QRP Labs QCX+ 20-meter CW Transceiver Build

Figure 1 - QRP Labs QCX+ 5W transceiver

I got the bug to get back into the build shop for a bit, and I settled on the QRP Labs QCX+ CW Transceiver (Figure 1) in the 20-meter flavor as my build of choice. At the same time that I ordered this kit, I also ordered their 50W RF Power Amplifier kit – that will be another article when I get around to building that kit.

The QCX+ is a compact transceiver offered in bare-board kit form with the exception of the two SMT IC's that are pre-soldered to the main printed circuit board (PCB). The heart of this radio is the ATmega328P microcontroller (see schematic diagram at Figure 2), accompanied by the Si5351 programmable clock generator and the FST3253 dual 4:1 multiplexer/demultiplexer bus switch, the two of which are the pre-soldered SMT IC's. I should mention that for an additional cost, the radio is also available factory-built and tested.

It almost goes without saying that this is a software-defined radio, though it does use a conventional low-pass filter in its design. The power amplifier is a grouping of three BS170 N-channel enhancement mode MOSFET's, which are heat-sinked in a rather unique manner, which we will discuss later. These three transistors are arranged in a Class "E" power amplifier arrangement. In addition to the three transistors used in the PA, three additional BS170 MOSFET's are used as well as a single MPS751, bringing the total discrete transistor count to seven transistors.

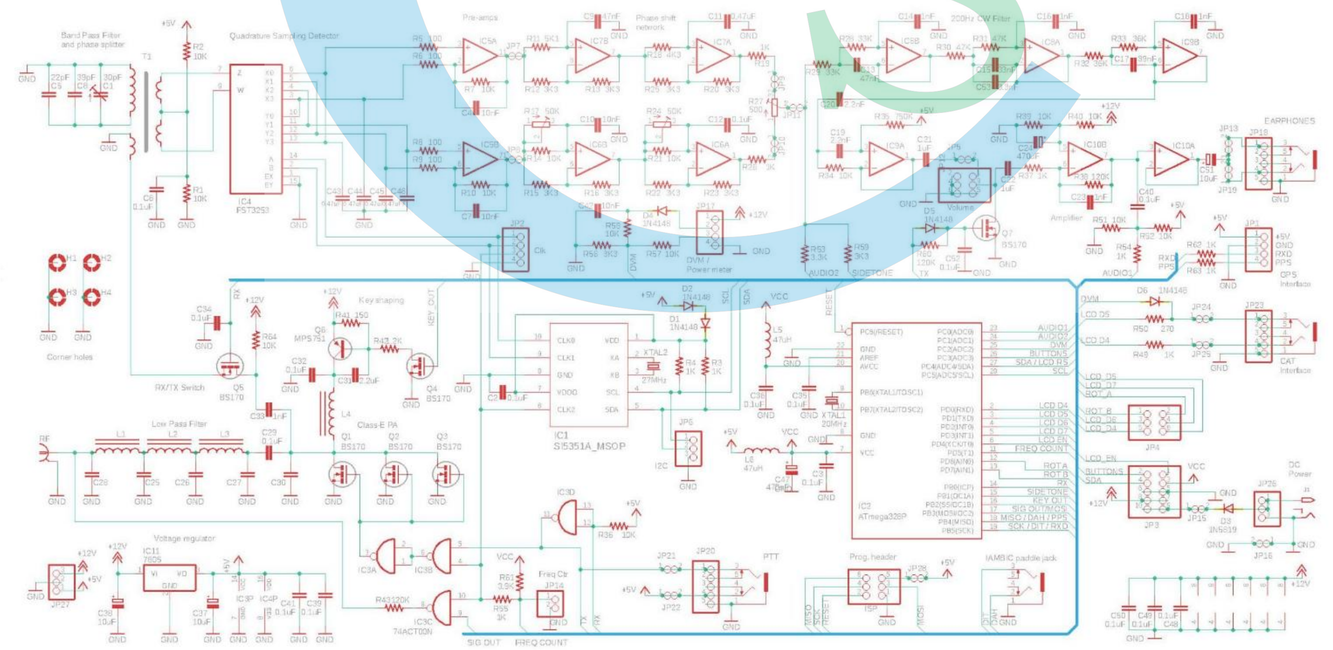

Figure 2 - Main PCB schematic diagram

This build requires the winding of a total of five toroids, one of which is a four-winding transformer. The radio is equipped with an ISP header for easy programming of firmware updates, and alignment is accomplished via the adjustment of one trimmer capacitor and three multi-turn potentiometers, a relatively simple though tedious procedure. In extreme cases, the alignment procedure may require modification of one or more of the toroids wound during the build, a circumstance largely driven by toroid winding technique and component tolerances.

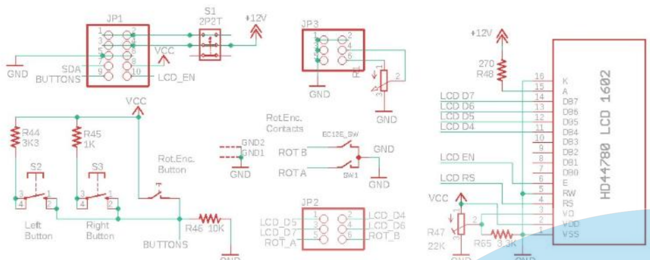

Figure 3 - Front panel schematic diagram

This radio has a lot to offer in terms of features, including the ability to store up to twelve canned messages for transmission, the first four of which can each be up to one hundred characters in length, while the remaining eight messages are limited to fifty characters each. Radio functions and features are accessed and adjusted through a nested hierarchal menu system which is in turn accessed through the front panel (see schematic diagram at Figure 3) controls, which include two pushbutton momentary switches ("Select" and "Exit") and a rotary encoder ("Adjust" and "Save"). The Operation Manual (linked later) includes a convenient "cheat sheet" that describes the nested menu system in sufficient detail for the average user to work with the menu comfortably.

The build is pretty straight-forward. I found it easiest to simply follow the assembly manual steps in their published sequence, even though I am extensively experienced in kit building. The manual sequence just makes sense, and it breaks the build down into logical steps that actually make the task simpler. For example, all of the resistors of a given value are installed together as a single step, instead of jumping around from one value to another and then back to the first again, as is often done in kit build instructions. The manual, available online at qrp-labs.com/images/qcxp/manual111.pdf, uses a white/grey/red color scheme in the PCB diagrams for each step, to indicate uninstalled components, previously installed components, and those components being installed in the current step, respectively. This makes it very easy to locate and identify the correct location for each and every component being placed.

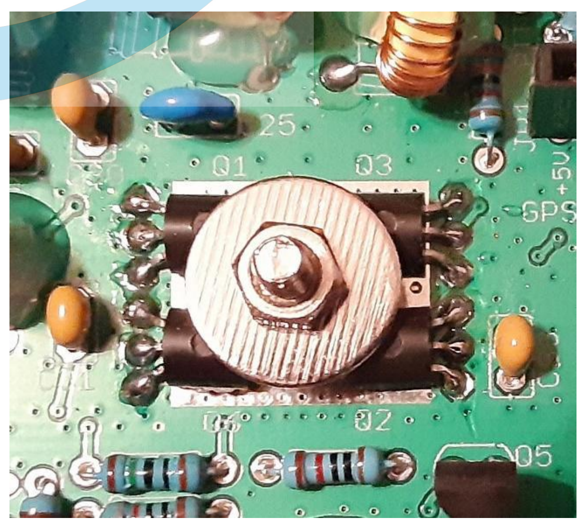

Figure 4 - PA transistor heat sinking arrangement

I will not bore the reader with a step-by-step litany of the build process, but I do want to talk about a few interesting points. Firstly, only the µC IC is socketed; all of the other IC's are soldered directly to the PCB. Next is the arrangement of transistors that form the power amplifier. In this build, a total of four transistors are installed to the PCB (three BS170's and the single MPS751) in an arrangement where the transistors are installed in two opposite pairs. The transistors are bent down with their flat sides flush against the PCB, in a large tinned area of the PCB that serves as a heat sink pad (Figure 4). The center of this pad has a hole drilled through it, around which the transistors are arranged and must remain clear. A machine screw is installed into this hole from the foil side of the PCB, a steel flat washer is placed onto the screw and down on top of the transistor bodies, and the whole arrangement is secured by a hex nut on the machine screw. This clamps the transistor bodies tightly to the PCB heat sink pad, allowing them to transfer their heat into the PCB. I added a touch of thermal compound to the transistor faces to facilitate better heat transfer, though that process is not mentioned in the manual.

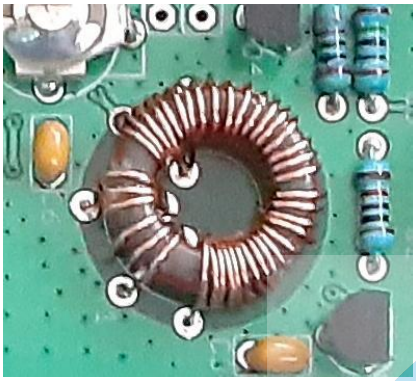

Figure 5 - Toroidal transformer

The other point I want to mention is the toroidal transformer with the four windings. In the case of my build, as the band selected was 20 meters, this transformer called for one thirty-turn winding and three three-turn windings. The transformer is then installed flat onto the PCB (Figure 5) with each of the winding leads inserted into the proper hole in the PCB. This was a bit fiddly to manage, but it was certainly doable. The four remaining toroidal inductors are all single windings of various differing turns counts, and they are installed upright on the PCB. I reinforced their positions with spot of hot glue on each toroid once installed and aligned. As always, the single most important point when winding toroids – or any inductor for that matter – is to properly remove the enamel coating from the wire in the area where the wire will be soldered. Failure to do so will cause open circuits and operational problems.

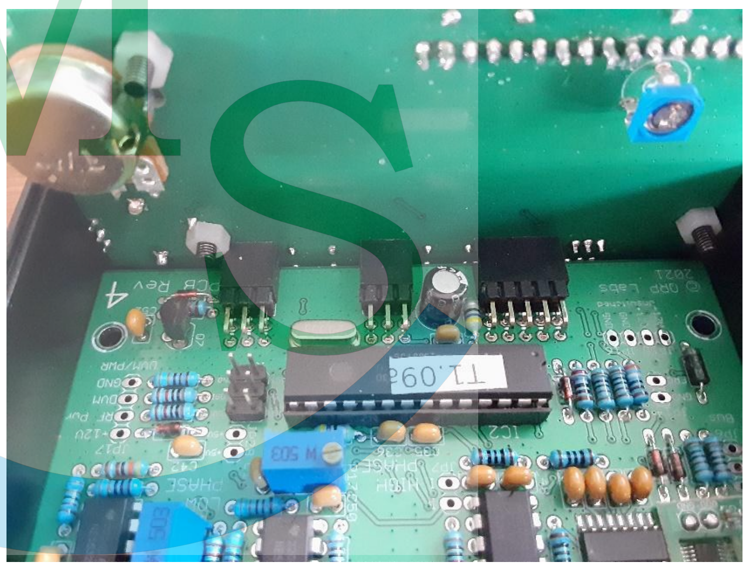

Figure 6 - Board interconnection

This radio is basically a two-board radio, with a main PCB carrying most of the working circuitry, and a front panel board carrying the display panel and the operating controls. In addition to those controls already described, there is a 10kΩ potentiometer installed as an audio gain control. At the lower left is a DPDT latching pushbutton switch used as the main power switch for the unit. The display, a 16x2 character blue and white LCD panel, is installed to the front side of the front panel board, while a 22kΩ contrast control potentiometer for the display is mounted to the back side of the board. Also installed to the back side of the front panel board are three dual-row pin sockets, one 2x5 socket and two 2x3 sockets (Figure 6).

Along what becomes the front edge of the main PCB is a series of three right-angle double-row pin headers, oriented so that they are pointing towards the board edge. These pin headers are intended to engage the pin sockets installed to the back side of the front panel board as the boards are assembled to each other.

Along the opposite edge of the main PCB, which becomes the rear edge of the PCB (Figure 7) when installed, are the DC power jack, the RF output jack, and a set of four 3.5mm TRS jacks designated for the PTT, CAT, paddle, and audio functions. In between the DC power jack and the RF output BNC jack is the +5VDC voltage regulator, a standard LM7805 three-pin IC in a TO-220 package. This IC stands upright on the PCB. More about that later.

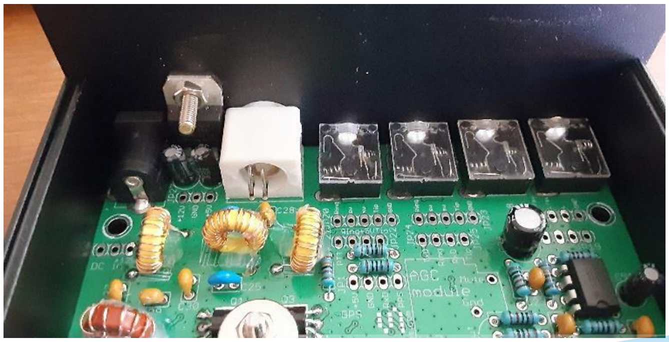

Figure 7 - Rear edge connections

Final assembly of the radio can go a couple of different ways. I chose to purchase the custom enclosure for the kit as offered by QRP Labs, so my finished radio is quite smart and looks like a factory-built unit. The enclosure consists of two identical extruded clamshell halves, which become the upper and lower halves of the enclosure, and two pre-drilled and labeled end-plates, which tie the whole assembly together.

The first step of the final assembly involves installing the front panel board to the front end-plate. This can be quite finicky because of some close tolerances. In point of fact, I had to modify the panel slightly by enlarging the pre-drilled hole for the power switch, as the switch button would not clear the edge of the hole when assembled. I did this using my tapered reamer from both sides of the plate, touching up the bright aluminum hole edge with a black Sharpie® marker to hide the modification. Another modification involves the nylon nuts used as spacers in the board sandwich that makes up the front panel. The kit includes four black 3mm x 20mm flat-head machine screws, four 3mm elongated nylon hex nuts, and eight standard 3mm nylon hex nuts. The assembly has the four elongated nylon hex nuts used as spacers directly behind the front end-plate – between the end-plate and the display board. Four of the standard nylon hex nuts are used as spacers (Figure 8) between the display board and the front panel board. The remaining four standard nylon hex nuts are used to secure everything. Of course, the 3mm x 20mm flat head machine screws are what all of these nuts are installed to. The problem is that it is extremely difficult to assemble these parts with the nuts, as the threads become problematic when the nuts are used as spacers. The solution, as suggested in the assembly manual, is to drill out the elongated hex nuts and one set (four pieces) of the standard hex nuts to remove the threads from the inside of the nuts. I did this using a 1/8" twist drill. Then, the screws will slide right through the nuts and the nuts will become proper spacers. The undrilled hex nuts are installed onto the four screws once the screws have all been inserted through the stack of boards and spacers.

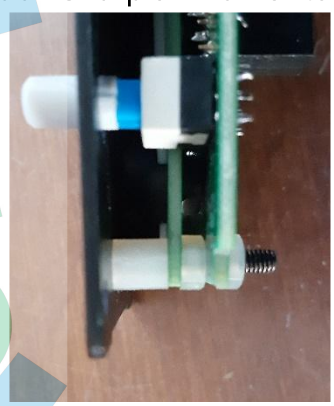

Figure 8 - Nuts as spacers

Figure 9 - Front panel assembled

With the front end-plate installed to the board, the knobs can be installed onto the audio gain potentiometer and the rotary encoder (Figure 9). Then, simply plug the front panel board onto the pin headers along the front edge of the main PCB, and then slide the whole board assembly into the provided groove in one of the clamshell halves. Secure the board assembly with two of the 3mm x 6mm black flat-head machine screws provided.

Install the rear end-plate to the rear end of the lower clamshell half again using two of the 3mm x 6mm screws, being sure to first remove the nut from the BNC connector, but leaving the lock washer in place on the connector boss. After the end-plate is secured to the lower enclosure half, install the nut back onto the BNC connector, and then install the provided 3mm Phillips round-head machine screw through the hole provided in the rear end-plate and through the tab on the LM7805 5VDC voltage regulator, securing the tab to the rear end-plate with the 3mm hex nut provided (Figure 10). This provides heat-sinking for the voltage regulator IC. Apply the four self-adhesive rubber feet provided to the underside of the lower enclosure half.

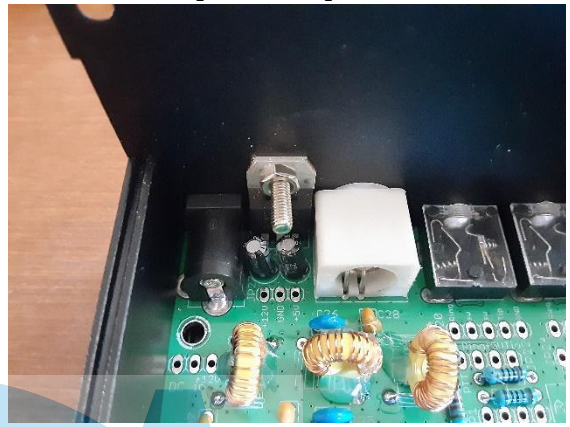

Figure 10 - Voltage regulator secured to rear end-plate

At this point, the radio is ready for alignment and testing. As expected, always use a dummy load when testing and aligning the radio. Connect the radio to a 12VDC supply using as 5.5mm x 2.1mm barrel connector with a center positive wiring scheme. A supply source of 12VDC is adequate for the alignment process. The alignment procedure is fully explained in the assembly manual, and is actually a quite simple procedure, though some of the adjustments can be somewhat touchy. Once the alignment is completed, simply install the upper enclosure half, securing it with the remaining 3mm x 6mm flat-head machine screws. The build is complete.

The QCX+ will operate from a wide range of voltages, but 13.8VDC is the recommended maximum. Excessive supply voltage will result in overheating of the PA transistors, as it will cause increased output power to be generated within the radio.

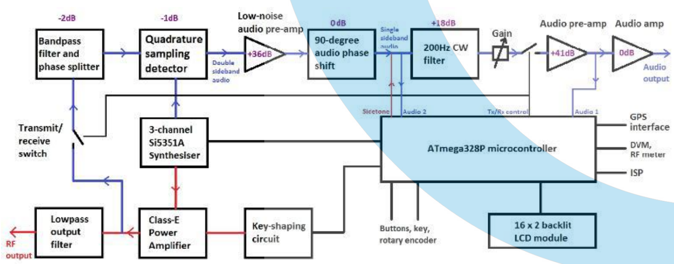

Figure 11 - QCX+ block diagram

On first power-up, the unit should prompt the user to select the band for which the unit was built. However, I did not receive that prompt, with the radio automatically locking in on the 160-meter band. In order to correct this problem and to be able to properly align the radio, it was necessary to do a "Factory Reset" of the radio. This is done quite easily by launching the menu, done by pressing the Menu button for a long press. Once the menu has opened, navigate to Menu 7, then to Menu 7.11 – Factory Reset. Select that menu item, and then use the rotary encoder to display "17" on the LCD panel. Press the select button to start the reset. All user settings will be deleted.

QRP Labs publishes a second manual besides the basic Assembly and Operation manual, available at https://qrp-labs.com/images/qcxp/firmware/1.08/OpMan108.pdf. While this manual is not specifically written to the current firmware in my radio, it is nonetheless quite useful. The current firmware is v1.09a, with the biggest difference apart from bug fixes being support for the 160-meter band. This makes the v1.08 manual quite applicable, and on its first page it is referenced for the v1.09 firmware as well as for the v1.08 release. One of the handiest features of this manual is the Menu Cheat Sheet found at the back of the manual, as mentioned earlier.

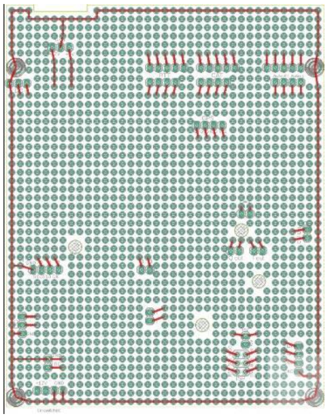

Figure 12 - Development Board

I also got some mileage out of the radio's block diagram (Figure 11) as provided in the Assembly and Operation Manual. The diagram was well drawn with the only obvious shortcoming being that one must intuit the location and connection into the system of the transmit & receive antenna. Now, any reasonably knowledgeable ham will understand exactly where the antenna must reside, but it would have been nice to include at least its connection point to aid the less experienced or knowledgeable among us. The block diagram makes it clear exactly what the signal path is like on both the transmit and receive chains through the radio, which is, after all, the intent of such a diagram. My compliments are due to the fact that the diagram makes the entire frequency synthesis operation more easily understood.

Several optional accessories are offered for the QCX+, including a development board (Figure 12), a GPS module, a TCXO, and an AGC control. I had ordered all of these except the AGC, which I went back and ordered after the fact. Due to the simplicity of the QCX+ design, it is an easy task to install the AGC after the fact, and then to make it active via the menu.

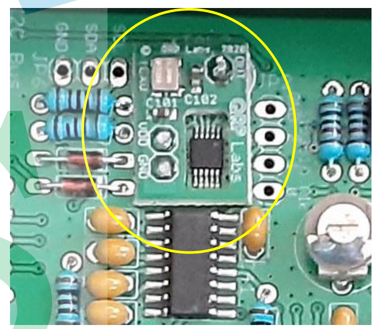

Figure 13 - TCXO installed

If installing the TCXO accessory (Figure 12), one crystal and one capacitor are omitted from the standard build sequence, as their replacements are on the tiny TCXO PCB. This PCB has a cutout to fit around IC1, the SMT Si5351 integrated circuit. The TCXO is connected to the main PCB via three bare wire leads that feed through aligned holes in the two boards, and it sits down directly on top of the main PCB. When the TCXO is in use, it is necessary to go into the radio menu system and set the reference frequency to 25.000MHz instead of the default 27.004MHz, as the frequency of the TCXO differs from the standard reference frequency.

Figure 14 - Audio amplifier circuit

The only factually weak point in the documentation, if there is one, is in the schematic diagram. First off, the main PCB schematic and the front panel schematic are drawn as two separate diagrams. However, while some component ID's are a continuation from the main PCB number sequences, others are not. This results in the existence of, for example, an R1 on both the main PCB and on the front panel PCB. Further, the schematic fails to identify certain component values, with a case on point here being the audio gain potentiometer, R1 on the front panel PCB.

There is sufficient height in the enclosure to install the development board with accessory circuits and still close the enclosure. The development board is connected to the main PCB via a set of pin headers and pin sockets, making it in essence a stack-on shield. Note that in the illustration of the development board at Figure 12, red lines indicate which holes in the board are connected to which other holes. One of the things that I added via the development board is an audio amplifier and an internal speaker, which through the pin header connections associated with each of the TRS jacks on the main PCB, was a seamless add-on. I designed a simple amplifier circuit using the LM386 audio amplifier IC and a handful of external components, all of which were installed to a small area of the development board, as shown in Figure 14. Note the two-pin header for the speaker connection near the amplifier IC.

I plan to add a battery system to the development board next for full portability of the radio. I will most likely install a pair of nine-volt snap-top batteries in series with an adjustable voltage regulator fixed to output 13.8VDC for the radio power feed. This will involve the use of an LM317 voltage regulator IC and some additional external parts, including capacitors for current purity and precision resistors to fix and hold the output at the desired level.

Let's take a quick look at that concept. Refer to Figure 15 for the location of the two resistors. The arithmetic provided by the manufacturer in the LM317 datasheet and used to calculate the required resistances for the LM317 looks like this, using a 1.25V reference voltage:

$$V_{OUT} = V_{REF} \times \left(1 + \frac{R_2}{R_1}\right),$$

or

$$V_{OUT} = 1.25 \times \left(1 + \frac{R_2}{R_1}\right).$$

Substituting our design goal of 13.8V into the equation, we get this:

$$13.8 = 1.25 \times \left(1 + \frac{R_2}{R_1}\right).$$

Re-arranging to get rid of the multiplication on the right side, we get this:

$$\frac{13.8}{1.25} = \left(1 + \frac{R_2}{R_1}\right).$$

Working out the division and removing the parentheses gives us:

$$11.04 = 1 + \frac{R_2}{R_1}.$$

Subtracting "1" from each side of the equation gives us the following:

$$10.04 = \frac{R_2}{R_1}.$$

Let's round it down to an even "10" and insert the factory suggested value of 240 ohms for R_1:

$$10.00 = \frac{R_2}{240}.$$

Multiplying both sides of the equation by 240, we get the following:

$$10.00 \times 240 = R_2 \ldots$$

…which means that R_2 is equal to 2400 ohms. All of this, when applied to the design of the voltage regulator circuit, will give us the schematic shown at Figure 15.

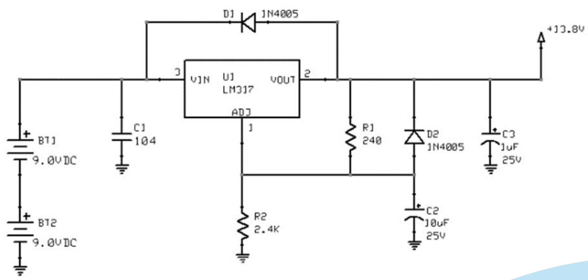

Figure 15 - Battery and voltage regulator circuit schematic

All in all, this was an interesting and enjoyable build. The radio works as advertised, with about a 5.3W maximum power output when supplied with 13.8VDC, slightly higher than the output of the sample radio used for the illustrations in the assembly manual. That graph (Figure 16) shows a power output of about 4.8W at 13.8VDC supply voltage. Current draw is about 125mA on RX and about 310mA on TX. The factory recommendation is not to operate at power levels in excess of 5W because excess heat and stress to the PA transistors will result when that power level is exceeded. I am OK with being 0.3W over that limit. Infrared thermal measurements of the PA transistors, of course with the enclosure open, came in at about 85°C across the securing steel washer during extended CW transmit operation into a 50Ω dummy load. Actual operating temperatures will be slightly higher due to the closing of the enclosure and also due to the placement of the development board above the main board. Key clicks are negligible, and key-down envelope rise times are on the order of just under 6mS as viewed and measured on my Siglent CML1102+ DSO.

I find the QRP Labs documentation to be fairly complete and mostly easily understood. I found the assembly instructions, as stated earlier, to be especially well-organized and professionally written. However, there are some shortcomings, and at least one big "plus". The basic Assembly and Operation manual includes an extensive voltage reference table, providing the voltages expected to be measurable at various points in the radio. That is the "plus". One of the major shortcomings is the fact that, despite a statement to the contrary, CAT information other than the CAT connector pinout data is *not* included in the manual. In fact, the user must look in the separate Operation Manual linked earlier to find the CAT reference details. In addition, there is a fairly comprehensive QCX troubleshooting guide available online at https://www.qrp-labs.com/qcx/qcxtrouble. That page has a link to a downloadable .PDF version of the web document. This guide was written to the earlier QCX design level and not specifically to the QCX+, but most of the material is applicable and will therefore be helpful to the builder in trouble. I would like to have seen one *complete and exhaustive* manual for the radio instead of the "two-manual plus a web page" system that is in place right now. On the whole, however, they have done an admirable job with this product, and I would happily recommend it to anyone who has an interest in kit-building a QRP CW machine.

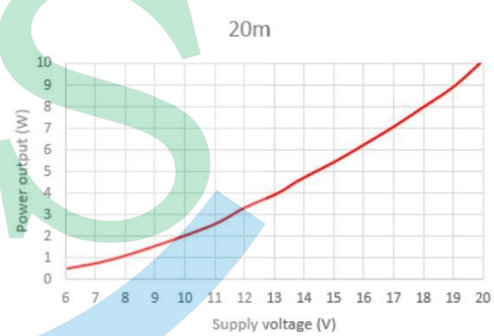

Figure 16 - Power output vs. supply voltage