



The Sudden Storm Kit

by QRPme

Builder's Guide

version4.2

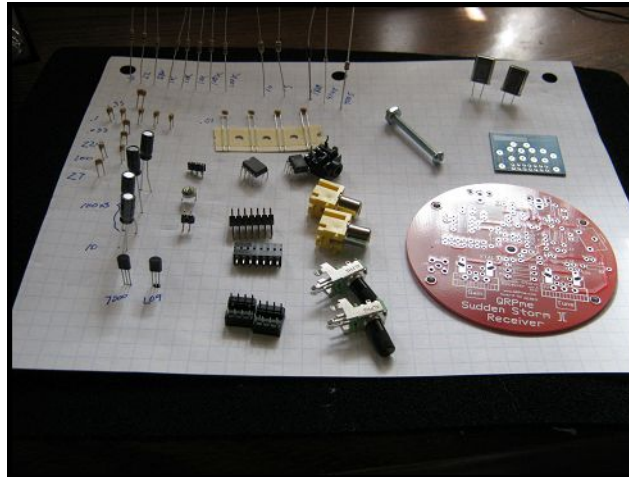
for

Sudden Storm][Ver4

(red pcb)

Updated 01/10/2012

Open the can and the adventure begins...



Organize the parts and take an inventory...

Bill of Materials

Resistors

R1 = 100K brn-blk-yel
 R2 = 10 ohms brn-blk-blk
 R3 = 22 ohms red-red-blk
 R4 = 10K brn-blk-org
 R5 = 680 ohms blu-gry-brn
 R6 = 27K red-vio-org
 R7 = 100K brn-blk-yel
 R8 = 1K brn-blk-red

L1 = 10uh brn-blk-blk
 L2 = 15uh brn-blk-blk

Q1 = 2N7000
 D1 = 1N4005
 D2 = unused
 D3 = 1N5818
 D4 = 1n4148
 VR1 = 78L09

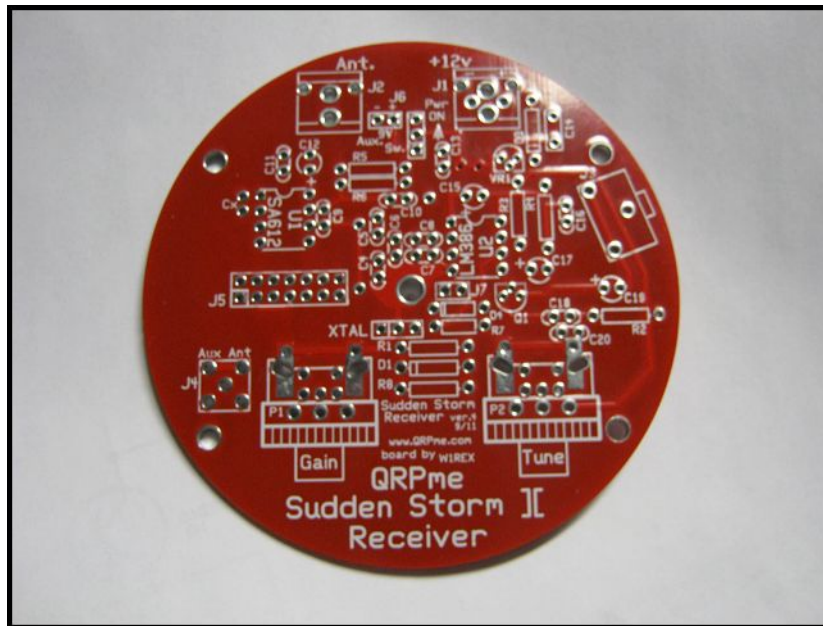
Capacitors

C1 = 27pf (270)
 C1' = 50pf trim
 C2 = 220pf (221)
 C4 = .01uf (103)
 C5 = .01uf (103)
 C6 = .033uf (333)
 C7 = .1uf (104)
 C8 = .1uf (104)
 C9 = 100pf (101)
 C10 = 100pf (101)
 C11 = .1uf (104)
 C12 = 100uf
 C13 = .1uf (104)
 C14 = .33uf (334)
 C15 = 10uf
 C16 = .01uf (103)
 C17 = 100uf
 C18 = .1uf (104)
 C19 = 100uf
 C20 = .01uf (103)

Miscellaneous

J1 = RCA jack
 J2 = RCA jack
 J3 = stereo jack
 U1 = 8pin DIP socket
 U2 = 8pin DIP socket
 U3 = 2x7x.1" f. hdr
 J1 = 2x7x.1" m. hdr
 P1 = 1x2x.1" male
 XTAL = 8pSI Psocket
 P1 = 10K linear pot
 P2 = 10K linear pot
 U1 = SA612 MIXER
 U2 = LM386 AMP
 round tuna can pcb
 band module pcb
 1.5" x 1/8" bolt
 1/8" x32 nut

OK! Here is a close-up picture of an actual ver4 board. It's RED!
I made a missteak on the order to the board house and received a batch of red boards!



The changes incorporated in the ver4 board are:

- The parts were spread out for easier building.

- The shorted traces mentioned in the ver3 board are fixed.

- Traces and pads were added to easily make the K4TWJ

 - 9 volt battery and switch mod if desired.

- The band module connector has been revised to just the

 - current dual pin header format. The old style DIP

 - carrier socket was eliminated.

- Slots were added to the pot mounting pads to facilitate firm mounting of the pots.

Only one error cropped up in the new ver4 board....

The trace from the center pin of P2, the Tune pot, was

accidentally dropped from the layout when the parts were spread

out. You will have to add a jumper from the center pin of the pot to the pad on C20 that is connected by a trace to R1.

Hopefully, everything else is there and you can get started!

You should review the following schematic and run through this builders guide a couple of times just to familiarize your self with the complete construction process. Experienced builders usually develop their own techniques in building projects. This guide takes a step by step approach to building the kit where all the parts are installed in functional stages. Within each stage, the parts are installed according to their profile; where the parts that hug the board closest are installed first and progressively taller parts are installed in order of height. This makes it easier to install successive parts. After completing each stage, it is either tested or inspected before moving on.

STAGE 1: THE POWER SUPPLY

The first stage is the power supply stage. The parts needed to install on the board for this stage are: J1, D3, C14, VR1, C13, R5, C12 and C13. Within each separate stage, the parts that lie closest to the board are installed first. The parts sequence for this stage is: C11, C13, C14, D3, R5, VR1, C12 and J1. If you don't have a bench top vice or pc board holder, you can use the now empty can as a board holder when soldering. The only problem is that if you do install the RCA connectors and the stereo headphone jack early in the build, the board will no longer fit snug in the can due to the connectors sticking out over the edge of the can. I have a small electronic vice I use to hold the board in position for soldering so I will show the connectors installed with the appropriate stage. If you need to use the can as a board rest, you should delay installing connectors J1, J2 and J3 until the very end of the build. You can tack in a couple of cut component leads at the J1 end of diode D3 and the ground side of either C13 or C14 to feed the 12 volts in for testing purposes.

- ü C11 .1uf = 104

- ü C12 100uf = 100uf 16 or 25 volt

- ü C14 .33uf = 334

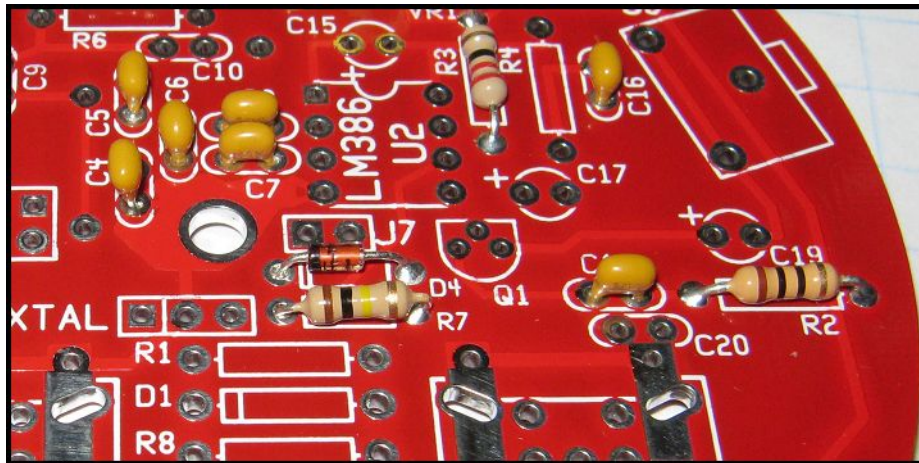
- ü R5 680ohms = BLU-GRY-BRN

- ü D3 1N5818

- ü VR1 LM78L09

- ü C13 .1uf = 104

- ü J1 RCA connector



View of completed first stage

Now you can test your progress. First inspect all your solder points for good solder fillets and that they are not shorting to adjacent pads or ground. Retouching a soldered pad with a hot iron usually will cause a 'suspect' solder blob to reform into a nicely wicked connection. When you are happy with your soldering, you can then do a preliminary power up of the board. You can either bring in +12 volts to connector J1 using a power cable with a male RCA connector or use alligator clips to hook up +12 volts to the test points that you tack into the circuit using the component leads. You should be able to measure +9 volts between the right side of resistor R5 and ground. A convenient location for the ground connection is any of the 4 mounting holes at the 4 corners of the board.

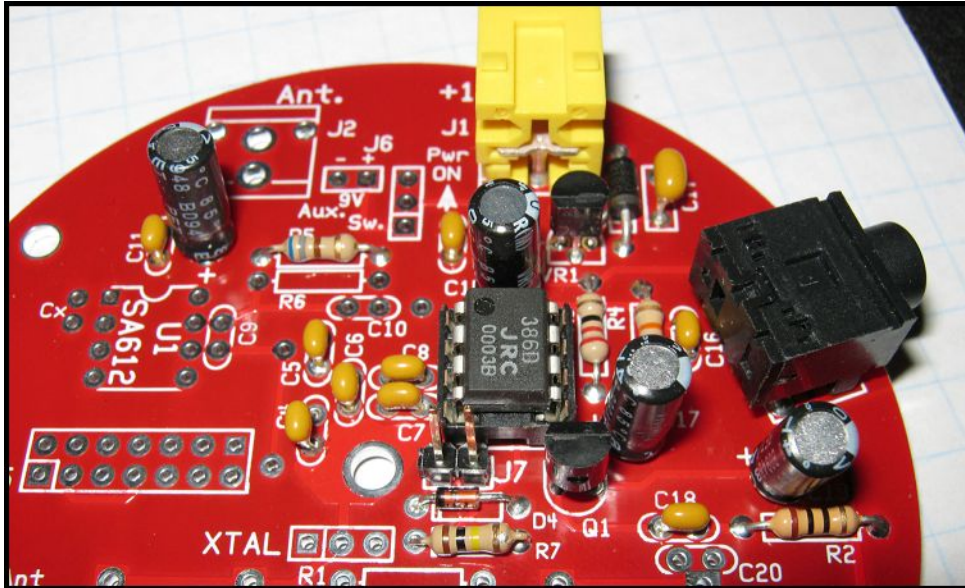
If you get +9 volts at the right side of resistor R5 then everything is normal and you can move on. Otherwise, you need to re-inspect the components, your soldering job or your +12 volt connection.

STAGE 2: THE AUDIO AMPLIFIER

Next we will tackle the audio circuit. The parts for this stage, in the order of installation are: C4, C5, C6, C7, C8, C16, C18, R2, R3, R7, D4, DIP socket U2, Q1, C15, C17, C19, J3, connector P1, and the LM386. Also included in this step is a special installation/MOD for resistor R4. The tiny little yellow caps C4, C5, C6, C7, C8, C16 and C18 are the smallest and lowest parts so they are installed first.

- ü C4 .01uf = 103
- ü C5 .01uf = 103
- ü C6 .033uf = 333
- ü C7 .1uf = 104
- ü C8 .1uf = 104
- ü C16 .01uf = 103
- ü C18 .1uf = 104
- ü R2 10 ohms BRN-BLK-BLK
- ü R3 22 ohms RED-RED-BLK
- ü R7 100K BRN-BLK-YEL
- ü D4 1N4148

Now install one of the DIP (Dual In-line Pin) 8 pin sockets. Make sure that the 'notch' on the socket lines up with the little 'notch' designator on the silk screen.



ü IC socket at U2

Install the transistor Q1 followed by the tall caps C15, C17 and C19.

ü Q1 2N7000

ü C15 10uf = 10uf 25 volt

ü C17 100uf = 100uf 25 volt

ü C19 100uf = 100uf 25 volt

The last two parts to solder are the jack J3 and mute connector P1. J3 snaps in for easy soldering but the p1 connector is a little tricky. You need to hold the connector in place for soldering by some means.

I've been soldering those pesky 2 pin Molex connectors for well over 25 years and I only use my fingers. I mount the soldering

iron into soldering position in my vice then bring the work to it now using 2 hands. My left hand is holding the board, upside down, with my forefinger holding the connector in place...TOUCHING ONLY 1 PIN OF THE CONNECTOR. I then position the board so I can solder in THE OTHER CONNECTOR PIN only. When the solder has cooled, I can remove my finger from the pin and then solder in that second pin, completing the installation of the connector. Soldering Molex connectors having 4, 6, 7 and 8 pins are pretty easy as there is plenty of spacing between the pin you are soldering and the pin you are holding. It is the 2 pin connectors that are tricky....because if you hit the wrong pin with the iron (50% chance of that), your finger will get very hot very quickly and you might end up saying a bad word or two.

- ü J3 stereo jack

- ü P1 1x2x.1" straight male header

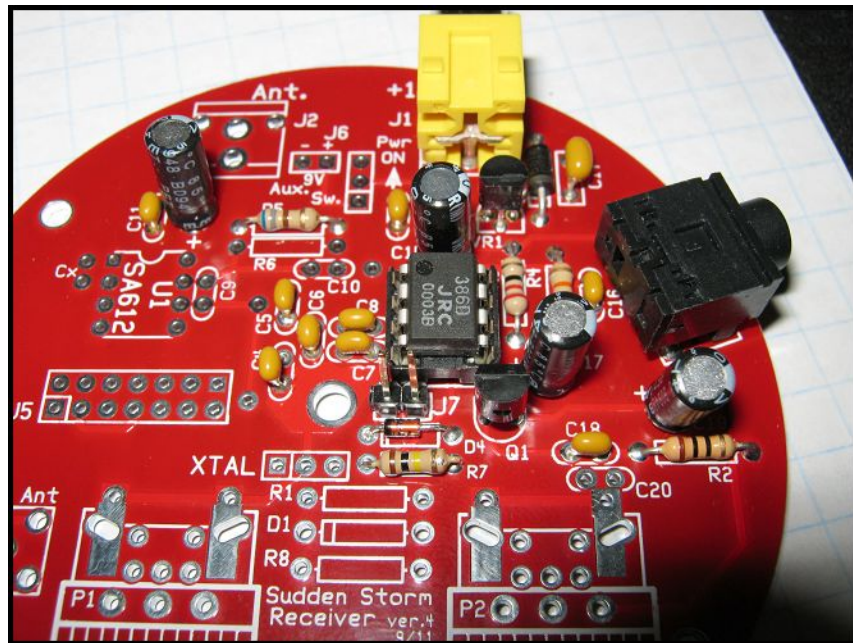
- ü R4 10K = brn-blk-org

Inspect all you soldering for this stage and make sure you have all the proper parts in the right places. If everything looks good, you can install the LM386 in the socket making sure the notch on the part lines up with the notch in the connector.

- ü LM386 at U2

Now we can test the audio stage. If you are holding back the installation of the RCAs and headphone jack so that you can use the empty can as a board holder, you will have to tack in a test jack for your headphones. Don't use the holes for the J3 jack

holes because you will then have to clean them up when you go to install the J3 jack later, Use already soldered pads like the test ground and the right side (top view) of cap C19 where it heads off to the stereo jack. With headphones in place and +12 volts applied to the power jack J1 or power test points, you should hear an audio pop when power is applied. When you touch any point around the tiny yellow caps to the left of the LM386, you should get loud hum in the headphones. Your audio stage is working! If you don't hear anything, you need to power down and go over your work for this stage. Check for proper parts in each location and all your soldering work. Look for cold solder joints, solder bridges to adjacent pads and such.



View of completed Stages 1 & 2

STAGE 3: THE MIXER/OSCILLATOR

Stage 3 parts in order of installation are: C9, C10, C20, D1, R1, R6, the 2nd DIP socket, the crystal socket and the SA612 mixer chip.

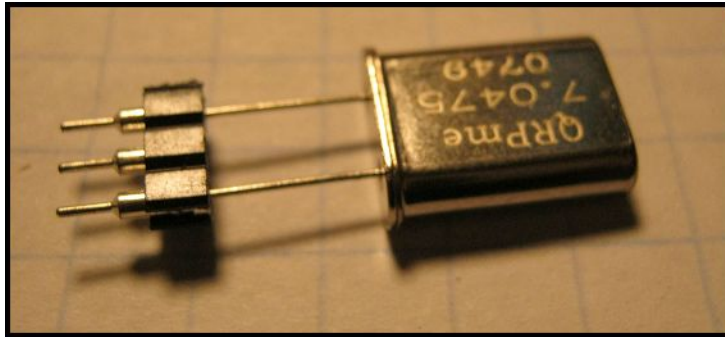
Again, the small yellow caps are installed first followed by the resistors and diode.

- ü C9 100pf = 101
- ü C10 100pf = 101
- ü C20 .01uf = 103
- ü D1 1N4005
- ü R1 100K = BRN-BLK-YEL
- ü R6 27K = RED-VI O-ORG

When installing the socket, orient the notch on the socket with the notch designator on the circuit board.

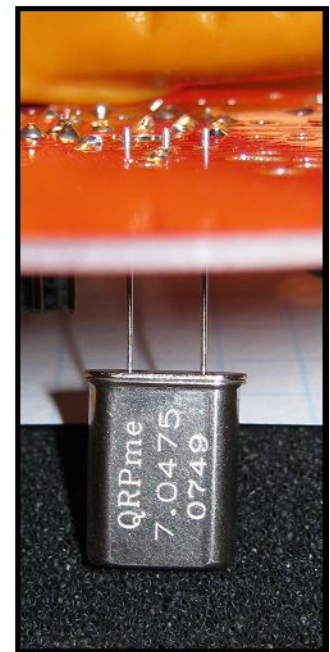
- ü IC socket at U1

The crystal socket is next. I usually insert a spare crystal into the socket to use as a handle for soldering.



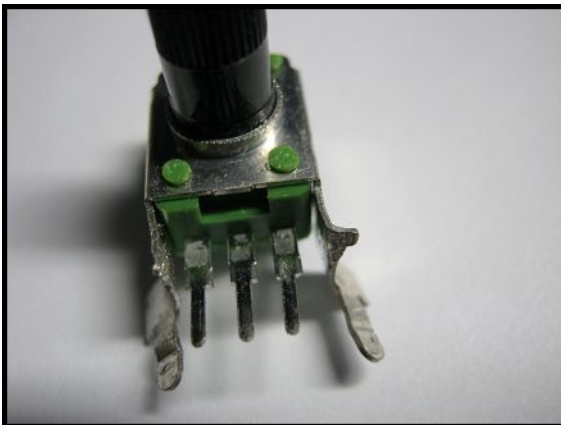
Crystal socket with resistor 'handle'.

Then I insert the socket into the board and rest the board on the resistor like a leg and solder 1 pin with your two free hands. Now that the pin is soldered, you can hold it in place while upside down with one hand, re-heat the SAME pin with the other hand while straightening the socket out at the same time. Now that 1 pin is soldered and the socket is straight up out of the board, you can solder the remaining pins.

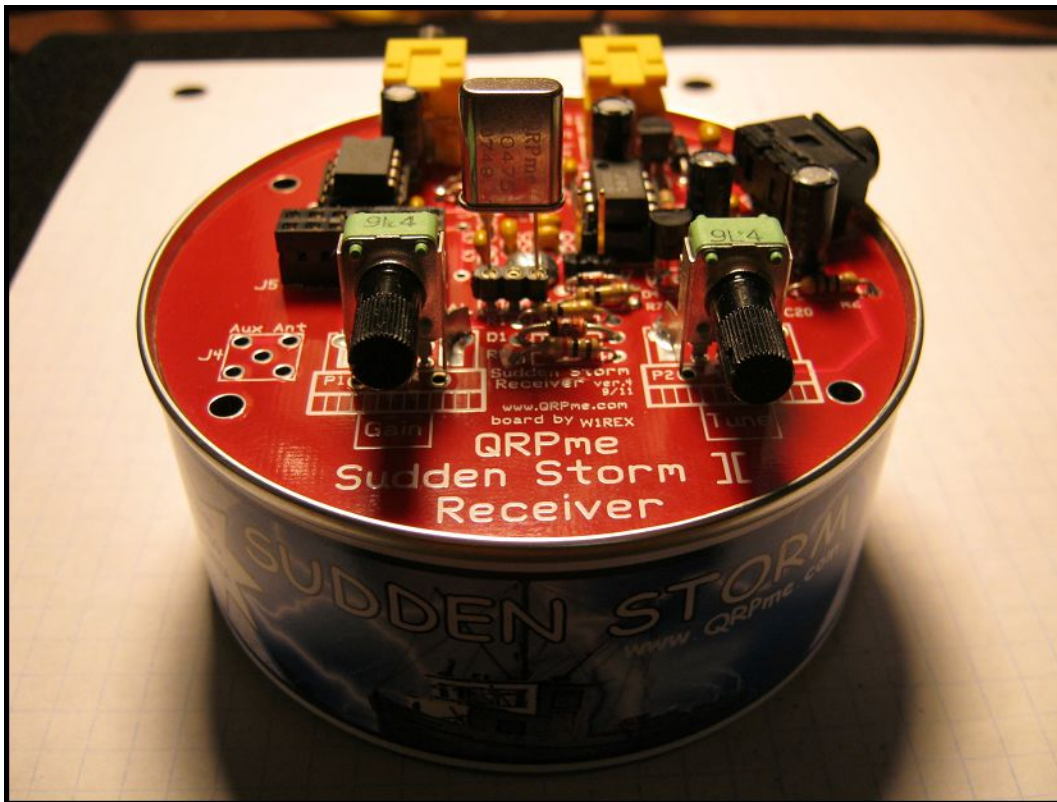


STAGE 4: THE POTENTIOMETERS

The potentiometers P1 and P2 are next. The pots that come with the kit are shown below. You have to spread the mounting tabs a little before mounting them in the slotted pads on the printed circuit board. Additionally, the circuit board is designed for the standard pots furnished with the kit AND the soup-up pots with lighted shafts. The soup-up pots fit in the front set of larger pads. A small problem exists when using the standard pots as supplied with the kits. The small 'feet' on the mounting frame of the standard pots come in contact with the outer pads for the soup-up pots. The inner pads for the soup-up pots (closest to the middle of the board) are grounded so the small 'feet' will be grounded which is OK. The outer pads are another problem! The outer pad of the GAIN pot is connected to the antenna input. The outer pad of the TUNE pot is connected to the 9 volt supply line. The small 'feet' of the pot frames cannot come in contact with the outer pads. You must EITHER cut off the corner of the pot frame OR bend the small frame 'feet' on the pots outward so that when the pot is mounted the 'feet' won't contact the pad below it.



Frame 'foot' bent outward OR Frame foot cut off

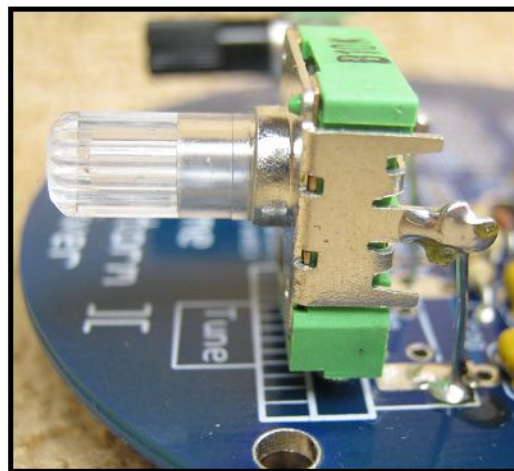
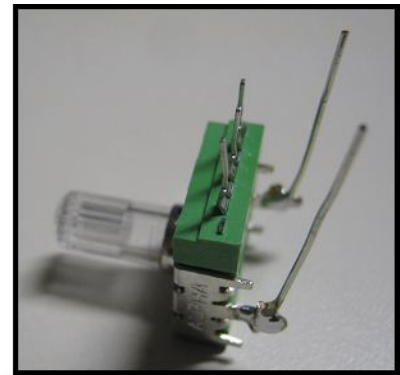


Sudden Storm][with both pots mounted

If you purchased the Sudden Storm Soup-up kit, you will want to install those pots instead of the regular pots supplied. The larger Soup-UP pots are designed for vertical mounting, so the mounting leads need to be straightened. The pots are mounted into the larger pads in the front of the pot locations. Two cut off component leads can be soldered between the mounting tabs on the pots and a set of extra holes on the pcb pads to provide extra stabilization to the pots.



Soup UP pots as supplied and modified for Sudden Storm use



Typical Soup UP pot mounted to the circuit board.

Unfortunately, there is not much you can do to test this stage until the band module is installed. So inspect your work while it is fresh in your mind and then move on to the next stage.

STAGE 4.5 THE POT MOD

When spreading out the parts (for easier construction) on the version 4 circuit board, I accidentally left off the trace connecting the center wiper of the tune pot to the rest of the tune circuit. You will have to add a jumper to replace this missing trace. Make the jumper out of a cut off component lead.



Mod jumper on pot P2

Add a jumper to connect the center wiper pin on potentiometer P2 to the right pad (as looking at the underside) of C20 as pictured above.

STAGE 5: THE BAND MODULE

Now it is time to work on one of the unique features of the Sudden Storm version 3 kit: the band module. The parts required for this stage are (in order of installation): C2, C3, C1' (trimmer cap), C1, L1, L2, J1, U3, pot P1, J2.

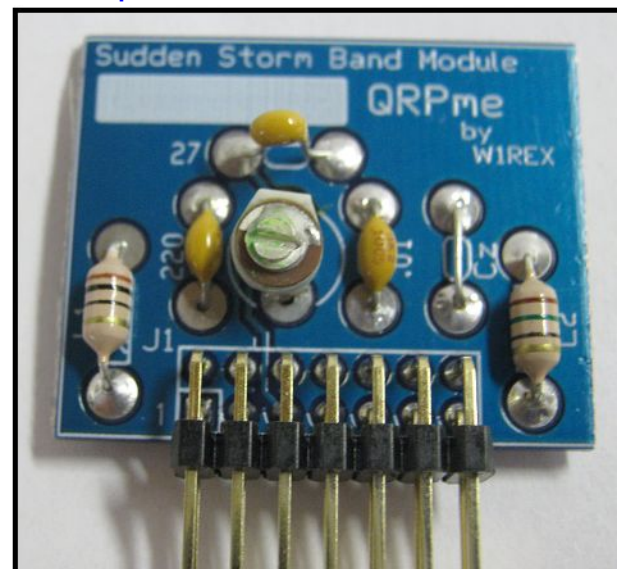
- ü C2 220pf = 221
- ü C3 .01uf = 103
- ü C1' the 50pf trimmer cap **also solder top pads**
- ü C1 27pf = 270
- ü L1 10uh = BRN-BLK-BLK
- ü L2 15uh = BRN-GRN-BLK
- ü Cz ...a shorting wire across Cz
- ü J1 2x7x.1" right angle male header



soldered on the top side.

Band module board with caps installed.

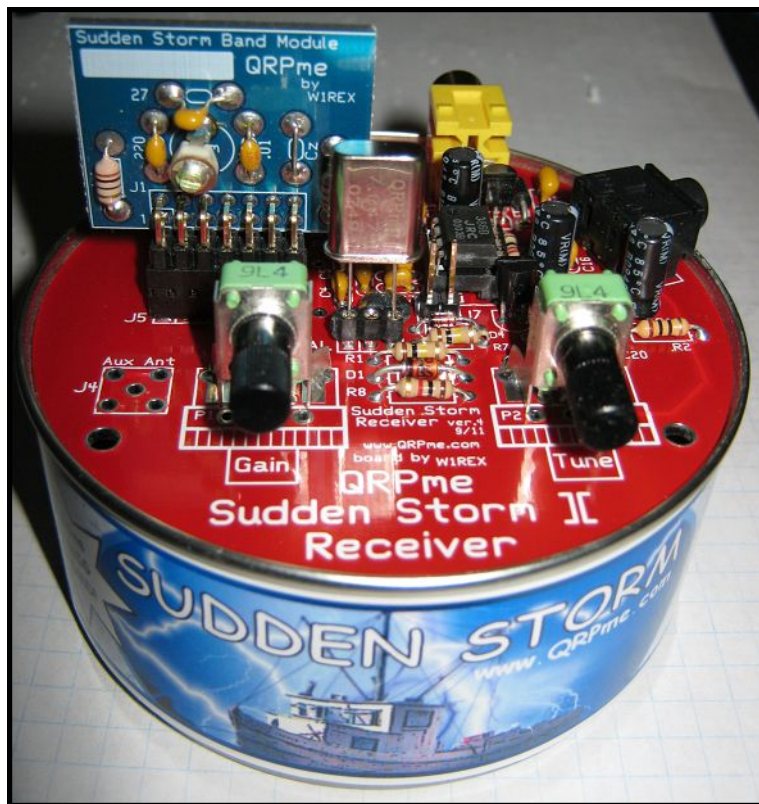
Note that trim cap pin towards top of board is also



View of completed 40m band module board
And now install the band module related parts on the round tuna
can board: U3, pot P1 and J2.

- ü U3 2x7x.1" female header
- ü P1 10k linear potentiometer
- ü J2 RCA connector

You can now insert the SA612 chip, crystal and band module
board and you are ready to operate!



Sudden Storm kit ready for action!