**SoftRock v5.0 Builder's Notes**

December 12, 2005

## Building a QSD Kit

Be sure to use a grounded tip soldering iron in building the QSD board. The soldering iron needs to have a small tip, (0.05 - 0.1 inch diameter), and be in the power range of 15 to 20 watts. A 2% silver-bearing solder with diameter of 0.015 inches works well for SMT work.

The schematic diagram, bill of materials and board information files for the QSD board and the OSC/BPF board may be down loaded from the HamSDR website or from the Yahoo SoftRock40 group website. These documents will be needed during the build of each of the SoftRock v5.0 kits.

Solder the three 0.1 uF 1206 SMT capacitors C3, C6 and C7 in place on the bottom side of the QSD board. This can be done by applying a small amount of solder to one pad and positioning the capacitor with the tip of the soldering iron and a toothpick for proper placement on the SMT pads. When the capacitor is properly placed and tacked down on one end, solder the other end of the capacitor to its pad. Re-heat the first pad and apply a little more solder if needed to get a small fillet between the end of the capacitor and the pad. Examine carefully with good lighting and magnification to make sure the capacitor is soldered down properly with a small fillet at each end of the capacitor.

Solder the four SOIC integrated circuits, U2, U3, U4 and U5 to the bottom of the board. Integrated circuit orientation on the QSD board is shown by small "1" marks in the bottom-side copper near the pin 1 corner of each IC. U2 and U3, (both LT1719 SOIC-8 devices), are in a line at one end of the QSD board and located near the 15-pin header connector. Pin 1 on the IC device is at the lower left corner when the IC is viewed with the text reading upright from left to right.

To position an IC for soldering, apply a small amount of solder to an IC corner pad on the board. Position the IC with the tip of the soldering iron tip and a toothpick so that the IC is properly centered on all pads and tacked down at the one corner. Double check the IC type, IC orientation and IC positioning on its pads! Apply solder to each of the remaining pads of the IC pattern to solder each pin of the IC to its pad. SOIC devices are large enough that soldering may be done a single pin at a time. Examine carefully with good lighting and magnification to make sure each pin of the IC is soldered down properly to its pad. Reheat any pads where the soldering looks questionable. Remove any excess solder and solder bridges between IC pins or pads with solder wick.

U4, (FST3126 SOIC-14 device), is next soldered in place with the same technique as with U2 and U3. Again be careful with device orientation, proper placement of the device on its pads, proper soldering at each pin and solder bridges between pins.

U5, (LTC6241 SOIC-8 device), is solder in place as with U2 through U4. Again be careful with device orientation, proper placement of the device on its pads, proper soldering at each pin and solder bridges between pins.

This completes the mounting of parts on the bottom side of the QSD board. Four 0.1 uF SMT 1206 capacitors, C4, C5, C8 and C15, are next soldered to the top side of the board to complete the SMT soldering of the QSD board.

Do not mount the 15-pin connector, J1, on the QSD board until the last operation where the QSD board and a OSC/BPF board are to be connected together.

Topside QSD resistors are next mounded hairpin style on the top of the board. Resistors provided in the QSD kit are typically 1% resistors with four color bands indicating the resistor value with the standard color code. However, the color bands are hard to view without good lighting and colors such as red, brown and orange look nearly the same on the blue body of each resistor. Remove all doubt as to the resistor value by making an ohmmeter check before mounting a resistor.

Bend the leads of R1, R7, R12 and R13 so that one lead of each resistor bends back with a small radius and is parallel to the body of the resistor. Insert each resistor into its mounting position with the body of the resistor positioned just above the silkscreen circle indicating the location of the resistor. Keep the resistor leads short by pushing the end of each resistor body against the board. The bent back lead goes through the via pointed to by a small line on the circle of each resistor silkscreen pattern. Spread the leads of each resistor on the underside of the board to hold it in place for soldering. Solder the four resistors from the bottom side of the board and cut off the excess lead with flush cutting wire cutters. (The four resistor will provide "legs" so that the circuit board can sit bottom-side up on a flat surface to make soldering easier.)

Diode D1 also mounts hairpin style with its band end to the square pad in the upper left corner of the board. Snug up the body of D1 to the board so that its height is no higher than the height of the resistors.

Mount the remaining resistors to the board in the same way as the above resistors.

The tantalum capacitors are next mounted to the topside of the board. Some of the 10 uF tantalum capacitors provided in the kit have a lead spread of 0.2 inches which does not fit with the 0.1 inch lead spacing on the board. The leads of such capacitors must be carefully reformed to a 0.1 inch spread for proper mounting to the board. Note the small "+" mark on the silkscreen pattern for each tantalum capacitor and orient each capacitor so that its positive end is connected to the positive via of the capacitor location. Snug up the tantalum capacitors to the board so that their height above the board is no higher than the height of the resistors.
Solder the remaining capacitors and U1, (78L05 regulator), in place. Keeping their leads short so that the height of each component is less than the height of the resistors. Ceramic capacitor values are typically marked with a three digit number where the third number indicates the number of zeros following the first two digits. For example: 221 is a capacitor with value 220 pF.

Carefully inspect all solder connection on the QSD board and touch up any connections that look questionable.

Fasten the board mounting hardware to the three QSD board mounting holes. Two 3/8 inch 4-40 machine screws along with nylon spacers and washers are used at each of the mounting holes near J1. A 3/8 inch machine screw with a 1/8 inch nylon spacer is inserted from the bottom side of the board through a mounting hole. A nylon washer and 4-40 nut go on the top side of the board on each of the machine screws.

The third board mounting hole receives a 1 inch 4-40 machine screw from the bottom of the board along with a 1/8 inch nylon spacer. On the top side of the board a 1/4 inch nylon spacer is placed on the machine screw followed by a 4-40 nut. The remaining two 1/8 inch nylon spacers and 4-40 nut are used to hold a OSC/BPF board on top of the QSD board.

Wait on mounting connector J1 on the QSD board until the QSD board and an OSC/BPF board are to be connected together.

The stereo audio cable may be connected at this time to the QSD board. Use a short piece of #22 bus wire to connect the ground via between the I and Q output vias to the barrel connection of the cable and wrap the end of the bus wire around the outside of the cable for strain relief of the cable. The tip of the stereo cable connects to the via that is electrically connected to C14 and the ring connects to the via that is electrically connected to C13.

Connect DC power leads and antenna leads as desired to the indicated vias on the QSD board as shown in the QSD information file.

**Building an OSC/BPF Kit**

Solder the resistors and capacitors to the OSC/BPF board as done on the QSD board. Resistors and axial lead capacitors should be mounted with leads formed in a hairpin style. Keep leads short so that the overall height of all components is no greater than the height of the resistors.

Diodes locations D21 and D22 are not populated at this time but may be used later when an exciter board is designed. The jumper location JP21 can be used to connect the antenna return connection to circuit ground by soldering a short piece of bus wire between the vias, however, no connection is required and it has been found that the SoftRock receiver operates better if JP21 is left open.

When mounting a crystal in the X21 location make sure conductive portions of the crystal case do not short between the pads in the crystal mounting location. The crystal pad arrangement is such that a number of SMT style crystals may be mounted as well as leaded crystals. A ground via is provided to solder a wire between the crystal case and ground for a HC49 type crystal. This will keep the crystal case slightly above the crystal pads to prevent the crystal case from shorting out the pads. Crystals will be mounted horizontally in the X21 location.

Inductors L21 and L22 are wound on the appropriate cores for the low band or high band with AWG #26 enameled wire. Each pass through the center of a core is counted as a turn when winding the inductors. The inductors are mounted horizontally in their locations. Be sure to remove the enamel coating on the wire before attempting to solder an inductor to the board. The enamel coating may be removed and the wire tinned by a holding the inductor lead in a hot blob of solder on the tip of the soldering iron.

Transformer T21 is also mounted horizontally. In winding T21, first wind the primary winding with AWG #26 enameled wire so that the primary winding starts and ends at the same point on the core and is uniformly spread around the core. Twist two pieces of #26 enameled wire together and wind the secondary windings bifilar with the windings starting and ending 180 degrees around the toroid core from where the primary winding starts and ends. After tinning each transformer lead at about 3/8 of an inch from the core, determine the two pair of leads that form each of the two secondary windings with an ohmmeter. Pairs of hole on the board are associated with the pairs of leads of each of the secondary windings. A secondary lead from the bottom of the transformer core is routed through the via marked 3 on the board. The second lead of that same secondary winding, coming from the top of the transformer, goes to the adjacent via next to the via marked 3. The lead from the top of the transformer of the other secondary winding goes to the via marked 6 on the board and the its second lead from the bottom of the transformer goes to the adjacent via next to the via marked 6.

Carefully inspect all solder connection on the OSC/BPF board and touch up any connections that look questionable. Use an ohmmeter to verify that the leads of L21, L22 and T21 connect to the appropriate vias.

## Connecting a QSD Board and an OSC/BPF Board

Carefully push the shorter leads of the 15-pin header P21 into the sockets of the connector that will be J1. The plastic bar of the header pins should be about 1/16 of an inch away from the plastic body of the J1 connector. The longer pins of the P21 header will go through the P21 vias of the OSC/BPF board from the bottom side of the OSC/BPF board and the solder tails of the J1 connector will go through the J1 vias of the QSD board from the top of the QSD board.

Place an 1/8 inch nylon spacer over the 1 inch machine screw on the QSD board and position the OSC/BPF board directly above the QSD board. This will sandwich the connector pair P21/J1 between the two boards. Press the two boards together so that the board are parallel to each other and the connectors are perpendicular to their respective boards. The plastic bar on P21 should be pressing against the bottom of the OSC/BPF board and the plastic body of J1 should be pressing against the top of the QSD board. The long pins of P21 should be extending above the OSC/BPF board. Use another 1/8 inch nylon spacer and 4-40 nut on top of the OSC/PBF board to hold the boards together. Carefully solder on the top of the OSC/BPF board at each of the pins of P21. Also carefully solder the solder tails of J1 on the bottom of the QSD board.

## Testing the v5.0 SoftRock Receiver

Before applying power to the v5.0 SoftRock receiver make sure there are no shorts between the DC power input and circuit ground on the OSD board. The resistance between any 5 VDC point on the QSD board and circuit ground, such as the lead of U1 closest to the board mounting hole, should be about 1.2k. If no power shorts are detected, apply 9 to 12 VDC to the board and check the 5 VDC regulated voltage level at the pin of U1 mentioned above. If the 5 VDC voltage is in the range of 4.6 volts to 5.4 volts continue with the operational tests, otherwise look for problems such as solder shorts between pads and vias. The current from the DC supply should be about 25 mA.

For the functional check out of the SoftRock v5.0 the Rocky program is recommended. Setup the Rocky program as listed on the website. Plug the stereo cable coming from the QSD board into the line-in jack of the soundcard. Set up the Rocky program for the selected soundcard and enter the crystal oscillator frequency as the center frequency in the Rocky setup. Connect a 50 ohm antenna to the QSD board antenna connections and enable Rocky operation. Watch for signals on the Rocky spectrum display indicating the receiver is functioning and tune to a desired signal by use of the mouse to point and click.

If the v5.0 SoftRock is not functioning refer to the schematic diagram to trace signals through the various circuits. A test RF oscillator with output in the range of 1 uV RMS to 22 mV RMS and a dual trace scope would be very helpful for signal tracing.