

SoftRock v7.0 Builder's Notes

May 15, 2006

Be sure to use a grounded tip soldering iron in building the v7.0 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, bill of materials and board information files for the v7.0 SoftRock 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 the SoftRock v7.0 kit.

Fasten the board mounting hardware provided in the kit to the v7.0 PCB with the 3/8 inch length 4-40 screws coming through the four corner mounting holes from the bottom of the board. A 1/8 inch nylon spacer goes on each screw from the bottom of the board. A #4 nylon washer and 4-40 hex nut are used with each screw on the topside of the board. The board mounting hardware will help protect the surface mount components as the board is built.

A total of twelve 0.1 uF 1206 SMT capacitors are soldered to the board, nine on the bottom side of the PCB and three 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 four SOIC integrated circuits, U2, 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 and U3, (both LT1719 SOIC 8 devices), are in a line on the bottom side of the board approximately under where the crystal mounts on the top side of the board. U4, (FST3253, an SOIC 16 device), is located under where T1 mounds on the board top side. U5, (MAX4489 SOIC 8 device), is located near the edge of the PCB. In general, pin 1 on an IC device is at the lower left corner when the IC is viewed with the text on the IC topside 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 the soldering iron 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. 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.

This completes the mounting of parts on the bottom side of the board. Three 0.1 uF SMT 1206 capacitors are next soldered to the top side of the board to complete the SMT soldering on the board.

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 value by making an ohmmeter check before mounting a resistor.

Bend the leads of R2, R14, R20 diode D1 so that one lead of each part bends back with a small radius and is parallel to the body of the part. 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. Orient D1 so that its banded end goes to the square pad on the D1 location on the board. Keep the resistor leads short by lightly pushing the end of each resistor or diode 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 and diode on the underside of the board to hold it in place for soldering. Solder the three resistors and diode from the bottom side of the board and cut off the excess lead with flush cutting wire cutters. (The three resistors and diode will provide “legs” so that the circuit board can sit bottom-side up on a flat surface to make soldering easier.)

Mount the remaining resistors to the board in the same way as the above resistors. The silkscreen marking for R6 and R7 can be misread so note that resistor R6 is closest to the crystal X1 and R7 is just to the right of a 0.1 uF SMT capacitor.

The electrolytic capacitors are next 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 end is connected to the positive via of the capacitor location. Snug up the electrolytic 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. Keep component 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.

If any of the capacitor leads are formed for 0.2 in spacing the leads will need to be carefully

reformed to 0.1 inch spacing. Use long-nose pliers to reform the lead spacing by squeezing on the lead at the bends in the lead to straighten out the lead.

A two-pin header is provided for mounting in the JP1 position along the lower left edge 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. A two-pin jumper is provided to short the header pins for normal operation.

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 X1 location.

Inductors L1 and L2 are wound with the specified number of turns of #26 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 on the board. Be sure to remove the enamel coating on the wire before attempting to solder an inductor leads to the board. The enamel coating may be removed by lightly pulling the wire through a fold of Emory paper or by scraping the wire with a blade edge.

Transformer T1 is mounted horizontally and raised above the board about 1/8 of an inch. 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 four pieces of #30 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. The #30 wire provided in the kit does not heat strip so Emory paper will be needed to sand off the enamel insulation. After tinning each transformer lead at about 3/8 of an inch from the core, determine the four pairs of leads, (pairs A--B through G--H in the diagram below), that form each of the four secondary windings by use of an ohmmeter.

The four secondary winding are shown below where A----B represents one winding with the A end coming out of the top of the core and the B end coming from the bottom of the core. C----D, E----F, G----H represent the other secondary windings.

(top of core) A----- B (bottom of core)
(top of core) C----- D (bottom of core)
(top of core) E----- F (bottom of core)
(top of core) G----- H (bottom of core)

The four secondary windings are correctly connected to the via 3 through via 8 the connections can be represented as shown below.

(via 3) A-----B (via 4) C-----D (via 5)
(via 6) E-----F (via 7) G-----H (via 8)

The primary lead from the top of the core goes to the via marked 1 and the other primary lead goes to the via marked 2. When the transformer is positioned horizontally with its leads to the appropriate vias, solder all the leads and cut leads flush to the bottom side of the board.

Carefully inspect all solder connection to L1, L2 and T1 and touch up any connections that look questionable. Use an ohmmeter to verify that the leads of L1, L2 and T1 connect to the appropriate vias.

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

A stereo audio cable may be connected at this time to the board at the three vias along the upper right edge of the board. Use a short piece of #22 bus wire to connect the middle via (ground) to the shield (barrel) 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 below the grounded via in the group of three vias. The ring connection of the stereo audio cable connects to the top via of the group of three vias.

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

Connect a length of 50 ohm coax to the antenna connection on the lower left edge of the board. The lower via is the coax shield connection and the upper via is the coax center conductor connection.

Initial Testing the v7.0 SoftRock Receiver

Before applying power to the v7.0 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 lead of C2, should be approximately 1.2k. If no power shorts are detected, apply 9 VDC up 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 look for problems such as solder shorts between pads and vias. The current from the DC supply should be about 30 mA.

For the functional check out of the SoftRock v7.0 the Rocky program is recommended. Setup

the Rocky program as listed on the website. Plug the stereo cable coming from the SoftRock 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 RF source of less than 2 mV RMS 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.