

SoftRock v6 RXTX Builder's Notes

October 28, 2006

Be sure to use a grounded tip soldering iron in building the v6 RXTX 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 RXTX board may be down loaded from the v6 RXTX folder within the v6 SoftRock folder of the Yahoo SoftRock-40 group website. These pdf files will be needed during the build of the RXTX board.

Construction of the v6 RXTX board

Use good electrostatic discharge, ESD, precautions in building the circuit board. This includes, if available, a grounded wrist band for the person doing the assembly of the board and a grounded conductive work surface mat.

To minimize the integrated circuits exposure to ESD, do not mount the SMT integrated circuits until other parts are soldered to the circuit board. Transistor Q1 and PA transistor Q2 should be mounted after initial electrical testing of the rest of the circuit board.

Documents required to construct the v6 RXTX board are the schematic diagram, the bill of materials and the board map. The v6 RXTX has many components and requires care in soldering components in place to prevent mounting a component in the wrong location or creating solder bridges between component pads. In some cases the component designators could not be included on the board and such component locations are identified on the board map pdf file.

The first step of construction is to fasten the board mounting hardware to the corner holes of the board. From the bottom-side of the board each corner hardware group consists of a 3/8 inch long 4-40 phillips machine screw, a 1/8 in long nylon spacer, the circuit board, a #4 nylon washer and a 4-40 hex nut.

All SMT capacitor locations on the bottom of the board are to be filled with 0.1 uF 1206 size capacitors. Tack one end of each capacitor to a pad and properly position the capacitor with the tip of the soldering iron and a toothpick. When the capacitor is properly positioned on its pads, solder the other end with enough solder to make a small fillet between the end of the capacitor and the pad. Reheat the first end of the capacitor and add a little solder, if necessary, to make a small fillet at the tacked down end of the capacitor. Excess solder may be removed with solder wick.

Electrolytic capacitors are mounted on the top of the board in all locations shown by a silkscreen open circle marking with a "+" mark indicating the positive terminal wire of the capacitor. Each electrolytic capacitor has a value of 10 uF. (Capacitors C2 and C9 are exceptions and have been replaced with smaller value ceramic capacitors.) Cut excess lead length flush to the bottom of the board after soldering.

Resistors are mounted on the board in a hairpin fashion with the body of each resistor located over its silkscreen circle. The other lead of each resistor goes to the hole that is pointed to by the small radial line on the silkscreen resistor pattern. Mount the resistors with the body of each resistor lightly snugged to the board. If resistors near the corners of the board are first mounted, the mounted resistors form legs to hold the board level when soldering on the bottom of the board.

Mount all resistors, ceramic capacitors and diodes with reference to the board map to help find the component locations. Diodes are mounted in hairpin fashion with the banded end to the square pad. Cut excess lead length flush to the bottom of the board after soldering.

After all resistors, capacitors and diodes are mounted to the board, the transistors, except for Q1 and Q2 may be mounted. Use the body shape silk screen marking to help in proper placement of each transistors.

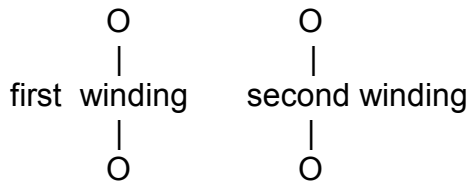
Do a careful visual check of soldering to make sure all component leads are properly soldered and no solder bridges exist between pads. Check for proper orientation of diodes and electrolytic capacitors. Measure the resistance from the DC power in pad to circuit ground and the 5 VDC line to circuit ground and make sure the resistance values are greater than 800 ohms.

The JP1 location needs to have a wire, (short piece of a resistor lead), connected between holes 1 and 2 located on the left hand end of JP1 location. This will set the board operation for divide by 4 to give 40m operation with 28 MHz crystals.

A three-pin header is mounted with the shorter length pins upward and the plastic bar holding the pins together on the top of the board in location JP2. (JP2 is used for jumper plug selection of either crystal X1 or X2.) The two crystals are mounted vertically to the board and two small ground holes near the back edge of the crystal locations may be used to ground the crystal cases with a short length of wire cut from a capacitor lead. The grounding of the crystal cases also provides mechanical support for the crystals.

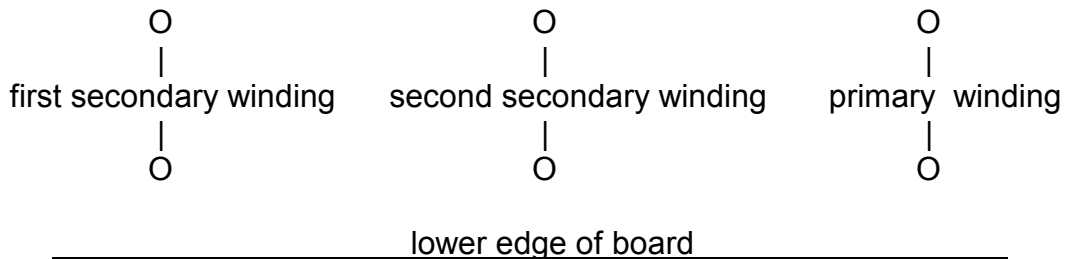
The 28.626 MHz SMT crystal may be mounted by soldering fine leads to each end of the crystal. (Either pad of each end-pair may be used as one connection to the crystal.)

Inductors and transformers may next be prepared for mounting to the circuit board. The following table of lead lengths allows for at least a half inch pigtail at beginning and end



Inductors, L1 through L7, are wound with the core type, number of turns and wire size as listed above and on the schematic. The inductors are mounted vertically or slightly leaning away from other surrounding components. RFC1 and RFC2 are wound on binocular cores with the winding turns listed above and on the schematic.

Transformer T3 is like T1 except the bifilar windings are now the secondary windings. T3 is mounted vertically near the bottom edge of the board as indicated in the diagram below.



When all parts are soldered to the board the SMT integrated circuits may then be mounted on the bottom side of the board. Exercise care in locating the dual op-amp integrated circuits since each of the three op-amps have the same 8-pin SOIC package. U1 and U2, TC2262 devices, are mounted close to each other as indicated on the board map. U8, a OPA2228 or LTC6241 device, is mounted below the crystal location. With each IC mounting location a small "1" in the copper indicates the pin 1 end of each IC.

Do not mount transistors Q1 and Q2 until initial testing of the v6 RXTX board is complete.

Connect stereo audio cables to the line-in and line-out hole groups along the left edge of the board. The middle hole in each group is the ground connection for the cable

ground. The Tip and ring connections of the line-in cable plug connect to the holes marked in the same way. The Tip of the line-out cable plug needs to connect to the hole marked with an "L" and the ring of that cable plug connects to the hole marked with an "R".

DC power, 12 VDC, connects to the board at any of the three adjacent holes as shown on the board map. (A 12 v switched DC output is just above the three input DC power input holes and care needs to be exercised to not connect an external power source to this board output.) DC power return can be connected to any of the three holes above the switched 12v output hole.

The antenna connection to the board is the top hole in the hole group along the right edge of the board and is marked ANT IN. The antenna return connection is just below the ANT IN hole and is ANT RTN.

Initial testing of the v6 RXTX board

Some initial testing may be done at this time if the external cables are connected to the two boards. Make sure the resistance from circuit ground to the DC power in connection and the 5volt connections to any of the ICs have a resistance greater than 800 ohms before DC power is applied to the board. Connect a current limited 12 VDC supply to the DC power-in leads of the board and make sure supply current is less than 50 mA. (If supply current is above 50 mA, disconnect the supply and look for shorts or component orientation problems on the boards.) Check the regulated 5 volt power to any of the IC supply pins to be in the range of 4.8 volts to 5.2 volts.

Receiver operation may be tested by connecting the line-in cable to the soundcard line jack, connect a antenna through a coaxial cable to the board and connect DC power to the board. An SDR program such as Rocky is a good way to initially check receiver operation. With Rocky operation enabled and the board electrically connected as indicated above, 40m signals should be heard and seen on the spectrum display.

PTT input function may be verified by connecting PTT-in line to 12 volts. (Exercise care in connecting 12 volts to the PTT-in line because a connection of 12 volts to the PTT-out line can damage U3.) The receiver should mute with the PTT input high and the switched 12v line should go to about 12 volts.

A quadrature audio source can be used for initial testing of the QSE functions. Quadrature audio can be provided from a PC soundcard line-out if a program such as IQ GEN by DL6IAK is installed on the PC. (Please see

<http://dl6iak.ba-karlsruhe.de/projects/2001-04-28.htm>

to download the IQ GEN program.)

With 5 kHz quadrature audio input to the line-out audio cable of the QSE board, check for DC voltage and an AC voltage at capacitors C9, C13, C18 and C20. The DC value at these capacitors should be about 2.5 volts and the AC component will depend on the amplitude of the line-out voltage and equal to the 20% of its amplitude. If PTT-in is connected to 12 volts, an RF voltage can be observed at the hole for the Q1 base pin. The frequency of the RF voltage should be at the center frequency + or - 5 kHz where the + or - frequency offset depends on the line-out signal phase relationship.

(More testing information to follow as this document receives additional work.)

Please post any problems found in the building or testing of the v6 RXTX board on the SoftRock-40 Yahoo Group website.

Thanks and 73,
Tony KB9YIG