.



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.

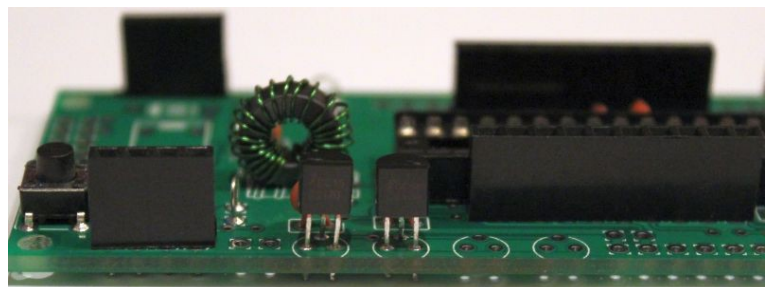
Resistor R4 is an optional resistor, which is in series with the top of the contrast adjustment potentiometer R1 and +5V. Typically the contrast voltage required is less than 1V and the adjustment is rather sensitive. If you wish, you can fit a 220K resistor in this position, which will make the contrast 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 transistors Q1 and Q2.

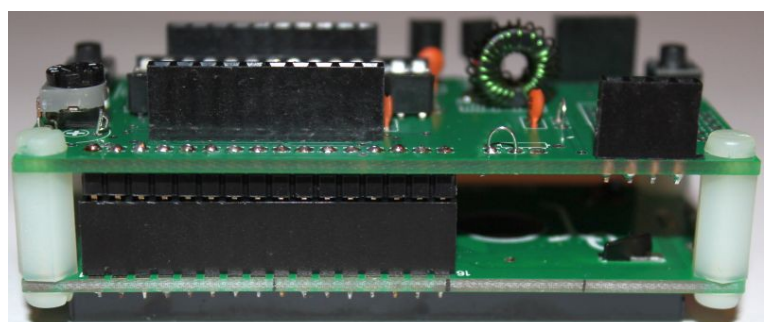
If you have purchased additional transistors for Q3 and Q4 to increase the power output, then Install these now too.

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.

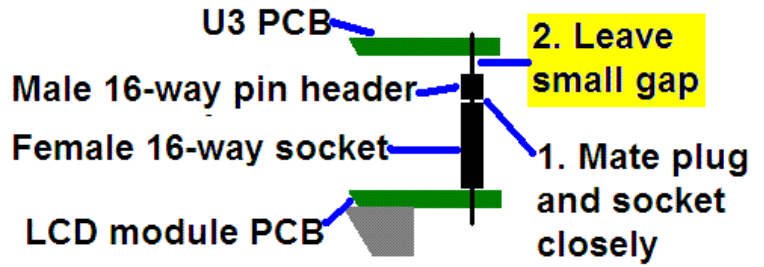
Either way is fine, but my recommendation for installing the LCD connectors, is to use the female (socket) connector on the PCB side and the male (plug) on the LCD. **Be sure to solder the socket to the reverse of the main U3 PCB!! Not the top side!** If you bolt together the LCD module and the PCB, with the 16-way connectors in place but not



soldered, then you will be able to 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.

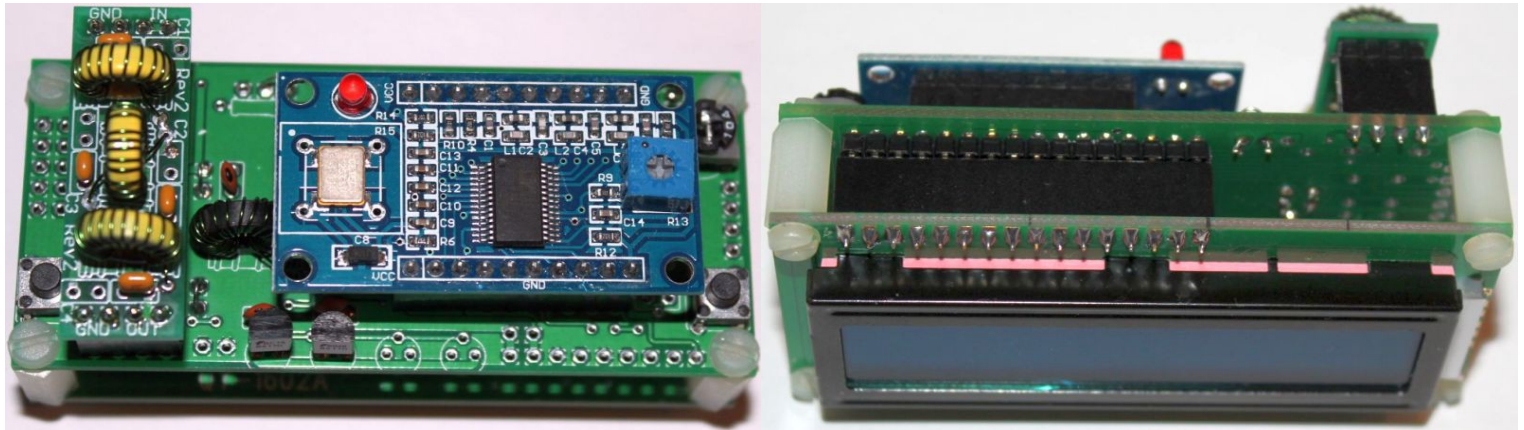
My suggested method is illustrated to the right here. I suggest closely mating the plug and socket. Leave the gap between the male header (plug) and the U3 PCB. I have found this method to work successfully.



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.



Ensure that the DDS module is inserted the correct way round. 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 anti-clockwise until the displayed text looks correct.

4.4 Notes concerning the DDS module

a) You will notice that the DDS module has a square, blue plastic adjustment preset potentiometer. This adjusts the threshold for the DDS chip comparator that turns its 1V p-p sine wave into a square wave to drive the microcontroller. Adjusting it will change the duty cycle of the DDS square wave. You should not need to adjust this potentiometer, as I have already adjusted it individually, when I tested every DDS module before putting it in a kit. However, some builders have noted that the potentiometer needs to be adjusted slightly for best output waveform on 10m (28MHz). The adjustments needed are extremely slight, the potentiometer setting is very sensitive.

b) The module uses the AD9850 DDS chip. The module is officially rated for output from DC to 40MHz. You may want to test it at higher frequencies than that. In theory the chip could output up to 62.5MHz (half the 125MHz reference clock frequency), but the output amplitude and spectral purity will deteriorate.

c) It is normal for the 125MHz reference oscillator (metal can on the right) to get hot in operation.

5. Hardware Options

5.1 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.

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
Tune	Future	No connection – for future enhancements
Sin	Options	RF output from DDS, 1V p-p sinewave
Out	Options	RF output from DDS, 5V p-p squarewave
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
Gnd	Power	Ground connection to Power supply
+5V	Power	+5V power supply
+PA	Power	PA power supply – may be connected to +5V
Gnd	RF	Ground for RF output
RF	RF	RF output

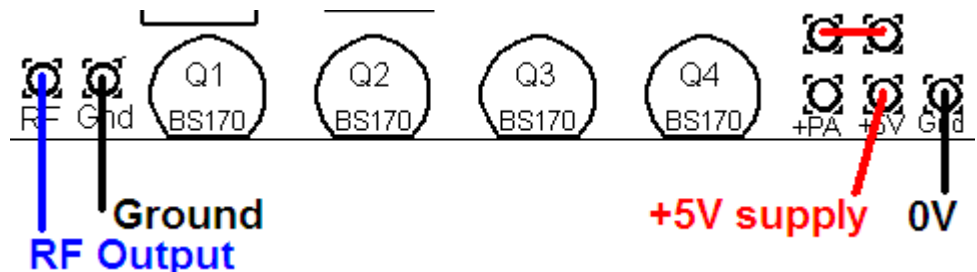
Note that the cluster of connections at the left board edge are for fitting the connector to the relay-switched LPF board kit. They can be ignored for now.

Connections A0, A1 and A2 allow options concerning the LCD backlight, as described below. For basic operation, simply connect A0 and A1.

5.2 Connection for basic operation

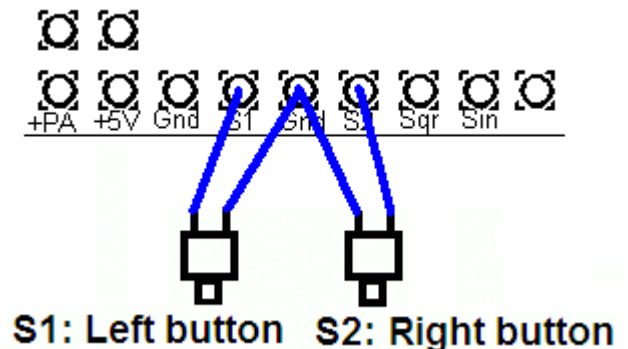
For the most basic operation of this kit as QRSS transmitter using 5V supply, it needs only a power supply and RF output connection.

To allow use of a higher voltage supply for the PA, the PA voltage is not connected to +5V on the PCB. So to run the transmitter using the 5V supply, be sure to connect a wire between the +5V and PA pins, as shown by the short red vertical line in the diagram (right).



5.3 Alternative button mounting option

If you wish 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.



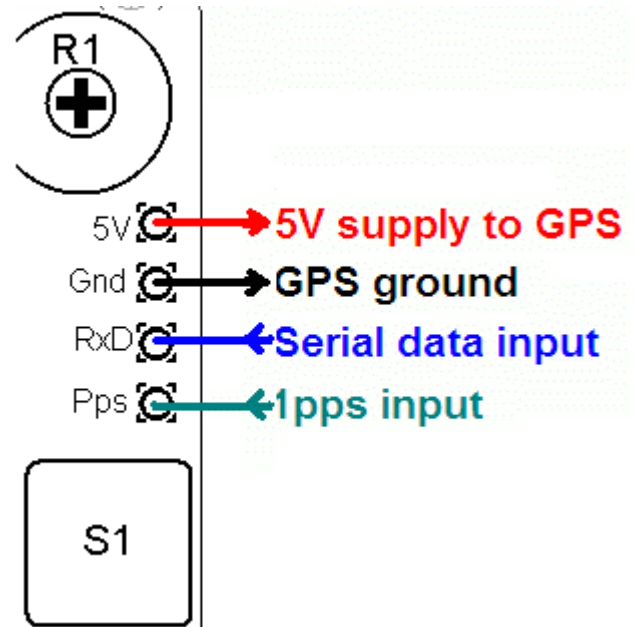
5.4 Optional connection of GPS module

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. 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.

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. 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).

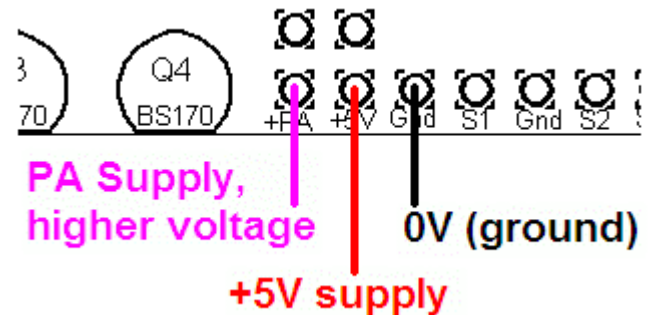
See operation instructions for details on how to configure the GPS interface.



5.5 Higher power output

The kit provides around 150mW of output power using a 5V supply and a single BS170 transistor (depending on the band). The transistor gets slightly 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.



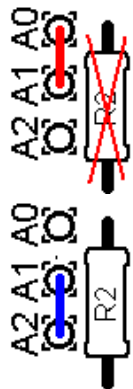
It may be necessary to fix a heat sink to the transistors if the temperature rise is excessive. The transistors are sited near the edge to the board to facilitate this. Experiment is needed in this area, with PA voltages up to perhaps 12V.

5.6 Audio frequency output

The kit can be used to generate audio tones for feeding to an SSB transceiver etc. In this case you can simply set the output frequency to the desired audio frequency, and feed the “RF” output to the microphone input. It is very important to note that this output is a just a 5V peak-to-peak square wave – to connect it to a microphone input (for example) you will need to reduce the signal amplitude considerably. In testing, I used a potential divider made of an 18K resistor and a 10-ohm resistor to ground. The mic input was connected across the 10-ohm resistor and provided a suitable signal amplitude.

Another possible way to use the AF output is to rectify it (diode + capacitor) and use that as a plain on/off keying signal to a CW transmitter. This would be suitable for CW and Hellschreiber modes, but not modes where a frequency shift encoding is involved.

5.7 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.

c) 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 off during operation.

5.8 DDS outputs

The connection points labelled Sqr and Sin provide direct connection to the DDS module's squarewave and sinewave outputs. The squarewave connection is the output pin of the DDS module's comparator and is a 5V squarewave. The sinewave connection is the analogue frequency output of the DDS module and has a 1V p-p amplitude. In both cases, you should refer to the AD9850 datasheet and ensure that you understand the specification, and do not draw excessive loads from these connections.

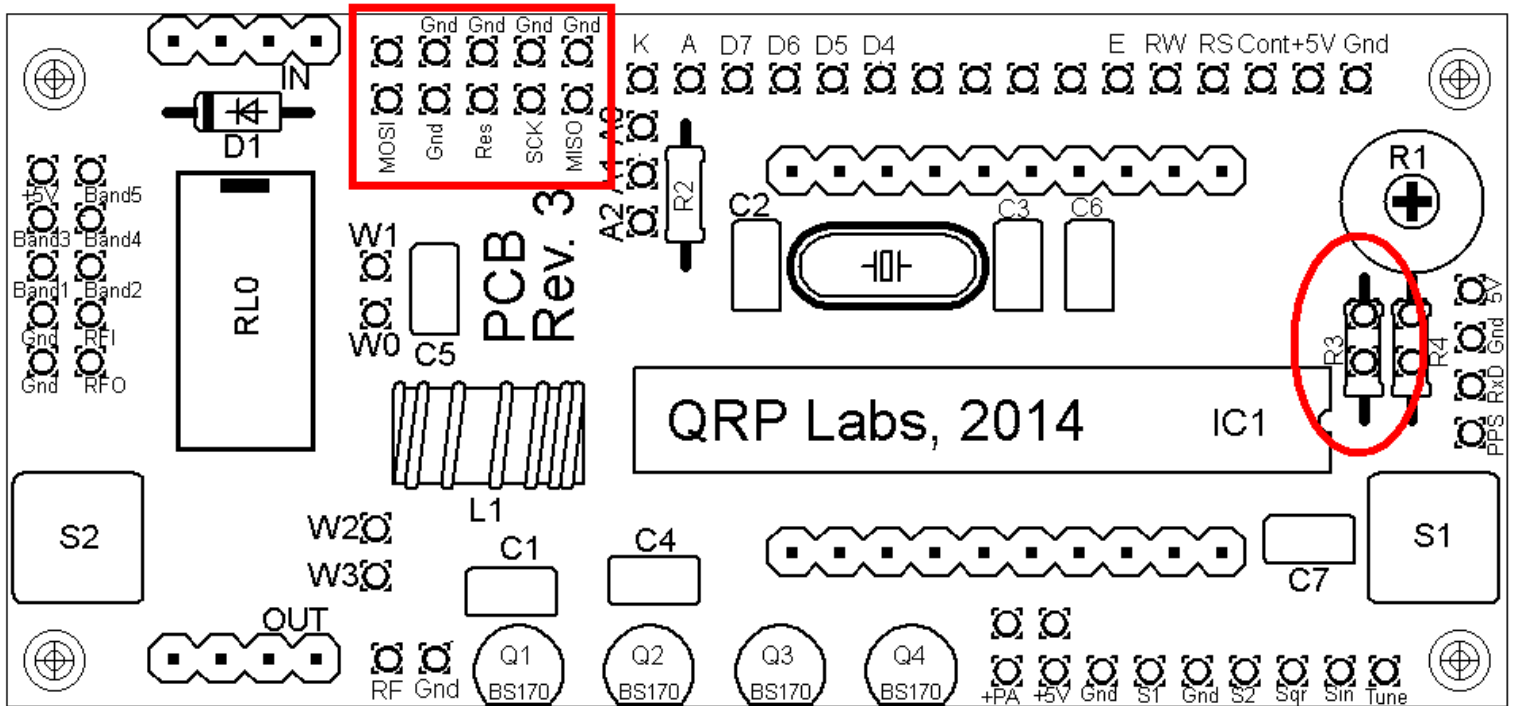
5.9 In-circuit programming of AVR

If you own an AVR programmer, you can make connections to the U3 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 circled 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 U3.

Notes:

- 1) Power needs to be applied to the U3 during programming! When you start programming, whatever the U3 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.



5.10 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.

6. 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 125MHz reference oscillator frequency into the “Ref. Frq.” configuration setting.

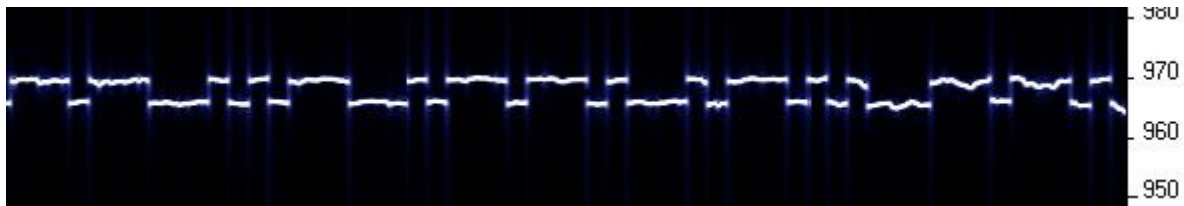
It is possible that you have an accurate means to measure the 125MHz reference oscillator frequency itself by probing the correct point on the DDS module. However most kit builders will not have the necessary equipment, and such a measurement at VHF is not easy to make without itself risking disturbing the frequency.

The easiest method is to set the output frequency to something convenient such as 12.500000 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 125MHz reference frequency and enter it in the "Ref. Frq." configuration setting. For example, suppose you set the output frequency to 12.500000 MHz but you actually measure 12.500075. Your output frequency is 75 Hz too high. Since your output frequency setting is 10 times less than the 125MHz reference oscillator frequency, this means that the actual reference oscillator value is 750Hz too high. So enter 125,000,750 in the "Ref. Frq." setting.

7. DDS Module Stability

Here's an extremely important tip regarding the stability of the DDS module. If you just wire everything up and go on air, you would surely be ashamed to see how horrible your signal will probably look:



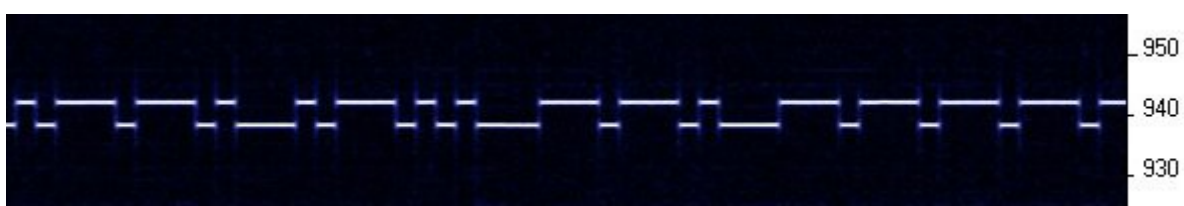
Obviously the DDS module was designed for something else, not just our QRSS passion, and the stability at Hz-level wasn't important. My own theory is that there are temperature sensitive components within the 125MHz crystal reference oscillator, which take time to all reach the same temperature. When they do, the temperature coefficients largely cancel each other out. But in the short term, while the different components are experiencing slightly different temperatures, the frequency can shift considerably. I believe that even small air currents over the oscillator case are enough to alter the local temperature and cause frequency instability.

I have found that in practice there are two easy ways to solve this problem. Preventing the instability is very important before putting the transmitter on air.

- 1) If you put the kit into an enclosure, such that you've removed any air currents around the crystal oscillator, the frequency instability disappears.
- 2) Attaching (e.g. with glue) a small heatsink to the crystal oscillator can also cure the frequency instability. I found that almost any heatsink will do. Even take the smallest, most pathetic coin in your country and place it on top of the crystal oscillator, and the frequency instability is cured. In fact even non-metal will do it – a piece of plastic, for example – which also adds weight to the air current theory, since anything you put on top of the crystal oscillator stops air currents altering the temperature so easily.

Arguably a heatsink may be a good idea in any case, to slow down temperature changes of the crystal oscillator anyway. Either way: just don't leave the crystal oscillator in the open air.

Below see the result of placing a coin on the crystal oscillator.



8 Resources

Please see the kit page <http://www.hanssummers.com/qrssultimate2> for any information on latest updates and issues.

Further references are listed in the Operation manual.

Analog Devices DDS information: <http://www.analog.com/en/rfif-components/direct-digital-synthesis-dds/products/index.html>

9. Version History

0 **15-Oct-2013**

- Initial draft version

1 **11-Nov-2013**

- First official version

2 **01-Feb-2014**

- PCB revision 2

3 **06-Mar-2014**

- PCB revision 3

4 **06-Jun-2014**

- Include PCB revision 4 (minor change to expose pads at the corner mounting holes)