



The Sudden Storm Kit by QRPme

The Sudden Storm kit is a simple, easy-to-build direct conversion receiver based on a design by the Rev. George Dobbs, G3RJV, presented in his Practical Wireless column. QRPme's version of the circuit board is designed to fit on top of a tuna can creating the perfect mate for the Two Tinned Tunas kit. With no coils to wind, a minimum of parts and a step by step pictorial builder's guide, this kit is ideal for beginners and experienced builders alike.

W1REX QRPme www.QRPme.com

Manual Ver. 4.3



Open the can and the adventure begins...



Organize the parts and take an inventory...

Hopefully, everything 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. This guide takes a step by step approach to building the kit where all the parts are installed in order. Experienced builders usually develop their own techniques in building projects. One method is to install the parts by their profile; where the parts that hug the board closest are installed first and progressively taller parts are installed in order of height. This guide follows that method of construction soldering: resistors, capacitors, ICs, Pots and jacks according to their height above the board. The reasoning behind this logic is that it makes it easier to install successive parts.

Sudden Storm Parts List

Resistors: Qty. 11:

- 2 x 0 ohm jumpers = black
- 10 = brown black black
- 22 = red-red-black
- 1.8K = brown-gray-red
- 27K = red-violet-orange
- 100K = brown-black-yellow
- 4x 10M = brown-black-blue
 - (previous versions included 5.1M)

Capacitors: Qty. 15:

- 68pf = 68
- 2 x 100pf = 101
- 220pf = 221
- .01uf = 103
- .022uf = 223
- 5 x .1uf = 104
- .33uf = 330nK (yellow or blue square)
- 10uf
- 22uf
- 100uf

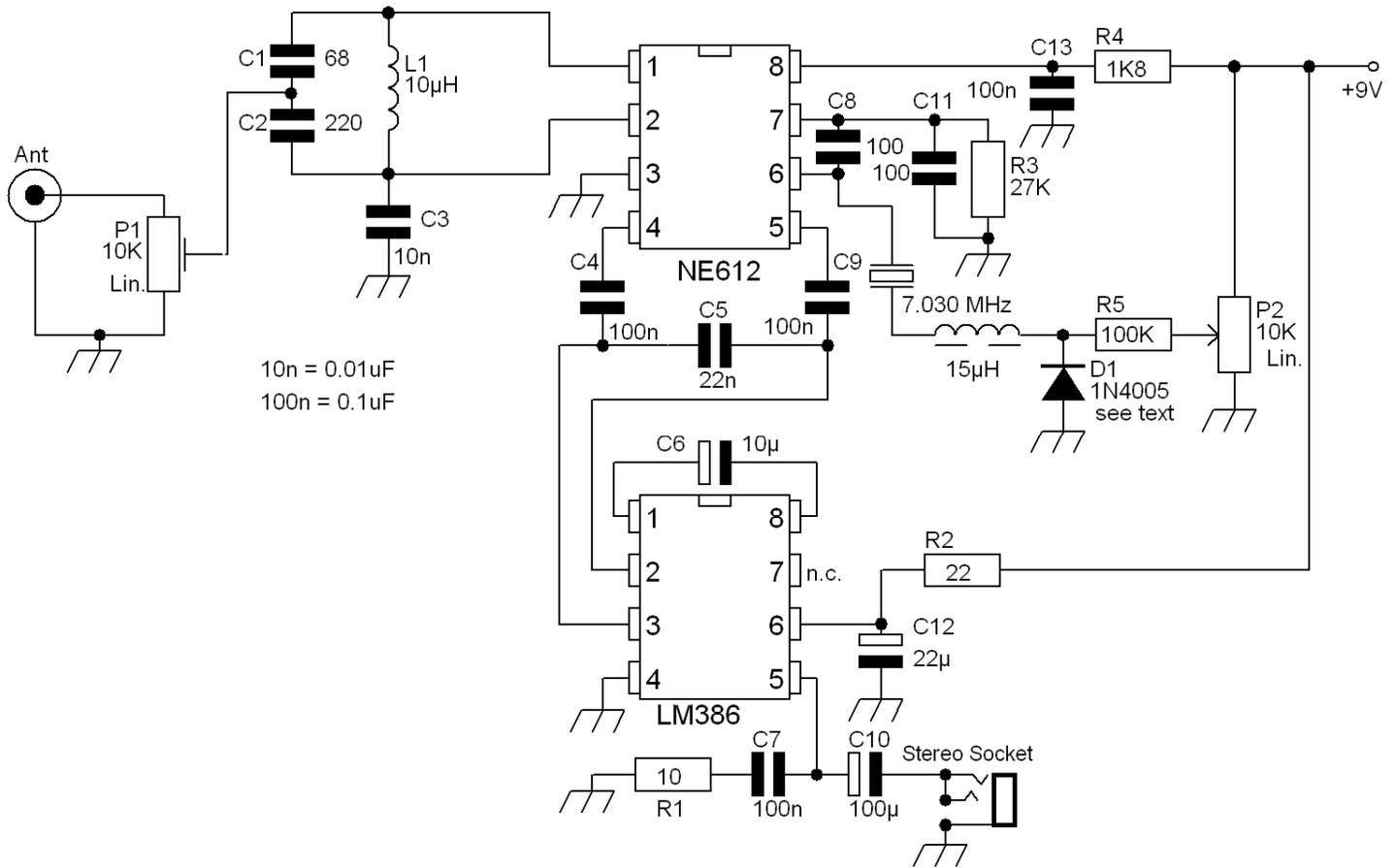
Inductors: Qty. 2:

- 2x 6.8uH = blue-gray-gold
- 10uH = brown-black-black
- 15uH = brown-green-black
- 2x 22uH = red-red-black

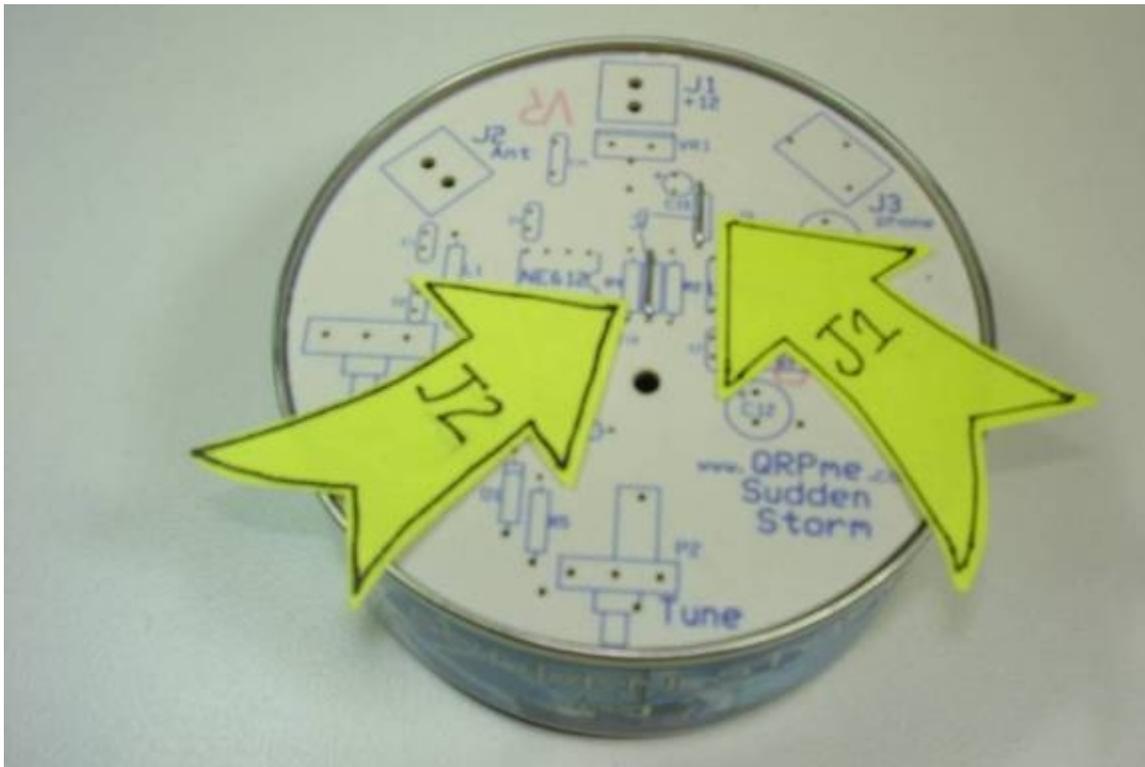
Miscellaneous:

- 2 x 10K linear potentiometers
- 1N4005 diode
- 2 x RCA jack - pcb mount
- 1/8" stereo headphone jack - pcb mount
- 7.040 crystal
- 8 pin SIP socket (2 pins each for xtal & diode and 4 pins for L1 & L2)
- NE602 or NE612 integrated circuit
- LM386 integrated circuit
- 78M09 9 volt regulator
- 1/8" x 1 1/2" stove bolt
- Printed circuit board

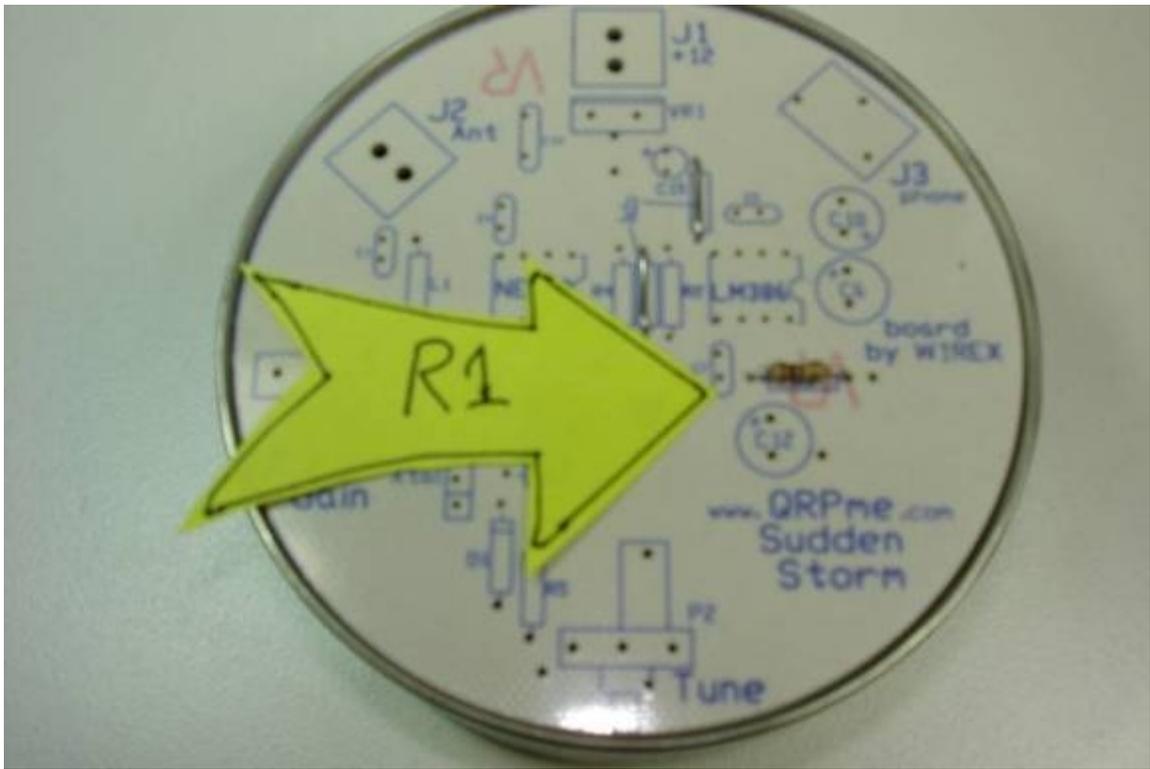
Sudden Storm Schematic Diagram



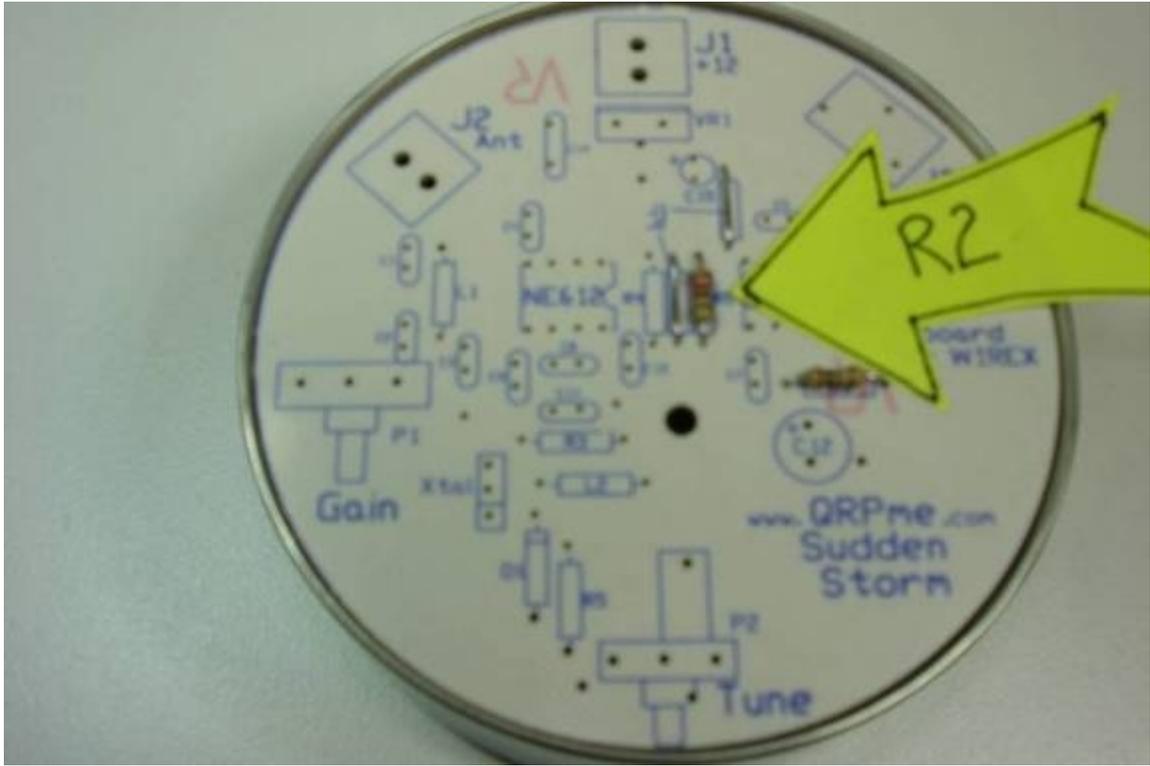
Design & Schematic by G3RJV PCB layout & kit by W1REX



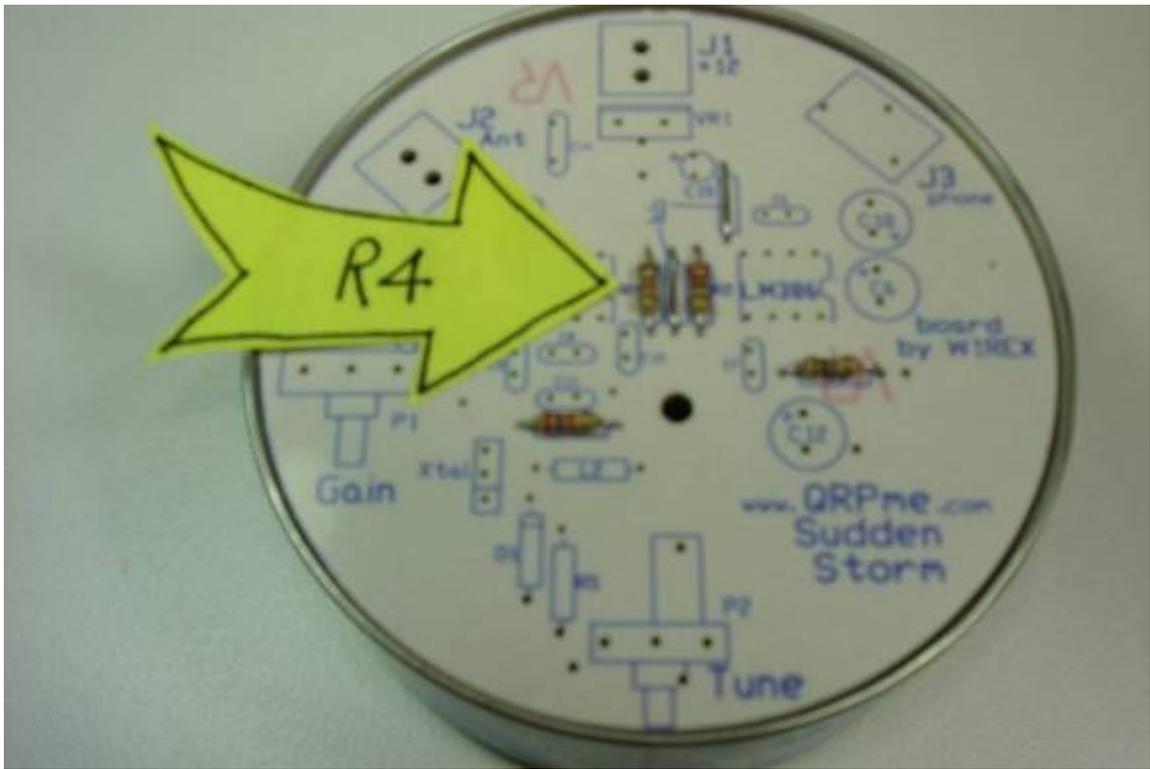
Install 2 jumpers at locations J1 and J2...that's jumper J1 and J2 NOT connector J1 and J2. Use the discarded end of a resistor or capacitor...or a zero ohm jumper (single black stripe) if one was included. Insert the jumper and spread the leads apart slightly to prevent it from falling back out when you turn the board over to solder. Solder the leads and clip off the excess leads just above the solder joint.



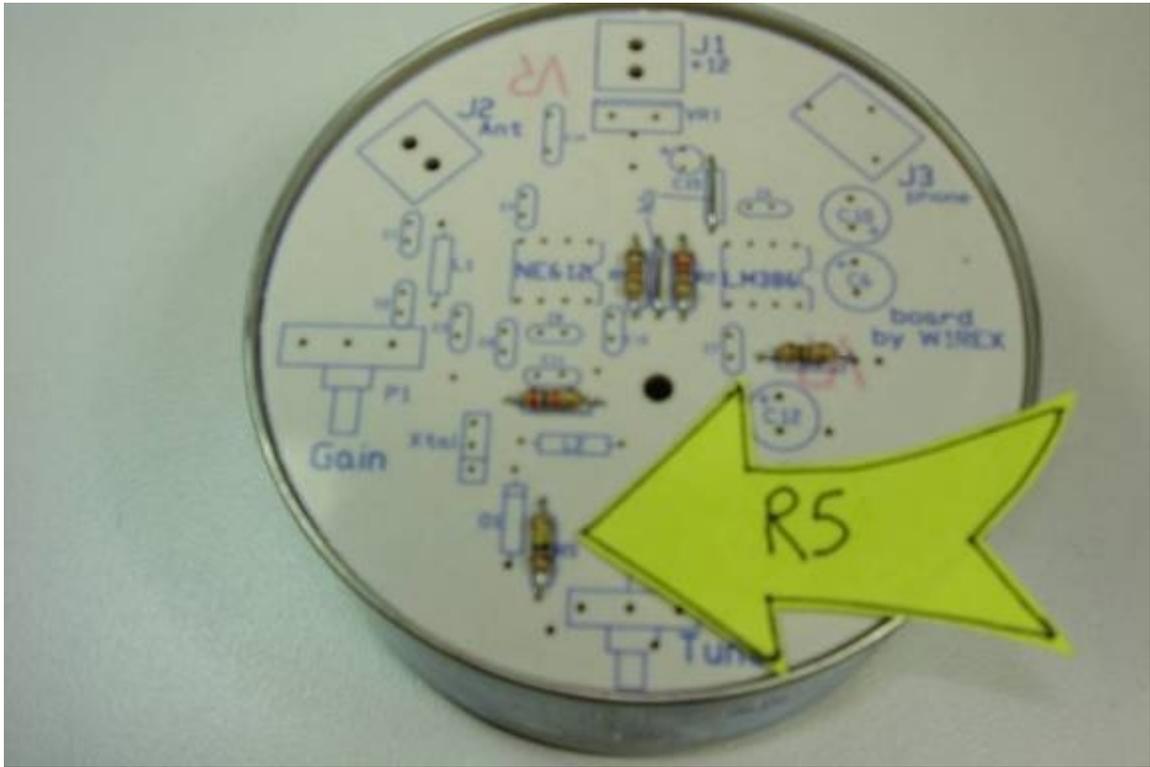
Install a 10 ohm (brown-black-black) resistor at R1



Install a 22 ohm (red-red-black) resistor at R2



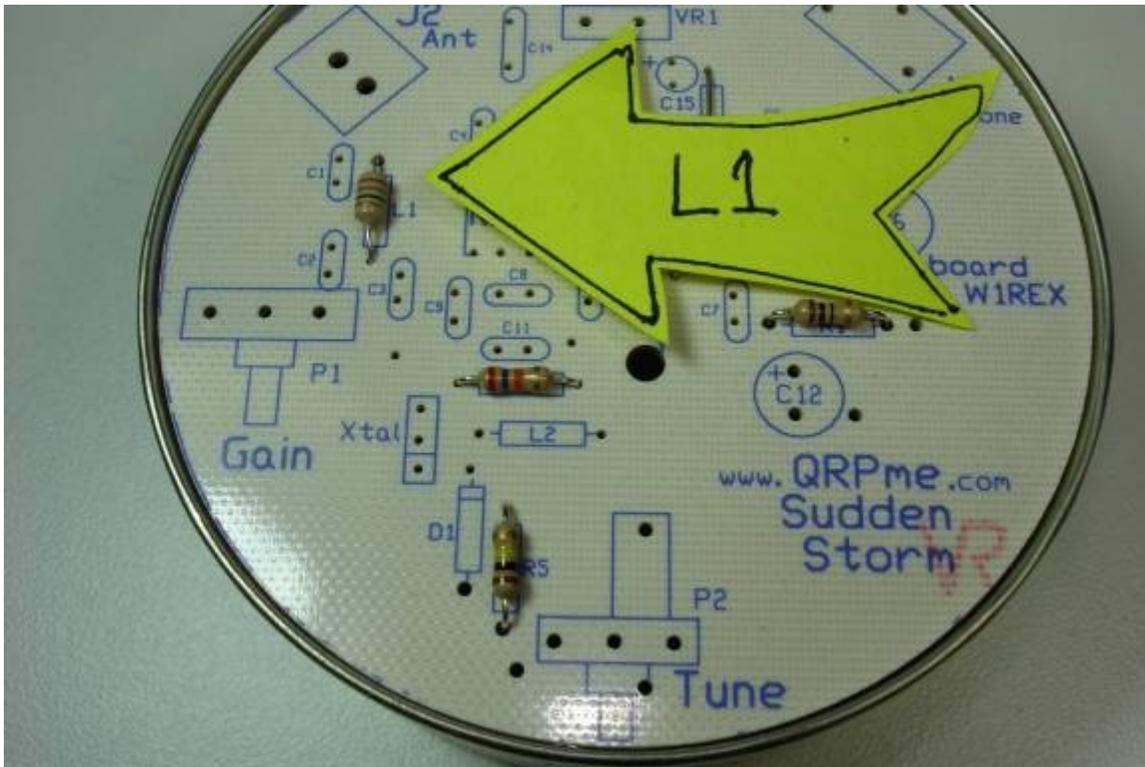
Install a 1.8K (brown-gray-red) resistor at R4



Install a 100K resistor (brown-black-yellow) at R5

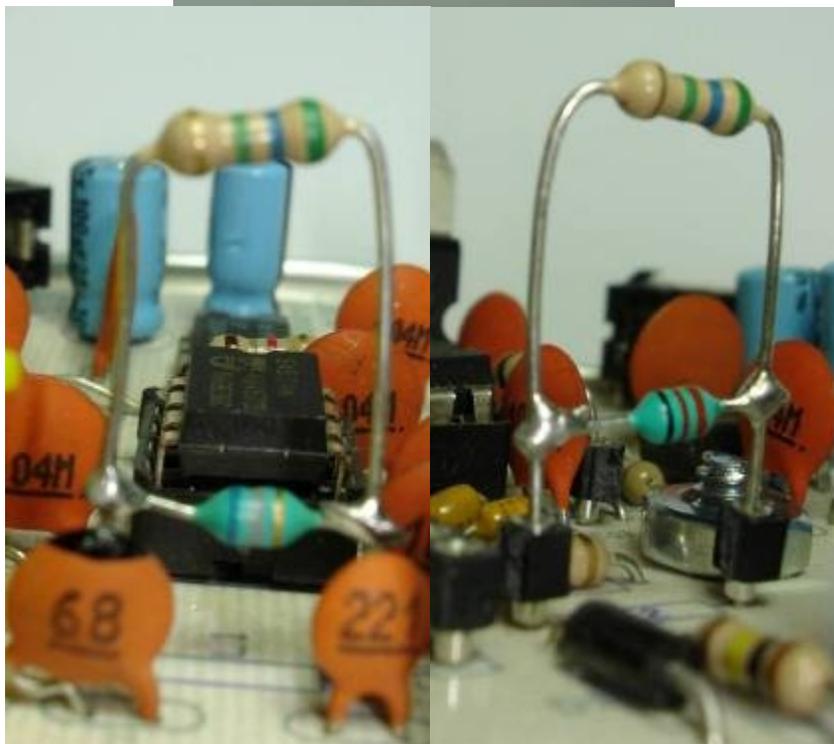


You can install and solder each part individually or install several parts and solder them all at once. Batching the installation and soldering of the parts will make the assembly process faster. Just don't try to batch too many parts as that will increase the 'difficulty' of navigating the tip of the soldering iron through the 'forest of leads' to reach the base of each lead for soldering. The empty can makes a perfect holder for the upside down board for soldering. No PCB soldering vice required here!



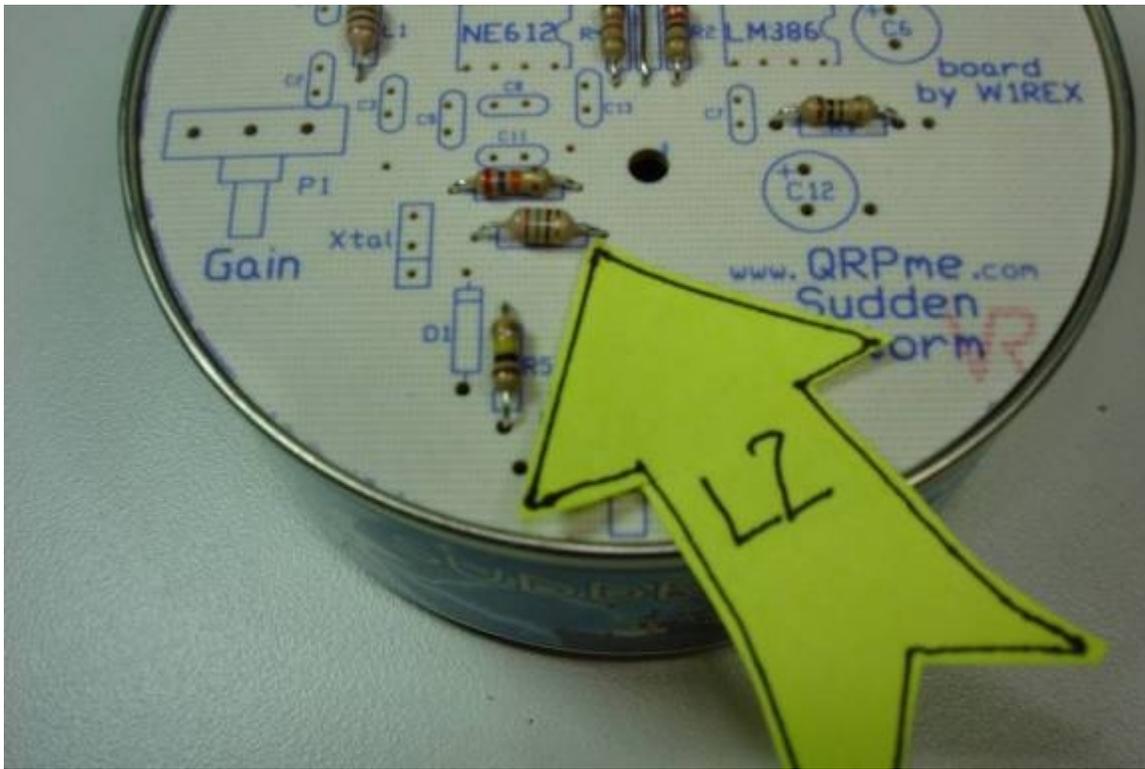
Stock 40m version: Install a 10uH (brown-black-black) inductor at location L1. The molded inductors look like resistors only they are a little 'tubbier'.

Multiband: Install 2 SIP socket pins at location L1. The inductor value for L1 varies for the intended band of operation. By installing SIP socket pins, you can then plug (and unplug) inductors depending upon the band you want to operate on... Note that the 22 and 6.6 uH inductors have finer leads than the 10uH. You can use the 10 Megohm resistors as the insertable piece and solder the inductors higher up the legs using the resistor as a handle.



Suggested starting values of inductance for L1. Since the inductors are pluggable, experiment with different values to find the best one for your situation.

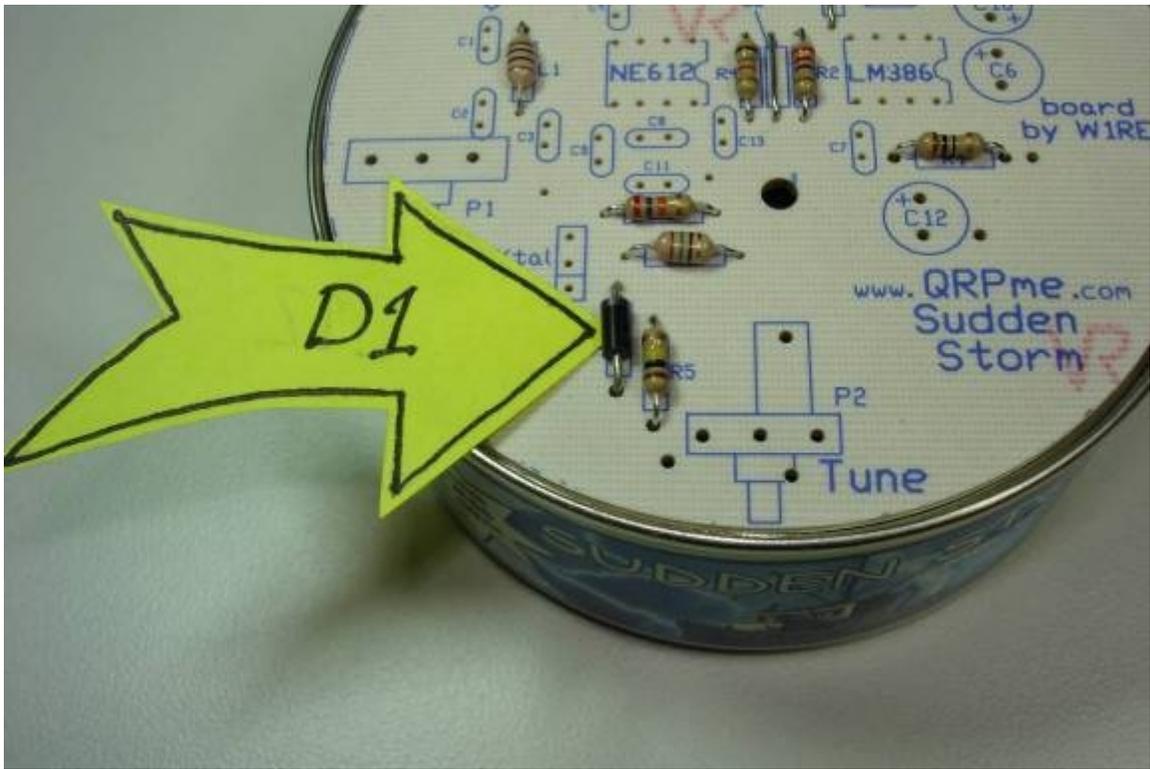
Band	L1 (in uH)	Markings
80	6.8	blue-gray-gold
40	10	brown-black-black
30	10	brown-black-black
20	22	red-red-black



Stock 40m version: Install a 15uH (brown-green-black) inductor at location L2.

Multiband: Install 2 SIP socket pins at location L2. The inductor value for L2 varies depending on the crystal in use. By installing SIP socket pins, you can then plug (and unplug) inductors to find the most optimum inductor to use. Suggested starting values for the inductance are:

Band	L2 (in uH)	Markings
80	22	red-red-black
40	10	brown-black-black
30	10	brown-black-black
20	6.8	blue-gray-gold



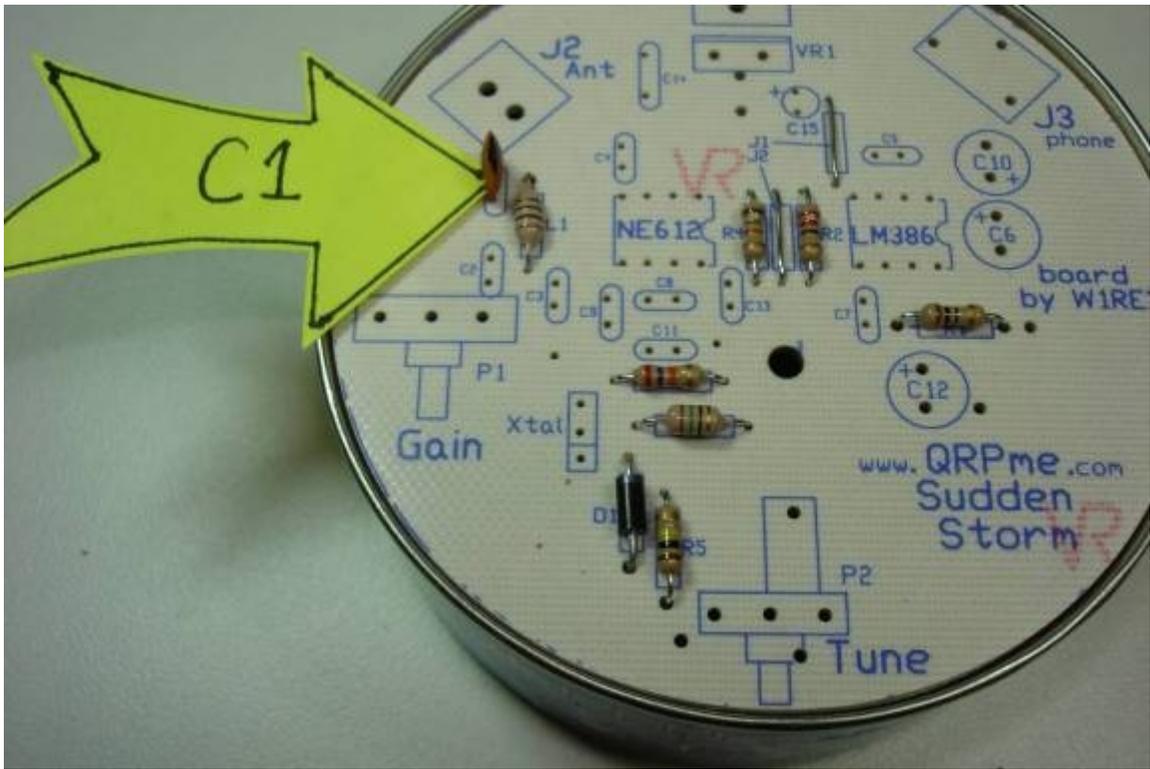
Stock Version

Install a 1N4005 diode at location D1. The 1N4005 acts as a pseudo-varactor diode.

Optional Experimenter's Version

You can optionally install 2 socket pins here to allow for experimenting with various diodes. The diodes will have larger diameter leads and will not fit into the sockets. Small pieces of 'regular' resistor leads soldered to the diode leads will allow you to plug in various diodes for experimenting. A very informative article on pseudo-varactor diodes can be found at:

<http://www.hanssummers.com/radio/varicap/index.htm>



Now we can start on the capacitors. Install a 68pF (marked 68) capacitor at location C1. For the small disc capacitors, the marking system is pretty simple:

When 1 or 2 digits appear, read that value as picofarads (pF).
When 3 digits appear, read the first digits as the value and the 3rd digit as the # of zeros to add to the digits.

Examples:

5 = 5 pF

68 = 68 pF

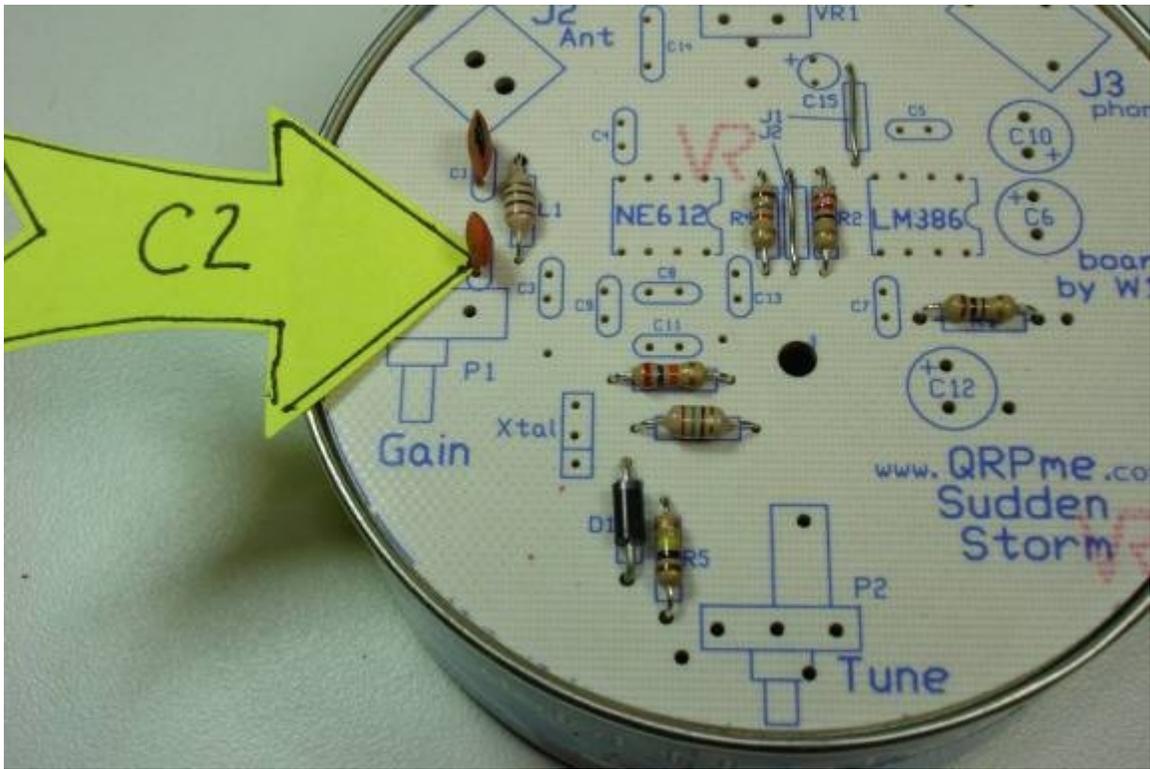
101 = 100 pF 10 + 1 added zero

221 = 220 pF 22 + 1 added zero

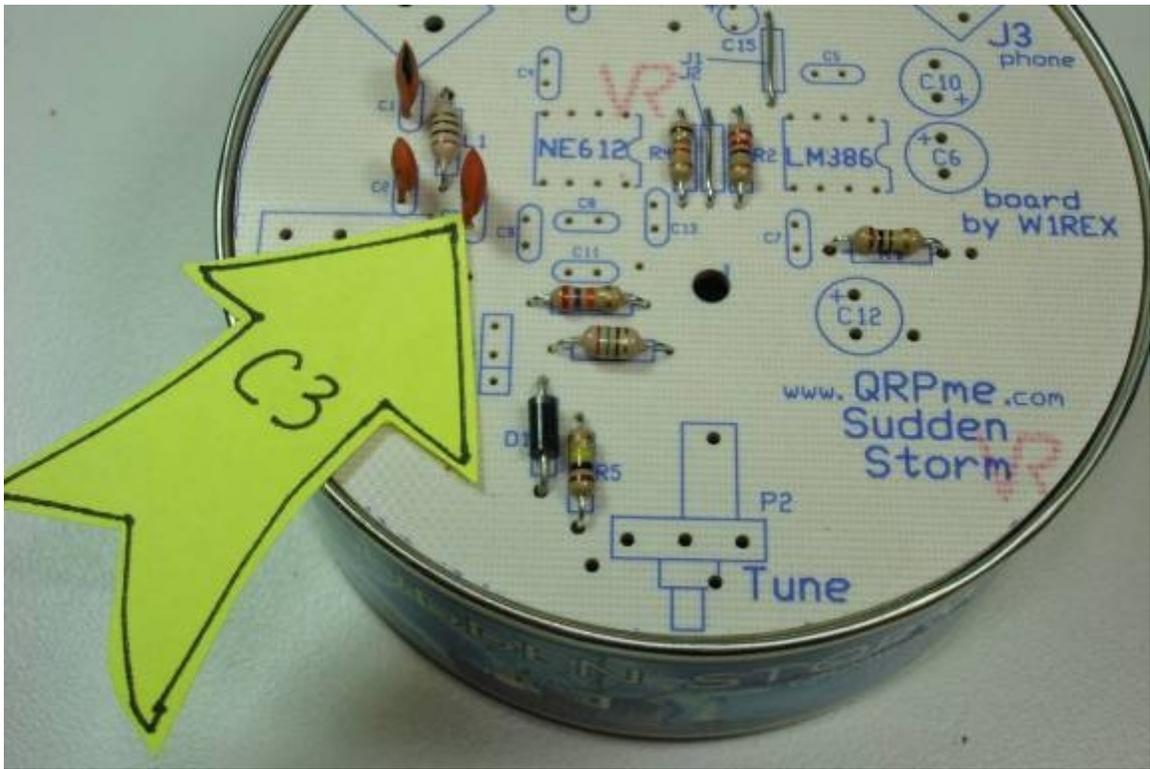
103 = 10000 pF = 10 nF = .01 uF

223 = 22000 pF = 22 nF = .022 uF

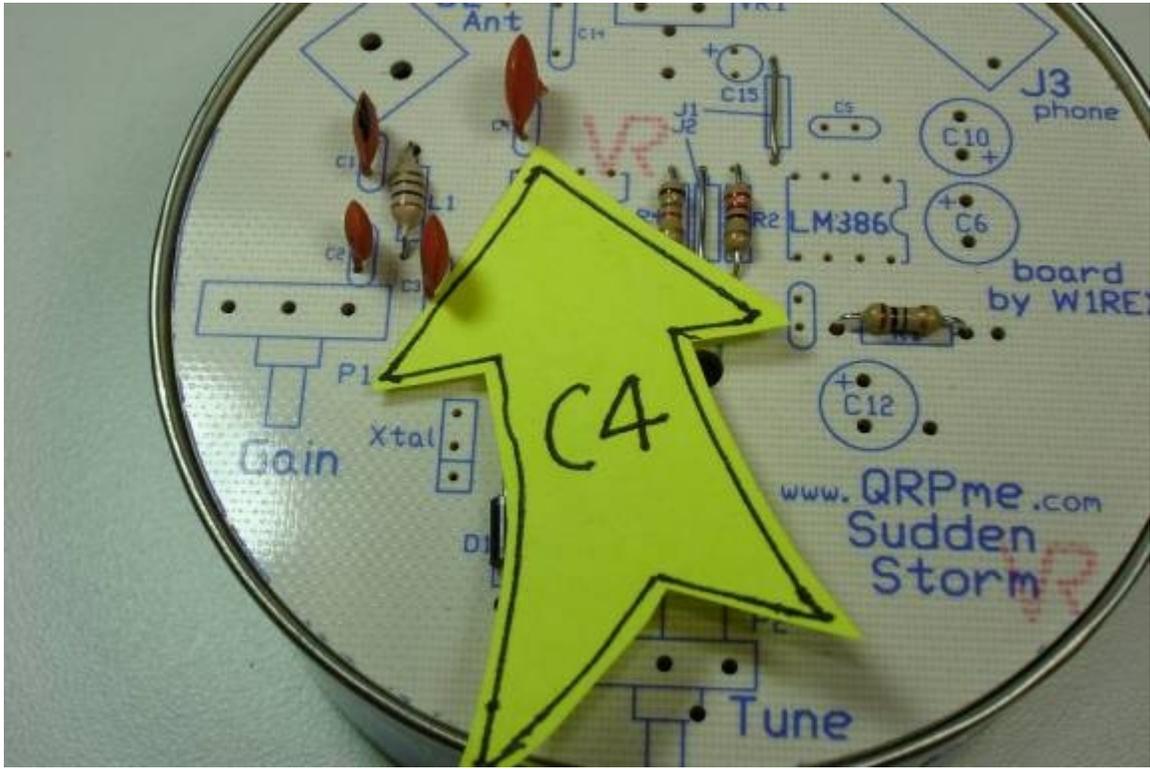
104 = 100000 pF = 100 nF = .1 uF



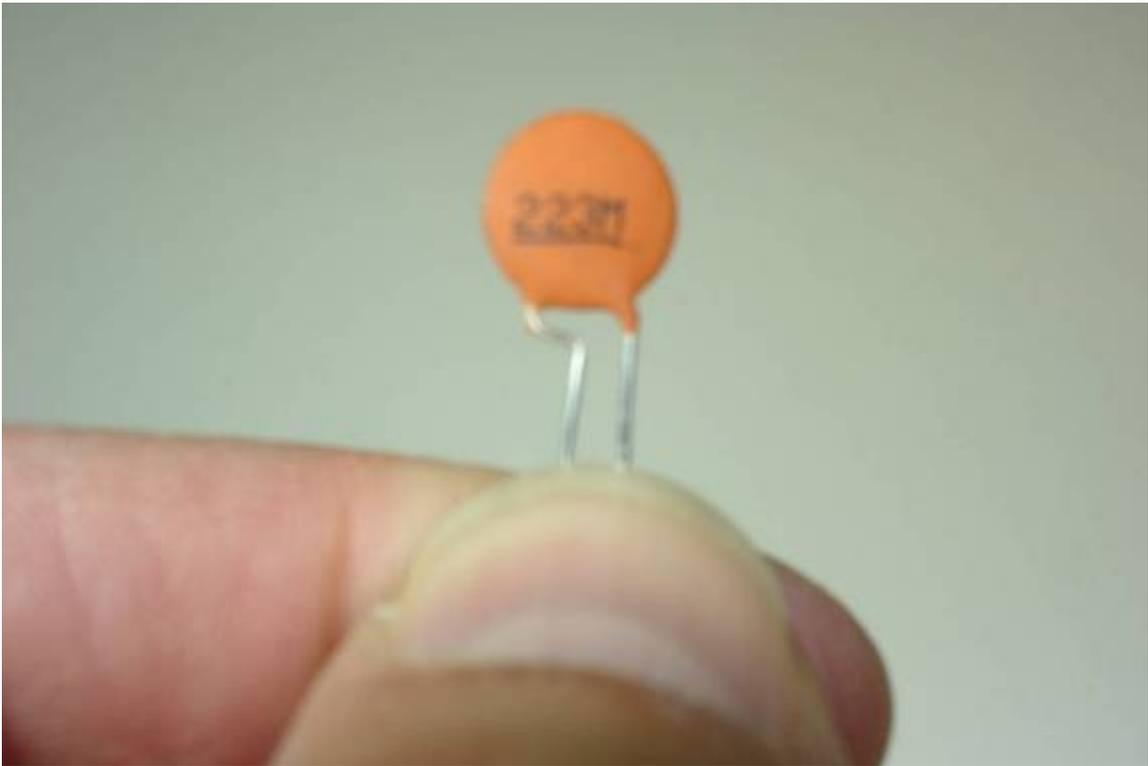
A 220pF (marked 221) is installed at C2.



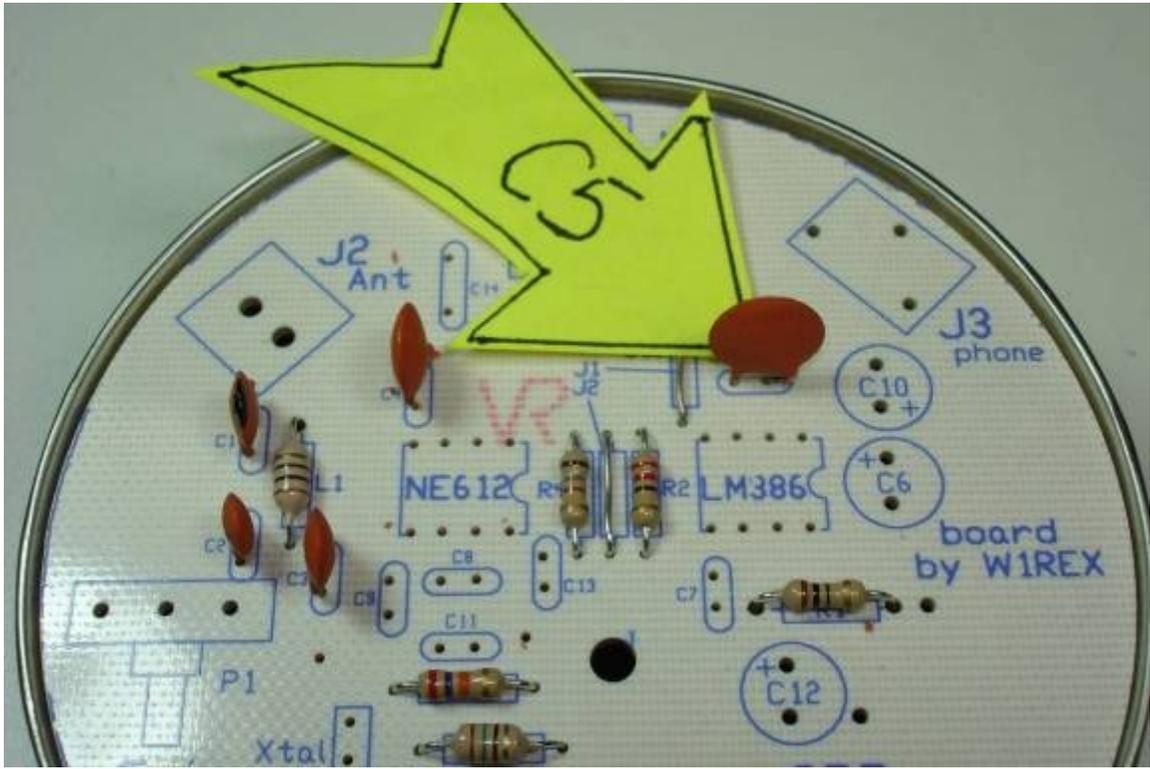
A .01 μF (103) is installed at location C3.



A .1uF (104) is installed at location C4



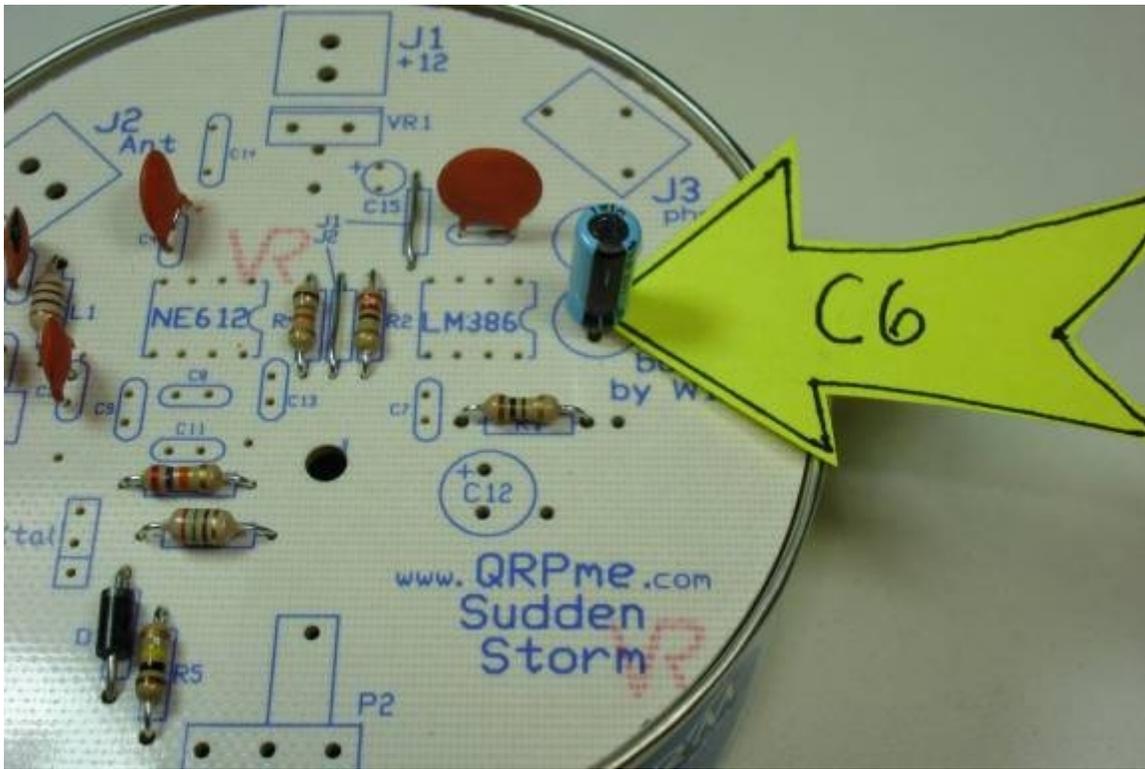
If the lead spacing of a capacitor are a little too wide for the hole pattern in the circuit board, they can be neatly bent with needle nose pliers to make a narrower spacing. Take care to not stress the lead where it enters the disc. I used the very tip of a very narrow pair of needle nose pliers to hold on to the lead where it exits the disc and a second pair of pliers to make the two bends.



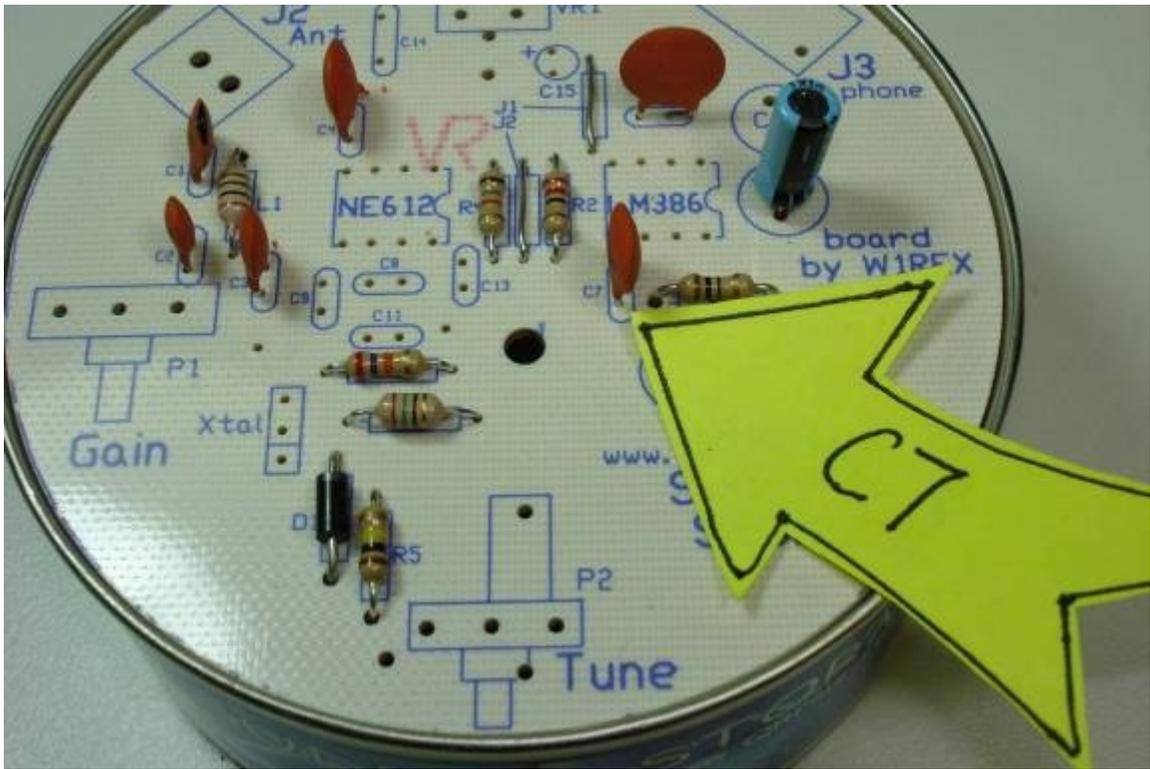
A .022 uF (223) is used at C5.



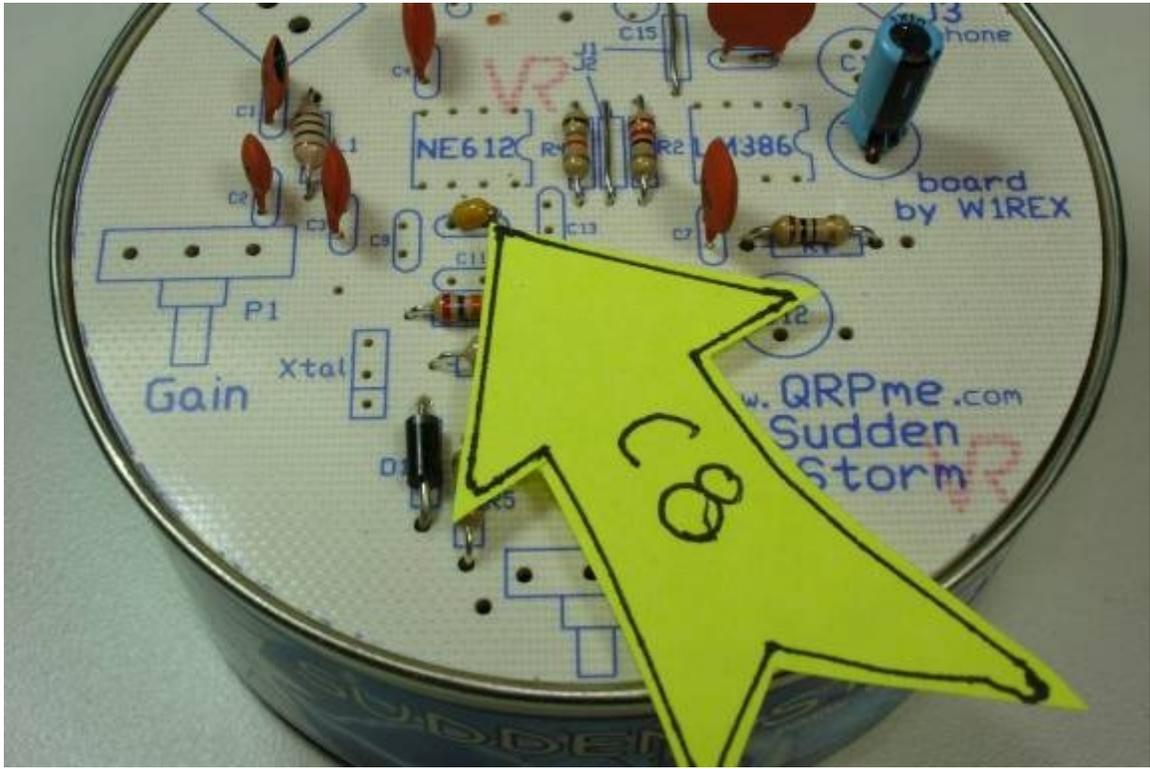
Again, you can batch install a few capacitors, bending the leads apart slightly where they exit the board. Then turn the board over, set it inside the rim of the empty can, and then solder the leads. Clip them off just above the solder joint with flush cutting pliers.



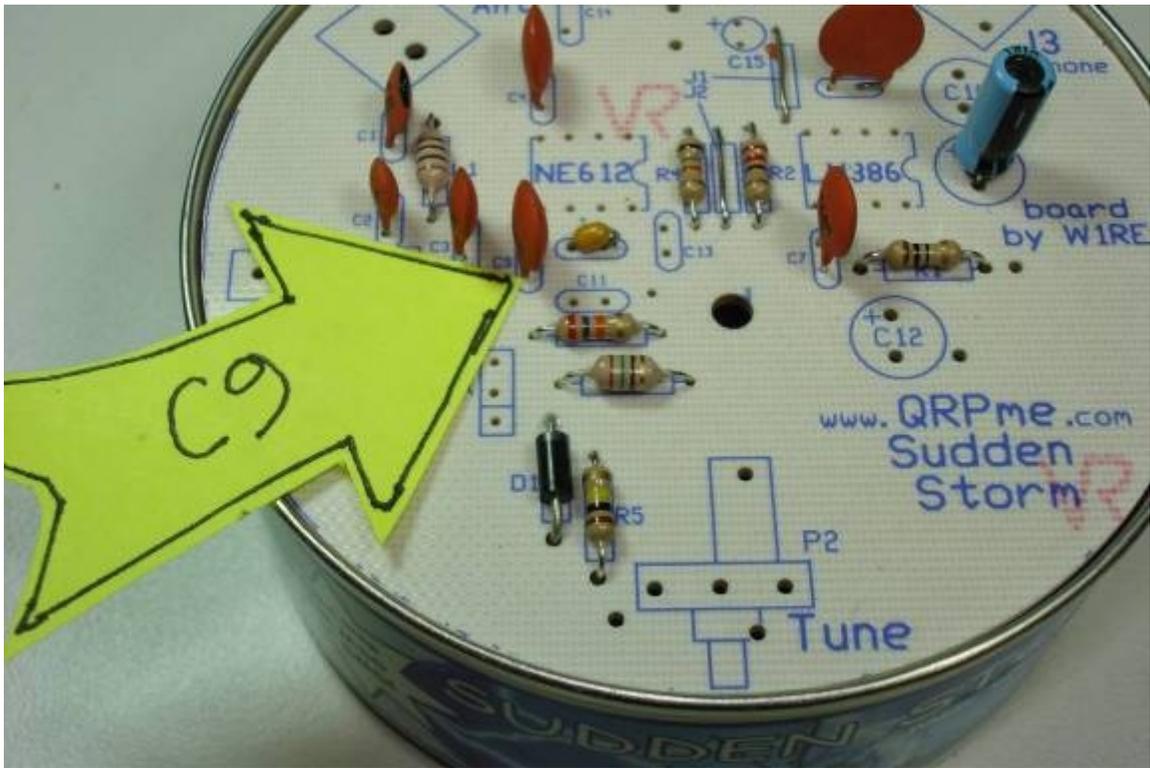
Install a 10 uF electrolytic capacitor at C6. Electrolytic capacitors have a polarity to their leads. The negative side is usually marked with a stripe and a negative sign on the case close to the lead. Align the capacitor so that the polarity of the leads matches the polarity silk screened on the printed circuit board.



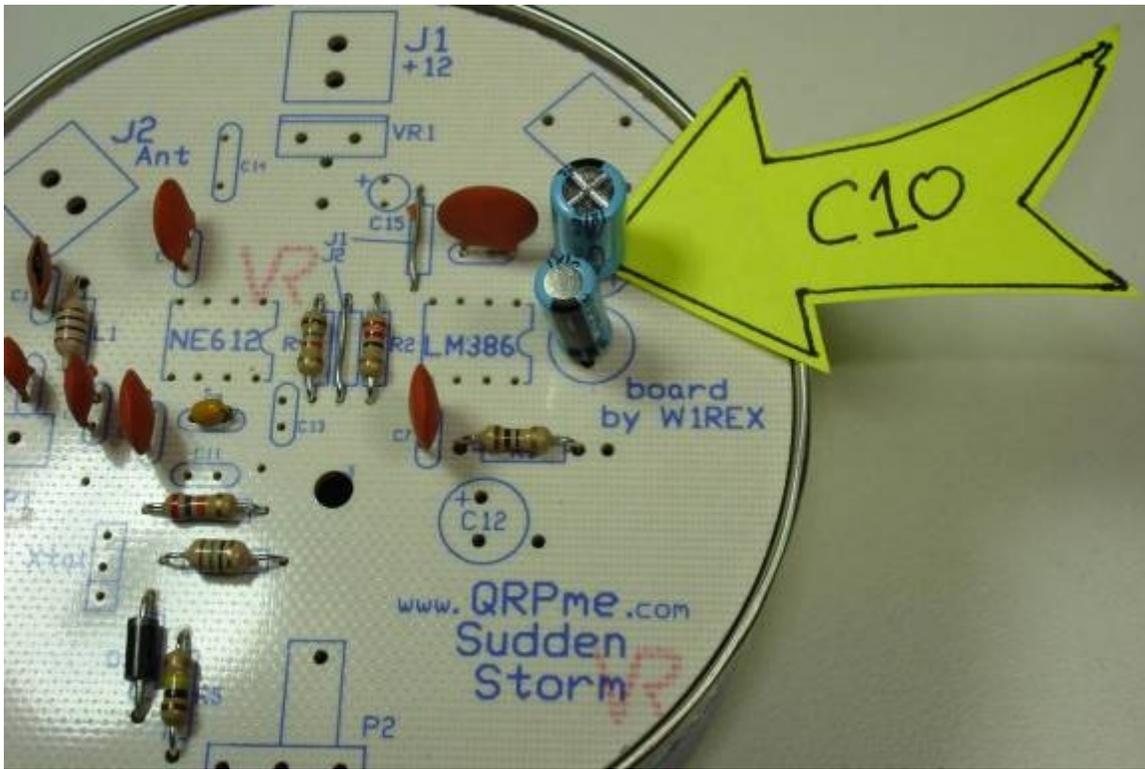
Install a .1 uF cap (104) at C7



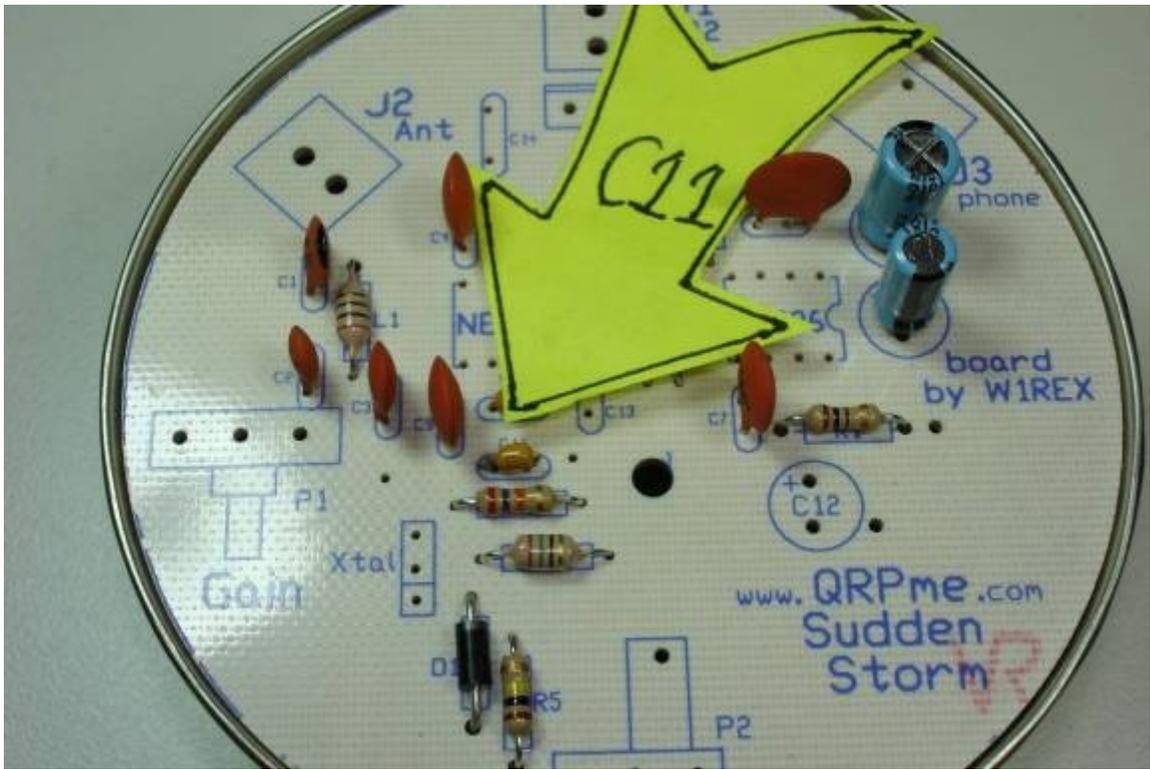
A 100 pF cap (101) is installed at location C8



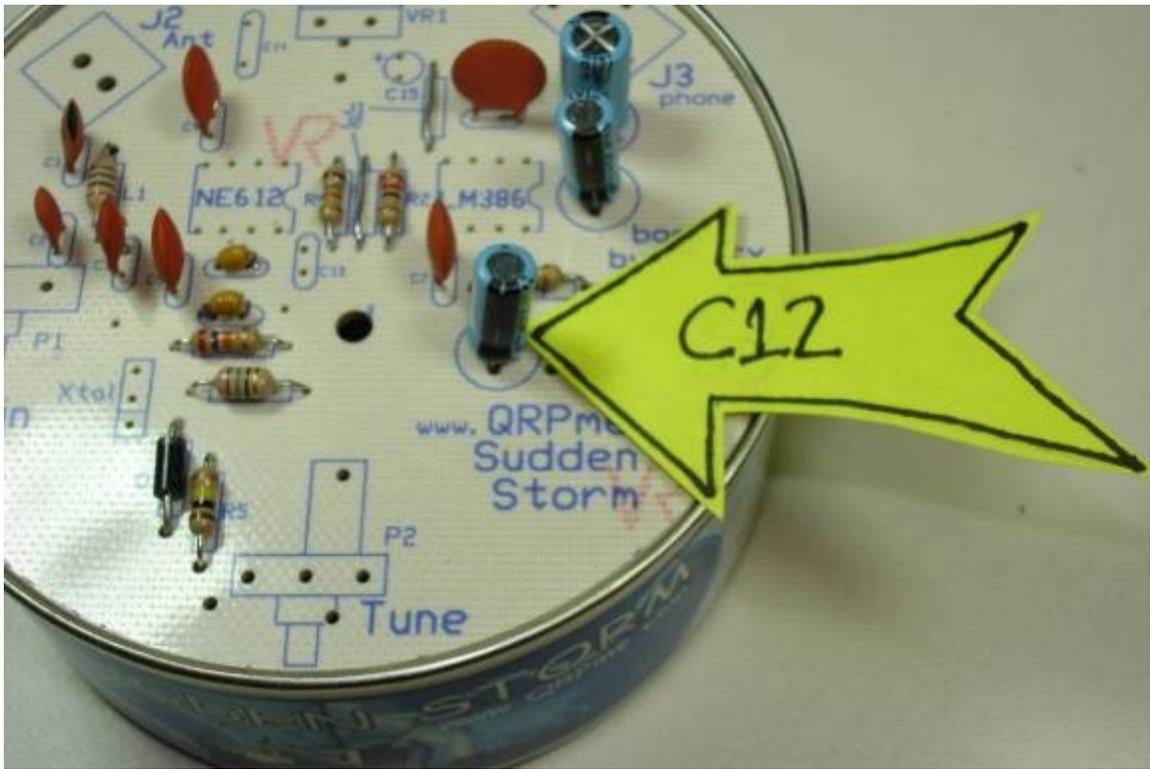
And a .1 uF (104) is installed at location C9



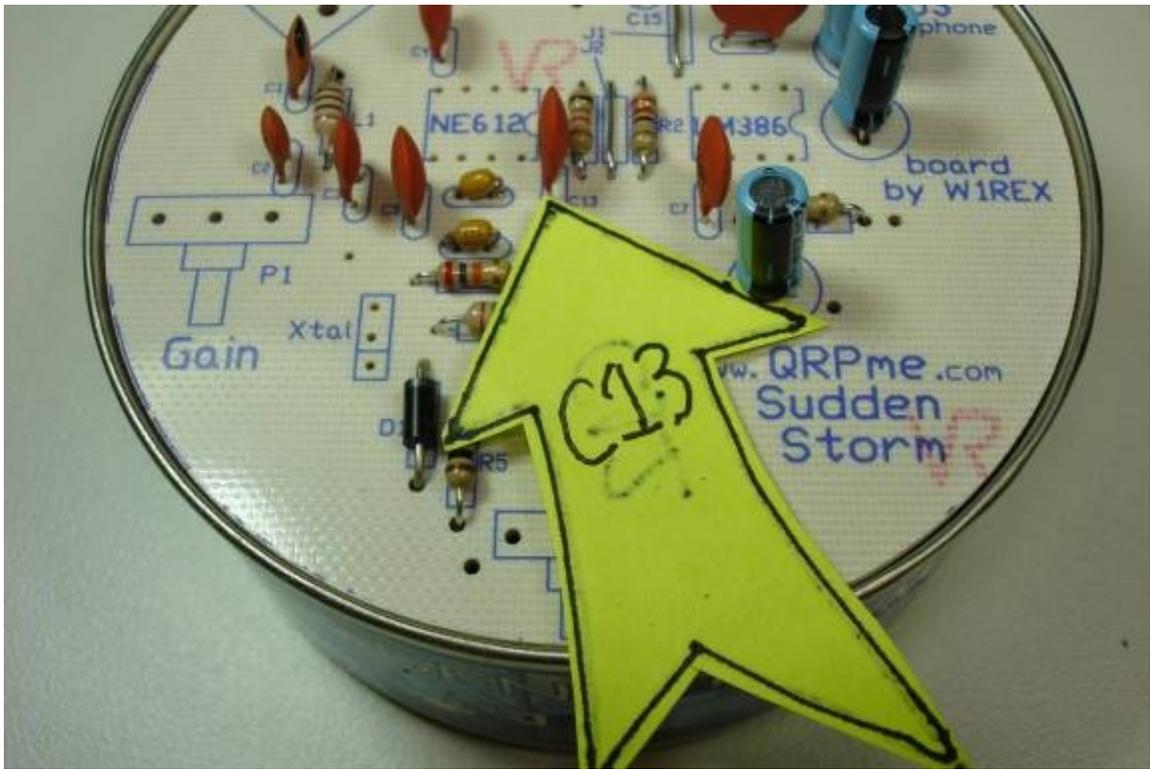
A 100 uF electrolytic capacitor is installed at C10. Pay attention to the polarity of the leads on the capacitor and align them to the markings on the printed circuit board.



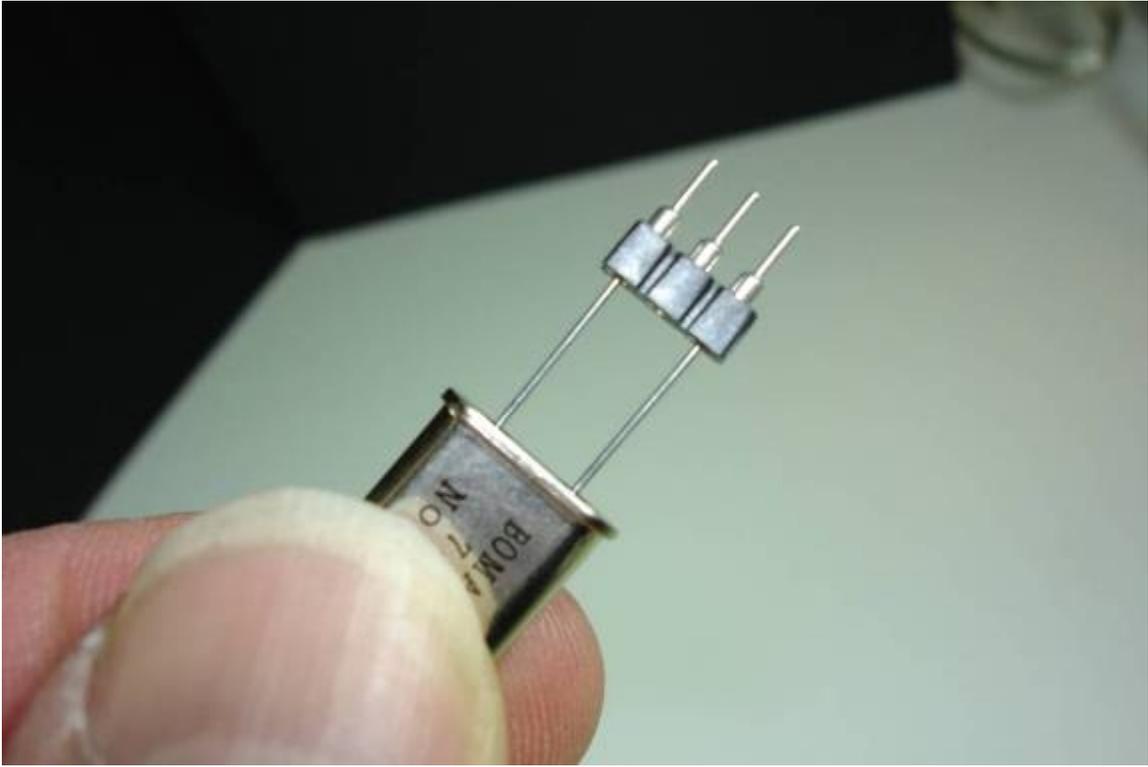
A 100 pF (101) capacitor is used at location C11



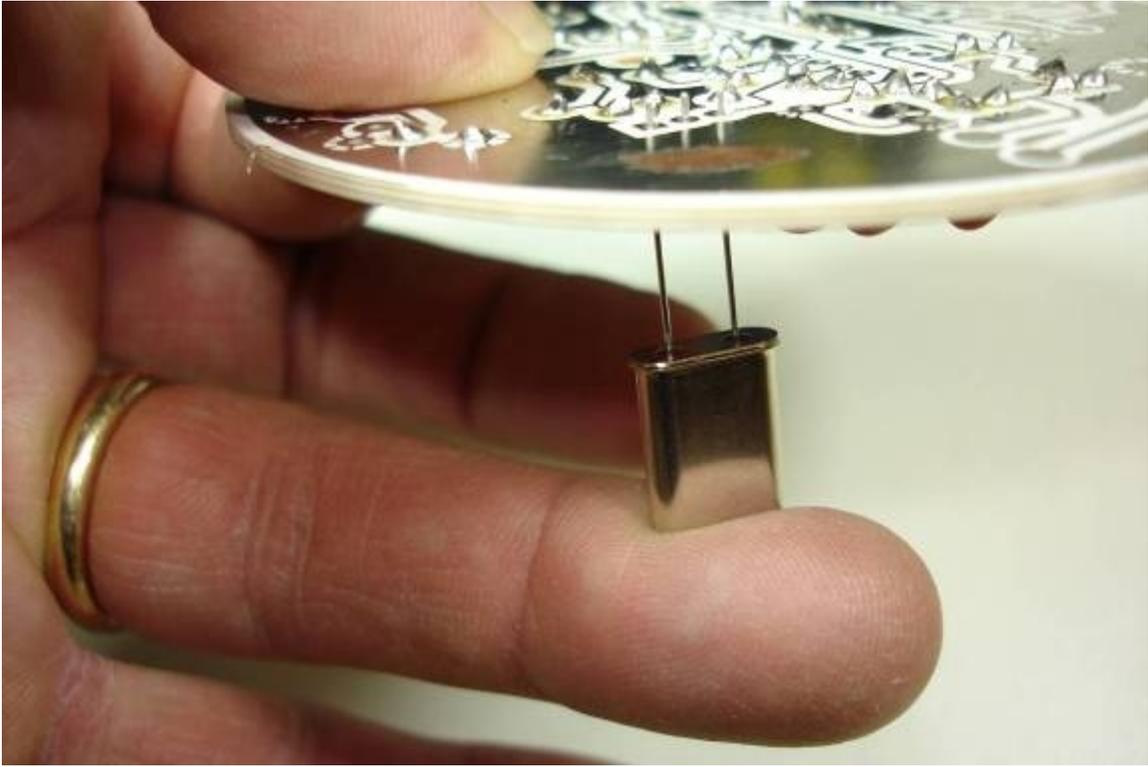
A 22 uF electrolytic capacitor is installed at C12. Remember to align the polarity of the leads.



And capacitor C13 is a .1 uF (104) disc capacitor. Skip capacitors C14 and C15 for now.



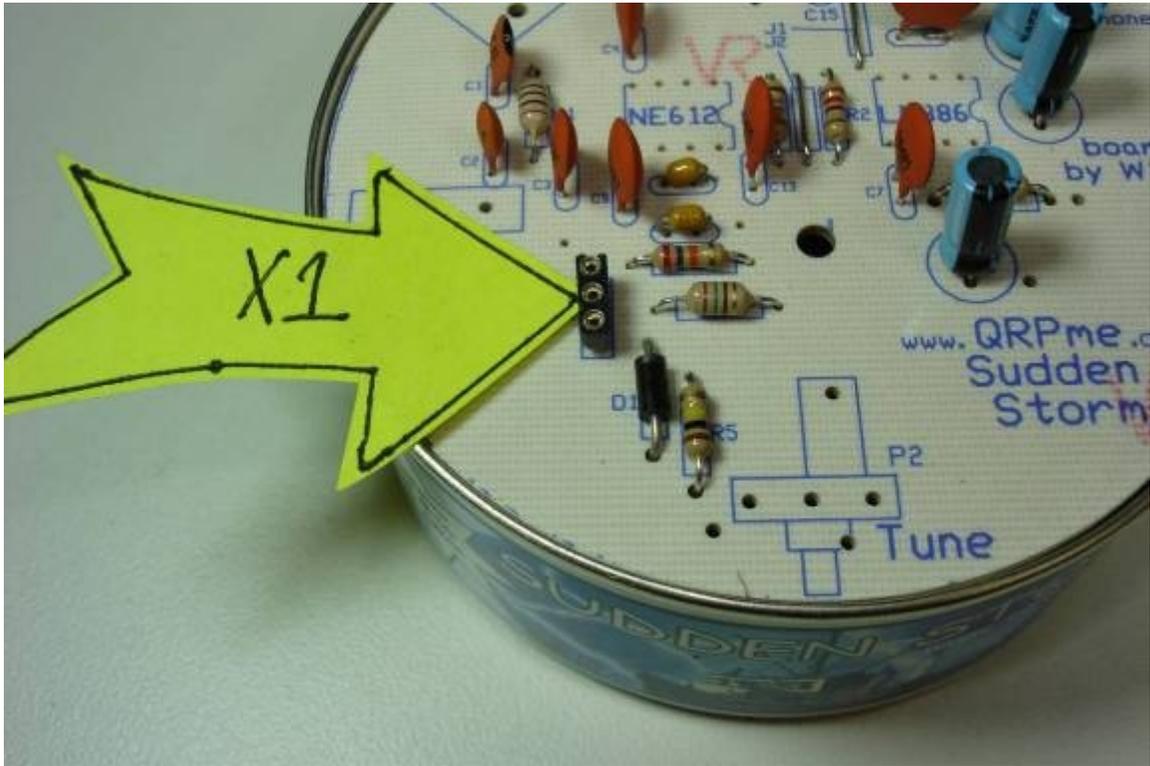
Next prepare the crystal socket. If you elected to use socket pins for the diode, you will only have 2 socket pins left. Split the 2 pins apart and install them on the 2 leads. If you elected to build a 'stock' version, you will have 3 socket pins as shown above. I insert a crystal, or a spare capacitor, into the socket as a holder....



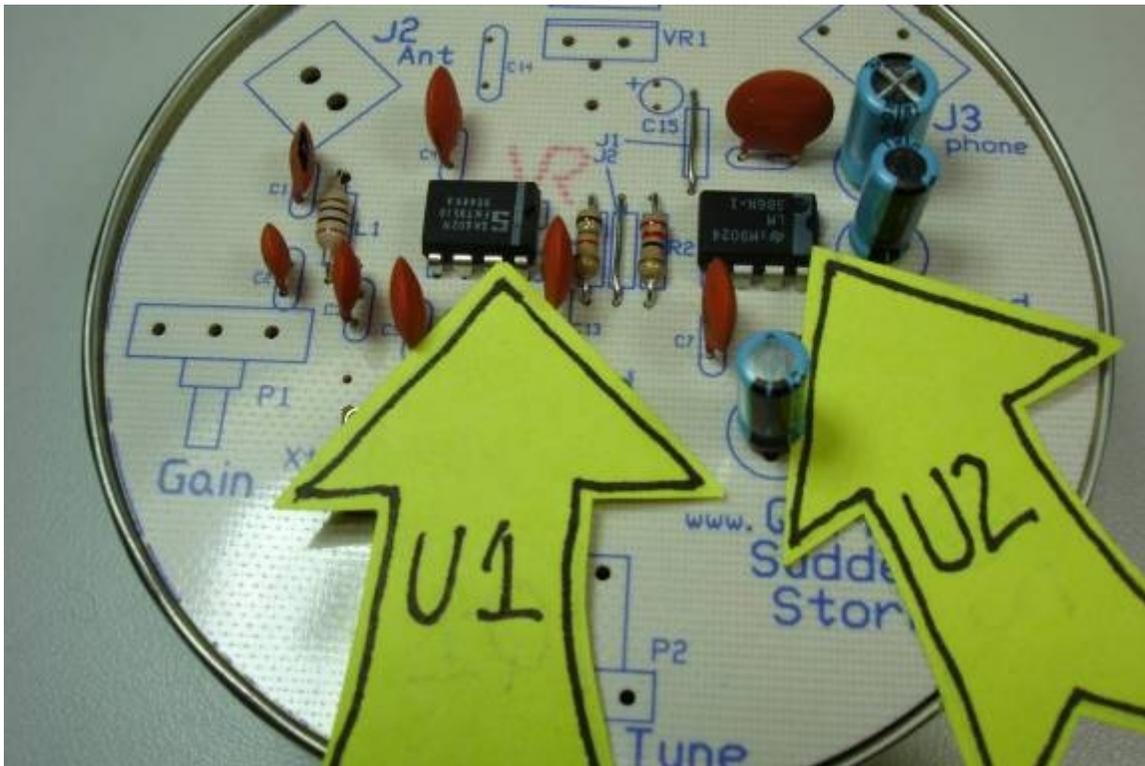
then insert the socket into the pc board at the location marked Xtal. I hold the socket to the board with a spare finger...



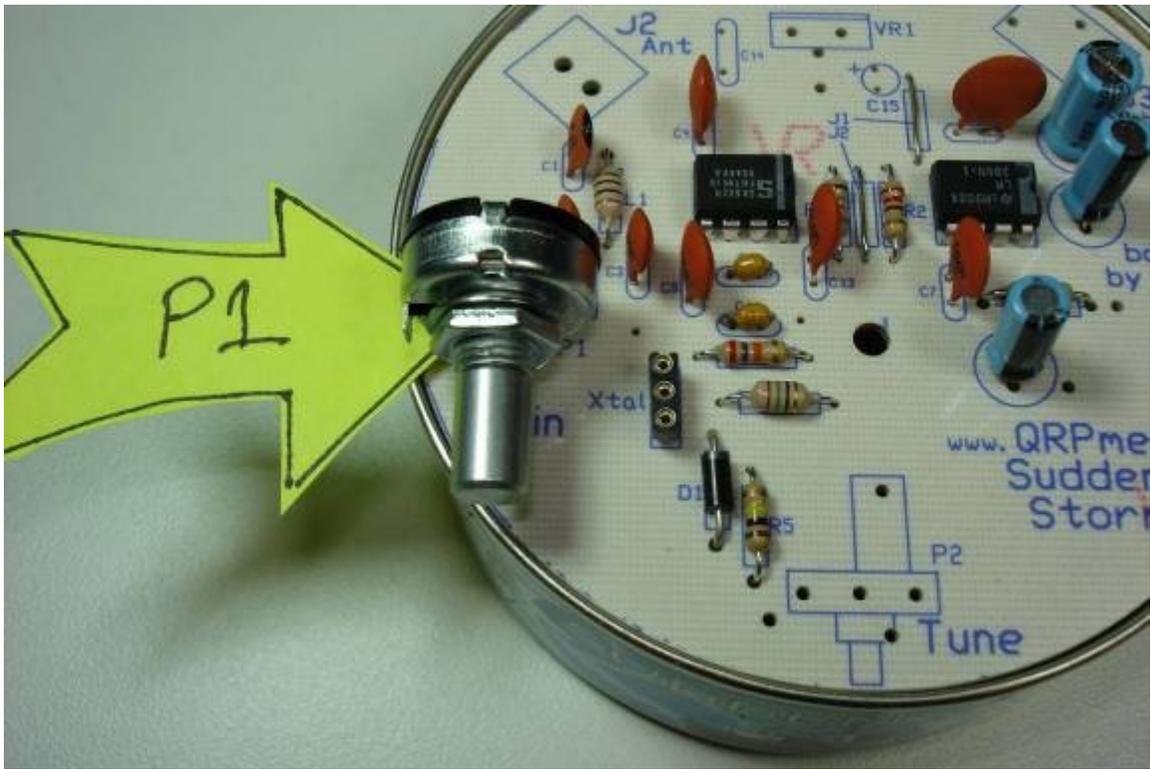
And solder 1 pin to hold the socket to the board while making sure the socket is aligned vertically. A freshly cleaned soldering iron tip with a small 'dollar' of fresh solder is all you need to tack solder one pin. You don't have to feed solder with a 'third hand'. Once the first pin is tack soldered, you can solder the other two as usual because you now have two free hands. Then go back a 'finish' solder the first pin. You could also rest the board on the 'holder' and 'something' under the opposite side to position the pc board flat, then solder using two hands.



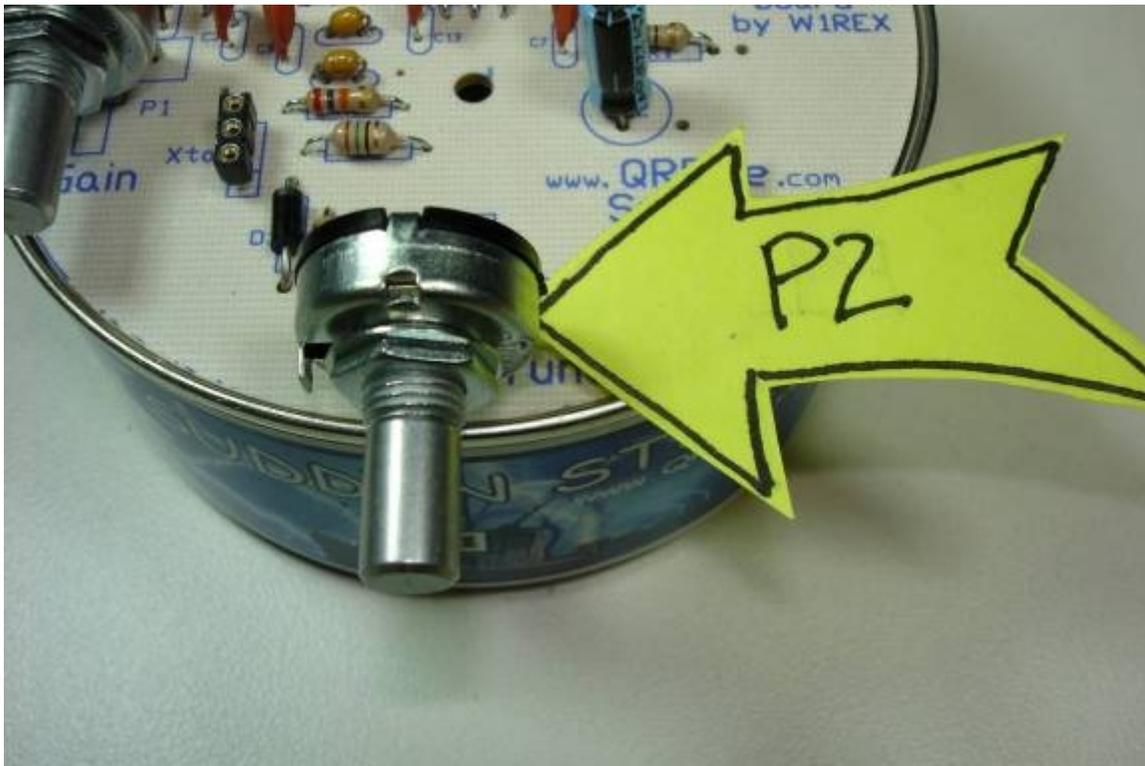
Leave the crystal out for now...



The 2 integrated circuits are next. U1 is the NE602 or NE612 while U2 is the LM386. The Sudden Storm kit now includes sockets for these ICs. Install the sockets at locations U1 and U2. Align the sockets AND the inserted integrated circuits so that the 'notched' end is aligned with the notches indicated on the silk screen.

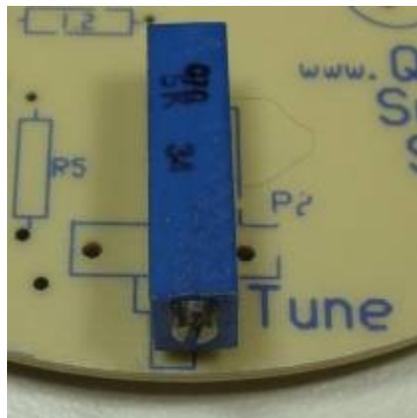


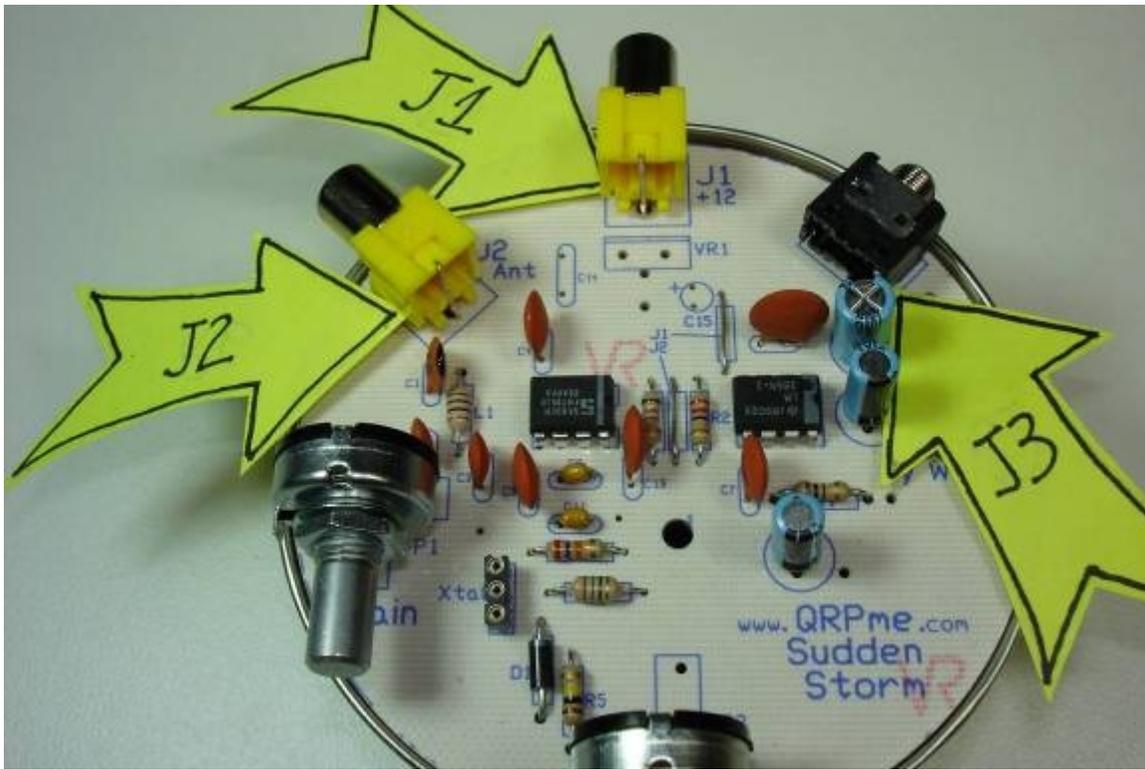
Now mount the 10K potentiometer at the P1 Gain location.



Install the second 10K potentiometer at the P2 'Tune' location. Notice that the silk screen outline for P2 is different from P1.

A multi-turn trimmer pot can be substituted for the single turn, actually 270 degrees, pot supplied. Additional turning resolution results in increased frequency selectivity. A certain degree of creativity is required to attach a tuning knob on these pots but it can be done.





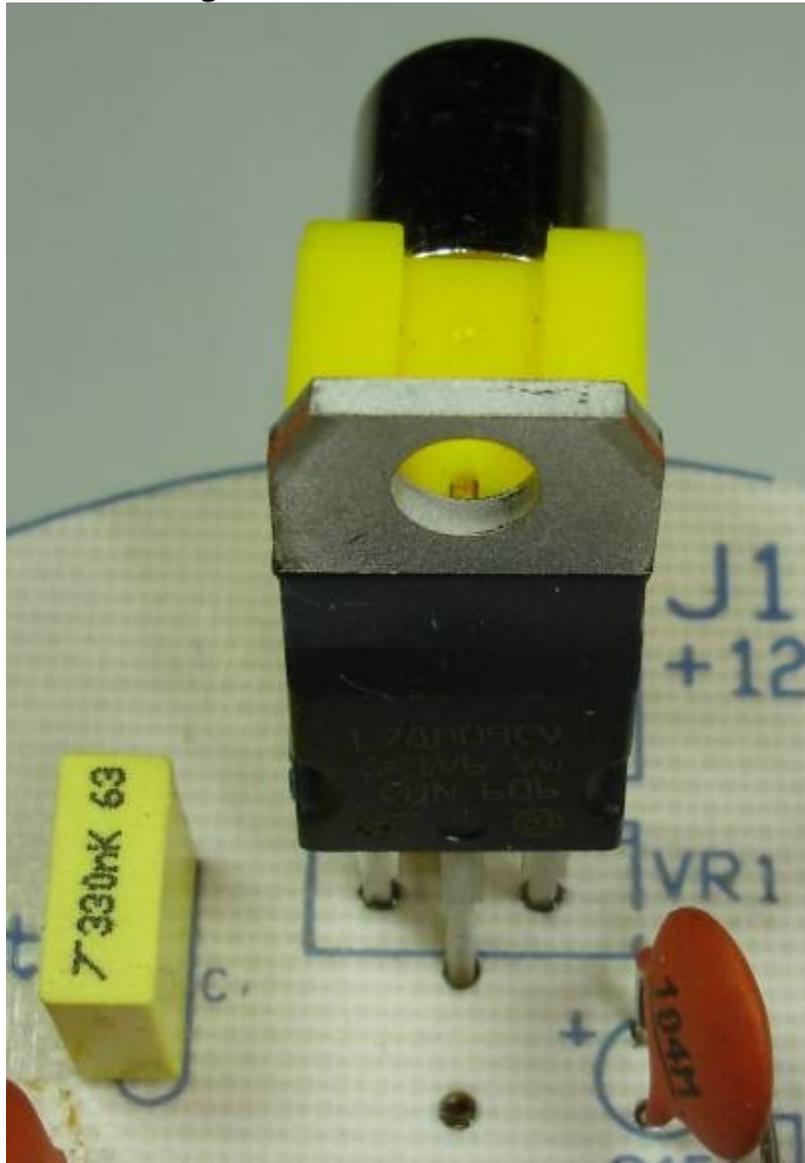
Install an RCA jack at J2 and the headphone jack at J3.

Now it is time to decide how you are going to power your Sudden Storm receiver. George Dobbs's design, article and schematic call for +9V input...possibly a 9V battery. Most Two Tinned Tunas and Tuna Tin 2 kits are being powered by 12V batteries. The QRPme Sudden Storm kit can be configured to operate either way.

Power Options:

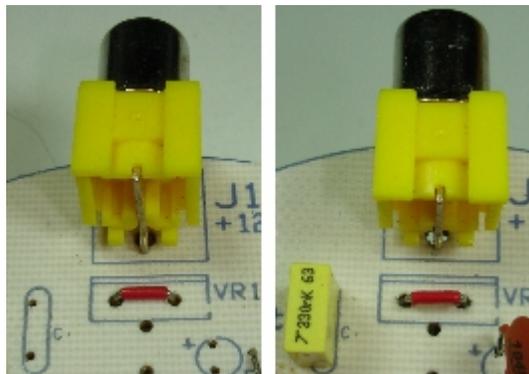
1.)12 Volt operation

If you decide that an external 12V source is desired, install an RCA jack at J1, capacitors C14 and C15 (.33 and .1) and voltage regulator VR1 (78M09) in their corresponding positions as shown below. Cable a 12 Volt source into the completed Sudden Storm kit at jack J1 as designated on the silk screen.



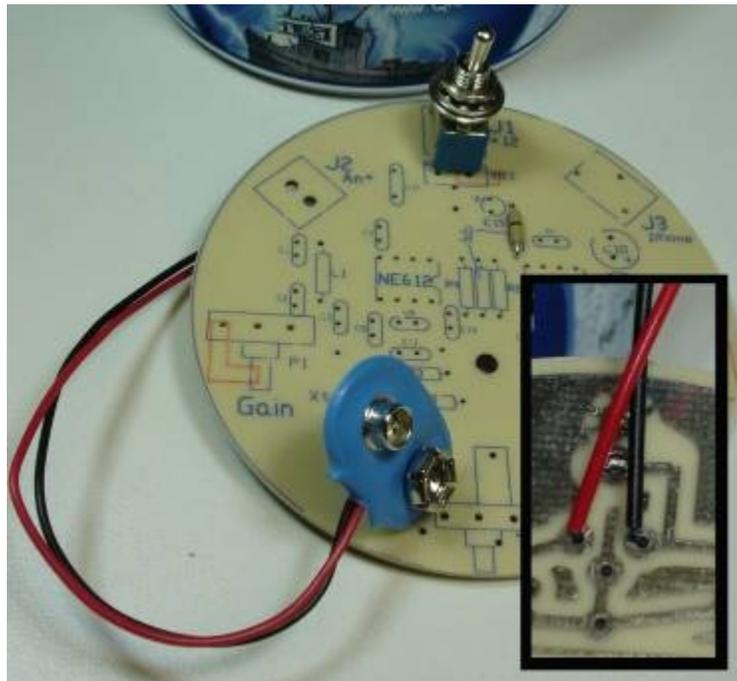
2.)9 Volt external source

Do not install the 9 volt regulator VR1. Solder a small shorting jumper between the left and right side holes of VR1. This jumper shorts the input of the nonexistent regulator to the output so that the 9 volts coming in from the RCA jack J2 makes it to the rest of the circuit. Now you can cable J2 to a 9 volt external source instead of the indicated 12 volts. If you are using a short cable to a 9 Volt battery, you probably do not need to install capacitors C14 and C15 (.33 uf and .1 uf). If you are cabling to an external power supply, the 2 capacitors will help reduce noise. It wouldn't hurt to install them in either case.



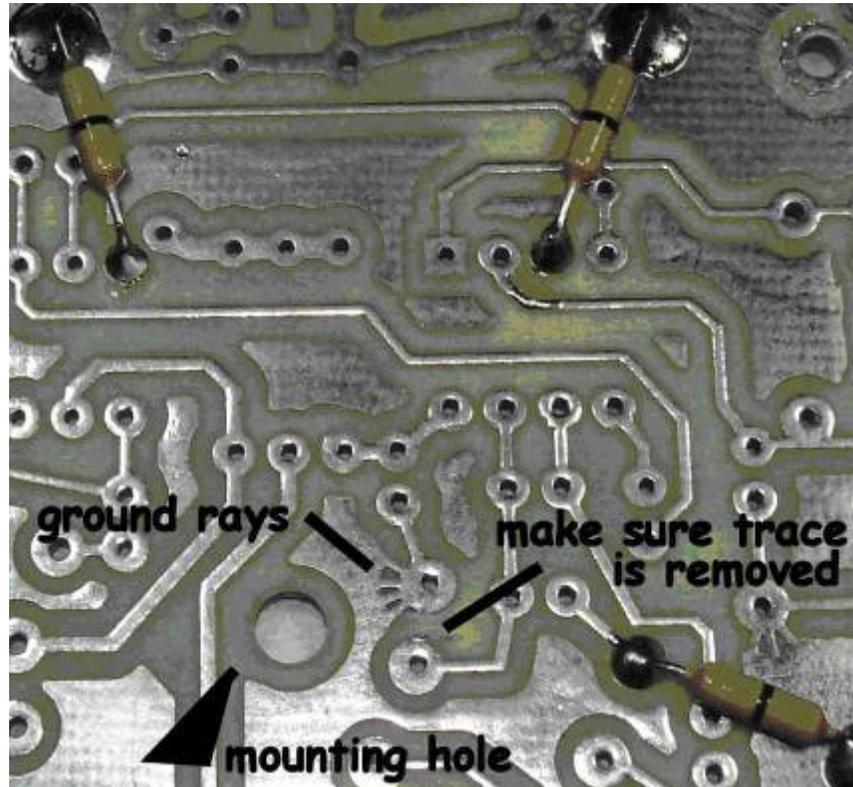
3.) Switched 9 volt battery

This nifty mod was sent to me by Dave Ingram, K4TWJ, of CQ fame. Replace RCA jack J2 with a SPST (single pole single throw) miniature toggle switch and substitute a 9 volt battery clip for the jumper across VR1. The positive side of the battery clip would be soldered to the right/output (C15) side of the nonexistent regulator as viewed from the top side of the board. The 9 volt battery leads should emanate out the bottom of the board. The negative side of the battery clip would then be soldered to the left/input (C14) side of the nonexistent regulator, again as viewed from the top side of the board. When the switch is turned on, the battery ground is switched to the receiver ground bus thus powering up the receiver. Caps C14 and C15 are not necessary. Now you can place the 9 volt battery inside the can and secure the PCB above it. Use a little double stick tape on the battery to secure it inside the can.



Patches to Version 2 pcb

- 1.) Install a zero ohm jumper from LM386 pin4 to the ground plane.
- 2.) Install a zero ohm jumper from NE602/612 pin3 to the ground plane.
- 3.) Install a zero ohm jumper from the ground plane to the side of R3 closest to the xtal.



If there is a trace coming from the pad and hole with the ground rays going to the pad and hole just below it then remove the trace.

Take a sharp Exacto knife and run through the trace close to where it meets each pad.

Now apply a hot soldering iron to the middle area of the trace...in a second or two you will see the trace either start to slide around (the connecting underside glue has melted) or just lift right off.

Clean up the area and remove any residue of the trace....



Now you can install the crystal...

Before powering up the board, measure the resistance between several pins:

Ground and NE602/612 pin 8 = 12.3K ohms

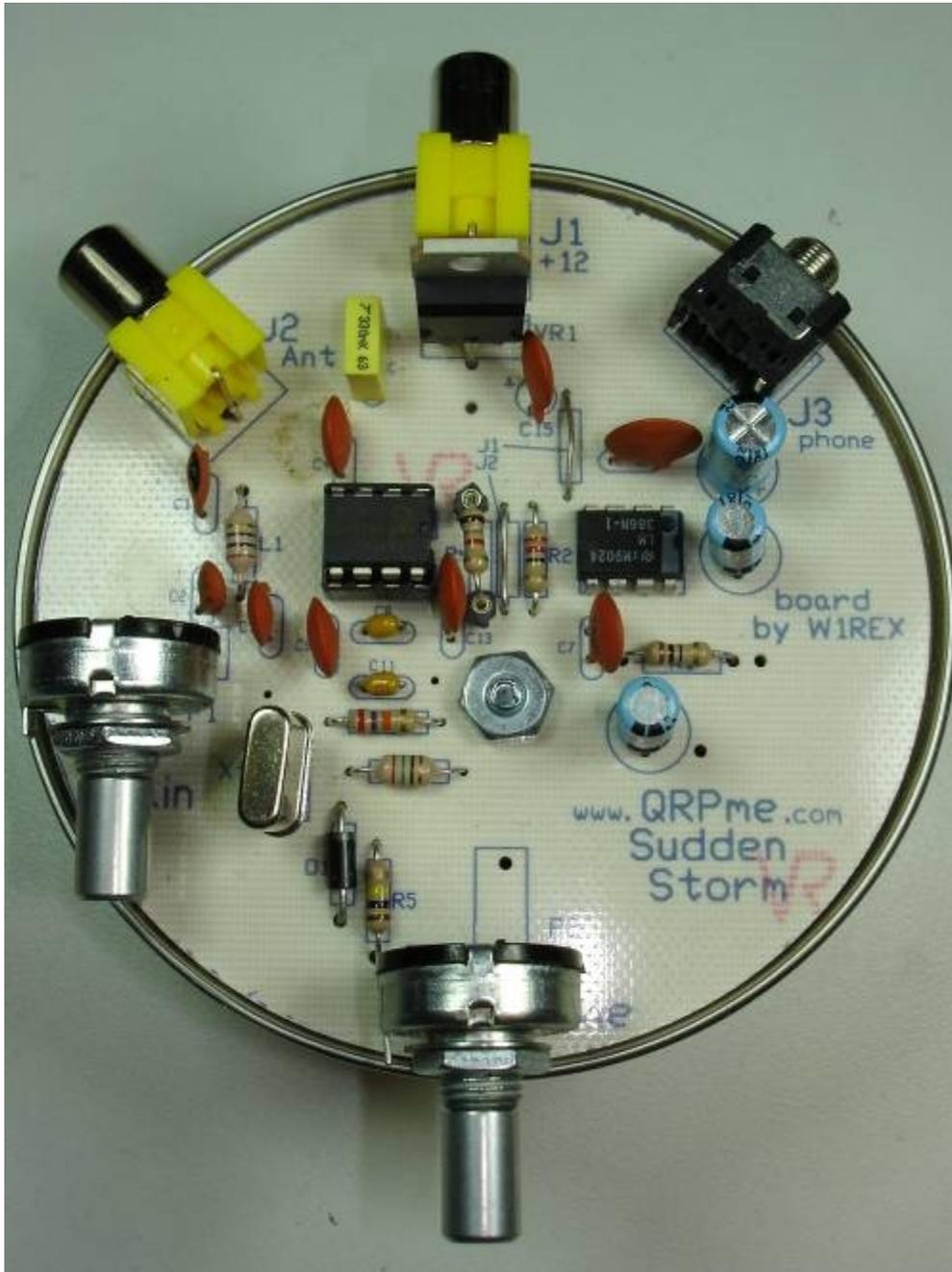
Ground and LM386 pin 6 = 10.6K ohms

A reading close to zero ohms here would indicate a solder bridge between power and ground traces or a misplaced resistor. The 10.6K ohm reading is also the DC load resistance seen by the +12 Volt output of the voltage regulator.

Jumper 1 and NE602/612 pin8 = 1.8K ohms

Jumper 1 and LM386 pin 6 = 22 ohms

Both of these readings are the voltage dropping resistors to the ICs.



Mount the board on the can, insert the bolt through the can and tighten the nut to clamp the board to the can. Don't over-tighten!



Congratulations!

You're done!

