Building the SWL30+

by Chuck Adams, K7QO
August 29th, 2003

This article was written some time ago and I have come back to clean it up and redo some sections. This article appeared in one issue of the NorCal QRPp Journal. The pictures were done with an old Epson PhotoPC digital camera. I am in the process of redoing some of them to improve the quality. So you may want to check back and watch the above date to see if any thing has changed. You may print this article off and give it to some young kids or old ones too that may have some interest in building their own radio. You do need a current amateur radio license to use this radio on the air.

Dave Benson, NN1G, has a company called Small Wonder Labs (SWL). He is located near ARRL headquarters in Newington CT and his web page is

`http://www.smallwonderlabs.com/`

Current information on his company, SWL, and the kits that he has available can be found there. Dave has kindly given me and others permission to use his SW-30+ design as a tool to use to help people learn and build. This project was part of the Elmer101 series on QRP–L. As a starting point for that goal I have purchased one SWL SW–30+ transceiver from Dave. I already had one such transceiver in operation and am using it almost daily until November 11th, 2001 to work all states and a hundred countries at half of one watt to a vee-beam. I bought this kit in order to build it from scratch and document the process of how I would build a kit. This write up is to be used as a learning tool for those that have yet to build a complete kit. I'm a regular customer of Small Wonder Labs and Dave can tell you and I get no special deals for doing this. I am just interested in getting as many QRPers on 30 meters as I can and to also teach building at the same time.

The schematic and the parts list are at the end of this material. I will use digital pictures to show you how to build a kit and this article is written assuming that you have never built a kit. The series of pictures and written material shown here is for those that want to start with a kit first before moving on to more difficult projects. This material may be used by those that wish to instruct others on kit building also. This information is step by step and filled with details of the building process and some notes on what each stage of the transceiver does. Not in real gory detail but in enough detail to help you comfortable with the construction and operation of a rig. You do not have to be a licensed radio amateur to build anything, but you do have to have a license to operate this transceiver on a real antenna and on the air, i.e. transmit RF energy into free space outside your home.

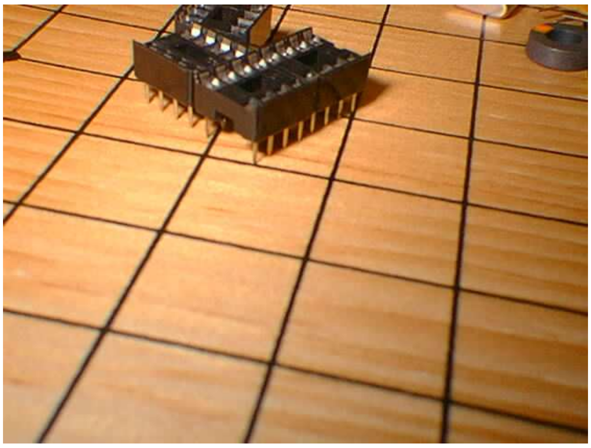
If you are an experienced builder you might want to consider using this project to teach new hams or want to be hams the craft of building and having a life long hobby. In this day and age of computers and high tech toys the expense of approximately $100 for a complete transceiver that will last for years to come is a cheap investment, IMHO. Feel free to use this material and give credit where due. Thanks and good luck.

Parts is Parts

When you get the kit from Small Wonder Labs the first thing you need to do is get organized. So let me tell you how I do it and you can either follow my suggestions or work out your own methods. My techniques are not the only way to solve a problem, but I have been building for many many years and these work for me. My success ratio is very high. In the package for the SW-30+ there are a number of bags. The clear zip lock, which I had in this kit, contains a large number of parts. Here is a photo of them out of the bag and on a small surface in a pile.

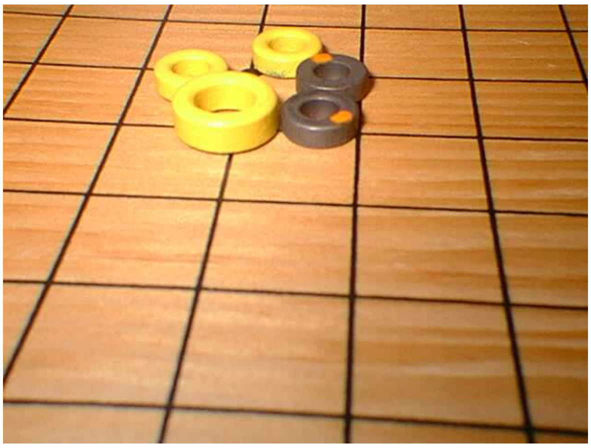
Parts in a pile.

The first thing that I do is break out the parts by type and put them in small bathroom Dixie cups (new and unused). But you can use muffin tins, small plastic cups, jars, etc. Do this with the zip lock baggie first. There is a another small black anti-static bag that you should not open yet. Here are pictures of the various parts to help you identify them if you have never done this before. I have given some references to the latest ARRL Handbook for reading material if you are very new at this and wish to learn more. Suggested reading is for all as you may have forgotten some of this stuff. If you do not own a 2003 HB, then any from 1995 to 2003 should work also. The page numbering may have changed slightly from issue to issue.

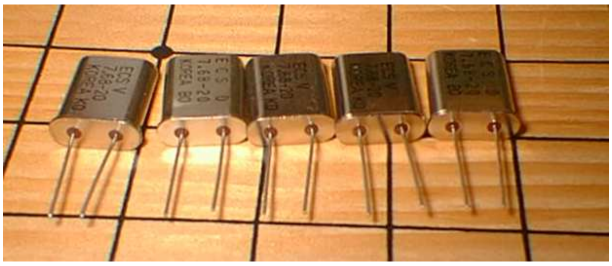


Integrated Circuits Sockets.

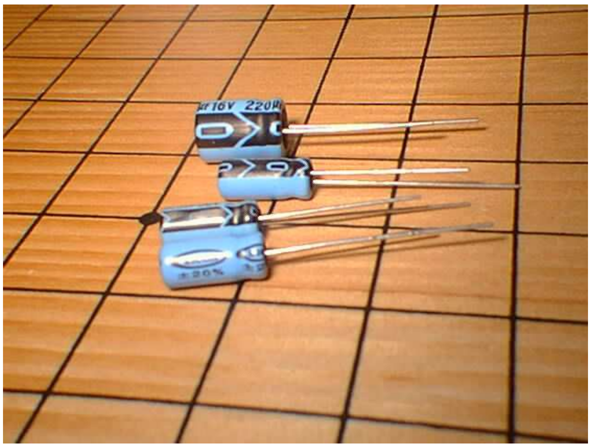
IC Sockets in Cup.



Toroids. See HB 10.8–10.14

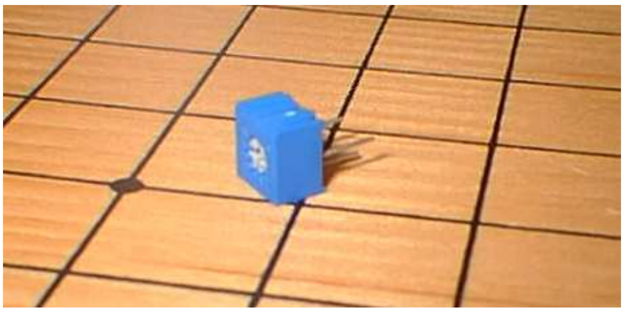


Crystals. See HB 14.21–14.28

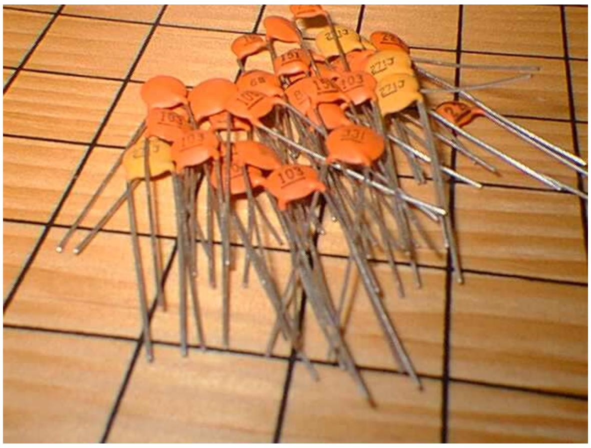


Electrolytic Caps. See HB 6.7–6.14

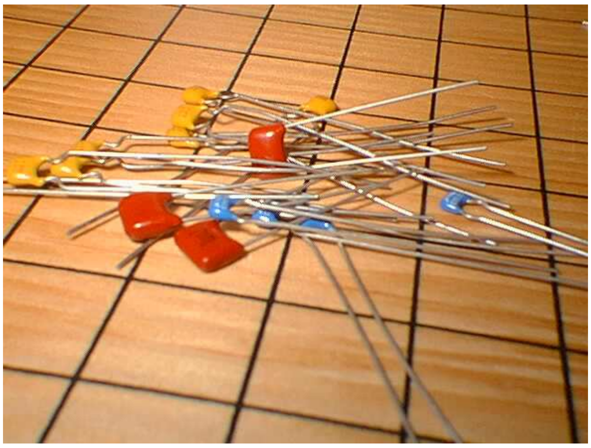
IF Cans. See HB 6.14–6.26



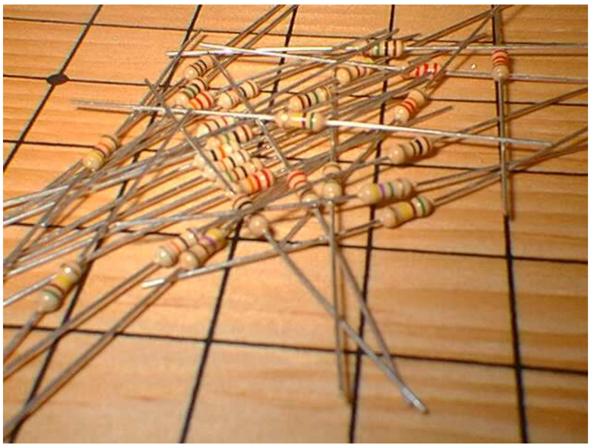
Variable Resistor. See HB 5.2–5.6



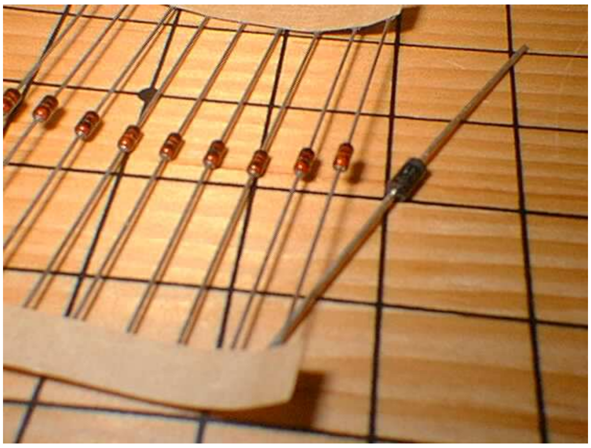
Disc Capacitors. See HB 6.7–6.14



Mono Capacitors. See HB 6.7–6.14



1/4 Watt Resistors.



Diodes.

Zener Diodes.

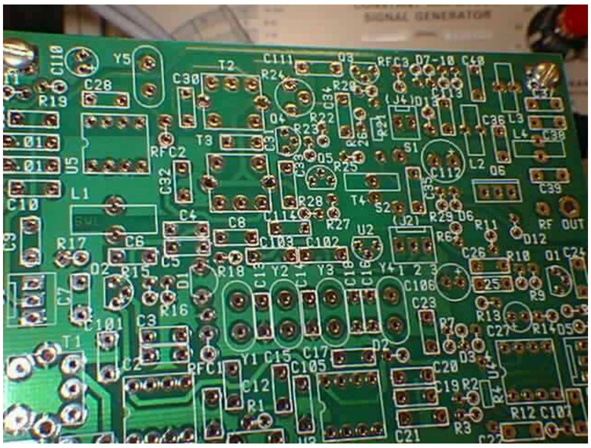
Do not remove the zener diodes from the paper.

Just put them into the paper cup as they are from the package.

Anti–static bag.

The anti-static bag contains ICs, transistors, etc. Do not open it at this time. Please.

OK, I know you are getting anxious to start building, so let's start slowly so that I can show you some tricks of the trade — so to speak. Here is a photo of the printed circuit board that you will building the transceiver kit on.



PC Board.

Note that the printed circuit board for the SW30+ is what we call a double sided plated through board with silk screened outlines and labels on the component side. The component side is obviously the side upon which all the components that go on the board will be placed and soldered from the other side. Later in life you may run across a kit or kits where parts may also be placed on the bottom or underside of the board. We will not be concerned with that here. Note how nicely the board is done and don't be scared by the fact that the board looks crowded. You are going to place one part at a time on the board, so don't rush and just take your time and double check each step. Even as an experienced builder I double check my work as I go along. I find that it saves me a lot of trouble at the end trying to find simple mistakes.

I am a person that likes to build kits and electronic circuits in stages and test them as I go along. It gives me positive feedback that I am doing things correctly. It is also much more easy to find problems as they occur instead of getting to the end and not having a clue as to where something or several things went wrong. Also, by setting up a plan at the first I can learn more about the system as I go. You just have to do this a couple of times and you too can be an expert at doing this. So allow me to show you just how I think through this process using the SW30+ as an example.

First take the schematic for the rig and study it for a period of time. In a transceiver the three main building blocks are the variable frequency oscillator or VFO for short (also know as local oscillator LO). Then there is the receiver and the transmitter sections. These will be broken down even further into smaller sections. The entire rig is powered from some outside source in most cases. Dave gave me permission to reproduce his schematic and I have redrawn it using a program that I wrote. I have used dashed outlines for some of the sections to help identify them. There may be one or two components inclosed within a section that you can argue about it's belonging there, but this is not rocket science here and I'm not being picky.

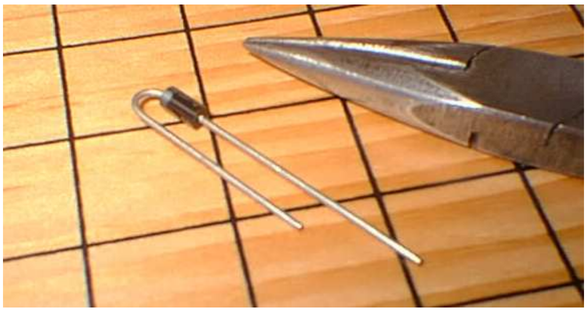
I have come up with what I call the K7QO coordinate system for beginners. I know that the first time you do something like this you are totally overwhelmed by the amount of stuff you have to learn. Determining just where parts go on the board is difficult and even after many years of building I have difficulty with a new kit and components aren't where I assume that would be. Dave has a nice parts outline with the kit documentation and that helps, but there are still a couple of items that a tricky to locate at first. So here is what I'm going to do. With the board oriented with the labeling where you can read it, let's call the lower left hand corner of the PC board the origin, i.e. X=0.0 and Y=0.0 as in a Cartesian coordinate system. I'll measure distances in centimeters (cm) from the corner and indicate a part location using the notation (X,Y) for distances in the horizontal direction (X) and the vertical direction (Y). Think of X as being the distance from the left hand edge going to your right and Y as being the distance from the bottom edge of the board (the edge closest to you when the board is in front of you). Again, all distances are measured in centimeters 'cuz I like the metric system a lot. So for something like D13 I'd indicate (7.5,5.5) and don't take the distances to mean that I got them exactly on the nose. They are just to get you as close as possible to make the part outline and label easy to find. OK? Neat idea if I do say so myself.

You might want to take the schematic and the PC board and study them together. See where each part on the schematic fits on the PC board. See if you can see the traces that correspond to the lines on the board. But don't start putting parts on the board yet. Let's do that together and in a specific order. Also read through Dave's manual as he has some excellent material. I am going to build this rig in a slightly different order and in detail to aid the beginners. If you have experience and you don't need me, then fire away and good luck.

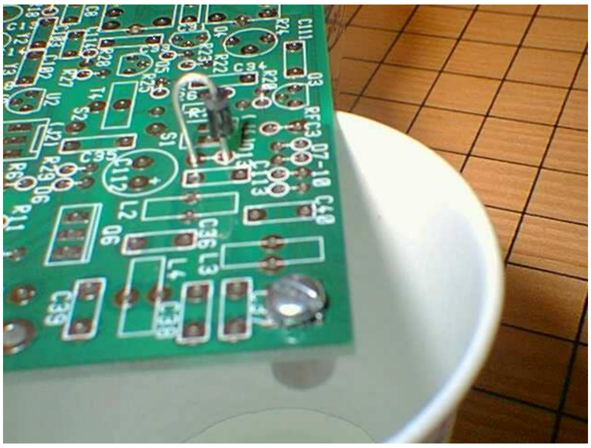
Power Setup

For this article let's start with the power distribution. If you are using the schematic that came with the rig, look for J4 on the center right hand side of the page. In my schematic (the PDF link above) there is no power plug shown where J4 should be and the connection is shown as VIN just to the right of D13. This is where the 12 to 15V DC comes into the rig from the outside world. Before the supply voltage can get to any other part of the transceiver it must pass through D13. Several reasons for this diode. It is too easy for any one in the world to connect a battery or other power supply into the rig backwards, i.e. with the polarity reversed from the correct connection. If this reverse condition ever occurs there will be considerable damage done to the ICs, transistors and other parts immediately. So the diode prevents a reverse polarity on the input plug from doing any damage. The diode will also burn out at 1 amp of current so it will act as a fuse in some other cases but, unfortunately, if one ampere flows for even a short period of time for whatever reason there will most likely be some damage done to the rig. This is critical information. You will note in all of the following pictures that I have four standoffs each placed at one of the four corner mounting holes of the PC board. If you can at all do this, then do the same thing with either nylon standoffs or if you don't have anything else use some small 1/4" long screws and nuts to just place the board above the table surface when it is just sitting. Later we will be powering up the board, i.e. applying 12V to test things. The worst thing you can do is have the board sitting on a work surface when there are metal wires under the board and you apply power. These wires will cause short circuits and burn out traces on one or more sides of the board. This is an expensive error. Don't do it.

So let's install the diode D13 first at (7.5,5.5). Here is the way I go about installing a part. First identify the part. In this case it is a black diode (in the diode cup) with 1N4001 printed on the body of the diode with a white band on one end. This white band is the cathode end of the diode and corresponds to the end of the schematic symbol with the line perpendicular to the arrow. All diodes in NN1G kits are installed with the band away from the PC board, i.e. that end is up. Here is a photo of the diode, the way it is bent, and the chain nosed pliars that I use.



D13.

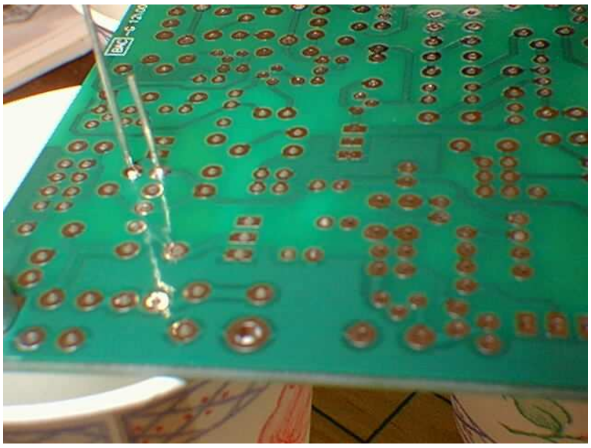


D13 on the PC board.

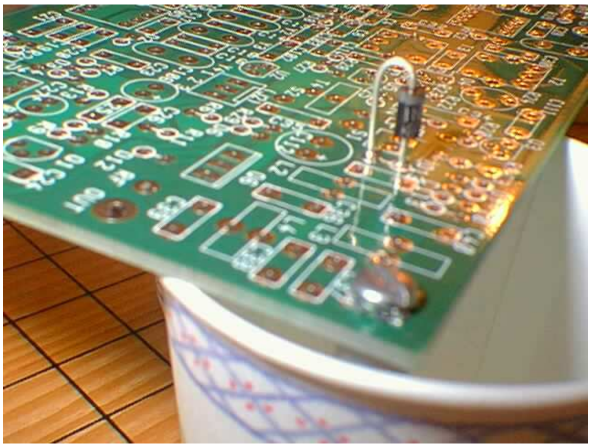
D13 on the bottom side of board.

D13 being soldered.

A side note on my soldering technique. Since I use a solder from Kester that is 62/36/2 based and has a low melting point and a 2 percent silver content, I like to place the solder between the iron tip and the component lead and PC pad and then apply heat. I know that a lot of books including the ARRL Handbook suggest different. But I find that the melting of the solder distributes heat to the joint rapidly and I spend no more than three seconds (the K7QO 3-second rule) in soldering and I remove the heated tip of the iron. I get the solder to melt and flow in this time period and I don't over heat the part on the other side of the board. See the Manhattan Article on my web page for more details on what solder I use, etc.

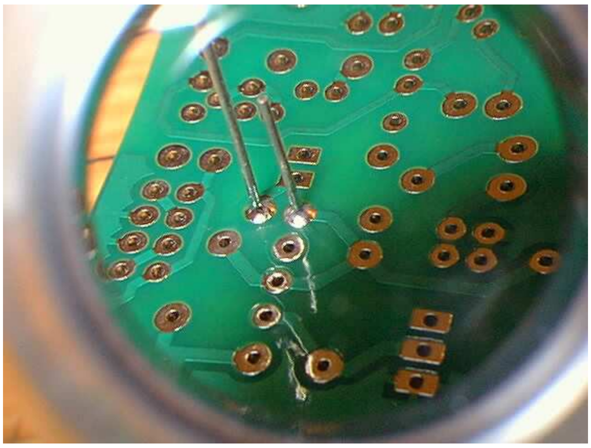


D13 soldered in place.



D13 soldered in place.

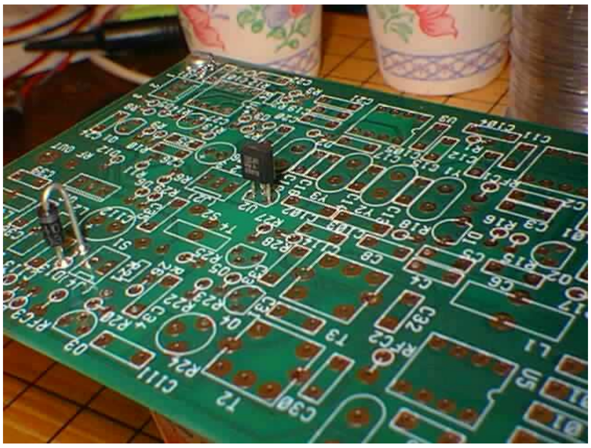
Diode D13 on top side of the PC board after being soldered. Note the solder that has wicked through the plated through hole. See Dave's illustration in the SW–30+ manual on soldering.



D13 soldered in place.

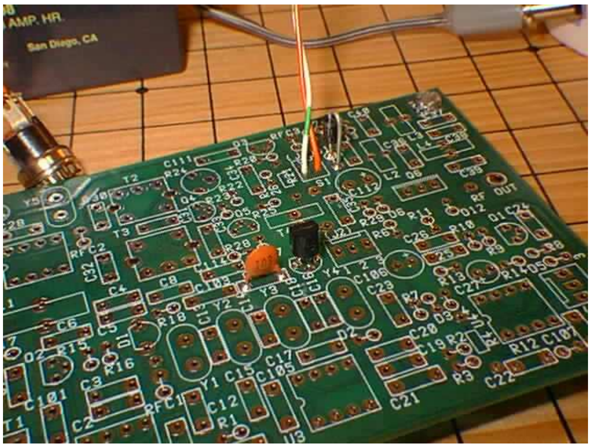
Diode D13 bottom side of the PC board after being soldered. This is a magnified view. I used one of the lenses from a 8x Loupe to do this one. I love it.

Now that you have successfully installed the first part, let's do two more. U2 at (6.0,3.5). U2 is a 78L08 8 volt voltage regulator. It is located in the black anti-static bag and use the techniques for working with these parts as described in Dave's manual. The 78L08 looks like a small transistor. It has three legs on the bottom. Install it at the location (6.0,3.5). Make sure you have the right part and make sure you put in the right direction by following the outline on the PC board. See the photograph if you have any questions. You do not want to ruin this part or any other part for that matter. Since no one is timing you just take it slow and easy and double check everything that you do. You'll do OK.



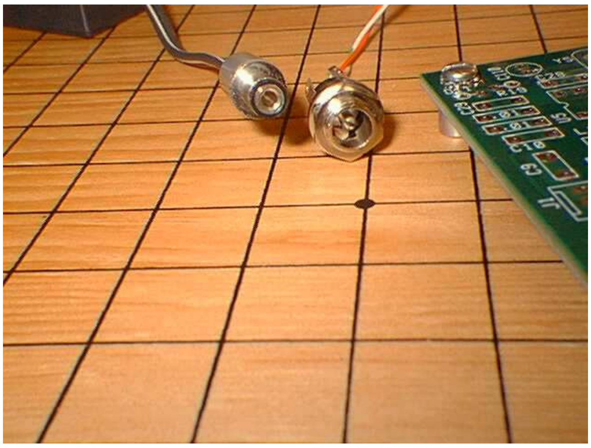
78L08 soldered in place.

Now look in the disc capacitors (caps) and look at the side for one that has 103 on it (103 means pF which is 10,000pF or 0.01uF. (A note: the British like to use nF for nanoFarads, which is what 1,000pF is, thus 10,000pF is 10nF and you'll see this notation used a lot in SPRAT and other British based publications.) This is a 0.01uF (microFarad) cap. Solder it in location (5.0,3.5) where C102 is labeled. Do not force the leads (legs) of the cap down into the holes so far that the covering material also is in the hole. I leave just a very very small amount of metal showing on the top side to allow the solder from the bottom side to wick or flow onto the top side of the pad. Don't leave a lot. I like the parts close to the board and not more than a few mm of bare lead above the board. Again look at the pictures next to see just how I do it. Do not force any part into the board. They break easily or you may create a small fracture that you will not see and the part(s) cease to work.



C102 left of 78L08.

At this point I want to show you something. This will be difficult for me to go into every little detail on, but let me try to convey enough info that you can do this. I am assuming that this is your first time to build and I hope you want to learn as you go. Time spent here will aid you the rest of your life in building. If you purchased the case, mounting hardware, pots, etc. then you want to install J4 (the power plug) on the board. If you do not have the case, then I do the following. Find two wires about 10 to 15 cm long. One red and one black. Lightly solder these into the two holes at J4 (7.0,6.0). The red one into the right hand hole and the black one into the left hand hole. With these two wires you can now apply a 12V supply to the board for testing. If you are building from the kit from SWL then look on page 15 to see the power connector setup at the top center of the figure. I use a 12V 0.8AHr gel cell for small rigs. I also use a plug and connector for battery supplies. All that is important you have a way to connect the battery to the transceiver for testing and hopefully you can disconnect rapidly if you need to in case of emergency. Also place a small fuse in the power line to prevent accidents and damage to parts. The center conductor is + and the outer conductor is -. Measure with a digital multimeter after wiring these. A word of warning. When working with batteries and I do this a lot. You must make extra sure that you do not short out the wires from the battery. If you are soldering wires to "live", i.e. charged batteries and you should always assume that they are charged, you can get into serious trouble in a hurry. Some of these batteries can easily generate 100 amps of current for a short period of time and that is a fire hazard to say the least. Get expert help if you are not sure about what you are doing. Better safe than sorry.



Power Plugs.

12V Gel Cell.

Now get one of the glass diodes, a 1N4148, and solder it into place at D2 located at (6.0,1.5). Do it in a similar manner that you did D13. Be careful as the glass easily breaks. Be sure to read Dave's instructions that all diodes mounted vertically have the banded end at the top away from the baord. Proud to say that this was my suggestion years ago as the first kits had different orientation for some diodes and it could cause confusion.

If you are following these building instructions then it is time for your first test. Make sure the bottom of the board is above any conducting objects. Use standoffs please. Connect the battery to the transceiver at this time. Watch for signs of trouble and gently touch the 78L08 to see if it is hot or the wires from the battery are hot. This means that a lot of current is flowing and you have a short somewhere on the board. At this time with so few parts this should not occur, but I'm not there to help you. Make very sure that you always connect the + terminal of the battery to the wire that goes to Vsupply (+) and the negative battery terminal to GND or negative (-) lead from J4. Pay attention at all times and if you are tired then you should quit. The rig won't run off in the middle of the night. Another warning. Touching parts is a dangerous practice. I have touched metal transistors that at the time were running a lot of current and were over 100 degrees C and I wound up with a blister on my finger. So don't do this as a general habit. Please.

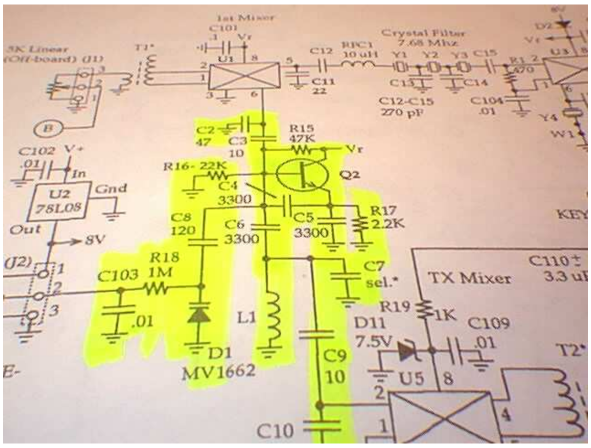
With power applied, use a digital multimeter in the VOLTAGE position and measure the voltage at J2 pin #1. This is at (6.5cm, 3.5cm) K7QO coordinates. You should read close to 8.00V. Oh, you might not know about DMMs. The red lead should touch ONLY the point at which you wish to measure the voltage. The black lead should touch one of the standoff screws or one of the pads at the four corners. The pads are square and tin plated. I did get exactly 8.00V, but this does not mean that you will. Parts will vary slightly and you may read something like 7.89V or even 8.07V. Anything within a 5% or so is fine. This is not rocket science. Look back at your schematic to see at what point you are doing your measurement. I emphasize the point that you double check to see that your meter is in the voltage position. I have lost several fuses in the multimeter by having it in the wrong position.

At this time with very few parts on the board making measurements is easy. Touch the probe from the DMM to the center of the pad since there are no leads from parts soldered at this time. Later, when you have to make measurements in a crowded area you have to pay attention at all times and pay particular attention that you do not touch the lead(s) to more than one pad or lead at a time or you can cause a short and destroy one or more parts in the process. I don't mean to be always warning you of things but be careful at all times.

OK. Look on the board at (2.0,1.4) in the lower left quadrant and you will see a rectangular outline labeled U1. This is for an IC socket. See the left side and the round dimple on the outline? This is used to show how the IC will be installed. Starting at the bottom left the pins are numbered 1, 2, 3, and 4 for the bottom row. Continuing in a CCW direction the top row will be pins 5, 6, 7, and 8 from right to left. If you look at the schematic you will see that pin 8 (the upper left pad) is connected to V_r (read as V-sub-r) which comes from D2. Measure the voltage at pin 8 of U1. This voltage will be less than what you measured as the output from the 78L08 because of the voltage drop across D2. I get 7.72V. Look carefully at the PC board and you will see a conductor running under the green solder mask up from pin 8 to the bottom pad of Q2. This is the collector of Q2. Measure the voltage at this pad of Q2. You should get the same reading as you did previous at pin 8 of U1 (mine was 7.72V). Follow the PC board trace from pin 8 of U1 to the right and you'll see it winds around (technical talk for it does not make a straight line path) to pin 8 of U3. Measure the voltage there also. Get the same voltage as before? Good. Look back at the schematic and you should see a line from pin 8 of U3 to a crossing of lines and off to the left an arrow pointing to V_r. This means that a conductor or wire goes from this point to all other points labeled in the schematic as V_r. It saves clutter in the schematic and is done regularly. For the schematic that I produced for this rig I show the wire. No big deal.

I'm having you do this exercise now to help you see what is going on and to check out work as we progress. It makes building a lot easier, gives you positive feedback, and trains you to make careful measurements. You may wish to keep a journal on each step for future reference and to aid in asking for help later on. OK, now unplug everything from the rig (transceiver) before proceeding. Make it a regular habit to turn off things when you leave the lab or area you are working in and before going back to soldering parts into the board.

VFO Section



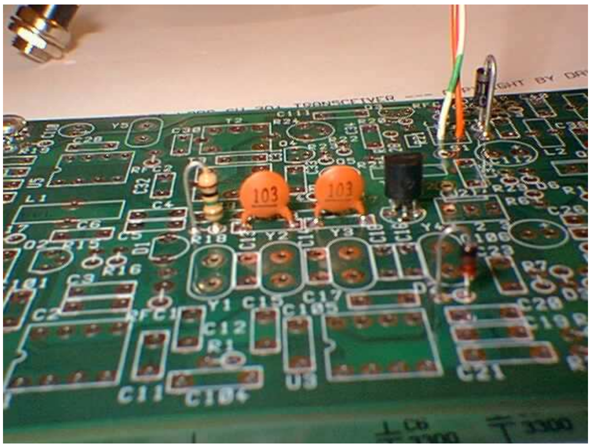
VFO schematic section.

Before you begin building anything may I suggest something. Make a photocopy of the schematic. You don't have to do that here as you can easily go to my web page, bring up the Adobe PDF reader and print out the schematic that I have already done for you. Then with a highlighter of your favorite color you should highlight each component as you install it. It will keep you out of trouble. I did that, but I forgot in this writeup to put in L1 when I first published it on the web. This caused me some embarassment on QRP-L when someone brought it up publicly. In doing a rewrite and update here is what I did. I printed off this article (19+ pages on my printer) and with the highlighter and red pen corrected things and modified text as I went along. I also printed off another copy of the schematic and highlighted parts as they appeared on the page. This way I guaranteed myself that I left nothing out. Well, as much as one can guarantee things. :-)

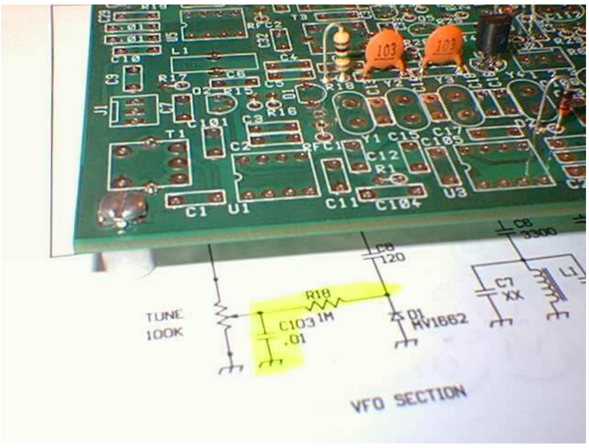
We are now going to build the variable frequency oscillator section of the transceiver. You may think of this as the heart of the transceiver. The oscillator will be used to determine the receiving and the transmitting frequencies while in operation. It is variable, i.e. the frequency may be changed. This is accomplished by using a variable capacitor (varicap) that is actually a diode with capacitance between the anode and cathode that is varied by applying a voltage across the diode. This diode is D1 and is a MV1662. C103 is a bypass cap to keep the radio frequency oscillations (RF) from the tuning variable resistor and the voltage regulator U2. R18 is a current limiting resistor and also serves as an isolation resistor so that the oscillator does not see C103 and the rest of the circuit to the left of D1 in the diagram. C8 keeps the DC voltage away from Q2 and the oscillator itself as C8 blocks the DC voltage from getting to the junction for R16, C3, R15, Q2 base, C4, and C6. More on the circuit in a bit.

Let's build up the VFO. I'll go from left to right in the schematic and just list the parts in order with their part designation, markings to help identify them, their actual values, and then their K7QO coordinates on the board. I will also at points show you a picture of what the board looks like at that point. This will scare you, but I will show you D1 in place but I will have it missing in a few pictures because I am going to do an experiment for the MH101 group. You install D1 when I show it in the following list.

Part	ID markings	Value	Coordinates
C103	(103)	0.01uF disc	(4.0,3.5)
R18	(brn-blk-grn)	1M resistor	(3.5,3.5)

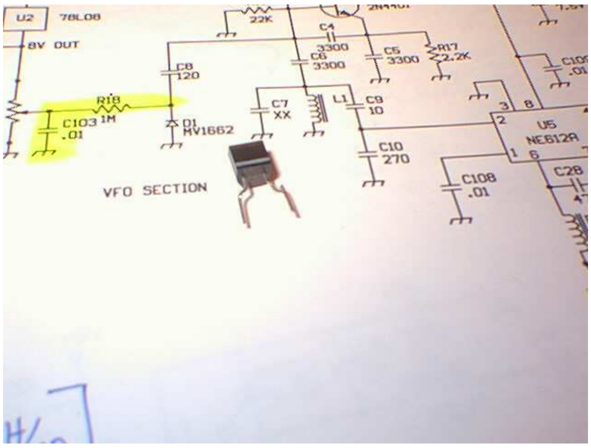


Parts in place.



Parts in place.

And schematic with parts highlighted that have been soldered in place.



MV1662 diode.

Part	ID markings	Value	Coordinates
D1	(no markings)	MV1662 diode	(3.0,3.0)
C8	(120)	120pF NPO mono	(3.5,3.6)

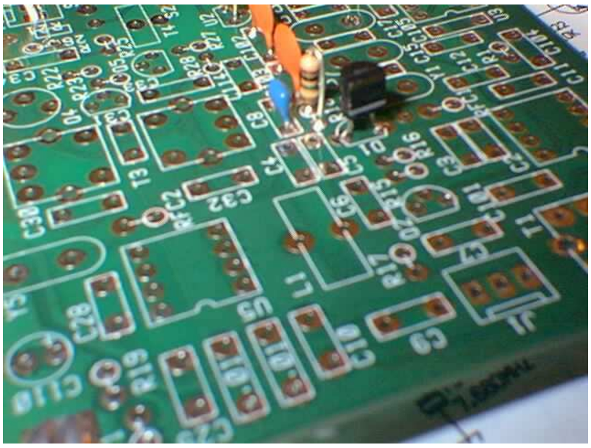


Photo of installation.

Part	ID markings	Value	Coordinates
C6	(332)	3300pF NPO mono	(2.0,3.5)
C4	(332)	3300pF NPO mono	(3.0,3.8)
C5	(332)	3300pF NPO mono	(3.0,3.5)

In the next step you will install R17. Before doing so look at its outline on the board and note the ground plane. You will install resistor reversed in order that the lead that is at the top will not be the ground plane just in case we need to measure this. If you have already installed the resistor in the direction on the board, then do not remove it. It will not affect the operation of the circuit at all.

Part	ID markings	Value	Coordinates
R17	(red-red-red)	2.2K resistor	(1.2,3.4)

R17 is left of C6 and note installation opposite of outline on board. See Dave's note in manual on page 9 left hand side margin of the diagram.

Part	ID markings	Value	Coordinates
C9	(10)	10pF disc	(0.6,3.5)
C10	(271J)	270pF disc	(0.7,4.5)

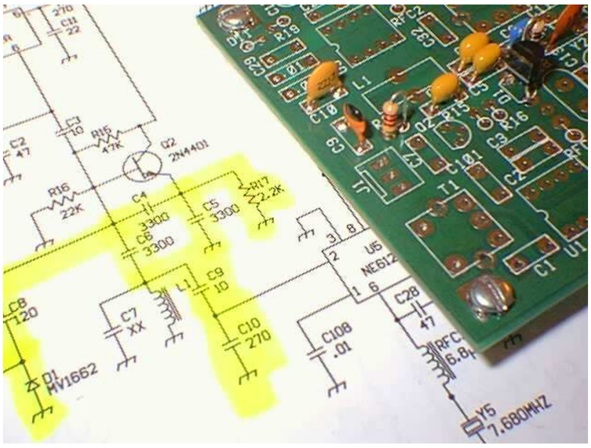


Photo of installation.

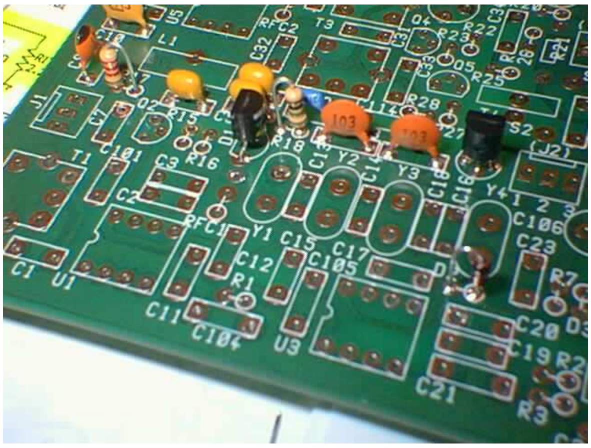


Photo of installation.

Part	ID markings	Value	Coordinates
R16	(red-red-org)	22K resistor	(2.6,2.8)
R15	(yel-vlt-org)	47K resistor	(2.4,2.8)

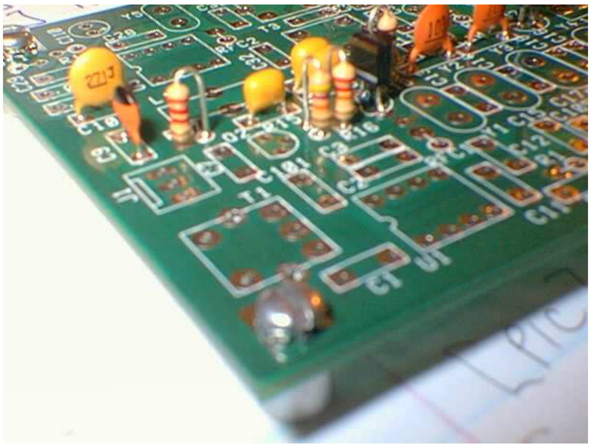


Photo of installation.

Part	ID markings	Value	Coordinates
C3	(10)	10pF disc cap	(2.5,2.1)
C2	(47)	47pF disc cap	(2.5,1.9)

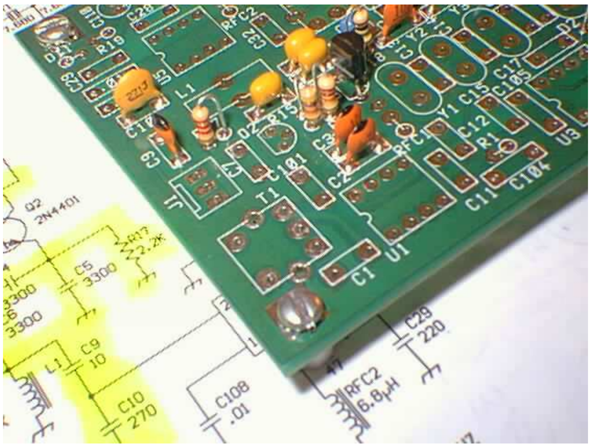


Photo of installation.

Part	ID markings	Value	Coordinates
Q2	(2N4401)	2N4401 NPN	(2.9,2.9)
L1	T50-6 0.50" diameter yellow toroid		

L1 has 29 turns of number 24 wire that is 47cm long. See building manual for details on toroids and winding or see the ARRL Handbook.

This completes the VFO section. Please note that we have not installed the capacitor at C7 yet. We have not connected the tuning pot that will be on the front panel yet. I prefer not to do this at this time in order to save the hassle of having it flop around during the rest of the assembly or having to unsolder it after doing some tests.

VFO Alignment

OK. Congratulations are in order. You have just completed a major project. It is now time to test the system to see how it is working. Let me break this down into several ways to do it all dependent upon your skill level and just how much money you have spent in your life up to this point on electronic test equipment. If you have been a regular lurker on QRP-L then you may have picked up some of the bargains that came along at Radio Shack and other places. Then there is some test equipment that you can build and this is how you can get started on the journey to building a lot of stuff.

DMM Only

A digital multimeter is a valuable tool and very inexpensive in this day and age. Harbor Freight just recently had one on sale for $9.99 and sometimes you can find DMMs on special sale at Radio Shack. Let me warn you. There are people out there that will give you 1,000 reasons why you should not use one or trust one. These individuals will tell you that doing a measurement will affect the circuit. Heisenberg, a famous physicist who may or may not have slept here or there, determined that if you measure anything you affect it. True here, but we are just trying to determine if things are working and this is not rocket science. So ignore every one but me. I do know what I'm doing here. :-)

I am going to have you take some measurements with the power applied to the circuit thus far built. I will have you measure voltages at points that are easy to reach. Please be careful and not drop the digital probe and make sure the pointy end thingy only touches one thing at any one time. Don't get distracted. Remember that the point at the end is a conductor made of metal. If you happen to allow it to touch more than one point at once then you can create a short circuit, i.e. 0 ohms resistance between two points, that might just cause a lot of current to flow and destroy all your work. Been there. Done that. It's not a pretty picture. I'm not trying to scare you here, just let you know that things do happen if you are not careful and even when you are and you think you know all there is to know about everything. :-) ;-)

For these measurements use the negative lead (-) of the DMM to connect to one of the standoff ground pads at either of the four corners of the PC board. Then use the positive lead (+) of the DMM to make these measurements. With the components standing vertically on the PC board we will touch the probe to the lead at the top of the part. This is easy to do and not very dangerous compared to trying to get a probe to touch a lead at the base of the part and between other close parts where we could easily create a short circuit. I will give you the part number and the voltage that I get. You DO NOT have to get the same exact voltage that I do. Just in the ballpark will do. Problem with digital stuff is that people tend to take the numbers as the only correct ones. :-)

We have not connected the tuning variable resistor to J2 and we don't need to. The oscillator will run just fine without it. We just can't adjust the frequency. We can do something later in this section if you have the equipment to do the adjustments.

```
R18     0.070V
R16     2.31V
R15     2.31V
R17     2.98V
```

If you installed R17 differently from the outline on the PC board as per Dave's note on page 9 and my suggestion previously in this writing. Otherwise you will get 0.0V even if the circuit is operating properly.

```
D2    7.49V
```

If you got close to the above voltages then you have a working VFO. In fact if you got a non-zero value at R18 then the oscillator is running. The diode D1 R18 and C103 combo acts like a RF probe and proves that RF currents and voltages are at work in the circuit!! Are we good or what? The A-team at work, you and me kid. If you don't have the equipment to do the following sections, then not to worry. Your VFO is working and you can move on to the next building section.

Frequency Counter Measurements

If you own a frequency counter, like the Radio Shack one that was on sale last year I believe. Model number 22-306 is the one that I have and I also have my trusty Heathkit IM-2410 that I dearly love. Anyway, using the frequency counter measure the frequency (while the VFO is powered up) at pin 6 of U1. If you look at the schematic you will see that we are getting output from the VFO through the coupling cap C3 which is very small, 10pF. This C3 cap helps keep the VFO somewhat isolated from the NE602/NE612 mixer chip but couples some of the RF energy to the local oscillator (LO) input of the mixer.

Also measure at pin 2 of U5, the transmitter mixer. You should get about the same thing. Because of the affect of the counter on the circuit may vary at the two places you won't get the exact same reading but pretty close.

The intermediate frequency (IF) for this rig is 7.680MHz. This means that in order to get the frequency range of 30 meters, 10.100MHz to 10.150MHz, you have to have the VFO working at 10.100MHz - 7.680MHz or 2.420MHz for the low end of the band and 10.150MHz - 7.680MHz or 2.470MHz for the high end. This tuning range is not possible with the SW-30+ VFO circuit as it is and I measured about 36KHz range, which is just fine.. As it turns out, without the tuning voltage applied to R18 and thus D1, the VFO is operating at its lowest frequency. The toroid that we wound with 29 turns, L1, helps determine the operating frequency of the VFO. After powering up my VFO and checking its frequency with the frequency counter I found that the frequency was 2.4748MHz which is too high. Since the frequency formula is:

$$f = \frac{1}{2\pi\sqrt{L * C}},$$

in order to lower the frequency we either increase L or increase C. Increasing L means removing the L1 toroid and rewinding with an added turn. Too much trouble I guarantee you. So Dave was smart enough to include in the kit some extra caps with values of 22pF, 47pF, 68pF, 82pF, 100pF, etc. These he shows in his parts list as C7A, C7B, etc. Here is what I do. Cut the leads on these parts to about 1.0cm and set their leads to slightly larger spacing than the two holes in the PC board for C7. I then put one cap in the C7 position with a friction fit to hold it in place on the board with the leads making contact. I then measure the resonant frequency of the VFO with the cap in place. That way I do not have to solder, measure, unsolder, solder the next cap into place, measure, unsolder, etc. Remember we are shooting for 2.420MHz here or within a few KHz.

Here is what I get for my VFO and the 3 caps that I had to use to get close.

```
C7= 0pF     2.4748MHz
```

```
C7=22pF      2.4523MHz
C7=47pF      2.4260MHz
C7=68pF      2.4070MHz
```

So the 47pF is close enough. I soldered it in. Now comes the tricky part, well kinda the first time you do this. You can further change the frequency by "squeezing" the turns on L1 closer together. Please note. This does not change the inductance of the coil. If it did, I can get you a nobel prize in physics if you prove otherwise. What you do is change the distributed capacitance across the coil. By moving the turns closer together you increase this capacitance and you further lower the resonant frequency of the circuit. Try it with just a small movement of the turns. By spreading the turns out a little you lower the frequency. Go the way that you need to go to get to 2.420MHz as the measured frequency out of your VFO. This will be the lower tuning range of your VFO. Good work. Take a break.

If by any chance you start out with the frequency of the VFO less than 2.420MHz, then you will have to unsolder L1 and remove one turn and resolder it back in place. It may be that you wound one too many turns, but then again the variance in the magnetic properties of the cores may have caused the slight problem. But you can fix it easily.

Now if by chance you happen to own an oscilloscope then use it to measure the waveform at pin 6 of U1 and pin 2 of U5. You should see a nice sine wave at each point. They do not have to have the same magnitude. I may come back with this measurement later if I get a chance to remember to do it. It is just nice to see the waveforms and know exactly what they look like.

Receiver Audio and Testing

Since I am assuming here that you are a first time builder and also that you don't have a lot of expensive test equipment, then let's build this puppy in an order that will give you some feedback as you go. Since you my not own any other radio equipment I have to consider that possibility and you can build this transceiver without a lot of equipment. Just some thinking and some planning and I'll do that for you.

Look at the schematic for the transceiver. You do not want to build the transmitter first as you may not have everything to test it out properly, so let's build the receiver. We are going to use your ears as a piece of test equipment. So I am going to start at the audio output and build the receiver from right to left on the schematic. And I'll show you some neat tricks that I use even though I have the necessary test equipment. By building this way I can do a quick test and not have to drag out some equipment, and get set up.

So here is the order we'll do this building stage. We are working from the top right hand side of the schematic. Starting with R14.

Part	ID markings	Value	Coordinates

R14	(brn-blk-blk)	10 ohm	(9.0, 2.1)
C27	(47uF)	47 uF	(8.5, 2.1)
R13	(brn-blk-grn)	1M	(8.0, 2.4)
C25	(821J)	820pF mono	(8.0, 2.9)
C26	(222J)	0.0022uF mono	(8.0, 3.0)
R11	(grn-brn-yel)	510K	(8.2, 3.4)

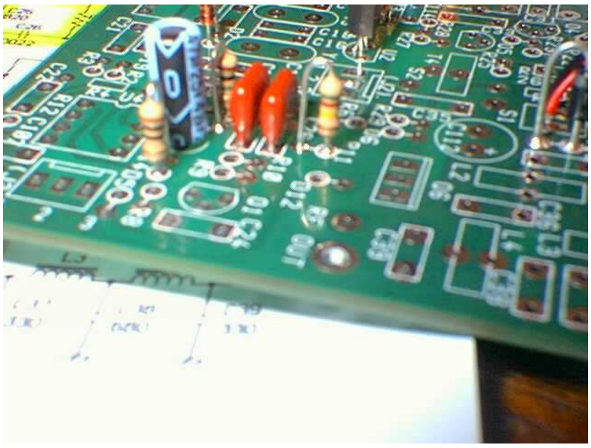
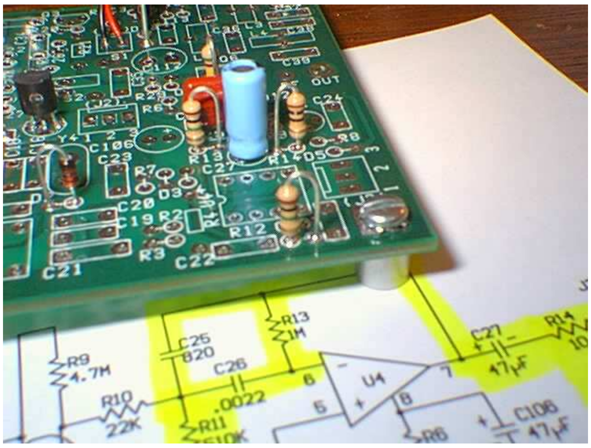


Photo of installation.

Continuing on....

Part	ID markings	Value	Coordinates
R12	(brn-blk-grn)	1M	(8.7, 0.3)



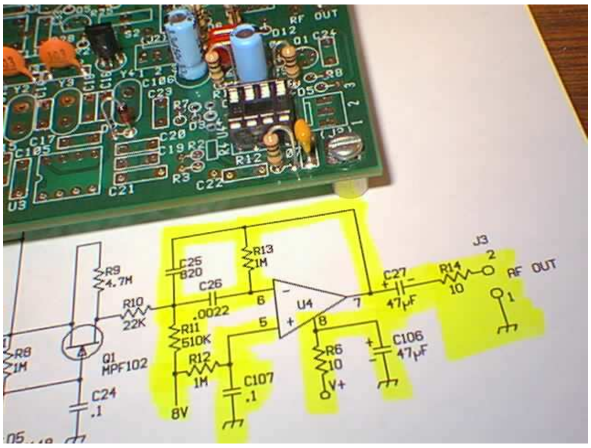
R12 on edge of board.

Part	ID markings	Value	Coordinates
C107	(104)	0.1uF mono	(9.1, 0.5)
R6	(brn-blk-blk)	10 ohm	(7.5, 3.5)
C106	(47uF 25V)	47uF	(7.4, 2.6)

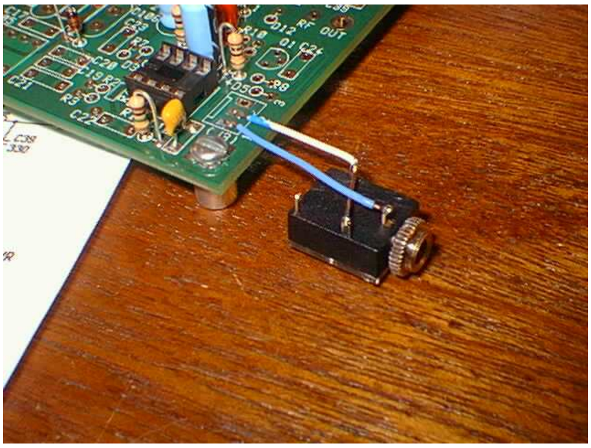
The 8 pin socket at U4 in the middle of the stuff you just put in. Make sure the little half-moon shape in the socket end matches with the white outline on the board. Hold socket in place with some masking tape and the solder ONLY TWO pins. I choose two pins on opposite corners to solder. Then turn board back over and look at the socket. Is it in the right place? Does the half-moon match the silkscreen outline? Is the socket down on the board all the way. Reheat the pins to adjust the socket if necessary.

I put a stereo 3.5mm jack at J3, pins 1 and 2 by using some small wire and tack soldering for temporary installation. See the following pic. Pin 1 is ground and pin 2 is the audio output. For stereo phones you wire pin 2 to both right and left (tip and ring of the jack.

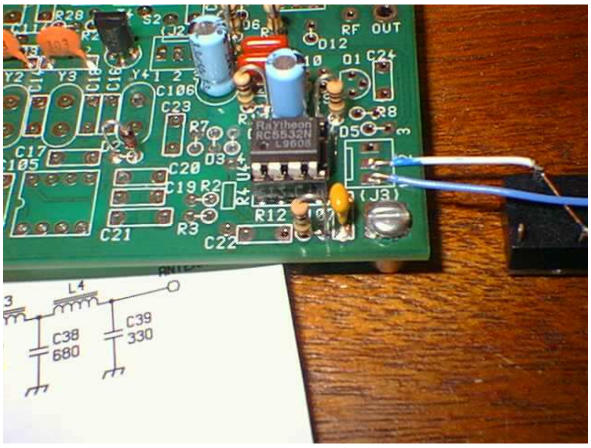
Look for the U4 IC labeled 5532. Mine was manufactured by Raytheon and had RC5532N on the top. Yours will most likely be the same or close to it. Note the small circular indentation on the top. This is pin 1 of the IC. It will go on the end where the half-moon is on the socket and board outline. Use Dave's instructions for making sure the pins are straight and carefully insert the IC into the socket. Do not force it with too much pressure. It may take a pound or so if it is stubborn, but make sure the pins are in the right position in the socket before you really push.



Audio stage wired.



Stereo jack.



IC in place.

Note the markings and you can see a dark circular dot that is the edge where pin 1 of the IC is located.

OK, take a break. Run off to the sink and wash your hands. You'll see why in a second.

OK, now the moment of truth. Plug in some Sony Walkman type headphones into the jack. Place the headphones just in front of the ears in case upon power up the audio stage starts screaming. Most likely it will not if you have been careful. Plug in the power source. In my case the gel-cell battery. You should hear a slight pop and then pretty much silence. OK. Lightly lick your pointing finger. (See why I had you wash your hands? :-)) Touch only the top of R11. You should hear some hum in the headphones at this time. Maybe a slight pop when you touch or rub the top of the lead to R11. This proves that your output audio stage is working correctly.

This pretty much concludes the building of the final audio stage of the receiver. Congratulations again on a fine job. We can now move on to the muting circuit next.

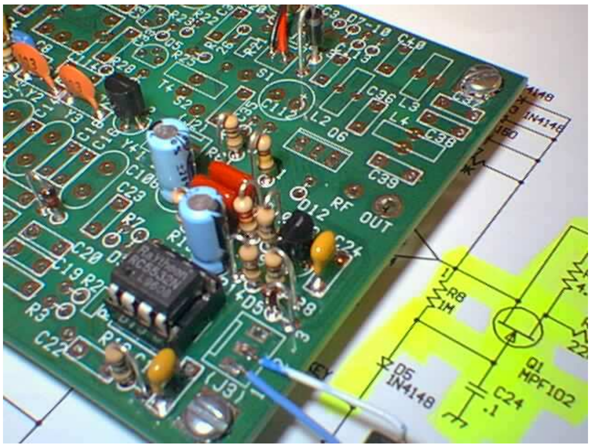
Muting Circuit and Testing

We now want to move to the next stage towards the front end of the receiver. This consists of Q1 and about 5 other parts so this goes quick. This circuit is something that Roy Lewallen, W7EL, came up with in his famous direct conversion rig for 40 meters. You'll see a lot of people using this circuit in their designs. By now you should be pretty comfortable identifying parts and figuring how just where they go on the PC board. Remember to concentrate and take your time. This is not a race.

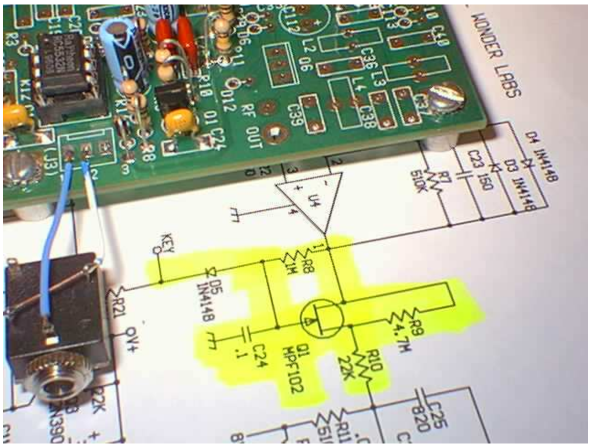
Part	ID markings	Value	Coordinates
R10	(red-red-org)	22K resistor	(8.6, 2.8)
R9	(yel-vio-grn)	4.7M resistor	(8.8, 2.8)
Q1	(MPF102 or 2N5485)	JFET	(9.3, 2.8)
C24	(104)	0.1uF mono cap	(9.7, 2.8)
R8	(brn-blk-grn)	1M resistor	(9.3, 2.2)
D5	(1N4148)	1N4148 Si diode	(9.7, 2.0)

Again, remember to place transistor oriented according to the board outline and do not force the transistor too far onto the board and break off one or more leads. I allow about 3 or 4 mm spacing above the board. (1/8")

See, that went pretty fast. Now here are some pictures of the completed section from different angles. Note the schematic and the highlighted parts that were installed. I find that when I do this I don't miss things. Forces me to take more time and care in building.



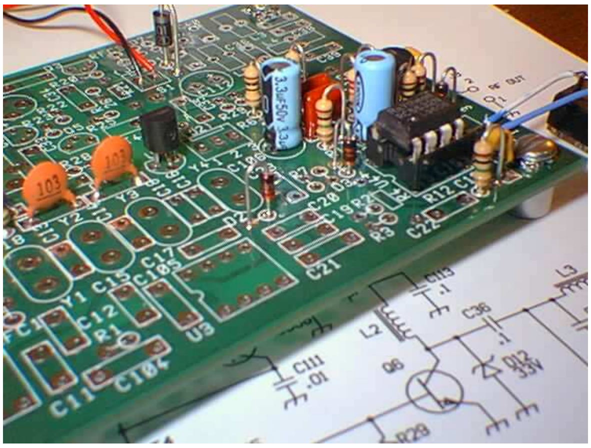
Muting circuit completed.



Muting circuit completed.

Now for testing. Hook up the headphones BUT DO NOT PUT THEM ON. This test will blow your ears off if you try to get ahead of me. Now hook up the battery or your power source. You should be able to hear a tone coming from the headphones. What we have built with the addition of the muting circuit is an audio oscillator. Pretty loud isn't it?

This next step must be done carefully. Find a short piece of wire about 2 cm long. Use it to connect pin 3 of J3 to pin 1 of J3. This in effect acts like a straight key and shorts the cathode of D5 to ground. This turns on the muting circuit and the tone in the earphones should stop. If it does you are home free. If it does not, then go back and check the parts that you just installed. Hopefully you did not zap the JFET with a static discharge. Check the underside of the board and look for bad solder connections, shorts, etc.

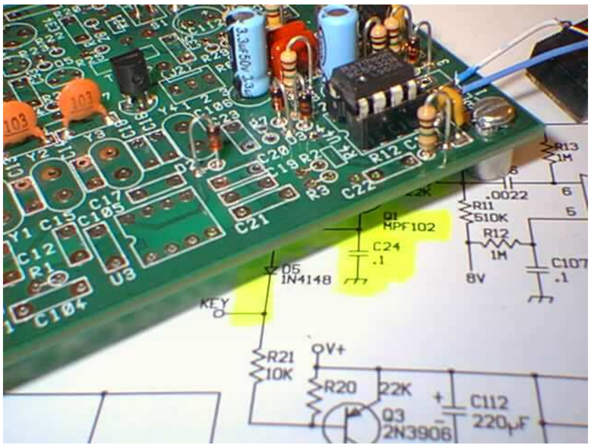


D4 in place.

OK, that again went pretty quick. Hook up headphones and battery and this time there should not be any tone coming from the headphones. Use your DMM probe and touch the pad just to the left of C22. This should cause a hum to be heard faintly in the headphones. Once again you are ever closer to the finish line. The PC board is looking pretty crowded in the lower right hand corner isn't it. And you did all that work yourself.

Audio Preamp Circuit

Moving right along we are now going to wire up the other half of the 5532 IC. This part of the receiver amplifies the audio generated by the U3 IC acting as a detector.



Current state of board.

Part	ID markings	Value	Coordinates
D4	(1N4148)	1N4148 Si diode	(7.6, 1.8)
D3	(1N4148)	1N4148 Si diode	(7.3, 1.8)
R7	(grn-brn-yel)	510K resistor	(7.1, 1.8)
C23	(151)	150pF ceramic cap	(6.8, 1.8)
C22	(151)	150pF ceramic cap	(8.2, 0.3)

Receiver Detector Circuit

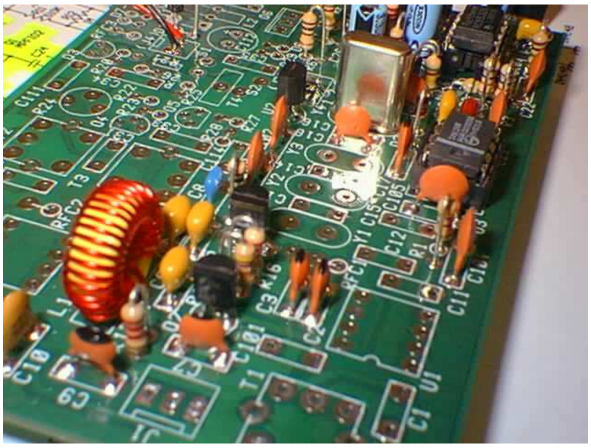
This section is the point in the receiver where the signal coming from the IF crystal filter (crystals Y1-Y3 and associated components is turned into audio tones. The NE612A (or its equivalent) has an internal oscillator which uses C17, C18, and Y4 to create a RF signal which is mixed with the incoming signal from pins 1 and 2 and the difference is output on pins 4 and 5. The output is an audio tone dependent upon how far from the Y4 frequency the incoming signal is. Read the ARRL Handbook on mixers and IF stuff. I'll come back later with specific pages and sections if you need me to do it for you. OK, so let's start building.

Part	ID markings	Value	Coordinates
R4	(grn-brn-yel)	510K	(7.7, 1.0)
R2	(brn-blk-org)	10K	(7.2, 0.9)
R3	(brn-blk-org)	10K	(7.2, 0.5)
C20	(104)	0.1uF mono cap	(6.5, 1.3)
C19	(333)	0.033uF mono cap	(6.5, 1.0)
C21	(103)	0.01uF ceramic	(6.5, 0.5)
C17	(68)	68pF ceramic	(5.5, 1.6)
C16	shorting wire		
C18	(151)	150pF ceramic	(5.5, 2.3)
U3	8-pin IC socket at U3		(5.3, 1.0)
Y4	7.680 crystal		(6.2, 2.3)

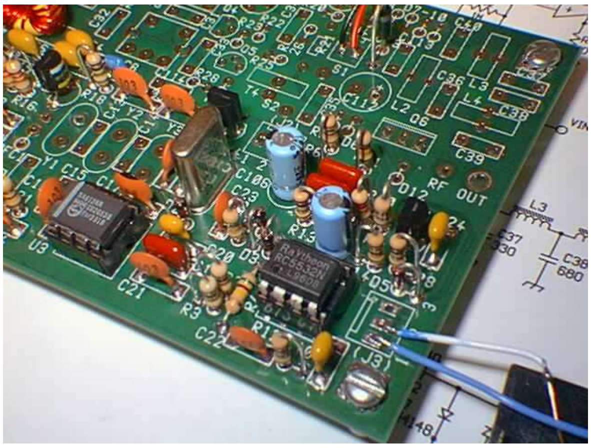
If you want to get the best IF strip you can get from this kit then you need to build a series mode oscillator and measure the output frequency for each crystal. I'm going to build this rig without doing the matching.

Part	ID markings	Value	Coordinates
R1	(yel-vio-brn)	470 ohm	(3.7, 0.8)
C105	(103)	0.01uF ceramic	(4.6, 1.0)
C104	(103)	0.01uF ceramic	(4.0, 0.5)
U3	NE612A or NE602A IC		

OK, ready to test again. Hook up headphones and battery. You should hear a slightly louder hiss that in the previous steps. This is caused by the internal thermal noise generated by the NE612A. If you touch the top lead of R1 with the DMM probe you will hear the noise level increase further, especially if you hold your finger on the tip at the same time. This proves the circuit is working. Congratulations again. You are now just about at the expert level of building.



Current state of board.

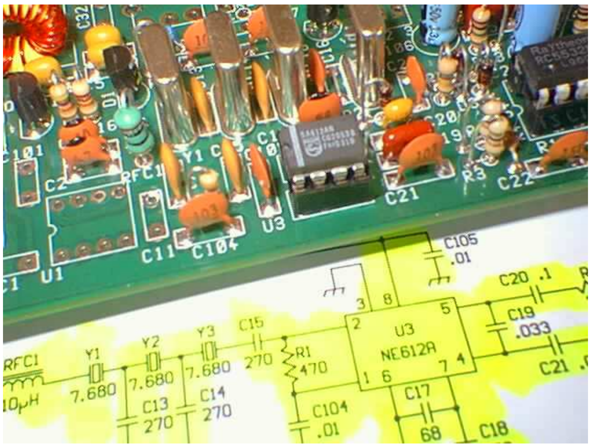


Current state of board.

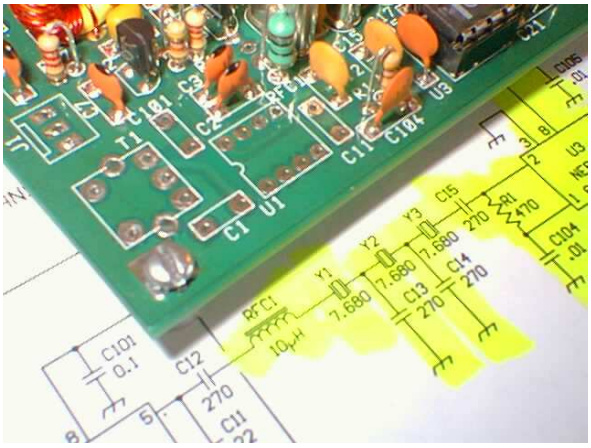
Receiver IF Crystal Filter Circuit

In this section we will be wiring up the 7.680MHz IF filter. There are not all that many parts.

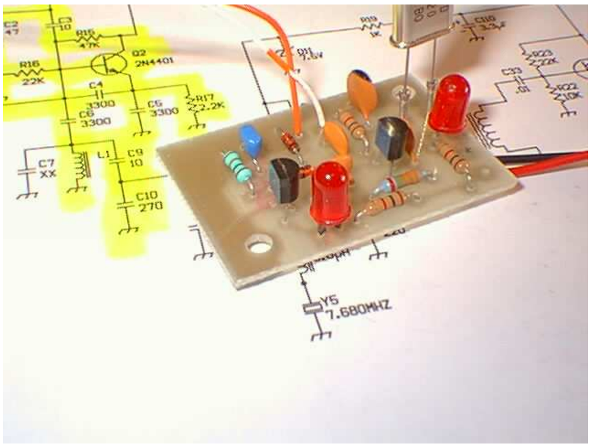
Part	ID markings	Value	Coordinates
C15	(271)	270pF ceramic	(4.2, 1.2)
Y3	7.680MHz crystal		(5.2, 2.3)
C14	(271)	270pF ceramic	(4.7, 2.3)
Y2	7.680MHz crystal		(4.4, 2.3)
C13	(271)	270pF ceramic	(4.1, 2.3)
Y1	7.680MHz crystal		(3.6, 2.3)
RFC1	(brn-blk-blk)	10uH choke	(3.2, 1.9)
C12	(271)	270pF ceramic	(3.5, 1.3)



Current state of board.



Current state of board.



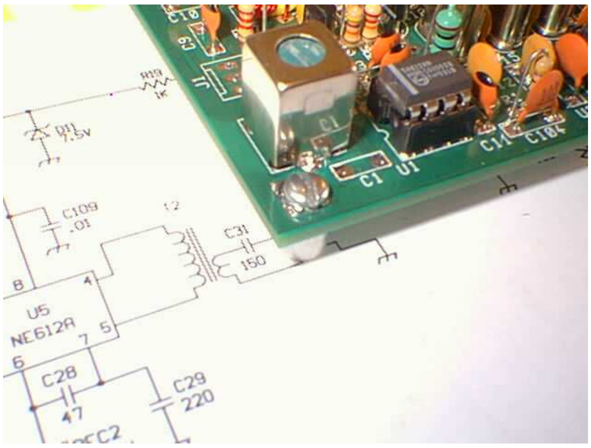
Crystal Checker.

To test this installation, once again hook up the earphones and battery and listen and you should be hearing the hissing type noise that you heard in previous steps. Not much else we can do here to check this out unless you have a crystal tester or crystal oscillator that you can take the remaining 7.680MHz crystal and place in the tester and power it up. WARNING. Do not put the crystal tester too close to the receiver in the SW-30+. The receiver is running wide open, i.e. at maximum gain and is very very sensitive. You should be able to hear the crystal oscillator when you turn it on. The tone is dependent upon the type of oscillator circuit you have for the crystal tester and how close the frequency is to the frequency of the crystal in the detector circuit or crystal Y4. This concludes this test. Power off the rig and remove the earphones and proceed to the next wiring stage.

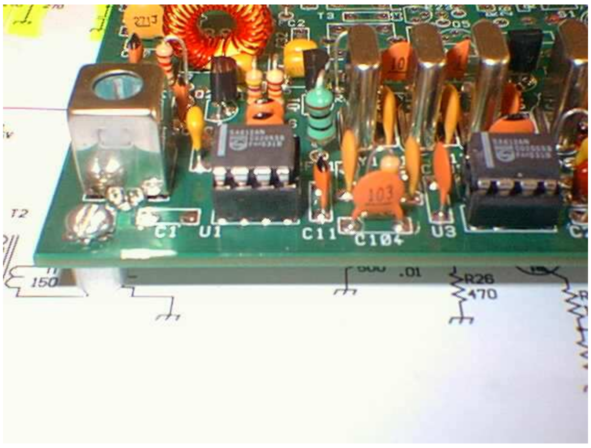
First Receiver Mixer Circuit

This wiring phase involves wiring the first mixer U1. After this you will pretty much have a working receiver.

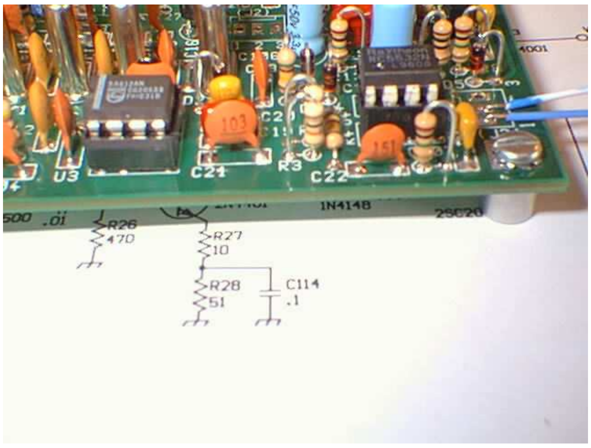
Part	ID markings	Value	Coordinates
C11	(22)	22pF ceramic	(3.2, 0.9)
U1	IC socket		
C101	(104)	0.1uF mono	(2.5, 1.2)
T1	42IF123	10.7MHz IF transformer	



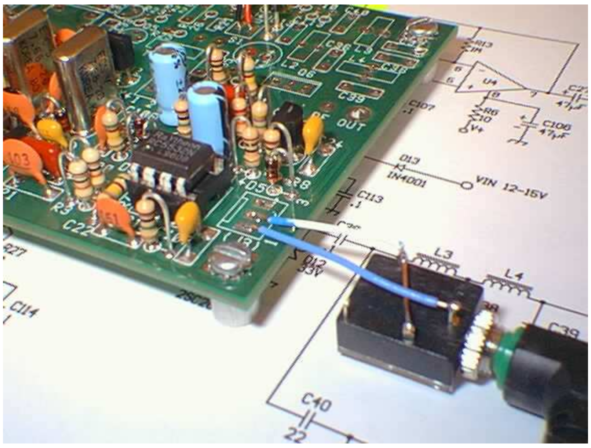
Current state of board.



Current state of board.



Current state of board.



Current state of board.

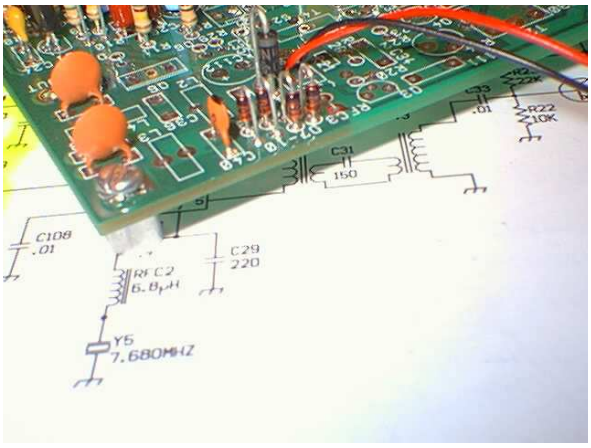
OK, now we have the receiver wired up. Connect earphones and battery and you should hear hiss/noise in earphones. Touch either pad where C1 would have gone next to T1. The noise level should increase. Also touch the top pad of J1. If you have a short piece of small wire, touch it to the top pad of J1 and the noise level should increase a bit more. Take a non-magnetic screwdriver or some plastic piece and adjust the tuning slug in T1 for maximum noise level with the wire at J1.

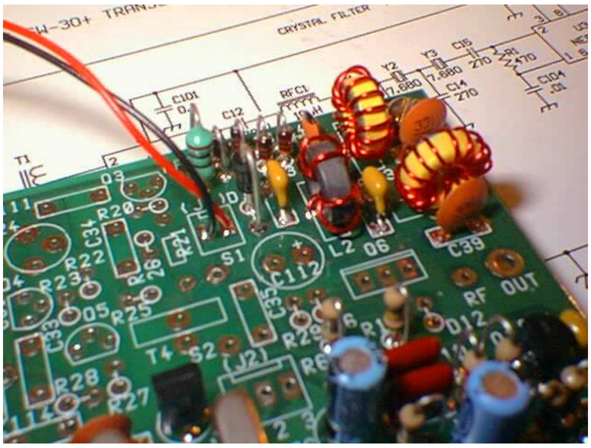
If you want to play just a little bit more here, you could power off the rig and temporarily solder leads from J2 to the tuning pot. See Dave's manual page 14 on where the 100K linear taper pot goes. But you may want to wait until the next stage to do this. It's all up to you and just how much you want to play around.

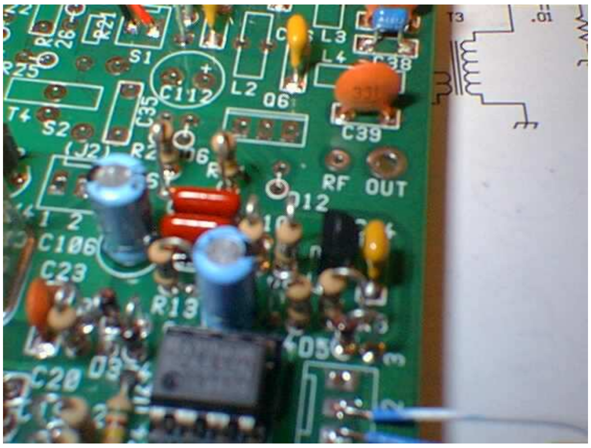
Output Low Pass Filter

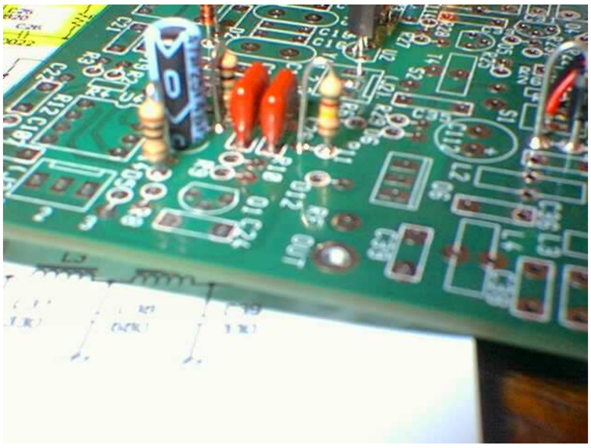
The following components will allow us to connect the receiver to an antenna and further check it out. It will also get us one step closer to getting the transmitter to work as this section has components common to both the receiver and the transmitter. I have put these in order that will make it easy to work in a small area and get the smaller/lower parts in first to avoid working crowded places.

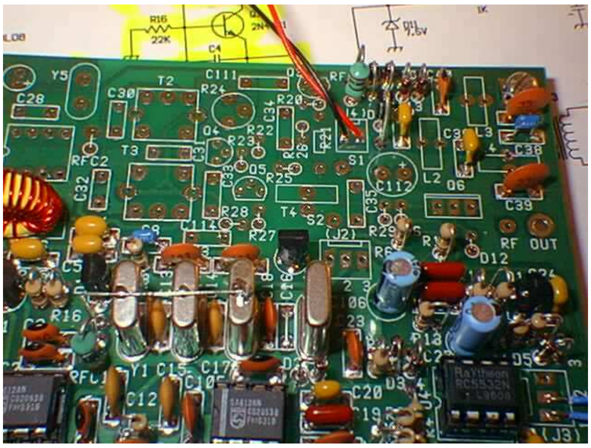
Part	ID markings	Value	Coordinates
C37	(331)	330pF ceramic	(9.8, 6.0)
C38	(681)	680pF mono	(9.8,5.6)
C39	(331)	330pF ceramic	(9.8, 4.6)
C40	(22)	22pF ceramic	(8.4, 6.5)
D10	(1N4148)	1N4148 Si diode	(8.0, 6.5)
D9	(1N4148)	1N4148 Si diode	(7.7, 6.5)
D8	(1N4148)	1N4148 Si diode	(7.5, 6.5)
D7	(1N4148)	1N4148 Si diode	(7.2, 6.5)
C113	(104)	0.1uF mono cap	(7.6, 5.8)
RFC3	(brn-blk-blk)	10uH choke	(6.8,5.8)
C36	(104)	0.1uF mono cap	(8.7, 5.3)
L2	(FT37-43 toroid)		(8.2, 5.5)
L3	(T37-6 toroid)		(9.0, 6.5)
L4	(T37-6 toroid)		(9.4, 6.5)











OK, wiring is done. First test. Using ohmeter in continuity check (the DMM makes a beeping sound when you measure a very low resistance or short) put one lead at pad labeled RF (the smaller pad at RF OUT) at (9.2, 4.0). Using the other lead see if you get continuity to each of the pads at L3 and L4. This insures that the leads are cleaned and soldered properly.

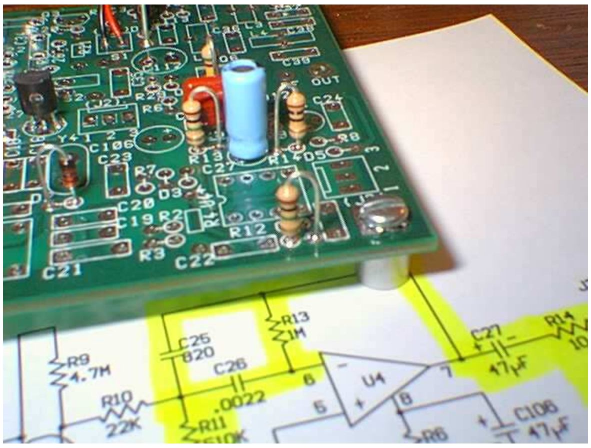
Second check. Test continuity from the top lead of RFC3 to center pad of J1.

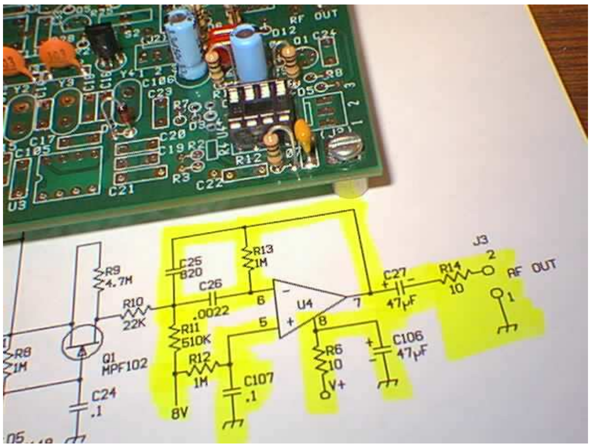
And an optional more advanced test. Take a small clip lead and bend it and friction fit it into the two top pads of J1. What this does is feed input of receiver direct to RFC3 and from there to the antenna connection. Take a small insulated wire of length one meter or so with a small part of the end stripped and tinned and insert it into the RF pad just above the right hand center of the board near C39. This wire will be a temporary antenna. I would not solder it, just let it sit in the hole and make contact. Now power up the SW-30+ and connect earphones. You should be hearing some atmospheric noise, especially if there are thunderstorms somewhere in the US. Removing wire should cause a drop in noise level in the headphones. Tune T1 for maximum noise level with the antenna attached. And if you have the tuning pot (100K) hooked up you just might be able to tune around and hear signals. The 30 meter band for hams is from 10.100MHz to 10.150MHz and if you had a frequency counter and were able to set the VFO range you can hear in this range. If not, then we will have to solve the problem another way later on.

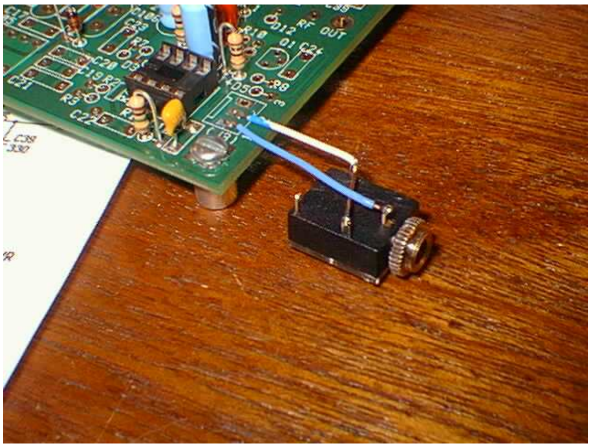
Keying Circuit

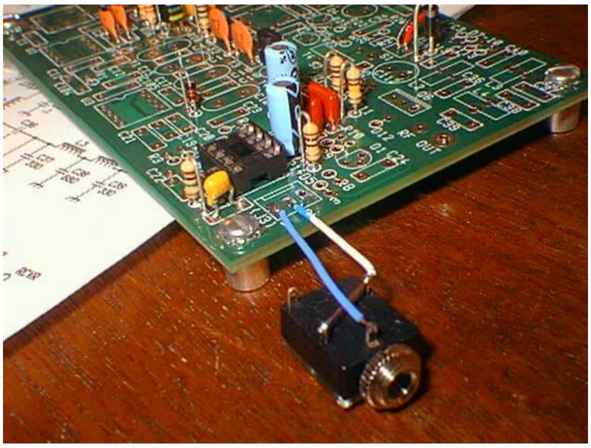
In this section we are going to wire up the parts that cause the transceiver to switch from receive to transmit whenever a key or keyer is used. The receiver never really completely turns off during transmit, but the muting circuit prevents the receiver front end output from getting to the final audio amplifier and only just a small fraction of the energy passes and provides us with a sidetone.

Part	ID markings	Value	Coordinates
Q3	(2N3906)	2N3906 PNP	(6.3, 6.5)
C111	(104)	0.1uF mono cap	(5.5, 6.6)
R20	(red-red-org)	22K resistor	(6.2, 6.2)
R21	(brn-blk-org)	10K resistor	(6.4, 5.5)
C112	(220uF)	220uF elect.	(7.5, 5.0)
C110	(47uF)	47uF electr.	(1.5, 6.5)
R19	(brn-blk-red)	1K resistor	(1.0, 6.0)
D11	(1N5236B)	7.5V Zener	(0.5, 6.0)
C109	(103)	0.01uF ceramic	
C108	(103)	0.01uF ceramic	









OK, testing time again. Hook up headphones and battery. You'll hear some background hissing type noise in the headphones as in previous steps.

Test 1: Short pin 3 of J3 to pin 1 of J3. This simulates a key down condition. You should hear the audio in the headphones mute, i.e. disappear or get real quiet.

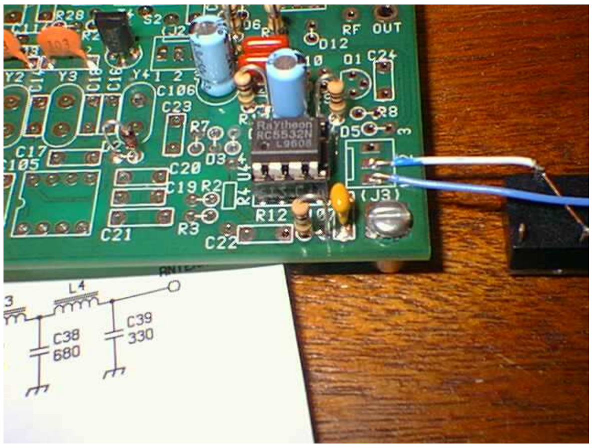
Test 2: Take DMM and measure voltage at pin 8 of U5. With key up condition you get almost 0.0V. With the key down condition, short 3 to 1 of J3, the voltage should exceed 7V. I got 10.87V and this because we don't have an IC in yet. When you release the key down condition you will note that the voltage does not immediately decrease to 0V, but takes some time. This is due to no high current drain on the caps in the circuit as we don't have an IC online at this time.

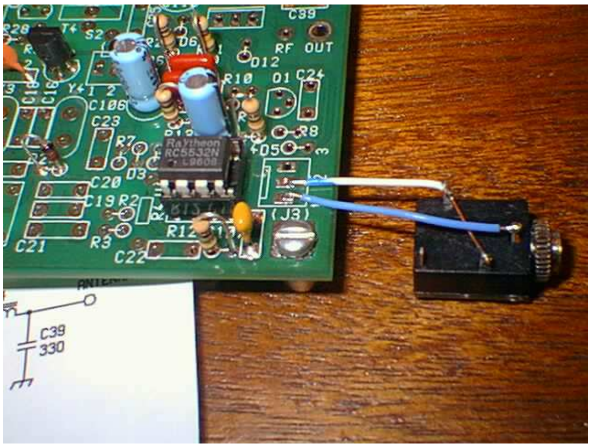
If the above two tests pass, then the circuit is working properly. Congratulations again.

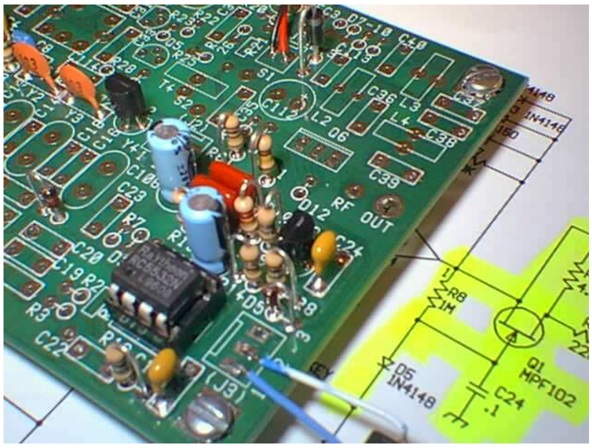
First Transmitter Mixer Circuit

In this section we will start from the front of the transmitter section and work our way to the final power amplifier (PA).

Part	ID markings	Value	Coordinates
C28	(47)	47pF ceramic	(1.8, 6.0)
C29	(221)	220pF ceramic	(0.7, 0.7)
RFC2	(blu-gry-gld)	6.8uH	(2.7, 5.5)
Y5	7.680MHz crystal		(2.5, 6.5)
U5	IC socket and IC		(2.0, 5.0)





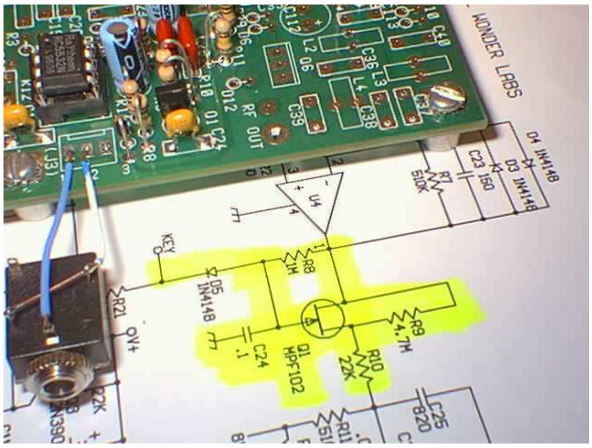


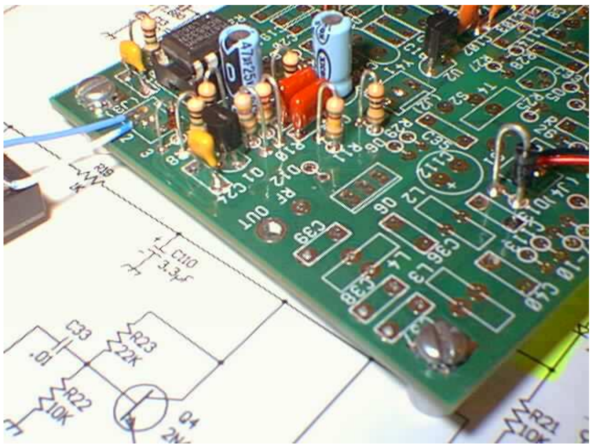
OK, back to testing again. Hook up battery and headphones. Now, redo the key down condition, i.e. short pin 3 of J3 to pin 1 of J3. This time you should hear a very weak tone in the headphones. This means that the VFO frequency is combined with 7.680MHz to generate a 10.1XX MHz RF signal. You first part of your transmitter is now working. If you want to play around just a second like a kid that I know you are, try sending some code using the wire that you have for shorting pin 3 to pin 1.

OK, that's enough, back to work. Remove the battery and the headphones and get back to soldering. :-) You are so good.

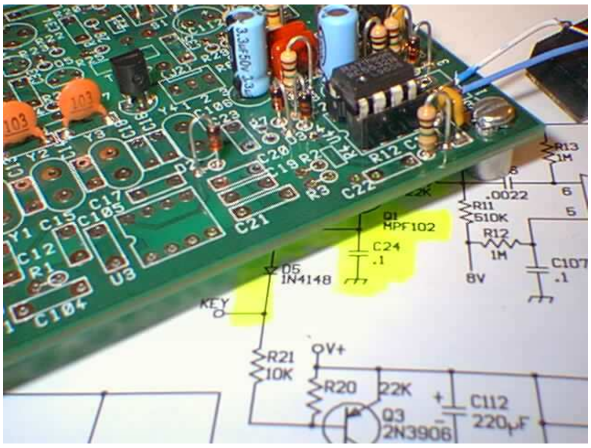
Transmitter Bandpass Circuit

Part	ID markings	Value	Coordinates
T2	(42IF123)	10.7MHz transformer	(3.8, 6.0)
C31	(151)	150pF ceramic	(3.0, 5.2)
T3	(42IF123)	10.7MHz IF transformer	(3.8, 4.5)
C33	(103)	0.01uF ceramic	(4.5, 4.5)





OK, back to testing again. Power up and put on the headphones. Once again, key down and you should hear the tone in the phones. You might adjust T2 and T3 for a maximum, but it won't make too much difference at this time. You're done for this test. Unhook battery and phones and continue wiring.

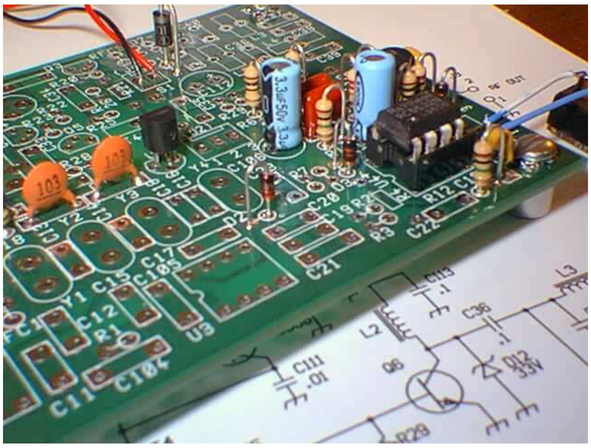


Transmitter Buffer/RF Amplifier Circuit

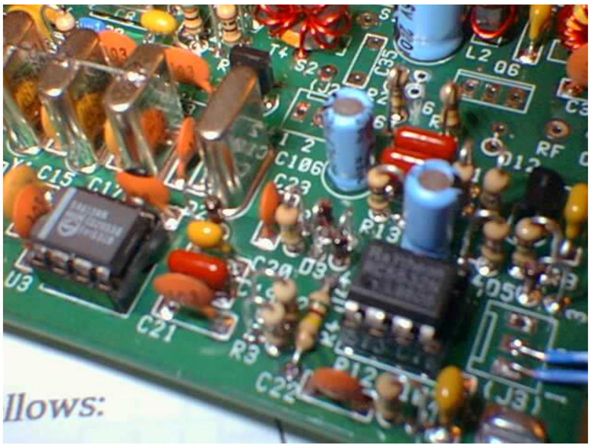
Part	ID markings	Value	Coordinates
Q4	(2N4401)	2N4401 NPN	(4.7, 5.4)
R23	(red-red-org)	22K	(5.0, 5.1)
R22	(brn-blk-org)	10K	(5.1, 5.2)
R24	500 ohm variable resistor		(5.0, 6.1)
C34	(103)	0.01uF ceramic	(5.8, 5.8)

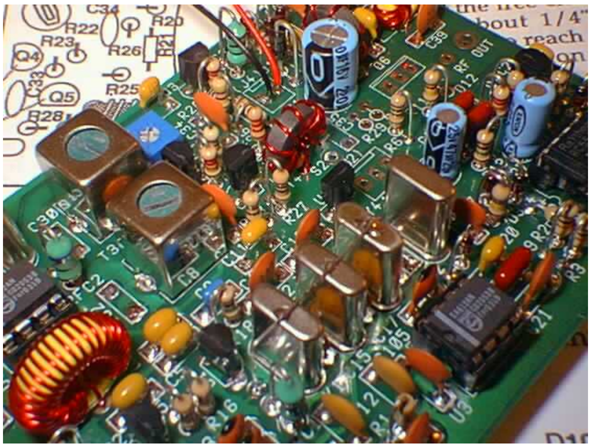
Carefully turn the adjustment of R24 (the variable resistor) fully counter clockwise (CCW). This makes the output from this circuit a minimum.

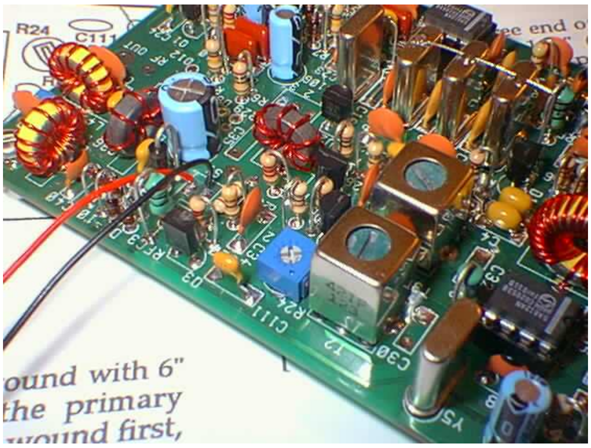
OK, headphones back on and battery connected. Key down and you should hear a tone. No need to mess with the variable as it won't do anything to effect the volume of the tone at this time.



Part	ID markings	Value	Coordinates
R26	(yel-vio-brn)	470 ohm	(6.1, 5.6)
R25	(red-red-red)	2.2K	(6.1, 5.6)
Q5	(2N4401)	2N4401	(5.2, 4.6)
C114	(104)	001uF mono	(4.6, 3.8)
R28	(grn-brn-blk)	51 ohm	(5.0, 4.0)
R27	(brn-blk-blk)	10 ohm	(5.3, 4.1)
T4	(FT37-43)	10T:1T	(6.2, 4.4)
C35	(103)	0.01uF ceramic	(7.0, 4.4)







Once it again it is time to test our wiring. Power up and key down and you should still hear a tone. That's all we can do at this time unless you have an RF probe or O'scope and look for RF out on the secondary winding or at C35.

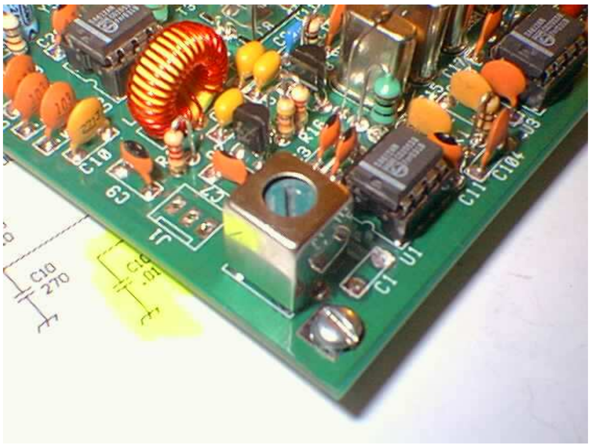
Transmitter Power Amplifier Circuit

This is it. You are four parts away from being done with the on board wiring and soldering. And I won't show the K7QO coordinates for these parts as it is obvious at this point where you are and where they go.

Part	ID markings	Value	Coordinates
R29	(grn-brn-blk)	51 ohm resistor	
D6	(1N4148)	1N4148 Si Diode	
D12	(1N5257B	33V Zener diode, the last one on the yel	
Q6	(2SC2078 or C2166)	Final PA transistor.	

Be sure to read the manual on putting the 2SC2078 or C2166 PA transistor in. It mounts straight up on the board and the metal tab is towards the back of the board. See the outline on the PC board and double check. Look at the pictures before installing if you are not sure.

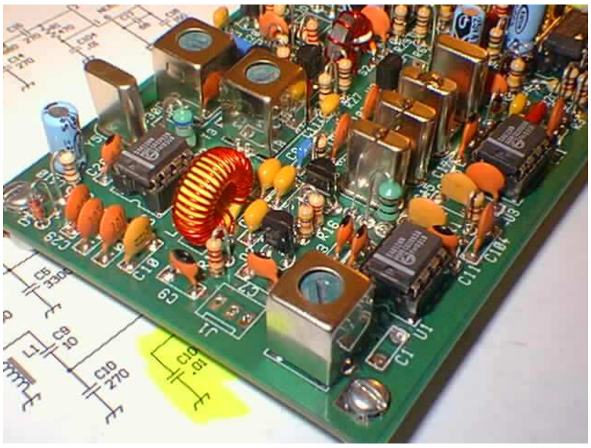
Well that's all there is for now. PLEASE. Do not power up at this time. You can ruin the final PA if you operate this transceiver without a dummy load or a good antenna for 30 meters attached. I'll add a section here pretty quick as soon as my paint dries on my homebrew case and I'll show you how to install into a case. If you bought the case and pots from Dave, NN1G, then follow his instructions on installing the transceiver into his case. It is a good one as I own two myself.



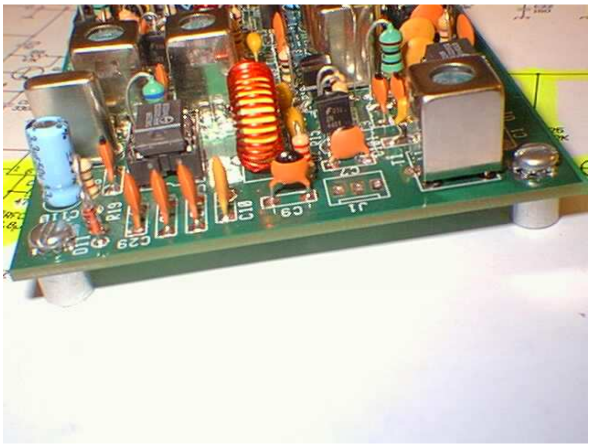
Final board completion.

Final PA Transistor.

Final board completion.



Final board completion.



Final board completion.

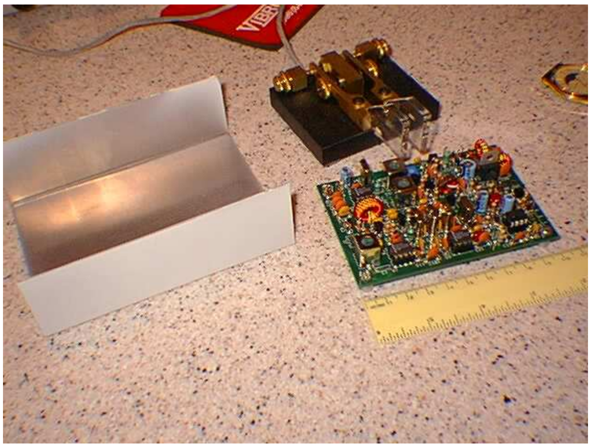
Congratulations are in order for your successful completion of building the board. Now we move on to putting it into a case with antenna and power connectors.

K7QO's Case

This is some photos to illustrate what I do to put a rig into a case. I use a Harbor Freight shear/brake/roller that will handle up to 12" sized metal sheets. I get my 0.040" thick aluminum sheet as one of the surplus metal places in Phoenix about 130 km away from Prescott. I go down there (meaning that it is south of me) once a month for the AZ sQRPion QRP group meetings, so if I need something I can drop by their and pick it up in the same trip.

I measure the size of the rig and figure on how much room I need for stuff on the front and rear panels, height of the panels, etc. Then on a blank sheet of paper lay out the total size of a sheet of metal and where it will be bent and drilled. I cut it out, bend it and double check. Then I drill the holes and using a hand tapered reamer enlarge the holes for the control sizes and start putting things together. I hope from the photos you can see how I do it. Ask questions if you have any and I'll try to add stuff to the photos and what I'm doing. The case costs me about $1.00 US to make in materials and paint. I get some OEM Krylon paint from Wal-Mart for less than a buck a can. Nice paint. I cook it at 170 degrees F in an electric oven with all the exhaust fans in the house going. Take Phyllis to dinner and a movie. I have yet to label the case with press-on lettering in these photos. Maybe later if I need to take it out for a drive and show it off. :-) Otherwise it will probably stay the way it is.

I'm now in trouble on this rig. I put it on the air. The receiver is about 20 dB more sensitive than the old SW-30+ and I now have to reverse engineer both and test them to see where the difference(s) are. More on this in a few weeks (June 2001 timeframe most likely) or later due to the real MH101 project work.

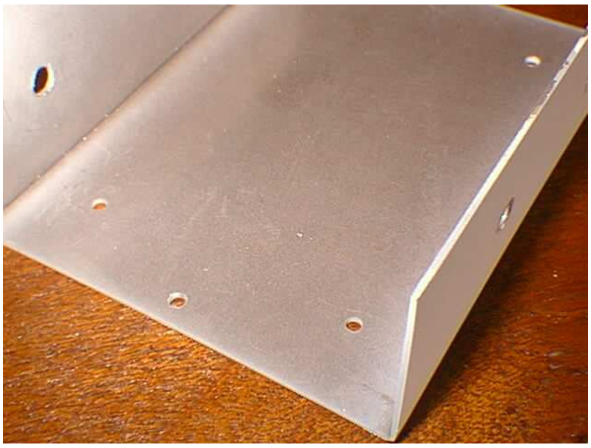


Bottom half of case.

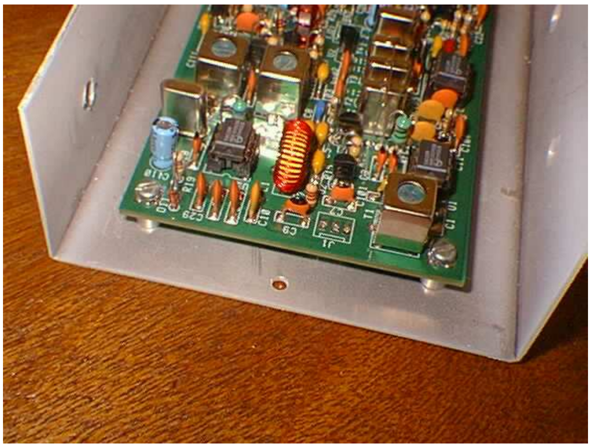
Bottom half of case.

Check for fit.

Different View.

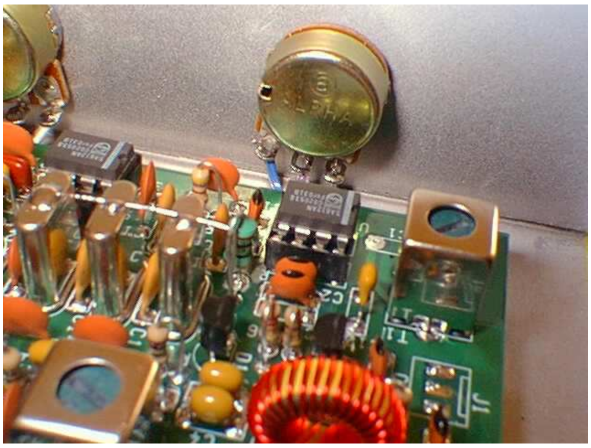


Holes for Controls and Connectors.

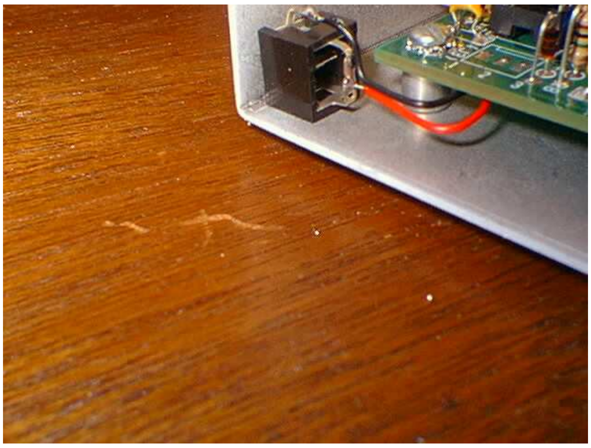


Check Standoff Holes.

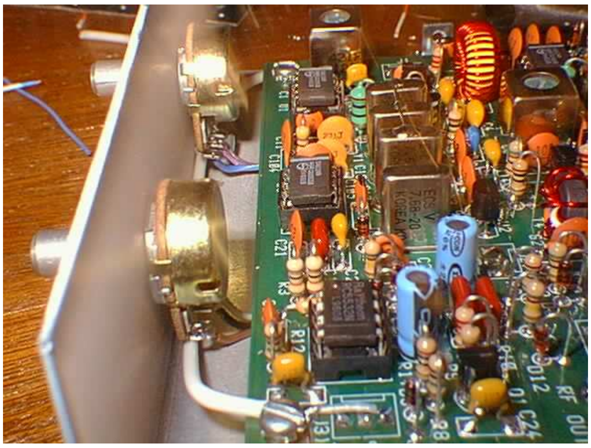
Tuning Control Pot.



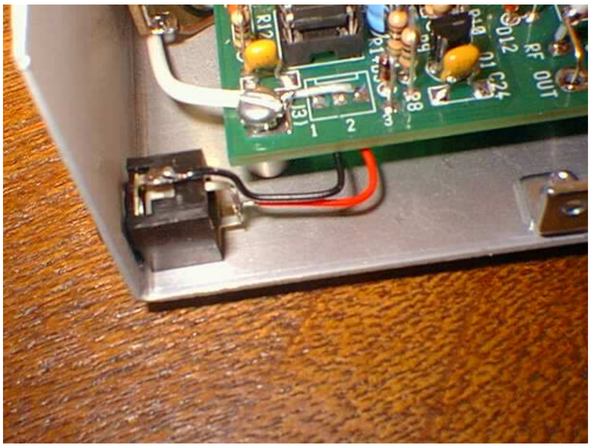
Gain Control Pot.



Earphone Jack.



Teflon RG–316U coax.

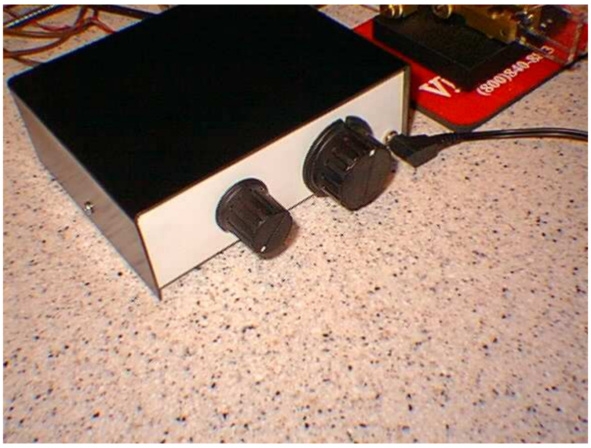


Teflon RG–316U coax.

Front View.

Front View.

Completed Station.



Completed Station.

That's it grasshopper. Your turn to build one.