RXTXv6.2 Builder's Notes for 160m Kit

July 31, 2007

Be sure to use a grounded tip soldering iron in building the RXTXv6.2 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 two schematic diagram sheets, BOM (bill of materials) and board map for the RXTXv6.2 80m kit may be down loaded from the RXTXv6.2 160m folder within the RXTXv6.2 folder on Yahoo SoftRock-40 group website. These pdf files will be needed during the build of the RXTXv6.2 circuit board. Other helpful files and pictures will be included in this folder.

Construction of the RXTXv6.2 160m board

Use good electrostatic discharge, ESD, precautions in building the circuit board. This includes, if available, a grounded wristband 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. FET transistors Q1, Q3, Q5, Q6, Q7 and Q8 should not be mounted until later in the board assembly. Also do not mount transistor Q2 until transformer T2 has been mounted to the circuit board.

The RXTXv6.2 board 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. Look carefully to determine component locations and their associated designators. Use good lighting and magnification when building the circuit board.

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. Add a little solder to one pad and tack one end of a capacitor to a pad. Reheat and 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.

Ceramic capacitors are next mounted in all appropriate positions on the circuit board. Capacitor value codes are typically expressed as three digits where the first two digits are the significant figures of the capacitor value and the third digit indicates how many zeros to add to express the capacitor value in pF. Examples would be: the code 471 would indicate a 470 pF capacitor and the code 220 would indicate a 22 pF capacitor.

Lightly snug each capacitor as close to the board as the lead formation will allow. Slightly spread the capacitor leads on the bottom side of the board and solder one lead to hold the capacitor in position. Cut both leads flush to the bottom of the board and solder the second capacitor lead.

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. The longer of the two capacitor leads is the positive lead. Slightly spread the leads of a capacitor to hold it in place and solder one lead. Check that the capacitor has been lightly snugged to the board and reheat the solder if necessary to adjust the position of the capacitor. Solder the second lead and cut both leads flush to the bottom of the board.

Resistors are typically mounted on the board in a hairpin fashion with the body of each resistor located over its silkscreen circle. Resistors R22, R23, R24, R31 and R32 are mounted parallel to the board with their bodies against the circuit board. Care needs to be exercised in mounting resistors R22, R23 and R24 so that the resistor bodies do not encroach on the area reserved for mounting transistor Q2.

Resistors are typically 1% parts with a four color bands to indicate resistance by the standard color code. However, several colors such as brown, red and orange may look nearly the same on the blue bodies of the resistors so it is best to use an ohmmeter to make sure of each resistor's resistance value.

The second lead of each resistor that is mounted in a hairpin fashion 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. Cut component lead flush to the bottom of the board after soldering.

Consult the RXTXv6.2 board map file if there are questions about a resistor's location. Resistors R15 and R16 have been a problem for some so special attention to the lead locations of these resistors is appropriate.

Diodes are mounted in hairpin fashion with the banded end to the square pad. The exception to this is D2 which is mounted parallel to the board with its body against the board. Cut component lead 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, Q2, Q3, Q5, Q6, Q7 and Q8 may be mounted. Use the body shape silk screen marking to help in proper placement of each transistors. U4 may also be mounted at this time. The three hole pattens for mounting each transistor and U4 are closely spaced and careful soldering is necessary to prevent solder bridges between leads and to make sure each lead is properly soldered. Note the silkscreen marking on the circuit board for orientation of each transistor.

Bridge the x8 hole pair, pins 2 and 3, at the JP1 location where the first pin is the square pad. This will set a clock division ratio of eight required for 14.xx MHz crystal use on 160m. Also mount a small wire loop in hole four of JP1 location to use as a ground test point.

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, transistors and electrolytic capacitors.

Measure the resistance from the DC power line to circuit ground and the 5 VDC line to circuit ground. The DC power line resistance to ground may be checked at a through hole near the center of the board marked +12V. The 5VDC line may be checked at a through hole marked +5V, also near the center of the board. The DC power line resistance to ground should be much greater than 10k and the 5VDC resistance to ground should be slightly greater than 800 ohms.

A three pin header is mounted at the JP2 location is used for crystal selection of the two mounted crystals. The header is mounted with the shorter pins upward and the plastic bar holding the pins together on the top of the board.

The two crystals are mounted vertically to the board and slightly raised above the board to make sure each crystal's metal case does not short the leads of the crystal together at the pads. Two small grounded 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. Make the solder connection in grounding each crystal case with as little heat as necessary by attaching the grounding wire to each crystal prior to mounting each crystal. The grounding of the crystal cases also provides additional mechanical support for the crystals.

Transistors Q1, Q7 and Q8 may next be mounted.

Transistors Q3, Q5 and Q6 are mounted with their flat side facing upward so that the entire flat side of each transistor will be under an aluminum TO220 heat sink. The leads of Q3, Q5 and Q6 will need to be bent at right angles away from the flat side of each transistor at a distance from the plastic body of each transistor so that its package end is very close to the heat sink mounting hole in the board. When a transistor is properly placed solder one of its leads from the top side of the board to tack the

transistor in position. After Q3, Q5 and Q6 are tacked in place with their flat sides facing upward, mount the heat sink with the Sil-Pad between the flat sides of the transistors and the bottom surface of the heat sink. The heat sink stack up should be as follows from the bottom of the circuit board: the ½ inch long 6-32 phillips machine screw, the circuit board, the transistors with flat faces upward, the TO220 Sil-Pad, the TO220 heat sink with fins upward, the #6 star washer, and the 6-32 hex nut. Tighten the 6-32 hardware carefully to firmly compress the stack while holding the heat sink in alignment with the rectangular markings on the circuit board. Be careful that the tightening of the #6 hardware does not cause a twisting motion of the transistors. When the transistors are firmly clamped between the board and heat sink with good alignment, complete the soldering of the transistor leads from the bottom of the board and trim the lead flush to the bottom of the board.

Inductors and transformers may next be prepared for mounting to the circuit board. Each pass of a wire through the central hole of a toroid core is counted as a turn. Winding turns count, wire gage and wire length needed, as well as core type for each inductor or transformer is given on the schematic sheets.

Transformer T1 is wound with its secondary winding first wound and then the two primary windings wound bifilar on top of the secondary winding. T1 is mounted vertically on the board near the upper edge of the board. Wires coming out of one side of the core go to the three holes on one side of the six hole pattern and wires coming out of the other side of the core go to the second group of three holes. The diagram below shows the wiring to the six holes associated with the T1 mounting position.

upper edge of circuit board

O------secondary winding------O

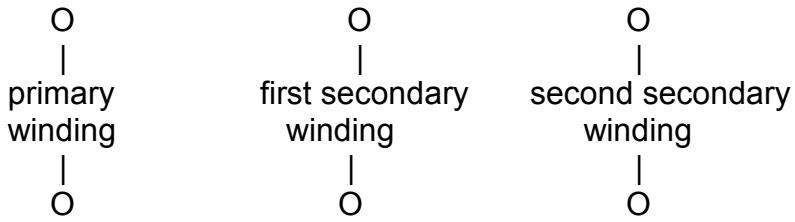
O-----first primary winding-----O

O---second primary winding--O

Transformer T2 is wound on a binocular core with three leads coming out of each hole on the same face of the core. A turn is counted when a wire enters the first hole and comes out the second hole on the same face of the binocular core. Exercise care in winding the binocular cores and direct the wire into each core hole so that the hard material of the core does not scrape off portions of the wire's enamel insulation. (It may be good to lightly spin with one's fingers a small drill bit in each of the binocular core hole openings to remove sharp edges that can cut through the enamel insulation on the wire.) The secondary windings are wound bifilar on top of the primary winding. Use a thin wooden toothpick to lightly compress the windings to get the final turns through the

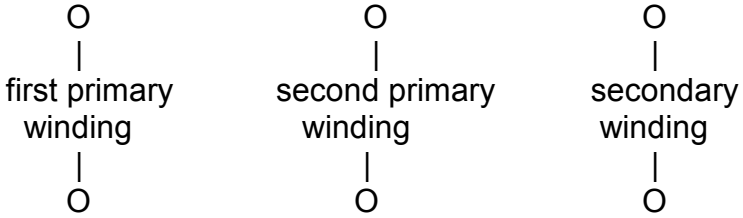
core holes.

T2 is mounted vertically and slightly raised above the circuit board with the three leads from each core hole going to a three hole group in the transformer mounting location. Use a 1/16 inch length of wire insulation on each of the middle leads of a three wire group to help support the vertical binocular core and lessen the chance for shorts between winding ends. Leads from one hole of the binocular core go to the upper set of holes and leads from the other hole of the binocular core to the lower set of holes as shown below.



Transistor Q2 may be mounted after T2 is properly mounted. Press the TO-18 heat sink onto Q2 by placing the top of the heat sink on a hard flat surface and then pressing uniformly on the lower rim of the transistor to fully insert it into the heat sink. A #4 nylon washer is included in the kit as a spacer between the bottom of the transistor and the circuit board. Slip the leads of the transistor through the nylon washer hole and snug the transistor to the circuit board. Exercise care to make sure the emitter lead of Q2 by the metal tab on the transistor case goes to the hole closest to the silkscreen emitter mark on the circuit board.

Transformer T3 is like transformer T2 except the bifilar primary windings go to the two circuit board hole pairs on the left and the secondary winding goes to the pair of circuit board holes on the right. The primary windings are wound bifilar on top of the secondary winding. Again use a 1/16 inch length of insulation on each middle lead. The diagram below shows the T3 connections to the circuit board.

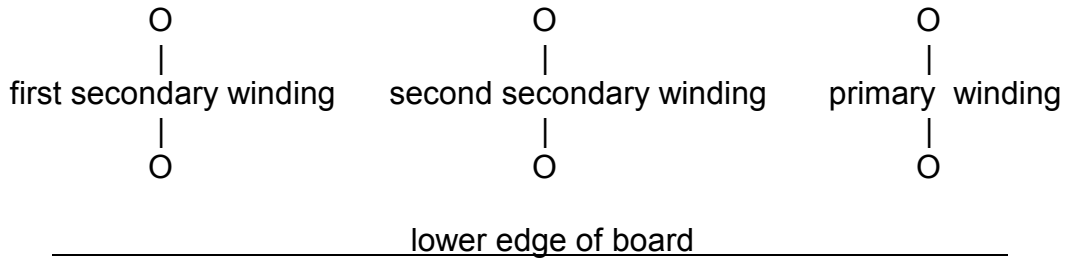


Inductors, L1 through L4, are wound with the core type, number of turns and wire size listed on the schematic. The inductors are mounted vertically or slightly leaning away from other surrounding components. Note the mounting hole pattern on the circuit

board for each inductor and plan the direction of core winding to provide the best orientation of each inductor lead.

RFC1 is wound on a binocular core and mounted vertically and slightly raised above the circuit board.

Transformer T4 is like T1 except the bifilar windings are now the secondary windings. T4 is mounted vertically near the bottom edge of the board as indicated in the diagram below.



When all other parts are soldered to the board the SMT integrated circuits may then be mounted on the bottom side of the board. The ICs in the kit are such that if an IC provided in the kit fits an IC mounting location on the circuit board, then the IC it right for that location. Orient each IC on its pads so that the pin 1 corner of the IC matches the small “1" mark in the copper on the bottom side of the board. In general, pin 1 of an SOIC packaged IC is in the lower left corner of the package when the printing on the package top reads upright, from left to right. Tack solder one corner pin of an IC and when the IC is properly oriented and the pins line up well with the pad, carefully solder the rest of the leads to their pads. Use solder wick to remove any excess solder or solder bridges between IC pins.

Please see the board map drawing for details of external electrical connections to the RXTXv6.2 circuit board.

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 and connects through each cable to the barrel of the associated 3.5mm plug. The tip and ring connections of the line-in cable plug connect to the holes marked Tip and Ring. 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 the PWR + marked hole at the middle of the right edge of the board. The power return lead connects to the PWR - hole just below the PWR + hole

The antenna connection to the board is the top hole in the two hole group above the DC power holes on the right edge of the board. The antenna return connection is just below the ANT hole.

Initial testing of the RXTXv6.2 board

Some initial testing may be done at this time if the external cables are connected to the board. Make sure the resistance from circuit ground to the DC power in connection and the 5volt connections to any of the ICs has 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 the circuit board draws less than 50 mA from the DC supply. (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 at the +5V test point to be in the range of 4.8 volts to 5.2 volts.

Receiver operation may be functionally checked by connecting the line-in cable to a soundcard line-in jack, connect a 50 ohm 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 properly configured and operation enabled along with the board electrically connected, signals should be heard and seen on the spectrum display.

PTT input function may be verified by connecting PTT input line, (I PTT), to12 volts. (Exercise care in connecting 12 volts to the PTT input line because a connection of 12 volts to the PTT output line, (PTT O), can damage U3.) The receiver should mute with the PTT input high and the switched 12v line, (S12v), should go to about 12 volts. With PTT input high the circuit board current demand should be less than 160 mA.

PA standing current is checked by connecting PTT-in to 12 volts and measuring the voltage across resistors R31 and R32. The DC voltage across each resistor should be 55 mVDC +/- 10mVDC.

A quadrature audio source can be used for initial testing of the transmit function. 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 the link below to download the IQ GEN program.

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

Connect the antenna coax cable from the circuit board to a 50 ohm load. With PTT input connected to 12 volts and with 5 kHz quadrature audio input to circuit board from the line-out audio cable, check for RF output across the 50 ohm load as the quadrature audio level is increased from the IQ GEN amplitude slider. The RF voltage across the 50 ohm load should go to 20V p-p, (1 watt output), when the quadrature audio inputs

are each at 2.4V p-p. The RF output waveform across the 50 ohm load should appear as a clean looking sine wave when viewed with a scope. The frequency of the RF voltage should be equal to the center frequency + or - 5 kHz where the + or - frequency offset depends on the line-out signal phase relationship between the two line-out channels.

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

Please post any problems found in the building or testing of the RXTXv6.2 board on the SoftRock-40 Yahoo Group website.

Thanks and 73,
Tony KB9YIG