Additional NorCal 20 Construction Notes

David Fifield, AD6A fifield@pacbell.net
31 Jan 99

Folks,

Thanks to input from several of you, here are a few things to note in addition to all the stuff in the manual:

1. The 82pF C0G capacitor (C108) for the VFO may be labeled either '820' or '82J', these mean the same thing.

2. Some rigs will have NE602AN chips (U4 and U7) as per the circuit diagram and parts list and some will have NE612AN chips instead. There is no need to panic. At HF, these two devices can be considered identical.

3. Use the drawing on page 8 to construct the VFO coil (L1) rather than the text. The screw is easier to fit from the top of the coil. I changed it for the drawing and then forgot to modify the text.

4. There is just enough red 26awg wire in the kit to do all the inductors. If you run out of the red stuff, use the green 26awg wire instead - there's plenty of that, and it'll make the rig more colorful.

5. When you have built the VFO, audio amplifier stages and AFA you may find that everything appears normal except the AFA won't announce anything for you. This means the VFO is off frequency too far - usually too low (e.g. 4.8MHz) - spread the turns of L1 more and twiddle with TC1 until you get an AFA response.

6. It's okay to put any/all of the IC's into sockets if you like. Of course, the better the socket, the better performance and longevity you are going to get out of the deal. Try to use gold plated turned-pin sockets if you can.

7. You can modify the values of R1 and R100 to use different values of 10 turn pots you have handy. However, only go up as far as 20K maximum - 50K pots, for instance, will have problems which I will explain privately if you like (it's very long winded...).

That's it for now. Please let me know if you spot anything else or have any hints for people, I'd like to keep everything compiled neatly together.

Have fun putting it together, and don't forget to get on the air and actually use it too!

72 de Dave Fifield, AD6A

NC20 Tips

Gary L Surrency, AB7MY

Gang,

Nice comments by Chuck on his kit.

Chuck, on your paddle problem:

This sounds to me like the classic "piece of component lead flash, or other trash under the socket". I have had this kind of hair-pulling problem myself a few times. Closely spaced component holes in the PCB will sometimes shave off a tiny bit of solder plating as you press a transistor or other device down near the board. S&S Engineering specifically mentions this in their kit manuals as a cautionary statement. I had some of the debris from a part slip under one of the IF filter crystals in my SW-40+ kit, causing it to periodically go "deaf". I finally fixed it by using a short blast of compressed air on the top of the PCB. Similar intermittents can easily occur under IC's and/or their sockets.

When I first built my prototype NC20, I had no AGC action. It was due to a transistor lead sliver bridging across the top of the J310 AGC amp. Since it was still partially stuck the the transistor's lead, I had to use a tiny tweezer to fetch it out. These are just some tips for anyone who might come up with a similar problem.

I grounded my crystals the following way:

Solder them to the PCB first, then pre-tin a small spot about two-thirds of the way up the side of can, on the same side as the ground hole in the PCB. A little extra flux may help wet the nickel plating. Then, solder a cut-off resistor lead straight into the hole. I merely used a resistor with full leads, as I knew there would still be plenty of resistor left after I "borrowed" some for the crystal. ;-) Solder it in, straight up-and-down.

Then, bend over the lead so that it contacts the crystal can where you just pre-tinned the crystal case. Cut off the resistor lead to the correct length, then hold it against the crystal with a small screwdriver or other tool while applying the soldering iron. Use a bit of fresh solder if needed, or pre-tin the resistor lead. Remove the soldering iron, and hold the lead still for a second or two until the solder cools.

The whole idea is to have both the crystal and grounding wire all prepared before actually doing the soldering, as you don't want to overheat the crystals and change their resonant frequency - thus altering the careful matching Doug has already done for us. You also do not want to rupture the hermetic seal at the bottom of the crystal package, since that is where the crystal is assembled. That is why it is better to stay up near the top of the crystal can when soldering.

As for the 84 KHz of tuning range Chuck got:

I also had about this much band coverage with mine. Originally, we were getting about 71 KHz. This was after the NC20 debug team decided to dispense with a 1N914 diode in the tuning pot voltage divider, as it was a small source of temperature-related VFO drift. That is why one extra diode appears in the kit, as Doug had already packaged them! ;-)

Since I don't do any operating above 14.071 MHz, I felt I would reduce the range a bit. R9 was reduced from 220k to 200k. I couldn't find a 200k resistor in my junk box, so I used two 100k resistors in series. One end of each resistor goes into the PCB holes, and the top leads of those resistors are bent together and tack-soldered. Other tuning ranges are easily accomodated this way.

Also, I wanted less RIT range than the +/- 3.5 KHz or so I was getting. So, I experimented and found that by using a 680k resistor for R5, instead of the original 330k, I could obtain about +/- 2 KHz RIT tuning. It varies some from the lower tuning end to the higher end.You may adjust this resistor to produce the RIT tuning you prefer. I found that the center detent control was making it difficult to move the RIT frequency just a little bit during a QSO, because the large original RIT range required only slight movement off of the detent, and therefore it would "snap back" to center too easily under these conditions.

If your rig seems a little bit lacking in sensistivity, here are some suggestions:

As Dave Fifield mentioned in the manual, some J310's are better than others. Dave Meacham and I both found we needed to replace some J310s in our kits to produce satisfactory VFO and RX sensitivity. Now, let's not all mass email Doug for replacement J310's un-necessarily!! Remember, we *were* building prototypes, and in the original prototypes we had to ground the Gate lead of RF amp Q5, since it was missed in one of the early PCB designs.

Also, we all did a heck of a lot of work around the VFO JFET, Q2. So it is possible that during our work, we partially damaged some of the J310's by handling or replacing them. You all should be careful to not static damage (ESD) any of the active devices in the kit while building. Winter time is especially conducive to static situations, so don't automatically dimiss the ESD precautions that should *always" take place with handling semiconductors.

Keep in mind that many JFET's vary a lot in performance. It's nice to have some spares for this rig and other projects, so get some for yourself and stock up. RF amp Q5 and VFO oscillator Q2 are probably the two parts most likely to affect performance that use the J310.

A small improvement in RX sensistivity and improved noise figure (NF) can be achieved by replacing the 2N5179 transistor at Q7, with a Motorola MRF904. This device is in a similar 4 lead metal package, and has a lower NF (1.5 NF vs 4.5 NF, and ft of 4 GHz vs 1.4 GHz) than the 2N5179. If you have some or can get them easily, you might give it a try. I saw a small improvement in my rig this way.

But beware! It's not easy to unsolder a 4 legged transistor unless you

really know what you're doing, so try it before permanently installing either device. Or, cut the leads off the original device before attempting to remove it.

You can just let the unsoldered leads temporarily hold the part in place during some comparisons. After the best part is found, you can solder it in. This procedure is part of my "kit blueprinting mods" I often use to eek out that last bit of performance. :-)

Although the MRF904 has a little more gain than the 2N5179 (16db vs 15 db), it does not affect the Q7-Q8 shunt-feedback amplifier gain, since this is set by the ratio of R36/R34, or about 25. But, the MRF904 is a much quieter device, as indicated by its location at the top of the Low-Noise Transistor chart in the ARRL Handbook; on page 24-18 in my 1998 copy. So it *is* a superior device, and direct comparisons by me bear that fact out in operation. If someone has the test equipment to derive the actual MDS figures, then the improvement can be verified and substantiated. My ears were good enough for me. ;-)

Other devices were also tried for Q8, with little improvement in NF, as it is less a factor in the weak signal levels coming from the crystal filter than Q7 is.

That's all for now. More tips will be made as we go, so stay tuned. ;-)

72,

Gary Surrency AB7MY QRP-L #571 Chandler, AZ (near Phoenix)