QRP

New Jersey

SOP Receiver Kit

A "universal" CW receiver for 80m and 40m operation. Easy to modify for use on other bands. Direct conversion for simplicity and performance. Low current drain from 9V-12V battery is perfect for portable operation. Open pc board layout - perfect for novice homebrewers, great for experimenting. Add-on accessories: FreqMite audio frequency dial, homebrew copper-clad enclosure, all controls, knobs & jacks.

Brought to you by the New Jersey QRP Club

http://www.njqrp.org

About this manual ...

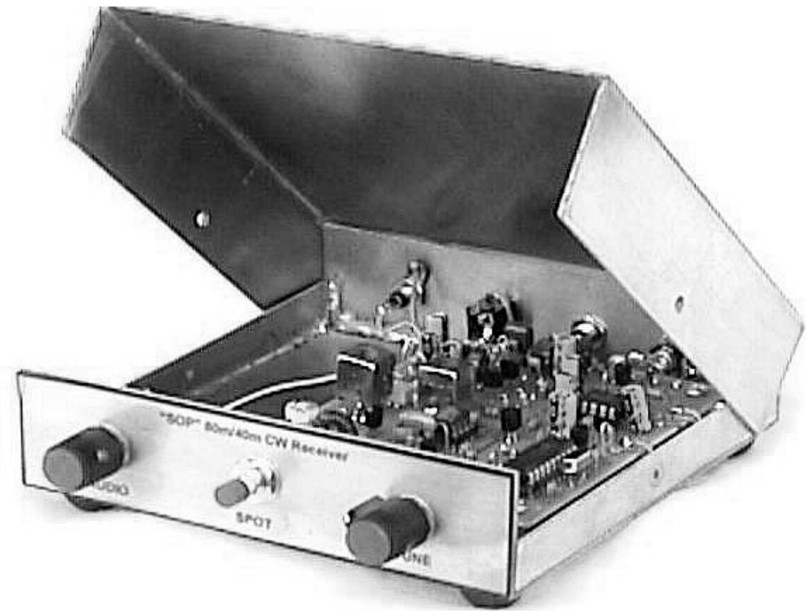
This manual actually consists of five separate manuals combined into a single and convenient reference manual for the NJQRP SOP Receiver.

Edited and published by N2APB, using Pagemaker v6.
Printed by Sir Speedy Printing in Baltimore, Maryland.

SOP Receiver Kit

Assembly Manual

The SOP is a basic "Seat Of the Pants" receiver designed by veteran elmer Joe Everhart, N2CX, and is being kitted and sold by the NJQRP Club. This kit is a very flexible and inexpensive direct conversion CW receiver for 40m or 80m. Providing better quality than a single-chip receiver, yet not as complex as a superhet, the SOP's straightforward design and construction is ideal for newcomers to the homebrewing scene as well as for experienced QRP builders.



WELCOME!

Thanks for purchasing the SOP Receiver Kit from the NJQRP Club. This section of the manual provides a basic technical overview of the design and operation, and details the assembly required to make it a functional receiver in your shack.

The SOP (Seat-Of-the-Pants) is intended as a utility CW receiver, useful in the shack to complement some of the simple transmitter projects available today (Tuna Tin 2, Fireball 40, etc.); or for use in the field because of its low power consumption and small size.

It uses a simple direct conversion technique and is well-suited for experimentation. Though designed originally as a companion to the NJQRP Fireball-40 transmitter, it is a "natural" to team up with any simple QRP transmitter. Another use is that of band monitoring. The SOP provides an elegant way to listen to a band wherever you might be.

The SOP incorporates an audio low pass filter to minimize off-frequency interference but by design is not so selective that nearby signals cannot be heard. While superheterodyne receivers offer better performance, the SOP's simplicity makes it easier to tune up on anywhere in the HF spectrum for CW (or SSB) reception.

An important concept to keep in mind is that the SOP is an experimenter's receiver. The building blocks used in the design are simple and straightforward, providing a clear functional representation of a relatively generic receiver platform. Thus the SOP is a perfect receiver platform with which to try out new ideas.

The printed circuit board for the SOP is similarly generous in size to provide novice fingers ample space to assemble and for oscilloscope probes and extra components to be connected. The functional blocks of the design are arranged in a manner to allow easy modification and substitution of components. In fact, one could easily provide little "daughter boards" to sit directly above any particular module to replace or augment its function. For example, if a better audio filter is desired (more poles, different rolloff characteristics, etc), one could create a 1" x 1" board containing the improved filter and suspend it above the stock filter on the middle-right end of the board, jumpering the input and output signals down to the original board. Great flexibility is provided in the SOP in this manner.

The SOP Receiver was designed for flexibility. Again, with the basic no-frills component construction, the design can be easily modified for use on other bands. Application notes will be available in the near future describing the changes necessary to put the receiver on any HF ham band, from 10m to 160m.

The SOP is not necessarily the highest-performing receiver you can find. But for its low cost, it will provide you many hours of quality service as a flexible receiver in the shack, as well as being a platform for you to modify and improve on as your needs and skills develop.

In the process of assembling your SOP and in modifying it to suit your particular needs, you will gain a better understanding of the principles of receivers, including oscillators, mixers, RF and audio amplification, and audio filtering.

We hope you enjoy your SOP Receiver kit!

MINI-SPECS

Here is a brief listing of the SOP receiver's specifications. See Figure 1 for a corresponding block diagram.

1) The SOP is a "universal" HF CW receiver;

- Low current drain; able to operate from 9V battery or 12V source;
- Small size enables use as custom rig or as standalone portable monitor receiver;
- Jumper selectable operation on 80m or 40m;
- Can be set up on any other HF band (30m-10m) with few component changes;
- No critical assembly or alignment required;
- Tailorable for advanced homebrewers - extra pcb space provided for experimentation;
- No odd-ball crystals, variable capacitors or other components;
- Plug and play design ideal for beginners or advanced homebrewers;
- Solid repeatable design;
- Open layout on PC board simplifies construction.

2) The RF Amp

- Provides front end gain and selectivity;
- Very stable grounded gate FET design;
- Provides adequate gain and enhances noise figure on higher HF bands;
- Can be tailored to any HF band via parallel inductor or capacitor (pcb space pro-

vided).

3) The Local Oscillator

- Simple, trouble-free VXO design provides variable tuning control;
- Uses ceramic resonator for 80m & 40m operation, easily placed anywhere on band with alternate resonator/crystal;
- Tuning range is > 50 KHz on 80m, and > 100 KHz on 40m, ensuring coverage of the standard QRP operating frequencies on each band;
- Tuning range can be limited simply by changing padding resistors;
- Can use other crystal or resonator to cover any HF band on 80m to 10m;
- Output tank is shielded inductor - eliminates fragile trimmer capacitors;
- Output tank tailored to any HF band via parallel inductor or capacitor (pcb space provided);
- Varactor tuning via potentiometer eliminates need for big, expensive variable capacitor.

4) The LO buffer

- Stable, repeatable design provides high-level drive to mixer stage for improved performance;
- No adjustments;
- Broadband performance.

5) The Mixer

- Single-chip, passive double balanced mixer (dbm);
- No diode matching or toroid winding, yet all the benefits of discrete dbm performance;
- Better performance in crowded HF bands than single IC mixers;
- Simple wide-band termination on output helps reject interference.

6) The Audio Stage

- Low noise bipolar amplifier provides amplification and filtering;
- Integrated circuit audio filter helps with cw reception;
- Low pass filter peaked for CW tones;
- No adjustments;
- No ringing or ear fatigue;
- Filters out high frequency “monkey chatter”.

7) The Audio Muting and Amplifier

- Proven muting design quiets receiver audio during transmit via keyline signal from transmitter;
- Bypass resistor on muting switch allows operator to hear sidetone of the transmitted signal;
- Allows verification of transmit frequency offset;
- Proven LM386 amplifier features high frequency hiss rejection;
- Anti motorboat components give clean audio output;
- Sufficient output for low impedance headphones or small speaker.

CIRCUIT OVERVIEW

Figure 2 shows the SOP’s schematic.

RF Pre-Amp

The SOP RF amplifier is a single-stage common-gate FET stage that provides front-end gain and selectivity for the receiver. The antenna input is link-coupled to a tuned circuit to provide impedance matching which preserves circuit Q while efficiently coupling receive energy. Resonance at 7 MHz is achieved by use of a 150 pF trimmer capacitor, allowing “on-frequency” tuneup. The transformer winding instructions can be scaled to other amateur bands as well. Energy from the tank is coupled to the FET source via a two-turn tap. Not that both the input link and output tap are 2 turns providing identical input and output matching. The input impedance of the grounded gate amplifier is in the range of 50 to 100 ohms so a reasonable match is achieved.

Source resistor R11 is used to provide a negative gate bias to stabilize FET operation with a source current of several milliamperes. The FET would probably give more gain without the resistor, but bias current would vary widely with de-

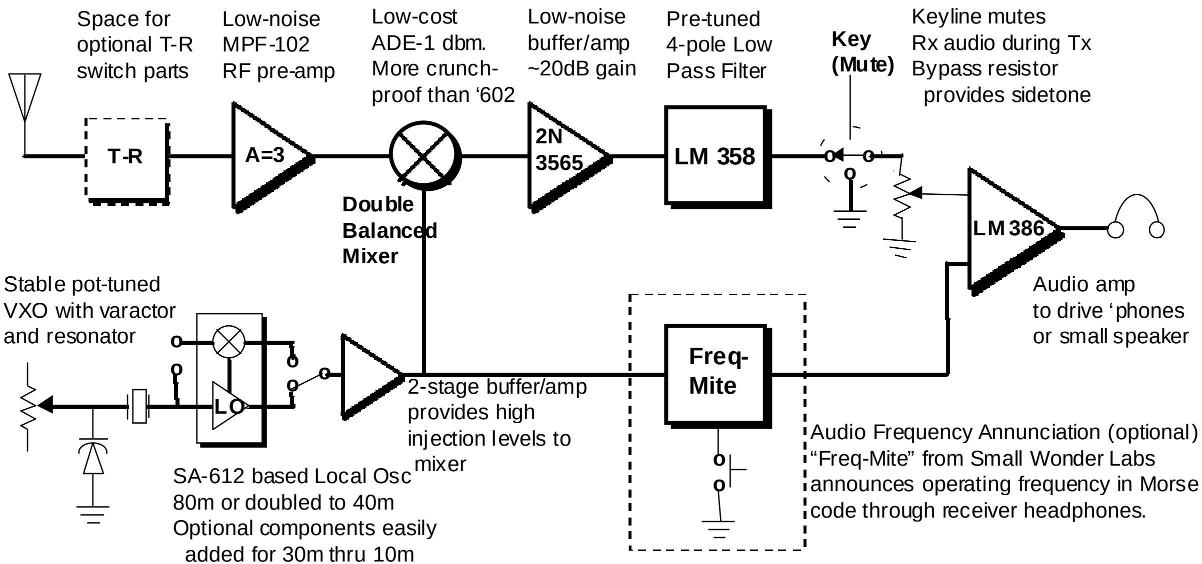


Figure 1 - SOP Receiver Block Diagram

vice-to-device tolerances. Gain stability is more important than raw gain. Capacitor C15 provides a low-impedance bypass across the R11 to prevent signal loss. The common gate configuration provides a small voltage gain with a low noise figure. The low noise is not too important at the low end of the HF spectrum but will aid ultimate receiver sensitivity on 20 meters and higher. An additional advantage of the common gate stage is that it is quite stable and repeatable, thus requiring no neutralization or tricky adjustments.

Output selectivity and matching are achieved via a common 10.7 MHz IF transformer. It is resonated on 30 meters and below by means of an external capacitor. On 20m and above, an external capacitor and inductor do the job. Link coupling steps down the high FET drain impedance to the 50 ohm level required by the ADE-1 balanced mixer. Tuning adjustment is done by the variable inductance of the shielded transformer. Tuning is rather broad since the transformer is heavily loaded by the following double balanced mixer.

Mixer

One of the most important areas of the SOP is the mixer. A Mini-Circuits ADE-1 double-balanced diode mixer is used. This device is a surface mount packaged equivalent of two small toroid transformers surrounding a ring of four diodes. It provides good sensitivity while giving much less audio rectification than the more common NE602 Gilbert cell mixer used in other direct-conversion receivers. (The NE602 has been recently been superseded by the SA612 so from here on out we will use the latter part number.) R12 and C18 provide a wide-band termination to the ADE-1 mixer.

The Local Oscillator

The SOP uses an SA612 to provide the local oscillator signal for the mixer. This device incorporates both an oscillator and a mixer in an 8-pin package — this dual functionality is used to good advantage. The "stock" SOP uses a 3.58 MHz ceramic resonator in a voltage controlled oscillator to tune a portion of the 80 meter band. The resonator has a lower Q than that of a quartz crystal so it can be tuned over a wider range while still being more stable than an LC circuit. The fundamental oscillator output is used for 80 meter operation. However if the oscillator signal is fed to the SA612 mixer section, it is doubled in frequency to 40 meters. A tuned circuit on the SA612 output minimizes 80 meter feedthrough. The receiver can then be used on 40 meters by retuning the RF amplifier to that band.

This "trick" works well for 80 and 40 meters because of the inexpensive resonator. Unfortunately there are no standard units available on other ham bands. However the SA612 can be set on other bands by using quartz crystals in place of the resonator. Tuning range can be enhanced by using the "super VXO" techniques of adding a series inductance and paralleling multiple crystals. Naturally the RF amplifier must be retuned to the band of interest. Application notes will soon be made available showing how to effectively modify the basic 80m/40m SOP for use on other bands.

The Local Oscillator is tuned by providing a variable DC voltage to a varactor diode. The TUNE potentiometer provides a portion of the regulated 6V DC level coming from the 3-terminal voltage regulator U2. C16 and R10 bypass the DC control signal to keep RF from getting into the tuning components. When varactor diode D3 sees a range of DC bias voltage applied, its characteristic capacitance changes and adds to the total capacitance in the Y1 resonator circuit. This varying capacitance "bends" the effective frequency of the oscillator, providing the range of frequencies under control of the operator.

The TUNE potentiometer is padded by a resistor on each end. R7 and R8 can be a jumper wire (i.e., zero ohms) to provide the widest swing of DC control voltage, and thus the widest swing of frequencies from the oscillator. However this might be too course an adjustment for some, so a smaller tuning range can be forced by placing actual in position at R7 and R8, thus providing a smaller range of control voltage for the same 300-degree turn of the pot. This will have the effect of providing "bandspread-like" operation and give the operator finer tuning control, making it easier to tune closely-spaced stations.

LO Buffer

The Local Oscillator Buffer is comprised of the Q1 and Q2 stages immediately following the LO. Transistor Q1 is configured as an emitter follower to serve as a high impedance buffer to minimize loading of the LO. Transistor Q2 is a common emitter amplifier with a wide-band transformer to provide a low impedance output (50-ohms) to drive the mixer stage. By constructing T1 as a trifilar-wound toroid transformer, we have an easy way to achieve the wide-band impedance transformation. R30, C10 and C37 decouple the buffer amplifier from the rest of the circuits on the +V power line and thus reduce the chance for feedback to and from the power bus. (Direct-conversion receivers are very sensitive to cross-coupling through common power bus connections.)

Audio Gain

The ADE-1 audio output is boosted by a low-noise 2N3565 transistor amplifier. As in all direct conversion receivers most of the gain is produced at audio frequencies. The mixer output is carefully filtered by C36 to prevent RF feed-through. If you don't do this, strong AM signals will be rectified in the audio amplifier! Again, DC power to the audio amp is carefully filtered by R19 and C34 to minimize feedback from other stages.

Low Pass Filter

A low-Q low pass filter is peaked in the 700-800 Hz area to give some audio selectivity but mainly to filter out off-frequency signals. This design originated with Wes Hayward some years ago and has been detailed in a number of publications and articles. Probably the most common reference is the ARRL "Solid State Design for the Radio Amateur" by Hayward and DeMaw.

This unity gain amplifier and filter is direct-coupled — the same voltage appears at R15/R16 junction, and at the op amp output pins 1 and 7. C22 and C25 are NPO-type capacitors for good frequency stability of the filter. C23 and C25 are also important for frequency stability, but here we use mylar capacitors for a cost effective solution.

Muting

Muting of the receiver is required during "key down" times in order to keep the very strong signal from blasting out the audio amp output stage when the local transmitter is keyed. This clever muting circuit, borrowed from Hayward and Lewellan, uses the key line signal from a companion transmitter to switch off an FET to reduce audio levels during transmission. However when the key line is closed and the FET is turned off (its gate will be negative with respect to its source), some audio signal is still provided through R23 to produce a ready-made sidetone for the operator.

Audio Amp

The SOP uses the ubiquitous LM386 in the audio output stage. This device has sufficient drive for headphones; a loudspeaker can also be used in a quiet room when receiving strong signals.

R13 and R25 keep the FreqMite audio annunciator from being affected by the setting of the AUDIO LEVEL potentiometer.

R26 and C29 serve to decouple the amplifier from the power bus. This prevents the strong output signals of the LM386 from getting back to other components in the receive chain.

C30 sets the amplifier to the maximum available gain. C31 provides low impedance DC decoupling. R27 and C32 provide feedback to lessen high frequency hiss in the audio output. R28 and C33 dampen the output of the amplifier to minimize high frequency oscillation and instability.

CONSTRUCTION

Here are the steps to follow in putting your SOP Receiver Kit together. Refer to the Parts List on the next page for a description of each part in the kit.

1. Parts Inventory

Check out the contents of your kit to ensure that all parts are present. As you identify each component, put a little checkmark next to that line item, or write the number of components you count up for the lines containing multiple parts of the same value.

The most of integrated circuits (ICs) and transistors will be found pushed into an anti-static foam pad.

The mixer integrated circuit (U3) is a "surface mount device", or SMD. It is contained in a little 1/4" x 1/4" square plastic container with a clear plastic cover. We'll describe how to mount this when the time comes, but don't worry about it right now ... it's a piece of cake!

2. Other Things Needed

You'll need some common tools and things normally found on the workbench.

Soldering tools: A fine-tipped, low-wattage soldering iron will be important to have available. Ideally, a temperature-controlled solder station has been found to be of great service to many homebrewers and kit builders. I've found that setting the tip temperature to about 650-degrees works best. Use of a wet pad or rag for wiping the tip just before soldering a component lead keeps the tip clean and free of oxidation and carbon crud. A supply of standard 60/40 solder from Radio Shack (p/n 64-009) will work just fine. And finally, to clean up the inevitable solder shorts, many builders have had great success using SolderWick, a braided wire to help soak up the solder bridges when heated.

The basic tools needed in putting together a kit such as this include fine-point needle nose pliers, wire (side) cutters, a small-blade screwdriver for adjusting the IF transformer cans, and an Exacto blade for general probing around and scraping off the insulation from the magnet wires when making the transformers. Other, more elaborate tools can be useful but are not required. These might include tweezers for holding very small parts, "third hand" device for holding the board at various angles, and a lighted magnifying glass .

Connecting wire: About 5" of thin gauge hook up wire will be needed for the single jumper on the board. And you'll need some additional wire to connect your favorite controls and jacks to the board. Individual wires, strips of ribbon cable, and thin RG-174 coax are examples of such hookup wire that you probably have this laying around your workbench or in the junk drawer.

Off-board components: In order to complete the SOP Receiver kit, you'll need to supply the two potentiometers for the "front panel" controls (Tune and Audio Level) and the four "rear panel" jacks

SOP RECEIVER PARTS LIST

Designator	Qty	Value	Description
C2	1	22pF Disc	capacitor, orange, "22"
C14	1	47pF NPO	capacitor, orange, "47"
C7	1	100pF Disc	capacitor, orange, "101"
C4, 5	2	220pF Disc	capacitor, orange, "221"
C22, 24	2	680pF NPO	capacitor, orange, "681"
C23, 25	2	0.1uF Mylar	capacitor, larger, dark red, "104"
C20	1	5-100pF	capacitor, trimmer, white, 3-pins
C8, 15, 27, 32	4	.01uF Mono	capacitor, blue, "103"
15, 16, 18, 20, 28, 33, 36	15	0.1uF	capacitor, orange, axial lead, "M074"
C19, 21	2	2.2uF Elec.	capacitor, blue, radial lead
C30, 34	2	10uF Elec.	capacitor, blue, radial lead
C29, 37	2	47uF Elec.	capacitor, blue, radial lead
C31	1	100uF Elec.	capacitor, blue, radial lead
R5	1	4.7	resistor, yellow-violet-black
R26, 28	2	10	resistor, brown-black-black
R12	1	47	resistor, yellow-violet-black
R4, 30	2	100	resistor, brown-black-brown
R11	1	220	resistor, red-red-brown
R3, 19	2	1K	resistor, brown-black-red
R31	1	2.2K	resistor, red-red-red
R25	1	4.7K	resistor, yellow-violet-red
R1, 2, 6, 11, 13, 15, 16, 27	8	10K	resistor, brown-black-orange
R14	1	220K	resistor, red-red-yellow
R22, 23	2	1M	resistor, brown-black-green
T1	1	T37-2	red "donut" core
L1, T2	2	455KHz IF	5-lead "can", 42IF124 (Mouser)
U1	1	SA612AN	8-pin, integrated circuit, DIP
U3	1	ADE-1	8-pin surface mount IC, tan
U4	1	LM358N	8-pin, integrated circuit, DIP
U5	1	LM386-1	8-pin, integrated circuit, DIP
Q1, 2	2	PN2222A	3-pin transistor, 2N2222A, TO92 case
Q3, 5	2	2N5485	3-pin transistor, 2N5485, TO92 case
Q4	1	2N3565	3-pin transistor, 2N3565, TO92 case
D1	1	MVAM108	2-pin varactor TO92 case, "V149"
D2	1	1N4148	glass diode, band = cathode end
VR1	1	LM78L06	3-pin 6V regulator, TO92 case
Y1	1	3.58MHz	ceramic resonator, 2-lead, "3.58G"
Magnet Wire	30"	toroid wire	red, enamel-coated magnet wire
PC Board	3" x 4"	pc board	green, ground plane bottom side

(Ant, Keyline, Phones, +V). It's up to the builder as to what kind of jacks are used, but the controls must be the correct value (10K-ohm) shown in the schematic. There is an optional controls and hardware "add-on kit" available from the NJQRP Club that conveniently provides these components at a nominal price. Section 4 of this manual overviews assembly of this add-on kit.

IC Sockets: The only other "nicety" that would be reassuring to novice builders would be to use sockets for integrated circuits U1, U4 and U5. Each of these ICs is an 8-pin DIP package, and use of a good-quality, low-profile, machined-pin socket would allow you to remove/replace the IC during troubleshooting and while making modifications further downstream. These IC sockets can be found at Radio Shack.

Test & measurement equipment: The basics needed here include a VOM (voltohmmeter) to measure DC voltage levels, check resistor component values, continuity, etc. An RF probe adaptor for your VOM is very useful in measuring RF voltages and AC signals at various points in the circuit. We've included plans in Appendix A for a simple RF probe if you don't already have one. Similarly, many builders find it quite helpful to be able to inject audio tones into their projects at various points to help determine if the circuits are working. We provide plans in Appendix B for a simple audio oscillator. Other more specialized (and expensive) pieces of equipment may also be effectively used. The first, and most versatile, is an oscilloscope (20 MHz or higher is preferable). Many guys find that an inductance-capacitance meter is invaluable in assuring correct component selection and toroid inductances ... the LC Meter - II from AADE is just fabulous and relatively inexpensive. Another extremely useful piece of equipment to have when working with RF circuits is an antenna analyzer, like the RF-1 Analyst or the MFJ-259 series. These serve well at injecting RF signals to receiver front ends, detecting RF output, and (with simple adaptors) measuring the values of inductors and capacitors.

If you don't have the basic items readily at hand, now would be a good time to make a quick trip over to your local Radio Shack to stock up on these wires, tools, solder and other standard items used in electronic kit assembly.

3. Start with the PC Board

Okay, let's get acquainted with the printed circuit board, or "pcb". If you orient the pcb with the green side up, you'll see that it contains a majority of the traces and component designators for this project. All components will be inserted from this top side of the board shown with the designators and traces on it.

You might need to hold it at just the right angle for the light to clearly show the component designators (e.g., R1, C21, etc.), and you might even need a magnifying glass to see it clearly. Again, not to worry because the PCB Layout figure on page ___ should be big enough for anyone to see things! This diagram will be your roadmap for placing the components onto the board.

(The next revision of the board will have a "silk screen" of component outlines and designators. Although this makes the board a bit more expensive to manufacture, it makes it even easier to populate the board.)

Place the pcb so you can read "FB Rx REV B N2APB (COMP. SIDE)" along the top edge. All input and output wires and cables will be soldered to the board at pads along the four edges of the board. Starting at the top left corner and proceeding counter-clockwise, you'll see pads labeled: ANT, TUNE, SPOT, PHONES, AUDIO, KEY, and +V. Most of these input/output pads have ground pads (labeled GND) close by to provide you with a convenient way to use shielded wire or twisted pair wire and have the ground line of the wire connected to the board ground plane right at the signal pad.

Mounting holes are provided in each corner of the board to allow use of standoffs to hold the board in your favorite enclosure. These holes are connected to the board ground plane, so if you are a purist, you might want to be careful to isolate the board from a metal enclosure (if used) so as to help eliminate the possibility of ground loops in your receiver ”system”. If you purchased the SOP Enclosure option, nylon spacers are provided for this purpose, thus isolating the board ground plane from the chassis ground.

Turning the board over you will notice a pervasive amount of tinned copper serving as the ground plane. This ground plane encircles most component pads, providing a good shield for the low-level signals in the circuit. You’ll be soldering almost all of the component leads to the respective pads on this bottom side of the pcb, so you’ll need to be careful not to accidentally bridge solder from the component pad being soldered over ontop the encircling ground plane. You should carefully inspect each soldered connection to check for these solder shorts and correct them right away.

4. Installing the Components

You are now ready to install and solder the parts onto the printed circuit card. Be sure to ear mark (or paper clip) the pages containing the Parts List and the PCB Layout, as you’ll often be referencing them. You’ll also notice that outlines of the component packages is indicated on the PCB Layout diagram. This will greatly assist in properly positioning the components onto the board before soldering. For example, polarity of the electrolytic capacitors is indicated on the PCB Layout diagram, shown with a “+” symbol near one of the two capacitor pads.

Be sure to check the full-page SOP Schematic in the center of this manual to help answer any questions you might have about proper positioning of components during assembly or connection of the off-board controls and jacks.

We’re going to assemble the SOP Receiver in stages, and test each stage as we go along. This will give you the best shot at having an operational receiver when assembly is complete.

With our cumulative years and years of kitbuilding experience in the NJQRP Club, we offer **some advice to make your kit construction a most pleasurable experience**. Take your time during assembly. Read through all the description in each section before starting to solder anything. As you accomplish each step within a section, put a little checkmark in the little box provided at the start of each operation in order to help you keep track of what’s been done and what’s remaining. Take a short break and give yourself a reward (coffee, chocolate, a quick QSO on 40m, etc) in between each section built up. And when you’re all done and the receiver is working as expected, do the Happy Dance throughout the home and neighborhood to let everyone know of your wondrous accomplishment!

LOCAL OSCILLATOR (LO)

We’ll start the ball rolling by building up the Local Oscillator. This is a sensible place to start in that this portion of the circuit requires no other signals for it to work. You build it and it works! ... Well, we hope it works; and if it doesn’t, we’ll hop in to find the problem right away. Let’s begin!

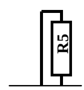
❑ **Install mixer IC “U1”** -- If you are using IC sockets (recommended) orient the socket in the holes for U1 such that the pin 1 corner of the socket is in the hole with the small square pad. If a socket is not being used, place the IC in these holes directly, ensuring that pin 1 goes into the square pad. (Pin1 is located just to the left of the detent-marking on the top side of IC package.) You might need to gently push the rows of pins closer together for them to fit into the rows of holes in the board. Holding the socket (or IC) in place, turn the board over and gently bend several of the pins outward so the part stays in place. Then, with a warmed and tinned soldering iron, solder each of the 8 pins in place. Inspect the pads when done to ensure that no solder bridges exist to ground. (Note that pin 3 will be connected to ground by small traces.)

❑ **Install IF Transformer "L1"--** Locate one of the IF transformer cans and align it in position just above and to the right of U1. Insert it into the holes - three on one side and two on the other, with the tabs of the can extending down into the larger holes. IMPORTANT: Make sure the can is sitting just a hair (approx 1/16') above the board so the metal can will not contact the trace running underneath the part. When soldered, this can will be "ground" and would short out the oscillator. Carefully turn the board over and solder the 5 transformer can leads, as well as the larger tabs. Check for solder bridges and correct if necessary.

❑ **Install Crystal Resonator "Y1"--** Locate the little dark orange resonator, marked as 3.58MG, and insert at the location for Y1, just to the left of U1. Holding it in place, turn the board over and carefully solder its leads. Using side cutters, snip off the excess lead length close to the solder joint. Check for solder bridges and correct if necessary.

❑ **Install Capacitors C13, C16, C2, C3, C4, C5, C6 and C7 --** Using the descriptions in the Parts List, locate the specified capacitors and install them at the designated positions. As you put each in place, slightly bend each component lead outward so as to hold it in place while you put the others in. When all of these caps are in, turn the board over and carefully solder each to its pad. Using side cutters, snip off the excess lead length close to the solder joint. Check for solder bridges and correct if necessary.

❑ **Install resistor "R10"--** Locate R10 and ensure that it is the right value. Carefully inspect the color coding on the resistor body to ensure you have the right one. A bright light and a magnifying glass might be helpful. Sometimes resistors lie flat against the board, but in this kit the resistors will be mounted in an upright position, as shown below.



Once again, solder and snip the excess lead length. Check for solder bridges and correct if necessary.

❑ **Install Varactor Diode "D1" --** Locate the 2-wire varactor component and notice the flat side of the package. Orient the package as shown in the PCB Layout diagram, and insert into its pad holes with the flat side toward the top of the board. Carefully spread the leads on the bottom side of the board to hold it in place, then solder the pads and snip the excess lead lengths. Check for solder bridges and correct if necessary.

❑ **Install Voltage Regulator "VR1" --** Locate the VR1 component and orient it as shown in the Layout diagram, with its flat side toward the bottom, and insert it in the three pad holes. Carefully spread the leads on the bottom side of the board to hold it in place, then solder the pads and snip the excess lead lengths. Check for solder bridges and correct if necessary.

❑ **Install R7 and R8 "jumpers"** -- Using short pieces of component leads (snipped off in prior steps), bend each over into tight "U" shapes and insert into the positions for R7 and R8 on the board. Spread the leads on the bottom of the board and solder in place. Snip off excess lead length and check for solder bridges. These "zero-ohm resistors" can be later changed to real value resistors and used to pad the tuning control R9. This will be described later in greater detail.

❑ **Install tuning potentiometer "R9"** Obtain the potentiometer you wish to use for the TUNE control (from your junk box, Radio Shack, or from the optional SOP Controls Kit). Using short lengths of hookup wire, connect the pot as shown in the PCB Layout diagram. You'll probably later use longer, shielded wire to connect this pot when installed in an enclosure.

Testing the Local Oscillator

It's time to test what you've done thus far. Carefully inspect all your work to make sure you didn't miss anything, that traces are not shorted, and that everything is soldered properly.

Connect up a 9V battery (or equivalent) to the +V and GND pads along the right side of the board. We're using a battery here to reduce problems in case of inadvertent shorts on the board ... the battery is self-current-limiting and doesn't supply tons of current like a regulated power supply when there are problems.

❑ **Check for +6V -- The first thing to do is to check to see that VR1 is working. Using your VOM, ensure that there is +6V on U1 pin 8. If you don't see this, stop right away and determine the cause. Is VR1 getting hot? (If so a shorted output line might be present.) Is the +V supply voltage making it to the VR1 input pin?**

❑ **Check for RF OSCILLATION -- Using the RF Probe (from Section 5, or equivalent) attached to your VOM, measure the RF voltage on U1 pin 7. You should see some millivolt reading to indicates that the circuit is oscillating. If you do not see a reading on the VOM, carefully check all components for proper orientation, and for solder shorts. An oscilloscope would indicate that the circuit is oscillating at approximately 3.58 MHz without jumper C in place.**

Insert a jumper between pads C-C just to the left of U1 (in a manner similar as the zero-ohm resistors R7 and R8), and measure the oscillator output on U1 pin 4. If you were using an oscilloscope, you would detect oscillation occurring at approximately 7.1 MHz, because of the "doubling" effect of U1. A small reading should also be seen on this pin with the RF Probe and VOM. The signal would "peak" at some point as you adjust the slug inside the IF can L1, indicating that the L1-C7 tank circuit is properly adjusted for 40m operation.

LOCAL OSCILLATOR BUFFER

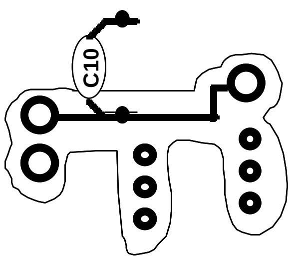
We'll next assemble and test the two-transistor buffer amplifier for the LO.

❑ **Install transistors "Q1" and "Q2"** Locate the two 2N2222A transistors and put them each into their 3-hole positions on the board with their flat sides facing toward the right. Slightly spread the component leads on the bottom side to hole them in place, and carefully solder the leads. Trim the excess lead length and inspect the pads for solder bridges.

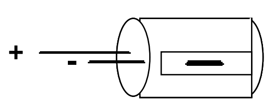
❑ **Install resistors R2, R3, R4 and R30 --** Locate these resistors by carefully studying the color bands, per the Parts List line descriptions. Install them in the "upright" manner previously described for R10. Gently spread the leads on the bottom of the board and solder them to the pads. Clip off excess lead length and inspect for solder bridges.

❑ **Install capacitors C8, C9 and C12** These caps are all the same. Install them through their respective pads on the pcb, spread their leads slighty on the bottom side of the board, and solder each one carefully. Snip off excess lead length and inspect for solder bridges.

❑ **Install capacitor C10 --** This capacitor actually does not have a pair of holes on the board (a slight oversight on this version of the pcb), so you'll install in on the bottom side of the board. Carefully locate the pad for R30 and its corresponding trace going over to T1. You'll solder one side of the 0.1uF C10 to this trace, and the other side of the cap to ground. See the diagram below for reference.

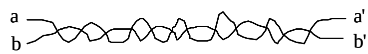


❑ **Install C37 electrolytic cap --** This capacitor also has something screwy about it ... it goes into the holes marked C14. But curiously, there are two C14 positions marked on the board (another minor error in the pcb) ... our 47uF C14 electrolytic goes into the position marked with a round symbol in the Parts Layout diagram. Make sure the positive side of the cap goes into the hole marked with a + symbol. (The positive lead is the longer one on the side <u>opposite</u> the lead marked with the "-" symbol on the side of the cap.)



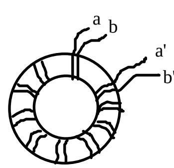
❑ **Winding and installing T1 --** This is a fun part of the project that some homebrewers worry about. There's actually nothing very hard in winding a few turns of wire through the toroid core, and we'll take you through it in a step-by-step manner.

T1 is a "bifilar-wound" inductor on a toroid core, meaning that you'll be combining two magnet wires together and winding them at the same time. Measure off two 7-inch pieces of red magnet wire. These wires should be twisted together as illustrated below.



(Just twist the wires loosely together by hand ... they don't have to be very tightly wound at all.)

You will then wind the combined, twisted wire pair around a black FT37-43 toroid core. See the diagram below for proper connection of the four leads:

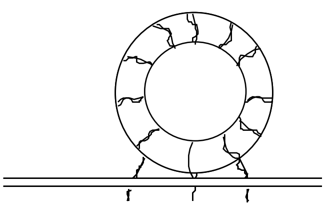


You'll next need to strip off the red enamel coating from the transformer leads. The heat strippable magnetic wire being used requires no scraping to clear the red insulation off the leads being soldered to the PCB pads. Once the wires of each inductor are trimmed to the right length (approximately 3/8" extending down past the bottom of the core), tin the ends of the wires by doing the following.

Using a good hot soldering iron, place the tip under the end of the wire to be tinned and add a little solder so that there is a small pool of molten solder and flux on top of the iron with the wire in the pool. After several seconds, the insulation will melt away and the wire will be tinned where it is in contact with the iron. Continue moving the iron slowly toward the toroid core adding solder as you go, until the wire is tinned within 1/16 inch or so of the core. Repeat the procedure for the other 3 leads and brush off any carbon residue from the ends of the wires.

You'll next connect two of the leads together to form the center tap of the transformer. When two wires of the same color are twisted and wound together on a toroid, it's difficult to know which ends to connect together to form the center tap. You will need to use an ohmmeter to determine proper ends. As indicated in the diagram on the previous page, one of the wires, a is the start and a' is the end. On the other wire, b is the start and b' is the end. **You should twist wires a' and b together to form the center tap.**

Insert T1 into position on the circuit board as shown below. The centertap is in between the two end leads.



Trim the leads to the correct length, prepare the ends with the soldering iron again, and solder them into their respective pads on the pcb.

We've found that over 90% of most kit assembly problems come about because of improper tinning and soldering of toroid inductor leads, so please be sure that you carefully followed the instructions in this section.

❑ **Install jumper J2 --** Solder a short length of a discarded component lead in place between pads 1 and 2 at jumper B. This is just to the right of IF can L1. This jumper will select the 40m signal from the LO.

Testing the LO Buffer

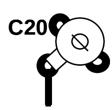
Now it's time to test the LO Buffer you just installed. Apply power again and using your RF Probe and VOM, measure some signal on the emitter of Q1 (the lowest of the 3 transistor pads). Next measure some significantly greater voltage on the top end of capacitor C12 at the Buffer output.

If you do not see signal readings at these locations, go back and check for solder bridges and proper component placement. Your receiver will not operate without sufficient "LO injection" to the mixer stage.

RF PRE-AMP STAGE

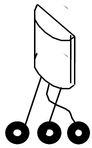
We'll next assemble and test the RF front end.

❑ **Install trimmer cap C20 --** Locate this trimmer cap ... it's a larger, 3-terminal component with a slotted adjustment on the top. Orient it in the 3 holes of the pcb such that its third "extended" tab is in the bottom-left pad. See below.



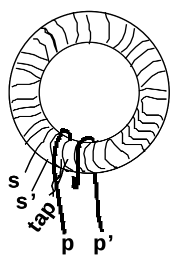
❑ **Install IF can T2 --** In a manner similar to what was done for the L1 IF can, install another at position T2. Again, be sure not to have T1 sitting flat on the pcb ... you must solder it in place such that it sits about 1/6" above the board surface.

❑ **Install FET Q3 --** You'll need to carefully swap the right two legs of this transistor when inserting it to the pads on the board. (There is a routing error on the board.) Reference the following diagram.



❑ **Install C15 and R11 --** Locate these two components and install them in position below trimmer cap C20.

❑ **Construct and install T1 --** Okay, it's time to make yourself another transformer! This one is also pretty easy to do. Measure off about 20" of red magnet wire and wind 36 turns around the red toroid core. But at the second turn extend a little extra past the core body and twist it together to form a tap. Continue winding the rest of the 36 turns. This will be the secondary winding of the transformer. Measure off another length of red magnet wire, this time only about 3" and put 2 turns around the core overlapping the lower two windings of the primary, right up to the tap you created.



Tin the leads as done before and insert onto the board at position T1. The two (short winding) primary ends will go into the two pads on the left, and the three secondary leads (two ends and the centertap in the middle) will go into the pads on the right side of the component. Solder all 5 leads in place, ensuring that no enameled wire is in the solder junctions.

Testing the RF PRE-AMP

Connect a temporary wire from the output of the LO Buffer (top side of C12) to the ANT pad at the input of the RF front end. This jumper will supply a known-good frequency that we can use to test the front end.

Apply power to the board and using the RF Probe measure the signal on the T1 secondary winding, located at the left side of R11. While watching the VOM reading, adjust the trimmer cap for maximum readings.

Move the RF Probe over to the secondary of T3 (can be probed on pad 3 of U3). Again while watching the VOM readings, peak the IF can T2 using a small screwdriver.

If you did not see noticeable signals at these points, or if the circuits did not peak as described, go back and check for proper components and solder bridges. Be especially careful in double-checking the construction job on T1.

Remove the temporary jumper from the LO Buffer.

MIXER STAGE

We'll next put the mixer chip U3 in position.

❑ **Install mixer U3 --** This integrated circuit is a "surface mount chip", but it's a rather large one and relatively simple to put in place. We were going to provide this part pre-soldered on the board but found it to be so straightforward that we felt most kit builders could easily accomplish it.

First tin pad 6 of U3 on the board. This is the top right pad in the array of 6 pads for U3. By tinning this pad, you are depositing a small ball of solder to the pad that will be re-heated with the IC in place, thus holding it down while you solder the other pins.

Next, locate the IC in the black plastic case with the clear top. Carefully peel back the top cellophane membrane and the IC will fall out onto the table.

Pick up the IC with your fingers or with your needle nose pliers and place over the pads on the board with pin 1 in the top left corner. (Pin 1 is denoted by a small dot in the tan plastic package.)

Making sure that the pins of the IC are properly positioned over their respective pads, carefully touch pad 6 with the tip of the soldering iron and "reflow" the solder while gently pushing the IC down onto the pad. With only a small amount of luck, the IC will be held firmly in place by that one pin, with the other 5 pins clearly oriented over their pads. (If not, reheat pad 6 and adjust the IC to be correctly positioned.)

Go to the each remaining pin and solder it to the pad beneath it. The pins have about the standard 0.1" lead separation, so this operation should be a piece of cake. Carefully inspect the pads for shorts.

There's nothing to easily test at this point, so we'll go on to add the audio filter and amplifier stages next.

AUDIO GAIN STAGE

You'll next assemble the Audio gain stage: Q4. (You'll actually assemble all four stages of the audio chain before we test them.)

❑ **Install resistors R14, R31, R19 --** Note that R31 is installed at position R29 along the top of the board. (A small typo on the board designator.)

❑ **Install capacitors C19, C36, C21, C34 --** Be careful to observe polarity of the electrolytic capacitors. The positive pad is indicated with a '+' on the board. Leave the left end of C19 up in the air, unconnected to the left pad. (We'll use this as a signal injection point during test in a few moments.)

❑ **Install transistor Q4 --** Note this package orientation on the Parts Layout diagram.

LOW PASS FILTER STAGE

You'll next assemble the Low Pass Filter stage.

❑ **Install resistors R15, R16, R17, R18, R20, R21 --** Nothing tricky here.

❑ **Install capacitors C22, C23, C24, C25 --** Nothing tricky here either except to ensure that you grab the correct caps for C23 and C25. These are the dark brown mylar .01uF caps.

❑ **Install integrated circuit U4 --** This is pretty straightforward, as long as you ensure you get pin 1 in the square pad. Again, you might want to use a socket to help with later debug or modifications.

MUTING STAGE

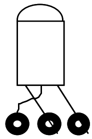
You'll next assemble the Muting stag: Q5.

❑ **Install resistors R22, R23 --** Nothing tricky here.

❑ **Install capacitors C27, C28 --** Nothing tricky here either.

❑ **Install diode D2 --** Just be sure to orient this glass-packaged part with its cathode (banded end) into the lower of the two pads.

❑ **Install FET Q5 --** Here again you'll need to carefully swap a couple of the legs (this time the left two) of this transistor when inserting it to the pads on the board. (There is a routing error on the board.) Reference the following diagram:



❑ **Install pot R24 --** Here's the first instance that you'll have to install an off-board component. Make sure that you select the right pot (i.e., the one with the extra two tabs on it) and wire it as shown in the Parts Layout diagram.

You'll now install the final stage of the receiver: the Audio Amp.

❑ **Install resistors R13, R25, R26, R27, R28 --** Nothing tricky here.

❑ **Install capacitors C29, C30, C31, C32, C33 --** Nothing tricky here either except to ensure that you are careful to install the electrolytics with the proper polarity/orientation.

❑ **Install integrated circuit U5 --** This is pretty straightforward, as long as you ensure you get pin 1 in the square pad. Again, you might want to use a socket to help with later debug or modifications.

OTHER OFF_BOARD PARTS

For the final installation of parts, you'll need to connect up the remaining off-board controls and jacks. This will enable you to fully test, and then later operate the receiver. Refer to the Parts Layout diagram in the center of this manual for graphic detail.

❑ **Connect J3 to PHONES--** Using your selected hookup wire or cables, wire the 3.5mm audio output jack to the PHONES pads.

❑ **Connect J1 to KEY--** Using your selected hookup wire or cables, wire the RCA phono jack to the keyline input at pads KEY.

❑ **Connect J4 to +V--** Using your selected hookup wire or cables, wire the coaxial dc power jack to the +V and GND pads.

❑ **Connect R9 to TUNE--** Using your selected hookup wire or cables, wire the remaining pot to the tuning pads TUNE.

❑ **Connect J2 to ANT--** Using your selected hookup wire or cables, wire the RCA phono jack to the antenna input at pads ANT.

Testing and Troubleshooting the Audio Chain

The figure on the next page represents a higher-level view of the SOP audio section. Debugging process starts at its output then proceeds backward toward the input. Going at it this way lets you hear what's going. In the interest of brevity (and saving trees) we'll show the debug steps and results in summary form.

1) Connect power - no smoke! Use a 9 volt alkaline battery to minimize damage if something was not right in the circuit.

2) Check pin 6 of U5- should be about 1/2 supply voltage.

3) Using an audio signal source (like the one in Section 5), apply audio to top of volume control. A tone should be audible in the speaker or headphones.

4) Apply audio to input of the mute switch. You should hear a weak tone.

5) Check the dc bias on the output of low pass filter (U4 pin 7). Should be 1/2 supply voltage.

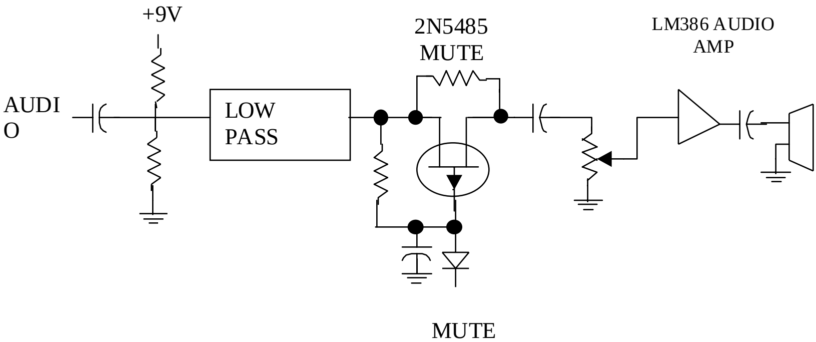
6) Check bias divider on the low pass filter input. Should be 1/2 supply voltage.

7) Apply the audio source to the input of the low pass filter (junc-

tion of R15 and R16). Should hear tone in loudspeaker and volume control should work normally.

8) Ground the KEY input pad with a clip lead. The tone should disappear.

9) Connect the audio signal source to the input of the entire audio chain (i.e., at the lifted end of C19 in the Audio Gain stage). You should hear a very loud signal, and it can still be adjusted using the R24 level control.



FINAL ASSEMBLY

The only assembly operation needed at this point is connection of the Mixer stage output to the Audio Gain stage input.

❑ **Connect left side of C19 to its pad**

THE SOP KIT IS COMPLETE! ... NOW WHAT???

CONNECT AN ANTENNA!

When you connect a proper antenna to the ANT input pads (or the J1 jack), you should hear a significant amount of activity as you tune through the SOP's frequency range (depending on the time of day and band conditions).

WHERE THE HECK ARE YOU?

Now would be a good time to find a known-frequency signal on the band, or use another, calibrated receiver in order to determine the specific frequency coverage of your SOP Receiver. Finding the band edges and marking your front panel accordingly would be real useful when it comes time to operate the receiver.

Of course, if you have purchased and installed the AFA option, you'll hear the exact frequency of operation come pounding out at you in Morse code from the on-board frequency counting circuit of the PIC.

PUTTING THE SOP ON 80M

Putting the SOP receiver on 80m is relatively easy. You'll need to add a few components, and change a couple of jumper settings on the board, but there's nothing major. Just follow the notations on the schematic. These minor mods are primarily in the Local Oscillator stage and the RF Pre-Amp stage.

GETTING A WIDER SWING

You'll notice that the TUNE pot is connected to the regulated 6 volt regulator. This was done to provide a nice, clean, stable signal for the voltage presented to the varicap tuning element. Without such regulation, the set frequency would drift when the power source (e.g., battery) drooped, dipped, glitched, sagged or otherwise varied due to being weak.

If one were to instead provide the higher +V unregulated voltage to the top of the TUNE pot (and its padding resistors), while at the same time provide a <u>stable</u> +V source, a wider swing in frequencies would be achievable.

Like we said up front, the SOP Receiver is an ideal experimenter's platform!

FUTURE MODIFICATIONS

There are many mods coming down the pike for this project.

- taking the SOP to other HF bands

- improving the gain and sensitivity

- using alternate designs for some stages

- provide multi-band operation with "daughter boards"

- providing a wider range in the LO

- etc.

There will be some "application notes" published and distributed to the SOP owners, allowing you to have some greater visibility into the fun times ahead. Lots of plans!

IF YOU HAVE PROBLEMS ...

That is, if you have problems in getting your SOP to operate, feel free to contact us by email or US Mail and we'll do what we can to help.

For a nominal, flat fee, we will provide a repair service that will guarantee you getting back a working SOP (as long as there was no major component or pcb damage). Contact us for details on this last resort option.

THANK YOU!

Thanks a whole bunch for buying the SOP Receiver Kit and experiencing the fun of homebrewing your own rig with us. The NJQRP Club put in a tremendous number of hours designing and putting this kitting project together for QRPers all over the world, and we sure hope you are pleased with the experience.

As we stated up front, the SOP is an experimenter's platform for learning about basic receiver operation, straightforward homebrew kit building, and when all complete it's a flexible little receiver for portable use. And if you've come this far in the manual, you surely have gained a whole lot of this good experience along the way!

72/73,

--George Heron, N2APB
n2apb@amsat.org

and

-- Joe Everhart, N2CX
n2cx@voicenet.ccom

And be sure to visit the SOP Receiver web page at the NJQRP Club website:

http://www.njqrp.org/sop

Section 2: Audio Frequency Annunciator ("AFA") Option

Assembly Manual

We were able to design in a great feature, in cooperation with Dave Benson, NN1G, to provide the Small Wonder Labs "Freq-Mite" audio frequency annunciator PIC chip and associated components as an optionally-purchased add-on to the SOP Receiver. This option provides an incredibly useful feature of an "audio frequency dial" that delivers the operating frequency of the receiver in Morse code through the audio chain of the receiver. With this option installed on the base PC board, the operator merely presses a front panel "spot" button to hear the operating frequency delivered in Morse code through the headphones or speaker. It's always accurate and there's no front panel space necessary to know where you are in the band. NN1G was very pleased to be able to contribute his Freq-Mite functionality at a very attractive price to those purchasing SOP kits.

Circuit Description

This circuit from Small Wonder Labs is really quite simple, yet clever. Referring to the schematic in Figure 2, a PIC16C621 microcontroller is programmed as a frequency counter.

Transistor Q6 buffers, amplifies and conditions the signal coming from the Local Oscillator and presents it to an input port of the PIC. The PIC then counts the number of low-to-high transitions of the signal during a specified period of time, and *voila*, the frequency is known.

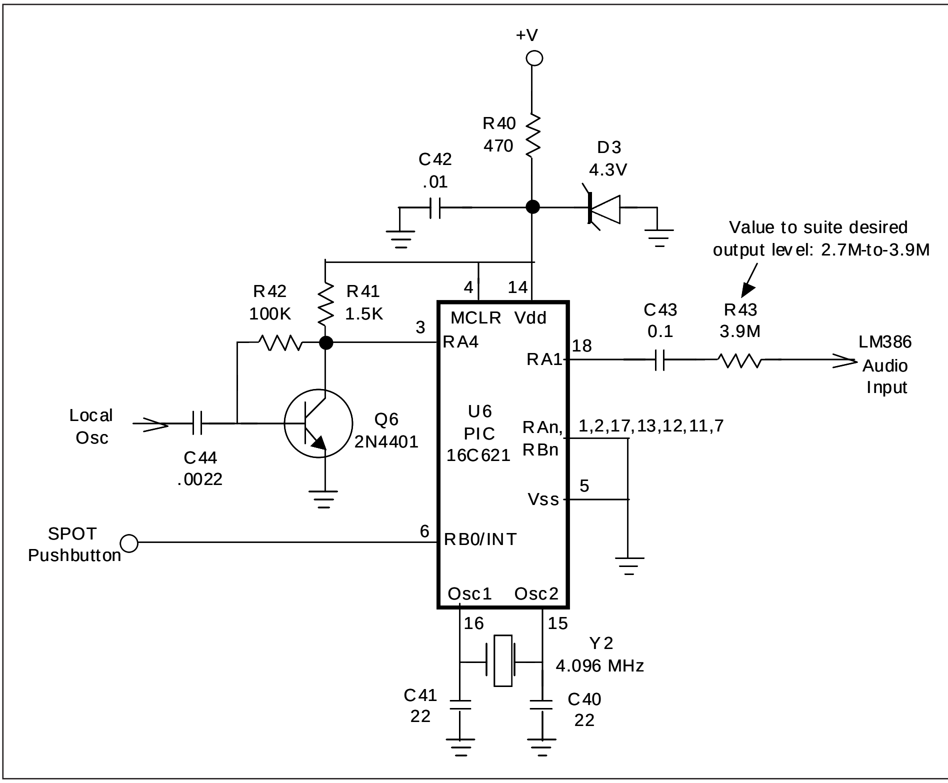
But wait, that's only half the beauty of this circuit. The other purpose of the PIC is to output the computed frequency in Morse code!

When commanded by the operator pressing the SPOT pushbutton, an output bit is wiggled at approximately 600 Hertz, and then modulated on-and-off to send the Morse code equivalent of the measured frequency through capacitor C43 and on into the audio amplifier.

Resistor R43 is used to reduce the amplitude somewhat so the Morse frequency doesn't wake the next door neighbors. (This is a high-value resistor, and you may need to adjust it up or down to suite your particular needs. Just substitute a higher or lower valued resistor if desired.)

Because this high-speed digital circuit is sitting pretty close to low-level RF circuitry, we took some special precautions to reduce electrical noise and cross-coupling.

AFA SCHEMATIC



The circuit is operated at the lower end of its voltage range, as determined by the 4.3 volt zener diode. This lower operating voltage produces significantly less energy in its digital waveforms and thus minimizes interferences to the receiver.

The PIC circuitry is tucked into a corner of the pcb where its effect would be least noticeable and where cross-coupling would be minimized.

And finally, the ground plane for this circuit on the bottom of the board is isolated, except for one joining point, thus preventing digital return paths from flowing past or through the RF or audio circuits.

In earlier prototypes of the SOP, we had wide ground traces on top the board defining the perimeter of the microcontroller circuit, allowing for a small pcb-cage to be soldered in place to further isolate digital RF coupling. But we found that this precaution was unnecessary and it was not included in rev B of the pcb.

For those who have previously used the Small Wonder Labs' FreqMite, you'll know that an "offset" can be set into the PIC's computation, thus yielding an adder to the base frequency being measured. This allows the FreqMite to be used to measure signals in multi-mixing stage superhet designs where the VFO frequency is not the actual "displayed" frequency. But here in the direct-conversion SOP, the Local Oscillator signal <u>is</u> the operating frequency and the PIC is configured through pins 1, 2, 17, 13, 12 11

AFA PARTS LIST

Designator	QTY	Value	DESCRIPTION
C40,41	2	22pF	220J
C42	1	0.01	brown, 103
C43	1	0.1	brown, 104
C44	1	0.0022	222
D4	1	4.3V zener diode	glass, band at cathode
Q6	1	2N4401	3-pin TO92 case
R40	1	470	yellow-violet-brown
R41	1	1.5K	red-green-red
R42	1	100K	brown-black-yellow
R43	1	3.9M	orange-white-green
U6	1	PIC	18-pin DIP IC
Y2	1	4.096 MHz crystal	2-wire metal case

and 7 to provide a Morse readout of the actual frequency being measured. Thus, when the LO is tuned to 7.040 MHz, the AFA will output “7040” to the phones/ speaker in Morse code when the SPOT pushbutton is depressed.

PARTS INVENTORY

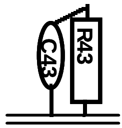
Once again, the first thing to do is to check the contents of the separate bag containing the AFA parts. (This bag has a label in it indicating that it’s the AFA parts bag.) This accessory only contains the parts required for the AFA feature, and the components will be placed on the main SOP pcb.

❑ **Install PIC uC U6 --** Locate the integrated circuit in the black antistatic pad and insert it in the holes for U6 on the pcb. Be very careful to ensure that pin 1 of the IC (at the left side of the end with a detent in the plastic body) goes into the square pad. (The rest of the pads of the IC are round.) It would be a good thing to solder in an 18-pin socket first, instead of soldering the IC directly in place. This would allow you to remove/replace/re-use the PIC in other design should you need to later on.

❑ **Install crystal Y2 --** Locate the low-profile 4.096 MHz crystal and put it in place in the pcb at Y2. Do not have it sitting flat on the pcb surface ... you should solder it in place such that it is raised about 1/6” above the board, ensuring that its metal case is not touching the traces beneath it.

❑ **Install zener diode D3 --** Locate this glass diode and insert it in an upright position (like you do for the resistors) at position D3. Be sure to orient the diode such that the cathode (i.e., the banded end) is the end connecting to R4. It would be best to have this cathode end be the one sticking up with its lead folded over. This way you’ll more easily be able to probe the circuit later for proper voltages.

❑ **Install C43 and R43 --** You’ll be putting these two components into “the same” location on the pcb. The left leg of C43 will be inserted in position just below Q6, and its other leg will be pointing straight up. Thus the component will be leaning to the left. R43 should be inserted to the right hole of C43 on the pcb, and the other end which is also sticking straight up should be soldered to the up-pointed leg of C43. Thus, these two components will be in series. See the figure on the next page for reference.



❑ **Install remaining components: C40, C41, C42, C44, R40, R41, R42, and Q6.**

❑ **Clip leads & inspects for solder bridges --** As always, trim all leads down after soldering and carefully inspect the bottom side of the board for solder bridges.

❑ **Install Jumper A-A --** The last thing you'll need to do is to install a jumper wire from pad 'A' at the middle-top of the board (just above U3) to the other pad 'A' just above pin 1 on U6. For neatness, you could install this jumper on the bottom side of the board - it doesn't matter. This jumper brings the LO signal down to the PIC.

OPERATION

Upon power-up, the PIC sends an 'S ?' if you press the SPOT pushbutton switch within about two seconds, the chip switches to a higher speed (26 WPM) readout rate. It acknowledges this entry with an 'R'. If you do nothing the chip maintains its default 13 WPM readout speed.

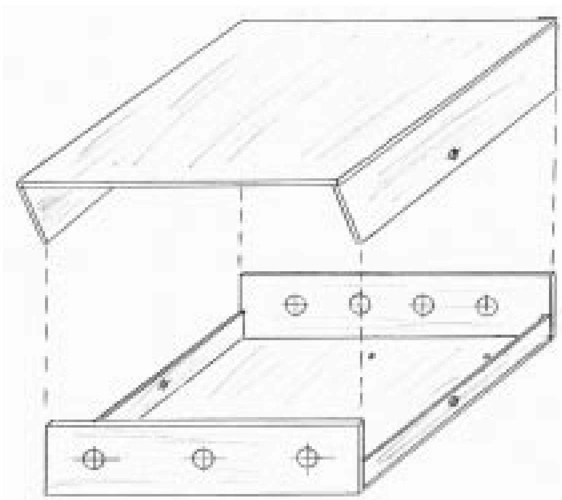
THANKS

Many thanks are due to Dave Benson, NN1G and Small Wonder Labs for allowing use of his FreqMite circuit, for providing us the pre-programmed PIC microcontrollers at an attractive discount, and for advising us on implementation of the design into the SOP Receiver. Dave is an active contributor to NJQRP design and project teams.

Section 3:
SOP Enclosure Option

Assembly Manual

This optionally-purchase Enclosure Kit is the ideal homebrew case for your SOP Receiver. Consisting of 8 precision-cut double-sided copper clad boards and assorted hardware, this kit can be assembled in less than 30 minutes and can be finished to the builder's liking or used with its *au natural* copper look.



The SOP Enclosure is a *homebrew* one - one that you will assemble from its most basic pieces. Don't worry though, you won't have to go through the hassles of scribing, bending and cutting metal; we've already done all that work for you!

This do-it-yourself case is a little different than most that you may be familiar with. It is not made of aluminum or steel or even plastic. It's fabricated from copper-clad glass-epoxy printed circuit board material. Being homebrew, it is a fitting companion to the SOP Receiver kit and adds to the enjoyment of your project. All you need to do is carefully follow the directions in this manual and you will find it easy to make a very attractive case. No drilling, sawing or punching is necessary; only a little soldering is needed, and perhaps some touchup with a file. In fact, once you built it you may even be inspired to make your own pc board case for the next homebrew project you have lined up.

Cases made of pc board stock have lots of advantages over other material. The raw material is relatively common and easy to machine. It is lightweight and, in small cases, quite strong. The copper surface provides a good continuous electrical shield for electronic circuits inside while presenting an outer skin that can be left "natural" or painted in any color you want.

If you want a short treatise on this construction method check out Joe's Quickie No. 34 in the Information Exchange column in the April 2000 issue of the QRP Quarterly.

The SOP Enclosure Kit consists of eight pieces of printed circuit stock which will be soldered to form the top and bottom of a cabinet. (Skip ahead to Figure 2 to see a set of diagrams depicting all supplied parts.) The case top is an open "U" which sits over another open "U" formed by the case bottom. The bottom also has internal side rails which lend strength to the front and rear panels, and to which the top is connected via sheet metal screws.

Also provided with the kit are the mounting hardware, knobs and jacks needed for the SOP Receiver. Clear acetate sheet overlays are even supplied to be used as attractive labels for the front and rear panels.

PACKAGE CONTENTS

It's a good idea first off to identify the kit parts and make sure that they are all present. The following list details them and identifies the cabinet parts by letter. Match them to Figure 2 on the next page and check them off in the list as you identify the contents of the parts bags.

Parts list -- Bag 1

a Top cover panel
b, c Two top cover side pieces
d Bottom plate
e Front panel
f Rear panel
g, h Two side rails
1 front panel clear acetate label
1 rear panel clear acetate label
1 right angle alignment gauge (approx. 1 inch square)

Parts list -- Bag 2

4 3/8" Nylon standoffs
10 #4 x 1/4" sheet metal screws
4 Self-adhesive rubber feet

General Guidelines

Before we start construction, let's start off with a couple of tips in case you are assembling a pc board enclosure for the first time.

You will need only a couple of small tools and supplies that are probably already at hand. They will be discussed in the text but here is a sample list:

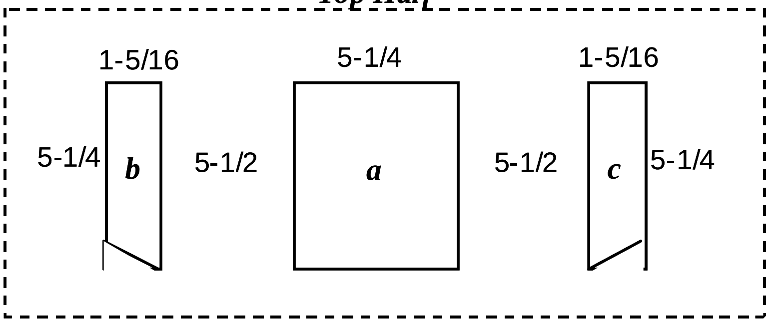
- Soldering iron
- Non-acid solder
- Scotchbrite or other soap-free non-abrasive pad
- Flat file
- Alcohol or other non-water-based solvent
- Clear adhesive to attach panel labels
- Mechanic's square (optional)
- Rosin core flux (optional)
- Paint (optional but an excellent choice for durability is Rustoleum)
- Clear spray (optional though Krylon clear protects copper from corrosion)

This process has a lot in common with assembly using surface mount components. While ordinarily solder joints should not be relied on for mechanical strength, good joints are the only source of strength for surface mounting and for pc-board cases. So every effort must be made to produce good joints.

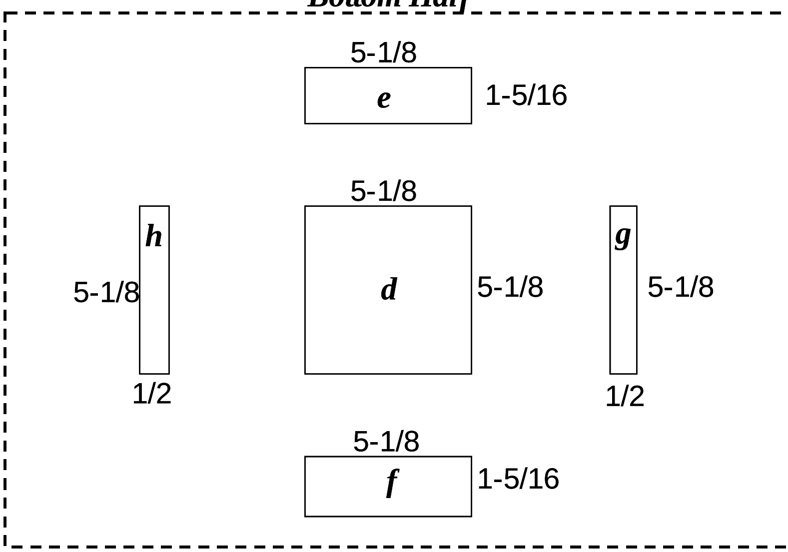
High quality solder connections begin with clean surfaces. While the copper-clad board pieces in this kit are shipped in good condition, the copper will form an oxide film over time and normal handling can contaminate the surfaces with finger oils and slight corrosion. A good practice to follow is vigorous scrubbing with a gentle abrasive such as a Scotchbrite pad followed by cleaning with rubbing alcohol or other non-aque-

Figure 2: KIT CONTENTS

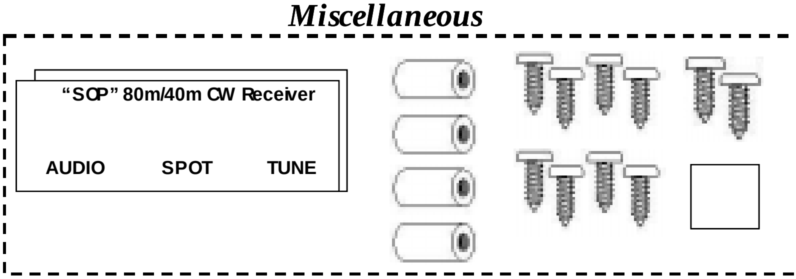
Top Half



Bottom Half



Miscellaneous



ous solvent. Once cleaned this way the copper should stay clean enough for several days so long as you keep it dry and don't handle it any more than necessary during assembly.

As with any soldering operation, the **proper iron and amount of heat** should be used. Generally any soldering iron suitable for electronics use with a 35 to 50 watt rating is fine. 600-deg to 800-deg F should do the trick if you are using a temperature-controlled solder station. Irons with less wattage will not have enough heat capacity to produce good joints while higher-wattage ones can overheat the adhesive that holds the copper onto the printed board base layer causing delamination. A flat chisel tip is preferred over a cylindrical or conical one so that enough heat can be transferred to the solder as a running seam is made.

A good quality **rosin core solder** is fine though some of the more modern "no-wash" type mixes are good too. Do <u>not</u> use acid core solder as it will destroy your soldering iron tip and eventually lead to severe corrosion of the copper surface of the enclosure you are building.

Since you want the top and bottom of your enclosure to fit snugly without unsightly gaps or seams, **alignment** of the pieces while you are soldering is quite important. Copper clad board boxes have their individual sides joined together at right angles. Accuracy in aligning each piece accurately is what makes the enclosure go together properly and look attractive.

A simple **alignment gauge** is included with the enclosure kit to help you make good right-angle joints. It is accurately cut on at least one corner for perpendicularity (love those big words!). It is a good idea to check all four corners against a machinists or carpenter's square to see which one is best. You may also care to use a file to "knock off" 1/8 or so on the exact corner so that it will fit flush against a corner of your box after soldering to ensure "squareness" in the finished product. This will also let you know at as glance which is the best corner! Using the gauge will be described in the detailed assembly steps so that the enclosure pieces can easily be aligned the correct way.

Finally, you need a **good clean uncluttered work surface!** That should be obvious, but it is important. When working with the pc board material, a fairly benign surface is necessary to keep the copper on the pc board material clean and scratch-free. A soft cloth or towel laid out on the table surface is best but in a pinch you can use cardboard or newspaper. A side benefit of using an overlay on you work bench is that you will help protect that surface as well as that of the project you are working on. Eliminating clutter helps you keep track of all the parts.

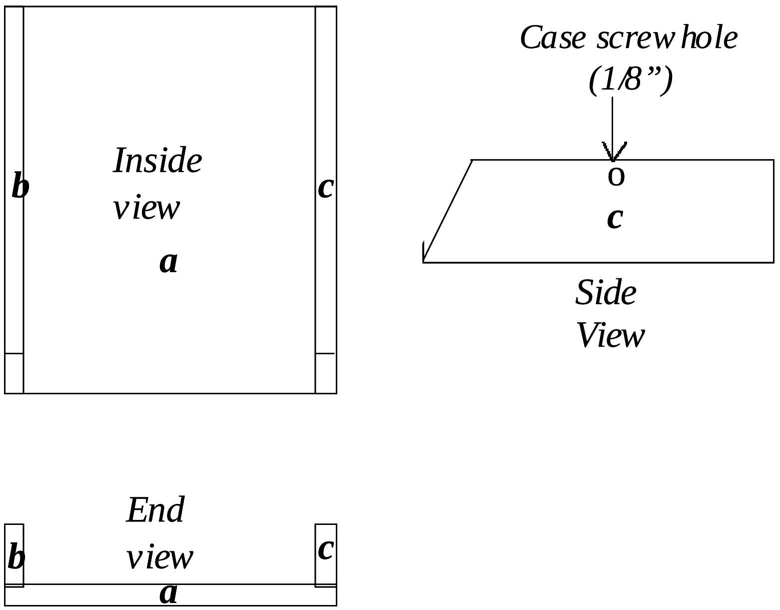
DETAILED DIRECTIONS

Ok, the preliminaries are out of the way so let's get started! Heat up your iron kick the kids and pets out of your workshop and turn on your rig so that you can be soothed by the sound of cw on 7040 as you work.

If you have already inventoried the parts, *great*! If not please go back to the **Package Contents** section on page 3 and do so. Select pieces *a*, *b* and *c* to build the case top first.

Please refer to "Figure 3: Top Half Assembly" drawing on the next page. Note from the end view that the two side pieces *b* and *c* will be assembled to the **inside** of top cover panel *a* rather than alongside it. If the sides are soldered alongside the cover, the cover will not fit on the bottom properly. Also observe that the top cover is longer than it is wide so that the longer side of pieces *b* and *c* will be against the long dimension of *a*. Be aware

Figure 3: Top Half Assembly



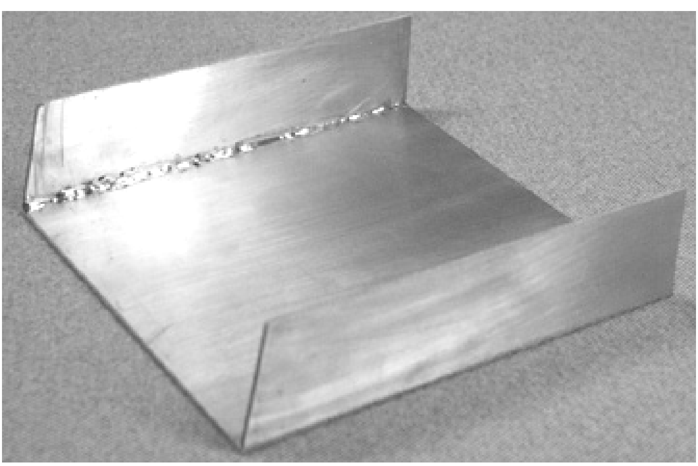


FIG 5: Completed Top Half

Figure 4: Use of Alignment Gauge

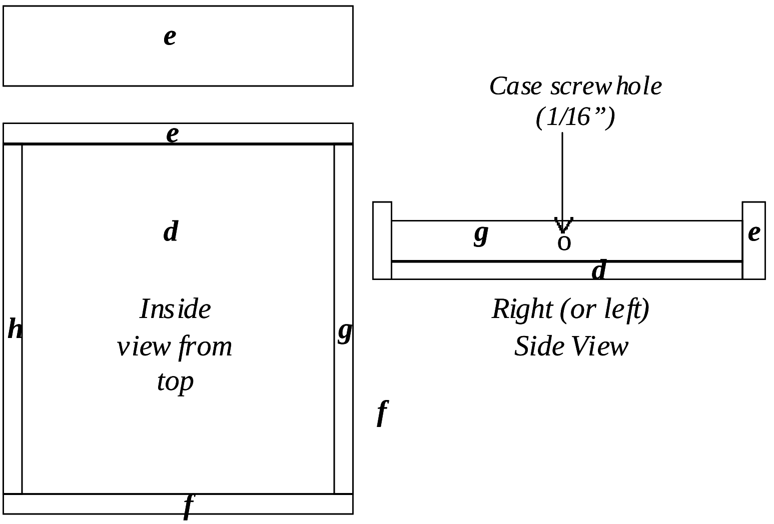
that the side view is shown upside-down. When the case is finally assembled the "overhang" will be at the top of the enclosure front panel. All right, that's enough visualization.

Lay piece *a* on the workbench and set one of the side pieces (either one) on top of it along one side. Line the side piece up flush with the edge of the top cover, using the alignment gauge to ensure that the two pieces are lined up at right angles. The photo in Figure 4 demonstrates how this is done. *[Note: The one photo we have of using the alignment gauge technique actually shows its use in soldering the side rails to the base (described in the next pages); but the technique is the same as used in soldering the top half sides.]* It may seem awkward at first since you have only two hands but you have to do it with only one hand since you will need the other for soldering!

Carefully load some solder on your soldering iron top and place a solder "tack" in the middle of the seam between the side and top pieces to hold it in place. Now add a couple more tack joints about ½ inch or so from the front and back edges while holding the pieces together with the alignment gauge. When complete, the side and top covers should be accurately at right angles. Check along the seam from front-to-back and carefully redo the tack joints as needed to get the proper alignment.

Once you are sure of correct alignment you can **complete the solder seam**. You have two choices here. If you are cautious and don't want extra bother, don't run the seam all the way out to the front and rear edges of the cover. That is, keep them back about 5/16 inch from the front face of the side pieces and about 1/8 inch from the rear face. If you are adventurous, run the seam all the way out. This will require a little filing on the bottom cover later, but will give you a somewhat stronger box. To make the continuous

Figure 6: Bottom Half Assembly



seam, start at the back end of the seam and slowly add solder to the joint as you move the soldering iron along the seam. Take care not to disturb the right-angle that the cover and side piece make with each other.

Now that you are an accomplished expert, repeat the above process with the other side piece. You now have a completed top cover! It should look like the photo in Figure 5.

Bottom Half Assembly

Next you will assemble the bottom cover. Begin by identifying bottom cover *d* front panel *f* and rear panel *e*. You can tell the front and rear covers apart by noting that the front panel has three holes while there are four on the rear.

Examine Figure 6, the Bottom Half Assembly drawing, as illustrated on the next page. It shows that the front and rear panels mount on the outside edges of the bottom cover and the side rails go between the end panels inside the whole shebang.

We'll first solder both side rails (*parts g and h*) to the base plate (*part d*). By placing the smaller, less-critical side rails in place first, the later step of soldering the front and rear panels in place will be made much easier.

Using the alignment gauge exactly as shown previously in figure 4, solder one side rail in place using the *tack-first, seam-solder second* technique already described. The side rails are identical and can be used interchangeably. Make sure the edges of the side rail are even with the front and rear edges of the base plate, as this flush right angle edge will be the mounting datum for the front and rear panels. Don't worry if the side rail is not exactly even with the base plate, as you'll have a chance to "square things up" with the flat file before putting the front/rear panels in place.

Next, solder the second side rail in place in the same manner as you did for the first one. Now is the time to file the edges to ensure flush perpendicularity.

You next **solder the front panel** *(part f)* onto the bottom plate (*part d)*. Ensure that the front panel is aligned square and flush with the base/rail assembly. Hold it in place with the claw grasp of one hand as you tack solder several points along the joint. Use the same solder-tack technique as you did for the top cover then run a continuous joint in the seam after ensuring proper alignment.

Repeat the same process to **solder the rear panel** *e* to the bottom plate *d*.

You're coming along nicely now! Your completed bottom assembly should look something like that shown in Figure 7.

It's a good idea at this point to **fit the top cover over the bottom** to be sure that everything fits. It should be a snug fit but not so tight that you will stress the top cover when it is installed. If you ran the solder bead all the way out on the top cover you will find that it interferes with the edges of the front and rear panels. Never fear – we'll take care of that next.

If you have continuous solder beads out to the front and rear panels on your top cover it is now necessary to perform a small amount of cosmetic surgery. File off the sharp corers on the front and rear panels so to clear the top-cover bead. See Figure 8. You only have to "knock off" about 1/16 inch so don't get carried away!

When you are sure that the covers nest together properly, put in the two sheet metal screws that hold the top and bottom halves together. The first time that you screw one of the screws into place, it will be a hard turn on the screwdriver.

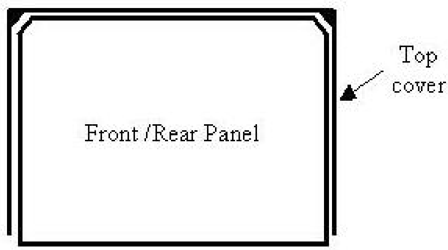


Figure 8:
Fitting Top Cover

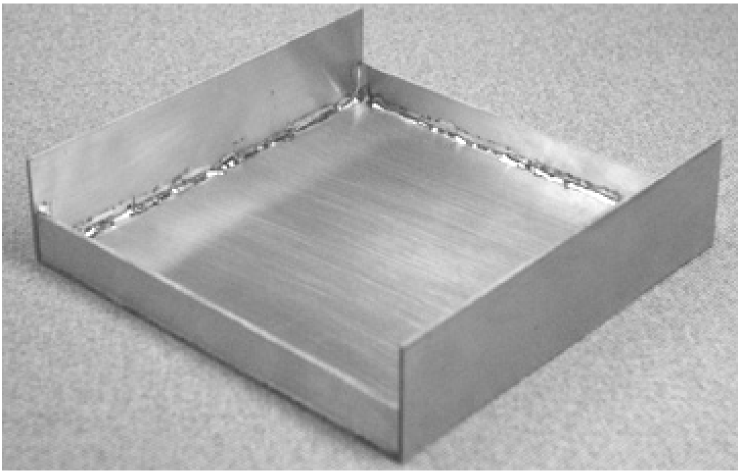


Figure 7: Completed Bottom Half

This is because you are cutting threads into the pcb material on the side rails. But once you have these threads in place, insertion and removal of the screws should be much easier to do.

Do not overtighten these screws! If you are careful in tightening they will last for many on and off cycles. If you inadvertently strip one out by over-tightening, drill out the top cover screw holes and go to the next larger size sheet metal screw!

The pc board edges and corners of the assembled enclosure are sharp and rather abrasive. You may care to bevel them to protect your fingers and keep the box from damaging tabletops and anything else it comes in contact with. A couple of swipes with a flat file will do the trick. Early samples of case like this had a nasty tendency to poke holes in bags and knapsacks before being refined this way!

Figure 9 below is a photo of the bottom of a complete enclosure with the side screws in place. Note the flush and even nature of all pc board edges.

FINIS HING

At this point you have a completed enclosure! All that remains is to do some final finishing as you see fit and to install the SOP Receiver and associated hardware.

Painting is a personal thing. While some care to leave a "natural" copper finish, others want to personalize their projects by painting them outrageous colors or match them to other equipment they may have. The whole process is beyond the scope of this manual but here are a couple of tips.

1) If you are going to paint your enclosure be sure that all soldering is finished. The heat from soldering inside the case will conduct through the wall and damage your nicely painted surface.

2) Surface preparation is very important to ensure that the paint adheres properly. Use a Scotchbrite pad to remove surface roughness and clean the metal immediately prior to painting with acetone, working in a well-ventilated area so that you don't suffer respiratory distress.

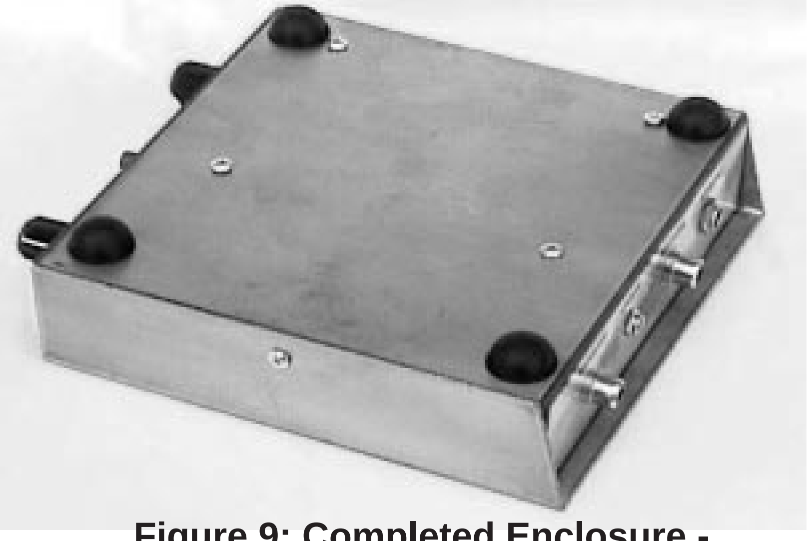


Figure 9: Completed Enclosure - Bottom View

Figure 10: Rear Panel with Labels

3) For a well-preserved "natural" copper finish that will not tarnish use a clear acrylic spray such as Krylon ™ clear spray.

4) Other spray-on acrylic colors will work okay for the short-term but may have a tendency to chip or get scratched in time. An oil-based paint like Rustoleum adheres well and gives a tough finish.

5) Chose a light color if you are going to use the acetate overlays provided with the kit. The lettering on them is black, but may not have much contrast against a dark surface.

FRONT & REAR PANEL LABELS

Acetate labels are provided in your Enclosure Kit to help you create a neat and professional-looking project case. These clear overlays are pre-printed but do not have holes cut in them for panel-mounted controls and connectors. Once in place on the panels, a sharp hobby knife such as an Xacto knife will cut holes in the appropriate control/jack locations quite nicely. A good treatise on labels is "Homebrew Chassis and Panel Labels" by Ed Roswell, K2MGM. It appeared in the Winter 2000 issue of the QRP Homebrewer.

If you choose to use the acetate labels provided in this kit, merely cut each one out carefully along the outside of the rectangle defining the panel perimeter. This will allow the rectangle to be present on the panel when complete.

Next, spray adhesive or apply a light coating of rubber cement on the panel and carefully position the acetate label sheet over the panel and press into place. In a few minutes, the adhesive will be dry and you can delicately cut out the holes in the acetate over the control/jack holes of the panel. (Trick: Let the panels dry thoroughly, or even wait overnight, and the holes will be easier to cut.)

It's a good practice to lightly spray the front of the acetate label with clear acrylic like Krylon. This will protect the black toner (comprising the actual labels) from inadvertent scraping.

See Figure 10 above showing the rear panel with labels applied. The front panel labels are shown in the photo on the cover of this manual.

See Figure 14 at the end of this section for actual-size graphics used to create the acetate labels. If you need to make another set of labels, you can run that page through a copier using transparency material instead of paper in the copier.

FINAL STEPS

All that needs to be done now is the finishing touches.

Rubber Feet - Carefully peel off the black rubber feet supplied in the strip and apply one each to the bottom side of the enclosure. Make sure they are aligned orthogonal and uniformly for maximum esthetics. (That is, put them neatly in place!) If the surface of the pc board material is clean, the rubber feet should stay in place for a long while.

Put the top half of the enclosure on the bottom half and secure it using the two previously-used screws.

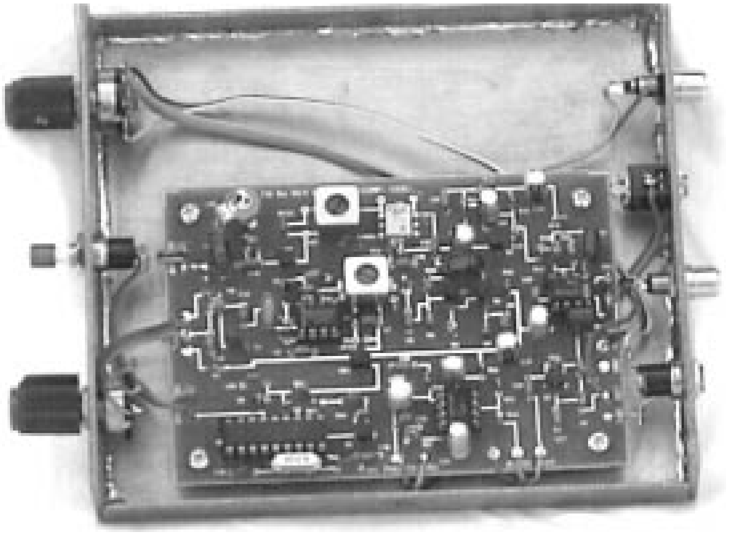
INSTALLING THE SOP BOARD

Now that your enclosure is all assembled, painted and labeled, all that needs to be done is to install the SOP circuit board and control hardware.

The SOP Receiver board will be supported by four nylon standoffs, secured by small #4 sheet metal screws.

Using small sheet metal screw (supplied), attach a nylon spacer to the bottom side of the board at each of the four holes on the SOP board. The screw will be inserted from the top side and protrude down and into the nylon spacer on the bottom of the board. You'll be cutting threads into the spacer, and it will be a tight turn of the screwdriver. Hold the spacer tightly with your pliers and turn the each screw down into its respective spacer until the board is tightly contained between the screw and spacer on each end of the board's rear edge.

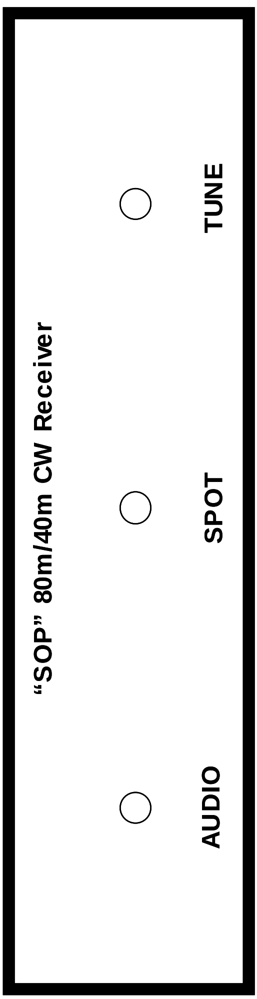
Next, carefully insert the board assembly into the enclosure and position it over the four holes in the base plate. Using the same technique as above, screw four #4 sheet metals screws (supplied) through the bottom of the case and up into the four nylon standoffs on the board. If the holes do not perfectly line up, try loosening the top side screws and let the nylon washer float into the correct position when screwing the bottom screw in place. Worst-case, if the holes cannot be made to line up, just use 2 or 3 screws on the bottom, and the board will still be held securely in place.

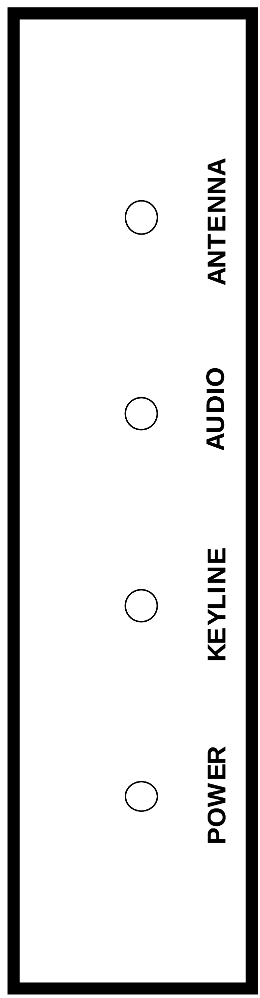


QUESTIONS

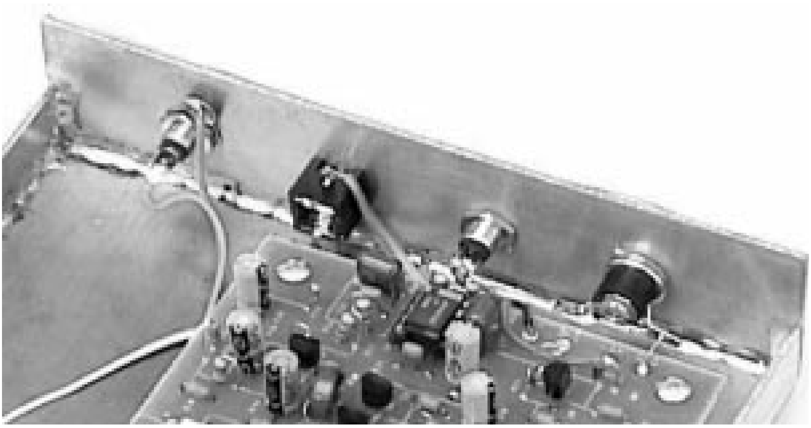
This method of homebrewing project enclosures is bound to be growing in the future. Opportunities for modifications and personal customizations are limited only by your imagination. One of our members of the NJQRP Club has a project in progress wherein he is making the enclosure for his 2N2/40 Transceiver completely out of pc board material, just like with this basic SOP design. In addition, he's also fabricating an entire, modular, multi-board inner structure out of the copper clad boards. Homebrewing boundaries have officially been extended to the horizon!

If you have any questions or problems with the enclosure aspects of this project, please feel free to contact us by email or by US ground mail. We'll do our best to help you fully enjoy all aspects of your homebrew enclosure.

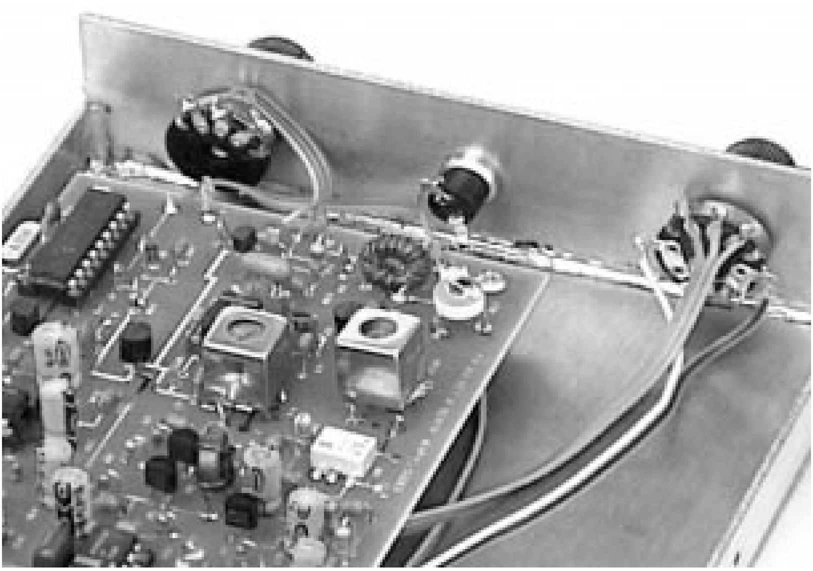




A view of the Rear Panel Jacks



A View of the Front Panel Controls



Section 4:
SOP Controls Option

Assembly Manual

This optionally-purchased Controls & Hardware Kit is another "ideal companion" to the basic SOP Receiver. All of the front panel controls and rear panel jacks are provided to enable the homebrewer to quickly and easily bring his SOP Receiver to a working state. These controls mate perfectly with the Enclosure Kit (another optional purchase) to form the completely operational receiver.

This will be an easy accessory kit to deal with. What you'll do is wire the controls and jacks to the board as shown in the figure on the next page.

Parts Inventory

The first thing to do is to compare the contents of the little plastic bag containing the controls and jacks to the Parts List shown on the next page. Make sure everything is present before you begin.

Carefully remove the nuts and washers from the two 1/8" jacks, the coaxial DC power jack, and from the RCA phono jacks. Install each into its respective location on the front and rear panels of your favorite enclosure. If you haven't yet selected or prepared your enclosure, you may of course just wire these parts directly to the board and leave them hanging in "free space" off the end of the connecting wire

Using short insulated hookup wire, thin RG174 coax cable, shielded audio cable, or whatever other wire you might have handy in your junk box, connect the ground and signal/power wires of each component to the respective pads along the edge of the circuit board.

J3 3.5mm jack -- The ground terminal on this PHONES jack is the little one on the side, and the signal terminal is the larger one on the opposite side.

J4 coaxial DC power jack -- The ground terminal is middle one, and the +V terminal is the one on the left. (The right terminal is unused.) Wire the +V terminal over to the switch on the Audio potentiometer. (See that section below.) Wire the ground terminal directly to the pcb pad for GND, next to the +V pad.

J1, J2 phono jacks -- The ground terminal of these is the tabbed washer (bend it slightly up and away from the case in order to solder the ground wire to it.) The signal wire should be attached to the center terminal of the phono jack. Wire one of the phono jacks to the KEYLINE pad, with its corresponding ground wire going to the GND pad next to the KEYLINE pad. Wire the other phone jack to the ANT pad on the pcb, with its corresponding ground wire (or shield of the thin coax used) to the GND pad next to the ANT pad.

R24 Audio Potentiometer -- This pot also contains the power switch, so wire the 2-tabbed side of this control between the power jack and the +V pad of the pcb. Wire the 3-terminal pot tabs to the pcb as shown in the diagram.

CONTROLS & HARDWARE PARTS LIST

Designator	QTY	Description
J1, J2	2	RCA Phono jack, nickel plated, ground, black insulator
J4	1	DC pow er jack, 2.1mm, molded, economy, w ith NC sw itch
Knobs	2	Black plastic knob w /blue top, .25" shaft
P4	1	DC pow er plug, 2.1mm x 5.5mm x 9.5mm black locking, barrel
P1, P2	2	RCA phono plug, black plastic insulator, hollow center pin
R24	1	Audio pot , 10K-ohm miniature potentiometer w /sw itch
R9	1	Tune pot, 50K-ohm miniature potentiometer
PB1	1	Pushbutton sw itch, normally open, momentary contact
J3	1	3.5mm jack, black, panel mount

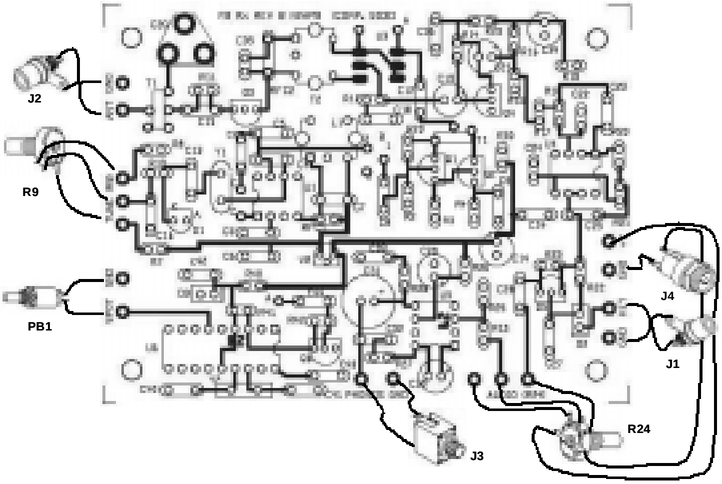
R9 Tune Potentiometer -- This pot is just a pot. Wire its 3-terminal tabs to the pcb as shown in the diagram.

Front Panel Knobs -- Using a small flat blade screw driver, partially unscrew the set screw from each knob until they are able to be slipped onto the shafts of the potentiometer controls. [The following applies if the pots are already mounted in a front panel.] With each pot shaft turned completely counterclockwise, rotate the knob (which is still loose on the shaft) such that its indicator is pointing to the seven o'clock position and tighten the set screw. This will give the pot and knob normal relative indications during operation.

PB1 pushbutton --This little red-topped pushbutton is used as the SPOT switch for the Audio Frequency Annunciator option. When pressed, it triggers the FreqMite microcontroller to sound the current frequency through the PHONES jack. (We figured that most people would be interested in the AFA option; but even if not, this pushbutton would nicely fill the front panel hole!)

Extra Plugs -- In a further attempt to ease the job of getting your SOP Kit operational, we've included the mating plugs to some of the jacks on the rear panel. Two phono plugs (P1, P2), and a coax dc power plug (P4) are included to allow you to wire up the mating cable for the antenna, keyline and dc power.

PC BOARD LAYOUT



Section 5:
Test & Measurement Circuits

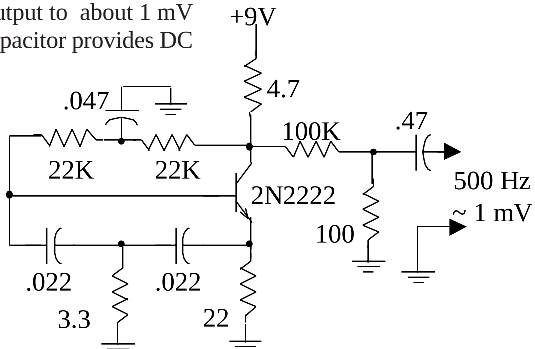
Here are two simple circuits that can be effectively used to get your SOP Receiver into tip-top shape. The first is a great little **audio oscillator** contributed by Joe Everhart, N2CX. With this useful gem you'll be able to inject low-level signals into your audio chain and trace them through to the speaker. The other circuit is a standard-but-nifty **RF Probe**, contributed by Chuck Adams, K7QO as part of his series of articles concerning "Manhattan-style construction techniques", published in QRP Homebrewer. With the RF Probe you'll be able to measure the signal coming from your local oscillator, the LO buffer and other RF signal sources. Very handy indeed!

Twin-Tee Audio Oscillator
by Joe Everhart, N2CX

This circuit is an old favorite of mine. It was the very first Joe's Quickie in the QRP Quarterly's Information Exchange column and is still a favorite on the N2CX workbench. Though very simple, this circuit is an excellent signal source for debugging and troubleshooting.

Power is supplied by a 9-volt battery (a carbon-zinc "cheapie" is fine) and the output is a 1 KHz sine wave. A simple resistive divider from the transistor collector reduces the output to about 1 mV while the 0.47 uF capacitor provides DC isolation. Check the original article for a table of resistor attenuator, which set the output levels.

It's a pretty non-critical circuit that can be built ugly or beautiful. I built mine on a scrap of perf board. There is no power switch; to turn it on I simply clip a 9-volt battery onto the power connector. And there's no fancy output connector, either. An output and ground lead are wrapped over the edge of the board for access to a clip lead or soldered-on wires.



Building an RF Probe

by Chuck Adams, K7QO

The first thing to build is an RF probe. Go to the ARRL Handbook to page 26.9 and look at figure 26.9(C) for a voltage doubler circuit. Don't use the entire circuit. All you need is C1, C2, D1, and D2. The AC input will be the RF signal and the point at which D1 and C2 meet will be the +DC output. Eliminate R1 and R2 from the circuit. The values here are not too critical and you can experiment with this. Use something like 0.1uF or smaller for C1 and something like 0.47uF or larger for C2. For D1 and D2, use either germanium or silicon diodes. I used some surplus printed circuit board leaded diodes that I got surplus somewhere and of type 1N4148 or 1N914 silicon diodes.

I won't go into the nitty-gritty details here, but you need three pads and just a small segment of printed circuit board. Because the leads were short I mounted the diodes horizontally. D1 is between two pads. D2 is soldered to the center pad and the other lead soldered to the ground plane. With longer leads you can mount the diodes in a vertical position. The probe will work the same with either configuration.

I have a Ballantine RF probe that I purchased for $10 at a Livermore Swapmeet on a trip to CA when I was working. This probe came sealed in an aluminum pouch with all kinds of military part numbers, etc. Probably cost the tax payers a few hundred dollars. I compared my readings with the RF probe built *Manhattan-style* and the Ballantine probe and got the same results to over 40MHz. The one here probably cost me about a quarter. The commercial probe is nice, but not everyone can find such a great deal. Besides, in the building of homebrew gear you can take extra pride in using something that you personally built and can repair.

Your VOM should be set for DC volts, and your reading will then be RMS volts, or Vrms = Vpeak * 0.707.