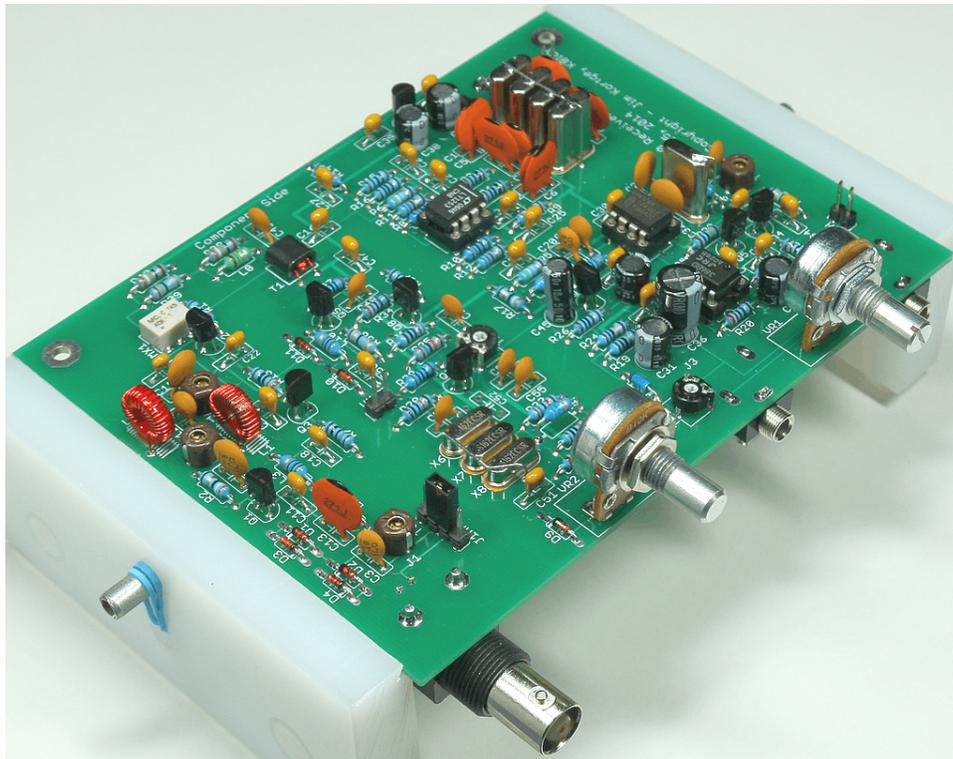


SS-40HT Receiver Assembly Manual for Printed Circuit Board V1.0HT (Manual V1.1F)

Copyright – 2014 Forward

**Jim Kortge, K8IQY
&
The 4 State QRP Group**



The First SS-40HT Receiver – Completed 03/29/2014

Introduced at Ozarkcon 2014

Introduction

Thank you for purchasing a SS-40HT Receiver. We hope you will enjoy building it and using it to listen to the 40-Meter band. This receiver concept was first conceived several years ago as part of a 2-Band SSB/CW transceiver. That project is “ongoing”. In the meantime, it seemed like a good idea to create a stand alone receiver which could be used with the 4 State QRP group NS-40 Transmitter (or other 40-Meter transmitter) and the K8IQY MagicBox, (also available from 4 State) to assemble a complete station.

This newest version of the SS-40 Receiver has header pins added so that it can be easily interfaced to a newly designed SS-40 Transmitter Board to create a complete 40-meter transceiver. In addition, an attenuator has been added at the output of the VXO so that the drive level into the mixer can be easily measured.

This receiver is a superhetrodyne design employing a very Stable VXO for frequency control. It has no perceptible drift, even from a cold start. It is also very Sensitive with an MDS of better than -130 dBm. Since it operates on 40-Meters, it was dubbed the SS-40HT Receiver. Other features of the receiver include room filling volume when using a speaker, AGC to keep strong signals at bay, and a 4-pole, 500 Hz Butterworth crystal filter. It tunes 25-30 KHz of the band including the two QRP watering holes at 7030 and 7040 KHz.

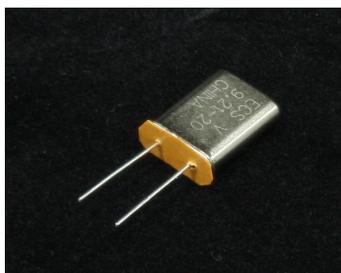
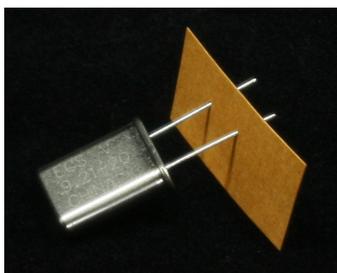
Printed Circuit Board construction is used with wide part spacing. All of the parts have leads except for the mixer, which is a large footprint surface mount unit. Only 2 toroids with a single winding each are used, minimizing frustration for new builders. All controls and jacks are PCB mounted, eliminating control wiring. The receiver can be constructed by beginners as well as experienced builders. Construction time is approximately 20 hours, depending on experience level. Building follows the schematic diagram layout, allowing sections to be completed and tested when possible before moving on.

First Steps

Before getting started with building the receiver, take some time to organize and familiarize yourself with the parts provided and check them against the Bill of Material. Building over a cookie sheet is recommended to minimize parts being lost. If parts are missing in your kit, send an email to Garrett Coleman, colemangarrett1@gmail.com. He will promptly provide replacements.

A schematic file in PDF format is available on the 4SQRP web site as part of documentation package. It is highly recommended to print a couple of copies of the schematic at 11 X 17 inch format at your local USP Store, Staples, etc. As you build, use a highlighter to mark off parts that have been soldered onto the PCB on one copy. When you think you are done, you can check that copy to verify that all of the parts have been installed.

When installing crystals, put insulators under them to space them off of the PCB. Kraft paper from a grocery bag or the envelope from this kit works well. To make a spacer, poke two holes at the width of the crystal leads through a 0.5 X 0.75 piece of paper with an awl. Slide the spacer on the leads and trim away the excess paper with scissors. Examples are shown in the photos below.



Let's build this kit!

Step 1 – VXO Tuning Pot VR2

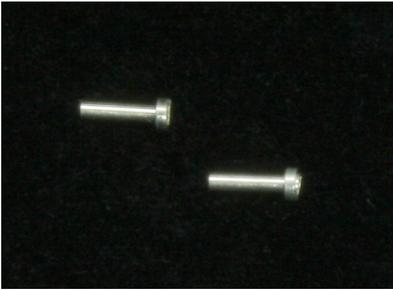
This component is mounted on the top of the PCB first, as it requires some extra steps, since it is not designed for PCB mounting. The previous version of this potentiometer was designed for PCB mounting, but could only be obtained as an audio taper potentiometer, which made tuning the higher end of the VXO range more difficult than desired. VR2 is now a 10K linear taper potentiometer and is used with a shunting resistor to optimize the tuning range versus potentiometer rotation.

Select two header pins shown in the photo below.



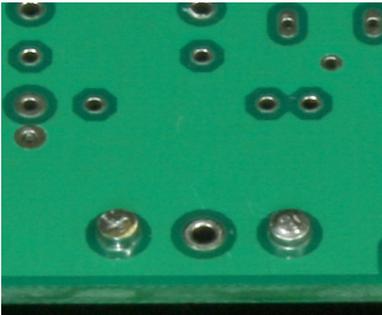
Unmodified Header Pins

Using a sharp pair of diagonal cutters, remove the short part of the pin below the collar area as show in the next photo.



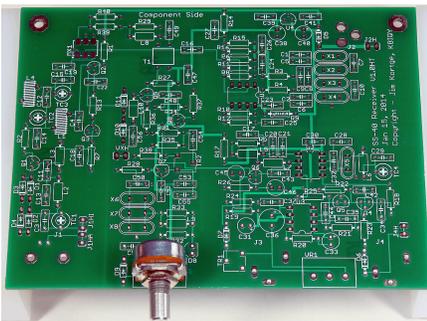
Modified Header Pins

Insert these pins into the outer holes of the PCB, from the bottom side, where the tuning potentiometer VR2 is mounted and carefully solder the pins to secure them. If you have liquid flux available, use a small amount of that at each pin location to make the solder flow better. The bottom of the PCB will look like the next photo.

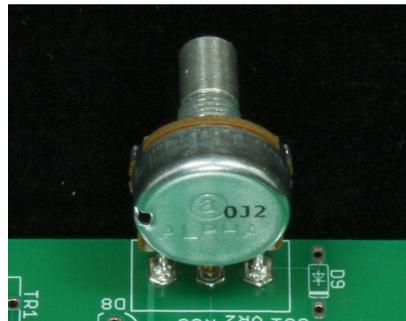


Modified Pins Soldered to PCB

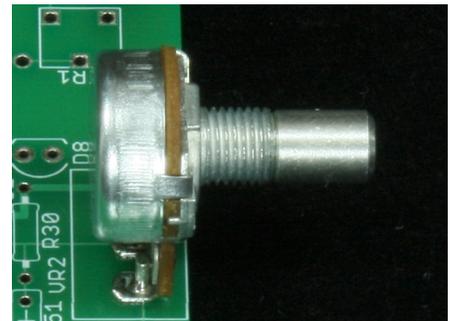
Turn the PCB over and place the 10K linear potentiometer solder lugs **over** these pins with the shaft pointing away from the board. If you have liquid flux available, use a small amount of that at each pin location to make the solder flow better. Carefully solder one of the potentiometer lugs to its corresponding pin while holding the potentiometer down against the PCB surface and pushed gently away from the edge of the PCB so that the potentiometer is aligned as far rearward as is will go. If you are happy with the alignment, solder the other lug, or reheat the first lug to more favorably realign the potentiometer. Once the outer potentiometer lugs are soldered, solder the center lug. When all of the potentiometer lugs are soldered, the PCB will look like the following photos.



VR2 Mounted – Front View



VR2 Mounted – Rear View



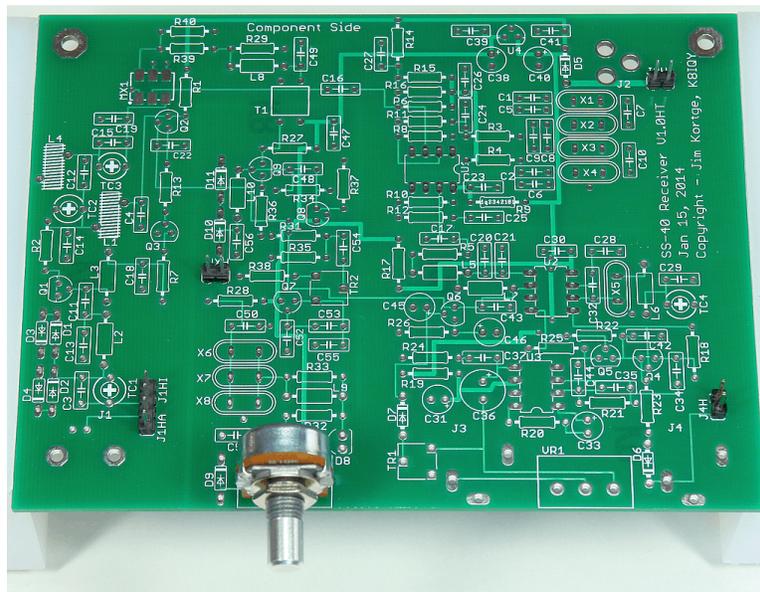
VR2 Mounted – Side View

Testing

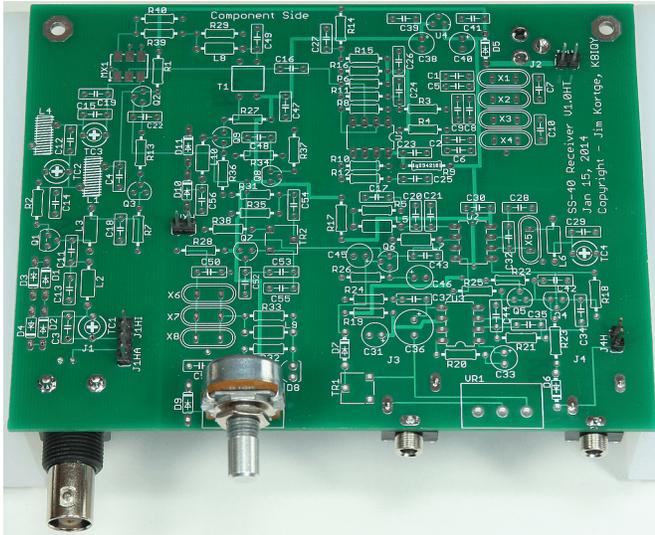
Confirm that the potentiometer is electrically connected to the PCB by measuring its resistance. Place one lead of an Ohm meter at the right mounting hole for resistor R30 and the other lead at the anode (arrow) of diode D9. You should get a reading of $\sim 10\text{K}$ Ohms. Move the Ohm meter lead from D9 to the left mounting hole for resistor R32. Confirm that the resistance varies from ~ 0 to $\sim 10\text{K}$ Ohms as the shaft of the potentiometer is rotated over its range.

Step 2 – Connectors

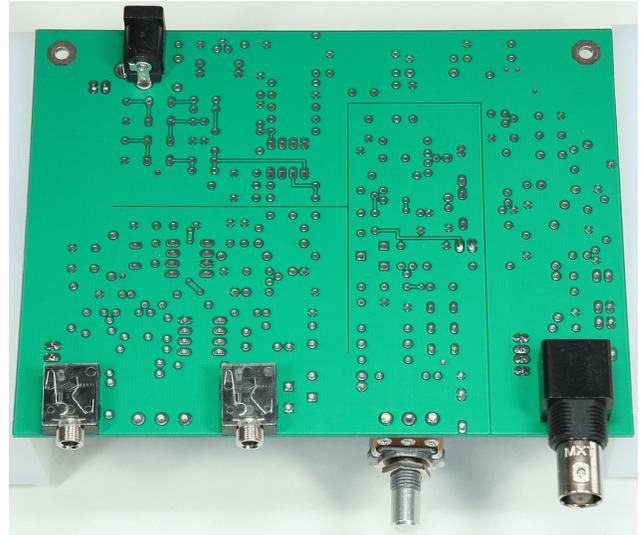
Install a 4-pin header spanning the locations J1HA and J1HI; install 2-pin headers at locations J2H, J4H, and VXH. Insert the shorter end of each header from the component side of the PCB and solder on the bottom side. Highlight them on the schematic when you are done soldering. Your PCB will look like that in the photo below.



Install connectors J1 through J4. They are all **mounted on the bottom** side of the PCB and soldered on the top or component side of the board. These are the only parts that mount on the underside of the PCB; everything else goes on the top. Highlight them on the schematic when you are done soldering. Your PCB will look like that in the photos below.



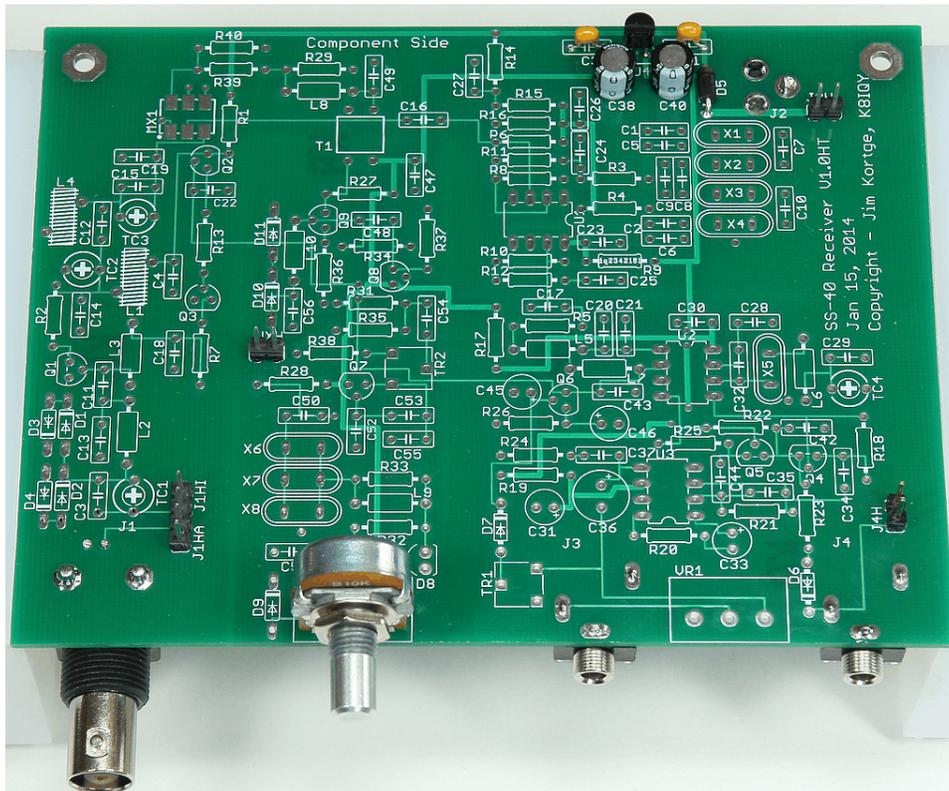
J1-J4 Connectors – Top



J1-J4 Connectors – Bottom

Step 3 - Power Supply Components

Install these components in the order they are called out: D5-1N5817 diode, C41-0.1uF capacitor, C40-22 uF capacitor (make sure that the polarity is correct), U4-78L08 8 volt regulator, C39-0.1 uF capacitor, and C38-22 uF capacitor (make sure that the polarity is correct). As each part is solder into the PCB, mark it off on the schematic with a highlighter. When all of them have been installed, your PCB will look like the photos below.



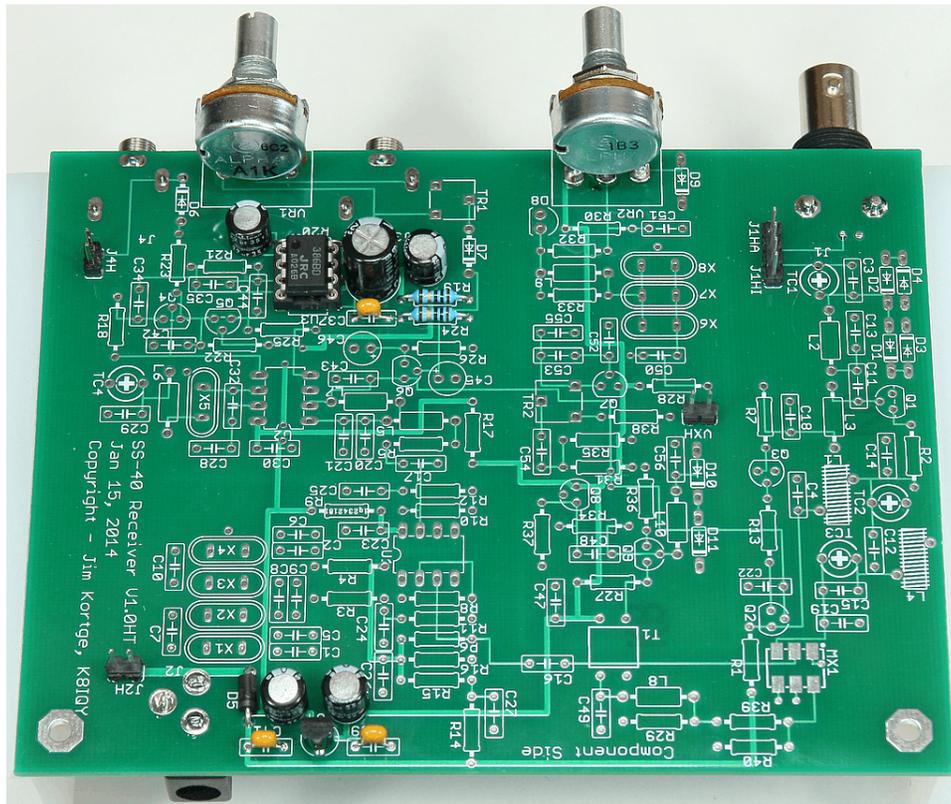
Power Supply Components

Testing

Apply 10-14 volts dc to power supply connector J2. Using a voltmeter, verify that there is approximately 9.5-13.5 volts at the right pad for resistor R9 and 8 volts at the rear pad for capacitor C47. Use one of the two rear mounting holes for the voltmeter ground lead. *(Note: the mating plug for J2 is the supplied 5.5 mm barrel/2.5 mm pin plug.)*

Step 4 - Audio Amplifier Components

Begin this build section by soldering in the 1 K volume control pot, VR1. Mount this pot just as you did for the VR2 tuning pot, using modified header pins. Refer to those photos for installation details. Next, install an 8-pin IC socket at location U3. The notch in the socket should match the outline on the PCB. Solder one lead and verify that the orientation is correct before soldering the remaining leads. Sequentially solder in the following parts: C36-220uF capacitor (make sure that the polarity is correct), C37-0.1uF capacitor, R24-10 Ohm resistor, R20-2.2 K Ohm resistor, C33-22 uF capacitor (make sure that the polarity is correct), C31-22 uF capacitor (make sure that the polarity is correct), and R19-10 Ohm resistor. As each part is solder into the PCB, mark it off on the schematic with a highlighter. When all of them have been installed, your PCB will look like the photo below.



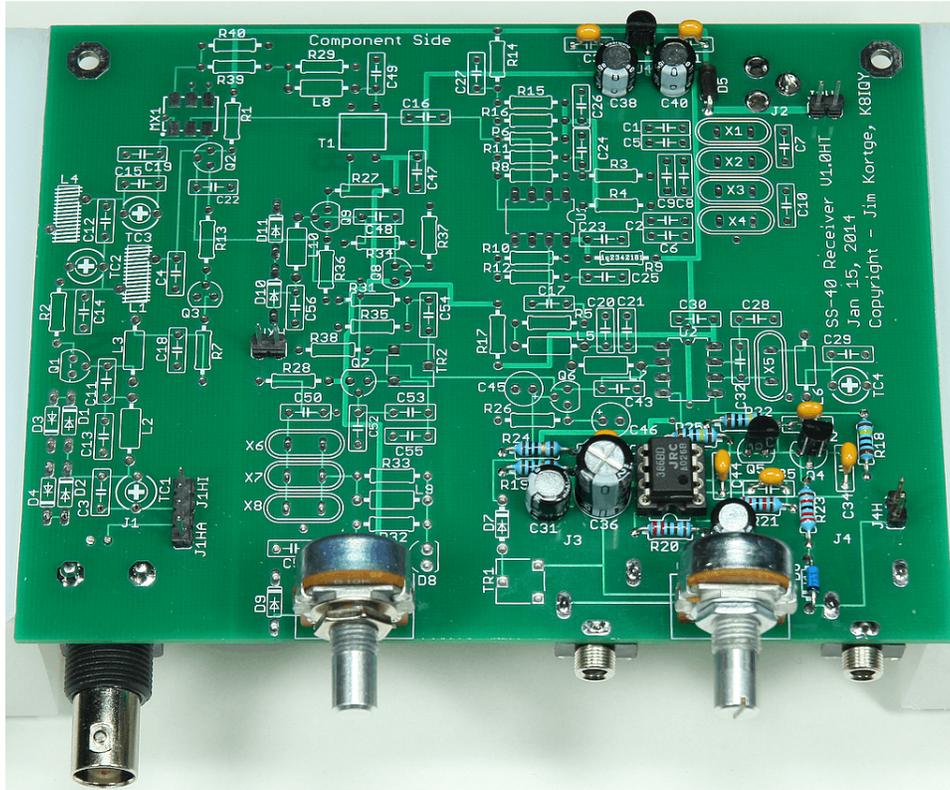
Audio Amplifier Components – Rear

Testing

Carefully bend the leads of the LM386 chip so that the pairs of leads are parallel to each other and perpendicular to the chip body. Install the chip into the U3 socket; please don't force it and break a lead. Plug a speaker or earphones into audio jack J3. Apply 10-14 volts to connector J2. Set volume control VR1 fully clockwise for maximum volume. Touch either pad for resistor R21 with your finger. A raspy 60 Hz signal should be heard. Rotate VR1 counterclockwise to verify that the volume of the signal can be lowered.

Step 5 – Audio Mute Components

Continue by installing the audio mute components consisting of R21-22 K Ohm resistor, C35-0.1 uF capacitor, D6-1N5711 diode, R23-22 K Ohm resistor, C34-0.1 uF capacitor, C42-0.033 uF capacitor, C44-0.1 uF capacitor, R18-47 K Ohm resistor, Q4-J310 transistor (*Note: J310 transistors are static sensitive, so touch the largest metal object near you before picking one up.*), R22-1 M Ohm resistor, Q5-J310 transistor, and R25-47 K Ohm resistor. As each part is solder into the PCB, mark it off on the schematic with a highlighter. When all of them have been installed, your PCB will look like the photo below.



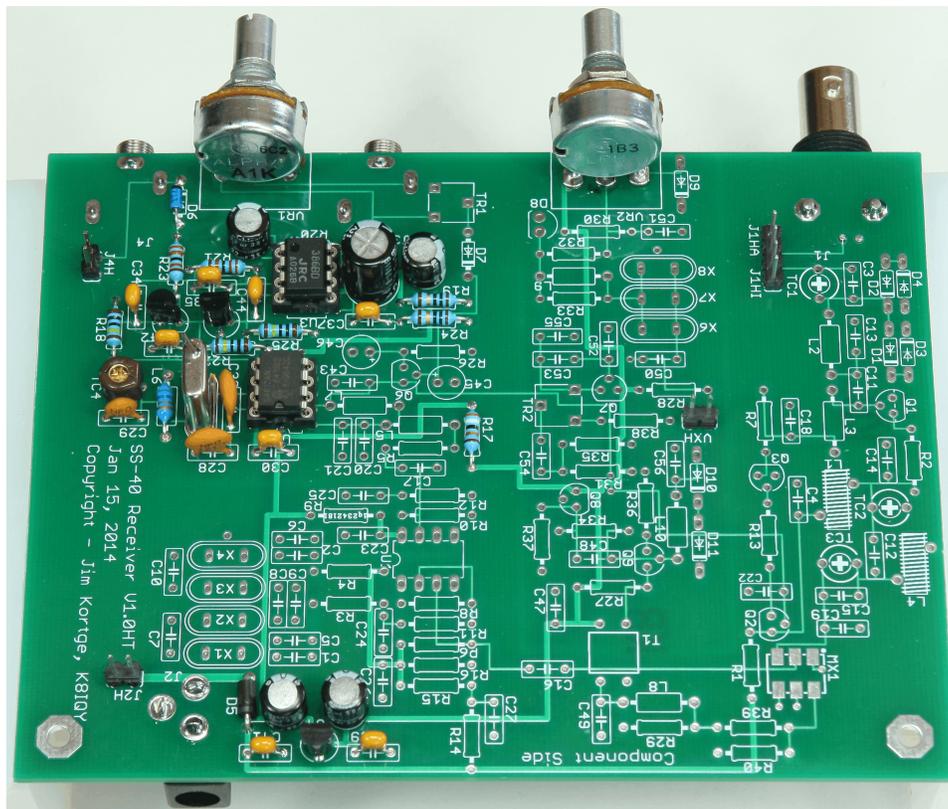
Audio Mute Components

Testing

No testing of this stage can be done now because there is no dc bias on the J310 transistors. We'll test this stage after completing the next step.

Step 6 – Product Detector Components

Install an 8-pin socket at the U2 position. The notch in the socket should match the outline on the PCB. Solder one lead and verify that the orientation is correct before soldering the remaining leads. Install the following components C29-15 pF capacitor, TC4-10-70 pF trimmer capacitor (remove the bends in the leads before inserting it into the PCB and orient it so that the rounded end is grounded), L6-5.6 uH inductor, X5-9.21 MHz crystal (marked "LO", use an insulator under the crystal and ground the crystal case to the pad provided), C28-120 pF capacitor, C32-120 pF capacitor, R17-39 Ohm resistor, and C30-0.1 uF capacitor. As each part is solder into the PCB, mark it off on the schematic with a highlighter. Carefully bend the leads of the SA612 chip so that the pairs of leads are parallel to each other and perpendicular to the chip body. Install the chip into the U2 socket; please don't force it and break a lead. When all of the parts have been installed, your PCB will look like the photo below.



Product Detector Components – Rear

Testing

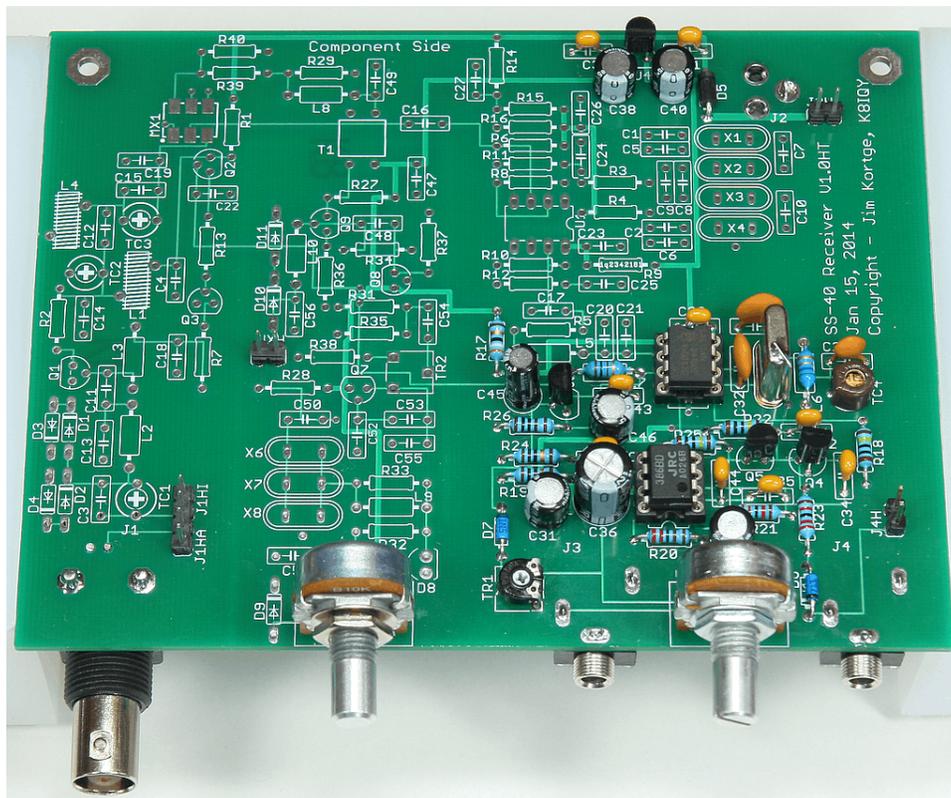
Plug a speaker or earphones into audio jack J3. Apply 10-14 volts to connector J2. Set volume control VR1 fully clockwise for maximum volume.

Product Detector – Touch the right pad of L7 with a 3 foot test lead. A hissing signal should be audible.

Audio Muting – With the test lead still connected to the L7 pad, ground the cathode end (bar) of D6 with another test lead. The hissing signal being heard should now be muted.

Step 7 – Automatic Gain Control 1 Components

Install the first set of automatic gain control components consisting of TR1-10 K Ohm resistive trimmer, D7-1N5711 diode, C45-10 uF capacitor (make sure that the polarity is correct), Q6-PN2222A transistor, R26-1 K Ohm resistor, C46-22 uF capacitor (make sure that the polarity is correct), L7-1 mH inductor, and C43-0.01 uF capacitor. As each part is solder into the PCB, mark it off on the schematic with a highlighter. When all of them have been installed, your PCB will look like the photo below.



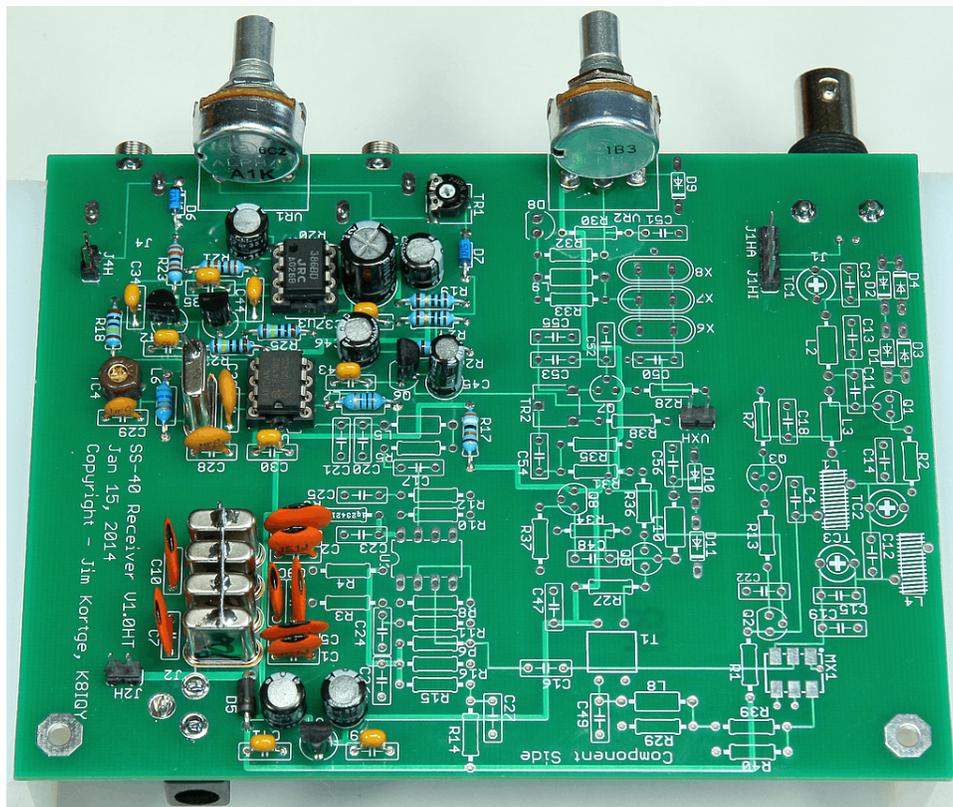
Automatic Gain Control 1 Components

Testing

No testing of this stage can be done now.

Step 8 – Crystal Filter Components

Install these components in the order they are called out: C2-150 pF capacitor, C6-270 pF capacitor, X4-9.21 MHz crystal, C10-270 pF capacitor, X3-9.21 MHz crystal, C9-270 pF capacitor, C8-150 pF capacitor, X2-9.21 MHz crystal, C7-270 pF capacitor, X1-9.21 MHz crystal, C1-150 pF capacitor, and C5-270 pF capacitor. Use an insulator under each crystal and ground the crystal cases to the pad provided, as shown in the photo. As each part is solder into the PCB, mark it off on the schematic with a highlighter. When all of them have been installed, your PCB will look like the photo below.



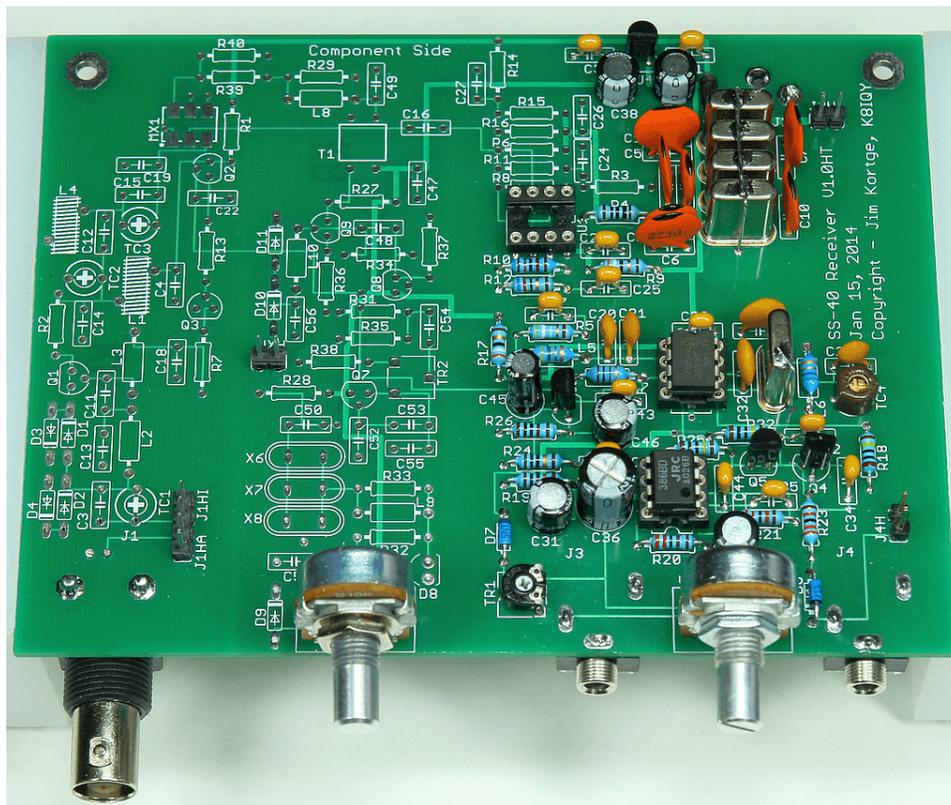
Crystal Filter - Rear

Testing

No testing of this stage can be done now.

Step 9 – IF Amplifier (U1B)

Install an 8-pin socket at the U1 position. The notch in the socket should match the outline on the PCB. Solder one lead and verify that the orientation is correct before soldering the remaining leads. Install the remaining components in this order: C21-15 pF capacitor, C20-47 pF capacitor, R4-100 Ohm resistor, L5-4.7 uH inductor, R5-49.9 Ohm resistor, C17-0.1 uF capacitor, R10-1.2 K Ohm resistor, R12-62 Ohm resistor, C25-0.1 uF capacitor, R9-39 Ohm resistor, and C23-0.1 uF capacitor. As each part is solder into the PCB, mark it off on the schematic with a highlighter. When all of them have been installed, your PCB will look like the photo below.



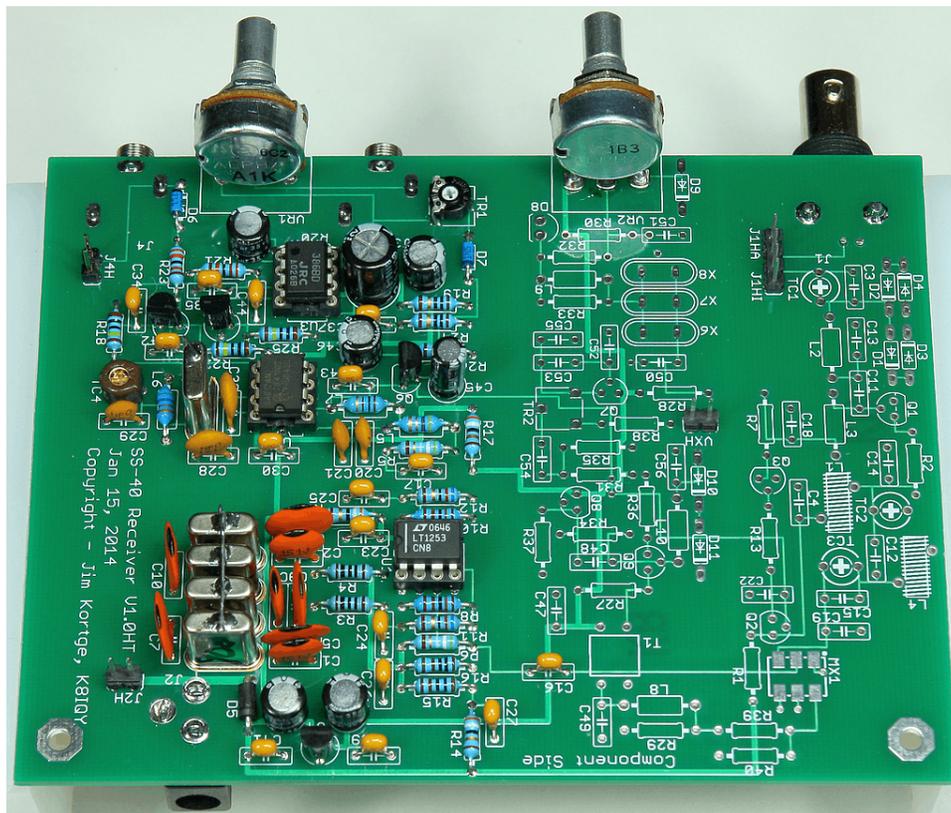
IF Amplifier_U1B Components

Testing

A number of tests will be performed after the U1A components are installed.

Step 10 – Post Mixer Amplifier (U1A)

Install the U1A, Post Mixer Amplifier components in this order: R3-100 Ohm resistor, R8-1.2 K Ohm resistor, R11-62 Ohm resistor, C24-0.1 uF capacitor, R6-49.9 Ohm resistor, C16-0.1 uF capacitor, C26-0.1 uF capacitor, R16-1 K Ohm resistor, R15-1 K Ohm resistor, C27-0.1 uF capacitor, and R14-39 Ohm resistor. As each part is solder into the PCB, mark it off on the schematic with a highlighter. Carefully bend the leads of the LT1253 chip so that the pairs of leads are parallel to each other and perpendicular to the chip body. Install the chip into the U1 socket; please don't force it and break a lead. When all of them have been installed, your PCB will look like the photo below.



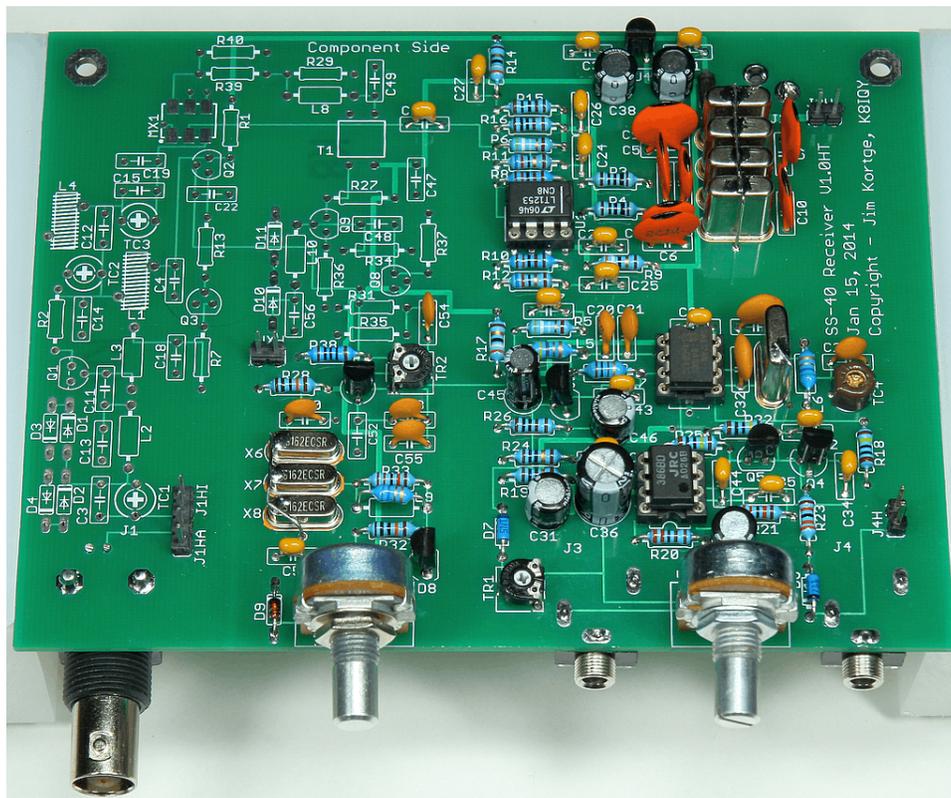
Post Mixer Amplifier_U1A_Components - Rear

Testing

Plug a speaker or earphones into audio jack J3. Apply 10-14 volts to connector J2. Set volume control VR1 fully counter clockwise for minimum volume. Now advance the volume control clockwise. At about 90% rotation, white noise should be heard and increase in volume as the control is advanced. When at full rotation, connect a 3 foot test lead to the mixer end of capacitor C16. More noise should be heard and possibly some static. The receiver is listening to 9.213 MHz, the nominal center frequency of the crystal filter. Rotate capacitive trimmer TC4 through 360 degrees. As that is done, a change in the audio tone of the noise should be noticed. We'll use TC4 to set the receiver to have a 600-750 Hz audio tone when listening to a CW signal centered in the pass band of the crystal filter.

Step 11 – VXO Components

Install the following parts: D9-1N4148 diode, R30-15 K Ohm resistor, C51-0.1 uF capacitor, R32-100 K Ohm resistor, D8-MVAM109 tuning diode, L9-5.6 uH inductor (space this part above the PCB by about 3/16 inch), R33-22 K Ohm resistor, X6-16.257 MHz crystal, X7-16.257 MHz crystal, X8-16.257 MHz crystal (use an insulator under each crystal and ground the crystal cases to the pad provided), C50-47 pF capacitor, C52 is not used, R28-100 K Ohm resistor, C53-47 pF capacitor, C55-47 pF capacitor, Q7-PN2222A transistor, TR2-1 K Ohm resistive trimmer, R38-100 Ohm resistor, and finally, C54-47 pF.. As each part is solder into the PCB, mark it off on the schematic with a highlighter. When all of these parts have been installed, your PCB will look like the photo below.

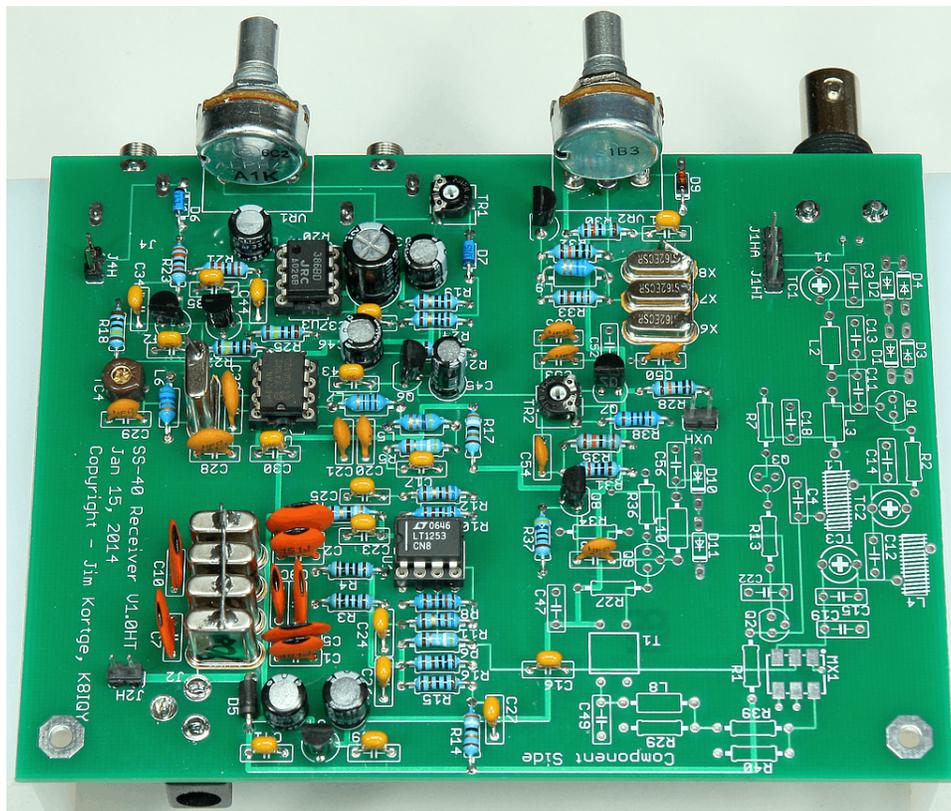


VXO Part 1 Components

Testing

A number of tests can be performed on this section of the VXO depending on the test equipment at hand. Apply 10-14 volts to connector J2. Set TR2, the resistive trimmer to the center of its rotation. All measurements are taken at the VXH header; the left pin is ground, and the right pin has the signal. Using an RF Probe with a DVM, a voltage of 0.8 to 1.0 volts should be seen. Using an oscilloscope, 2.5 volts p-p should be seen. If a counter is available, the VXO frequency can be measured. The VXO tunes 25 to 30 KHz, depending on the crystals and varies from receiver to receiver. The unit in the photo tunes from 16.2260 MHz to 16.253 MHz, which is a bit low from the typical. When finished, this receiver should nominally tune from 7.017 MHz to 7.045 MHz.

Continuing the VXO build, install these parts: R31-10 K Ohm resistor, R35-15 K Ohm resistor, Q8-PN2222A transistor, R37-470 Ohm resistor and C48-47 pF capacitor. As each part is solder into the PCB, mark it off on the schematic with a highlighter. When all of these parts have been installed, your PCB will look like the photo below.

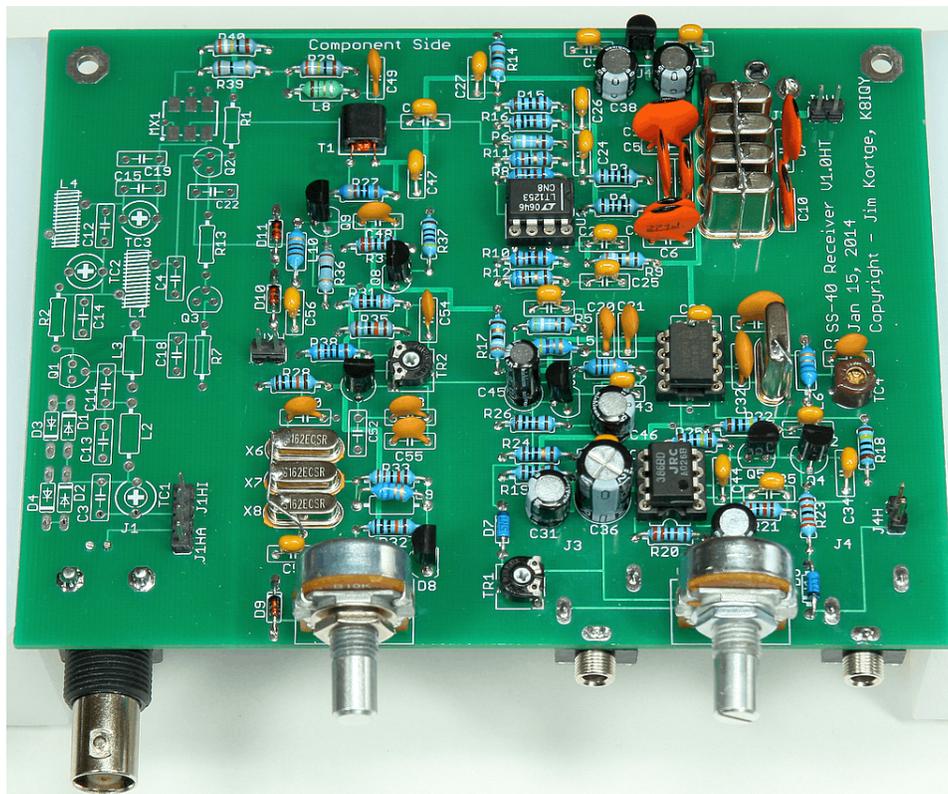


VXO Part 2 Components – Rear

Testing

The same tests that were performed in the previous step can be repeated with the test point now being the base lead of transistor Q9. The results will be nominally the same, since this stage is a gain of 1 (well almost) buffer.

Finishing the VXO build, install these parts: R34-2.2 K Ohm resistor, L10-1 mH inductor, D10-1N4148 diode, D11-1N4148 diode, C56-0.1 uF capacitor, C47-0.1 uF capacitor, R27-820 Ohm resistor, Q9-PN2222A transistor, R36-18 Ohm resistor, T1-4T Pri/4T Sec transformer (use 6 inch lengths of #30 gauge wire for the windings – see T1 details page), C49-47 pF capacitor, L8-1.8 uH inductor, R29-294 Ohm resistor, R39-18 Ohm resistor, and finally, R40-294 Ohm resistor. As each part is solder into the PCB, mark it off on the schematic with a highlighter. When all of these parts have been installed, your PCB will look like the photo below.



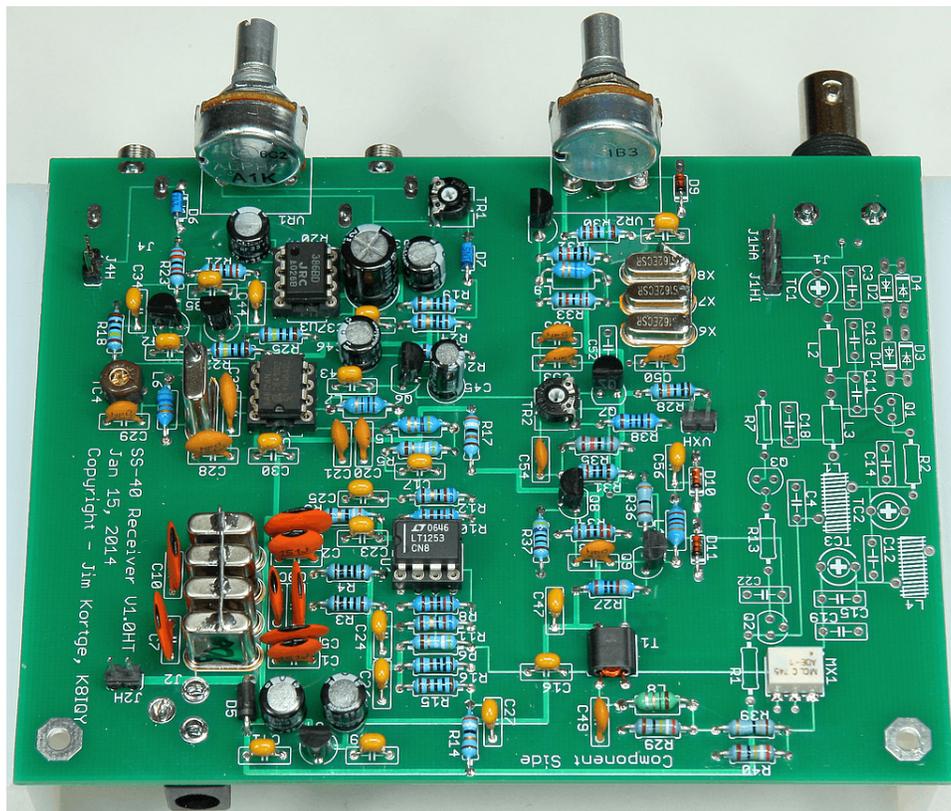
VXO Part 3 Components

Testing

A number of tests can be performed on this section of the VXO depending on the test equipment at hand. Apply 10-14 volts to connector J2. Set TR2, the resistive trimmer fully counterclockwise. Measuring at Mixer MX1-pad 6 and using an RF Probe with a DVM, a nominal level of 270 millivolts should be seen. As TR2 is rotated clockwise, the voltage should rise to a nominal level of 1.8 volts. Across the range of TR2, an oscilloscope will show 730 millivolts p-p at the low setting and 4.4 volts p-p at the high setting. On a spectrum analyzer, at the low setting, the signal available is nominally -8.4 dBm and at the high setting, 7.3dBm. At all settings of TR2, the 2nd harmonic of the 16.257 MHz (nominal) LO drive from the VXO is 25 dB below the carrier level.

Step 12 – Mixer Component

Notice that “component” is used as there is only one part to be installed in this step. However, it is probably the most difficult for most builders, especially those who are new kit builders. The key to soldering the MX1 Mixer is to use a tiny amount of liquid flux, if available, on each pad before setting the mixer on the PCB. If using liquid flux, let the part set for 15-20 minutes so that the flux “glues” the chip in place. The other key is to only solder one pin so that if the mixer moves position, that pin can be reheated and the chip pushed back into position. Also, use a small soldering tip for this part. Something on the order of 1/16 inch +/- should work fine on a 25-40 watt soldering iron. Once one pin is soldered and the chip alignment is correct (make sure the dots align), solder the other 5 pins. **Note: The mixer must be installed correctly for the SS-40HT receiver to work. This step must be done right!** After this part is soldered onto the PCB, mark it off on the schematic with a highlighter. When this step is done, your PCB will look like the photos below.



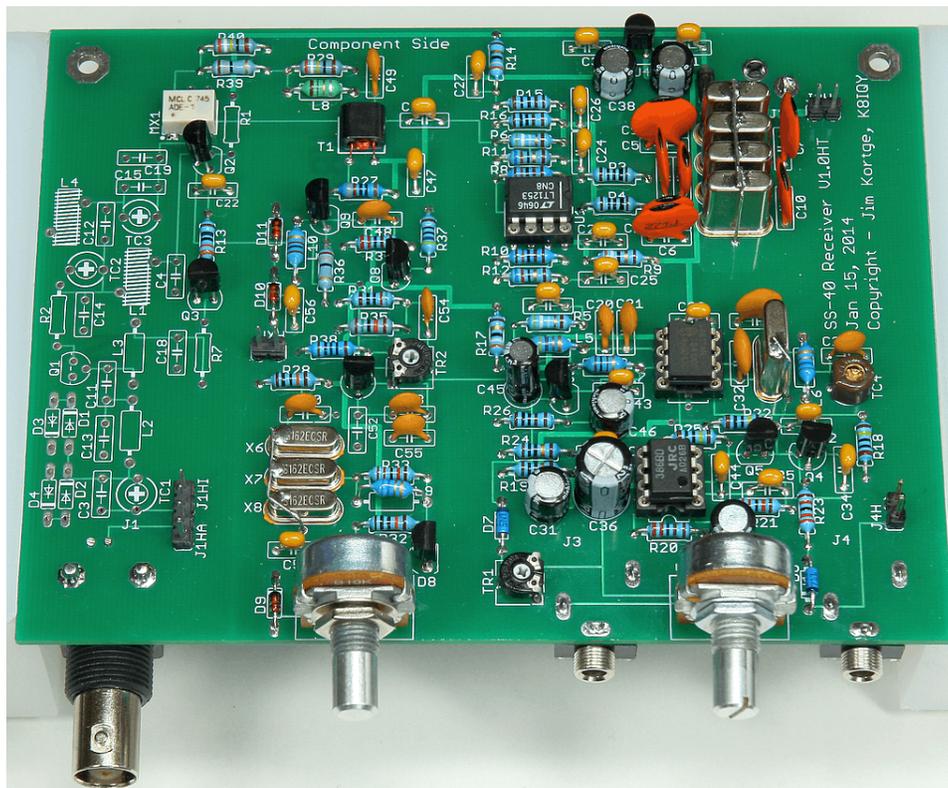
Mixer MX1 Component – Rear

Testing

Fun time! Plug a speaker or earphones into audio jack J3. Set volume control VR1 for minimum volume. Apply 10-14 volts to connector J2. Attach an antenna to the right pad for capacitor C19. Advance the volume control until band noise is heard. Using the VXO VR2 tuning control, tune around to hopefully hear a CW signal on the lower end of 40 meters. Adjustment of TC4 may be required to set the CW tone you like when receiving. Since no front-end filtering is in place, signals that are much higher in frequency, i.e. those that are the mixing product of the VXO + IF, or nominally 25.458 MHz might also be heard. The receiver performance will improve significantly when the Front-End parts are installed, but for now, enjoy the sounds of a mostly working receiver!

Step 13 – Automatic Gain Control 2 Components

Install the second set of automatic gain control components consisting of Q3-PN2222A transistor, R13-330 Ohm resistor, Q2-PN2222A transistor, and C22-0.01 uF capacitor. As each part is solder into the PCB, mark it off on the schematic with a highlighter. When all of these parts have been installed, your PCB will look like the photo below.



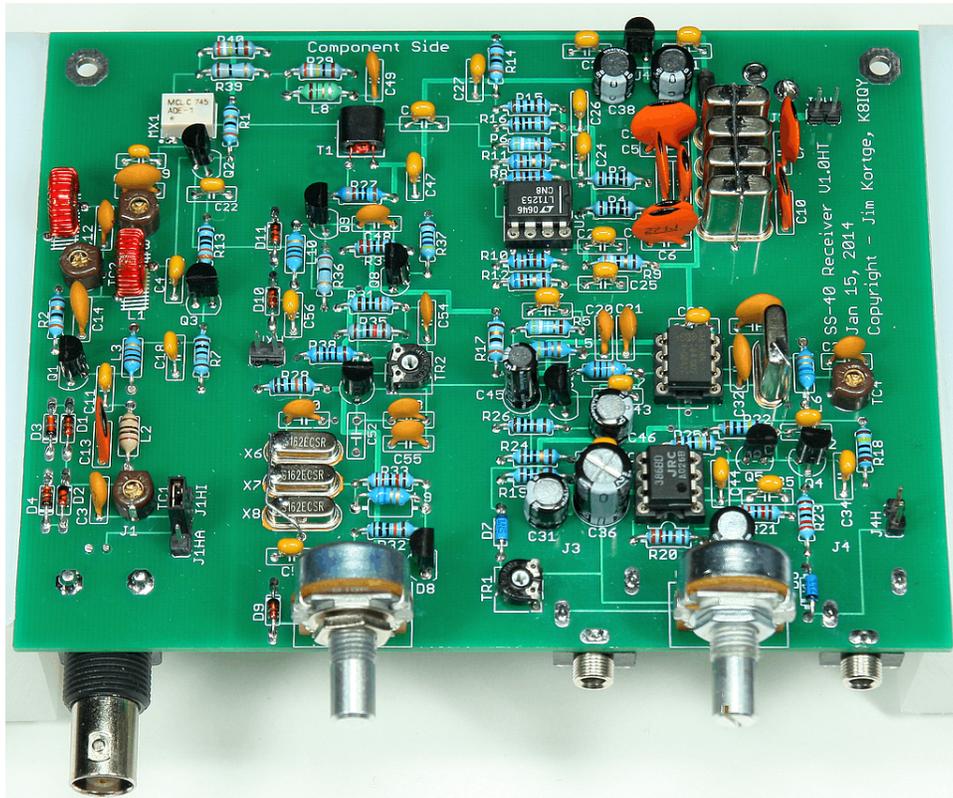
Automatic Gain Control 2 Components

Testing

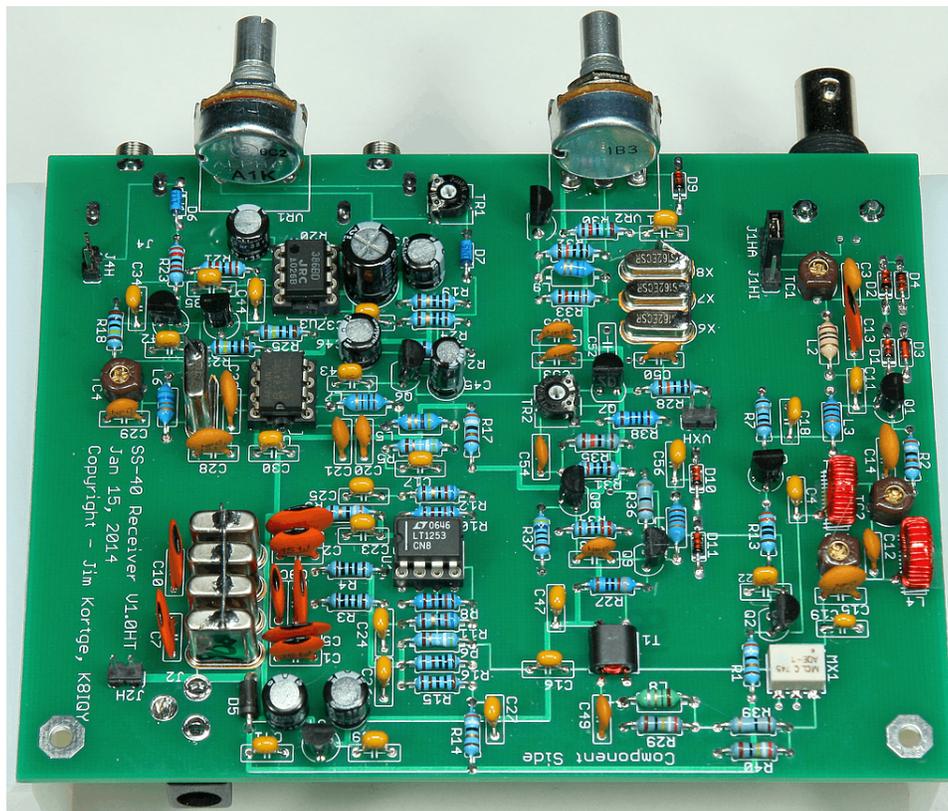
No testing of this stage (nor any others) can be done until the remainder of the receiver is built.

Step 14 – Front-end Components

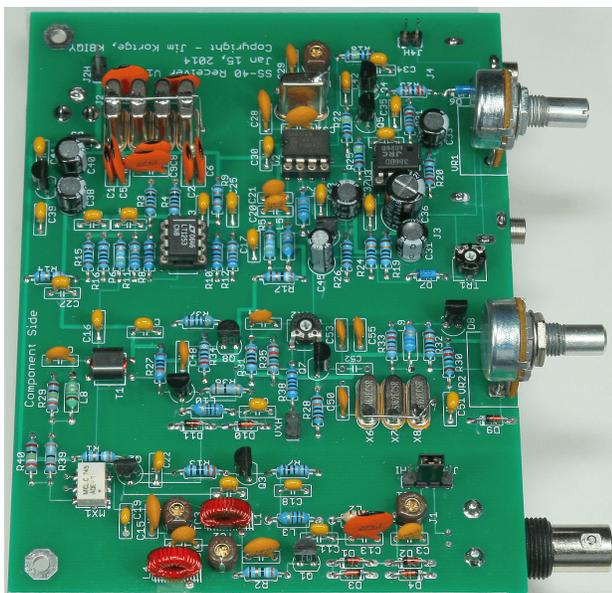
This is the final set of components. When they have been installed, the SS-40HT Receiver will be complete and ready for some simple alignment steps and then put into service. Install the supplied 2-pin shunt over the two center pins of the J1HA-J1HI header pair to connect the antenna BNC to the input of the receiver. Install these components in the order they are called out: C3-33 pF capacitor, TC1-10-70 pF trimmer capacitor, *Remove the bends in the leads of any trimmer capacitor before inserting it into the PCB and orient it so that the rounded end is the lower impedance or grounded end.* D1-1N4148 diode, D2-1N4148 diode, D3-1N4148 diode, D4-1N4148 diode, L2-10 uH inductor, C13-270 pF capacitor, C11-0.01 uF capacitor, L3-1 mH inductor, C18-0.01 uF capacitor, R7-180 Ohm resistor, Q1-J310 transistor, R2-39 Ohm resistor, R1-39 Ohm resistor, C4-0.1 uF capacitor, L1-28T 30 gauge wire wound tightly on T37-2 toroid core, C14-120 pF capacitor, TC2-10-70 pF trimmer capacitor, C12-4.7 pF capacitor, L4-29T 30 gauge wire wound tightly on T37-2 toroid core, TC3-10-70 pF trimmer capacitor, C15-120 pF capacitor, and finally C19-1000 pF capacitor. As each part is solder into the PCB, mark it off on the schematic with a highlighter. When all of these parts have been installed, your PCB will look like the photos below.



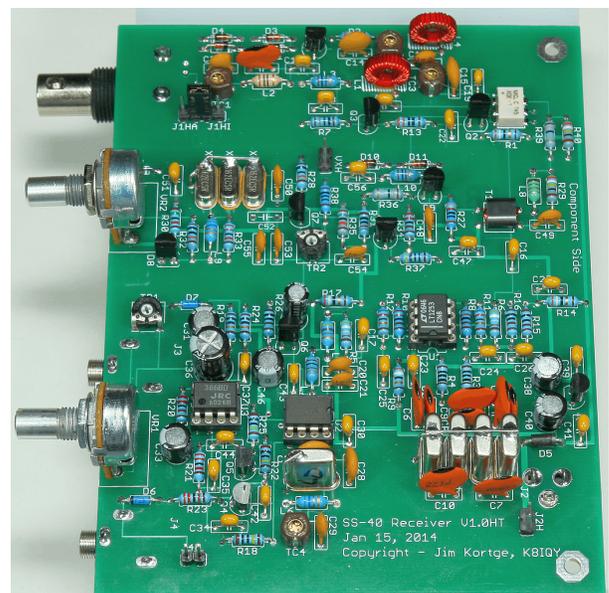
Front-end Components



Front-end Components – Rear



Front-end Components – Left



Front-End Components – Right

Testing

A lot more fun! Plug a speaker or earphones into audio jack J3. Set volume control VR1 for minimum volume. Set the VXO mixer drive level by adjusting trimmer TR2 to the center of its range. At that setting, approximately 7 dBm of LO signal is being supplied to the mixer. Apply 10-14 volts to connector J2. Attach an antenna to the BNC antenna connector, J1. Advance the volume control until

band noise is heard. Peak each of the trimmer capacitors beginning with TC3, then TC2, and finally, TC1. After the initial pass, go back and touch each of them up for the loudest band noise. Using the VXO VR2 tuning control, tune around to hopefully hear a CW signal on the lower end of 40 meters. Adjustment of TC4 may be required to set the CW tone you like when receiving. A better job of aligning the receiver can be performed using a signal generator set to 7.035 MHz with a signal level of 10-50 microvolts driving the input (J1) and an RMS voltmeter connected to the audio output connector J3. The three trimmers are adjusted for maximum signal.

To set the AGC level, turn the TR1 trimmer to the approximate center of its range and measure the DC voltage at the “hot” end of C43 with a DVM. With an antenna connected to J1 and the receiver tuned off of any signals, adjust TR1 until a voltage reading of 1.1 volts is measured. Then, while listening to a moderately strong signal, tweak the adjustment of TR1 until the best audio is achieved without any “clicks” or “thumps”.

TC4 adjusts the frequency of the Local Oscillator by moving the frequency of the LO crystal around. TC4 can be adjusted by ear for the audio tone you prefer. Tune across a weak signal and stop when it is the loudest. That places the signal in the middle of the crystal filter pass band. Then adjust TC4 in the direction that makes the resulting audio tone optimal for your hearing. Many CW operators like the audio tone to be in the range of 600 to perhaps 800 Hz, but it can be adjusted above and below those frequencies.

This completes the assembly and setup of the SS-40HT receiver. We hope building it has been a pleasant and rewarding building/testing experience.

Other Useful Information

The SS-40HT built and documented in this Assembly Manual uses the same set of parts that you received in your kit.

Performance measurements for a typical SS-40HT receiver:

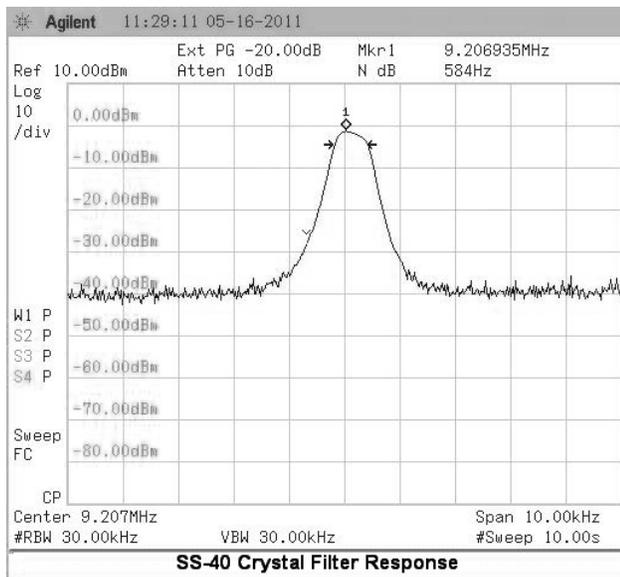
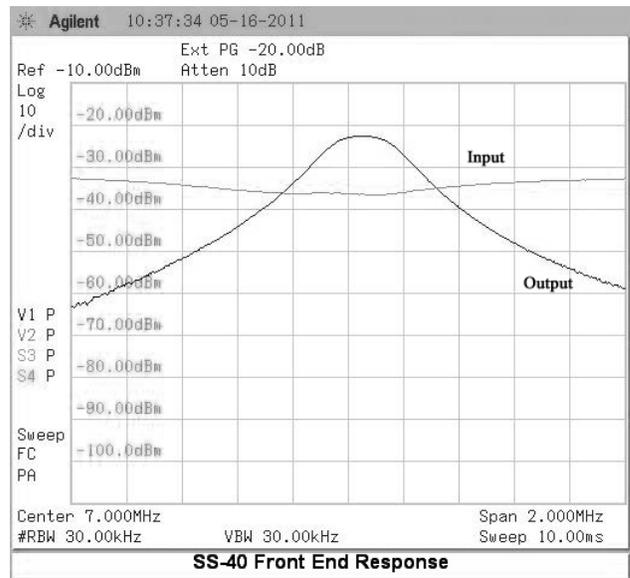
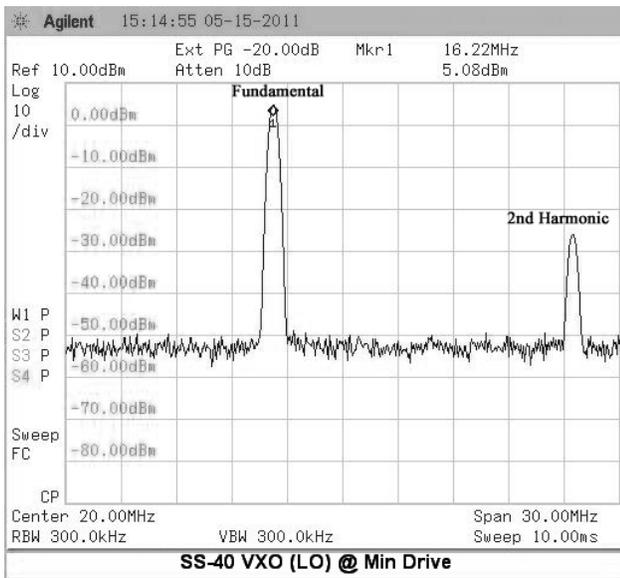
Minimum Discernible Signal (MDS) = -133 dBm

Opposite Sideband Suppression = ~50 dB

IF Signal Rejection = ~90 dB

Image Frequency Rejection = ~95 dB

Select Spectrum Analyzer plots for a typical SS-40HT receiver.



Not detailed is the ability to mute the audio on the receiver using the “key line” which comes out on connector J4. If the lead going to the tip of the mating connector for this jack is grounded, the muting circuitry is activated and the receiver audio will be greatly attenuated. This feature can be used to mute the receiver when it is used as part of a station and the user has implemented their own transmit/receive switching setup.

SS-40HT Specifications

The SS-40HT is a 40-Meter, VXO tuned, high performance receiver

- Super sensitive; Minimum Discernible Signal less than -130 dBm
- Super quiet; receiver noise floor at least 40 dB below band noise
- 4-pole IF filter for nominally 50 dB of opposite sideband rejection
- IF Rejection greater than 90 dB
- Image Rejection greater than 95 dB
- 25-30 KHz of “no perceptible drift” band coverage (7.030 & 7.040 included)
- Audio derived AGC (it really does work well)
- “Key Line” audio muting
- Current consumption less than 60 ma.

PCB construction using leaded parts

All PCB parts supplied

Approximately 165 parts

All controls board mounted – no wiring

Construction time ~ 20 hours

5.25" X 4.0 " PCB, will fit in a TenTec TP-41 case, but TenTec can no longer supply this case. (Case supplied by builder) Note: If the receiver will be used with the companion SS-40 Transmit Strip, the TP-41 case will not work. A different case will be required to fit that transceiver arrangement.

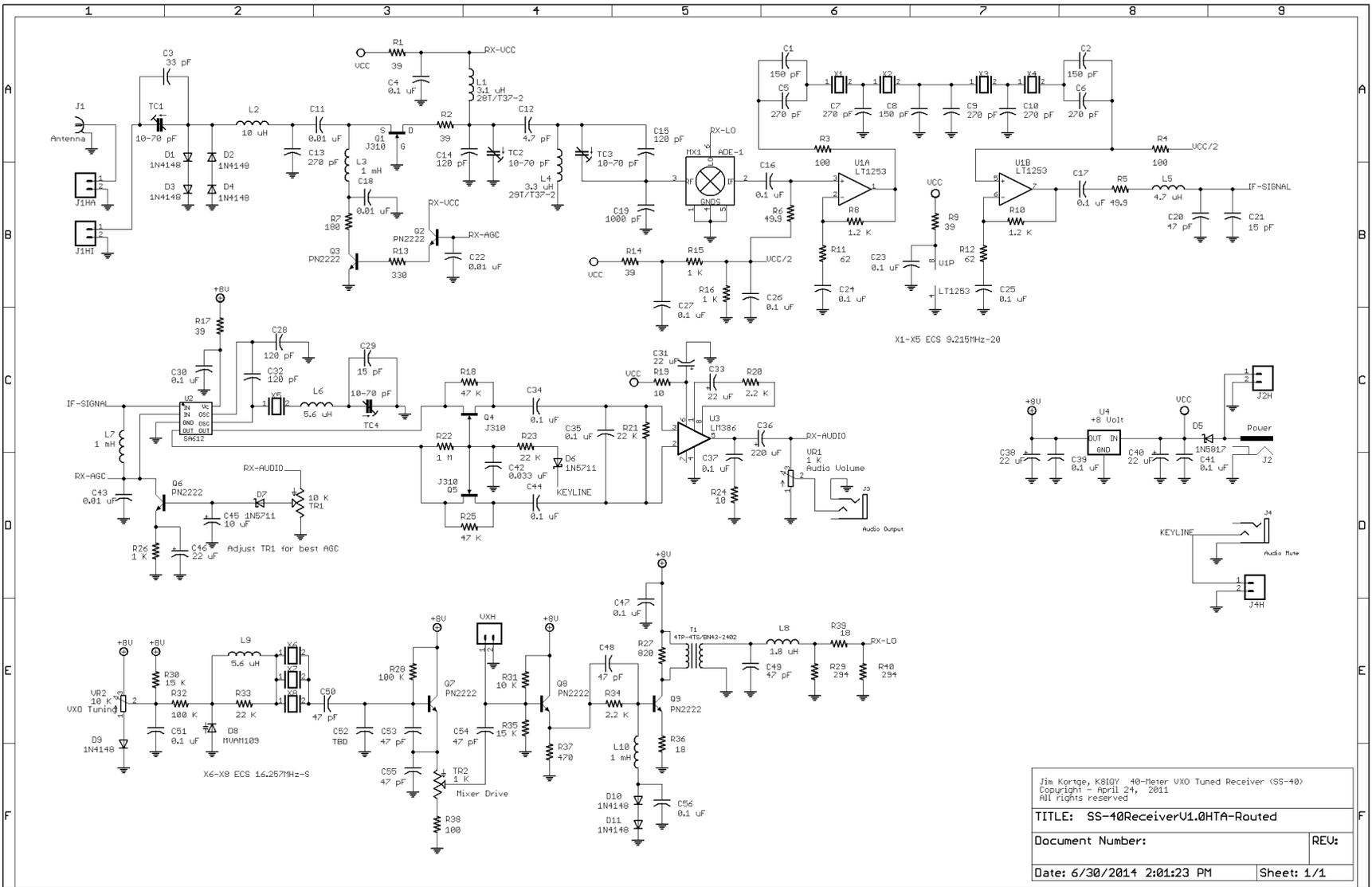
Targeted at Beginning/Intermediate builders; usable by everyone.

SS-40HT Bill of Material

Part	Value	Part	Value	Part	Value	Part	Value
C1	150pF	R1	39	D1	1N4148	U1	LT1253
C2	150pF	R2	39	D2	1N4148	U2	SA612
C3	33pF	R3	100	D3	1N4148	U3	LM386
C4	0.1uF	R4	100	D4	1N4148	U4	78L08
C5	270pF	R5	49.9	D5	1N5817		
C6	270pF	R6	49.9	D6	1N5711	MX1	ADE-1
C7	270pF	R7	180	D7	1N5711		
C8	150pF	R8	1.2K	D8	MVAM109	Q1	J310
C9	270pF	R9	39	D9	1N4148	Q2	PN2222A
C10	270pF	R10	1.2K	D10	1N4148	Q3	PN2222A
C11	0.01uF	R11	62	D11	1N4148	Q4	J310
C12	4.7pF	R12	62			Q5	J310
C13	270pF	R13	330	L1	3.1uH	Q6	PN2222A
C14	120p	R14	39	L2	10uH	Q7	PN2222A
C15	120p	R15	1K	L3	1mH	Q8	PN2222A
C16	0.1uF	R16	1K	L4	3.3uH	Q9	PN2222A
C17	0.1uF	R17	39	L5	4.7uH		
C18	0.01uF	R18	47K	L6	5.6uH		
C19	1000pf	R19	10	L7	1mH		
C20	47pF	R20	2.2K	L8	1.8uH		
C21	15pF	R21	22K	L9	5.6uH		
C22	0.01uF	R22	1M	L10	1mH		
C23	0.1uF	R23	22K				
C24	0.1uF	R24	10				
C25	0.1uF	R25	47K	J1	Antenna	J1HA	Ant. Header
C26	0.1uF	R26	1K			J1HI	Rx Input Header
C27	0.1uF	R27	820	J2	Power	J2H	Pwr. Header
C28	120pF	R28	100K	J3	Audio Output		
C29	15pF	R29	294	J4	Audio Mute	J4H	Mute Header
C30	0.1uF	R30	15K			VXH	VXO Header
C31	22uF	R31	10K				
C32	120pF	R32	100K	T1	4TP-4TS		
C33	22uF	R33	22K			X1	9.215MHz-20
C34	0.1uF	R34	2.2K	TC1	10-70pF	X2	9.215MHz-20
C35	0.1uF	R35	15K	TC2	10-70pF	X3	9.215MHz-20
C36	220uF	R36	18	TC3	10-70pF	X4	9.215MHz-20
C37	0.1uF	R37	470	TC4	10-70pF	X5	9.215MHz-20
C38	22uF	R38	100	TR1	10K	X6	16.257Mhz-S
C39	0.1uF	R39	18	TR2	1K	X7	16.257Mhz-S
C40	22uF	R40	294			X8	16.257Mhz-S
C41	0.1uF						
C42	0.033uF	VR1	1K		Miscellaneous		
C43	0.01uF	VR2	10K	3	8-pin DIP sockets		
C44	0.1uF			4 ft	#28 wire		
C45	10uF	C51	0.1uF	1	PC Board		
C46	22uF	C52	TBD	1	BN-43-2402 *		
C47	0.1uF	C53	47pF	2	T37-2 toroids**		
C48	47pF	C54	47pF	2	Header Pins		
C49	47pF	C55	47pF	1	Shunt		
C50	47pF	C56	0.1uF	1	5.5 mm OD Plug		

* for T1

** for L1 & L4



Schematic Diagram

Jim Kortge, K8IGY 40-Meter VFO Tuned Receiver (SS-40)
 Copyright - April 24, 2011
 All rights reserved

TITLE: SS-40ReceiverV1.0HTA-Routed

Document Number: _____ REV: _____

Date: 6/30/2014 2:01:23 PM Sheet: 1/1

Metal Film Resistor Color Codes

The resistors used in the SS-40HT are metal film, ¼ watt, and 1% tolerance devices. They were selected for their lower noise characteristics. They have a somewhat different color coding than is used with the more common carbon film 5% tolerance resistors. Below is a table for resistor values used in the SS-40HT receiver and their color codes.

Value (Ohms)	Color Code
10	Brown Black Black Gold Brown
18	Brown Gray Black Gold Brown
39	Orange White Black Gold Brown
49.9	Yellow White White Gold Brown
62	Blue Red Black Gold Brown
100	Brown Black Black Black Brown
180	Brown Gray Black Black Brown
294	Red White Yellow Black Brown
330	Orange Orange Black Black Brown
470	Yellow Violet Black Black Brown
820	Gray Red Black Black Brown
1 K	Brown Black Black Brown Brown
1.2 K	Brown Red Black Brown Brown
2.2 K	Red Red Black Brown Brown
10 K	Brown Black Black Red Brown
15 K	Brown Green Black Red Brown
22 K	Red Red Black Red Brown
47 K	Yellow Violet Black Red Brown
100 K	Brown Black Black Orange Brown
1 M	Brown Black Black Yellow Brown

If you can't reliably read the color code on the resistor, check its value with your Ohm meter.

Molded Inductor Color Codes

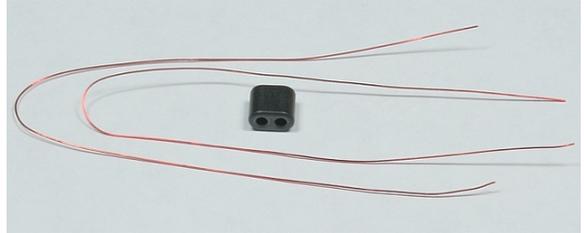
The molded inductors used in the SS-40HT are also color coded and shown in the table below.

Value (uH)	Color Code
1.8	Brown Gray Gold Silver
4.7	Yellow Violet Gold Gold
5.6	Green Blue Gold Gold
10	Brown Black Black Gold
1000 (1 mH)	Brown Black Red Gold

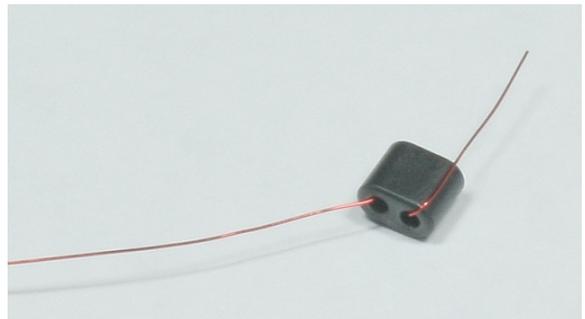
T1 Winding Information

Here is a short tutorial on how to wind transformer T1

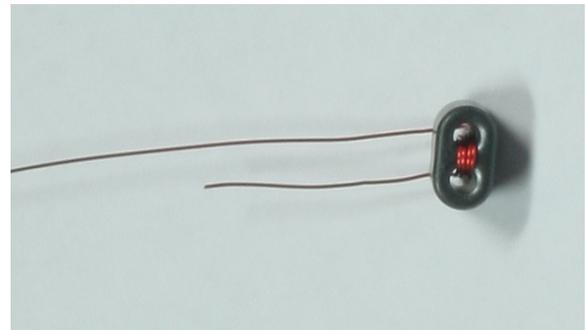
Begin with the BN2402-43 binocular core and two 6 inch lengths of #28 or #30 wire.



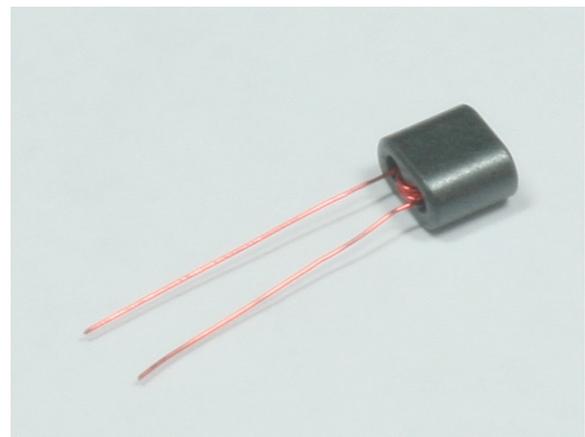
Bend a wire back on itself about 1.25 inches and insert into the core from one end. Snug this first turn up tightly, but not to the point of breaking the wire.



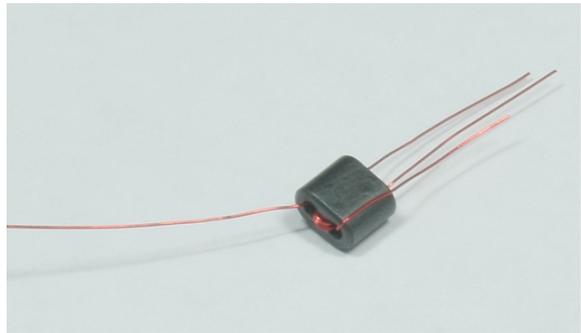
Wind on an additional 3-turns, keeping the turns tight and organized on the core. A turn is counted when the wire goes through a core hole and back through to the opposite side. When 4-turns have been wound on the core, the opposite end from where the wires exit will show 4-turns.



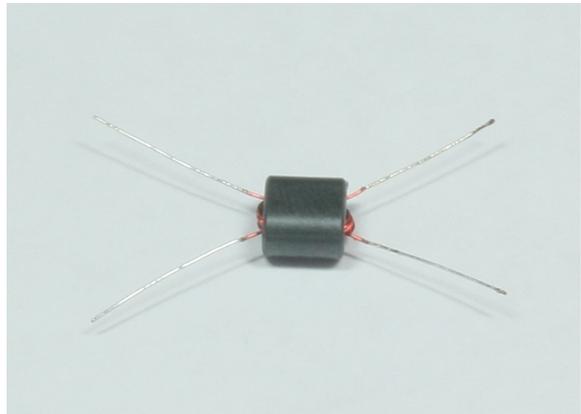
When 4-turns have been wound on, trim off the excess wire leaving about 1 inch long leads.



On the opposite end of the core, repeat the process of winding 4-turns on the core, keeping the turns tight and neat.



Trim all of the wires to approximately 1 inch, remove the insulation, then tin each lead as shown. T1 is now complete and ready to install into the PCB.



SS-40HT Receiver changes when used with the SS-40 Transmit Strip

One minor change has to be made when the SS-40HT Receiver is used with the SS-40 Transmit Strip. This change is needed to increase the side tone volume during transmit periods when the receiver is muted. As designed, the side tone coupling resistors, R18 and R25, are 1M Ohm parts, assuring the side tone level is very low. With the SS-40 Transmit Strip attached, more audio is desired to hear what is being sent. The 1M resistors should be removed and replaced with 47K Ohm resistors. This will set the side tone at a comfortable level for most operators with normal hearing. To improve the QSK keying, capacitor C42 should be changed to a 0.033 uF value. The above values are included in future releases of the kit.