Notes On Building the Iowa QRP 10 Transceiver
By *Jim Kortge*, K8IQY

My first step in building this transceiver was to do a somewhat quick inventory of the parts Doug, KI6DS had sent. After looking at these goodies, I proceeded to cut the sides down on a small cardboard box for making a parts holder. This technique was used back in the days of assembling Heath Kits, and still works well today, although the parts back then were a bit larger. In some instances with the smaller parts like diodes, several were bundled together and inserted in a single hole, to keep them from disappearing down one of the "cardboard tunnels".

Once the parts were organized, my attention was focused on preparing the printed circuit board substrates that the three section of the transceiver would be built upon. The board material used was some older, phenolic based 1/16-inch double-sided board stock that was waiting to be used. I decided to use this instead of glass based board material, as it is easier to shear, and does less dulling of the cutting edge on the shear. I marked the board sizes at 2 X 5 inches, giving myself an extra inch of width, just in case I needed more room. One of the important axioms of Manhattan-style construction is to use a larger substrate than you think you will need. It's always easy to hack off the extra, and really difficult to add a piece and have it look good, if you run out of building space before the parts are gone.

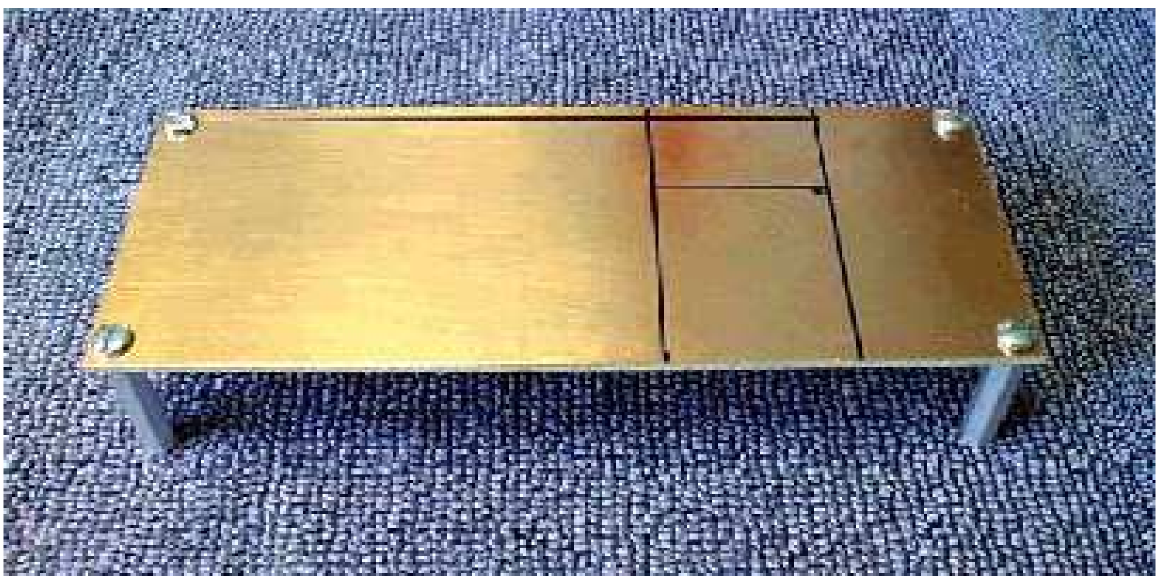

Fig. 1 VFO Board Substrate (1" oversized)

When the boards were cut to size, one of them was marked off for building the VXO, +8 volt supply, and optional TiCK keyer. Additional lines were drawn for the final cut at 2 X 4 inches, assuming all the parts would fit, and another line along the edge, 1/8 inch inboard, so that area would remain clear of parts and wiring. This was to allow a final cut for fitting the board into the supplied case, if indeed it is used. Mike, N0MF's boards apparently were 1 7/8 X 4 inches, but the QRPp illustrations show 2 X 4 inches. I believe the boards must be less than 2 inches wide to fit into the "Bud" minibox Doug provided.

Construction started with the +8 volt power supply, building it into the area marked on the substrate. It is always a good idea to start a Manhattan-style building project with a simpler part of a circuit. Even if you are an experienced builder, it takes some "parts orientation noodling", lead bending, fitting, and soldering to get your mind and hands into the flow. Once you're in the groove, building is much like a chess game, and you can see the next several steps that will happen. After completing the +8 volt supply, it was tested by applying 13.8 volts to the appropriate pad, and measuring the output. My regulator was putting out 7.96 volts. A note of caution, don't inadvertently short the output of the 8-volt regulator, they don't like that. I did it, and didn't let smoke out of the part, but I turned it into a 7.5 volt regulator after doing it a second time. It had to be replaced!

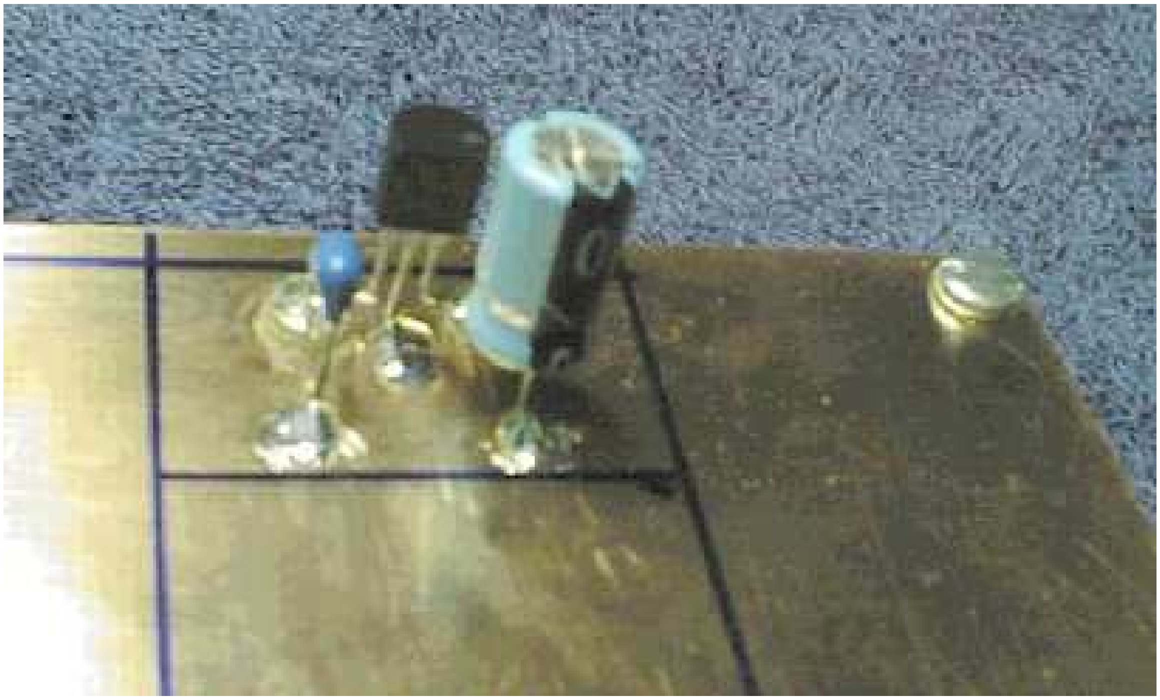

Fig. 2 U6 and associated components.

Since I built the +8 volt regulator farther to the right edge than Mike, I gained some space on the substrate that could be used for the VXO circuitry. VXO construction began by mounting Q4, oriented with the drain lead up, the gate lead down, and the source to the left. My basic plan was to have the frequency control components, X5 and RFC5, mounted across the shorter direction, and the diode doubler components along the longer direction. This would minimize crowding, and keep the frequency control components away from the higher-level r.f. output of the doubler. An extra pad was used between the gate lead and the crystal, just in case an extra capacitor was needed to set the VXO high end. [I've done this on other VXO based projects and was glad that I did. It's one of the benefits of many building projects I guess.] A 3.3 uH-molded inductor was used for inductor RFC5 instead of the T50-2 toroid. Having a large core inductor suspended by its leads could cause unwanted stability problems. The molded unit was mounted between two pads. A larger value inductor should work in this VXO for wider frequency coverage, but won't because of some design limitations.

Fig. 3. Q4 and components.

Before testing could proceed, the MV209 varicap was tack soldered to the pad containing the junction for switch S1. The control point at the end of R11 was grounded with a clip lead. Power was applied to the 12-volt terminal, and the VXO came to life. With the S1 control point at ground, the varicap is at maximum capacitance, and the output frequency was around 15.995 MHz, about where it needs to be to cover 28.06 MHz. What math was just done? Well, the VXO frequency is going to be doubled, so 15.995 MHz turns into 31.99 MHz. From this number, subtract off the i.f. frequency of the rig, or 3.9315 MHz to get 28.0585 MHz. How do I know that the i.f. frequency is 3.9315 MHz? The answer to that question will be addressed in detail later, when a “quick tutorial” on crystal matching takes place. For now, accept the fact that the crystals specified have their series resonance at nominally 3.9315 MHz. Getting back to testing, removing the S1 jumper from ground, and connecting it to the +8 volt supply raises the VXO up about 6 KHz, so 28.060 will be somewhere in the lower part of MV209 varicap tuning range. As the voltage is increased on a varicap, its capacitance decreases. To get a feeling for the total range covered by the VXO, a 470 pF capacitor can be placed across the MV209 leads while the S1 junction is at +8 volts. This capacitor will simulate the maximum capacitance of the 1SV149 varicap, and we’ll get a peek at the VXO low end. On my rig, that will be around 28.055 MHz, actually not as wide of a frequency swing as I had anticipated since the crystal is at 16 MHz.

Building the diode doubler circuitry began by winding the T3, T37-61 core with 10 bifilar turns. Since this core was wound with #28 gauged wire, the recommended method would be to twist the two strands together for 7 to 8 turns-per-inch of length, and wind the core with the two wire bundle. However, I didn't do it that way, but instead wound the 10 turns with parallel wires. I can do that more rapidly, and the results are virtually the same, if the leads are kept together. Once T3 was mounted on a line with the Q4 source lead, the remaining parts were soldered in approximately the same locations that Mike had used. However, I oriented the output transformer, T4 at 90 degrees to T3, to minimize any chance of coupling between these two. Output transformer T4 was wound with a 16-turn primary, so its inductance would be the same as many of the other tuned circuits in the rig. For reference, the primary inductance was measured at 0.9 uH. Making this change allowed using a 5-50 pF trimmer capacitor at location C33, making this value common with all of the other trimmers. The secondary of T4 uses 2 turns instead of 3, to partially compensate for the fewer primary turns.

Fig. 4 VXO Board Layout

With this circuitry completed, applying 12-volt power produced nominal 32 MHz output from the VXO. Trimmer C33 had two relatively sharp peaks, and no instability was seen. The unloaded output is a clean sine wave, at a level of approximately 800 millivolts peak-peak, more than enough signal to drive the receive and transmit mixers. The 16 MHz fundamental signal driving the diode doubler is more than 30 dB down from the main signal at 32 MHz, and all harmonics and spurious signals are also better than 30 dB down.

Fig. 5 View of completed VFO

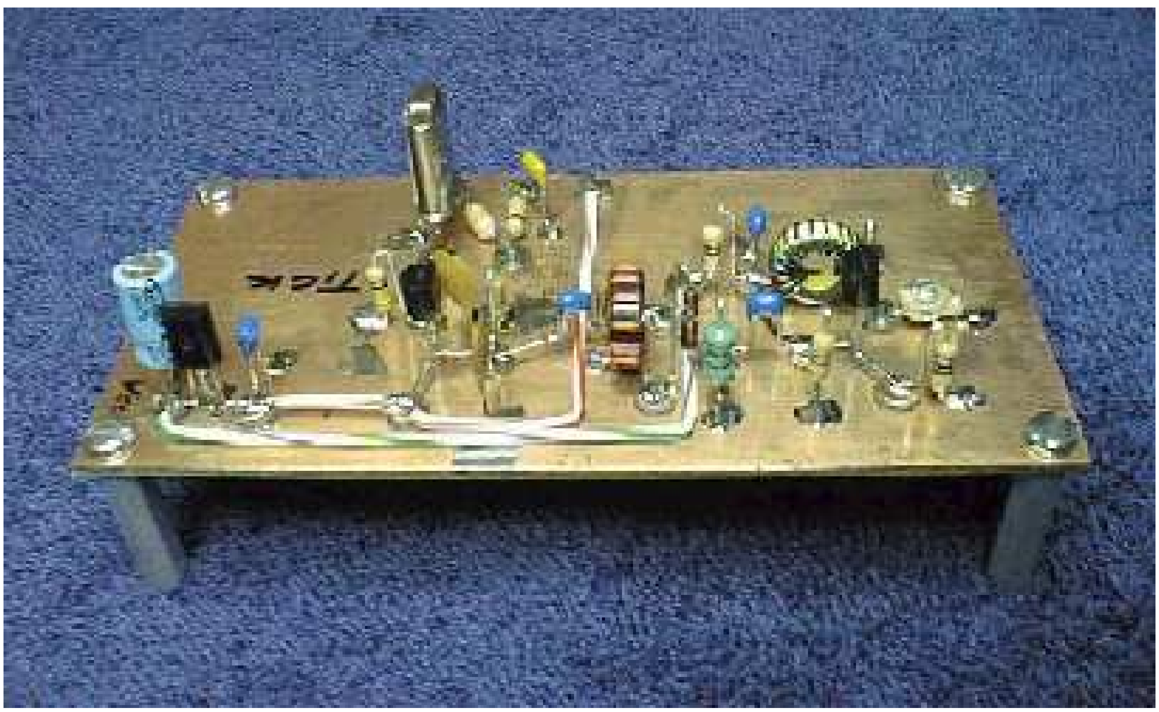

Fig. 6 Reverse View of VFO

In this section, I'll discuss my experience building the input circuitry of the IA QRP-10. I'll refer to this portion as the Rx T/R Switch, as that is its function. It takes the incoming signal from the antenna, via the transmit low pass filter, and passes it on to the receive r.f. pre-amplifier. Part of the functionality of this section is to also protect the pre-

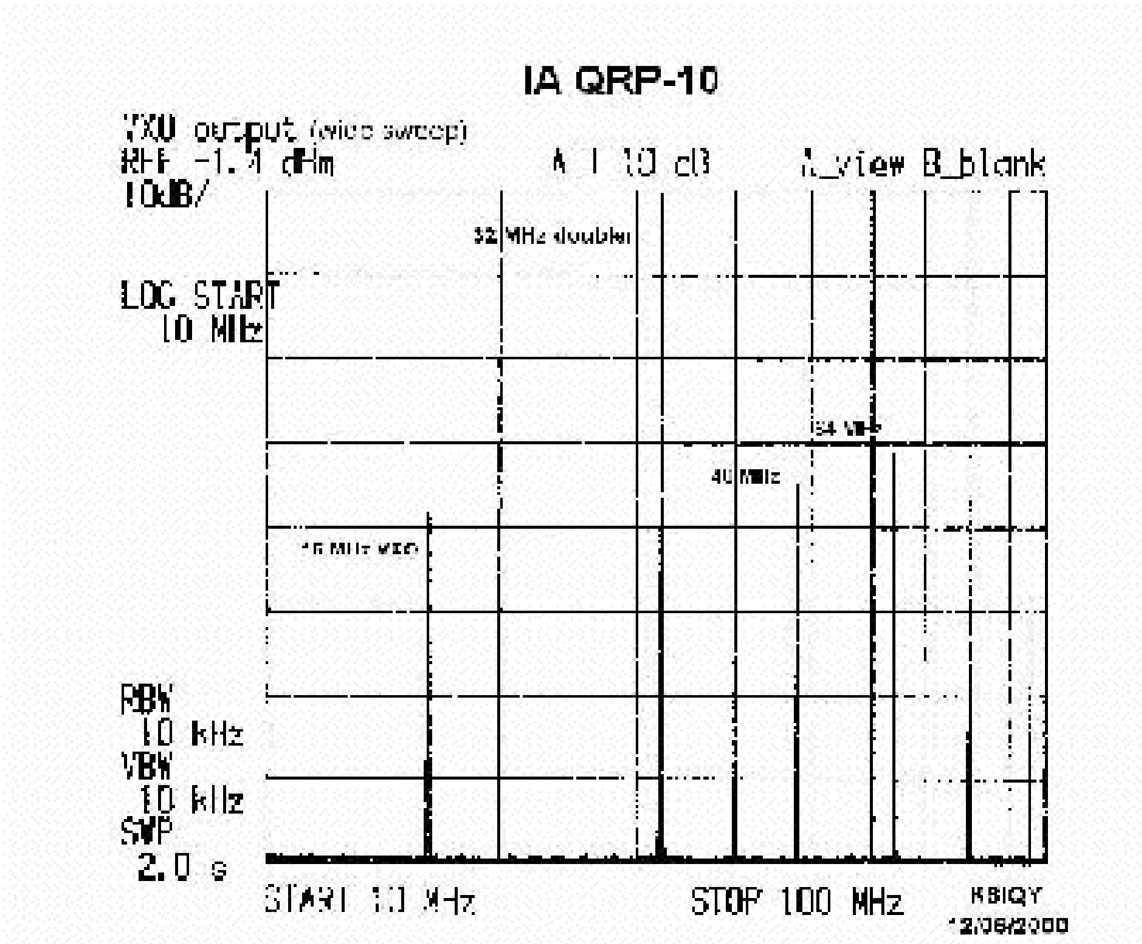

Fig. 7 VXO output spectrum plot

amplifier from being burned out when the transmitter is "on the air". Mike, N0MF essentially carried over the original SST front-end, but in doing so missed an important aspect of Wayne Burdick, N6KR's original design intent. Wayne used the bias voltage from the NE602 input pin to supply current to the MPN3700 PIN diode during transmit. In addition, the input circuitry worked directly into the 1500-ohm impedance of the NE602. That allowed the incoming signal to be properly coupled with a 5 pF capacitor. The input circuitry employed in the QRP-10 does not provide a suitable bias source for the MPN3700 PIN diode, nor is the 5 pF coupling capacitor appropriate for working into a 50-ohm impedance. Some quick Electronic Workbench modeling shows that using the original input circuitry would result in a signal loss in excess of 25 dB, far more than the 8-10 dB gain provided by the pre-amplifier in the next stage. In addition, when the key line is grounded, no change in the signal level occurs. With that knowledge in hand, I felt compelled to make some minor changes to the configuration, and achieve better performance.

The final design kept both the 1N914 and MPN3700 diodes, and added an input trimmer capacitor, 1.5 uH molded inductor, 1.5 K ohm resistor, and another bypass capacitor. Included with the set of pictures for this write up is a revised schematic diagram showing what was built, along with two spectrum analyzer plots of the new Rx Input T/R Switch. Overall, the performance is markedly better. The measured signal loss is now only 0.4 dB during receive, and the attenuation during transmit is over 60 dB, well in excess of the amount needed to protect the pre-amplifier from being overloaded or burned out.

With a new input design in hand, the next step was to mark off the receiver substrate into sections, as was done with the VXO. To reiterate, partitioning the substrate lets you visualize the amount of space available for each section of the design. With some experience, you can sense whether the allotted areas are compatible with the complexity of a section and the number of parts to be soldered down. This substrate was essentially split in half, and each half was then partitioned into thirds, except for the input T/R and pre-amplifier, which shared a larger area. One of the pictures shows the partitioned substrate. Once this layout was done, there was an immediate concern that building the receiver on a 2 X 4 inch substrate wasn't going to work; that all the parts wouldn't fit with ease, and everything would be jammed together. When the VXO was built, the pads used were 5/32 inch in diameter. I placed several pads of this size on the receiver substrate, and immediately decided that this size pad was too big. Using a pad of this size would cause me to run out of room before all the parts were down. With that thought in mind, the dies in the Harbor Freight hand punch were changed to 1/8 inch, and several dozen pads of that size were made. This size still had more than enough room for several connections, and could be placed closer to each other, thereby allowing greater parts density, but without over crowding. Once this decision was made, actual building commenced.

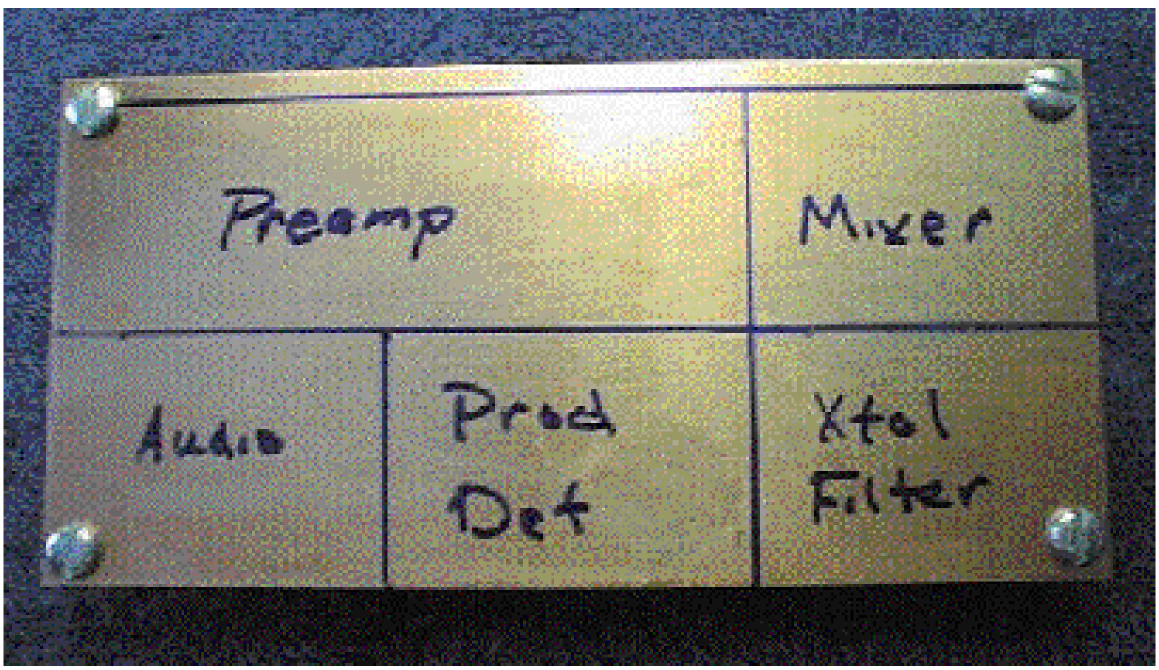

Fig. 8 Receiver Board Substrate

Several pencil and paper sketches were doodled showing possible parts arrangements for the Rx T/R switch before any pads were glued down. On the third or forth layout, everything jelled, and I was ready to build. The first five pads were placed,

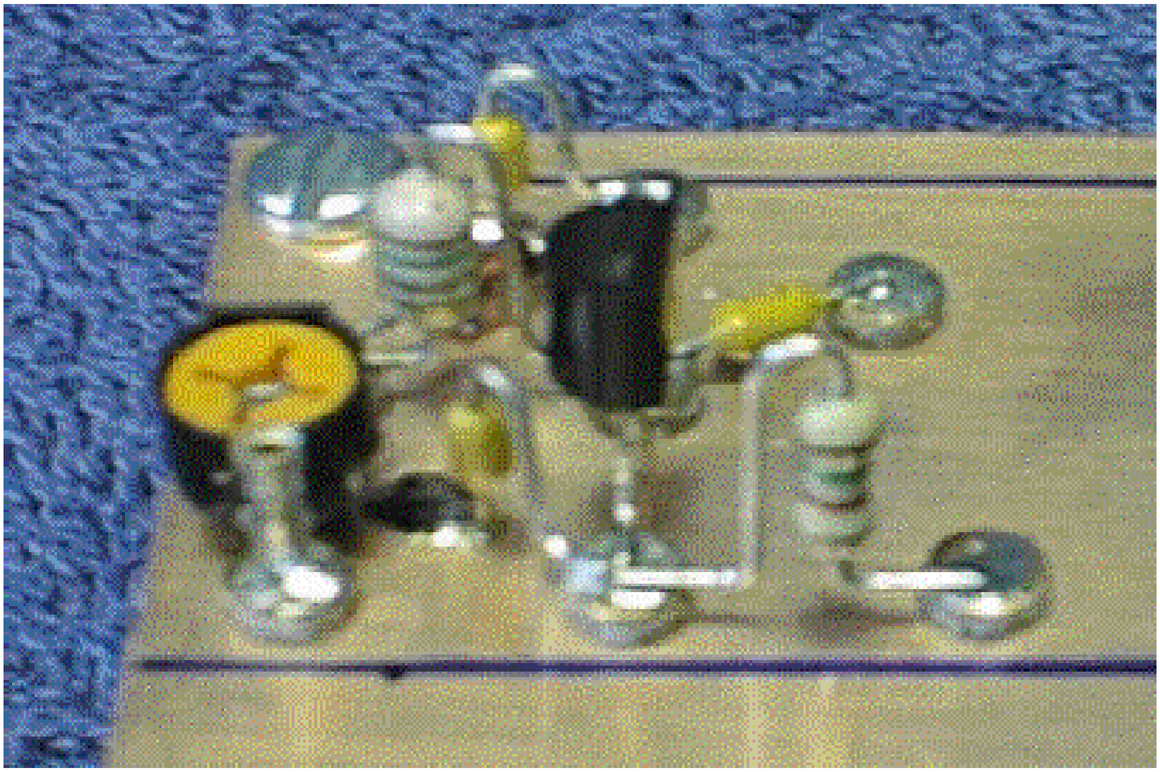

Fig. 9 RX T/R Switch

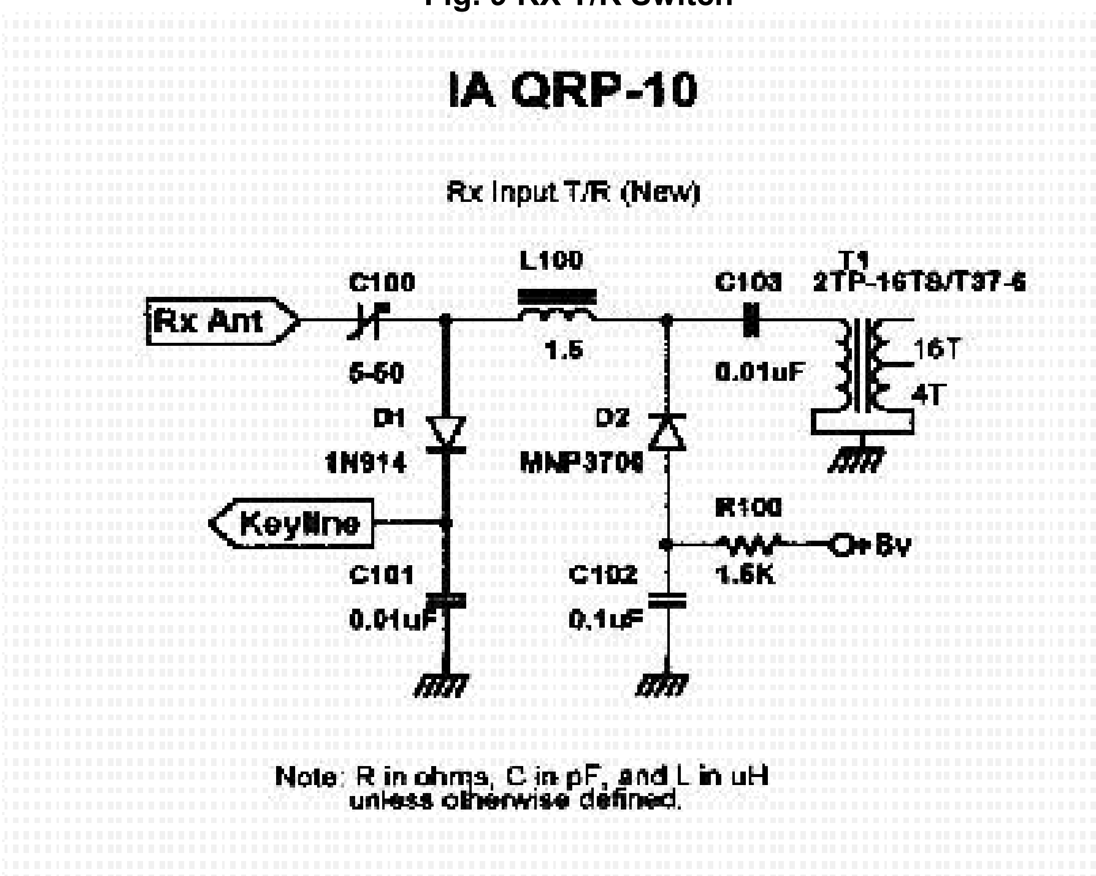

Fig. 10 Replacement T/R Switch Schematic. Note: R in ohms, C in pF, L in uH unless otherwise defined.

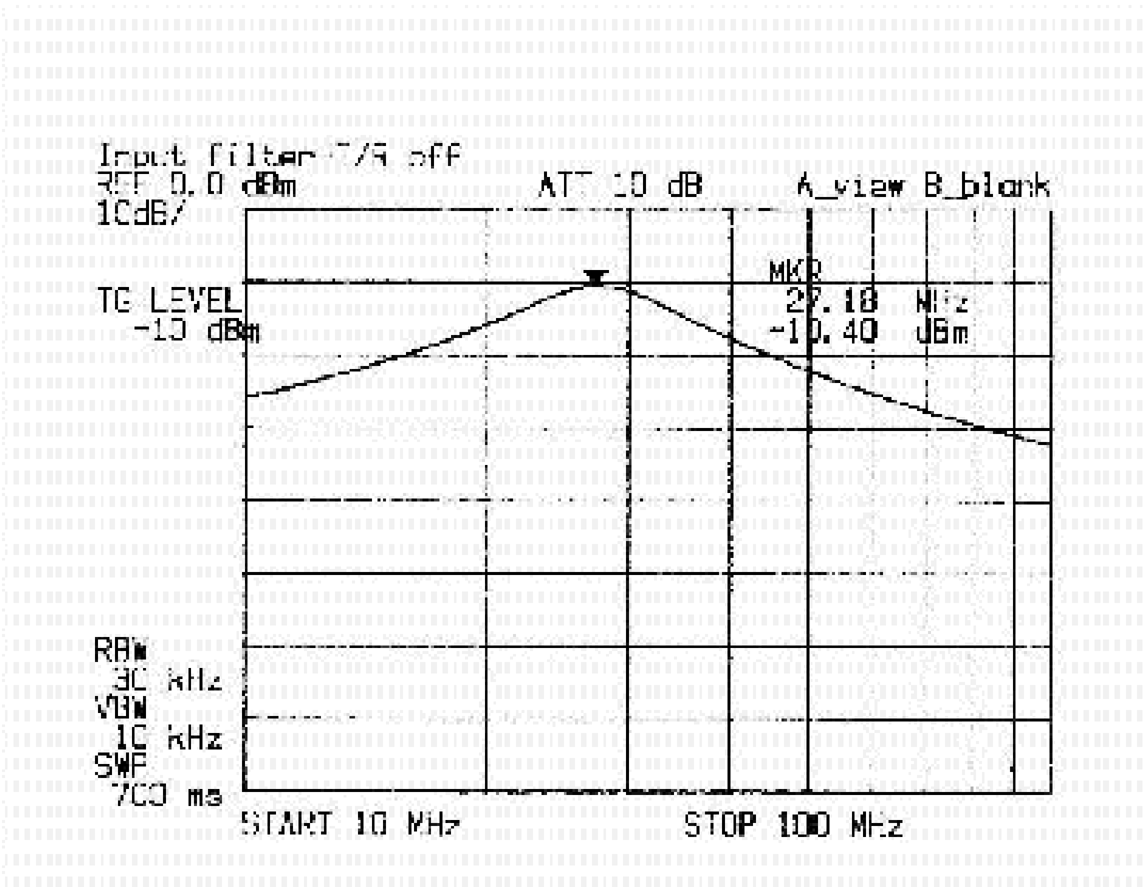

Fig. 11 Rx T/R switch "rcv mode" spectrum plot

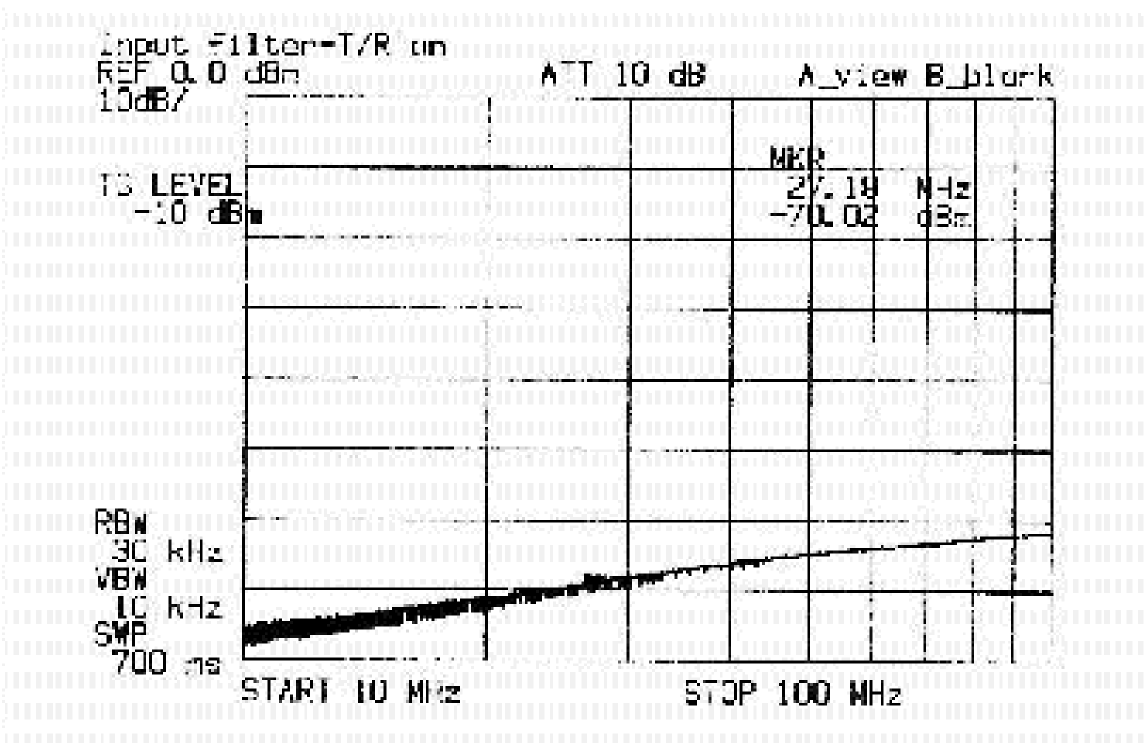

Fig. 12 Rx T/R switch "transmit mode" spectrum plot

and parts were soldered in, starting with the input trimmer capacitor, then the molded inductor, etc. After approximately half of the parts were mounted, the remaining two pads were cemented down, and those parts soldered. With all the parts on the substrate, the layout was given a final check against the schematic. There have been many times when I've built something entirely different than what was shown on the schematic. It happens to all of us; your brain gets tired, and you wander off to "Solder Land".

Once the input circuitry was finished, a few minutes of testing on the spectrum analyzer confirmed the robust performance of this new circuitry. Two spectrum analyzer plots have been included showing how well this circuitry works during receive and transmit.

Continuing construction of the rig began with winding transformer T1 with the appropriate number of turns. Since the input circuitry was changed to work with a low impedance (50 ohms), the primary winding on T1 was changed from 8 turns down to 2 turns. With the secondary tap at 4 turns, this presents approximately 200 ohms to the source of FET Q1, right in line with the input impedance of a grounded gate design using a J310. The transformer was wound with #26 gauge wire on the secondary, then the two turn primary winding was added using #28 gauge wire. As the secondary was being wound, after 12 turns were applied, a large loop was formed, twisted several times, and an addition 4 turns applied. The loop became the tap shown in the T1 detail picture. My

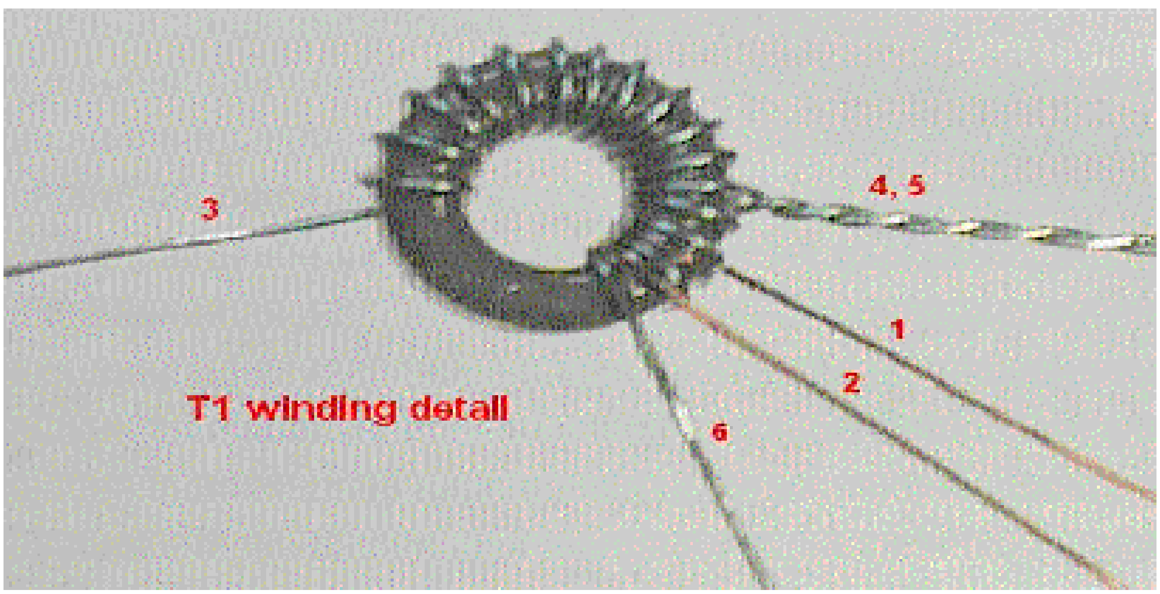

Fig. 13 T1 Winding detail

usual winding technique is to hold the core in my left hand, and wind the wire by pulling it through the core from the front side, while moving around the toroid in a clockwise direction. As it turn out, the transformer would have fit a bit better had I held the core in my right hand, and wound in a counter-clockwise direction, to place the winding ends on opposite sides of the core from where they ended up. I could have rewound T1, but just couldn't bring myself to spend the extra time for saving about 1/8 inch of space on the substrate.

Transformer T1 was mounted nearly identical to the position Mike, N0MF used, as was the trimmer capacitor, C3. The source lead resistor and capacitor, R1 and C4 respectively, were then soldered in place. A pad was placed for the drain lead for Q1, and it was then soldered into place. At this point, had I been thinking, resistor R2 should have been added, along with a pad for its opposite end. However, I was thinking about the need of winding T2, and completely forgot to put it in. Fortunately, the pre-amplifier is very stable without it, so its need may not be very great.

Winding transformer T2 is very straight forward, as it doesn't have the complication of a tap. It was wound with 16 turns on the primary, and 2 turns on the secondary, instead of the 8 shown in the design. My rationale for making this change is that the output of T2 is driving a rather low impedance load consisting of a series resonant LC made up of capacitor C7 and inductor RFC1. RFC1 is r.f. grounded at one end by capacitor C9, so the total impedance seen by the secondary winding of T2 is the parallel loss resistance in inductor RFC1, and that's probably a few ohms at best. With the 4:1 winding ratio on the original T2 design, the transformed load to the drain of Q1 is 16 times the parallel loss value; probably no greater than 50 to 100 ohms at best. This low impedance would destroy the selectivity on the output side of the pre-amplifier. By using an 8:1 turns ratio, the selectivity is markedly improved. No attempt was undertaken to optimize the turns ratio on T2. It may well be that a 1 turn secondary is better than the 2 turns that I chose.

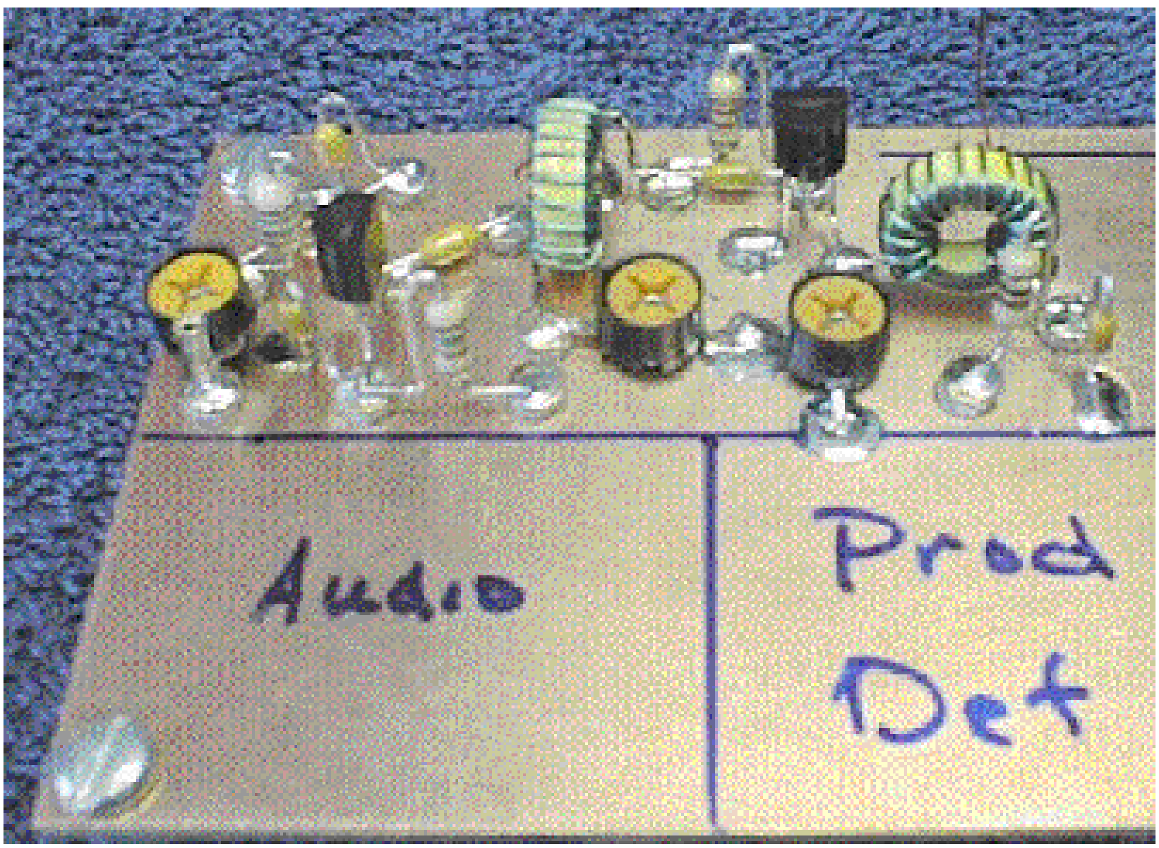

Fig. 14 Receiver Pre-Amp

After T2 was wound and installed, the completed receiver front-end was tested. In the first test, a signal from the tracking generator of the spectrum analyzer was connected to the "Rx Ant" terminal, and the input to the analyzer was connected to the secondary lead of T2. All trimmer capacitors were peaked twice for optimal signal amplitude. The spectral plot shows the overall gain to be about 8 dB, so the pre-amplifier is running at

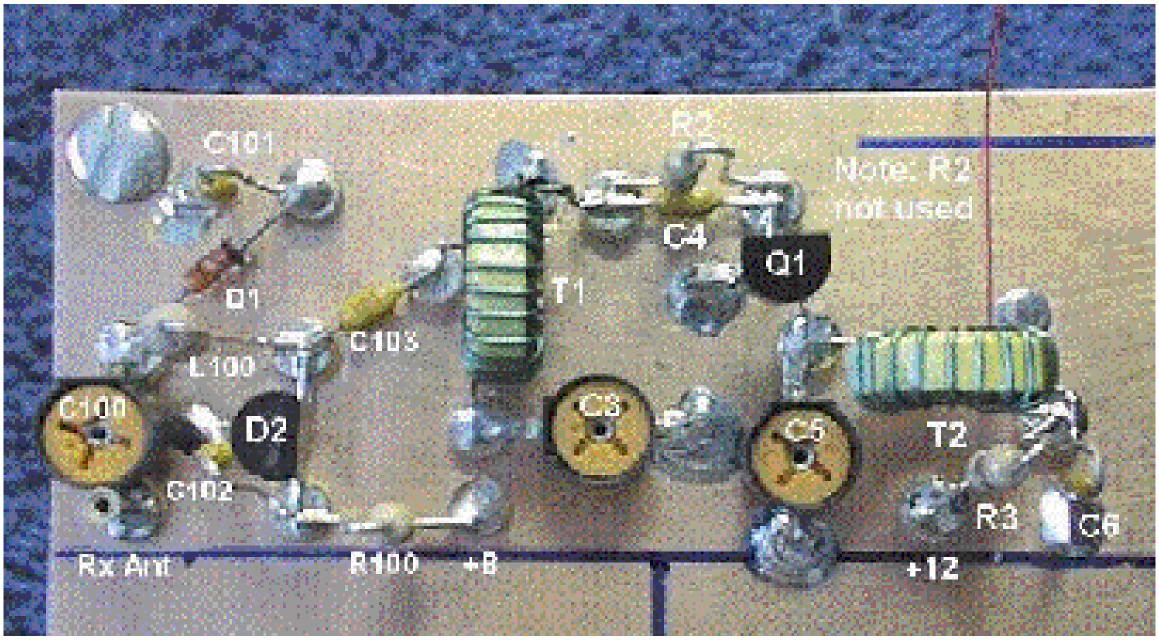

Fig. 15 Receiver Pre-Amp Layout

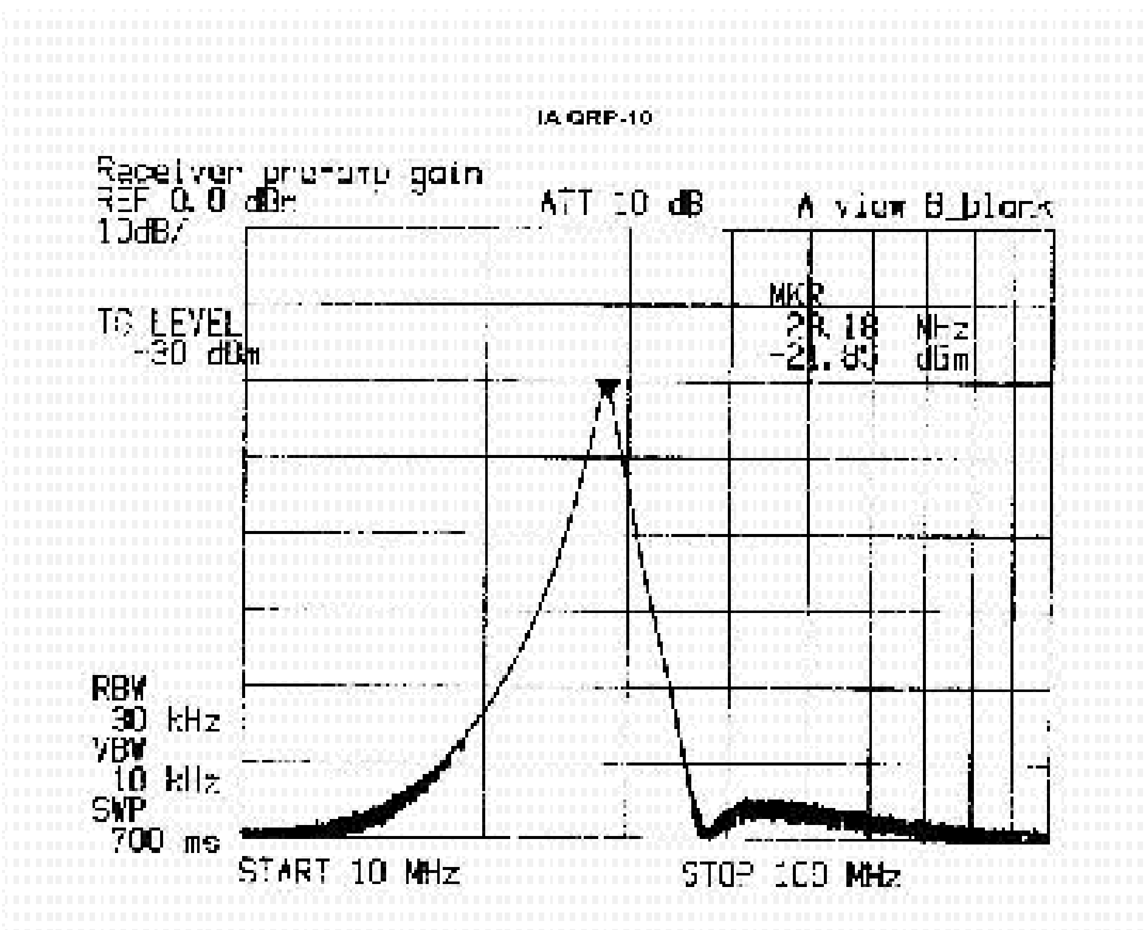

Fig. 16 Receiver Pre-Amp Spectrum Plot

about 8.5 dB, since there is a loss in the T/R switch of nominally 0.5 dB. The second test made use of my FT-990 receiver. An antenna was connected to the QRP-10, and the output from T2 connected to the FT-990. Signals on 10 meters were heard and their S meter readings noted. The same signals were then re-measured with the antenna connected directly to the FT-990, and observed to be about 1-1/2 S units lower. Added gain from the QRP-10 pre-amplifier appeared to be approximately 8 to 9 dB, consistent with the spectrum analyzer measurement. No detectable increase in the noise floor was noted while the QRP-10 pre-amplifier was being used, which is very good.

After the front-end was finished, it was time to tackle the first of the three ICs that make up the receive strip of the IA QRP-10. In many previous Manhattan-style construction projects which used ICs, my approach had been to use small rectangular pads made with the ADEL nibbling tool to mount the IC socket on. With more recent projects however, I started using small pads that approximate this arrangement. These pads are 0.6 inch long, and 0.4 inch wide, and have eight mounting surfaces cut into a piece of PC board material from with the pad is made. The cuts could be made with a small saw (a musical instrument fret saw would probably be excellent), milled with a milling machine if available, or etched as a small PC board if you have that capability using PnP Blue film or equivalent method. Mine were made on a drill press, using a milling table to control the X-Y movements. The cutter was a discarded dental burr.

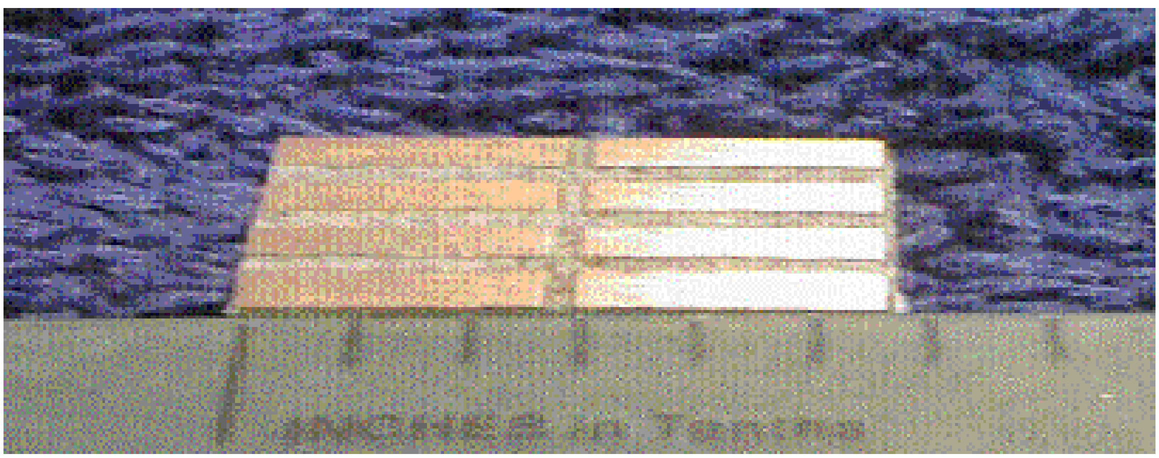

Fig. 17 IC Pad

The mixer IC socket was mounted to one of these pads, and the pad in turn glued to the substrate. Small glue dots were placed only at the corners, so the pad could be removed in case it needed to be relocated, or reoriented. The circuitry on the input and output sides generally follows the layout that Mike, N0MF used on his rig. The only deviation is in capacitor C9A, which is a 15 pF, instead of the 56 pF shown in the schematic. This change was made to reduce the LO drive to the mixer. With the 15 pF value, the drive is about 500 millivolts peak to peak, more than enough for the NE602 mixer. Overdriving an active mixer causes numerous spurious frequencies to be generated on the output.

After all of the parts are soldered in place, this much of the receive strip can be tested if you have a general coverage communications receiver. To test the operation, connect the VXO output to the VXO input terminal on your receive board. Then, temporarily solder a 0.01 uF capacitor to either pin 4 or pin 5 of the IC socket, and

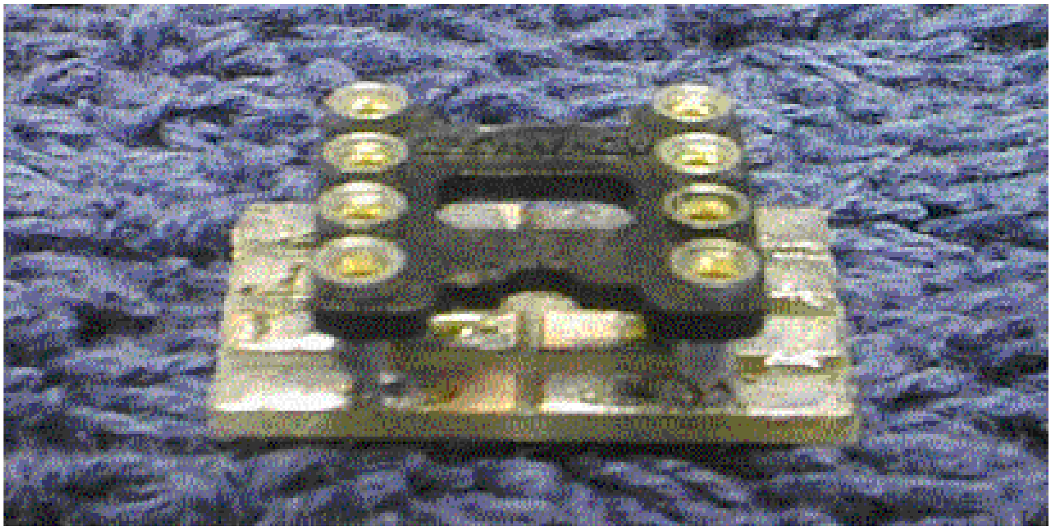

Fig. 18 IC Pad Mounted on IC Pad

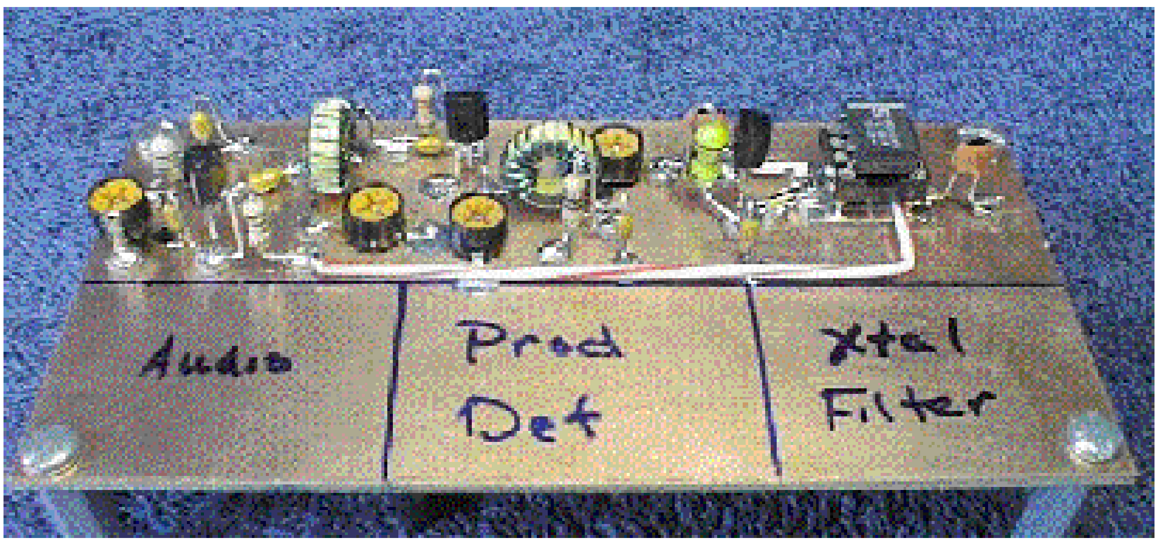

Fig. 19 Receiver Mixer, View 1

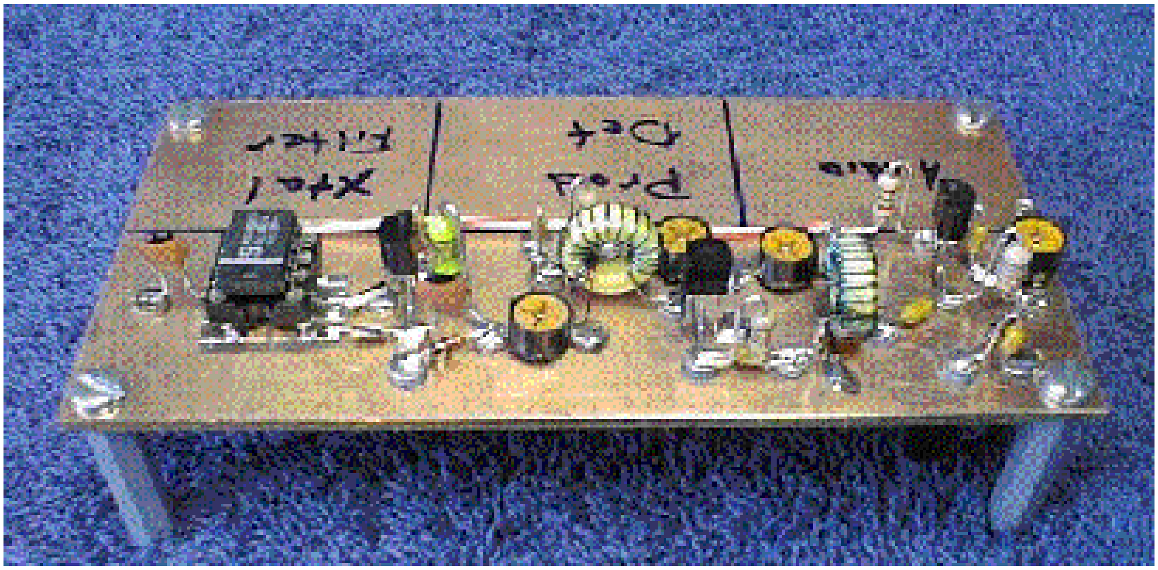

Fig. 20 Receiver Mixer, View 2

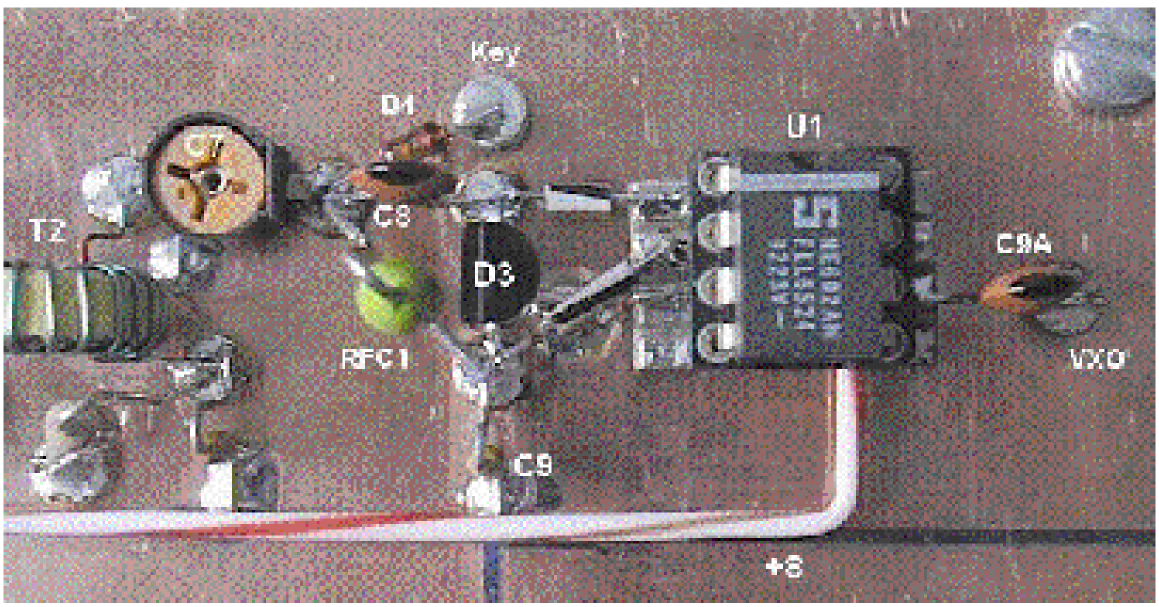

Fig. 21 Receiver Mixer Parts Layout

connect the other end of this capacitor to the antenna input on your receiver using a piece of coax. Tune the communications receiver to 3.932 MHz. Place an antenna on the input of the QRP-10 and apply power so that the receive pre-amp and mixer are active. Peak all of the trimmers for maximum noise in the communications receiver. If 10 meters is open, you should be able to tune the communications receiver around 3.932 MHz, and hear a signal. When I performed this test, there was a contest going on 10 meters, and finding a signal was not a problem. In fact, numerous 20 over S9 signals were heard from one end of the band to the other. Remove all of the temporary connections after this test, so the receive board is ready for further building. The crystal filter comes next.

These notes begin with a discussion on measuring crystals for use in the IA QRP-10 filter. My method for matching crystals is borne out of reading many articles on the subject, numerous experiments using various suggested methods, and some basic criteria that to my mind are invariant. This last item is probably the most important to the discussion. Based on all that I have read and found experimentally, it is clear that crystals in a filter are operating in a series resonant mode. This means they are exhibiting an "equivalent series resistance (ESR)" at the center of the filter pass band, and no reactive contributions. Above or below the center of the filter pass band, the crystal begins to exhibit greater reactance the farther away from the filter center one applies the excitation.

Given that the above statements are true, the only way one can measure the resonant condition of the crystal, i.e., no reactance, is to drive the crystal in a low impedance circuit, and find the condition where the output is a maximum. My test apparatus does just that. It consists of pair of 4:1 broadband transformers, one on the input, and one on the output. Ahead of the input transformer is a 4 dB attenuator, to provide isolation from the driving signal generator. The output transformer is terminated with a 51 ohm resistor, followed by a high impedance r.f. probe. Output from the r.f. probe is taken to an oscilloscope, which is used as the dc detector. The reason for using the oscilloscope is to take advantage of the screen graticule markings for setting up limits on acceptable ESR. Limits on what is acceptable as an ESR are determined with a small sample of crystals, since crystals of a given manufacturer and batch, as well as frequency, will exhibit differing ERS values. Normally, a marker is positioned at what appears to be the mean ESR for the group, and a second marker is then set at a value 3 dB lower. This marker becomes the lower limit at for which crystals will be accepted for filter use. Those that don't make this limit are destined for use in local oscillator service, assuming they have reasonable activity in an oscillator circuit.

The generator used to drive the tester is a RACAL-DANA 9087 digitally synthesized unit that has resolution down to 1 Hz. A short is placed across the crystal holder, the generator is set to the marked frequency on a sample crystal, and the amplitude of the generator is advanced until the trace on the oscilloscope is at full scale. A crystal is then inserted into the holder, and the generator frequency increased or decreased in 1 Hz steps until the oscilloscope display is again at a maximum. The trace location is recorded. A resistor is chosen from a low value set and inserted for the removed crystal. If the trace is higher, a the next higher resistor is selected, and the process repeated. Eventually, a resistor matching the ESR of the first sample crystal is found, and its value recorded. This process is repeated for the sample, which usually contains four to six crystals out of a batch of 50 to 100 units. The mean resistor value is then computed, and that value inserted into the tester. This output level then represents the median ESR for the batch, and a oscilloscope marker is set to this level. With the "mean ESR value" resistor still in place, the generator output is reduced by 3 dB, and a second marker is set. This marker then becomes the lower limit for accepting crystals from the batch for filter elements.

With the test fixture now calibrated, each crystal in the batch is measured by finding its series resonant frequency, and the last 3 digits of that frequency is marked on the crystal. Those whose ESR is below the lower limit, are marked "LO", and kept for use in local oscillator service. Occasionally, a crystal will exhibit really high ESR values, or cause the oscilloscope trace to continually wander in amplitude. These are rejected and not used at all.

Once all the crystals in a batch are measured, the leads are set into a foam board, with the crystals arranged from the lowest frequency to the highest, with spaces left for the sample set. The sample set is then further used to determine the characteristics of the batch in another test apparatus. Here is how that is done.

The second test apparatus consists of a Colpitts oscillator that uses 220 pF capacitors in the feedback positions and has a switch in the grounded leg of the crystal. Across this switch is a 12 pF capacitor. A sample crystal is inserted into the holder with the lower lead grounded. After a suitable amount of time for the crystal to stabilize, the frequency of oscillation recorded. Opening the switch places the 12 pF capacitor in series with the crystal, and its oscillating frequency increases. This new value is also recorded. Through the use of another switch on this test apparatus, the crystal is then placed into one arm of a resistive bridge circuit. The bridge is driven by the same 9087 generator that was used to excite the first apparatus. On the center nodes of this bridge are r.f. probes, one measuring the voltage drop across the crystal, and the other measuring the voltage drop across the reference lower bridge leg. This bridge leg is a 50 ohm, 20 turn non-inductive trimmer potentiometer. The generator is set to the frequency marked on the crystal, representing series resonance. It is moved up and down a few Hz just to verify a resonant condition. The resistor in the reference leg is then adjusted until the voltage drop across this leg and the crystal are the same. By operating another switch, the reference leg resistor is switched out of the bridge, and into a digital multimeter set to measure resistance. This value is recorded, and compared with the value obtained for ESR from the first test apparatus. The mean of these two values, measured by these two independent methods, along with the same set of measurements from the other samples becomes the crystal lot Rs value. To obtain the motional inductance, Lm, and motional capacitance, Cm values, and the Q of the crystal, requires a set of calculations. These are performed on a programmable TI-92 calculator. To obtain the holder capacitance, Co, of the crystal batch, each sample crystal is measure on an AADE digital L/C meter. The mean capacitance of the samples represents the Co value. The formulas used can be found in the paper "Refinements in Crystal Ladder Filter Design" by Wes Hayward, W7ZOI. This paper was published in the June 1995 issue of QEX magazine. The technique of using an oscillator circuit and derived formulas to obtain Lm and Cm values was developed by G3UUR.

Over the past several years, I have experiments with various oscillator circuits to simplify the process of matching crystals. None of them has worked satisfactorily. The best approximation to the accuracy that my present method provides is to use a Colpitts oscillator, with feedback capacitors that are very large. Values above 1000 pF will begin to yield approximately correct values for the actual series resonance in a given crystal. The feedback values have to be adjusted to match the frequency of the crystals being tested, with the limit being a capacitor value that just will let the circuit oscillate with the crystals being tested. As the feedback capacitance value is lowered, the measured series resonant frequency will begin to error upward in frequency, and the spread among the crystals being tested will reduce until they all measure nominally at the same frequency. By intuition, we know that cannot be correct.

The crystals supplied by Doug Hendricks, KI6DS for use in the IA QRP-10 building challenge have the following parameters:

Manufacturer	ECS
Marking	3.93-17

Measured characteristics (mean values)

Fo	3.931420 MHz
Rs	22.7 Ohms
Co	3.58 pF
Cm	9.61 X 10-15 Farad
Lm	0.1706 Henry
Q	185,600

Why are these parameters important? The answer is that they dictate how the final filter will be configured. While not going into all of the details, these parameters are used with other design criteria to define the actual topology of a given filter, and its theoretical performance such as bandwidth, input and output impedances, insertion loss etc. Without knowing them, we can only make educated guesses regarding terminating impedances, coupling capacitor values, and insertion loss. Certainly an optimal filter will not result without knowing the crystal's characteristics to some reasonable degree of accuracy.

Another nagging question that come ups often is "How close do the crystals have to be matched to work properly. There are two answers. The best filter would result with all of the crystals having exactly the same frequency. Practically, that isn't going to happen, so a better answer is to match them to within 10% of the desired filter bandwidth at the very worst, and to within 5% if at all possible. So for a filter with a 500 Hz bandwidth, the would mean the frequency spread within in the group should be no greater than 50 Hz, but 25 Hz would make a better filter. When a group of 50 to 100 crystals is available, getting many groups of 3 or 4 crystals within 5 to 10 Hz is quite easy. If you only have 10 or 20 units to start with, the 25 or 50 Hz targets are more realistic.

The set of crystals used in my QRP-10 unit had a total spread of 1 HZ. Two crystals were measured at 3.931282 MHz, and the other unit was one Hz lower in frequency. Within the 49 crystals in the group, there were at least 10 sets of 3 crystals which would meet the 5% criteria, for a filter with an assumed bandwidth of 250 Hz. That's quite typical for a lot of nominally 50 crystals. Of course if more elements are required in the filter, then fewer sets would be available.

Knowing the crystal parameters also allowed me to optimize the values used in the IA QRP-10 that I'm building. The topology was also changed a bit, which results in a better performing filter. One note of caution is in order. If you are building your rig from the QRPp Fall 2000 issue, there is a mistake in the illustration shown on page 26 regarding the crystal filter. The illustration shows capacitors C13 and C14 in series with the crystals X1 and X2, and X2 and X3 respectively. That isn't correct. One end of each of these capacitors is grounded, as shown in the schematic. My changes include adding a capacitor in series with RFC2 and X1, and another capacitor at the other end of the filter in series with X3 and RFC3. In addition, all of the filter coupling capacitors are 220 pF, which results in a filter that has a 250 Hz, 3 dB bandwidth. Making these changes also requires changing the value of RFC2 and RFC3 to 22 uH, and capacitors C10 and C16 to 56 pF to correctly match the input and output impedance of the filter at 270 ohms. The resulting filter looks very good when modeled with Electronic Workbench, and its measured performance on the spectrum analyzer isn't much different. Those plots have are available for viewing along with the construction pictures of this section.

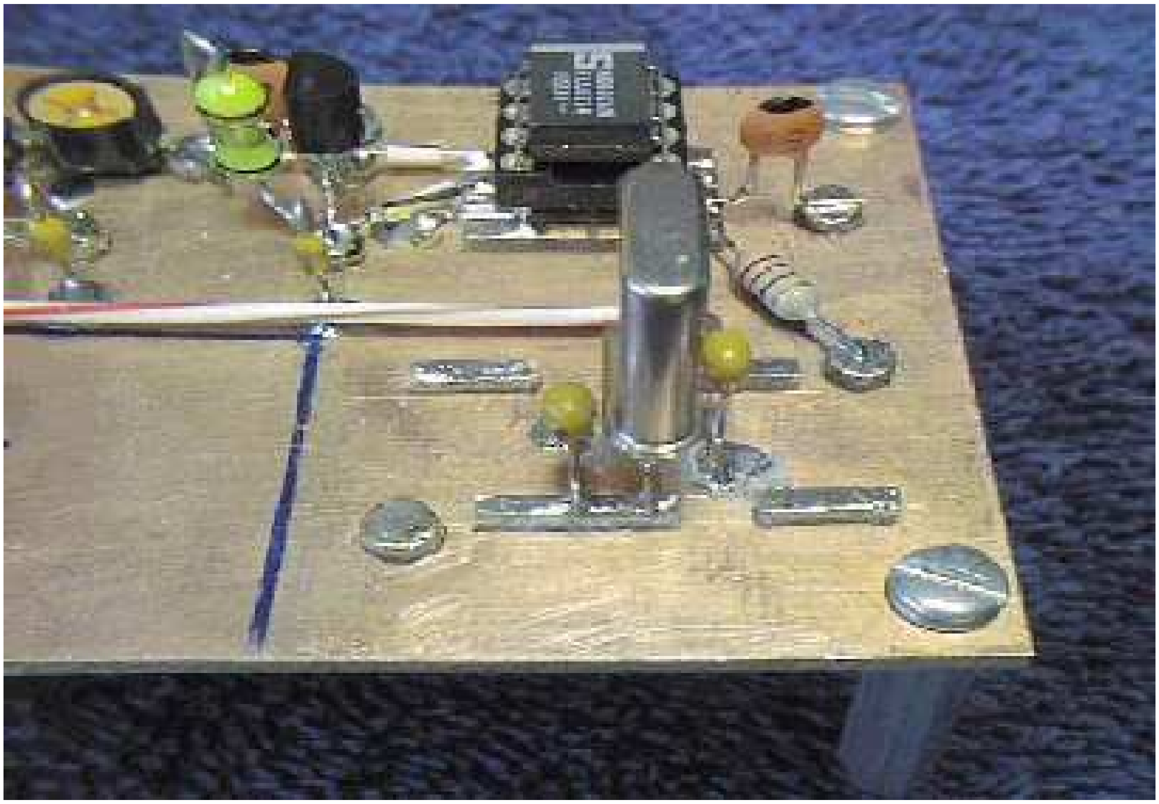

Fig. 22 Crystal Filter - Step 1

Fig. 23 Crystal Filter - Step 1, Another View

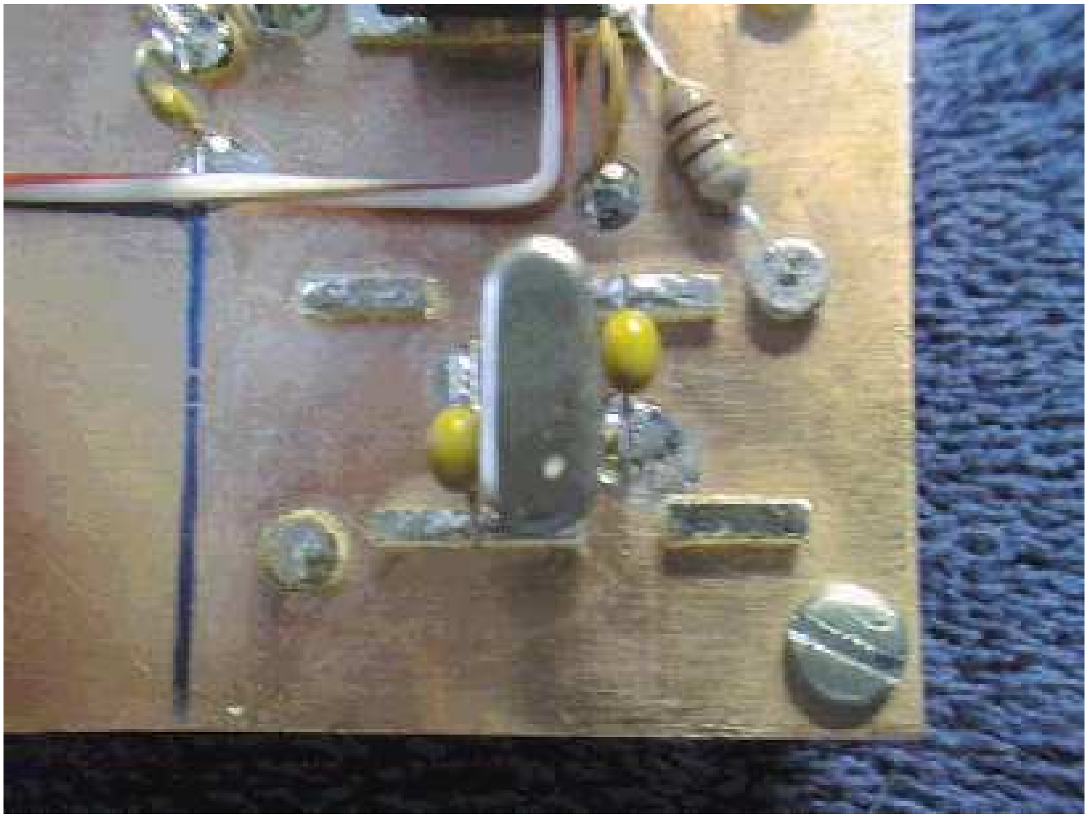

Fig. 24 Crystal Filter Step 1 Viewed from Above

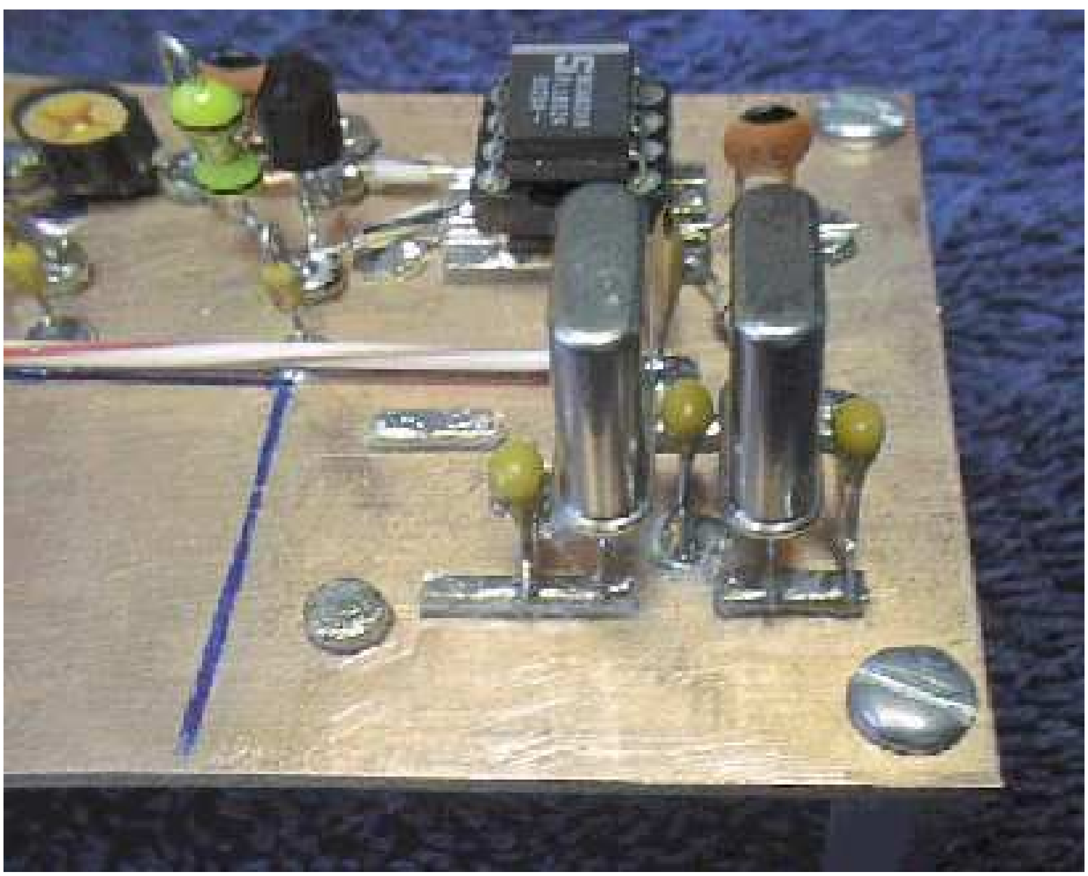

Fig. 25 Crystal Filter Step 2

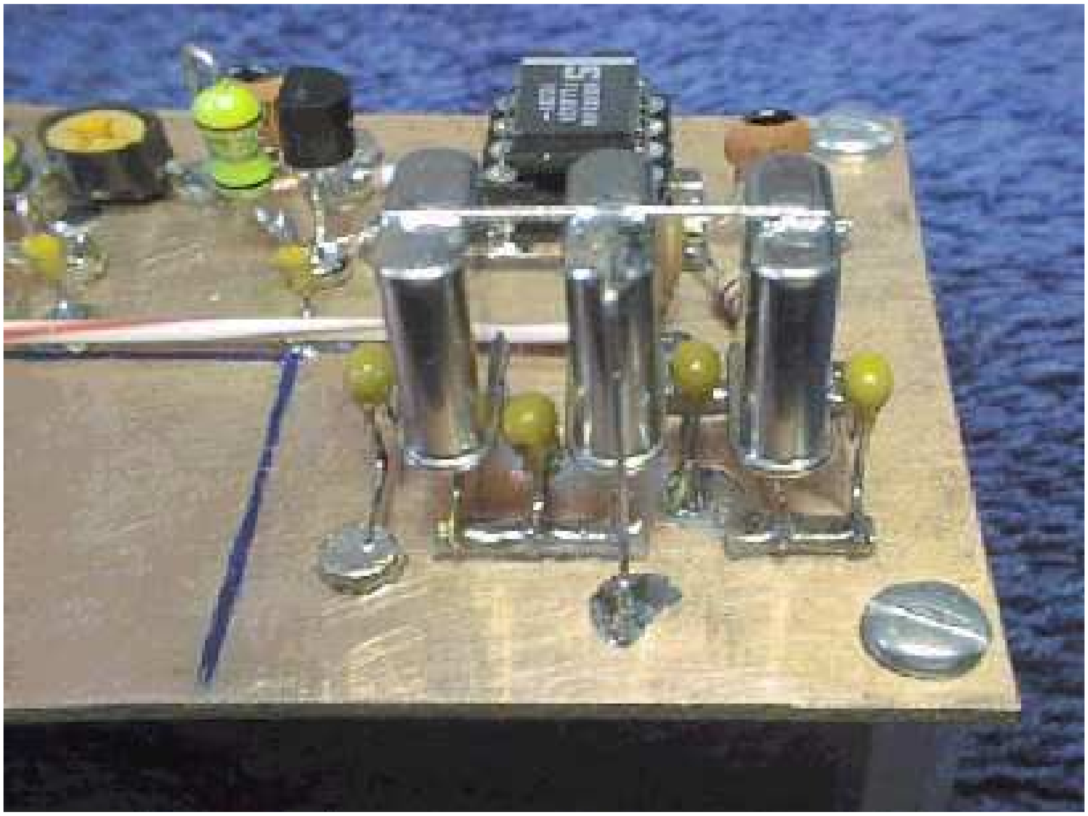

Fig. 25 Crystal Filter - Step 3

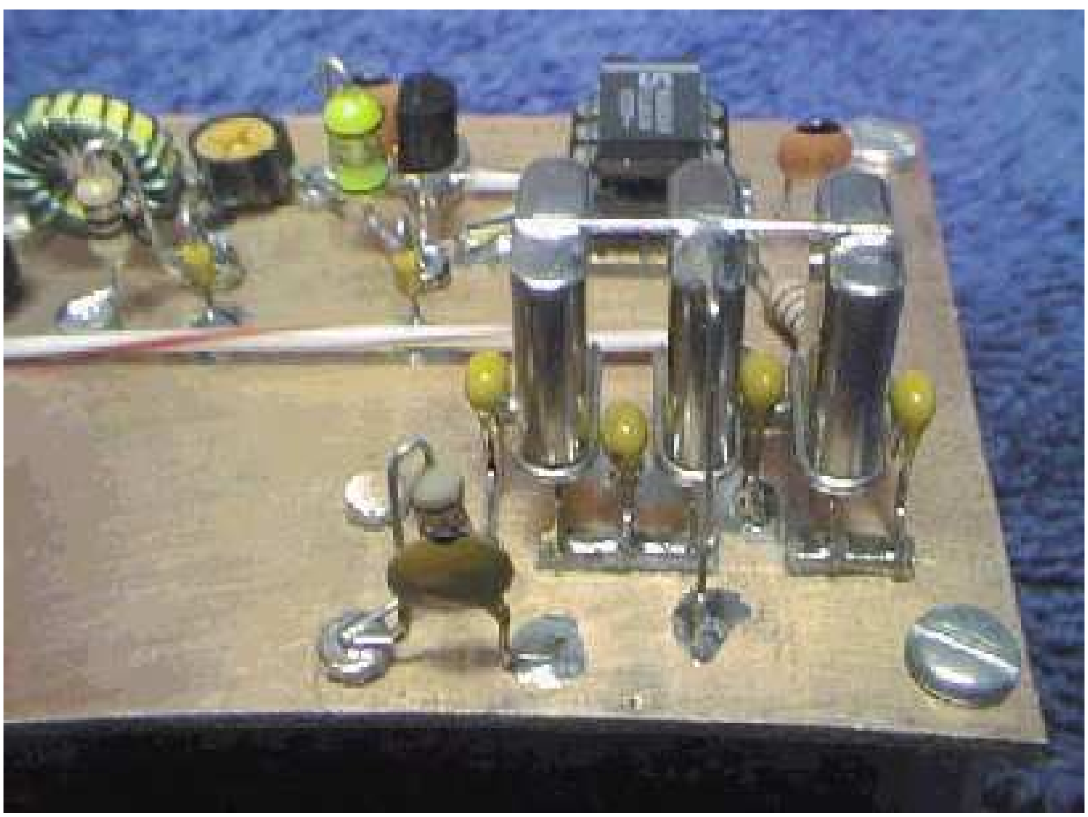

Fig. 27 Crystal Filter - Finished Version

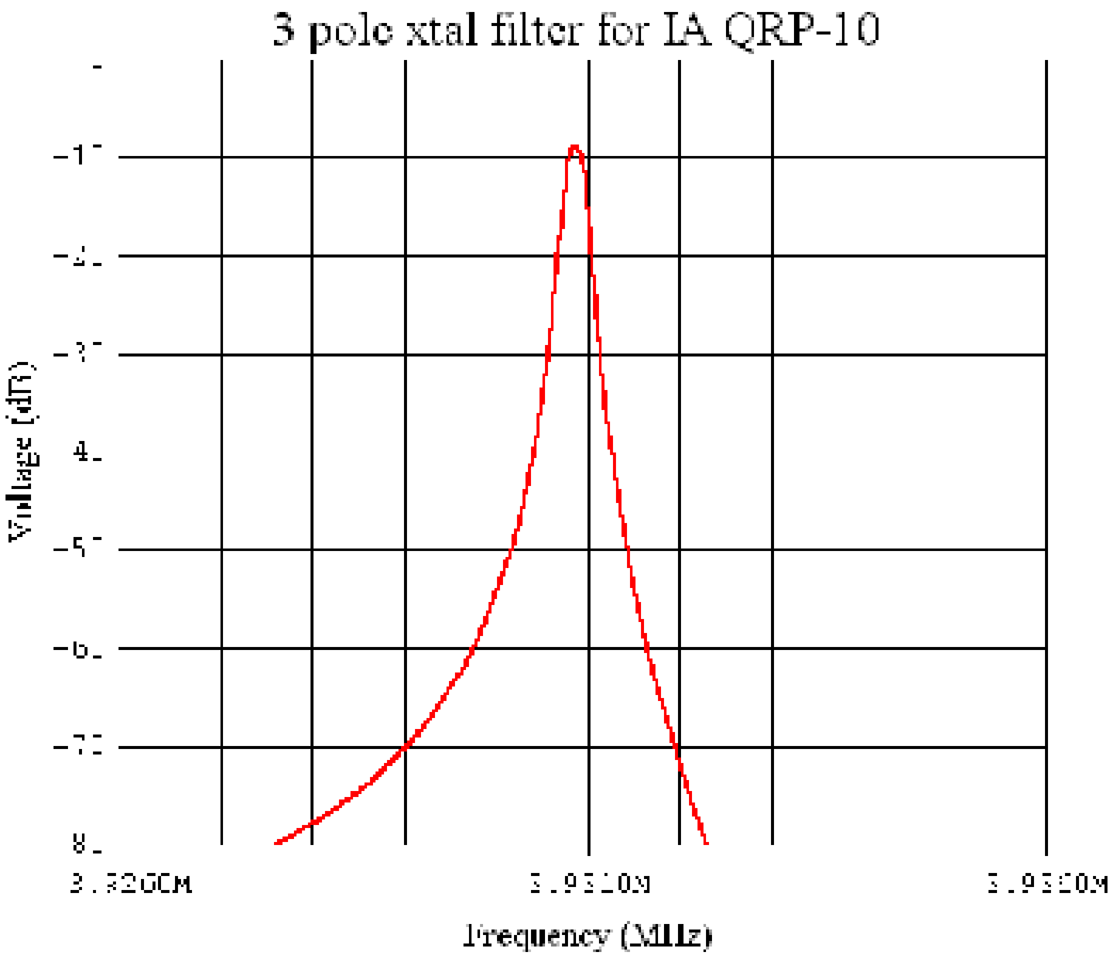

Fig. 28 Crystal Filter Simulated Frequency Response

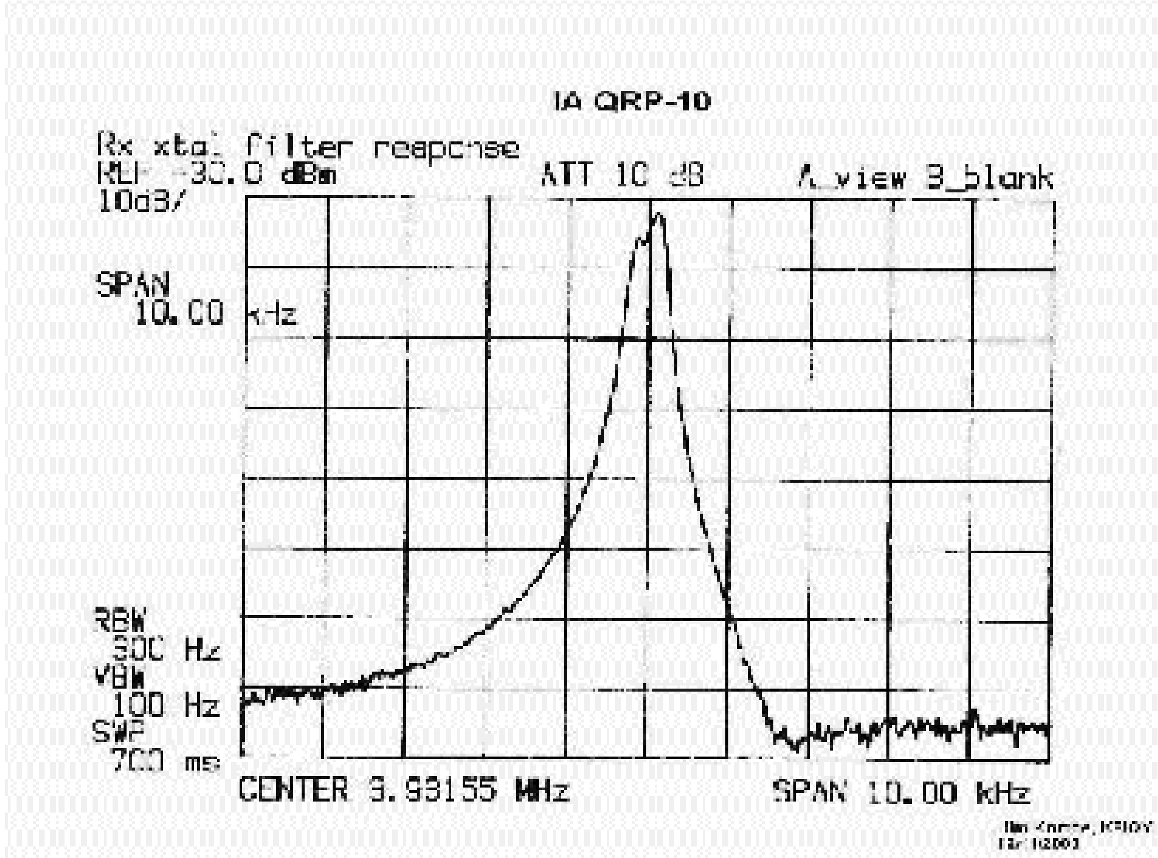

Fig. 29 Crystal Filter - Measured Frequency Response

Building the actual filter entails using a slightly different technique than the normal round pad that I have been using. The pads for the filter are 1/16 inch wide, and 3/8 inch long for the larger pads, and 1/16 inch wide, and 1/4 inch long for the shorter ones. The shorter pads were made with the trusty ADEL nibbling tool and the longer ones on the Harbor Freight shear. The pad pattern basically uses an offset of slightly less than 1/2 of the pads length on opposite sides of the layout. A spacing of 3/8 inch was used between the parallel pads. That can be seen in the pictures. All of the pads are laid first, and then the parts are added beginning in the middle of the pad array, and working toward the ends. This approach maximizes the space available for getting the soldering iron into the connection. The offsets are to accommodate the interior coupling capacitors, which for this rig are C13 and C14.

This section covers the building of the receiver product detector section, which turned out to be easier than expected, as several modifications to the original design were made. Primarily, neither of the J310 FET switches were built, as the changes provided by the redesigned front-end T/R switch are expected to keep the monitored transmit signal level to something reasonable. If I'm wrong about that assumption, then some or all of those parts will be installed after the transmit section is completed and working. I would test that premise, but my good signal generator is out for repair, so I don't have a reliable 30 MHz source at the required amplitude to test the receiver "as built". Also, only one output from the product detector was sent to the audio amplifier, instead of driving the audio amplifier differentially. My feeling here was that the improvement in the receiver front-end more than compensated for this 6 dB loss.

As a bit of a refresher course on product detectors, here is what is going on in this stage. First of all, a product detector is essentially a mixer, with the output signals being the original incoming signals, and the sum and difference of these two. Usually in a mixer, the incoming signals are the r.f., from perhaps the antenna, and a local oscillator, differing from the incoming r.f. by several MHz. One of the output signals becomes the i.f. frequency, and the other is the image frequency, and is not used. A product detector operates in the same manner, but with one distinction. The difference between the incoming signal and the local oscillator is in the audio spectrum; a difference of only a few hundred Hz normally. The local oscillator operates at or very near the i.f. frequency, for the IA QRP-10, at about 3.932 MHz. With this rig, the local oscillator is above the center of the crystal filter by about 750 Hz. The actual amount is adjustable via trimmer C17. When C17 is adjusted correctly, an incoming signal centered in the crystal filter is heard as an audio note of 750 Hz. The image signal coming out of the product detector is the sum of the i.f. signal and the local oscillator, and is at about 2 times the local oscillator frequency, or 7.864 MHz. This r.f. signal is essentially shunted to ground by capacitor C24. The audio signal is passed to the audio amplifier, U3 by coupling capacitor C19.

One more comment is appropriate. Since the local oscillator frequency is above the center of the crystal filter, the crystal filter is operating as a lower sideband filter. That's appropriate for all ladder filters using the design employed in the QRP-10, where the upper skirt is steeper than and lower one.

The parts for the product detector are laid out in a similar manner to that used by Mike, N0MF, although I did have to rotate the NE602 socket to the right by 90 degrees to fit the available space. This then required the attached parts to rotate too. However, if you look at Mike's layout rotated 90 degrees clockwise, and mine, they are quite similar. On this section, as I did with the mixer, the power to the NE602 comes into the socket above the socket mounting substrate, and is soldered inside the socket pin, instead of on the outside of the pin. That's clearly shown in several of the photos.

Fig. 30 Product Detector #1

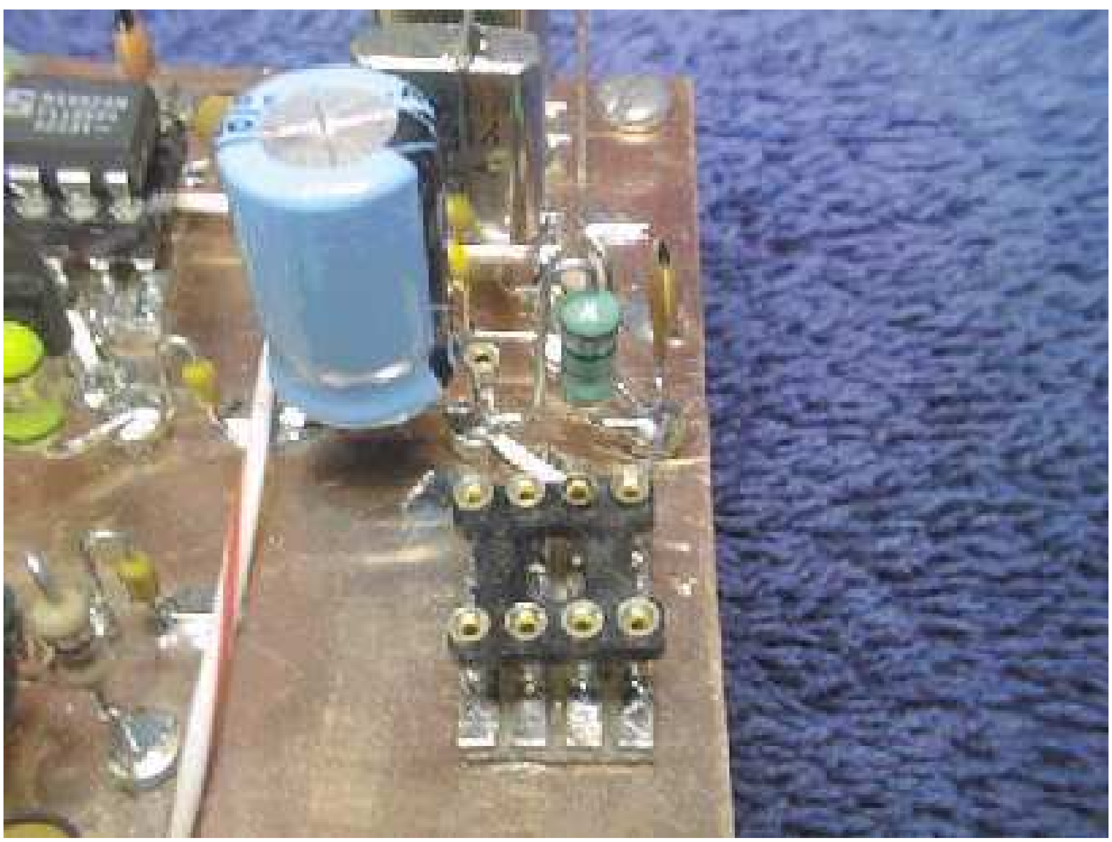

Fig. 31 Product Detector #2

Fig. 32 Product Detector #3

The parts layout photo has the parts labeled for the crystal filter, as well as this section. I felt it was easier to visualize how these two sections fit together doing it that way. After the product detector is built, it can be tested if you have a small high gain audio amplifier around that you can press into service. If you do, and it has about 40 dB of gain or more, you can solder a 0.1 uF capacitor to one of the NE602 output pins, (either 4 or 5) and a short run of shielded wire from the other end of this capacitor to your audio amplifier. Attaching the receiver to either an antenna or a signal generator should produce some noise. Peak the noise by adjusting all of the input and r.f. amplifier trimmers, and then adjust trimmer C17 for the loudest audio. If the product detector is working, 10 meter signals can be heard, or at least, you should be able to hear your signal generator. Trimmer C17 can be tweaked for the strongest signal.

These notes cover the construction of the last remaining receiver section, the audio amplifier. When I started this section, my thoughts were that it would be a “piece of cake”. After all, it’s only an audio amplifier. How hard could that be? Turns out this was the most difficult section to complete, and to get working properly. Read on....

I made a comment in the last section that I was planning to leave out the FETs, Q2 and Q3 that do the audio switching, since the front-end of the receiver had been redesigned, and it should have enough attenuation to keep the receiver from being overloaded by the transmit signal. As I started the build of the receive audio section, I indeed left out those parts, along with the associated resistors, capacitors, and diode. The only parts that I actually used were capacitors C19 and C24. Capacitor C19 takes the audio from the output of the product detector to the input of the LM386, U3, and C24 bypasses any r.f. coming out of the product detector to ground. Along with not using Q2

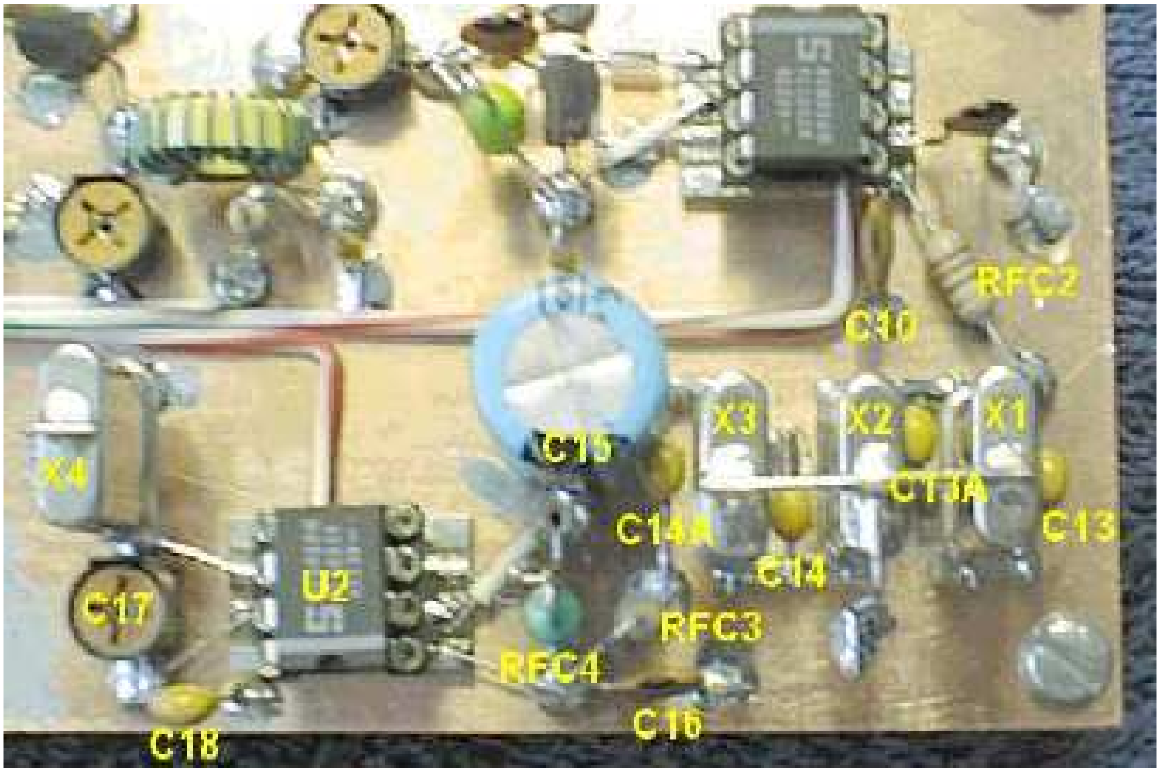

Fig. 33 Crystal Filter and Product Detector Parts Layout

and Q3 came another decision not to use both audio inputs on U3, but to run the audio single ended. This reduces the total audio gain by a factor of 2, but I was guessing there would still be an adequate amount, and it makes the building easier, especially if Q2 has to be added back in. The unused input pin on the LM386 was grounded.

After building the audio amplifier input circuitry, the output components were added, exactly as shown in the IA QRP-10 schematic, and in the same locations shown in the illustrations. The only components not installed were the LED in the AGC feedback circuit, and the audio output potentiometer, R9. The headphone jack was wired directly to the negative end of capacitor C27.

With this configuration, the receiver was powered up. My expectation was to hear some moderately loud audio noise, but not much was heard. I suspected a problem because this amplifier is running at a gain of 200 and that ought to produce some noise on the output. I looked over the circuitry again, just to make sure all of the connections were correct, and that I hadn't installed one or more of the electrolytic capacitors backwards, causing excessive leakage, and potential destruction. No problems were noted. This is the point in time when you're glad you have a fairly well equipped lab.

The oscilloscope was turned on, and the output probed. There was indeed a problem, a rather large sinusoidal signal (4 volts peak to peak) on the output, at a frequency of 10.1 MHz. Not at all like what is supposed to be there. I tried changing the values of R8 and C23 to see if that would help, since they are providing feedback from the output to an intermediate point of the LM386's input, but that didn't change the conditions much. I still had an oscillator, and I needed it to be an amplifier.

What to do? Go to the source, of course. I fired up my computer, connected to the National Semiconductor web page, and downloaded the product sheet for the LM386 device. In looking at the sample applications using this device, I kept noticing that each amplifier circuit had a 0.05uF capacitor and a 10-ohm resistor from the output (pin 5) to ground. It was there in every circuit except the one for a square wave oscillator. So those parts were added. With that change, the amplifier became very stable, and was producing noise, as it should be doing. The remaining change that I did was to remove the large electrolytic capacitor, C25 on pin 6 (power) and replace it with a 0.22uF tantalum capacitor. The large electrolytic didn't seem to be adding any stability to the circuit, and I felt the LM386 would operate better by having any r.f. on the +8 volt line bypassed to ground with the smaller capacitor. Ideally, both could be used, but space is at a premium at this point in the construction.

Having gotten the amplifier stage working, it was time to do some testing again. The receiver was powered up, and an antenna connected to the input. Immediately, 10-meter signals could be heard. They weren't very strong, and it turns out not from hams, but carriers from CB/Free Band operators running AM or FM. However, they were quite a good source to do some final peaking on the input trimmer. After that was done, the receiver was left operating. During the day, several CW signals, both DX and domestic were heard, some quite strong. I also found out the receiver will drive a small speaker with useable volume.

At this point we have two-thirds of the IA QRP-10 built and operating. When my signal generator gets back from the repair shop, I'll make some quantitative measurements on it. My guess is that the MDS is around 115 dBm, but it may be better than that. We'll also get some insight into whether the input attenuation is adequate to control the receive signal level during transmit without Q2 and Q3. I'll drive the input at +13 dBm, close the key line, and measure the audio output. While +13 dBm is below the signal level from the transmitter, it is at least in the ballpark. Once those measurements and others are done, I'll post an update.

Part 7 Update

Well, the UPS guy drove up the driveway this afternoon, and took this really big package off of his truck. It turned out to by my Racal-Dana 9087 signal generator that had to go back to New York for repair. I unpacked it, put it on the bench, and fired it up. Yes indeed! Working again like it was supposed to, and just in time for some timely measurements on the recently completed IA QRP-10 receiver.

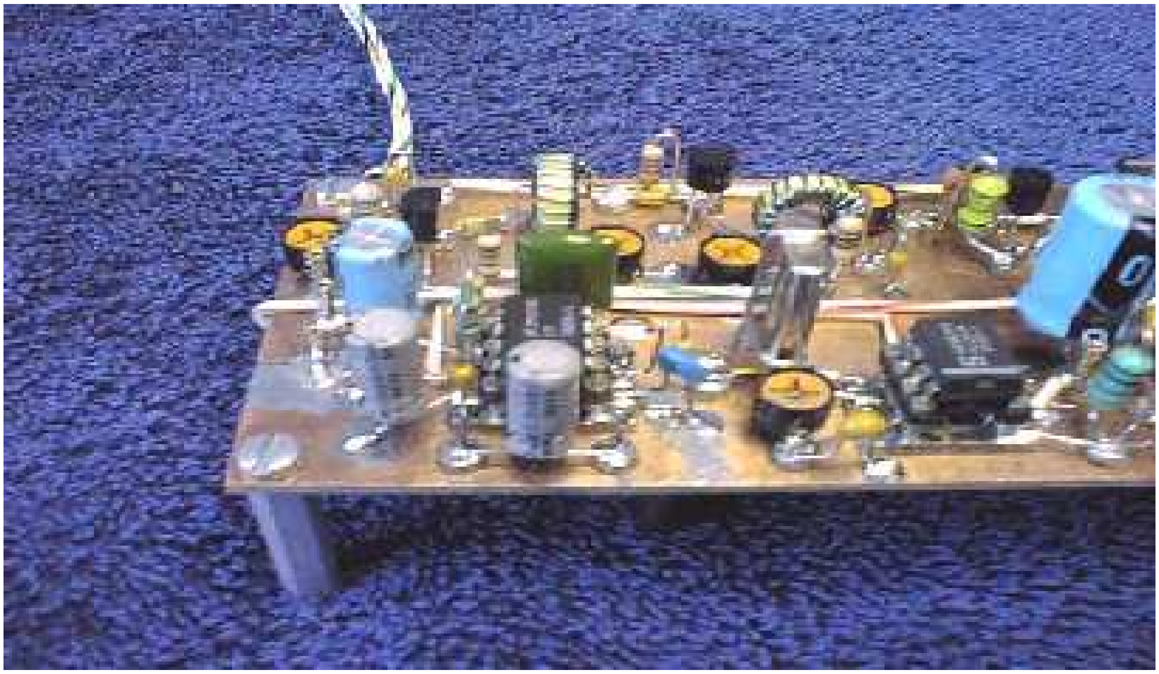

Fig. 34 Audio Amplifier

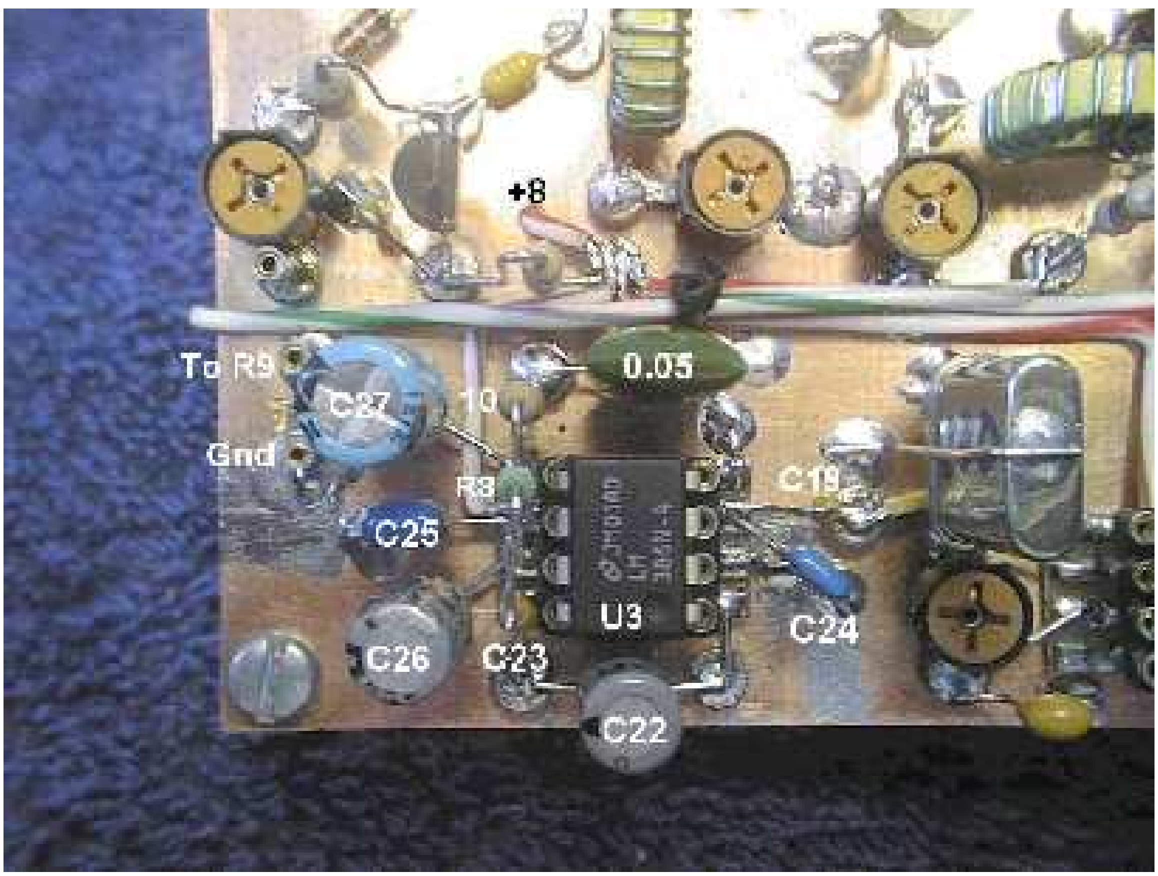

Fig. 35 Audio Amplifier Parts Layout

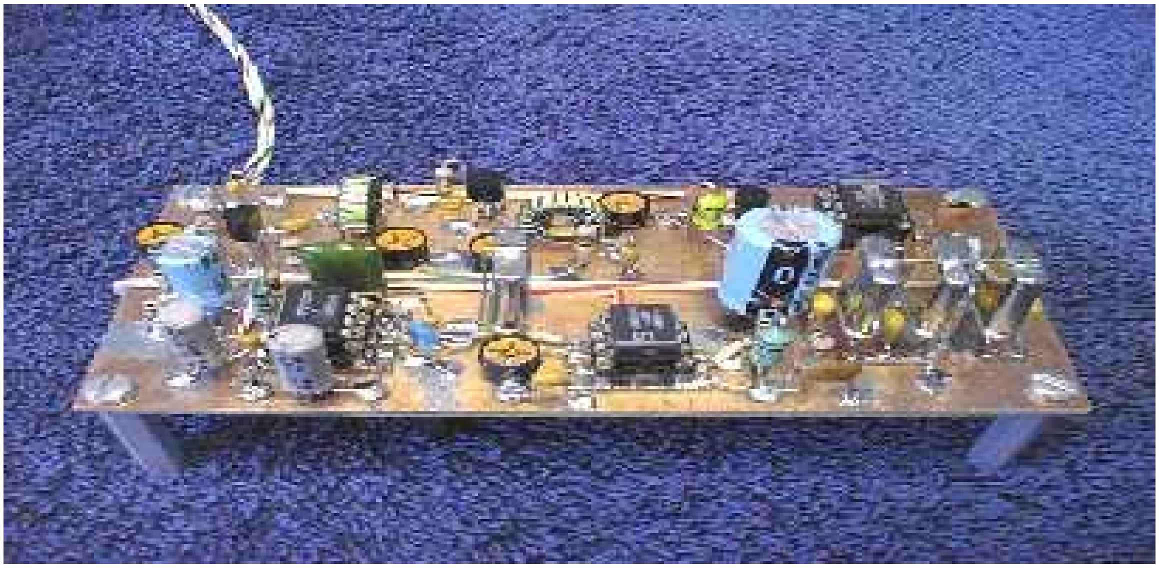

Fig. 36 Finished Receiver

Here are the measurements that I made, and my comments about them:

MDS = -130 dBm (That's less than 0.1 micro volt folks)
Filter Bandwidth = 230 Hz (Missed the design bogie by 20 Hz)
Minimum Opposite Sideband Rejection = 62 dB (Very good for a 3 pole filter)
IF Rejection = 110 dB (Excellent for any rig)
Image Rejection = 70 dB (Very good for a simple superhet design)
Rx Current Draw = 35 milliamps (VXO and receive strip combined)

Overall, this is really wonderful performance for a receiver as uncomplicated at this design. If you have been holding back on building this rig because you maybe thought it was "not that great", you ought to reconsider.

I'm really excited about getting started on the transmit strip, so I can see what the completed rig will do. It is a keeper in my book! I hope the gang is giving some serious consideration to providing a parts kit for this fine project.

I'll start the part 8 discussion by sharing a bit more information concerning how the VXO and receiver were integrated together. If you have followed this project to this point, you probably thought that the VXO and the receiver were built on separate substrates. That's the way Mike, N0MF built them. It is a good way, especially if you were to build a similar rig from scratch and not sure how much room is needed, or you're not sure what method will be used, VXO or a VFO, for frequency control. In this project however, that decision is made. Since the design requires two single sided PC board substrates for the VXO and receive strip, these can be combined into one, double-sided PC board substrate. On one side is the VXO, and on the other side is the receive strip. The advantage comes in connecting the two together, which is easy. The way the two sets of circuits are built, the VXO output is right under the receive mixer, U1. Within a few inches of the +8 volt power is a major connection point on the receive strip. The 12-volt power is similarly close by. None of these connections leave the substrate; they just have to be routed to the opposite side, keeping lead lengths short and direct.

My reason for mentioning all of this is that one of the first pictures in the part 8 group shows some of the VXO and receiver. I was keeping this detail secluded as long as possible in hopes of gaining a small, but significant advantage on my very worthy building opponent! It also made a lot of sense to build the rig this way.

Fig. 36 Transmit Substrate On Top

Fig. 37 Transmit Substrate Marked Off

The first steps in building the transmit section were to once again prepare another 2 X 4 inch substrate. This was done, and matching holes were drilled in the corners so that it could be stacked on top of the receiver/VXO board assemble during building. Why did I want it up there? Easy, so that the VXO signal would be close by during testing of the first stage of the transmit strip. Conveniently, the VXO output is also directly underneath the transmit mixer, U4. While the details have not been finalized, some set of mating connectors will be used take all of the signals to and from the transmit board, so that it can be removed from the stack.

After the transmit strip board was cut and drilled, four sections were marked on it, one each for the transmit mixer, transmit band pass filter, driver amplifier, and final amplifier/low pass filter assembly. As was done with the VXO and receiver, these areas help keep the layout clean and organized. While that is the goal, they are only lines drawn with a marking pen, so if a part needs to "spill" into an adjacent area to make it fit, let that happen.

With the transmit substrate ready for building, it's time to heat up the soldering iron, and get out the pads and glue.

Building the transmit mixer section starts with putting the IC socket on pads of some kind, or gluing down a socket/pad combo. My version uses the latter, and I won't go into details here, as that was already done in a prior discussion. There is at least one important consideration thought, and that is to place the socket away from the edge enough so that when the transmit band pass filter is added, (in part 10) there is room for either a trimmer capacitor or toroid. Following that approach will result in a neat layout.

After the socket for U4 is secured, the capacitors on pins 3 and 2 can be soldered in place. Don't worry about the keying lead that will also be soldered to pin 3. It will be added when the driver amplifier stage is built. U4 has an added capacitor soldered on pin 1, along with the 4.7-pF capacitor (C34) that is shown in the schematic. I've called this added capacitor C34A, and its value on my rig is 22 pF. It was added after some initial testing showed the r.f. drive level on U4 was too high. Capacitor C34A forms a voltage divider with capacitor C34 to reduce the level down to about 150 millivolts peak-to-peak. I've included an oscilloscope picture in this construction set showing the signal on pin 1 of U4.

Having completed one side of the socket, the parts for the other side can be added. Start by putting down the pads for the trimmer capacitor, C39. I made sure the pads were spaced enough apart that the trimmer body could fit between them. After these pads are placed, the leads on capacitor C55 (27 pF) can be bent so that it can be soldered to socket pins 6 and 7, with the remaining lead ends soldered to the C39 pads. Normally, I would use separate leads for this kind of a connection, but I felt the values that were shown in the schematic made sense, so there was little chance that C55 might need to be changed.

Fig. 38 TX Mixer #1

Fig. 38 TX Mixer #2

Fig. 39 TX Mixer #3

An additional pad is then placed for the mounting of inductor RFC7. Note in the pictures that this part, like most of the resistors and inductors, is oriented vertically with the upper loop end serving as a test point. After the inductor is soldered in place, the crystal leads can be bent, and it too soldered. The only remaining part to place is capacitor C40, and the pad for it is already there, since it shares a pad with trimmer capacitor C39.

As with the lead to pin 3, don't worry about adding the +8 volt power lead to pin 8 at this time. It will be easier to figure out the correct routing after the driver amplifier stage is finished.

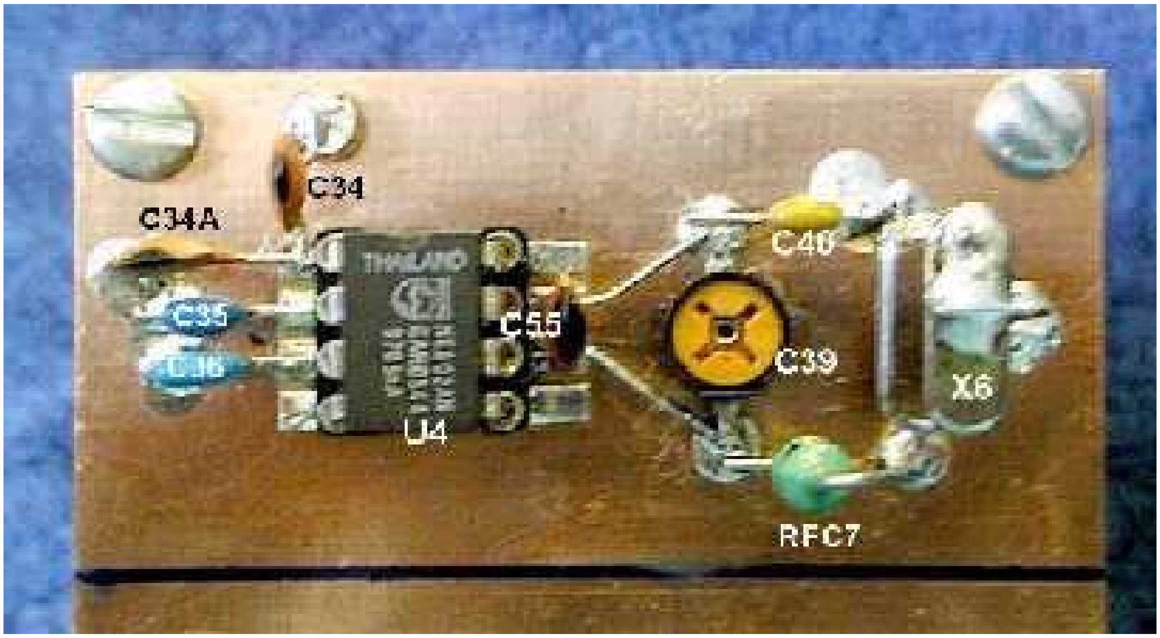

Fig. 40 TX Mixer Parts Layout

Once this stage was completed, I did some detailed analysis of the input drive level and output spectrum. It was during my first look at the output spectrum that I suspected the r.f. drive level from the VXO doubler circuit was too high. The first spectrum plot holds the key. If you look at the peak at nominally 28 MHz, which is the output we will be using, and the peak at 32 MHz, which is the r.f. drive coming in, you see a difference of about 12 dBm. Too much of the 32 MHz drive is showing up on the output, an indication of overdrive for this mixer. The difference should be greater but at least 10 dB or so for an NE602. If this were a diode double balanced mixer, the difference would be about 50 to 60 dB.

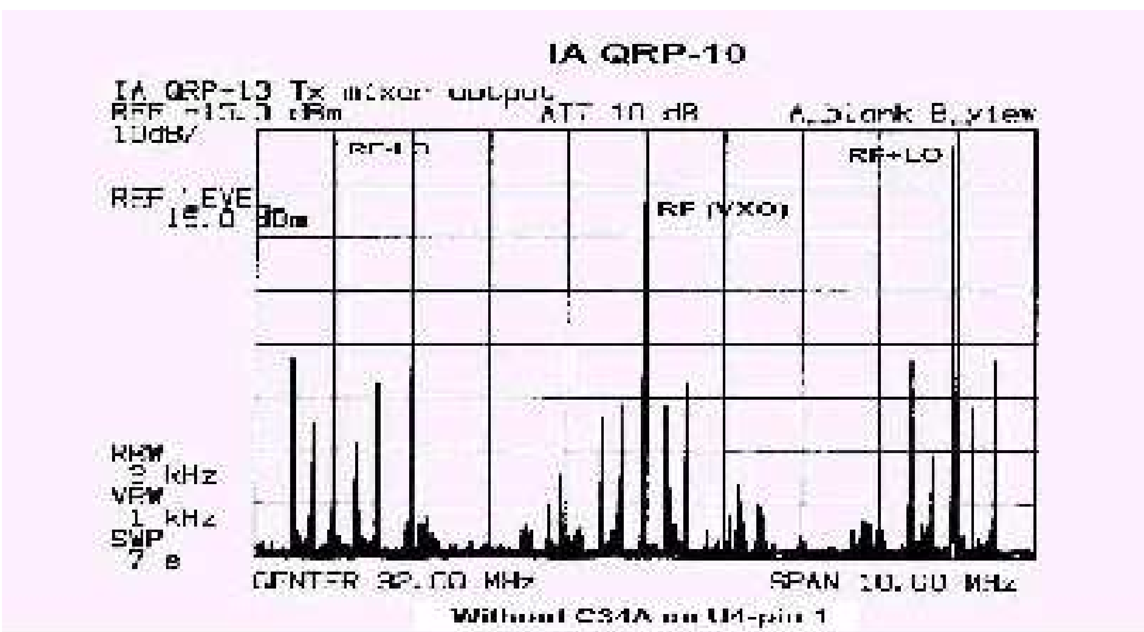

Fig. 41 TX Mixer Output Without C34A

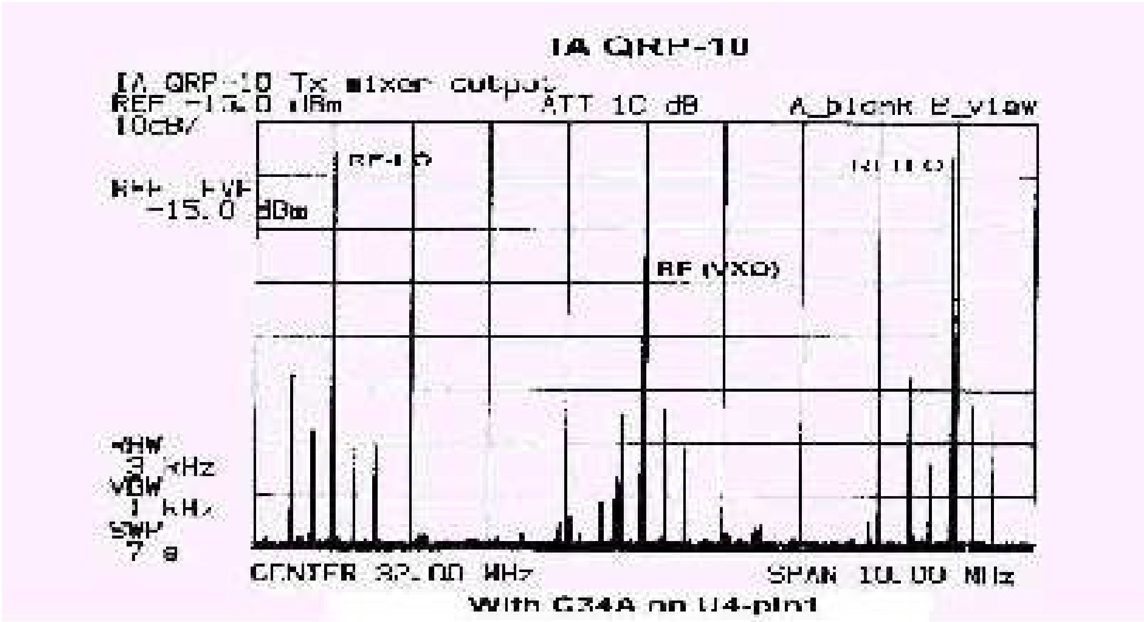

Fig. 42 TX Mixer Output with C34A

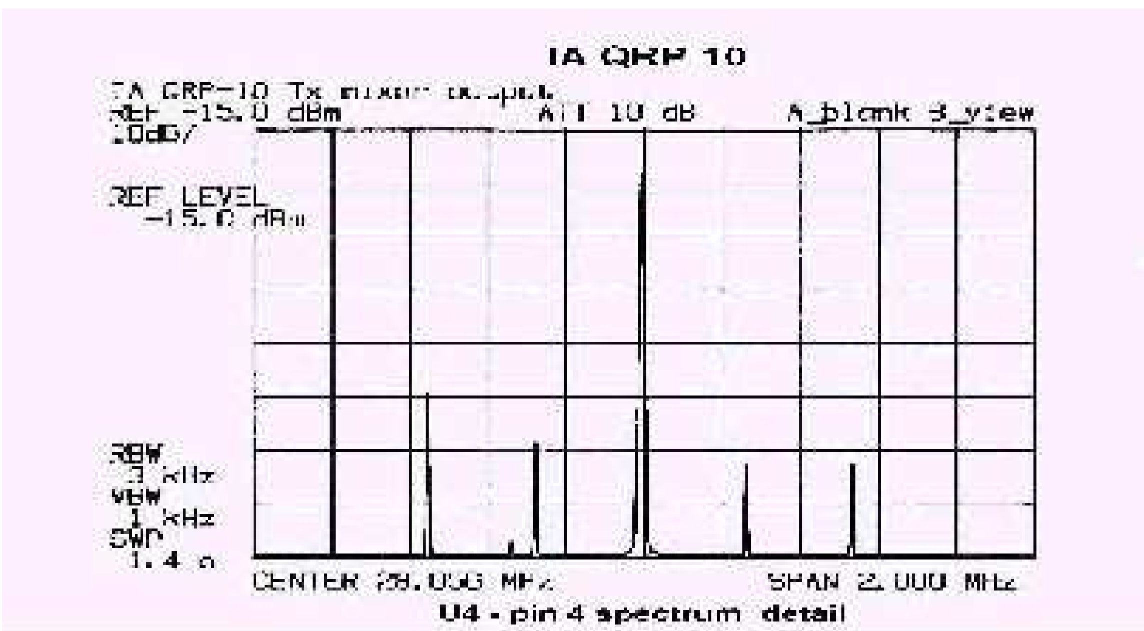

Fig. 43 TX Mixer Output Detail

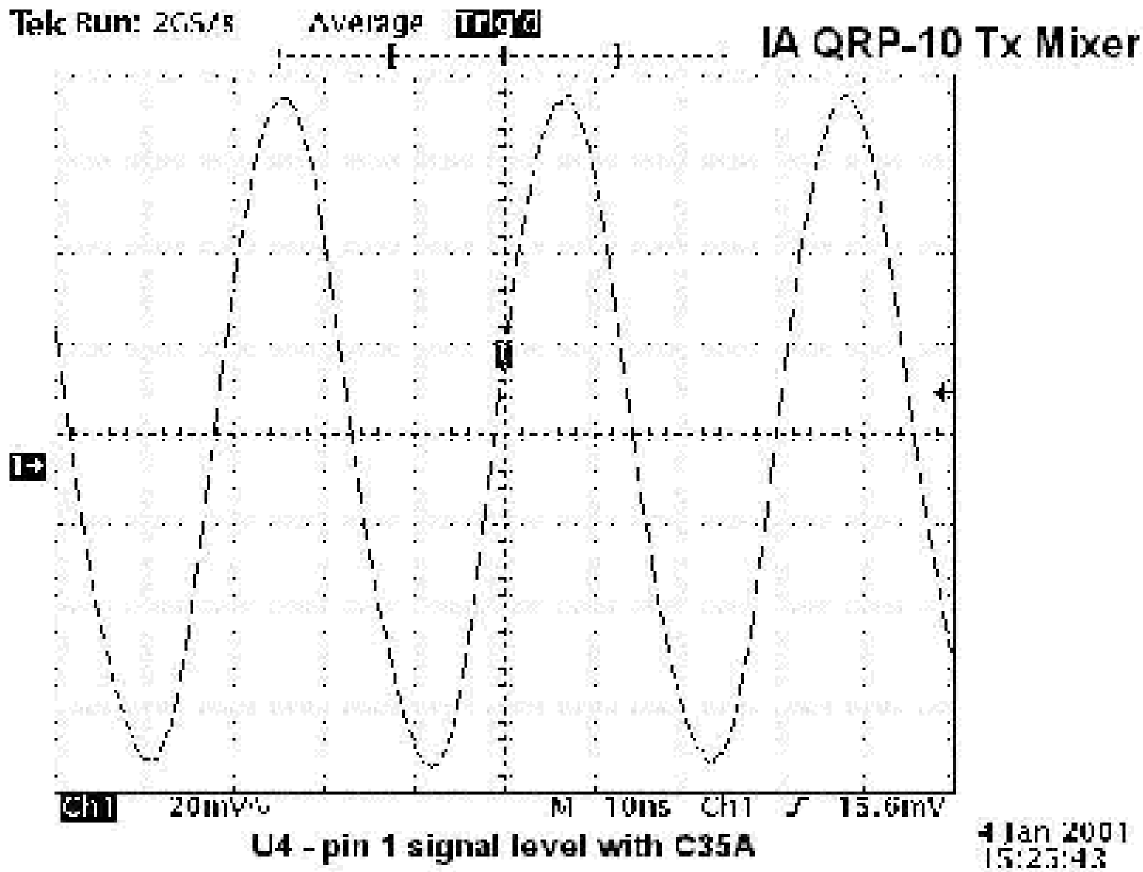

Fig. 44 TX Mixer Drive Level

By trial and error mostly, capacitors were added to ground on pin 1 of U4, until the largest difference between the 28 MHz and 32 MHz signals occurred. That happened with C34A at 22 pF. A value of 27 pF was also very good. The second spectrum plot shows these two signals with the capacitor soldered in place. The 32 MHz drive signal is now at least 20 dB below the output level at 28 MHz. Also, notice that many of the higher order spurs are reduced significantly too. This is another key that the mixer was being over driven. The signal at 28 MHz is the “minus” output, the difference between the VXO at 32 MHz, and the local oscillator at 3.93+ MHz. Look also at the other signal on this plot at roughly 36 MHz. This is the other principal output from the mixing process, the “sum” signal. It is this signal and the r.f. drive feed through, that the band pass filter, which follows this stage, can effectively attenuate.

That last spectrum plot shows the desired 28 MHz signal in detail. All of the close in spurs are at least 40 dB below the desired signal. These must be low, as the band pass filter will be tuned to 28 MHz, providing very little attenuation of these high order products. If the spurs were high at this point, they would be passed on to the driver stage, then to the final amplifier, and show up in the output.

In part 10, the transmit band pass filter gets constructed. Since this section has only a few parts, it won’t take long to discuss it, nor very long to build.

As was mentioned in part 9, the location of the U4 socket was selected to allow sufficient room for a toroid to be placed between the board edge and its connecting pad. To get started building, a pad was glued just over the section separation line, and at the same height as pin 4 of U4. Across that opening is soldered capacitor C37, a 4.7 pF unit. Toroid L1 was then wound with 17 turns of #26 wire and measured on an AADE capacitance meter. The inductance for L1 was 0.972 uH. After cutting its leads to length, it was soldered from the C37 pad to ground. Opposite this inductor, trimmer capacitor C38, a 5-50 pF unit was soldered.

Next, another pad was laid, and capacitor C41 leads were formed and trimmed, and it was soldered from the C37 pad to this new pad. Trimmer capacitor C42 is also soldered to this pad, located between it and the side of the substrate, and parallel to inductor L1. Finally, inductor L2 was wound with 17 turns of #26, its inductance measured, and soldered in place adjacent to trimmer C38. Interestingly, both inductors were wound identically, but the measured value for L2 came out at 1.05 uH. That represents an 8 percent difference in inductance between the two unit, and is typical. The differences come about from subtle winding variations, and differences in core permeability. One might wonder why were the inductors mounted on opposite sides of the center line on signal path. That's to keep them separated so their magnetic fields can't link, thereby changing the coupling between the two sections. The coupling is controlled by capacitor C41.

To finish the buildup of the filter, an additional pad was placed and the output coupling capacitor, C43 was soldered into place. This last pad won't remain. It will be removed when the buffer amplifier socket is placed. In the meantime, having a terminus for C43 provides a stable point for connecting our test gear. With the filter constructed, it was time for another test.

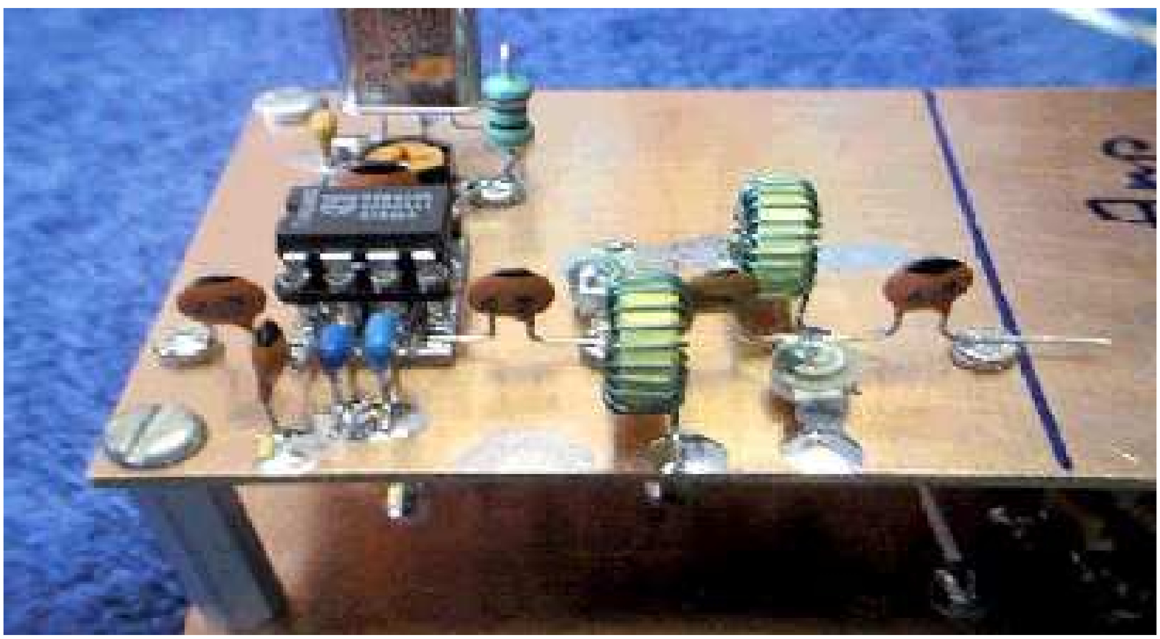

Fig. 45 TX Bandpass Filter #1

Fix 46 TX Bandpass Filter #2

Fig. 47 TX Bandpass Filter #3

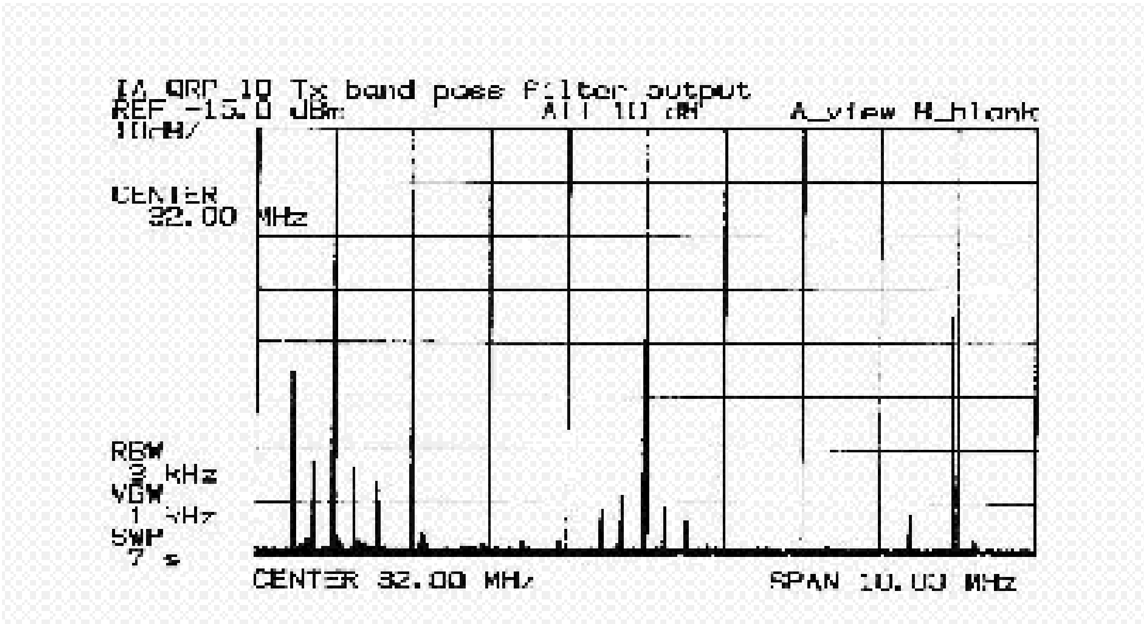

Fig. 48 TX Bandpass Filter Output Spectrum

I've only got one plot to show you for this section, but it reinforces the theory discussed in part 9. As a refresher, I commented that this filter would affect the level of the r.f. drive signal and mixer image output, but not the 28 MHz signal that remains. Indeed, if you look at the filter output plot, the r.f. drive level has been reduced by an additional 15 dB, and the image signal by 30 dB. If this transmit filter were even narrower in its frequency response, the two unwanted signal would be reduced more. Making it narrower involves reducing the value of capacitor C41. For this 28 MHz filter, 1 pF is probably the lowest practical value that can be used. However, doing that would make adjusting this filter more difficult. When we finally get the rig finished, if there is too much output of either of these two unwanted signals, we know exactly the place to go to make a change, and the part that is affected.

In the next part, we will build the buffer/driver amplifier.

In this part, we will do the construction and testing of the transmit buffer amplifier/driver stage. This stage is base on a rather unique IC, a Linear Technology LT1252 video amplifier. While not specifically designed for r.f. amplification work, it has enough gain-bandwidth to be very useable up to 10 meters.

Before starting the actual buildup of this stage, an 8-pin socket was configured with a 620-ohm resistor soldered underneath it between pins 6 and 2. The reason for doing this was to allow the layout of the driver stage to be more neatly built and without parts crossing over each other. That would not have been doable had this feedback resistor been wired external to the socket. Along the way, a series of three pictures were taken to document how this was done. While I don't recommend using this method for a new design, as you may need to change the value, I felt it was safe enough to use here. After the resistor was soldered to the socket pins, the socket was soldered to a header, similar to the other 8-pin sockets used in the construction.

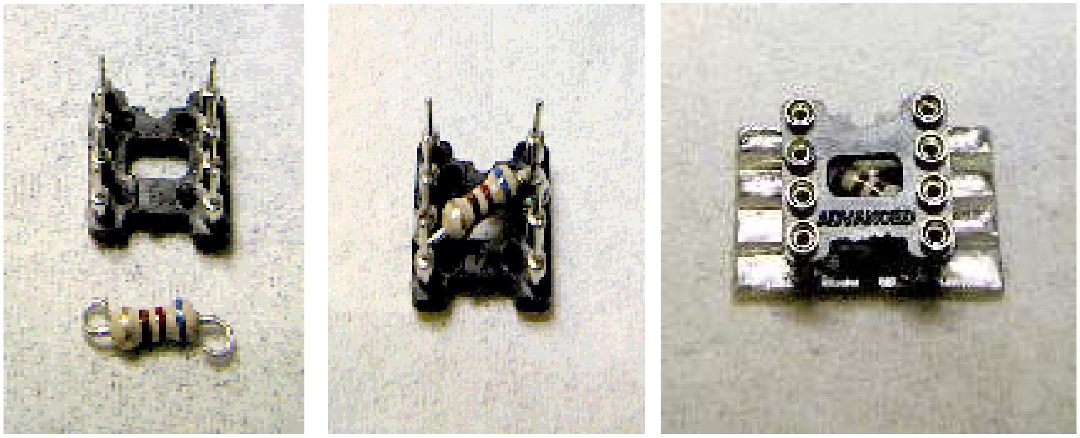

Fig. 49 U5 IC Socket Preparation

The U5 driver stage socket was aligned on the PC board substrate so that it was the same distance from the substrate edge as the U4 mixer socket. This alignment also placed pin 3, the non-inverting input just above the centerline of the Tx mixer filter output capacitor, C43. Sufficient room to the left of the socket was provided for bias resistors R18 and R19. A short jumper from the common pad containing these three components to pin 3 completes this part of the circuit.

The two remaining components, C44 and R20, which finish the feedback around the inverting input, were then added to pin 2. Resistor R62 was already in place, being the resistor soldered under the socket. Capacitor C46, an r.f. bypass for the Vcc line was installed on the pad containing the upper end of resistor R19. A short lead was taken from this pad to pin 7, the power pin for the IC.

To finish the wiring of this stage, capacitor C45 was then soldered between pin 6 and a pad secured to the right of the socket, and centered between pins 7 and 8. This pad also is one of two to which is soldered the 100 ohm trimmer resistor, R22. The trimmer used was a 1/4 inch diameter Bourns cermet type, and oriented so that the wiper moves away from the grounded end as the potentiometer is turned clockwise. That motion raises the drive level to the following stage. With the potentiometer mounted, the buildup of this stage is nearly complete. The remaining item is wiring pin 4 back to pin 3 of U4, the transmit mixer so that these two stages can be key together. I also added a 0.01 uF capacitor to pin 4 of U5, just to assure that any r.f. picked up on this lead would be bypassed to ground.

Testing can be accomplished by applying power to the rig and assuring that +8 volts and Vcc are on the appropriate chip pins. Since the output of the buffer/driver stage just built is a 100-ohm potentiometer, we already have a suitable dummy load for testing purposes. Grounding either U4, pin 3, or U5, pin 4 will cause both the transmit mixer and buffer/driver stages to be active. The junction of capacitor C45 and resistor R22 (the 100 ohm potentiometer) can be monitored with either an r.f. probe, or an oscilloscope. Trimmer capacitors C38 and C42 are adjusted until the output is peaked. If a receiver is available, or if the IA QRP-10 receiver is operating, the 10-meter signal being generated can be heard. I’ve included some spectrum plots in the picture set to show the output of this stage. The first plot shows the spectrum centered on the VXO frequency. I’ve show this plot format before after the transmit mixer and transmit band pass filter. This version shows the gain provided by the buffer/driver stage. The second plot is centered on the 28 MHz output and shows the close in spurious responses.

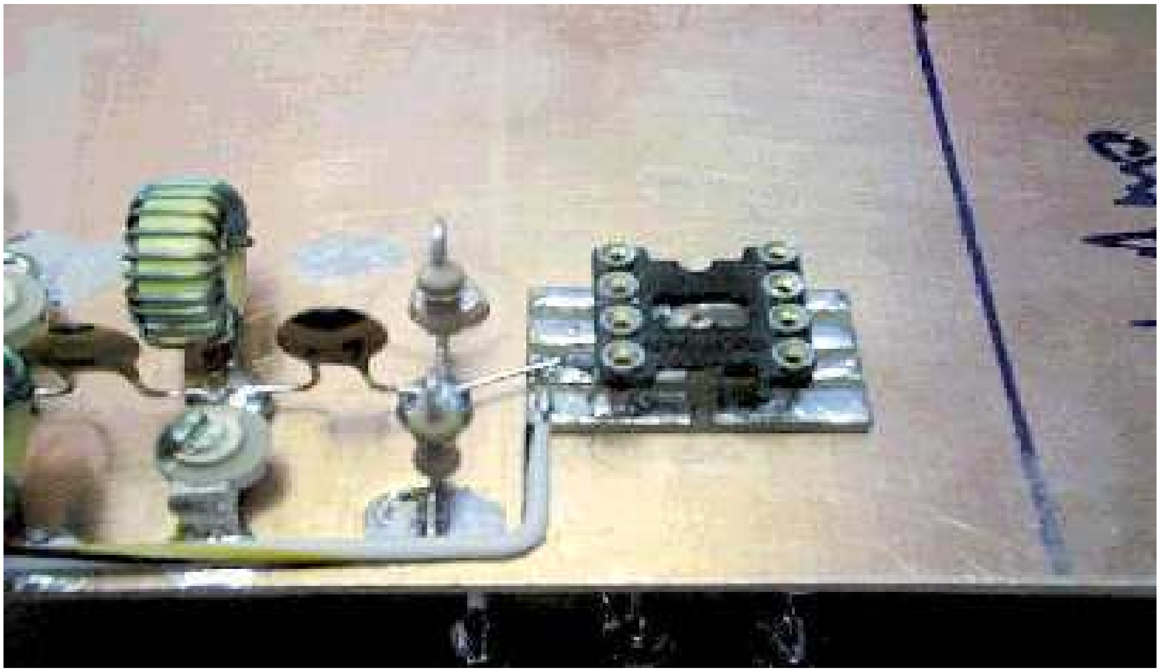

Fig. 50 Transmit Driver #1

Fig. 51 Transmit Driver #2

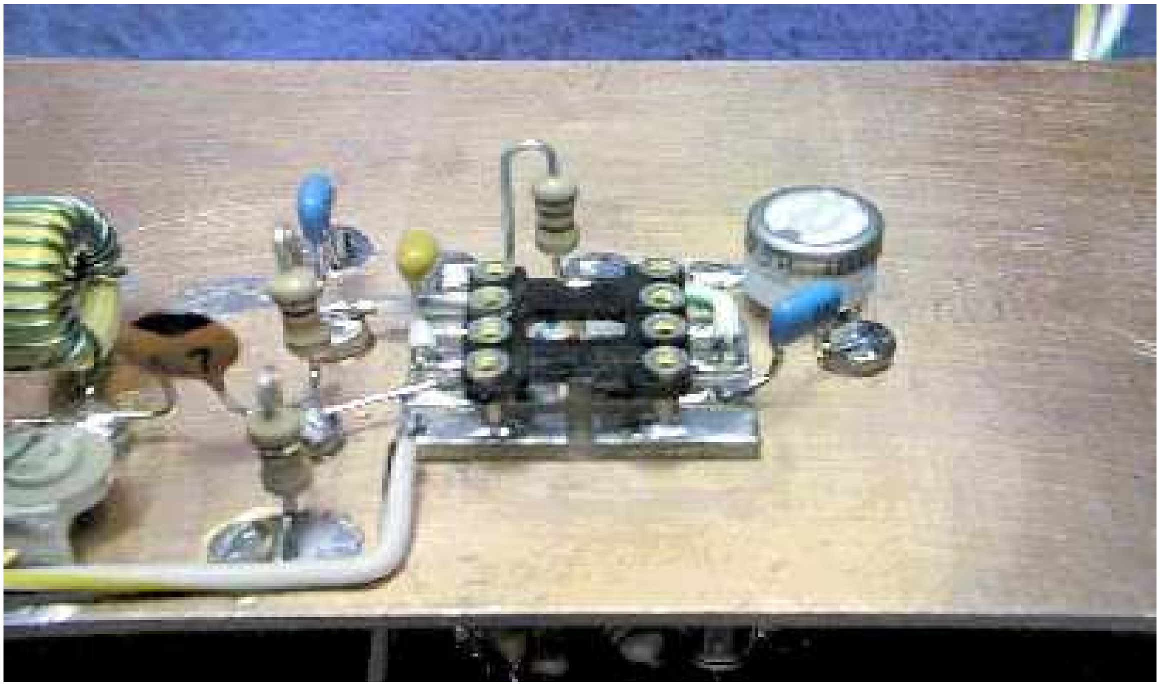

Fig. 52 Transmit Driver #3

Fig. 53 Transmit Driver #4

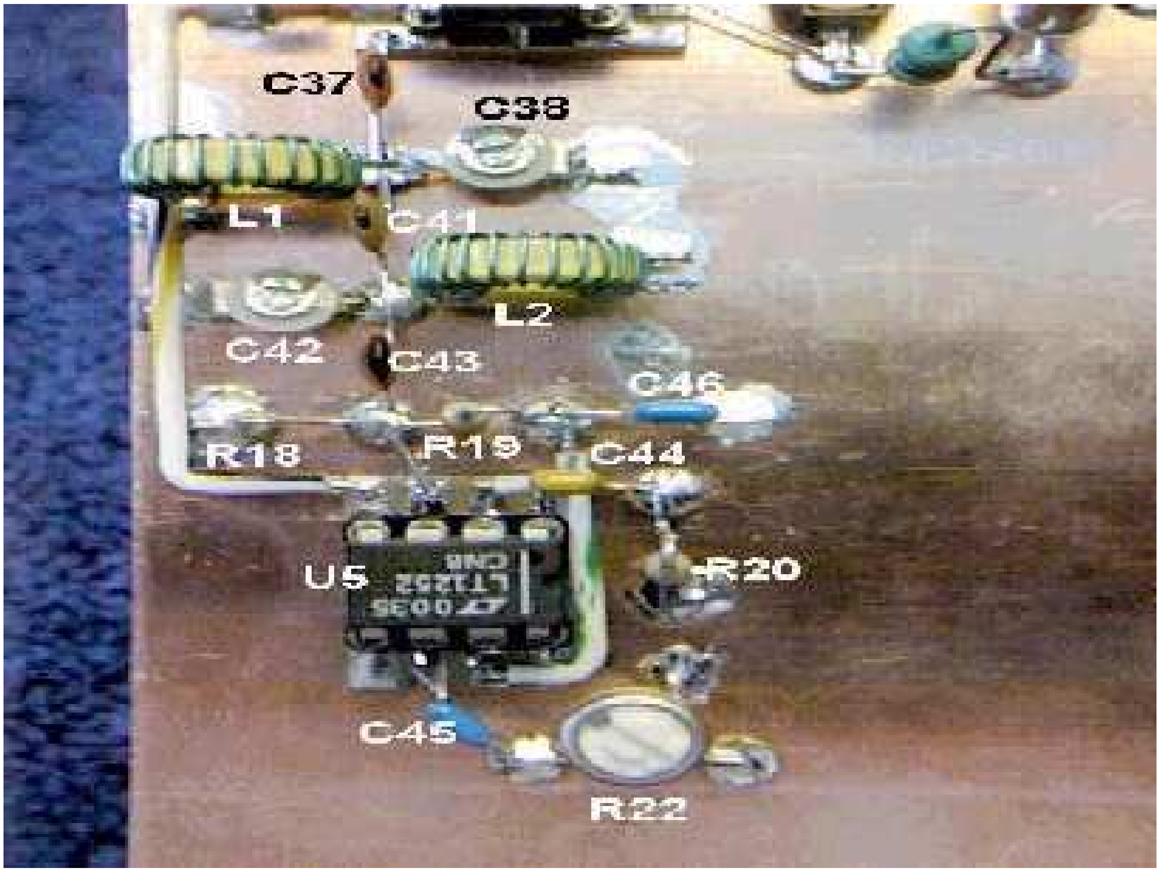

Fig. 54 Transmit Bandpass Filter and Driver Parts Layout

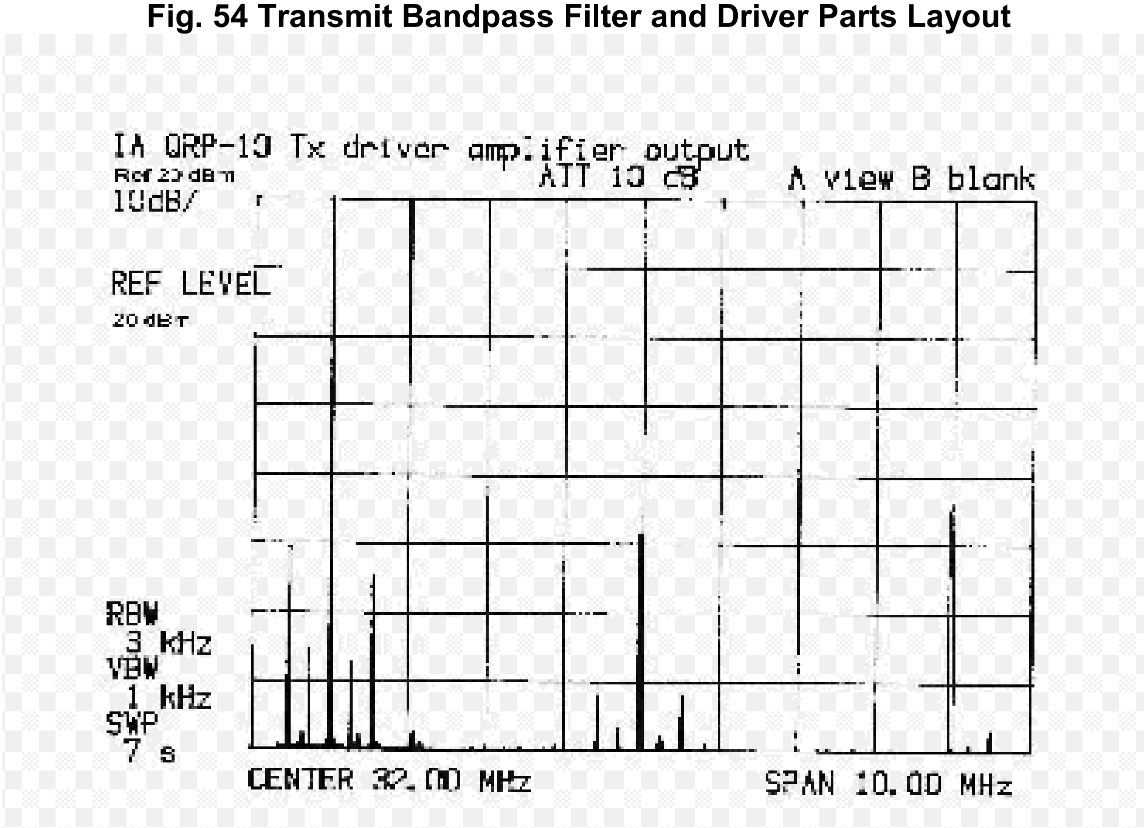

Fig. 55 Transmit Driver Output Spectrum Plot - “Wide”

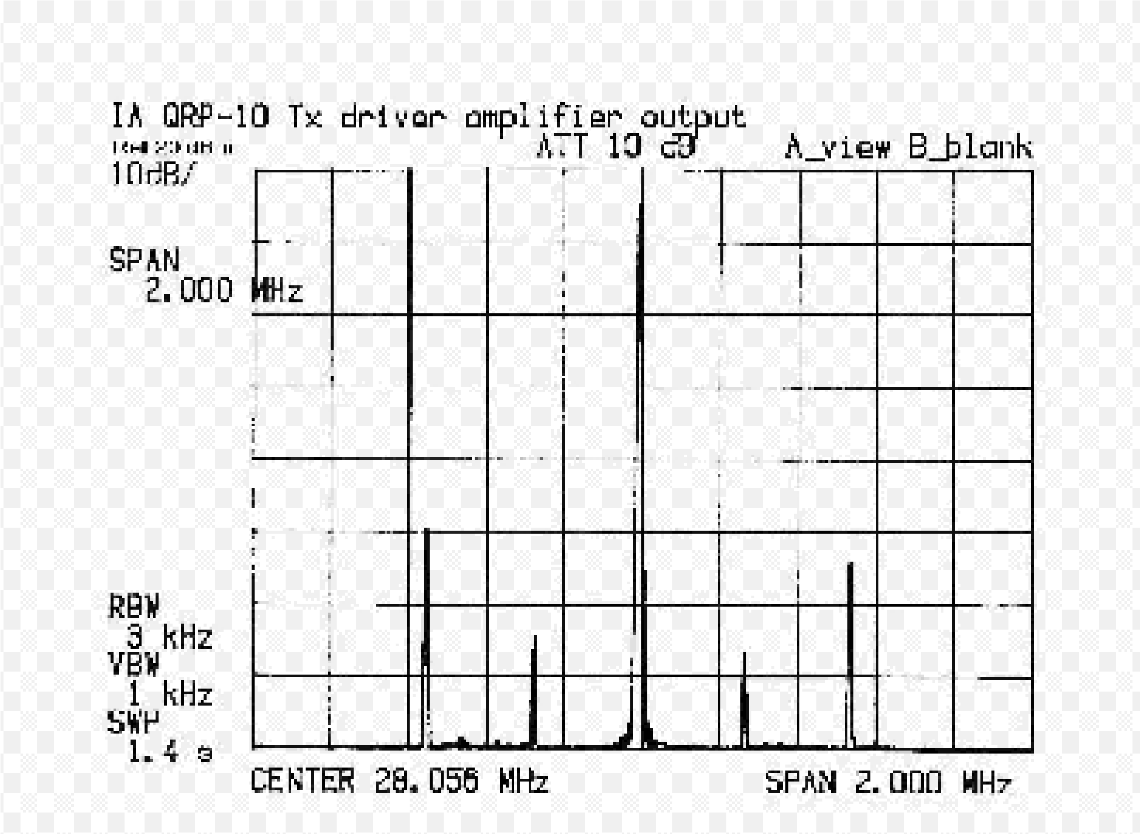

Fig. 56 Transmit Driver Output Spectrum Plot - "Narrow"

In the next installment, we'll build the final amplifier and output low pass filter. When those are complete, the rig will be finished to the point that it can be put on the air.

The final amplifier and low pass filter; hard to believe we are nearing the end of this phase of the project. Let's get started.....

I tried to follow Mike's, N0MF layout for this section, since it was laid out well. My having U5 oriented 90 degrees clockwise from that used by Mike brought about the only departure. This changed the location of the drive potentiometer a bit, and moved Q6, the MRF237 final amplifier a little farther away from the right board edge. Another change that I made was in orienting the output inductors, L3 through L5 at 90-degree angles to each other, to prevent coupling among this set of inductors. I'm not sure that change was completely necessary, but that's how I did the buildup. As a point of reference, inductors L3 through L5 measured as follows: L3 = 0.35 uH, L4 = 0.42 uH, and L5 = 0.35 uH. My measurements were done with an AADE L/C meter.

My first approach to these sections was to build them with the components specified in the schematic with the exception of RFC8. I wanted to see how well the LT1252 would drive the final without adding this inductance. The early pictures don't show this part on the layout, and that's why.

Capacitor C48 is also missing in some of the early pictures. The reason for leaving this out was suspecting that the capacitance of the Zener diode, D12, had not been accounted for in selecting the 82-pF value for C48. After corresponding with Mike, my suspicions were confirmed. A quick measurement of D12 showed that it provided about 40 pF of capacitance. Another measurement of receive strip input trimmer C100 would be providing another 25 pF, leaving about 18 pF required to achieve the 82-pF value. This is probably still a bit high, as the MRF237 has about 15 pF of output capacitance for which no compensation has been provided. However, 18 pF was the value used for capacitor C48.

Another change that should be mentioned is the addition of a pair of r.f. bypass capacitors to the junction of Vcc and RFC9. The way the temporary wiring worked out made the Vcc lead quite long. Therefore, adding two additional capacitors here was appropriate. One of the capacitors was a 1000 pF unit, and the other, a 0.01 uF unit.

Inductor RFC9 is also shown in the early pictures was 6 turns on it, and it is wound on a FT37-61 core. The reason for do this was to provide a choke with about 10 times the collector impedance. Using a choke with 5 to 10 times the collector impedance is a common practice for achieving stable final amplifier configurations. Six turns on this core provides 2.2 uH of inductance, or an impedance of about 390 ohms. I used a collector impedance value of 38 ohms, i.e. 13.8 volts for the supply, and 2.5 watts of output power.

Once all of the parts were solder in, I was anxious to fire up the transmit strip, and see how well is was going to work. However, I quickly realized that I had used up all of the TO-5/TO-39 heat sinks in my parts inventory. Nuts! Time to build a TO-39 heat sink.

Building the heat sink actually turned out to be a fun diversion. I had previously purchased some 0.010-inch thick tin plated brass at the local hardware store along with a small sheet of 0.020-inch thick copper. I "guesstimated" the amount of tin sheet need to make a small cylinder the diameter of the MRF237, by 3/8 inch in height. Using the handle of an Xacto knife as a mandrel, I formed the cylinder, and trimmed off the excess. A little bit of "tin smithing" with a pair of round nose pliers got the final diameter to fit the transistor tightly. A piece of copper sheet about 3/4 inch wide and 1 1/4 inch long was then cut. After a bit of cleaning with a "Scotch Brite" pad, the two elements were ready for the soldering iron. The cylinder was positioned vertically and centered on the copper pad, and tack soldered in place. After this step, the cylinder was held in place with a small wooden stick, while additional solder was added, making a continuous joint around the perimeter of the cylinder. When the assembly had cooled, it was cleaned with some lacquer thinner on a cotton swab.

With the heat sink installed on the MRF237, is was "show time". First, a power meter with a dummy load on its output was connected to the rig with a coaxial cable. Next, potentiometer R22 was turned to its lowest drive position. Power was applied and pin 3 of IC U4 was grounded, keying the transmit strip. As R22 was increased, power began to be delivered to the dummy load. Trimmer capacitors C38 and C42 were alternately peaked for the highest power output with R22 adjusted for about 2 watts. After the rig had operated perhaps 15 seconds, I felt the heat sink. Wow! Much hotter than it ought to be for a final running class C was my first thought. In addition, peaking C38 and C42 was most difficult; they seemed much too sensitive. Some of that sensitivity had been observed when the band pass filter was first constructed and tested, and again when the LT1252 driver was added.

Looking at the output on the spectrum analyzer provided a starting point to unravel what was going on. When the rig was first keyed, everything looked normal, and would stay that way for a few seconds. However, as the final heated up, something was becoming unstable. A very high spur at nominally 16 MHz would appear in the display, and grow in size at the temperature increased. Additional lower level spurious outputs would also appear if the rig continued to operate and heat. What was not known now was the cause. Were the observations the result of a single problem, or several related problems, as is most often the case.

Solving this mystery turned out to be a bit time consuming, but not highly technical, and is the basis for the next installment of the IA QRP-10 building saga. Until the next section is written, enjoy the pictures and spectrum plots. The two spectrum plots included with this set show the output when the rig is running in its stable mode. They show the output to be reasonably clean, and well within the FCC requirements for spurious outputs to be 30 dB below the main carrier. Also, compare these plots against those of the band pass filter, and LT1252 driver, noting the increasing power output. The reference for these plots is at +34 dBm, which is 2.5 watts. On the previous plots, we were seeing only +15 dBm, or about 32 milliwatts.

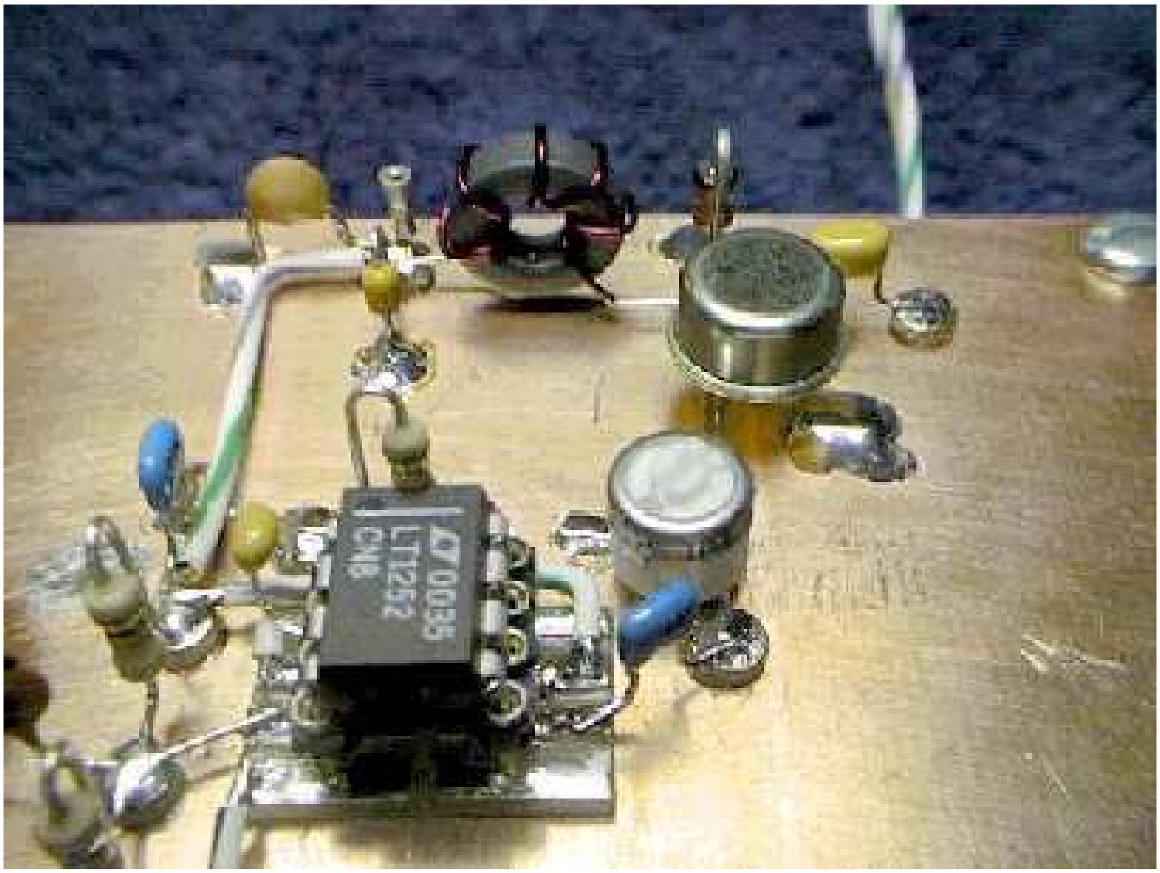

Fig. 57 Transmit Final #1

While the rig isn't quite ready to put on the air at this point, it is nearly so. The changes which will need to be made are actually rather minor, and typical of the kinds of things that are common with a brand new design. So, stay tuned.......

Our discussion is going to begin with a few general comments and pictures of the completed rig, built to the original schematic except for the receiver front-end, which has already been documented. If you have been following along, you've also seen the revised schematic for that section.

The first three pictures shows the rig with the boards stacked on top of each other, basically the configuration that is shown in the QRPp article. With the rig in this physical configuration, and the transmit strip built to the schematic, I could not get it to operate in an acceptable manner. There were three primary problems. First off, the tuning of the Tx band pass filter was much too sensitive. Getting trimmers C38 and C42 peaked was a maddening exercise. The solution to this problem was adding some fixed capacitance

Fig. 58 Transmit Final #2

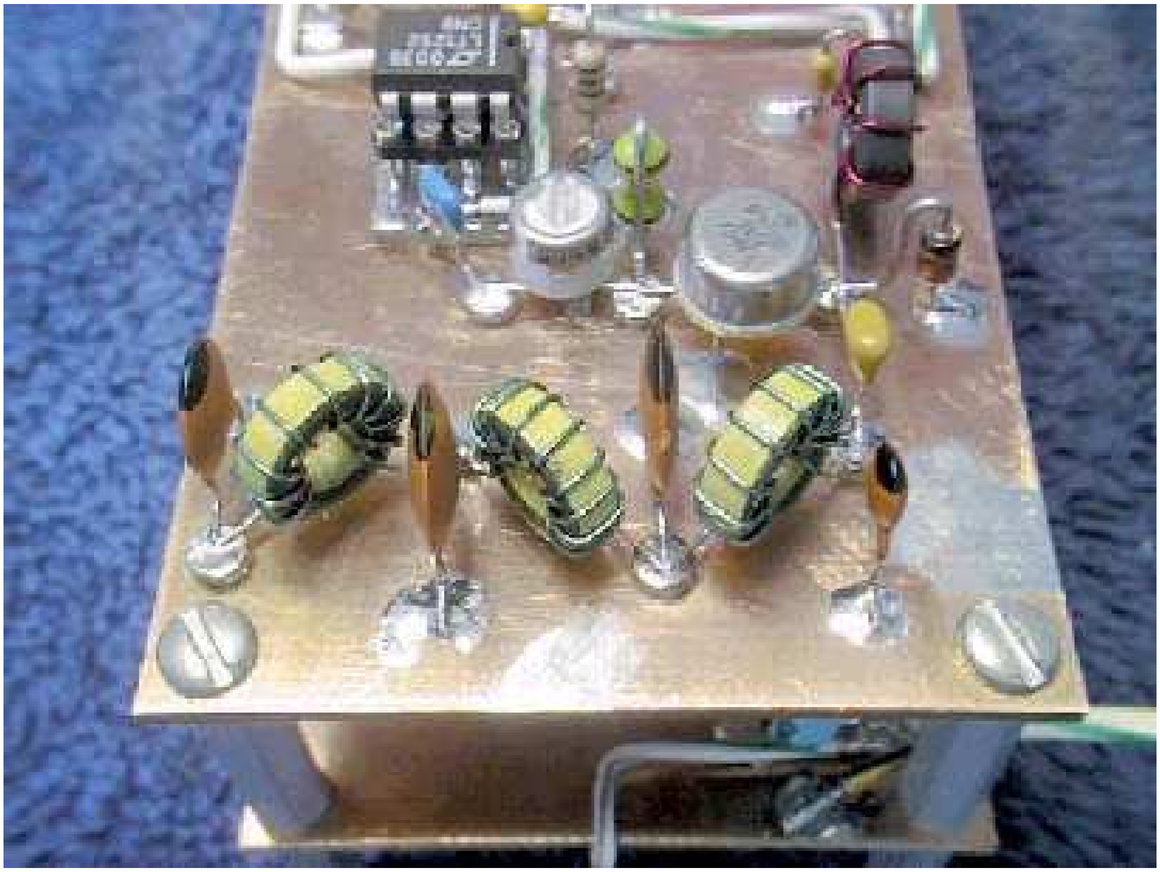

Fig. 59 Transmit Final #3

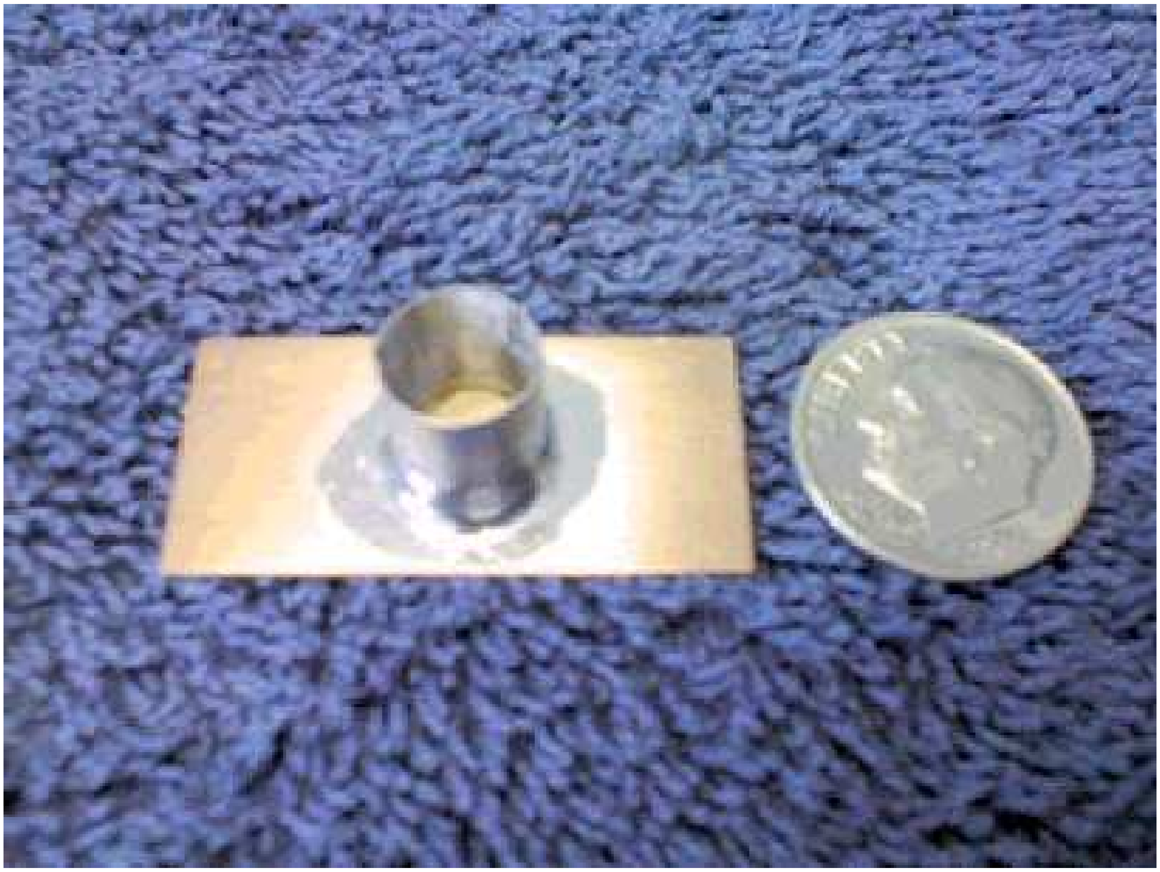

Fig. 60 Final Heat Sink Detail

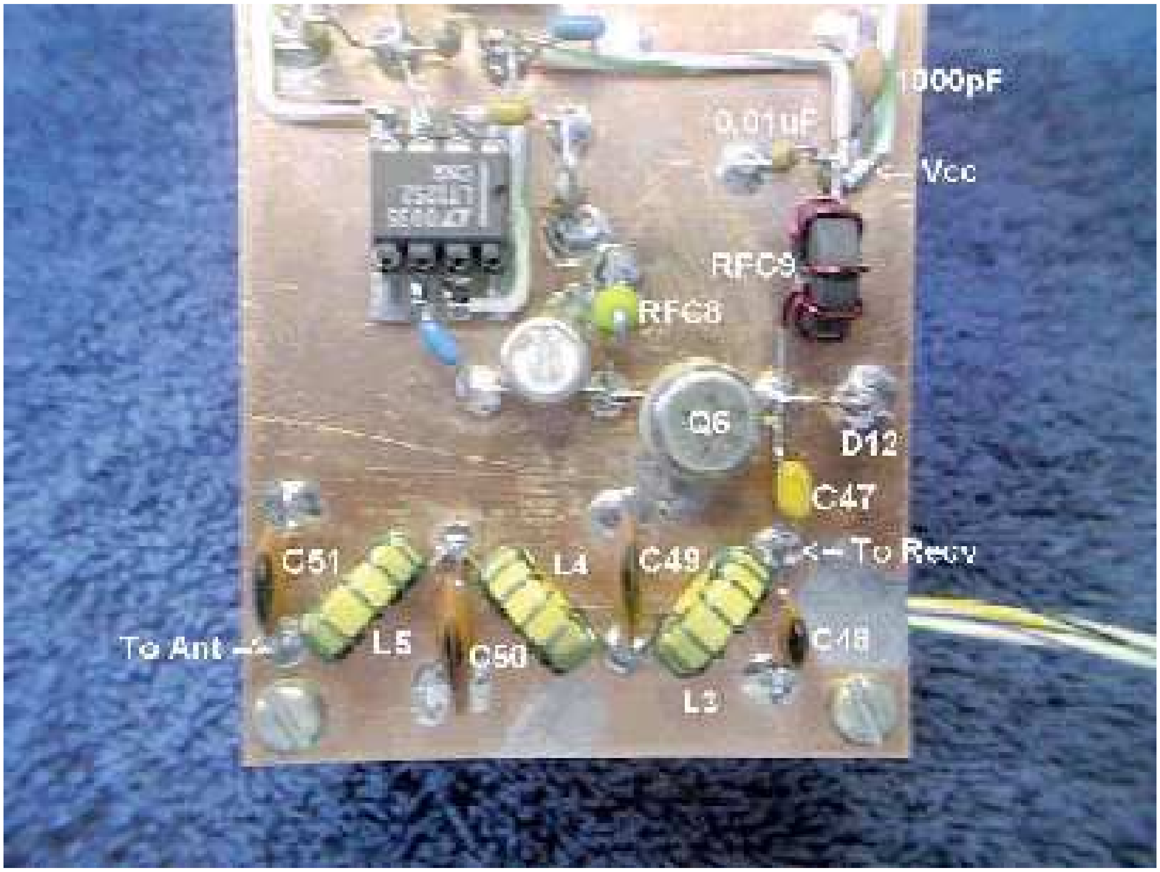

Fig. 61 Transmit Final Parts Layout

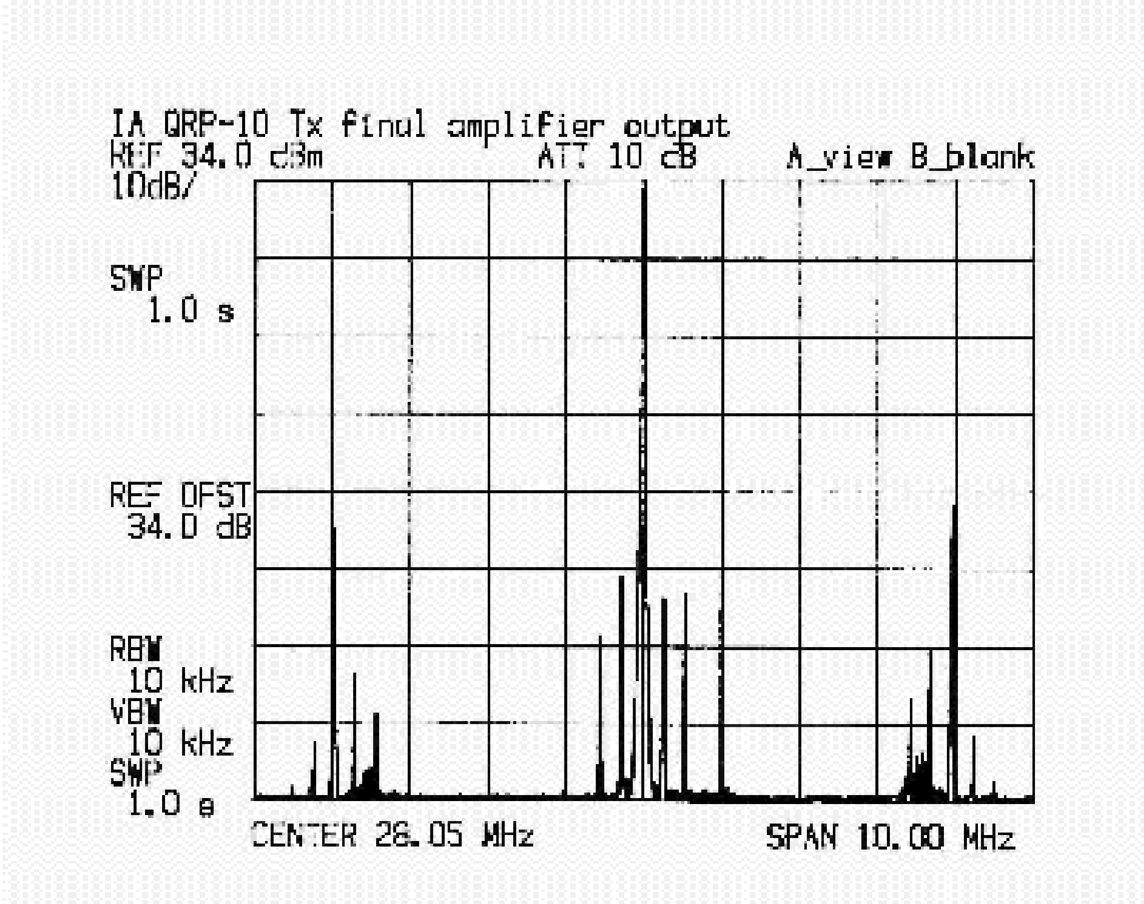

Fig. 62 Final Output Spectrum Plot - "Narrow"

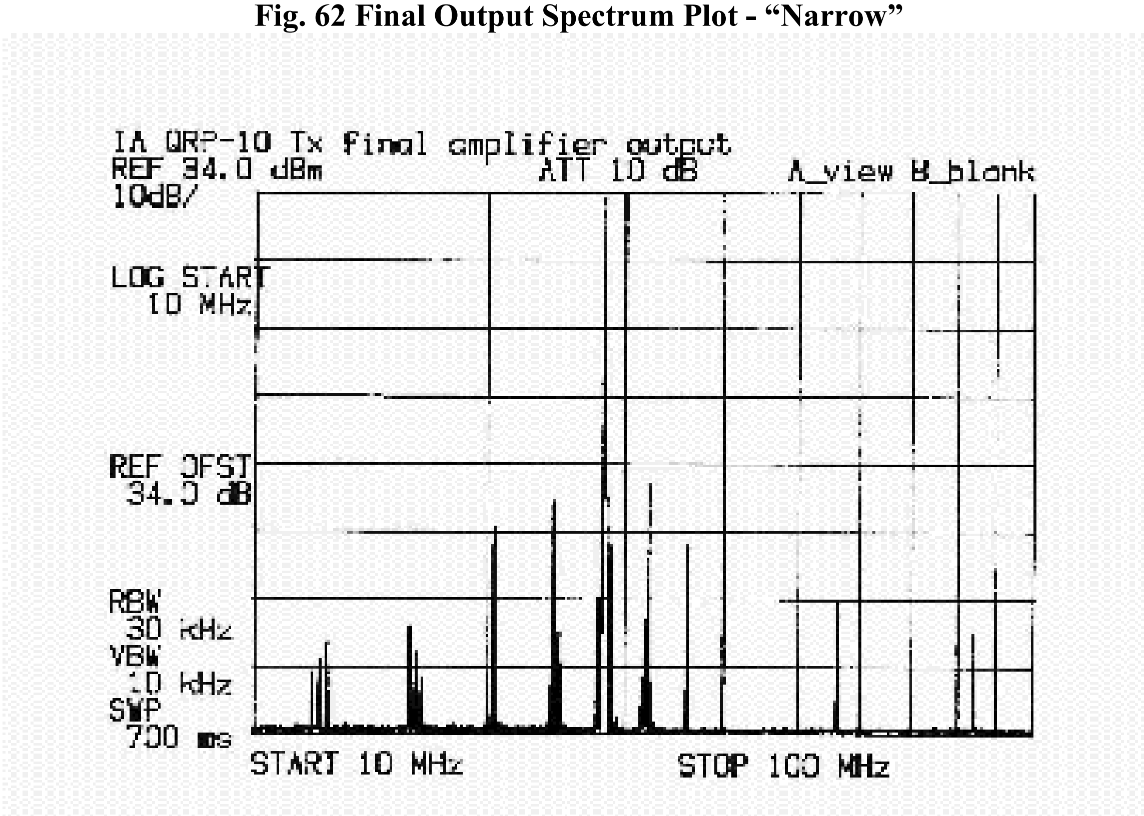

Fig. 63 Final Output Spectrum Plot - "Wide"

across each trimmer so that the trimmer capacitance was now a fraction of the total. However, to make this work well, and keep the trimmer somewhat centered in its tuning range required changing the number of turns on inductors L1 and L2. The added padding capacitors, labeled C38a and C42a on the revised schematic, were made 22 pF. This is nominally one-half the capacitance of the trimmer. Inductors L1 and L2 were then rewound with 13 turns of wire, and a resulting inductance value of 0.6 uH, as measured on an AADE capacitance meter. With these changes, the peaking of the Tx band pass filter could be easily accomplished, and was stable when the tuning tool was removed.

The next problem tackled was the overheating of the MRF237 final amplifier transistor. Solving this actually involved two areas of the circuitry. It was found that the gain of the LT1252 needed to be raised some and the value of the base r.f. choke needed to be lowered substantially. The gain of the LT1252 was raised to 11 by changing R20 from 120 ohms down to 62 ohms. Capacitor C44 was also changed to 1000 pF, but probably could be left at its original 820-pF value. My reason to making it 1000 pF was because that is the value used by Elecraft in the K1 design. They have much more experience using the LT1252 than do I, so I deferred to their value. The r.f. choke, RFC8, in the base circuit of the MRF237 was lowered to a final value of 0.56 uH. This value was found experimentally, based on finding a value, which raised the base drive, and consequently lowered the device temperature, without causing instability. Choke values below 0.47 uH caused the final amplifier to be unpredictable in its operation. Along with these changes, capacitor C45 was changed to 560 pF to reduce the coupling between the driver and the final. With these values “in circuit”, the transmit strip operated with much greater stability, lower operating temperatures on the final transistor, and predictable and stable tuning. The final still gets hotter than I think it should running in class C, but attempts to reduce it more have failed. My advice is to put a rather large heat sink on it and keep the temperature down that way.

The last change to be made was discovered after the rig had been operated for a few days. During this time, the QSK keying was less than adequate, since the audio output contained significant “thump” when keying. The solution was to move the location of diode D11 from after the connection between U4, pin 3 to U5, pin 4, to between these two devices. With that change made, the keying was much improved. The downside, however, was that the higher reverse bias normally supplied by the current flowing out of U5, pin 4 was not there to reverse bias diodes D1 and D4. However, the reverse bias supplied by U4, pin 3 seems to be sufficient. The thump is gone, and the audio output during transmit is clean, undistorted, and at an appropriate level.

If you look at the “as built” transmit schematic that is included with this discussion, you will see all of the changes that have just been detailed.

The last item for this section is a final spectrum plot that was created after the above changes were in place. This plot shows the improvements resulting from the changes, especially the close in spurs that have been reduced in amplitude. The worst spur just above 21 MHz is down approximately 48 dBc, and everything else is down approximately 60 dBc. When this plot was made, the rig was operating at 2.5 watts into a 50-ohm dummy load.

While the rig still needs to be packaged, it is complete electrically and has been operated a few times. The first contact was with PA3CVR in the Netherlands with a 529 signal report. Subsequent contacts over the next few days included DX stations in Germany and Sweden. So far, 4 countries and 3 states, including Alaska have been worked, all at 2 to 2.5 watts into my 180 foot end-fed long wire antenna. All reported the rig sounded great.

The last installment for this project will detail the packaging used for this terrific little rig so come back for one more visit!

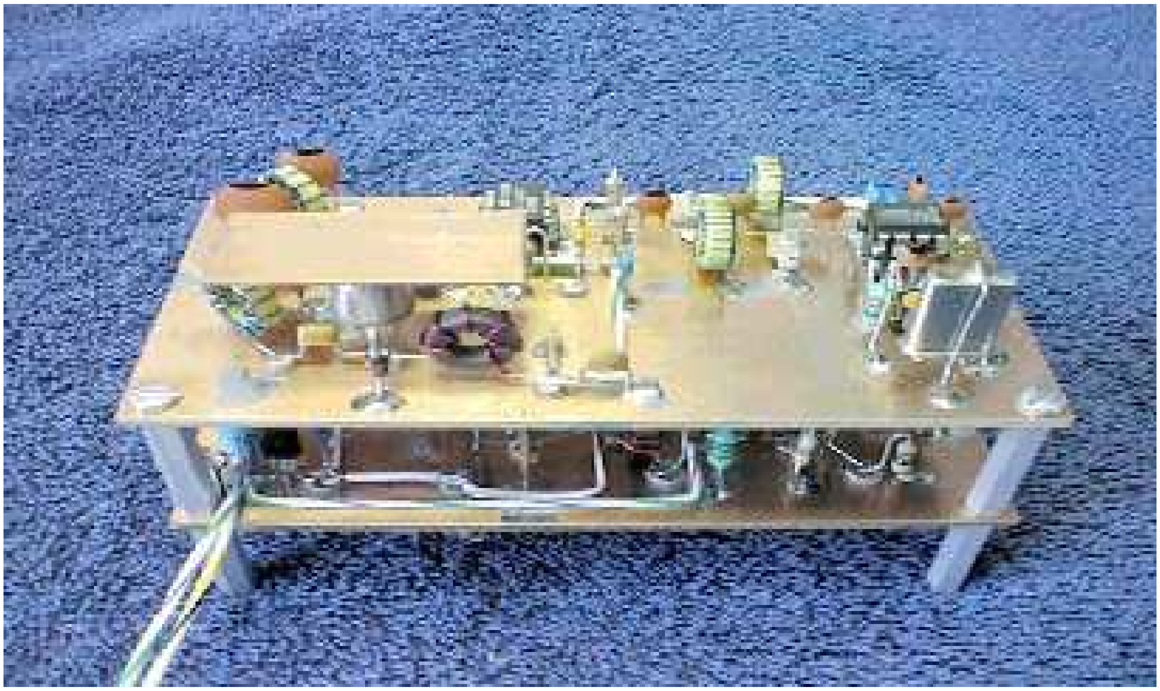

Fig. 64 The Finished Rig

Fig. 65 The Finished Rig

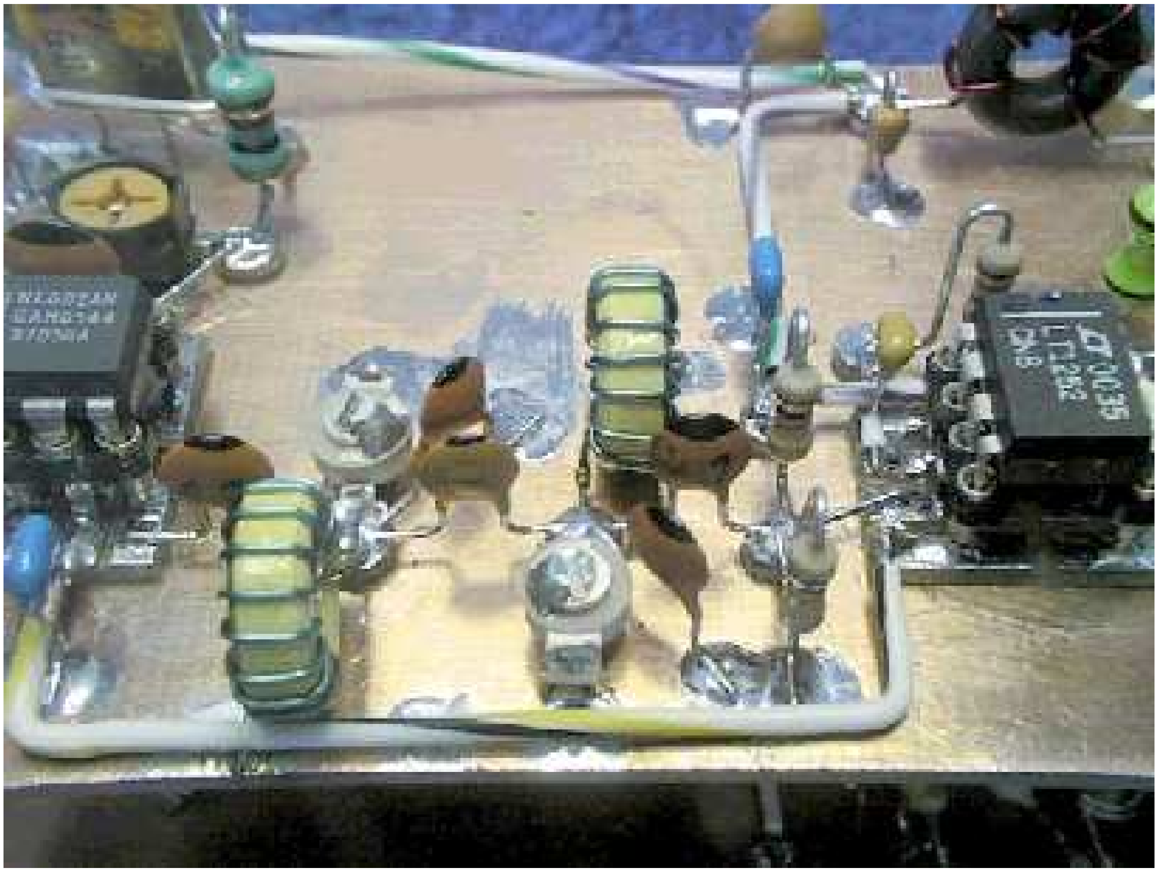

Fig. 66 Revised TX Bandpass Filter

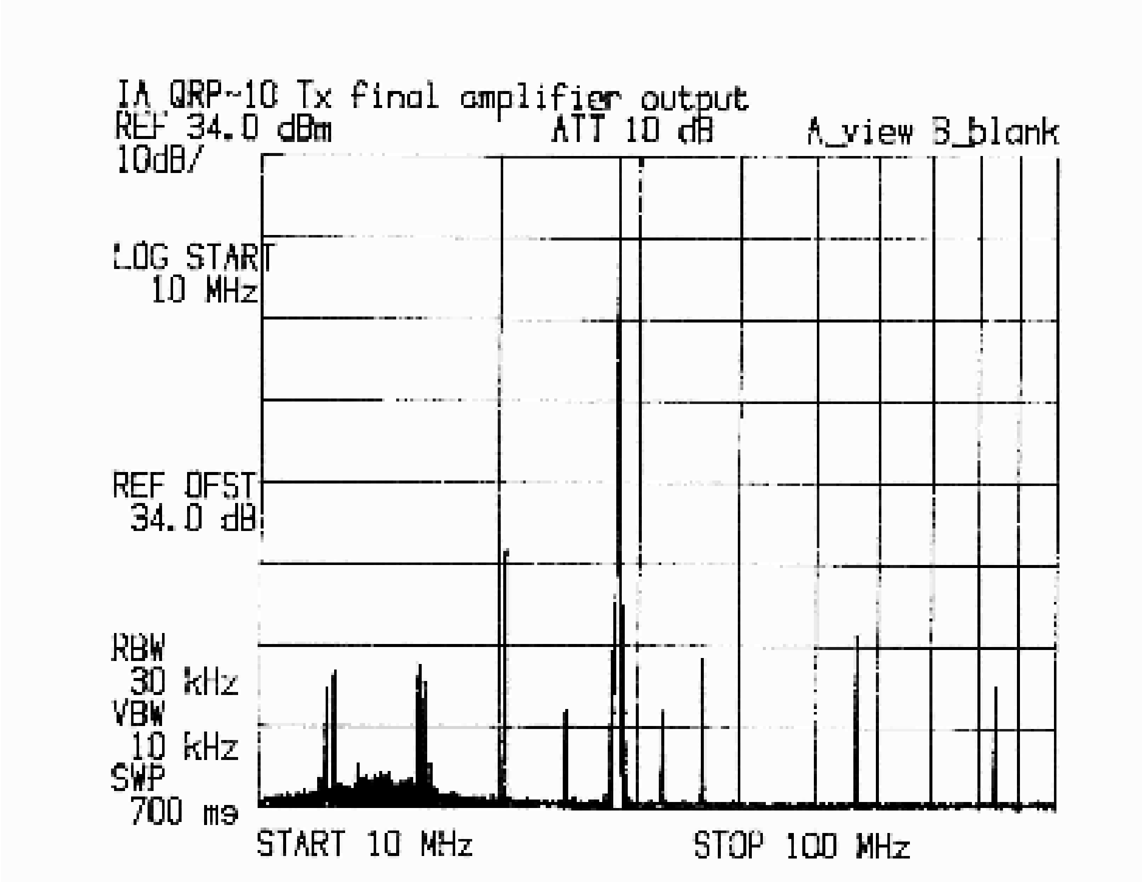

Fig. 66 Final Output Spectrum

Here we are at the end of what has been a delightful and interesting project for me. As always, I've learned some new things, had a blast building the IA QRP-10, and I hope, stimulated some desire on the part of many readers to build one. Modifying the design slightly has added to the robustness that Mike, N0MF had already built into the rig.

There are still some areas that could be improved, mainly in the way the VXO is done. If one were to add a buffer amplifier between the oscillator and the doubler circuit, loading of the oscillator would be minimized. This would allow a single crystal to be pulled over 25 KHz probably. By adding a second 16 MHz crystal in parallel, the tuning range could well reach 50 KHz. Certainly worth doing for the cost of another transistor and a few more parts.

Changing the physical layout would enhance the stability of the design. I'd like to see someone build the complete VXO and receiver on one side of a 3 X 5 or 4 X 6 inch substrate, and the transmit strip on the opposite side. Keeping the number of substrates down to two would minimize ground loop problems. Using three substrates allows smaller packaging, but aggravates r.f. stability with the multiple grounds required.

The remainder of this part will be short, and focus on the packaging of the rig. I decided not to use the same case as Mike, as I wanted more panel space for the nifty big knob that Doug, KI6DS, had sent with the box o' parts. After pawing around in my stash of cases, I found another TenTec TP-42, the same case used for the original 2N2/40 transceiver, and my 2N2/6 transverter. This case is 2 3/8 inches high, 5 3/4 inches wide, and 4 3/4 inches deep. After removing the spacers holding the QRP-10 boards together, it became clear that the transmit board would fit very nicely installed vertically at the back of the case, with the VXO/receive board installed horizontally. This configuration gave access to all of the trimmers with those on the transmit board being through appropriately placed holes in the back panel. The boards were assembled in this configuration, and the first series of pictures is a walk-around that configuration. As can be seen in these pictures, all of the between-board wiring has been completed. There are also pigtails for the key line, antenna coaxial connector, and the tuning controls are wired.

Once the board assembly was interconnected, the case was drilled. Hole layouts were created with WinBoard, the schematic capture program that I use. Back and front panel layouts were printed and affixed to the respective panels, and the appropriate sized holes drilled. After deburring the holes, the case was ready for the electronics. Two pictures of the drilled case are included in the picture set. Painting will have to wait for warmer weather, as I don't spray paint in the house. It's just too noxious and dangerous.

The board assembly was then installed in the case and connectors added for which pigtails existed. An assembly consisting of the volume control, headphone jack, and AGC LED was then wired up using several pieces of flat, multi-conductor cable. Once this assembly was built, the controls were added to the front panel, and the ends plugged into the pin receptacles on the receive strip. As a refresher, the pin receptacles are single terminals removed from an IC socket. The final set of pictures is another walk-around of the completed IA QRP-10 rig installed in its case. The only missing item is the heat sink that will be used, which can handle more dissipation than the homemade unit shown in these pictures. When the rig is being used, the case is open, and a small fan is allowed to blow across the heat sink to keep the final transistor cool. The commercial heat sink is sized such that it will transfer part of the heat load to the rear panel.

We are at the end! Thanks to all who have followed along, and especially to those who took the time to send email with comments and encouragement. It is always gratifying to know that work you are doing is appreciated and being used. If you don't have a 10 meter QRP rig, consider this one. It is easy and fun to build, and even more fun to operate. 72, Jim, K8IQY

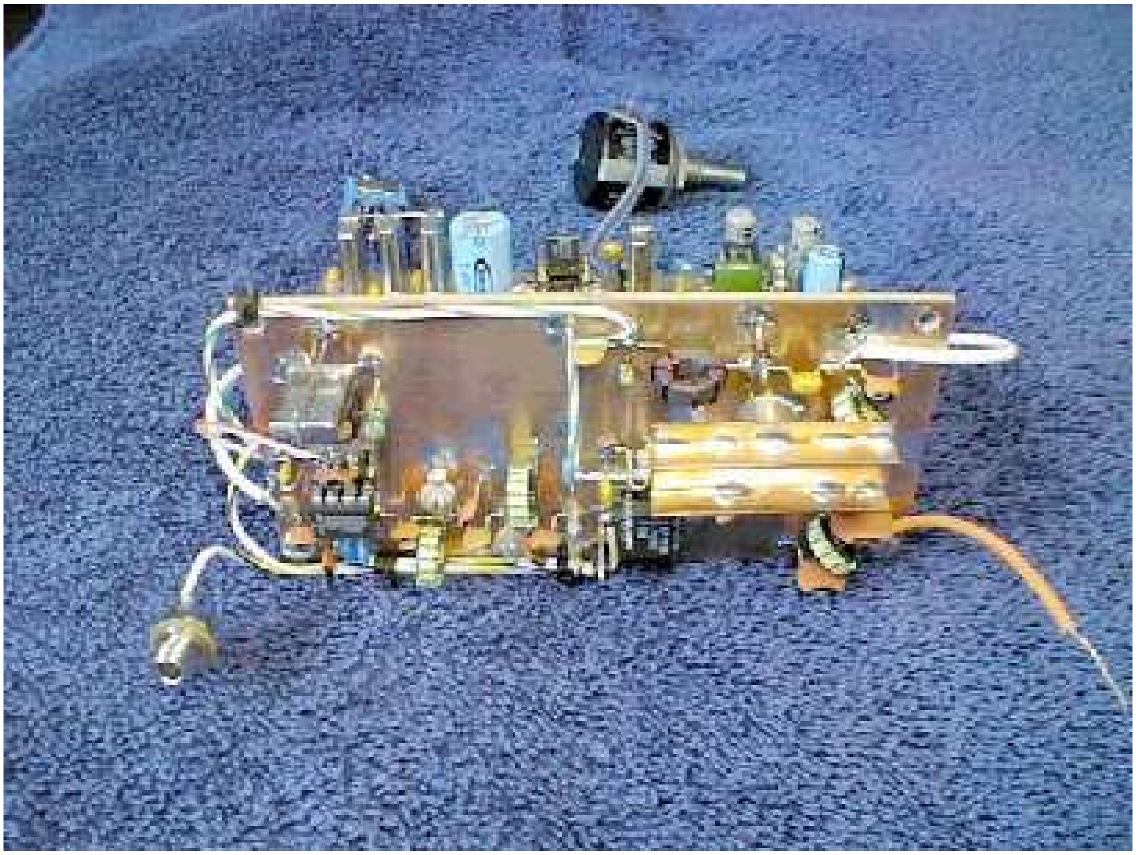

Fig. 67 The Finished Rig Ready for Packaging

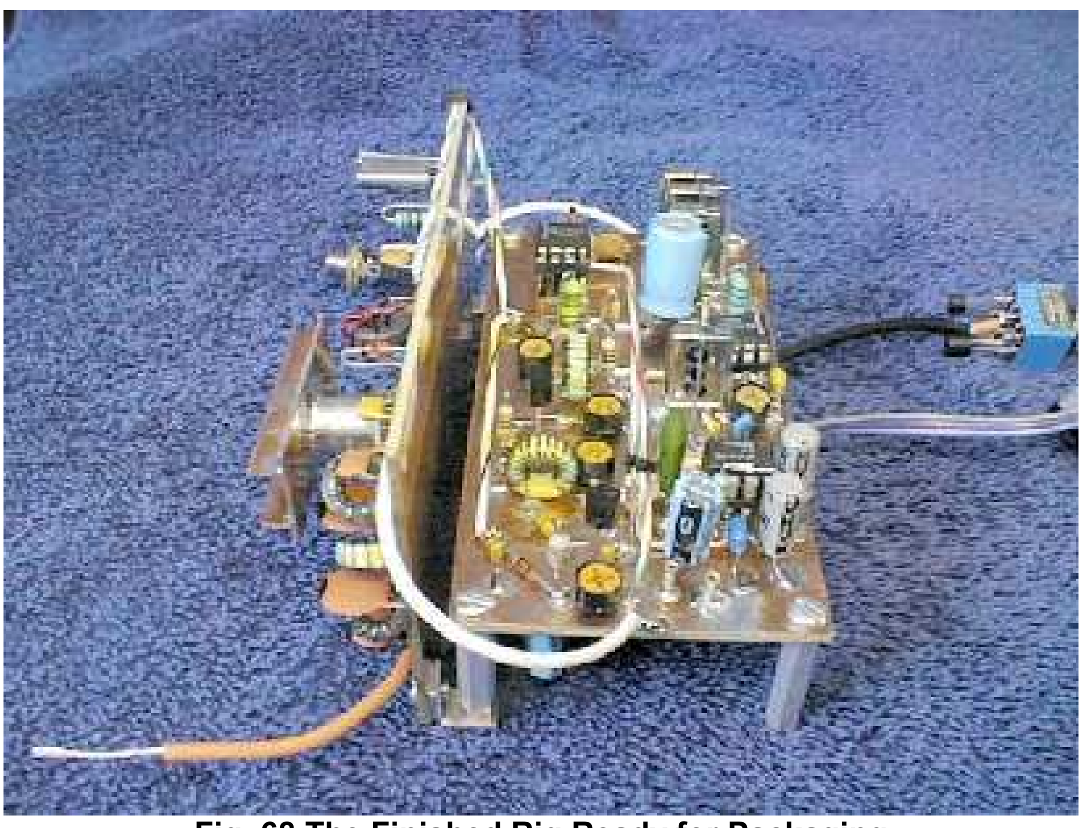

Fig. 68 The Finished Rig Ready for Packaging

Fig. 69 The Finished Rig Ready for Packaging

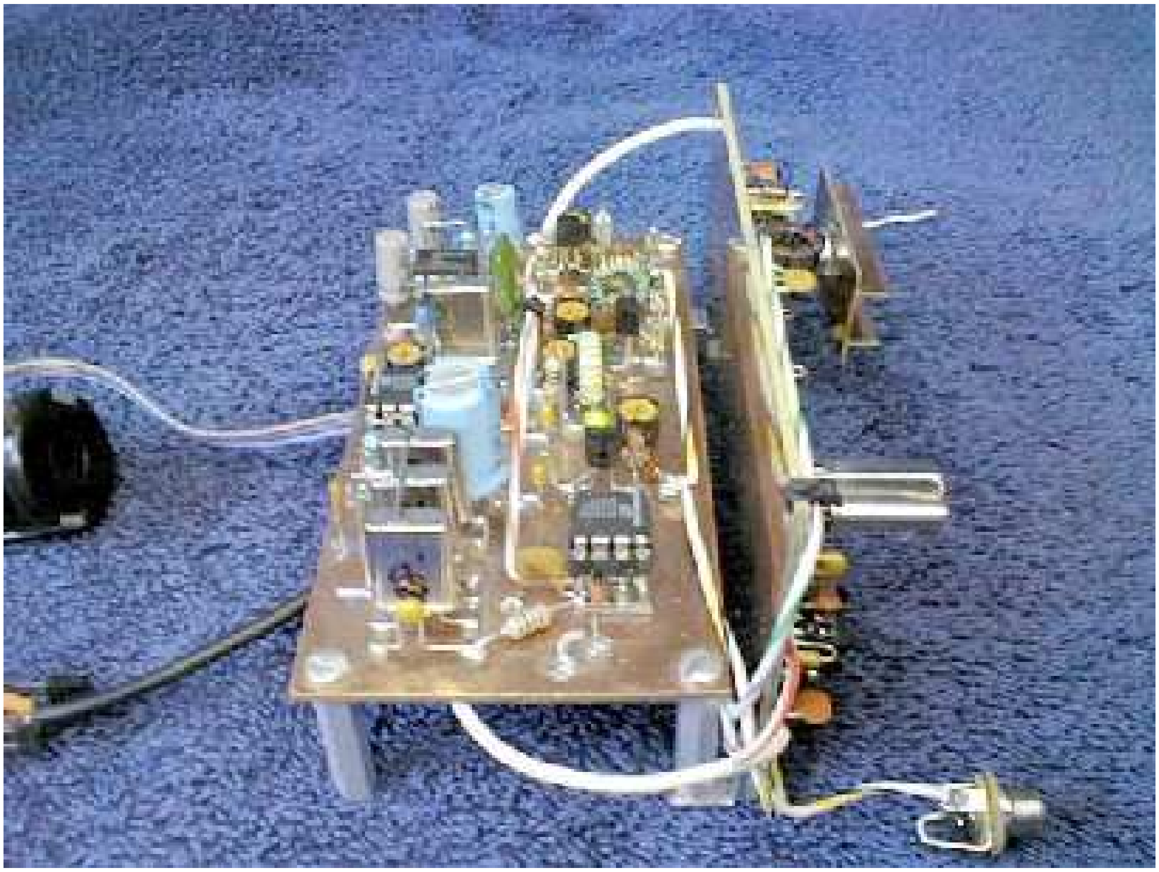

Fig. 70 The Finished Rig Ready for Packaging

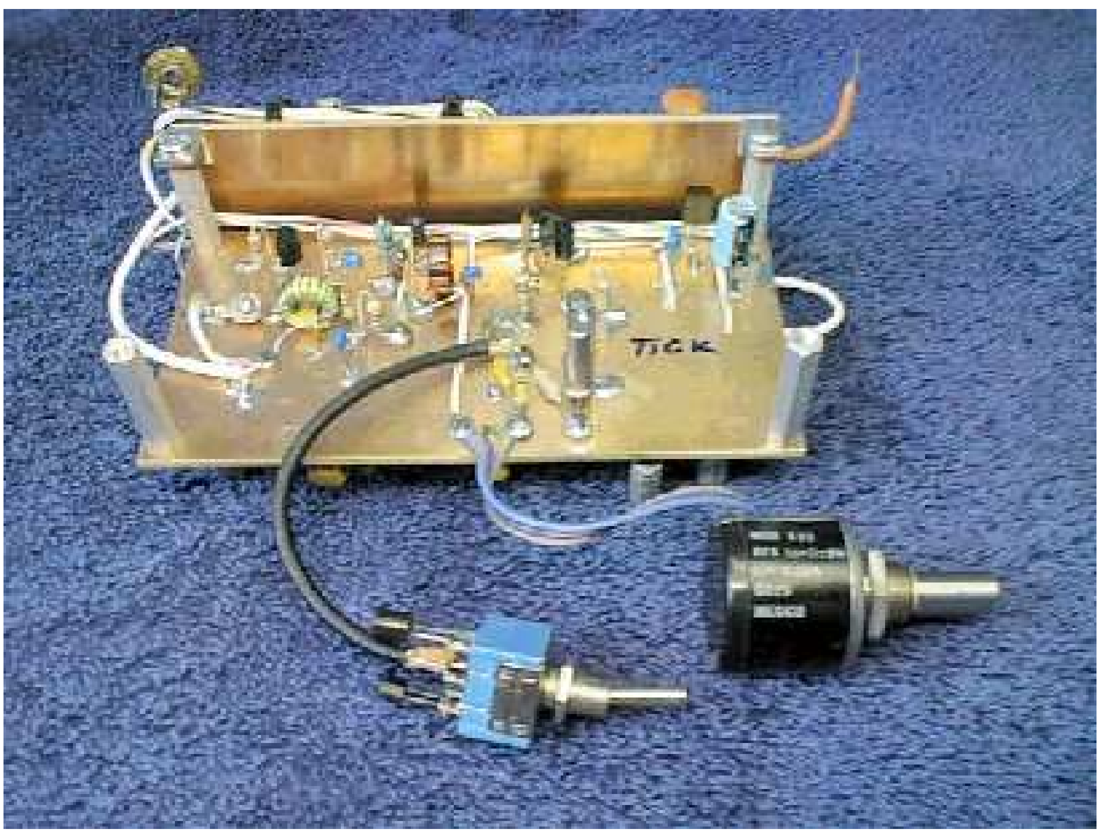

Fig. 71 The Finished Rig Ready for Packaging

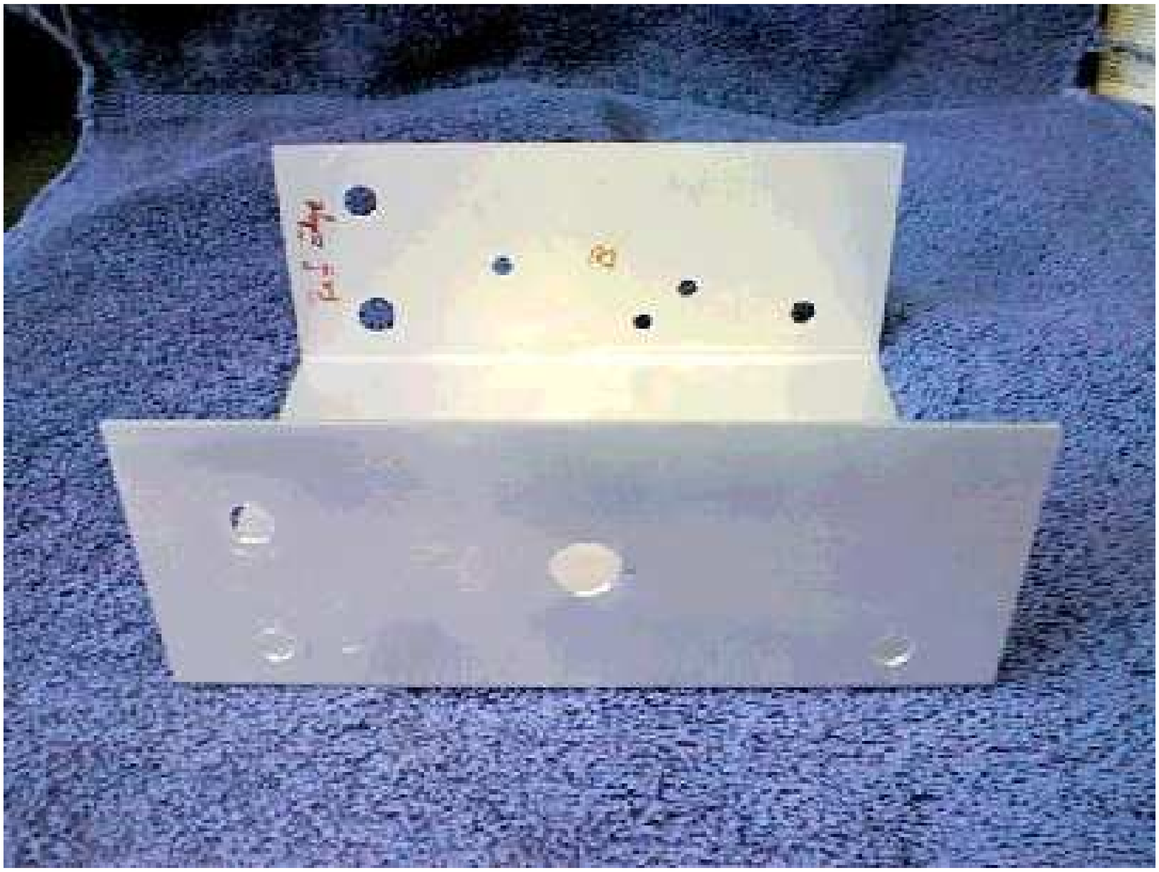

Fig. 72 The Drilled Case Front View

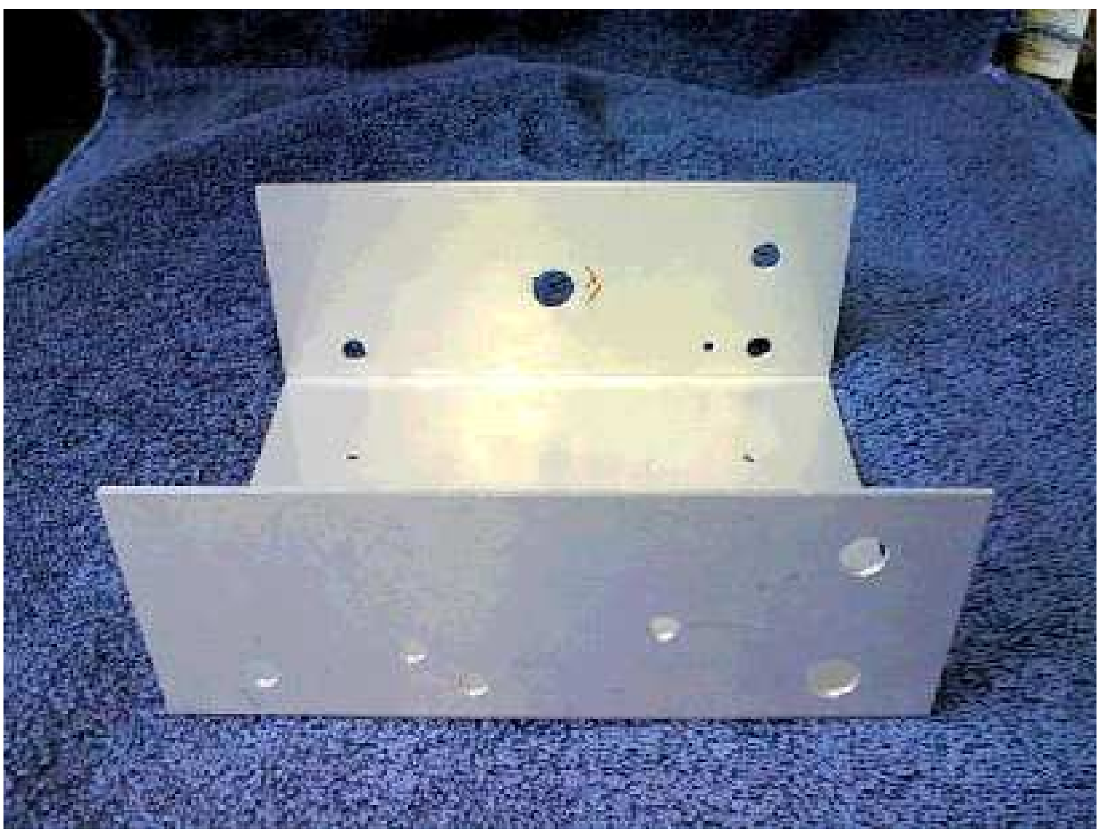

Fig. 73 The Drilled Case, Back View

Fig. 74 The Competed Rig, Front View

Fig. 75 The Complete Rig, Top Front View

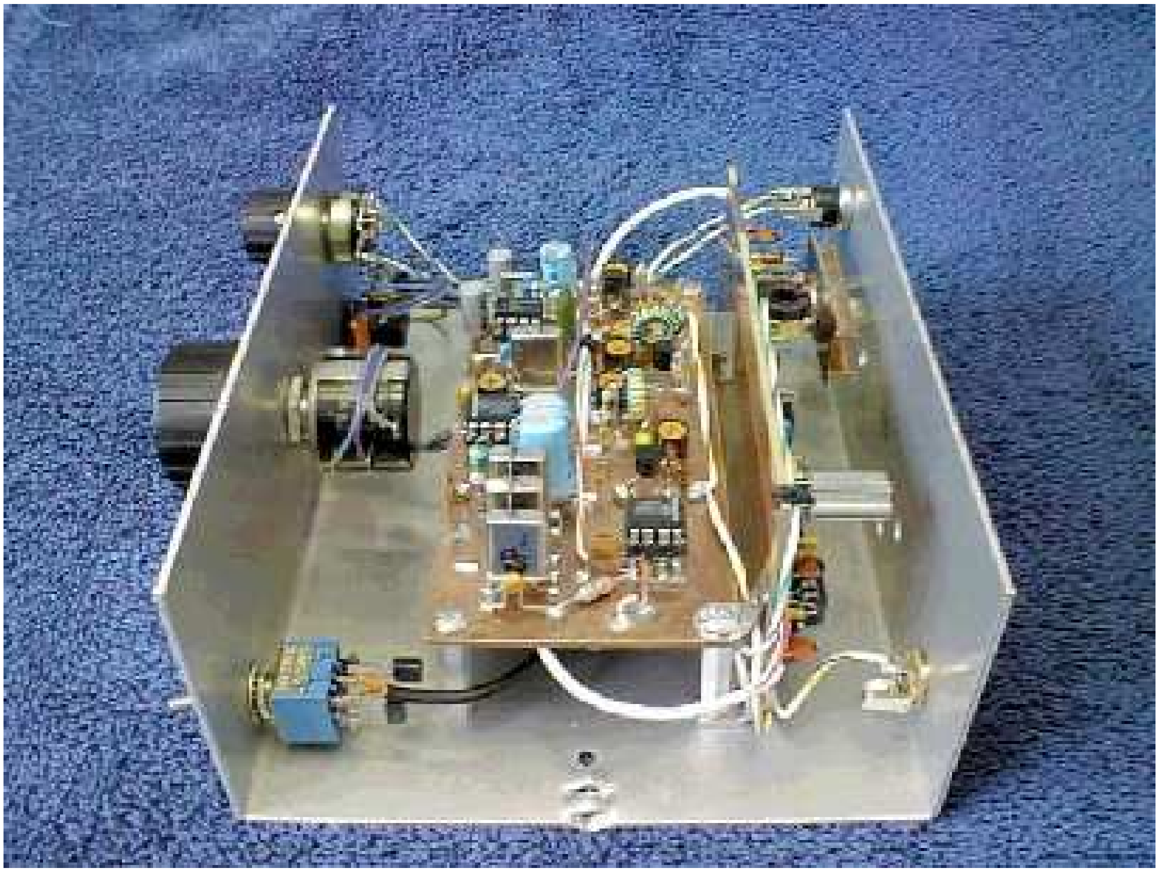
Fig. 76 Complete Rig Right View


Fig. 77 Complete Rig Back View

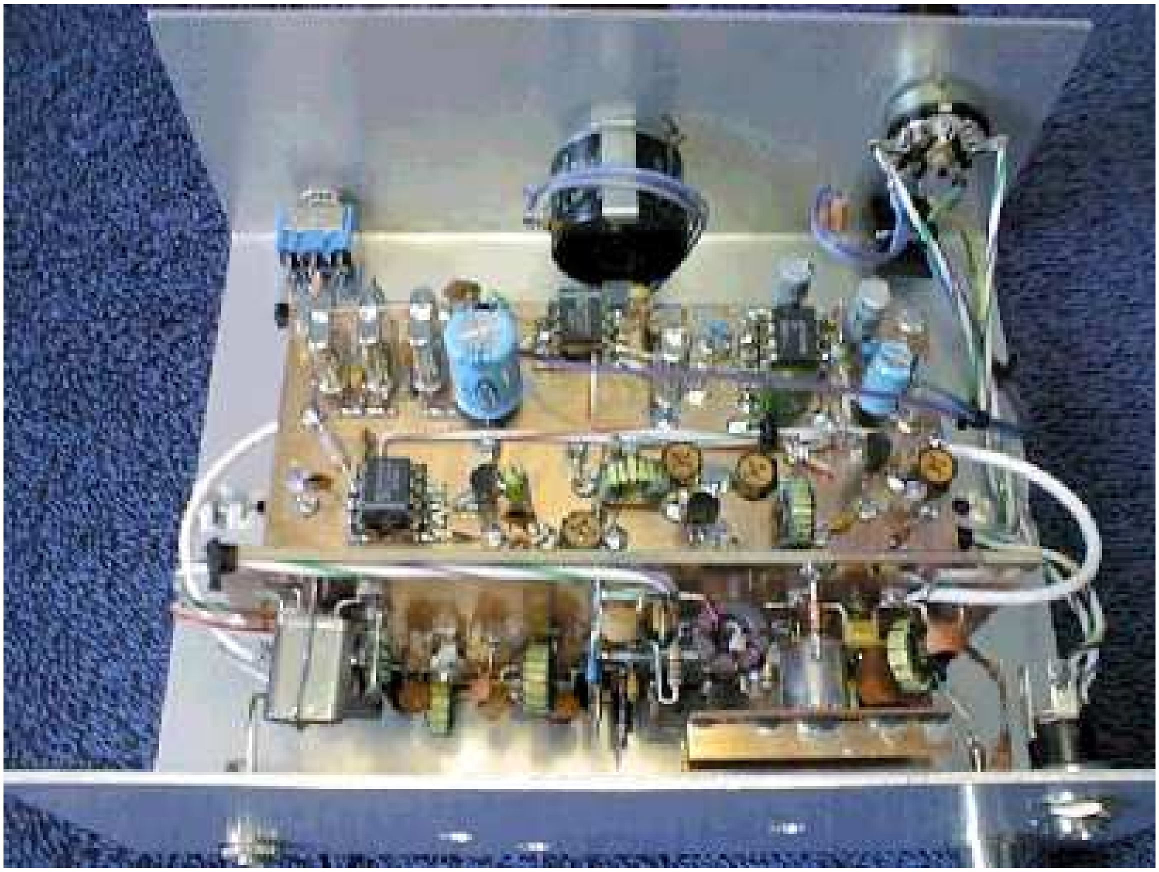

Fig. 78 Complete Rig Back Top View

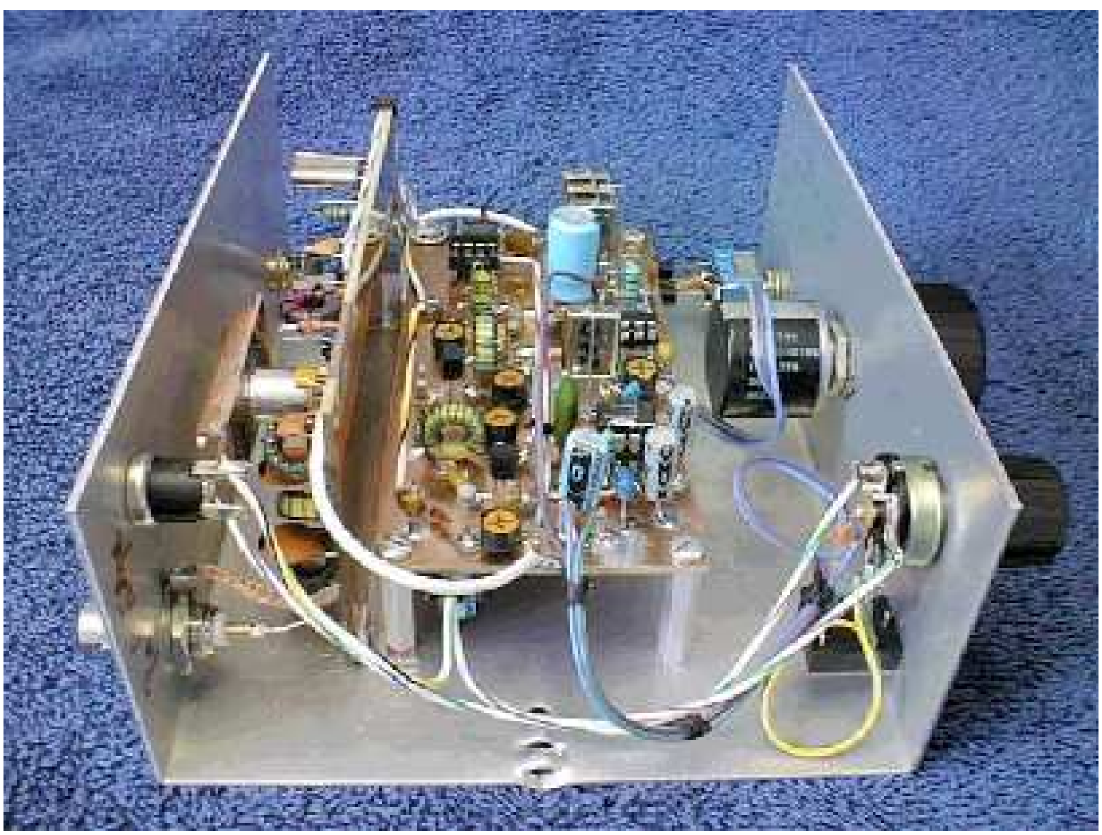

Fig. 79 Complete Rig Left View