RXTXv6.1 Builder's Notes

February 10, 2007

Be sure to use a grounded tip soldering iron in building the RXTXv6.1 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 sheets, BOM (bill of materials), board map, and heat sink files for the RXTXv6.1 board may be down loaded from the RXTXv6.1 folder on Yahoo SoftRock-40 group website. These pdf files will be needed during the build of the RXTXv6.1 circuit board.

Construction of the RXTXv6.1 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, Q2, Q6 and PA transistors Q3-Q4 should be mounted near the end of the board assembly.

The RXTXv6.1 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 except for the locations C75, C76 and C77 which are omitted. 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. Silkscreen marking for electrolytic capacitors C28 through C37, near the center of right half of the board, have not been placed on the board but those locations each are to be filled with a 10uF electrolytic capacitor. Cut component lead 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. Cut component lead flush to the bottom of the board after soldering.

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 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, Q4 and Q6 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.

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, (pin 14 of U5 for example), to circuit ground and make sure the resistance values are greater than 800 ohms.

The JP1 location is filled with a three-pin header where the first header pins go to the square pad at the left end of JP1. The header is mounted with the shorter pins upward and the plastic bar holding the pins together on the top of the board. The forth hole of JP1, at the right end, is a grounded via that can be used as a test point. Solder a short lead to the grounded via for a convenient circuit ground test point. Silkscreen marking at JP1 identify which two pins need to be bridged by a two-pin shorting plug for clock division by 4, x4, or clock division by 8, x8. For example: on 40m the 28.224 MHz crystal frequency is divided by 4 to give a center frequency of about 7.055 MHz.

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 slightly raised above the board to make sure the crystal case does not short the leads of the crystal together. Two small grounded holes near the

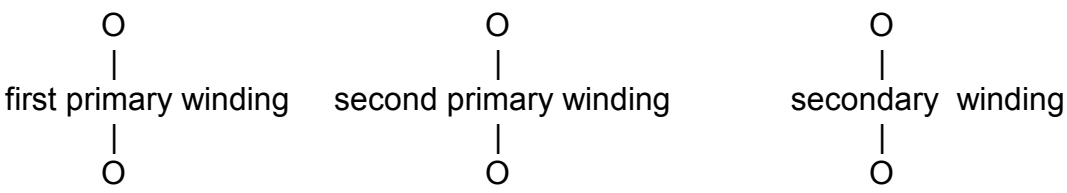
back edge of the crystal locations may be used to ground the crystal cases with a short length of wire cut from a capacitor lead. The grounding of the crystal cases also provides mechanical support for the crystals.

Transistors Q1 and Q6 may now be mounted. **Very important is the mounting of a 0.1uF SMT capacitor on the bottom of the board from the gate pad (middle pad) of Q6 to the grounded pad of capacitor C40.**

Transistors Q3 and Q4, the FETs used in the PA, are mounted with their flat plastic side facing upward so that the entire flat side of each transistor will press against the brass washer used for PA heat sinking. The leads of Q3 and Q4 will need to be bend at right angles at a distance from the plastic body of each transistor so that each end of a transistor case is very close to the central 1/4 inch diameter hole in the board. Use the 1/4-20 brass hardware, without use of the Sil-Pad, to hold each transistor in place for soldering of the leads. After the Q3 and Q4 are soldered in place, remove the brass hardware and then remount the brass hardware with the Sil-Pad between the flat sides of the transistors and the bottom surface of the brass washer. Tighten the brass machine screw while holding the brass hex nut and washer to prevent rotation of the nut and washer until the transistors are firmly compressing the Sil-Pad material. (Note: the 1/4-20 brass machine screw dome head will need to be filed flat so that the height of the head is less than 1/8 inch. Also the Sil-Pad should be trimmed to the OD of the brass washer and the brass washer should be sanded to present a flat face toward transistors Q3 and Q4.)

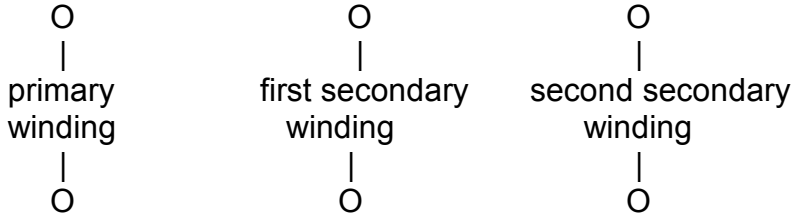
Inductors and transformers may next be prepared for mounting to the circuit board. Each pass of a wire through the central hole of a toroid core is counted as a turn. Winding turns count and core type for each inductor or transformer are given on the schematic sheets.

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 away from the edge of the board wires coming out of the other side of the core go to the three holes closest to the edge of the board. The diagram below shows the wiring to the six holes associated with the T1 mounting position.



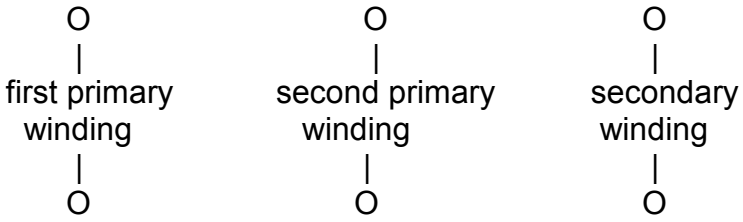
Transformer T2 is wound on a binocular core with three leads coming out of each hole on the same face of the core. A turn is counted when a wire enters the first hole and comes out the second hole on the same face of the binocular core. Exercise care in winding the binocular cores and direct the wire 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 1/8 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 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 leads toward the board and going to the nearest hole as shown by the mounting diagram below. 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. Exercise care to make sure the emitter lead of Q2 goes to the hole closest to the silkscreen emitter mark on the circuit board. Q2 is mounted raised above the circuit board enough that the top of the heat sink clears transformer T2.

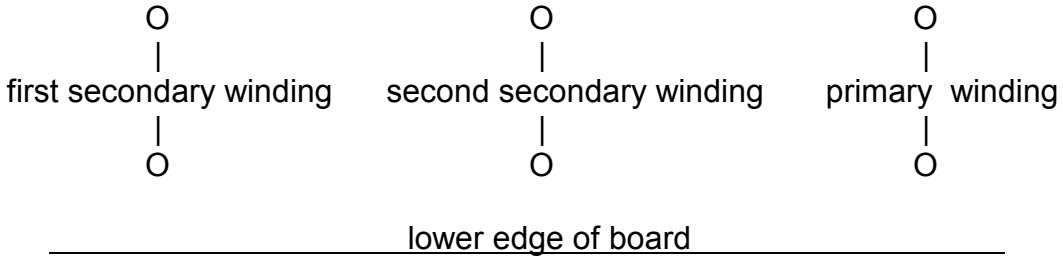
Transformer T3 is like transformer T2 except the bifilar primary windings go to the two circuit board hole pairs on the left and the secondary winding goes to the pair of circuit board holes on the right. The primary windings are wound bifilar on top of the secondary winding. The diagram below shows the T3 connections to the circuit board.



Inductors, L1 through L6, are wound with the core type, number of turns and wire size listed above and 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 leads.

RFC1 is wound on 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. 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. The IC 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 to 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, solder the rest of the leads to their pads.

Please see the board map drawing for details of external electrical connections to the RXTXv6.1 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 to the through the each cable to the barrel of 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 above the PWR + hole

The antenna connection to the board is the top hole in the two hole group near the upper right corner of the board. The antenna return connection is just below the ANT hole.

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 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 to any of the IC supply pins 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 operation enabled and the board electrically connected, signals should be heard and seen on the spectrum display.

PTT input function may be verified by connecting PTT-in line to12 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. With PTT-in high the circuit board current demand should be less than 80mA where pot R29 is rotated fully CCW.

Adjust the PA standing current by connecting PTT-in to 12 volts and measuring the voltage to ground at the top of resistors R30 and R31. Rotate pot R29 CW until the voltages at R30 and R31 are each around 50mV DC, +/- 15mV of each other.

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-in 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 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