V6.2 SoftRock Lite Builder's Notes

November 17, 2006

Be sure to use a grounded tip soldering iron in building the v6.2 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. Use good electrostatic discharge, (ESD), percussions when building the board by use of a wrist band grounding strap and grounded work surface when building the kit.

The schematic diagram, BOM (bill of materials) and board information file for the v6.2 SoftRock board may be down loaded from the files area of the Yahoo SoftRock 40 group website. Look in the SoftRock Lite folder under the v6 SoftRock folder. These documents will be needed during the build of the SoftRock v6.2 kit.

A total of ten 0.1 uF 1206 SMT capacitors are soldered to the board, nine 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 74HC74 SOIC 14 integrated circuits, U2, in place on the top of the board. Integrated circuit that solder to the bottom of the board may be solder in place after all leaded components are soldered in place on the top of the board. This will minimize the exposure of the integrated circuits to ESD damage during the kit build. 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.

Topside resistors are 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 identify as to color 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 R2, R5, R14 and R15 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 hole 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 four resistors and cut off the excess lead with flush cutting wire cutters. (These four resistors provide "legs" so that the circuit board can sit level, bottom-side up, on a flat surface to make soldering easier.)

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 its mounting location. Also mount the TO-92 parts, U1, Q1 and Q2. Be careful to identify Q1, a 2N3904 NPN transistor, and Q2, a 2N3906 PNP transistor, from each other since the markings on Q1 can easily be misread.

The electrolytic capacitors are mounted to the topside of the board. Note the "+" mark on the silkscreen pattern for each electrolytic capacitor and orient each capacitor so that its positive lead is connected to the positive hole 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 pads are small and closely spaced.

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: 152 is a capacitor with value 1500 pF and 220 is a capacitor with value 22 pF.

The JP1 location is three holes in a row just above U2 and parallel to U2 on the topside of the board. A short length of a resistor lead needs to bridge the two holes marked 4x when

a 30m or 40m kit is being built. If an 80m or 160m kit is being built then the two holes marked x8 need to be bridged with the wire lead. (This sets the division ration on the crystal frequency to divide by 4 or divide by 8.) The QSD mute line needs to be grounded for normal operation of the SoftRock receiver by bridging the two holes in the lower left corner of the board with a short wire loop on top of the board. (This wire loop will also be a convenient ground connection when testing the board.)

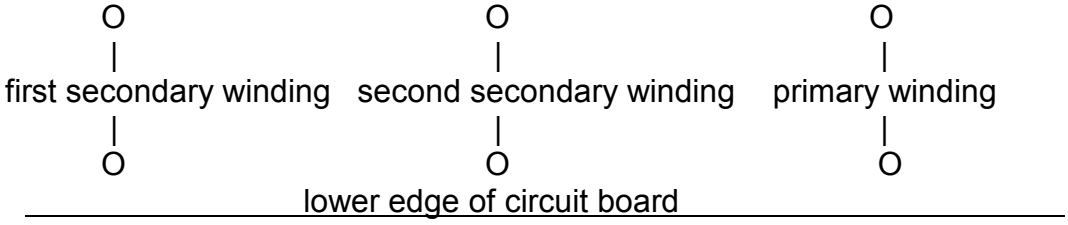
The HC49 crystal mounting location is in the upper right corner of the board and crystals are to mount vertically to the board. A small platted through hole in the lower left corner of the crystal mounting position provides a place for a grounding wire to be soldered to the metal crystal case. The grounding wire also provides additional mechanical support for the crystal. Make sure the crystal is mounted slightly above the board since a trace runs on top of the board from one of the crystal holes to the base of Q1.

Mount the integrated circuits on the bottom side of the board if not already done at an earlier step in the board construction.

Inductors L1 is wound with the specified number of turns of #30 AWG magnet wire on the specified core. Each pass through the center of a core is counted as a turn when winding the inductor. L1 is mounted vertically and supported by its leads. Be sure to remove the enamel coating on the wire before attempting to solder an inductor lead to its associated mounting hole. The enamel coating on the #30 wire provided in the kit does not heat strip very well but may be stripped by use of a small folded over piece of Emory paper where the lead is pulled through two facing surfaces of the Emory paper multiple times to sand off the enamel coating on the wire end.

Transformer T1 is mounted vertically and raised above the board about 1/8 of an inch. In winding T1, first wind the primary winding with #30 AWG 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 just slightly clockwise around the core from where the primary winding starts and ends. After striping and tinning each transformer lead at about 1/8 of an inch from the core, determine the two pairs of leads of each of the secondary windings by use of an ohmmeter.

Mount T1 according to the following diagram where “O” represents a plated through-hole of the T1 mounting location.



Either secondary winding may be designated as the first secondary winding.

When the transformer is positioned vertically 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 790 ohms between each secondary winding plated through-hole and circuit ground. Each of the primary plated through-holes should show about zero ohms resistance to the antenna return hole, (RTN), on the upper right edge of the board.

Carefully inspect all solder connection on the board and touch up any connections that look questionable. Problems in getting a SoftRock to function 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 lower 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 plug connects to the plated through-hole that is marked with the letter T on the board and the ring of the stereo cable plug connects to the plated through-hole marked with the letter R.

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 upper of the two plated through-holes is the antenna return connection to the coax shield and the lower plated through-hole is the coax center conductor connection.

Initial Testing the v6.0 SoftRock Receiver

Make sure the resistance between the input power leads is greater than 1000 ohms before applying DC power to the SoftRock board. Also make sure the resistance from the 5 VDC line circuit ground is approximately 1.2k ohms. 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 C8. 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 porwer to the baord and look for problems such as solder shorts between pads and plated through-holes. The current from the DC supply should less than 40 mA for any of the SoftRock Lite receivers for any band of operation.

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 (or 8) 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.

A couple quick voltage checks are as follows: DC voltages referenced to circuit ground measured at pins 1, 3, 5 and 7 of U5 should all be close to 2.5 VDC. The voltage at the secondary winding connections of T1 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 above and below the period of the “SR 6.2" marking in the center of the board on the top side. The frequency of these square waves should be the same as the center frequency with a 90 degree phase difference between the two square waves when 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.