K8IQY's "2N2/40" Forty Meter CW Transceiver

Introduction

The beginning of the 2N2/40 came about when Wayne Burdick, N6KR, proposed his "post apocalyptic" design contest for Dayton 1998. The premise of the contest was that "no matter what happens to us or the planet, you'll still be able to find them (2N2222's) in huge quantities." It is as Wayne termed it, "The cockroach of the transistor world." The contest challenge was to design a system, capable of transmitting and receiving, using only 2N2222 transistors as the active device, and using no more than 22 of them. A corollary to the constraints was that only other "cockroach parts" could be used, such as 1N914 diodes, but not three terminal regulators, PNP transistors, ICs, or anything of that ilk!

My interest in accepting this challenge was to design and build a transceiver from the ground up, something that I had always wanted to do, but never set aside the time to accomplish. Here was the "golden opportunity", if one ever existed. Along with this, I had been experimenting with MicroSim's PSPICE off and on for several months, but didn't feel that I had a very good understanding of its capabilities, nor how to use it very well. Combining the two elements, design and computer circuit simulation, would provide a valuable learning experience, provide a robust design assuming I was successful, and maximize the "fun" that could be derived from such a project. With those thoughts in mind, it was time to get serious and design and build a rig.

Design Criteria

At the onset, I struggled with some basic questions: "What will I design and build?" "Should it be on CW, and if so, what band?" "Maybe it ought to be something for SSB, then I could use it for bicycle mobile, if it works well." However, reality set in, and the recognition that I had never done this before, so keeping it relatively simple was most appropriate. Forty meters seemed like a good band for a CW rig, as I didn't have a QRP rig for that band, and kept missing out on the Fox Hunts. Sure, I would listen to the gang on my FT990, but the desire to put it on the air wasn't there; it just didn't fit my notion of what that contest is all about.

Another self imposed constraint was to not just lift circuits or designs from various books, but attempt to design the rig from the knowledge gained over many years of reading, listening, experimenting, observing, and operating. That sounds like re-inventing the wheel a bit, and to some degree it is, but it also frees the mind to consider other approaches. Some of that thinking shows up in the rig, and I'll discuss it later in detail.

So 40 meters got the nod for the band, and CW was the mode selected, mostly because of ease of design and construction. A number of the other issues that one would consider resolving at the beginning of a project were left, because I didn't know how difficult it might be to implement various sections of the rig, and how many of the 22 transistors would get used in each section. However, there were some desired basic features in the rig that did not impact the transistor count. These included an r.f. gain control, and a variable bandwidth filter. Features that did depend on transistor count included having an r.f. amplifier to make up for input filter losses, power output of 1-2 watts, and the ability to drive a speaker. All of these were "desirable", but would be reconsidered if all the transistors got used up elsewhere.

Another criterion was to build the rig as small as practical, so that it could go into a QRP-size cabinet. However, at the onset, I had no idea how large or small that might be. The building approach would be to start out with a fairly large piece of single-sided PC board material and begin building in the middle of it. As the design and construction progressed, the circuitry would be built outward, toward the edges. If all of the room wasn't needed when the rig was finished, then a quick trip to the band saw would remove the unused and unneeded board material.

The anticipated approach to use was to design a particular section, build the computer model for that design, optimize the computer model, build and test the section, and finally, compare the test results with those predicted by the modeling. This approach had been used once before on a two band SSB rig that was started for bicycle mobile use, but never finished. For much of that rig, I didn't know enough about modeling to build the models of the ICs that were being used. However, the approach had been used for the antenna T/R switch and the receiver input filter, and it was amazing how closely the actual circuit matched the frequency response predicted by the modeling. What wasn't known was whether this approach would work for a complete rig, and how many design iterations would be required before getting it to work satisfactorily.

As it turned out, the approach worked very well. There were a few iterations on one or two circuits, but for the most part, circuits were built with the component values shown by the modeling to be optimal. The total time to do the design and construction was considerable. I started around the middle of November 1997, a short time after the contest was announced and some 2N2222 transistors had been acquired. It wasn't until nearly the end of April 1998 that the job was done. A good portion of that time was spent doing the actual construction. My building speed could best be described "slow and precise", spending far longer on thinking about and visualizing how a section should be done than most builders would. However, the finished product is quite nice; almost "art like" in its appearance. The first part of May 1998 was spent feverishly getting the rig into a case and assembling the documentation package so that I could take it to Dayton.

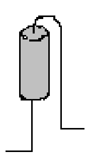
Construction Background

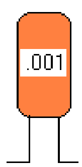
Over the past five or so years, many small projects have been built using a method that I was told years ago is called "Manhattan Style Construction". It's also called "Paddyboard" in some circles. The "Manhattan" name I believe comes from the fact that the little pads and parts that are used look a lot like a city in miniature, when they're all on the substrate. The substrate is usually a piece of single sided PC board material, copper side up. The pads are glued to the surface of the substrate, also copper side up, and become the junction points for the circuitry that requires support. The copper ground plane is available for soldering all component leads that go to ground.

My favorite method of making the small pads is through the use of an ADEL nibbling tool. This is a tool designed for cutting thin sheet metal, but can handle 1/16 inch thick PC board material quite well. The resulting pads that are produced by this tool are about 3/32 inch wide by 1/4 inch in length. While that might seem a bit small to most, there is ample room on one pad to solder the ends of 3 or 4 components. I use common Cyanoacrylate adhesives, (DURO Super Glue) although any of the available, thin adhesives of this type should work. My method for attaching a pad is to hold it in place and put a very small drop against the bottom edge of the pad and the substrate. The thin glue will wick under the pad and attach it in about 5 seconds. Others have tried this method and have had problems. The secret is to make sure the surface of the substrate is very clean, and devoid of any sensitizing films, grease, or other contaminants. I scrub my board material with soap and water and a piece of 3M Scotch Bright pad until the copper is shiny. It is then wiped with lacquer thinner to remove any remaining contaminants. When this method is successful, removing a glued down pad requires twisting it off with a pair of pliers. Each pad is also wiped across a piece of 400 grit wet and dry sandpaper several times on the copper side before gluing to the substrate. This cleans it and makes soldering to it very easy.

Mounting the various parts to the pads is mostly what could be called a "common sense" approach. I let the "geometry of the environment" guide how a part will be mounted, since there is no standard or accepted way of soldering a part into the circuit. For example, most of the resistors I mount vertically, for two reasons. First, I think they take up less space that way, allowing one to build more compactly. Second, the higher end also makes a convenient test point; that's why a small loop is put in the higher end lead. It is nice to install resistors with the color codes running from the higher end to the lower end, for easier readability, just in case a mistake is made in building. Occasionally, resistors should be mounted horizontally, either to better span the distances involved, or maybe fit under another component. That's done in several places in the 2N2/40, especially in the audio amplifier section.

There is not a lot to discuss regarding capacitor mounting, since the predominant type used has radial leads. When capacitors are mounted, I try to orient them so that their value can still be read. With all components, one lead needs to be approximately 1/16 inch longer than the other if it is going to be soldered vertically to the substrate. This difference is due to the pad height where the other lead is connected. For leads that are soldered to the substrate, I bend the lead at a right angle at the appropriate length, and leave about 3/16 inch of length for soldering. Leads that will attach to a pad are bent at a right angle also, and have about 3/32 inch of length for soldering. Generally, the leads on transistors and diodes should be bent about 1/4 to 3/8 inch away from the body. One additional comment is that component leads are prepared one at a time, as the component is being added to the circuit. Here are some examples of components that are suitably prepared.





That's enough background. Let's get on with building our 2N2/40 rig.

2N2/40 Specifications

Receiver

Narrow front end: 150 KHz bandwidth
Sensitivity: -122 dBm MDS (~0.2 uvolt)
Diode DBM first mixer
Very low noise
VFO: 100 KHz band coverage
       200 Hz maximum drift
       Linear varicap diode tuning
3 pole VBW crystal filter: 300-700 Hz
Push-pull audio output for speaker operation

Transmitter

1.5 watts output: three parallel 2N2222A
Excellent r.f. stability
QSK keying

Overall

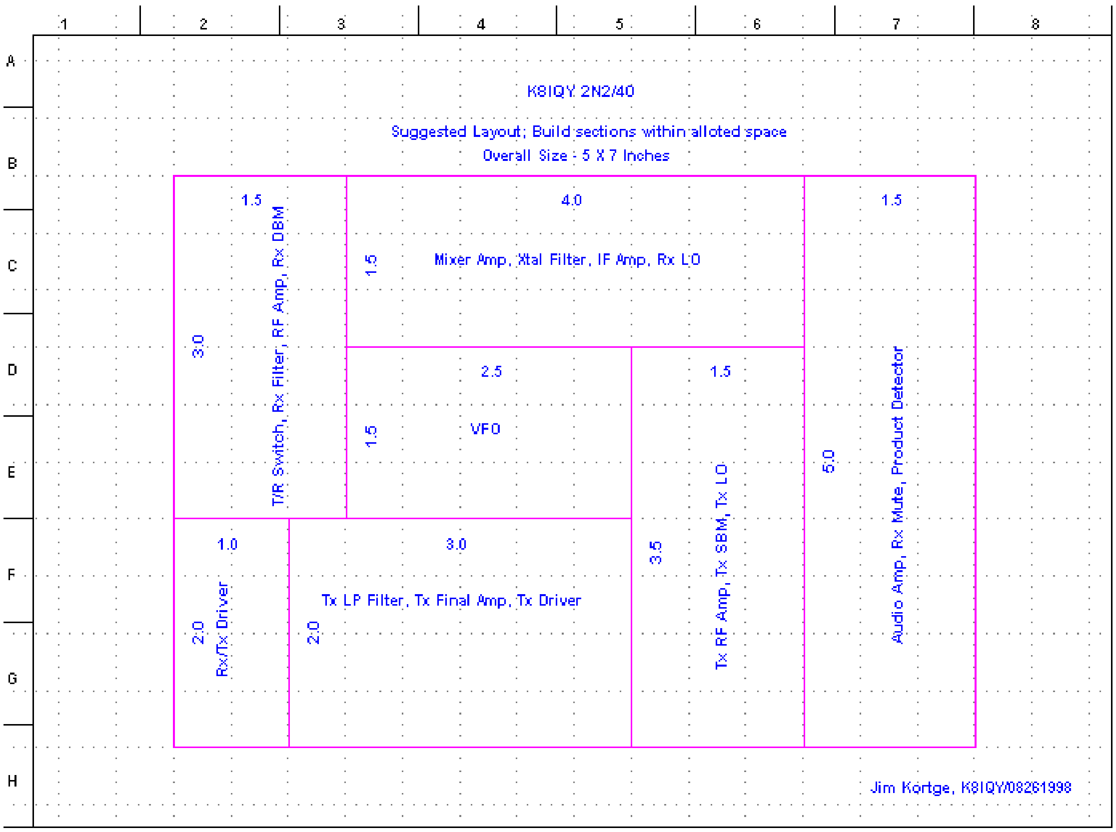
22 - 2N2222 transistors; four are 2N2222A metal case types
All circuits modeled with MicroSim DesignLab or Electronics Workbench
Manhattan style construction
4 inch by 5 inch footprint size

Physical Layout

Let's talk a bit about the layout used in the 2N2/40 and some of the thoughts and ideas that drove the design. We'll start with the VFO, since it's shared by both the receiver and transmitter. Then we will do the receiver, starting at the input, and ending at the audio amplifier. And finally, we'll tackle the transmitter. By the way, that's exactly the order in which the prototype rig was designed and built!

As each section is highlighted, I'll also try to impart some insight into the various layouts that I've prepared, as an aid in reproducing the rig. While these layouts aren't exactly as the prototype rig was built, they are quite close. However, the size was opened up to 5 inches by 7 inches so that there is significantly more room, hopefully making it easier for the first time builder to construct.

An overall layout for the 2N2/40 is shown below. It closely follows how the prototype was built. Building started with the VFO, located near the center of the substrate, but somewhat closer to the left edge than the right. Although VFO construction started with the circuitry around Q2, I kept reminding myself that the main output would be used by the receiver DBM, which was left and upward of the starting point.



The original receiver T/R switch, input filter, r.f. amplifier, and DBM were built along the left edge of the board, toward the VFO output transformer. When I got the DBM finished and mounted, (more on that later), it seemed like a good place to "turn the corner", so that was done. Before continuing on though, some testing of the existing circuitry seemed appropriate. Using my FT990 tuned to 4.915 MHz, the 2N2/40's intended i.f. frequency, I connected a test lead from the 990's antenna connector to the output of the receiver DBM. I connected a short antenna to the 2N2/40 input filter and powered it up. I could hear 40 meter signals, and they tuned with the VFO pot! Eureka.....it was working.

Having made a right turn on the substrate, building again started with the mixer amplifier. Next came the crystal filter, i.f. amplifier, and the local oscillator for the product detector across the top edge of the board. This brought construction to a point near the right edge of the substrate. It was necessary to make another right turn. That allowed construction of the product detector, mute switch, and audio amplifier along the right edge, going from top to bottom.

At this point, the receiver portion was essentially done, and I spent about a week just listening to it, and marveling that it actually worked. In fact, it worked very well, far better than I had expected for just 2N2222 transistors. Spurred on by the success of the receiver, I was anxious to see how the transmitter might fare.

Looking over the remaining unpopulated areas of the substrate, it was pretty clear that the transmitter would need to be placed adjacent to the receiver mute switch and the audio amplifier.

That also placed the transmit single balanced mixer reasonably close to the VFO, so that getting drive for it would be easy. I built the transmitter local oscillator, the single balanced mixer, and finally the cascode r.f. amplifier. At this point, there was no more room in the direction of the bottom of the substrate. Time to turn yet another corner, and start building toward the left edge of the substrate.

However, I started to get a bit uneasy. The buildup of the transmitter r.f. driver, the final amplifier output section, and the low pass filter, was yet to be completed, and not much space was left. It was then that the decision was made to go to a “second story” if the rig was all to fit within the substrate footprint. The details at that point were unclear, but it was certain the final amplifier(s) were not going on the main board, but somewhere else. After building the r.f. driver section, as expected, most of the space was used up. That which remained, I surmised, would be used to build the T/R driver switching circuitry, shown on the layout in the lower left corner.

On the prototype rig, the three 2N2222A transistors used in the final amplifier are located on a 1 1/4 by 2 1/2 inch piece of double sided PC board material. The transistors are on the top side, and the low pass filter components are on the bottom side. The whole affair is mounted on two 1 inch standoffs. Details of that construction are also shown in the Summer 1998 QRPp. As Paul Harden, NA5N pointed out, building the output final amplifier and low pass filter as a separate structure allows it to be replaced easily with another unit, maybe using a different bipolar or a MOSFET. The layout I’ve suggested leaves ample room to put the three 2N2222A transistors, and the low pass filter on the substrate, without building upward! However, you could build the driver, PA, and LP filter on a separate 2 inch by 3 inch piece of material, just so you have the option of replacing it some time in the future with a different set of components.

That completes the rig in terms of overall physical layout. Let’s now move on to the various circuits and circuit sections and discuss them in more detail. Before leaving this diagram, one additional comment is appropriate. As a building aid on a 5 X 7 inch substrate, use a ruler and black marking pen with permanent ink and layout the lines as shown. When you build, don’t put any component closer to a line than 1/8 inch. This will leave 1/4 inch “gutters” between each section which can be used for routing power and various signal lines. That approach worked really well on the second rig I built using the documentation from this article.

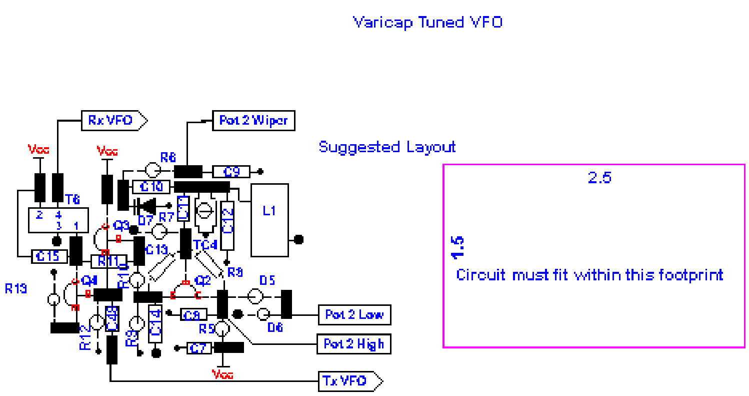
<u>Circuit and Construction Details</u>

Varicap tuned VFO **-** The VFO is a classic Colpitts design, using a MVAM109 voltage variable capacitance diode for the tuning. It tunes from nominally 2.085 to 2.185 MHz, 100 KHz of band coverage. Trimmer TC4 is used to set the lower frequency limit, and TC3, if included, sets the upper frequency. Some adjustment of the turns on inductor L1 may be required to get the correct frequency range, with the values shown.

Important features include the main inductor, L1, which is wound on a T50-7 powered iron core. With the 44 turns required on this core, the inductance should come in around 8.5uH. Type 7 cores have the best temperature characteristics of any of the commonly available cores. What frequency drift occurs is downward in frequency as the temperature increases. To compensate for this drift, a negative temperature coefficient capacitor is employed, C12a, which moves the frequency upward with increasing temperature. Polystyrene capacitor C12a, in conjunction with C12b (NPO type) provides the correct amount of compensation to keep the frequency stable with changing temperatures. The combination of Zener diode D5 and power diode D6 provide a total of 6.9 volts from the 13.8 volt supply. Keeping the collector voltage low (~7 volts) on transistor Q2 and its components reduces heat dissipation, also helping VFO stability. All of the capacitors in the VFO are NPO types except for C12a and C13 and C14. C13 and C14 are 5% tolerance polyester capacitors, which are quite stable with temperature. VFO drift from a cold start is under 200 Hz. The VFO tuning linearity is improved by swinging the varicap diode, D7, between 6.9 and 0.7 volts, avoiding the most non-linear portion of the capacitance curve, which occurs near 0 volts. Resistor R63, also helps linearize the VFO tuning by effectively changing the linear potentiometer, POT2, into a non-linear unit which approximately matches the tuning diode capacitance versus voltage curve.

Transistor Q3 serves as a buffer, keeping load changes from affecting Q2, the oscillator. Output from the emitter of Q3 is used to drive the transmitter single balanced mixer. The VFO signal is further amplified by transistor Q4, to provide the +10 dBm drive level (0.7 volts rms) required by the receiver double balanced mixer. Signal output is transformer-coupled to the receiver mixer from the secondary winding of T6. The primary winding is 16 turns on a FT37-61 core for 14 to 15 uH of inductance, depending how it is wound. It is tuned by capacitor C15 to provide the required driving power, and to reduce harmonic content output.

Here is the layout for the VFO.



As you can see, the construction starts on the right, and flows to the left, where the main output, the secondary of T6, is positioned to be close to the receiver DBM. The output going to the transmitter single balanced mixer (Tx VFO) will be routed through shielded cable, to reduce radiation into surrounding circuitry. There is some latitude for constructing the VFO in terms of alternate part locations. It doesn’t have to be built as shown. Just try to keep the overall size within the footprint, so that you have enough room for the other sections.

The varicap diode, D7 is not show in pictorial format, but schematically. This part looks like a TO-92 transistor, but with only 2 leads. It will fit within the space provided, however. Here is how to identify the leads. Hold the device with the leads pointing down and the flat face toward you. The left lead is the anode and the right, the cathode.

Once built, this section can be tested quite easily if you have a general coverage receiver. The VFO needs to tune nominally from 2.085 MHz to 2.185 MHz. Attach a short length of wire to the pad marked "Rx VFO". Power it up by applying 12 to 13.8 vdc (use a fused supply) to all of the pads marked Vcc, and tune your receiver until you hear the VFO. It should be quite strong. If you have a counter or oscilloscope, they can also be connected to the same output pad for measurement and signal observation. The unloaded output from the secondary of T6 should be around 3 volts peak to peak.

Rx T/R Switch, Input Filter, RF Amplifier and Diode Double Balanced Mixer - These elements make up the complete front-end of the receiver.

The T/R switch configuration used in the 2N2/40 is not the first design that I actually built. My first design was more robust, but required too many transistors to generate the drive signals, and in the end, had to be scrapped. This design, I believe, comes from Roy Lewellan, W7EL and has been used by many others. While it works very well for such a simple design, it does have some limitations. Its main fault is that it doesn't handle strong signals well, which could lead to third order intercept problems. However, if your 2N2/40 isn't used at field day or in similar situations, where really strong signals are present, it does just fine.

As originally implemented, the circuit used a trimmer at TC9, a 12uH inductor at L8, and the series resonance of this pair could be tuned. However, the loaded Q of the circuit is quite low, about 4, which makes the tuning so broad that this capability is wasted. I'd recommend building this circuit with TC9 being a fixed 47pF capacitor, and L8 being a 10uH molded inductor. That saves a trimmer, and the performance is virtually unchanged. The provided layout is shown that way also.

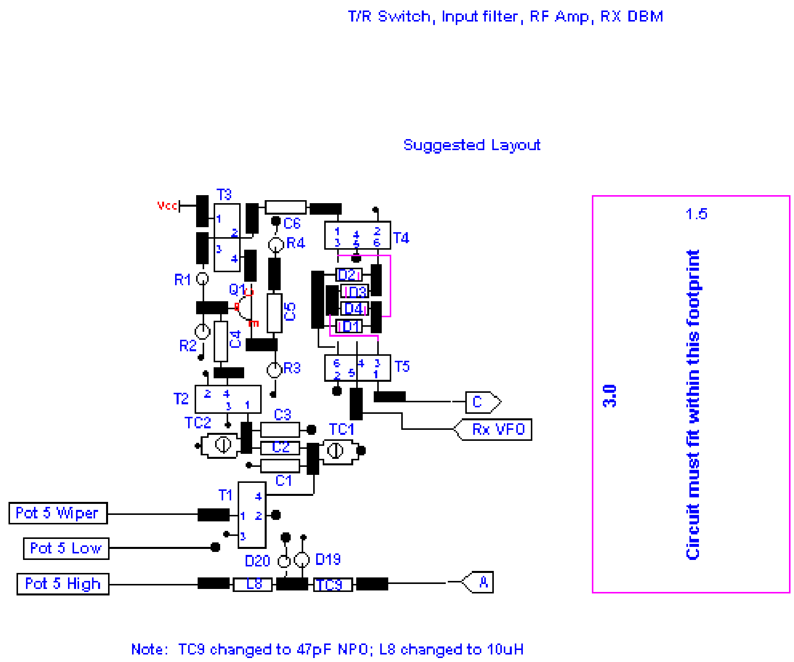
Once the receive signal passes through the T/R switch, it reaches the input filter. This filter is a classic double tuned band pass filter, using light coupling between the two resonators. Its half power bandwidth is about 150 KHz with the component values used. The secondary of input transformer T1 has an inductance of 3.6uH, and is resonated at 7.05 MHz with capacitor C1 and trimmer TC1. The three-turn primary provides a 50 ohm match to the antenna.

Output transformer T2 uses a 7-turn link, to match the 350 ohm input impedance of the r.f. amplifier. T2 is tuned to resonance with capacitor C3 and trimmer TC2. The coupling between the two resonators is 3pF. In the prototype rig, this capacitor is 3.9pF, but additional circuit tests suggests a 3pF value provides flatter frequency response over the desired 7.0 to 7.1 MHz region, without appreciable change in overall 3dB bandwidth. It should also make input tuning a bit less critical.

The r.f. amplifier is based on a common emitter design. With the component values used, it has 10 dB of power gain. Computer modeling shows that it can handle an input signal in excess of +13 dBm without distortion or going into gain compression. Another 10 dB of gain can be accomplished by paralleling emitter resistor R4 (82 ohms) with a 12 ohm resistor (R65). My 2N2/40 has a front panel switch for doing this, although I don't use the 20 dB gain position very often. Running the higher gain also reduces the input impedance to about 100 ohms, causing a mismatch with the input filter. However, it does let you to hear really weak signals, under the right conditions. A 4:1 impedance ratio bifilar transformer, T3, is used to couple the output to the receiver's diode DBM.

In the prototype rig, the receiver diode double balanced mixer was constructed on a very small (1/2 inch X 3/4 inch) piece of single sided PC board material, instead of on the main substrate. The reason for doing this was so that it could be easily replaced (after Dayton, of course) with a commercial DBM if my home-brew unit didn't work well. That DBM implementation was detailed in Paul Harden, NA5N's "QRP Hints and Kinks" section of the Summer 1998 issue of QRPp. I'm not sure that I would recommend building yours quite that small, unless you have lots of patience! The provided layout essentially uses the same geometry, but with more room between the various components. However, keeping the all the leads as short as possible helps maintain the balance necessary for the DBM to work well. Also, the 1N4148 diodes should be matched for forward resistance. Just measure a bunch, and pick 4 that are within a 1 ohm of each other. Since the diode leads are cut short, don't keep the soldering iron on these connections very long, or you will overheat the diode.

Shown below is the layout for the receiver input section.



This section should be built from the bottom to the top, and from right to left. Start with the input pad for the T/R switch (Port A) and end with the DBM on the top right. Once this portion is built, it too can be tested, using a general coverage receiver. I'm assuming that you have the VFO built and working correctly and its output is routed to the pad labeled VFO.

Solder a temporary jumper across the two pads labeled Pot 5 Wiper and Pot 5 High. Set the VFO to its mid-frequency, 2.135 MHz. Attach an antenna or 4-5 foot piece of wire to the pad labeled A.. Connect the output pad labeled "C" to the antenna input on your receiver with a piece of coax cable. Tune the receiver to 4.915 MHz. Using a signal generator, or a QRP rig fed into a dummy load, generate a signal on 7.050 MHz. You should find a signal very near 4.915 MHz, the 2N2/40 i.f. frequency. Once you find the signal, peak the input trimmers TC1 and TC3. Go back and forth between these two a couple of times until it is as strong as you can get it. If you built this section using a trimmer for TC9, peak that trimmer also. At this point, you should be able to leave the receiver tuned to 4.915 MHz, and tune the 40 meter CW band using the VFO. The general coverage receiver is acting as our crystal filter, i.f. amplifier, and detector. We'll build those next.

Mixer Amplifier, Variable Crystal Filter, I.F. Amplifier, and Rx Local Oscillator - These elements make up the next major section of the receiver. With the addition of the Product Detector in the next section, we have essentially a complete receiver, down to detected audio. We'll get to that point shortly.

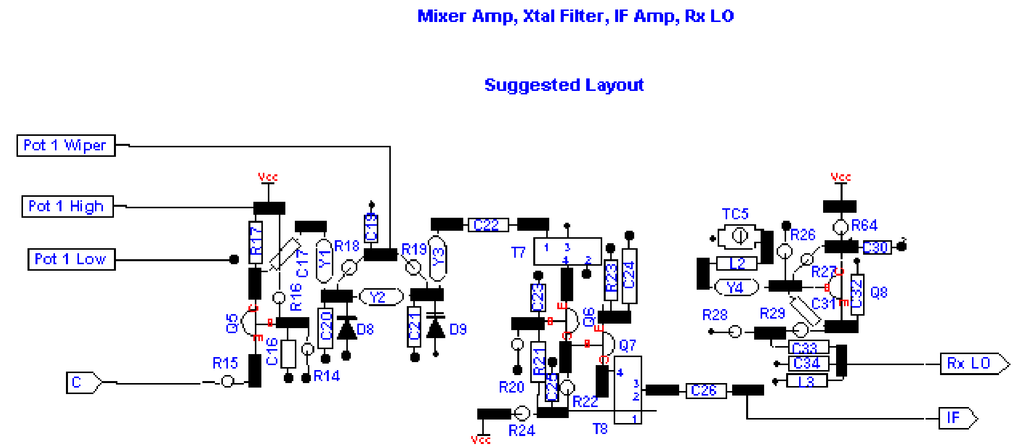
The diode DBM signal is fed to a common base amplifier. I've not seen this done before, and am not sure why. Since one of the goals in terminating a DBM is to have it working into a constant impedance load, this amplifier fits the bill nicely. As configured, the input impedance is constant at 50 ohms resistive, from DC to beyond 30 MHz. Using this circuit, we don't need a diplexer, nor do we need an attenuator, to keep the load on the mixer i.f. port constant. In addition, with the 270 ohm collector resistor, this amplifier produces about 6 dB of power gain. It also has reasonable output to input isolation, so that impedance changes from the downstream crystal filter don't reflect back to the mixer.

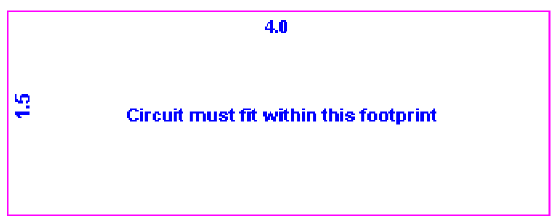
Following the mixer amplifier is a 3 pole, variable bandwidth (VBW), Cohn style crystal filter. It uses matched 4.915 MHz series mode crystals. The bandwidth can be changed from about 700 Hz, down to 300 Hz. Bandwidth control is accomplished using a pair of MV2115, voltage variable capacitance diodes. These diodes provide a capacitance change of about 100pF, as the voltage on them is varied from 0 to 13.8 vdc. Bandwidth control voltage is provided by a variable potentiometer, POT1. The 270 ohm source and terminating impedances for the filter are provided by collector resistor R17, on the input side, and the transformed input impedance of Q6, the first i.f. amplifier transistor, on the output side. The output impedance transformation is done with transformer T7, which has a 15-turn to 2-turn turns ratio, and approximately 56 to 1 impedance ratio. The 2-turn secondary should be wound on the "cold" (grounded) end of the primary.

Following the VBW crystal filter is the intermediate frequency amplifier, using another somewhat unconventional design. My first thought for the i.f. amplifier was to use a common cascode arrangement, i.e. a common emitter amplifier driving a common base amplifier. That configuration was modeled, and although it provided plenty of gain, it showed a wide variance in the input impedance with frequency. I felt the crystal filter ought to be working also into a constant load, just like the receiver DBM, for optimum performance. This led to building a new computer model which had the common base amplifier first, driving a common emitter amplifier. As with the common cascode configuration, the two transistors are direct coupled. Modeling once again showed this configuration could supply more than enough gain, but more importantly to me, the input impedance was constant over a very wide frequency range. However, the input impedance is only a few ohms, which resulted in the wide turns ratio transformer, T7, being required to couple the crystal filter into the amplifier. With the component values shown, the power gain of this stage is nearly 50dB, with a response peak at 4.9 MHz. Output is taken from a bifilar wound, 4:1 impedance ratio transformer, T8. The output impedance from this stage is 50 ohms, suitable for driving the product detector. This amplifier also shows no stability problems when properly terminated.

The last element of this section is the receiver local oscillator. It is a Colpitts configuration, just like the VFO. Perhaps the only unique feature of the circuit is tapping the output off the split emitter resistor pair, R29 and R28 and using a parallel tuned circuit to shape the waveform and reduce harmonics. I'm not sure going these extra steps has any payoff, but the output waveform is a clean sinewave. Something that may not be obvious about this oscillator setup is that its frequency is below the i.f. passband because this rig uses the Cohn filter as an upper sideband filter, instead of the more traditional, lower sideband arrangement. Lowering the crystal frequency is accomplished by adding inductive reactance from L2, in series with the crystal. Trimmer TC5 lets us adjust the amount of inductive reactance. For proper receiving, the frequency of this local oscillator is adjusted to be 750 Hz below the 4.915 MHz passband. When this signal is mixed with the i.f. signal in the product detector, our desired 750 Hz CW note is recovered.

Here is what the layout for this part of the receiver looks like.





I'd recommend building this portion of the rig by starting at the left, and building toward the right until you're done with the i.f. amplifier. Then I'd move over to the right edge, and build the local oscillator circuit, moving leftward. That should leave a small space between the output of the i.f. amplifier and the local oscillator. Probably the only critical thing here is keeping as much room between the i.f. input and output transformers, T7 and T8, as you can. This will help to insure stability. Placing them at right angles to each other also minimizes coupling, but is not required. Note that the symbols for the MV2115 varicaps are again schematic symbols, not pictorial symbols. There is ample space provided to fit them in.

If you would like to test the rig at this point, connect the output of the i.f. (port pad marked "IF") to the antenna input of your general coverage receiver, again using coax or a shielded scope probe. Tune the general coverage receiver to 4.915 MHz and power up the 2N2. This time, signals should be very strong, since we now have another 50 dB of gain from the i.f. amplifier. In fact, on very strong signals, you may have to reduce the r.f. gain of the communications receiver to keep from overloading it. You can also play around with the VBW crystal filter by grounding the pad marked "Pot 1 Wiper" or by taking this point to the Vcc supply. When this pad is grounded, the filter will be in its most "narrow" position, and when at Vcc, the filter will be running in its "wide" position. Of course, if you hook up POT1, you can set the filter to any passband width within its capability.

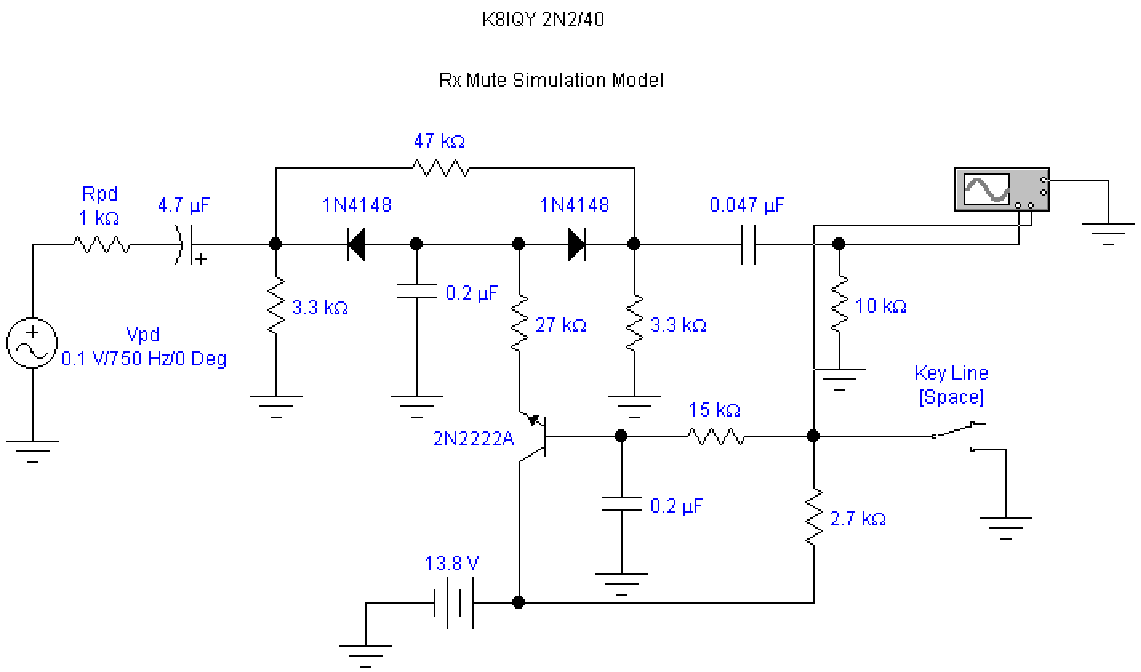
Product Detector, Receiver Mute, and Audio Amplifier - These three circuits complete the receiver portion of the 2N2/40.

The product detector is simply a two-diode, single balanced mixer. The i.f. signal is mixed with the receiver local oscillator producing principally two frequencies. The sum frequency is at 9.83 MHz and is shunted to ground by the reactances of C27 and C28. The difference frequency is 750 Hz, and adjustable by changing the frequency of the receiver local oscillator via trimmer TC5. This is the recovered audio that will drive the speaker after it is amplified.

The receiver mute circuit passes recovered audio to the audio amplifier when the rig is in receive mode. In this condition, diodes D12 and D14 are forward biased, and have very low forward resistance. The bias is provided by transistor Q9, which is turned "on" by the "Rx" signal applied to the base. When the rig transmits, drive is removed from Q9, causing the bias on D12 and D13 to be removed. These diodes now appear as open circuits, and audio is blocked. A small amount of sidetone audio is passed on to the audio amplifier during transmit by resistor R31. Sidetone audio is provided by having the receiver listen to the transmitter. Capacitors C35 and C36 provide delays to keep audio "thump" to a minimum during receive-to-transmit and transmit-to-receive transitions.

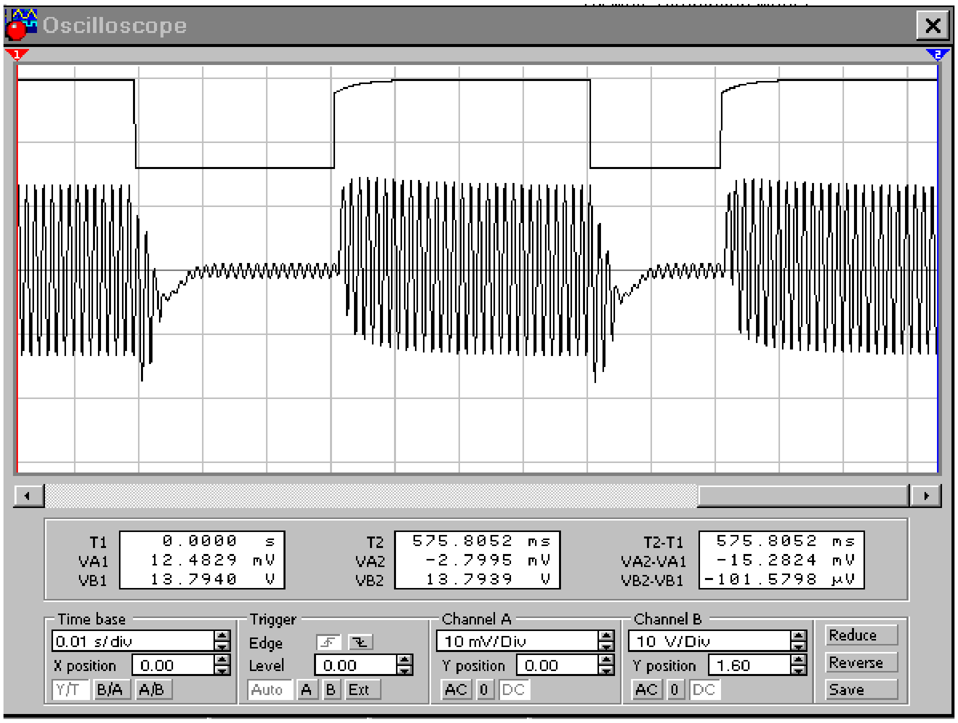
Before leaving this section, it seems appropriate to show an example of the analysis that can be done with today's computer modeling tools. In this case, the software product is the Personal Edition of Electronic Workbench. The company that produces this system offered it at reduced pricing this summer, so EWB was purchased. It works much like the MicroSim DesignLab (PSPICE) demo that I had been using, but is even more full-featured. When the educational Manhattan/Elmer 300 Project is done this winter over the internet, many more of these analyses will be discussed and shown. This one is just to whet your appetite!

Here is the computer model of the receiver mute circuit.



As you can see, this simulation circuit is a duplicate of the real thing shown in the 2N2/40 schematic. Product detector output, which is the signal source for this simulation, is the ac source labelled Vpd, set for 100 millivolts rms at 750 Hz. The product detector source impedance is the 1K resistor labelled Rpd. The small oscilloscope in the upper right hand of the model is a simulated scope within EWB, and can be expanded to show waveforms while the simulation is running. That will be shown next. The "key line" can be toggled while the simulation is running by tapping the space bar on the computer keyboard, so that the transitions from "receive mode" to "transmit mode" and back again can be studied. Actual CW could be simulated by hooking up the EWB digital word generator in place of the key line switch, but that's overkill for this simple circuit.

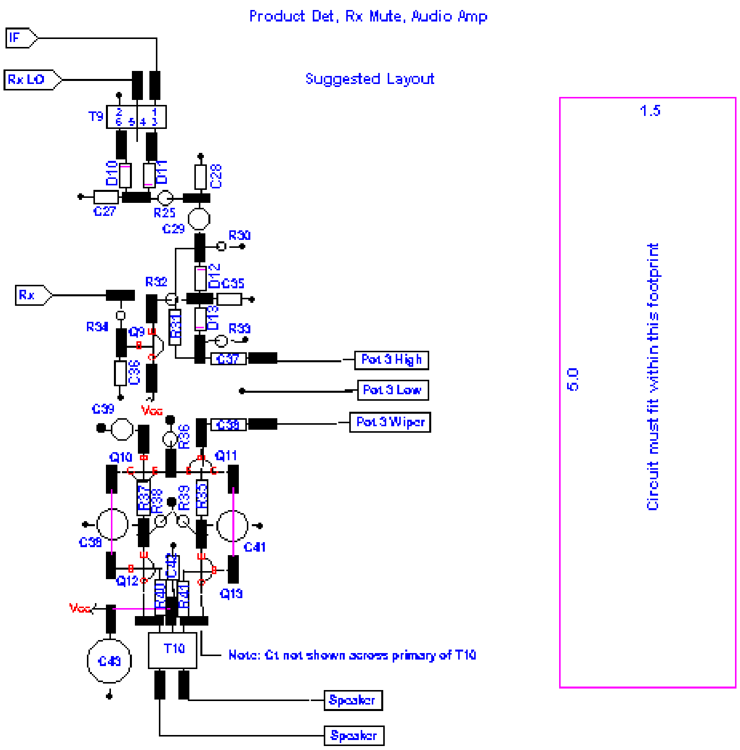
These are the resulting waveforms as the circuit is keyed.



The upper trace is the "key line" signal shown at 10 volts/division. The bottom trace is the output that is present on the wiper of the volume control potentiometer, POT3, at the full volume setting, shown at 10 millivolts/division. You can clearly see the effects of R31, the 47K resistor which "leaks" a bit of audio through during transmit so that we have keying sidetone. It should be apparent from this example what a marvelous tool modeling is, and the help it can be for checking out ideas, studying circuits, etc. without having to build anything physical.

Finally, getting back to the actual 2N2/40, a push-pull audio amplifier rounds out the receive chain. This amplifier is another circuit of my design. Incoming audio is applied to the base of transistor Q11. As can be seen, Q11 shares a common emitter resistor, R36, with Q10. Q10's base is grounded for audio purposes, so it becomes yet another common base amplifier, being driven in its emitter from the signal provided by Q11. The collector signals on Q11 and Q10 are of equal amplitude and 180 degrees out of phase with each other. These signals are then directly coupled to the bases of Q13 and Q12, where they are further amplified. Transformer T10 couples the push-pull signals from the collectors to the speaker. Note that the biasing of the input pair of transistors is derived from the emitters of the output pair. The large capacitors on the output pair emitters bypass all audio. Capacitor Ct is chosen to resonate the primary inductance of T10 at 750 Hz, thereby providing additional CW signal selectivity. In the prototype rig, this capacitor is 0.082uF. While I haven't measured the power output of this amplifier, it is more than enough to drive a speaker to uncomfortable levels at full volume. It also has very low internal noise.

And finally, the layout for the remainder of the receiver.



This layout is a bit more open than the others, since the size of T10 could vary depending on the part source. Circuitry should be built from the top down. Nothing is very critical here, since we're now dealing with audio, and it's a bit easier to control. When you get to the audio amplifier, note that the leads of base bias resistors R35 and R37 need to pass under their respective transistor bodies, Q10 and Q11. This is also true for resistors R40 and R41. I'd recommend that all of these resistors be soldered in first, before adding the other components. Transformer resonating capacitor Ct is not shown on this diagram for clarity. It needs to be soldered to the pads on either side of the T10 primary winding. Before testing. you should wire in POT3, so you have a volume control.

At this point, we have a complete receiver. The only test to perform at this juncture is to see if it works in its entirety. Connect an antenna to the Port A pad, and a speaker on the T10 secondary pads, and apply fused power. If you've done the building correctly, and have passed the previous tests, you should have a 2N2/40 receiver that is working, maybe even hearing signals, but as yet, not aligned. Here is how to do the alignment.

Receiver Alignment - Set trimmer potentiometer TC5 to its maximum capacity position by either listening to the receiver local oscillator on another receiver tuned to approximately 4.914 MHz, or by measuring the output frequency of the local oscillator by attaching a probe to the ungrounded end of L3 and adjusting TC5 for the lowest frequency obtainable. Tune the receiver to a signal, or if possible, generate a signal about mid-band, around 7.050 MHz. Tune across this signal by rotating tuning potentiometer POT2. Notice that as the tuning potentiometer is rotated clockwise, the audio pitch of the signal goes lower. Listen for a peak in audio response as the signal is tuned. We want this peak to occur at about 750 Hz. With TC5 set for maximum capacity, the peak is probably upwards of 1500 Hz. Slowly rotate TC5 to a lower capacity setting. Go a little at a time, and retune the receiver, listening for the audio peak. You will hear it progressively moving to a lower pitch. Repeat the adjustment of TC5 and retuning the receiver until the pitch is where you like to listen to CW, and is the loudest you can make it. That's it.....the receiver is aligned.

What we just did was place the receiver local oscillator frequency about 750 Hz below the center of the crystal filter passband. Remember, the crystal filter is being used as an upper sideband filter. To confirm that everything is working as it should, find a SSB station and see if you can tune in the audio so that it is intelligible. If you've done the alignment correctly, you should not be able to properly tune in the signal, since the station is operating on lower sideband, and the receiver is listening to the upper sideband.

Now let's build the transmitter portion.

Tx Local Oscillator, Single Balanced Mixer, and Cascode RF Amplifier - These elements make up the first section of the transmit strip.

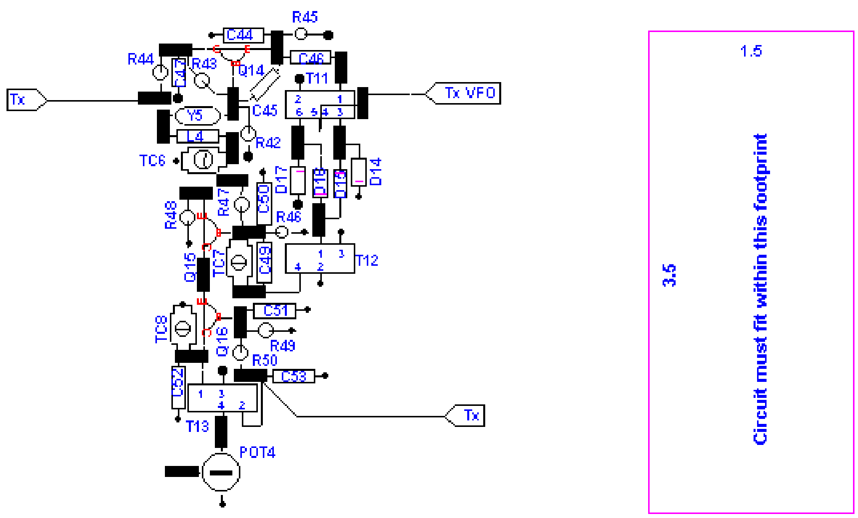
The transmitter begins with another crystal oscillator, Q14 and associated circuitry, used to generate a CW signal at 4.915 MHz. This circuit is virtually a duplicate of the receiver local oscillator. Output signal is taken from the emitter through a 47pF capacitor, C46. This signal is fed to a diode, single balanced mixer (SBM) along with a signal from the VFO, which is applied to the SBM through capacitor C48. The SBM consists of a trifilar wound transformer T11, along with four 1N4148 diodes, D14 through D17. As with the receiver DBM, the diodes should be matched for forward resistance. The sum of the Tx LO signal and VFO signal produce an output at 7 MHz that is used in the transmit strip. The difference frequency, along with the original mixer signals and higher order mixer products, are filtered out by the tuned input and output circuits of the next stage, a cascode RF amplifier.

The Tx cascode RF amplifier uses a conventional grounded emitter stage, Q15, direct coupled to a grounded base stage, Q16. Total power gain for this transistor pair is on the order of 40 dB. The input is a link-coupled, tuned circuit comprised of T12 and capacitors C49, C50, and trimmer TC7. A tuned output is employed using another link-coupled transformer, T13 and capacitor C52 and trimmer TC8. Output from this stage is taken from the 5-turn secondary link, and feeds the power control potentiometer, POT4. As a point of reference, the 26-turn windings on T12 and T13 should measure at 3.0uH. Maximum power output from this stage is about 10 milliwatts, or 0 dBm. However, with the 2N2/40 running at 1.5 watts output, this stage only needs to feed the driver with 0.2 milliwatts, or -6 dBm.

Here is the layout for this section of the transmitter.

Tx RF Amp, Tx SBM, Tx LO

Suggested Layout



This section should be built from the top down, and right to left. I'd recommend starting with the single balanced mixer and then follow that with the local oscillator. These two sections could almost be done together, since they are adjacent to each other, and tightly integrated.

The Tx local oscillator can be tested by itself by applying power to the Tx lead, and listening for a signal at 4.915 MHz on a general coverage communications receiver. You can also attach a counter probe to the mixer side of capacitor C46 to verify that the circuit is oscillating at the correct frequency.

When the local oscillator and single balanced mixer have been completed, build the cascode RF amplifier. Items that are critical here are the placement of the input and output transformers. They should be as far apart as you can get them, and ideally, at right angles to each other. They're not show in that orientation on the layout for clarity, and will also work fine as shown.

Once all of the elements are complete, they can be tested following this procedure. Connect a short test lead or your counter probe to the wiper pad of POT4 and adjust POT4 to maximum by turning the screwdriver adjustment to the full CW position. Apply power to the receiver in the normal manner. Apply power to the pads labeled Tx through a 1N4004 or equivalent diode, to simulate the approximate voltage level that will be present when the Rx/Tx switch is active. Listen for a signal at 7 MHz on a receiver. Verify that the signal changes frequency when the VFO is tuned. With the VFO set to a frequency of 2.135 MHz, its mid-frequency, adjust trimmers TC7 and TC8 for maximum signal. Go back and forth between these two a few times as there is some interaction. When you are done with this test, we are ready to build the other half of the transmit strip.

Tx RF Driver, Final Amplifier(s), and Output Low Pass Filter - These elements make up the second section of the transmit strip.

The Tx RF driver consists of Q17 and associated circuitry. Transistor Q17 is the first of the 2N2222A metal transistors used in the rig. A heat sink can be used on this transistor to manage the power being dissipated, but is not necessary. This stage is an untuned, class A amplifier, which produces the drive necessary for the final. Its output is via a 2-turn link on transformer T14. This transformer translates the higher collector load impedance down to the lower input impedance of the three parallel 2N2222A metal output transistors. Power output from this stage is about 10 milliwatts, or +10 dBm, with -6 dBm of drive from the previous stage.

The output is fed through capacitor C57 and on to the input circuitry of the final. This circuitry is a 1N4148 diode, D18, in parallel with R55, a 100 ohm resistor. Capacitor C57 charges minus to plus on the negative excursion of the drive signal. On positive going excursions, the voltage on C57 is added to the positive signal, thereby doubling the effective positive level, and providing more drive to the final transistors. This circuit is often referred to as a “dc restorer”.

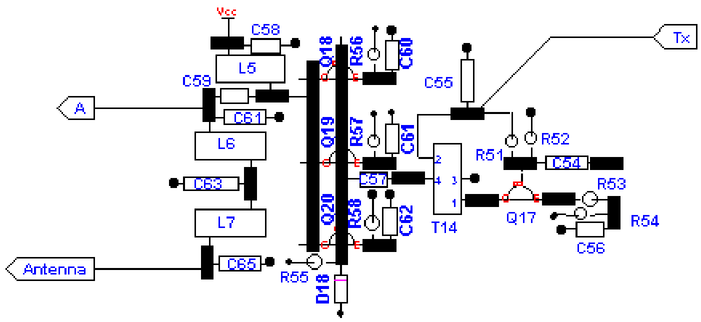
The final transistors, Q18, Q19, and Q20, are also 2N2222A metal case types and should be run with heat sinks. This amplifier stage runs in class C, and measured efficiency is around 70 percent. Each transistor uses a bypassed 2.2 ohm resistor in its emitter to help keep the collector currents balanced, without having to gain match the 3 devices. Three transistors were used in the final because that’s how many were left after building all of the rest of the rig. I had expected to get about 1 watt output from the three, and was pleasantly surprised to find that one can easily get 2 to 2.5 watts of output without excessive heating. I originally ran about 1.5 watts of output power without the heat sinks, but added them later, just to be on the safe side. The output impedance is about 50 ohms if the calculations are done with a Vcc of 12.5 volts and a power output of 1.5 watts, i.e. Vcc^2/2*Po.

That leads us to the output filter, which is a very standard, 5 pole, Chebyshev low pass filter taken out of table 11, page 2-44, of the 1988 ARRL handbook. It is filter number 86, using standard E24 capacitor values. The capacitors for this filter should be C61 and C65 at 430pF and C63 at 820pF. However, capacitor C61 is reduced to 360pF to compensate for the output capacitance of the 2N2222 finals, and the Rx T/R switch capacitance, that of TC9. These two sources add a total capacitance of 70pF in parallel with C61. All output capacitors should be silver mica units. The inductors, L6 and L7, should measure at 1.6uH. Mine have fewer turns than the formula predicts, but measure at that value, using an AADE L/C Meter IIB. By the way, if you don’t have one of these meters, get one! It is one of the best investments you will ever make in an inexpensive, very accurate, component test instrument.

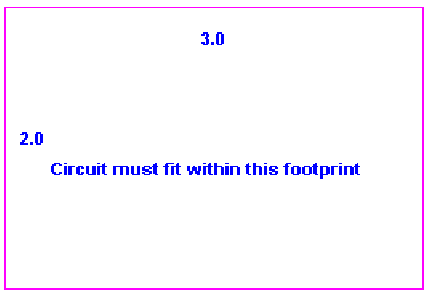
The layout for the driver, final amplifier(s) and LP filter is shown below.

Tx Driver, Tx Final Amp, Tx LP Filter

Suggested Layout



Note: Metal 2N2222s use different pinout



This circuitry will build the best from right to left, starting with the driver stage. There is nothing critical about the driver, just use common sense in parts placement. The final transistors also are not very critical, although having a degree of symmetry of the parts makes it look better. The bases and collector are connected together using long pads that can be cut from a scrap of PC board material using a fine blade in a hacksaw, or a jeweler's saw. A bit of cleanup with a mill file will make the edges smooth. If you don't plan on putting heat sinks on the final transistors (not recommended), they could be placed closer together than the layout shows.

The output filter is also not critical, except that you want to keep the inductors, L5 through L7, as far away from driver transformer, T14, as you can. Note that T14 also is oriented 90 degrees to these other inductors, just to minimize any coupling. On the prototype rig, inductors L5, L6, and L7 were lying flat on the substrate, and were spaced from the surface with small cardboard disks cut from a shirt box. The intent was to put nylon washers under them later, but that hasn't happened yet. I'll get to it sometime!

It would probably be better not to hook up the connection labeled "A" at this point until we have tested this stage, and finished the rig by building the T/R Switching circuitry, which is up next. That way, if something is amiss with the transmit section, such as it's parasitic, or oscillates all over the spectrum, we won't be putting that unwanted energy into the receiver. However, don't forget to do this at the end, or the receiver won't hear anything!

To test the completed transmitter, we need to do the following. Connect a suitable wattmeter and a 50 ohm dummy load to the pad labeled "Antenna" and a ground on the substrate near the "Antenna" pad. Make sure the dummy load will handle 2 to 3 watts. Connect a test lead from your power supply to the final amplifier pad for Vcc. Make sure this lead is fused with no more than a 1 amp slo-blow fuse. Take a second lead from your power supply and add a 1N4004 diode or equivalent <u>and</u> a 0.5 amp slo-blow fuse in series, with the diode polarity being anode to cathode, so that current can flow. Set POT4 to its lowest setting, i.e. wiper at ground (fully CCW). Apply power to the finals only. Verify there is full power supply voltage on the collectors of the final transistors. Apply the diode reduced voltage to the Tx leads. Verify that the local oscillator is running by listening for a signal at 4.915 MHz, or in the lower 100 KHz of the 40 meter band. Set the VFO to approximately the middle of its range. Slowly advance POT4 until you start to see some power output indicated on the wattmeter. Adjust trimmers TC7 and TC8 for maximum power output. Keep reducing POT4 to keep the output power below 0.5 watts while these adjustments are being made. Don't leave the Tx lead connected for more than about 1 minute, or driver Q17 will probably overheat. After you have things adjusted for maximum power, advance POT4 beyond the 0.5 watt setting, until the rig is running between 1 and 1.5 watts. Remove the Tx lead as soon as the power level is set. Keeping the rig on the air continuously at higher power levels is not a good idea! That's it. With any luck, it will be putting out a clean signal, which can be verified with another receiver.

Rx/Tx Driver - This section completes the construction of the 2N2/40. (I'll bet you never thought you would get this far.)

The Rx/Tx driver switching circuit provides receiver and transmitter control. Transistor Q21 is normally "on" due to base drive from bias resistor R59 and R60. The port labeled "Rx" supplies current to audio muting transistor, Q9, via resistor R59 also. Since the collector of Q21 is near ground potential, (0.2 volts actually) transistor Q22 is turned "off", and no current is flowing out of its emitter.

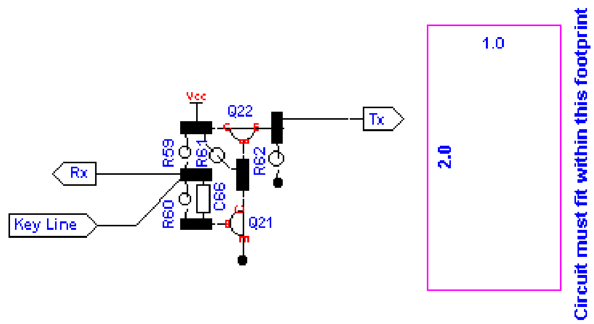
When the "key line" is brought to ground via a straight key or keyer, the two transistors revert to their opposite states. Transistor Q21 is turned "off", and the current that was flowing through its collector-emitter junction is now flowing into the base of Q22, turning it "on". The emitter of Q22 now provides current to all of the transmitter sections that require current from the port labeled "Tx". These include the Tx LO, Tx Cascode Amplifier, and the Tx RF Driver stages.

The voltage available at the Q22 emitter is the supply voltage, Vcc, minus its base-emitter forward drop, a value of approximately 0.7 volts, minus its collector-emitter saturation voltage, another 0.2 volts, or Vcc minus 0.9 volts. Had we been able to use a PNP transistor for Q22, the base-emitter drop would be eliminated, but then we wouldn't be using all 2N2222 transistors, as the contest rules required. Not having full supply voltage available to the transmit stages reduces their gain and output power by perhaps 10 percent, not enough to keep the design from working.

The layout for this very last 2N2/40 section is shown below. It is by far the easiest section in the whole rig to build.

Rx/Tx Driver

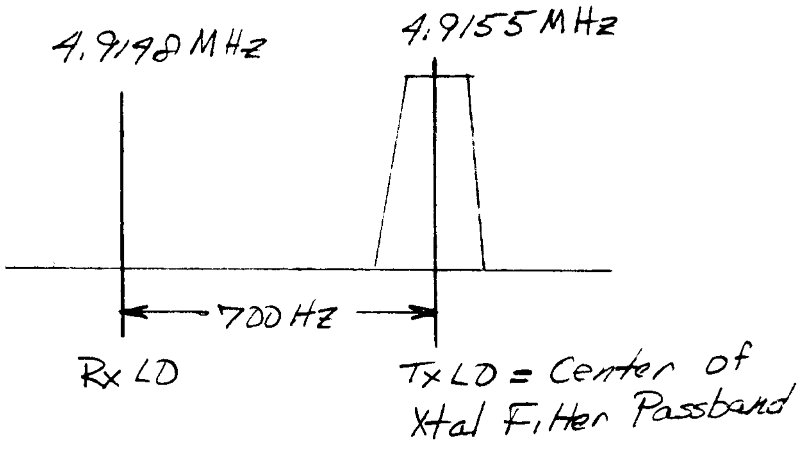
Suggested Layout



As you can see, this section is really simple-nothing critical in terms of layout, and lots of space to build it. Once it is completed and its port pads are wired to the corresponding port pads in the receiver and transmitter, your 2N2/40 is complete, except for wiring in the controls, jacks, and putting it in a case. Also, don't forget to connect the "A" pads together so the antenna signal can get to the receiver.

For testing, you can add the external components with leads long enough that they will be appropriate when the rig is assembled in a case, and run it on the bench. I did this with mine and had a blast making contacts with it sitting "nude" on the benchtop. To me, there is a certain fascination and beauty with having an operating rig on the bench, uncased, so that you can see all of the components, while watching the output power meter swing up as you key it in QSO. One can almost imagine all of the electrons moving here and there, doing their part to allow reception, or generating RF to be radiated. Had it not been for Dayton, and the need to package it, it might still be setting there! Before we move on, let's finish the alignment of the rig, by setting the transmitter to the frequency to which the receiver is listening. Here is how we do that.

Transmitter Alignment - Before aligning the transmitter, be sure the rig is connected to a dummy load, since it we will be keying it and generating r.f. Begin the transmitter alignment by setting the transmitter local oscillator trimmer, TC6, to the minimum capacity position. Use the same method we used with the receiver to determine where this is. With TC6 at minimum capacity, the transmitted signal will be far above the center of the crystal filter. Our goal is to slowly move the transmit frequency down until it is in the center of the receive passband. When that happens, we will hear it at the same pitch as we hear a properly tuned CW station. With the transmitter keyed, slowly turn trimmer TC6. As you do, you will eventually hear the transmit signal at a very high pitch as it enters the crystal filter passband from the higher frequency end. Keep turning until the pitch of the note heard is the same as a properly tuned CW station. When they are nominally the same, stop adjusting TC6. Don't keep the transmitter on the air continuously for longer than 20 seconds while making this adjustment. If you have a keyer, let it send a series of dits while doing this alignment. Once you're on the air with the rig, you can trim up the setting of TC6 so that the receiver and transmitter are "dead on". The station you are listening to and your transmitted signal will be heard at the exact same pitch. Below is a diagram of a properly aligned rig, showing example Rx and Tx local oscillator frequencies and their relationship to each other and the crystal filter passband.



Miscellany - This section will cover everything else that didn't fit under one of the previous categories. There are some general information items that are worth mentioning, which might make building a 2N2/40 a bit more successful.

First, let's discuss the wiring that is used to connect the various pads together. I used 22 gauge stranded, Teflon coated hookup wire for this task. The Teflon wire is really nice because you can't melt the coating with normal soldering iron temperatures. This makes for better looking wiring, I think. Routing the wires between pads is more of following logic than anything else. I tried to add the "Vcc" wiring as the sections were built, so that it could be routed against the surface of the substrate. In some cases, it passes underneath the leg of a resistor or capacitor as an aid in keeping it tight to the surface. The "Tx" and "signal" wiring was done in a similar manner. If you look at the overall layout of the sections, it is apparent that there really are not very many wiring runs required.

The only two places that shielded signal wire was used were from the VFO to the Tx SBM and from the i.f. amplifier to product detector transformer T9. In these cases, a short length of RG-174 was prepared, with the shield grounded only at the VFO or i.f. amplifier end. The shield at the mixer and product detector ends were cut off, and the outside jacket pulled over the shield so that it could not make contact with the substrate surface. Doing the installation this way prevents having the shield grounded in two locations, which could cause a ground loop to exist.

After all of the substrate wiring was done, it was secured at a number of locations with old fashioned, waxed nylon lacing cable to keep the wires together. This is one of those cosmetic items, and has no effect at all on the performance of the rig.
The lead from the "Antenna" pad to the coaxial connector was so short in the prototype rig that I didn't bother with coax. A pair of 22 gauge wires was twisted together and used to make the connection. However, the lead serving as the "shield" was terminated on the sub-board holding the PA transistors and output LP filter, and the BNC connector ground terminal. The total length of this run is about 1.5 inches.

Wiring to the power connector was also done with leads twisted together. Wiring to the potentiometers was done with flat, multiconductor cable, my favorite way to connect controls. If you haven't used this method, I encourage you to try it. You'll never go back to your old ways. You can buy 2 feet of 64 conductor, multicolored cable from many of the electronics suppliers for a couple of bucks, and that's enough wire for many, many rigs.

The Zener diode, D21, is just wired between the fuse holder "hot" lead and its ground terminal. Switch S1 is part of the audio control potentiometer, POT3. By all means, make sure that you put in the fuse and the Zener diode. They will prevent smoking the rig if the power is hooked up backward, or if a power supply with an output higher than the Zener voltage (15 volts) is connected.

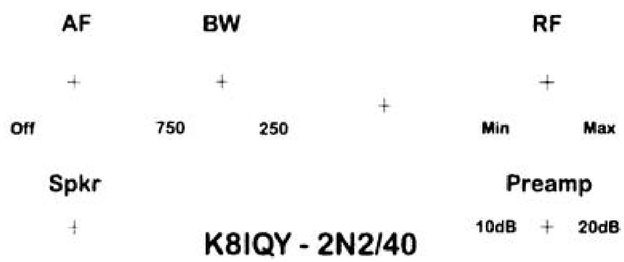
As mentioned earlier, I made my pads with an ADEL nibbling tool. Round pads on the order of 3/16 inch diameter will work as well, and have slightly more surface area than the ADEL produced pads. The second rig that I built to verify the accuracy of the material in this article in fact uses a mixture of the two pads. The transmitter was built with ADEL pads, and the receiver built with 3/16 inch diameter pads, to verify these pads would also work.

A good soldering iron is a must. Mine is a 25-watt, temperature controlled unit, with a 1/8 inch chisel point. For normal soldering, I keep the tip temperature at 320 degrees C. When I am soldering a lead to the substrate, I raise the temperature to 350 degrees C, which gives a bit more heat capacity to melt and flow the solder. Two of the musts in doing this type of construction are to have some desoldering braid on hand and some pure rosin liquid flux. The soldering braid is used to wick up extra solder on a connection, or remove all of it, if a component gets installed incorrectly. While all electronic grade solders contain flux, most have the minimum amount needed to properly "wet" a joint. A tiny amount of liquid flux can be brushed on a connection that appears to look overheated or "dry", and reheated to produce a superior solder joint.

When winding the various non-bifilar/trifilar transformers, put the windings on as close together as you can get them. Make sure that each turn is tight against its neighbor on the inside, and spaced on the outside. This method is usually frowned upon, but I find the results to be much more consistent, toroid to toroid, and builder to builder. Most people wind toroids without the use of any tools. The job is much easier if you use a small crochet hook. For T37 size toroids, an anodized aluminum #2 size, or its steel equivalent, is ideal. Each turn is wound by reaching through the center of the core with the crochet hook, and pulling the wire through the hole as the hook is retracted. Winding the core this way keeps the windings tight to the surface, and is much faster. Use 26 or 28 gauge wire for all of the conventional transformers, but make sure there is a 20 to 30 degree gap remaining after all the windings are on a core. Transformer T14, and inductors L5 through L7 can be wound with a heavier gauge; number 24 is a good choice. All of the bifilar and trifilar transformers were wound using about 300 degrees of the core circumference. I use three strands of number 28 gauge, and twist them together at 6 to 8 turns per linear inch with an electric drill motor. Make a long piece (3 feet) of 3 strand, and another shorter piece (2 feet) of 2 strand, and then cut suitable lengths for winding each transformer. I'm not going to provide a tutorial on winding bifilar and trifilar transformers. If you don't know how this is done, you should look it up in any good electronics reference book.

No rig is complete until it is installed in a case. For the 2N2/40 prototype, "home" became the inside of a TenTec TP-42 aluminum case. The case was painted flat black on all surfaces except the front and rear panels. These were painted light gray. A front panel overlay was drawn using COREL Draw. This overlay image was then reversed, and printed using a laser printer on a piece of transparency film. After trimming to size, the overlay was attached to the front panel with the printing on the inside, to keep it from being rubbed off. The control nuts from the various front panel controls hold it in place. If you build a 2N2/40 from this article, you will need find a larger case, as the TP-42 won't accommodate a 5X7 inch board. However, it appears a TenTec TP-46 or TP-47 case is the correct size case for the larger footprint. A custom case could also be built. The front panel layout shown won't work directly either, but you can use it for general control placement.

Here's what the front panel of the prototype rig looks like.



One of the things that probably isn't obvious, although it is mentioned elsewhere in the article, is that the rig's On-Off switch is incorporated in the Audio Gain potentiometer. This was a convenient place to add it, and saved finding another place on the already crowded front panel for a power switch. Another item that was mentioned earlier was the ability to switch the RF amplifier to a higher gain. That switch is the one the lower right hand corner labeled Preamp.

<u>Acknowledgments</u> - here is my chance to publicly thank all of the people that have helped along the way. There have been many, and they have helped immensely. First off is one of my heroes, Wayne Burdick, N6KR. It was Wayne who had the insight to propose the contest, and was the driving force behind keeping it pure. Many thanks to HB Electronics, and their generosity in supplying ready-made 2N2222 semiconductor packages, at absolutely bargain basement prices. Their components are the heart of my original 2N2/40, and the second unit built for this article. Then comes Steve "Melt Solder" Weber, KD1JV. It was Steve who led the 2N2222 design charge, building the first working rig, and demonstrating to all of us that a viable design could be accomplished. Next comes Doug Hendricks, KI6DS, and his silent twin, Jim Cates, WA6GER. Without this "dynamic duo" there would be no NorCal, nor 2N2222 building contests, nor a long, admirable, legacy of great club kits. We are all indebted to these two fine gentlemen. Next on my heroes list is Paul Harden, NA5N. Paul was most helpful in putting the prototype through his "top 10 tests", to verify that what I had observed qualitatively, was indeed based on quantitative performance measures. He alone is also responsible for all of the wonderful illustrations and pictures that are now a part of the 2N2/40 story. Without his help, this article

would not have happened in the format you see it. And the last three that I'd like to thank are tall, skinny, Chuck Adams, K5FO, from big D, Preston Douglas, WJ2V, from the "big Apple", and another of my heroes, "Professor" Glen Leinweber, VE3DNL. It was Chuck (along with Doug H.) who provided much of the "push" to get the 2N2/40 project "out to the public", almost to the point of making me "go ballastic" with his timeline. He is a "let's do it" person. Preston Douglas' building a working 2N2/40 from an inaccurate set of schematics is almost an unbelievable feat. I had sent K5FO (and NA5N) an early set of schematics after Dayton, so he could see the general layout of the rig. While Preston was visiting Chuck, he also got a set. On his flight back to New York from Dallas, Preston decided that he was going to build the rig. To his credit, he did ask my permission and concurrence to proceed. Having him do that served two purposes: it gave me more confidence that the rig was reproducible, by hams who do not "bend wires or chase electrons" for a living; and we found one major and several minor, but none-the-less important, errors in the documentation. Kudos to Preston for his work. All but two of our communications took place via the internet, for those interested. And finally, Glen, VE3DNL for taking the time out of his busy fall schedule at the university to look over an early schematic and layout package. His observations always inspire me to "look out of the box" for unique solutions and better designs.

The very last person who needs to be thanked is here, all alone, because of her importance to me. That is my dear wife, soul mate, and lifelong friend, Kathy, KB8IMP. She has put up with untold hundreds of hours of my being squirreled away in the ham shack, working on the rig's design, building them, operating them, and putting the documentation together. Through it all she has been most patient, understanding, and supportive. Among her many areas of expertise is written English, and she contributed to this project by being my primary proof reader, and resident advocate for "getting the project done".

Summary - I indeed hope that this article has inspired you to get the parts and dive into scratch building a 2N2/40, or another design that you have fancied, but never built, using "Manhattan Style" construction methods. It is really a whole lot easier than it might appear at first blush. From personal experience, I can tell you that nothing you will ever build as a kit will compare to the satisfaction realized with doing a complete rig from scratch. If the dimensions of the layouts I provided look still look too small, build it on an even larger platform, maybe 6 X 8 inches, or even 7 X 9 inches. Or you could split it up into sections and build it that way, with the receiver on one board, the transmitter on another, and the VFO and Tx/Rx driver on a third, all appropriately wired together. I think the design is forgiving to the extent that it will work in almost any reasonable configuration. What I'm encouraging you to do is learn to build from scratch. It's fun, and a wonderful way to satisfy those creative desires. Happy building!

Voltage chart - Shown are typical voltages for all of the transistors in the 2N2/40. These measurements were taken with a typical 3 1/2 digit digital multimeter. The supply voltage (Vcc) was 13.8 volts dc. In some cases, it will be evident that the emitter voltage is higher than the base voltage, a condition that should not occur. These readings are the result of r.f. being present during the measurement. The most obvious examples include Q8, the receiver local oscillator, and Q14, the transmitter local oscillator. During Tx measurements, the keyline was grounded.

	Q1	Q2	Q3	Q4	Q5	Q6
E -	8.7	3.8	3.3	2.6	0.1	0.0
B -	9.4	4.3	3.9	3.3	0.8	0.7
C -	13.8	6.9	13.8	13.4	12.5	2.3

	Q7	Q8	Q9 Rx	Q9 Tx	Q10	Q11	Q12
E -	1.6	5.4	12.7	0.0	3.2	3.2	4.7
B -	2.3	4.0	13.1	0.0	3.9	3.9	5.4
C -	13.3	11.0	13.8	13.8	5.4	5.4	13.3

	Q13	Q14 Tx	Q15 Tx	Q16 Tx	Q17 Tx	Q18-Q20 Tx
E -	4.7	4.9	1.4	5.0	1.5	0.1
B -	5.4	4.0	2.0	5.5	2.1	0.2
C -	13.3	10.3	5.0	12.2	10.6	13.8*

	Q21 Rx	Q21 Tx	Q22 Rx	Q22 Tx
E -	0.0	0.0	0.0	12.1
B -	0.7	0.0	0.1	12.8
C -	0.1	12.8	13.8	13.8

* This value was not measured during transmit, as too much r.f. is present. The level shown is always present on the collectors of the Q18 - Q20, when the rig is operating.

Appendix

The NorCal 2222 Design Contest
Wayne Burdick, N6KR

Introduction
The object of this contest is to build a ham-band transceiver using only one kind of active device, the venerable 2N2222 NPN transistor. NPN transistors can function at all stages of a radio--oscillators, RF or AF amplifiers, mixers, switching and timing circuits--and you get to use up to twenty-two of them!

Specifically encouraged is out-right theft of existing circuits to build the transceiver. In fact, you don't necessarily have to design a thing; if you want, just glue together existing circuits. But do it with style!

This contest will appeal to those who are all thumbs, since the finished rigs will NOT be judged by appearance or construction technique, just by what they do and how well they do it. The winner will have the honor of having his or her design named the official "NorCal 2222 Transceiver"! There will of course be tangible prizes, too, and there's always the possibility that the winning design might become a NorCal kit.

Rules
1. The 2N2222 is the cockroach of NPN transistors: no matter what happens to us or to the planet, you'll still be able to find them in huge quantities. Your task is to use these ubiquitous parts to design a radio for a post-apocalyptic world; a radio that could be built in any country, no matter how remote it may be from Silicon Valley.

2. You may use up to twenty-two (22) 2N2222-family transistors, including the 2N2222, 2N2222A, PN2222, PN2222A, exact NTE equivalent, etc. You can use as many other electronic and mechanical components as you like (including diodes), as well as any kind of packaging. But DO NOT use ICs or other transistor types.

3. Feel free to incorporate parts of published circuits into your design. Cite all references used, and try not to "borrow" more than 25% of your design from any single article. Also let us know which circuits you designed from scratch, or modified, and explain what you learned or observed in the integration process.

4. The transceiver may operate on any ham band(s) and any legal mode(s),

but must meet FCC regulations.

5. For each design you submit, please provide the following:

A. Working prototype of the design
B. Complete, readable schematic on one 8.5" x 11" page
C. One-page, typed description of the design, including operating instructions
D. Results from your own bench tests

Judging

Entries will be judged strictly by how creatively the designer applied the 2N2222 in his or her design. Entries will be NOT be judged by appearance, construction techniques, finish detail, etc., nor on how many 2N2222's were actually used. (For example, the judges will not be impressed by the use of a 2N2222's base-emitter junction as a simple switching diode.)

In general, performance of the radio will be inferred from the schematics and from the test results you supply. However, those rigs that are deemed safe to operate may also be tested on the air using real antennas. (If the judges have a lot of fun with your radio, it can't hurt your chances. If they fry their power supply and get dirty faces from exploding electrolytics, it *will* hurt your chances. Since you never know who might be judging--we certainly don't--try to make your radio foolproof!)