

SoftRock v6 Builder's Notes

August 25, 2006

Be sure to use a grounded tip soldering iron in building the v6.0 SoftRock circuit 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, BOM (bill of materials) and board information file for the v6.0 SoftRock board may be down loaded from the HamSDR website or from the Yahoo SoftRock-40 group website. These documents will be needed during the build of the SoftRock v6.0 kit.

Fasten the board mounting hardware provided in the kit to the v6.0 PCB with the 3/8 inch length 4-40 screws coming through the four corner mounting holes from the bottom of the board. The hardware stack at each PCB mounting hole includes a machine screw, a 1/8 inch nylon spacer, the PCB, a nylon washer and a hex nut. The board mounting hardware will help protect the surface mount components as the board is built.

A total of eleven 0.1 uF 1206 SMT capacitors are soldered to the board, ten on the bottom side of the PCB and one on the topside of the PCB. No other SMT capacitors or resistors are used on the board so simply fill each 1206 SMT position with a 0.1 uF capacitor. 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 each capacitor is soldered down properly with a small fillet at each end of the capacitor.

Solder the three SOIC integrated circuits, U3, U4, and U5 to the bottom of the board. Integrated circuit orientation is shown by small "1" marks in the bottom-side copper near the pin-1 corner of each IC. U2, on the topside of the board, orientation is identified by a silkscreen pattern showing the pin-one end of the IC. In general, the pin-1 corner of each IC device is in the lower left corner of the device when the text on the device topside can be read upright from left to right. Some devices also have a dimple in the package indicating pin-1 location. (The SOIC IC devices in the kit are such that if a device provided in the kit fits the PCB footprint, then the IC is the right one for the location.)

Exercise care to properly orient each IC and center the IC on its pads. To position an IC for soldering, apply a small amount of solder to a board corner pad where the IC is to

be soldered in place. Position the IC with the tip of the soldering iron and a toothpick so that the IC is properly centered on all pads and tacked down at one corner. Double check the IC orientation and IC positioning on its pads! Apply the soldering iron tip and 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. Use a small amount of solder to avoid solder bridges. 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. In the same manner solder U2 to its position on the topside of the board.

This completes the mounting of SMT parts. Topside resistors are next mounded hairpin style on the top of the board. Resistors provided in the 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's value by making an ohmmeter check before mounting a resistor.

Bend the leads of R1, R2, R7 with finger tip force so that one lead of each resistor bends back with a small radius and is parallel to the body of the part in a hairpin fashion. 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 lightly pushing the end of each resistor body against the board. The bent back lead goes through the via pointed to by a small radial 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 three resistors and cut off the excess lead with flush cutting wire cutters. (These three resistors provide "legs" so that the circuit board can sit bottom-side up on a flat surface to make soldering easier.)

Bend the leads of R14 in the same manner as above. Carefully note the R14 location in the lower left corner of the board. The two leads of R14 go to holes that are in a line parallel to the long side of the board. A large ground via just above the R14 location does not receive a lead of R14.

Mount the remaining resistors and diode D1 to the board in the same way as the above resistors. Diode D1 is oriented with its band end to the square pad in the location where D1 is mounted. Also mount the TO-92 parts, U1, Q1 and Q2. Be careful to identify Q1, a 2N3904 transistor, and Q2, a 2N3906 transistor, from each other since the markings on Q1 can easily be misread.

The electrolytic capacitors are next mounted to the topside of the board. Note the small "+" mark on the silkscreen pattern for each electrolytic capacitor and orient each capacitor so that its positive end is connected to the positive via of the capacitor

location. Lightly snug up the electrolytic capacitors to the board so that their height above the board is no higher than the height of the resistors. Exercise care in soldering these capacitors in place since the vias are small and closely spaced.

Special note for the 30 meter kit with v6.0 circuit board:

An extra resistor, R17, is added to the circuit for the 30 meter kit. R17 is mounted in the location where C6 and C7 are indicated on the PCB silkscreen. One lead of R17 goes to the grounded hole of the C6 location and the other lead of R17 goes to the left hole of the C7 location. The leads of R17 will need to be bent tightly to the ends of the resistor body to fit R17 between the two holes. C6 is not used in the 30m kit. C7 is then soldered to its plated through-holes from the bottom of the board with the body of C7 flat against the underside of the board.

Solder the remaining ceramic capacitors and in place. Keep capacitor 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.

A three-pin header is provided for mounting in the JP1 position near U2 on the topside of the board. The kit builder has an option to have long or short header pins. Cut the soldered ends of the header pins flush from the bottom side of the board. Mount JP2, near the edge of the board, in the same way as JP1. Two-pin jumpers are provided in the kit to short the header pins for normal operation on the selected band.

The crystal pad arrangement is such that a number of SMT style crystals may be mounted as well as leaded crystals. (If a SMT style crystal is to be used, a jumper wire will be needed from the crystal plated through-hole which connects to the base of Q1 to a rectangular pad provided on the board.) A ground plated through-hole is provided to solder a wire between the crystal case and ground for a HC49 type crystal. The crystals will be mounted horizontally in the X1 location.

Inductors L1 and L2 are wound with the specified number of turns of magnet wire on the specified core. Each pass through the center of a core is counted as a turn when winding the inductors. The inductors are mounted vertically in their locations with an insulated piece of bus wire used as a strap through the center of a core to isolated plated through-holes on the board to hold the inductors in place. Be sure to remove the enamel coating on the wire before attempting to solder an inductor lead to the board. The enamel coating on the #30 wire provided in the kit does not heat strip very well. Emory paper may be used to remove the enamel coating prior to tinning the lead.

Transformer T1 is mounted horizontally and raised above the board about 1/16 of an inch. In winding T1, first wind the primary winding with enameled wire so that the

primary winding starts and ends at about the same point on the core and is uniformly spread around the core. Twist two pieces of enameled wire together and wind the secondary windings with the windings starting and ending 180 degrees around the toroid core from where the primary winding starts and ends. After striping and tinning each transformer lead at about 3/8 of an inch from the core, determine the two pairs of leads of each of the secondary windings by use of an ohmmeter. Route a lead coming from the top of the core of the first secondary winding through the plated through-hole marked 3. The other lead of the first secondary winding goes through the unmarked plated through-hole next to the plated through-hole marked 3. The lead from the top of the core of the second secondary winding also goes through the unmarked plated through-hole. The second lead of the second secondary winding, from the bottom of the core, goes through the plated through-hole marked 5.

The primary leads of T1 are routed through the plated through-holes marked 1 and 2. When the transformer is positioned horizontally with its leads to the appropriate plated through-holes, solder all the leads and cut leads flush to the bottom side of the board. Use an ohmmeter to verify the transformer lead connections by checking for about zero ohms between plated through-holes 1 and 2 and zero ohms between the unmarked plated through-hole and plated through-holes 3 and 5.

Carefully inspect all solder connection on the board and touch up any connections that look questionable. Problems in getting a SoftRock to operate properly are most often associated with soldering problems on the board.

A stereo audio cable may be connected at this time to the board at the three plated through-holes along the left edge of the board near the lower left corner. Use a short piece of #22 bus wire to connect the middle plated through-hole (ground) to the shield (barrel) of the cable and wrap the end of the bus wire around the outside of the cable several turns for strain relief of the cable. The tip of the stereo cable jack connects to the plated through-hole that is marked with the letter I on the board and the ring of the stereo cable jack connects to the plated through-hole marked with the letter Q.

Connect DC power leads to the pair of plated through-holes just below D1 on the left edge of the board. The plated through-hole nearest to D1 is the positive connection to a DC power source and the lower of the two plated through-holes is the power supply negative connection or circuit ground.

Connect a length of 50 ohm coax to the antenna connection on the right edge of the board near the upper right corner. The lower of the two plated through-holes is the coax shield connection and the upper plated through-hole is the coax center conductor connection.

Initial Testing the v6.0 SoftRock Receiver

Place the two-pin jumper plugs between pins 1 and 2 on JP1 and JP2. This will enable

a oscillator divide by four for I/Q clocking. Before applying power to the v6.0 SoftRock receiver make sure there are no shorts between the DC power input and circuit ground. The resistance between any 5 VDC point on the board and circuit ground, such as the leads of C2, should be approximately 1.2k. If no shorts are detected, apply 9 to 12 VDC to the board and check the 5 VDC regulated voltage level at the positive lead of C2. If the 5 VDC voltage is in the range of 4.6 volts to 5.4 volts continue with the operational tests, otherwise immediately disconnect DC power to the board and look for problems such as solder shorts between pads and plated through-holes. The current from the DC supply should be 35 mA or less for the 40m/80m receiver.

The Rocky program is recommended for the functional check out of the SoftRock v6.0. Download the Rocky program from VE3NEA's website and setup the program as specified on the website. Plug the stereo cable coming from the SoftRock board into the line-in jack of the soundcard selected in the Rocky settings window. Enter the crystal oscillator frequency divided by four as the center frequency in the Rocky settings. Connect a low level (less than 2 mV RMS) RF source with a frequency equal to the center frequency minus 5 kHz frequency to the SoftRock antenna connections and enable Rocky operation. (Alternately, connect the SoftRock to a 50 ohm antenna.) Watch for a signal on the Rocky spectrum display indicating the receiver is functioning and tune to the signal by use of the mouse to point and click. Note that the image of the input signal, if visible, is much lower in amplitude and an equal distance above the center frequency as the input signal is below the center frequency.

If no signal is observed or if a strong image signal is also observed start debug efforts with a careful visual check of the circuit board. Make sure all IC devices are oriented properly and that there are no open pins or solder bridges between IC pins. Look for components mounted in the wrong locations or with a lead to an improper plated through-hole. Reheat any solder connection that looks doubtful. Use an ohmmeter to verify that transformer T1 is connected as shown on the schematic.

A couple quick voltage checks are as follows: DC voltages measured at pins 1, 3, 5 and 7 of U5 should all be close to 2.5 VDC. The voltage at the unmarked plated through-hole on T1 as well as the plated through-holes marked 3 and 5 should be close to 2.5 VDC. The DC voltage at the base of Q1 should be close to 2.5 VDC and the emitter of Q1 should be close to 1.9 VDC.

The I and Q square wave clock signals may be viewed with a scope at the two plated through-hole located in the center of the board on the top side. The frequency of these square waves should be the same as the center frequency and a 90 degree phase difference between the two square waves should be observed with a dual trace scope.

Additional debug help is available by posting problems to the SoftRock-40 Yahoo group website or by sending e-mail to Tony, KB9YIG, at raparks@ctcisp.com.

