mcHF Hardware modifications as of November 27, 2014
as compiled by KA7OEI for
Version 0.3 UI and Version 0.4 RF boards

Comment to new builders – PLEASE READ:

It is ***strongly recommended*** that new builders <u>do not</u> make these board modifications, other than those involving simple ***<u>substituting/changing of component values</u>,*** before verifying that their transceiver is fully operational.

<u>ONLY</u> after you have verified that the transceiver operates normally and have worked out other problems should you proceed to perform the actual circuit modifications and changes, ***one at a time***!

Note: *Most, if not all, of these modifications may be applicable to board versions other than the Version 0.3 UI board or Version 0.4 RF board, provided that care be taken to accommodate board/circuitry differences.*

- Use 47uH inductors instead of 4.7 in a few locations (RFC5, RFC6 and RFC7) - one possible symptom being low TX power on lower bands (mostly due to RFC5 and RFC6.) The updated BOM reflects this. (Not really a mod, though as the diagram was correct, but the original BOM had a typo.)

- **Do NOT install Q2.** (Not really a mod, either!) The updated firmware (versions newer 185 or newer) make this obsolete: ***Its installation will degrade sensitivity.*** There is no need to install resistor R40 (1k) either. At the time of assembly, capacitor C64 may be optionally replaced with a jumper *(e.g. a zero-ohm resistor, piece of wire or "blob" of solder)* as the DC-blocking function C23 performs this function with Q2 removed. ***There is no need to remove C64 on already-built boards as its presence does not degrade performance.***

- If, for RF board Q1, you use a BFR93R or BFR93AR, it will be required that its leads be bent backwards over the body of the device before soldering. If the ***more-common*** BFR93 or BRF93A is used, the footprint is correct. ***Note:*** *The Bill-of-Materials dated <u>prior</u> to 20141116 called out a Mouser part number that referenced a BFR93AR, requiring that the leads be bent before soldering. As of 20141116 this has been corrected. The Farnell part number is believe to have been correct.*

- The use of a low-dropout regulator for U5. Not really a mod, just a different part from the "original" BOM.

- Tantalum capacitors C5 and C6 are shown backwards on the footprint of boards ***prior*** to the version 0.3 board. The silkscreen is correct on the Version 0.3 board.

Important note:

- Some of the RFI/noises that can reduce sensitivity – particularly those that remain ***after*** all of the modifications below are completed ***(e.g. the "Helicopter" noise from the LCD's update)*** can be significantly reduced by placing a metal shield between the Ui and RF boards – even if it is ***not*** grounded to anything. This shield should be covered with an insulating material to prevent it from shorting against any components/connections.

More recent modifications:

Low Frequency oscillation prevention of the Final Amplifier stage:

It has been observed that at higher PA bias settings that the final amplifier stage can oscillate at relatively low frequencies, specifically in the range of 100 to 1000 kHz. This occurs due to the fact that inductor RFC8 and C106 can form a low-Q "tank" circuit and when the finals are heavily biased, oscillate at very low frequencies due to very high DC gain.

The symptoms of this may be a bit difficult for the average user to notice, but these can include:

- RF power when keyed down in SSB mode, but no audio present – particularly power that goes up suddenly with higher PA Bias settings.
- A noticed higher SWR reading on a frequency that you ***KNOW*** to have a good match. This is due to spurious oscillations at frequencies ***other*** than your operating frequency being reflected.
- Higher than expected DC operating current and/or hotter finals.
- Reports of "oddly" distorted audio – particularly if it seems to clear up if you reduce the PA bias.
- Sudden and unexpected loss of loss of RF power at times, but proper RF power at other times.
- With an oscilloscope attached to the junction of RFC8 and C106, this oscillation will be see when the transmitter is keyed, with the output terminated into a 50 ohm load, with no RF drive, at high RF levels.

The fix:

- Placing another capacitor in parallel with C106. A 10uF, 16 volt ***tantalum*** capacitor is suitable for this. The value isn't critical and anything from 4.7uF to 22uF should work just fine.
- If this turns out to be insufficient *(I'd be surprised...)* parallel a 10 ohm resistor across RFC8 as a "Q" spoiler.

This is a <u>highly-recommended</u> modification!

* * * * * * * * * * * * * * *

Increase of conduction current through PIN Diode D3 on RF board:

With the value of R54 set to 22k it had been observed that the current through PIN diode 22k was on the order of 200uA which, according to the data sheet for this device, would set its series resistance to approximately 30 ohms.

In testing, I found the insertion through D3 to be on the order of 4-6dB, which represents about 1 S-unit. Practically speaking, this was difficult to detect when test equipment was used to analyze the receiver sensitivity – probably because of the fact that the receiver is noise-limited elsewhere in the signal chain.

Nevertheless, the modifications to reduce this loss are quite simple:

- R53 = 220 Ohms
- R54 = 3.9k

This combination yields a (measured!) current of 1.03 mA through D3 (do not forget that there is a 0.6 volt drop across D3 and some resistive losses in RFC2 and RFC3) which, according to the specifications should set the series resistance of D3 approximately 6 ohms. The voltage across D4 is only 0.26 volts, which is very comfortably below its "ON" threshold - especially considering the fact that these are PIN diodes and will not react to incidental RF waveforms in the way "normal" diodes would.

In transmit the current through D4 would be approximately 19 mA, well within the safe drive capability of RF board's U7's outputs.

* * * * * * * * * * * * * * *

Improvement in receiver low-frequency response:

Capacitors C71 and C73 on the RF board, near U16, cause excessive low-frequency rolloff. These capacitors should be replaced with zero-ohm resistors (e.g. jumpers.)

The DC blocking function that these capacitors provide is actually redundant in that C26 and C31 on the UI board perform the same function.

* * * * * * * * * * * * * * *

The addition of sink current for the PA bias:

It was observed that the voltage at C96/RFC7 was not properly following the setting of the **PA Bias** adjustment. This turned out to be due to the fact that U18, an LM2931, like many regulators, has a ***minimum*** load current requirement for regulation.

If you look at the diagram you will note that there is ***NO*** DC loading on the DC output from this regulator as the gates of the PA transistors are purely capacitive. If leakage currents from U18 exceed those of C96, C101, Q5 and/or Q6, the voltage on this line can exceed the regulated output. Similarly, if there ***are*** leakage currents through C99 and/or C100 due to the components, board contaminants, moisture or other sources, this can also cause the voltage to rise.

What this means is that the bias voltage can not only exceed the setting for **PA Bias**, but it may not turn off when in receive mode which could mean that the PA transistors may conduct current and generate heat ***EVEN IN RECEIVE MODE!***

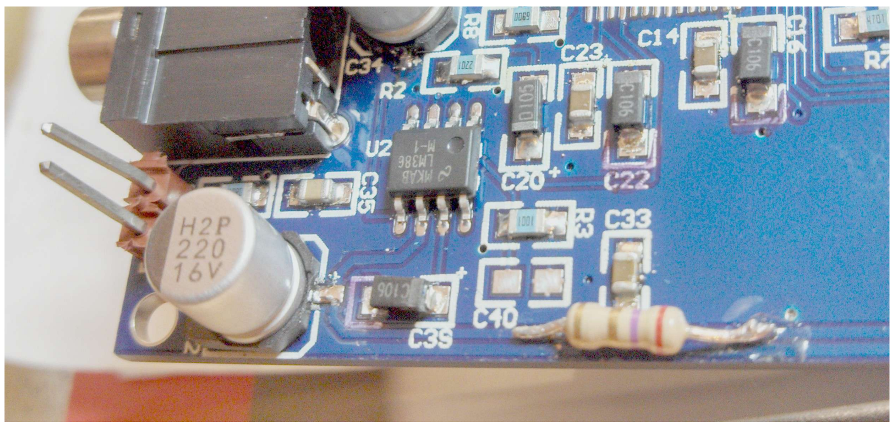
The fix is simple:

> Install a resistor in parallel with C96. The value of this resistor may be anything from 1k to 10k: I used 2.2k since I had plenty of them left over from construction.

* * * * * * * * * * * * * * *

Reduction of conducted audio on the 8 volt line that causes audio feedback:

The insertion of a 4.7 ohm, 1/2 watt resistor in series (or two 10 ohm 1/8 watt resistors in parallel) with the +8 volt line, near the LM386 audio amplifier.



The 4.7 ohm resistor added to the UI_8V line feeding U2, the audio amplifier.

This is an important modification that fixes one source of oscillation at high volume levels.

I'd originally suggested a 2.7 ohm resistor, but I later found 4.7 ohms to be better, as Chris pointed out.

This modification prevents audio from appearing on the 8 volt line which, in turn, causes audio to appear on the ground busses and other power supplies in the radio which can cause feedback, particularly at high volume levels with wider audio filters.

If you have done the other modifications, and you still get feedback when using a speaker, at high volume levels:

It has been noted that even with this modification having been done, if all of the other noise reduction modifications described in this are also done the receiver sensitivity is increased enough that the addition of just the 4.7 ohm resistor may not be enough to prevent feedback at high volume levels. This can sound like a howl, or just a “hollow” sound at high volume levels, with worsened feedback in the 300 Hz setting.

If this describes what is happening to you, this fix is also quite simple: Increase the capacitance at C32. It has been found that adding another 1000 uF of capacitance across C32 seems to prevent audio feedback, having done all of the other modifications.

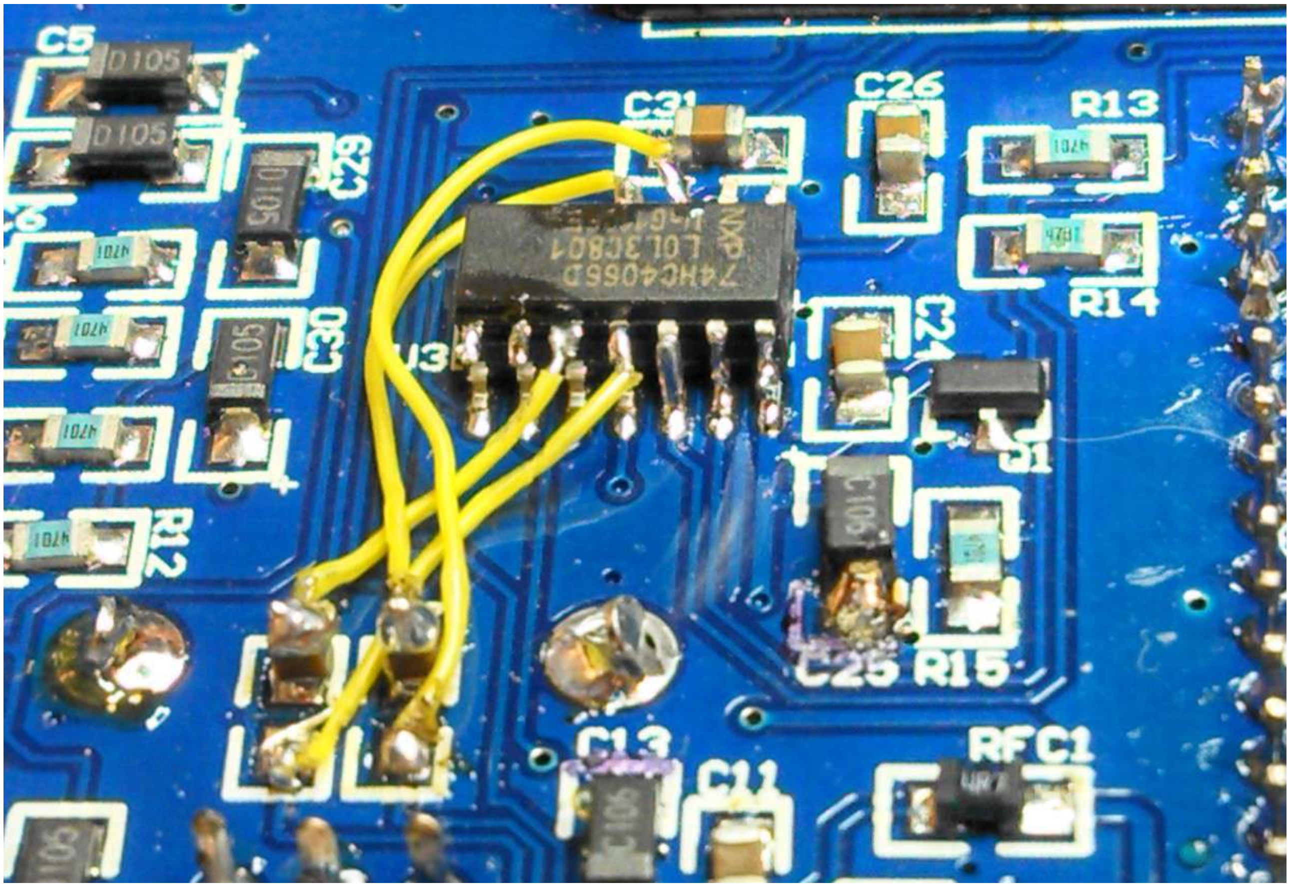
You may also prevent this feedback by increasing the value of the 4.7 ohm resistor even more at the expense of a reduction of audio output power.

* * * * * * * * * * * * * * *

Prevention of RX audio “breakthrough” on the TX mixer:

The installation of a which, which we call "U3a", is a 74HC4066 switch, piggybacked atop U3, and is a pretty easy mod if you have soldered the rest of the board!

This fixes a source of distortion/oscillation in the receiver, particularly at high volume levels and on high bands: **This is a *very* important modification that all users of this transceiver should perform**.



The "U3a" modification, piggypacking another 74HC4066 atop U3 to prevent feedback into the receiver.

Take a 74HC4066 chip, of the same type and package as U3 and piggyback it atop U3 (on the UI board) and use its power supply and control signals. This chip will now be able to turn on/off the TX I and Q signals with a bit of fairly simple rewiring. Let us call this new chip "U3a"

This modification is relatively simple and it seems to be effective in solving this particular feedback problem: This 'mod, in addition to the installation of a 4.7 ohm resistor in the 8 volt supply line for U2, the audio amplifier (see message #144) has caused my receiver (board revision 0.3) to be (almost) completely stable on all bands at all volume levels.

The procedure is approximately thus:

- On the new 74HC4066, U3a, bend pins 5, 6, 7, 12, 13 and 14 straight down using a VERY

FINE pair of needle nose pliers or tweezers. Bending these leads down will cause them to be just long enough to allow them to be overlaid atop U3 and connected to it.

- Bend pins 1, 2, 3, 4, 8, 9, 10 and 11 of U3a straight out using a the same pair of very fine needle nose pliers or tweezers.
- Set this chip atop U3, making sure to align pin 1 with pin 1.
- Using a VERY FINE tip, solder pin 14 of U3a to pin 14 of U3.
- Align pin 7 of U3a to U3 and solder.
- Now solder pins 5, 6, 12 and 13 of U3a to U3.
- Remove capacitors C7 and C8 and stand them on end, attaching them to the pad closest to U3, the upper pad. This will leave the lower pads of C7 and C8 empty. *You may turn C7 and C8 around, the "open" end facing U3 rather than standing them on end, if you prefer.*
- Using fine-gauge wire such as #30 AWG wire-wrap wire, run leads to sections 1 and 4 of U3a. For example, the wires connecting across the connection of C8 would go to pins 1 and 2 of U3a and the connection of C7 would go to pins 10 and 11 of U3a. Which wire goes to which side of the switch does not matter, nor does which connection goes to which switch.

How this works:

U3 is used to switch the A/D input of the codec (U1) between the Line input and receive I/Q audio lines. This chip already has a complement of logic levels that follow the PTT (Push-to-Talk) lines.

When in receive mode, the TX I and Q lines (AUDIO_IN_I and AUDIO_IN_Q) are disconnected and left floating. Fortunately, U19 on the RF board, the lowpass filters, remain stable in this condition. Importantly, this method of muting the TX audio does not cause a DC shift which should prevent/minimized a broadband "click" or thump when going to transmit mode.

Note that the other two sections of "U3a" (2 and 3) are connected to the inversion logic (Q1 on the UI board) and are active when in receive mode and the corresponding I/O pins (pins 3, 4 and 8,9) are left floating (which will cause no problems on a '4066) - but these may be useful for something else...

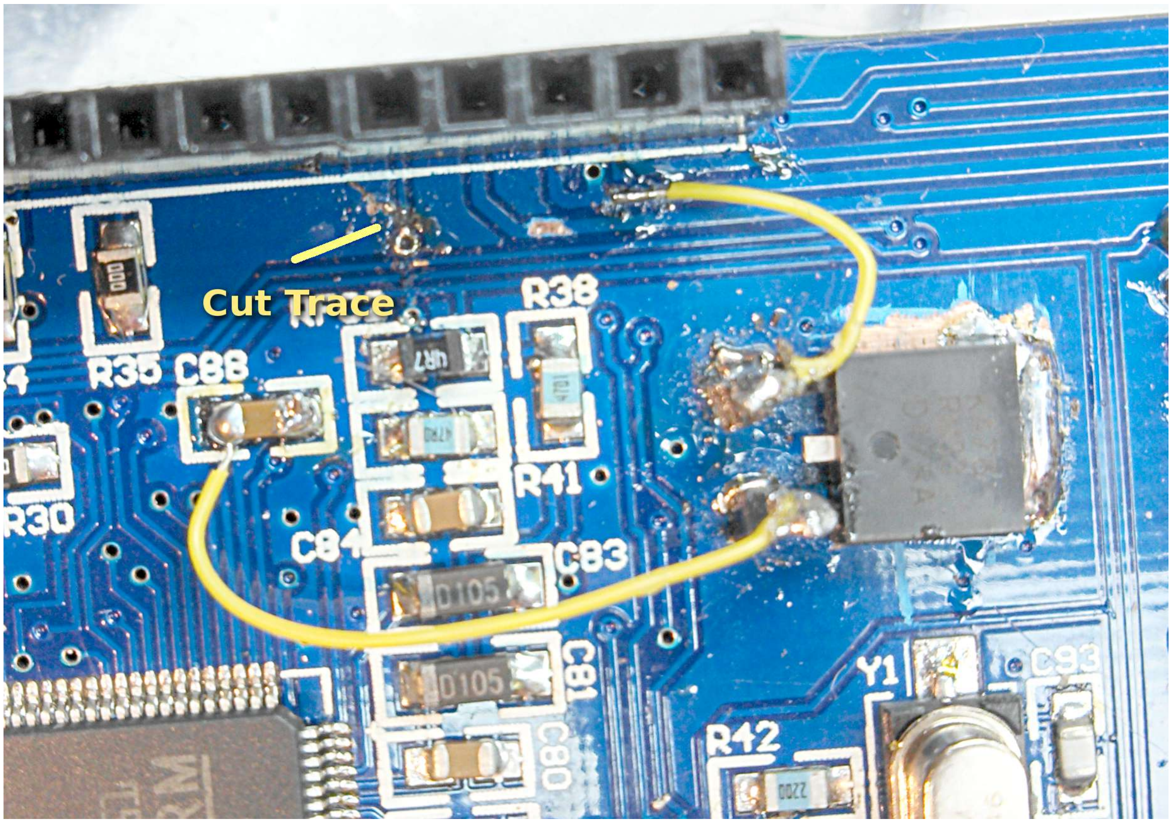
This modification prevents "breakthrough" of the TX QSD mixer (U17) caused by the fact that the Codec's DA is shared both for RX audio and TX modulation. Because the audio drivers for the TX modulator (U20-U23) are powered from 8 volts it is possible for high RX level audio to violate logic levels at U17, turning it "on" on voltage peaks: The result of this is highly-distorted, on-frequency noise/feedback.

* * * * * * * * * * * * * * *

A separate 3.3 volt regulator for the MCU and additional LCD +5V filtering:

The recommended installation of another 3.3 volt regulator on the UI board for the MCU: This mod is actually easier than the "U3a" mod and uses the same type of parts as those that you'd already have gotten (e.g. another 3.3 volt regulator and two more 10uF capacitors.) This reduces one source of low-level noise – much of it in the form of a 1.5 kHz "whine" – getting into the receiver due to the MCU's modulation of current on the 3.3 volt line shared with low-level audio circuitry. Taken together, these two modifications can improve the receiver sensitivity by 2-3 S-units, depending on band.

The trace to isolate the 3.3 volt supply feeding the MCU is indicated on the picture and it should be cut **just above the** via that goes through to the other side of the board. The 3.3 volt output from the added regulator is shown connected to the left-hand side of C88, below.



The added 3.3 volt regulator to deliver an isolated power supply to U4a, the MCU. This reduces a noise source that can get into low-level RF and audio stages. The trace is cut just ***above*** *the via.* ***Note that this picture was taken <u>before</u> the addition of the 4.7 ohm resistors and additional capacitors described below and is included for clarity.***

It was noted that when the above modification was completed that there was residual noise on the 5 volt line so further investigation was carried out. It turned out that the remainder of the noise was from two sources:

- The +5 volt supply for the LCD
- The transmission of noise ***through*** the 3.3 volt regulator.

While the regulator's contribution directly to the 3.3 volt line was significant and moving it to the 5 volt line made a significant improvement, it still put noise on the 5 volt bus which powers devices many other low-level circuits, such as the QSD mixers and, to a lesser degree, the RF signal sources (e.g. local oscillators.)

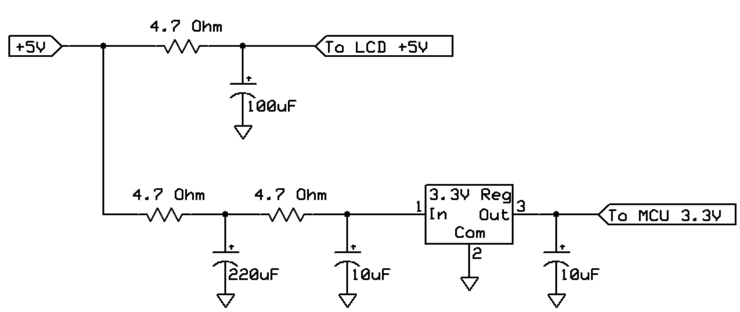


Diagram of the filter/regulators of the LCD and MCU power supplies. A low-dropout 3.3 volt regulator is used.

The diagram below shows the filtering in schematic form.

There is a voltage drop associated with each resistor, but each is well within the range of the associated device.

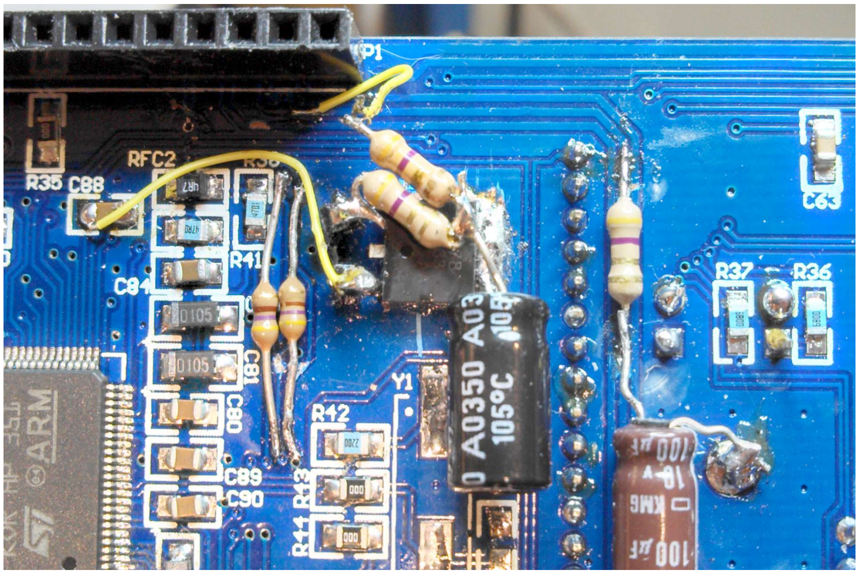
- The LCD has an internal 3.3 volt regulator that has a (worst-case) dropout voltage of 600mV – and less than 250 mV.
- The MCU regulator drops less than 250 mV per 4.7 ohm resistor, leaving well over a volt of headroom for the low-dropout regulator.

The 100uF capacitor, which provides bypassing, on the LCD supply would ideally be placed between the 5 volt and GND at the LCD, but I hadn't done that at the time the picture was taken.

The 220uF capacitor on the input side of the 3.3 volt regulator, along with the 4.7 ohm resistors, form a low-pass filter that provides an impedance path between the noise source at the input of the regulator and the capacitor, greatly improving its ability to filter. The other 4.7 ohm resistor provides isolation between the noise that remains at the 220 uF capacitor and the rest of the 5 volt bus in the transceiver.

Comment:

With the increase in sensitivity due to the reduced receiver background noise there may be the additional tendency for audio feedback to occur at high volume levels, particularly with no antenna connected ***when using a speaker***. This is related to circulating currents on the UI board, some of which were addressed with the addition of the 4.7 ohm resistor in series with the 8 volt line on U2 noted earlier in this document.



The modifications showing the 3.3 volt regulator (center) and filtering and the additional filtering of the 5 volt LCD line (right). The resistor to filter the "tick" (see next section) are also visible.

Increasing the capacitance of C32 to 1000uF can reduce this effect, as can the use of headphones or an external audio amplifier.

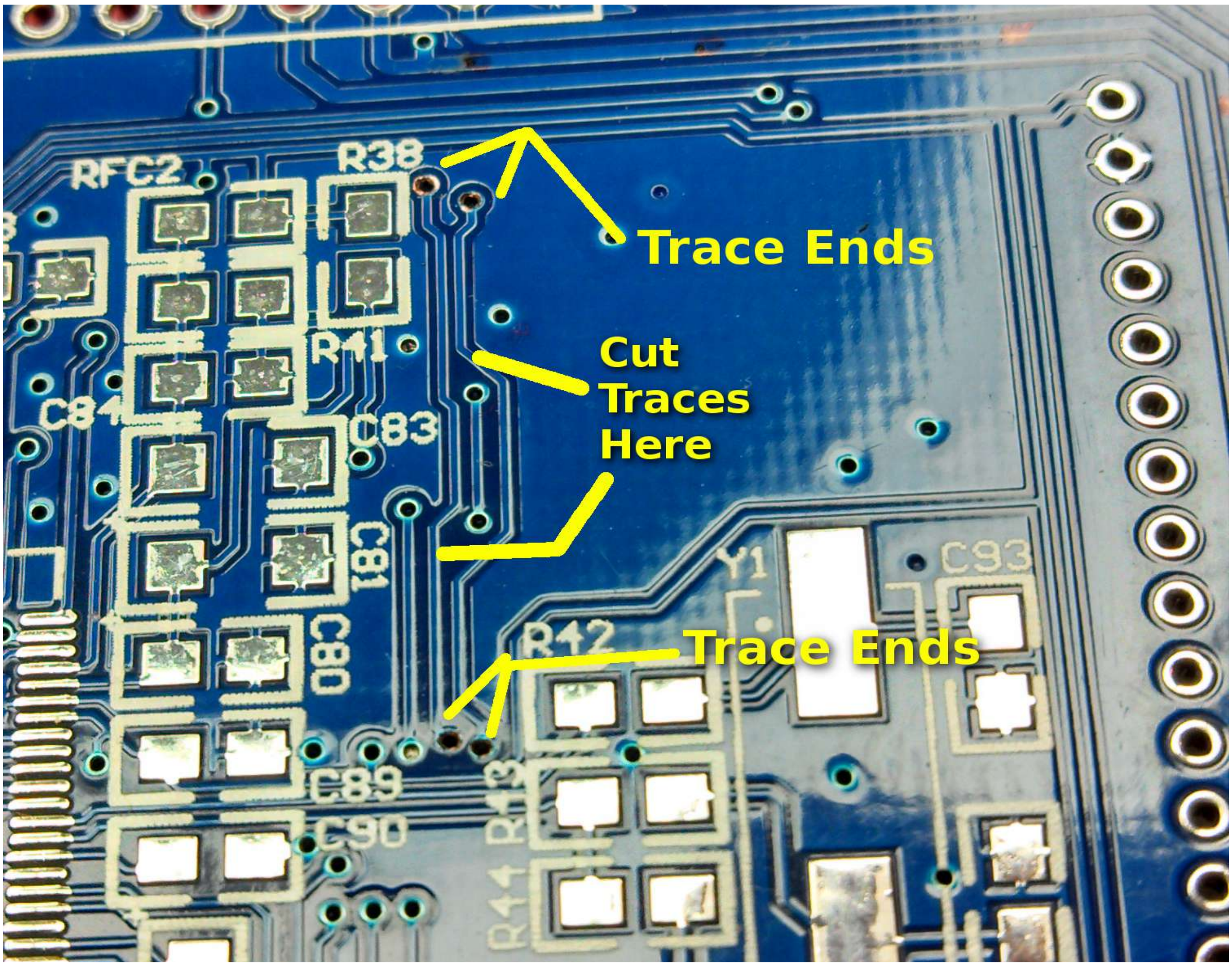
Further modifications to reduce this source of feedback are being investigated – but it's a good thing that the receiver's being improved to where this is a problem!

* * * * * * * * * * * * * * *

Elimination of the 1-second "tick" noise in the receiver – particularly on the higher bands:

If the above modifications are made, with no antenna connected – or even with an antenna connected when listening on higher bands (especially 20 meters and above) a fairly strong "tick" may be heard every second. This is due to the polling of the onboard temperature sensor and the fact that the "fall" time of the on the I2C line from the MCU is extremely fast and rich with harmonic energy. This problem can be easily "fixed" with the addition of resistors on the UI board in series with the SCL and SDA lines which effectively reduce the slew rate of these lines, quashing the RF energy.

To identify the traces into which the resistors are to be installed, refer to the picture below:



This picture shows the location of the traces and where the cuts should be made. This is found to the right and above U4a, the MCU. A picture of a "bare" board is used for clarity.

A reasonable value of resistor to install is 470 ohms and the installed components may be seen in the picture.

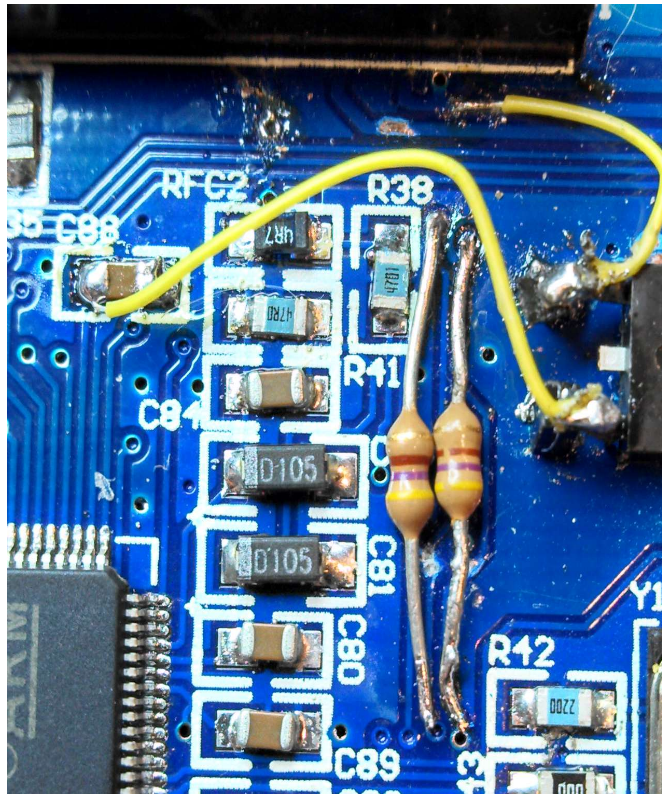
In the picture can be seen the two added 470 ohm resistors. As you can see, I used leaded 1/8 watt resistors, but surface mount resistors can be used, instead if that is your preference. (The 3.3 volt regulator used for the MCU – see the modification earlier in this document – can be partially seen.)

This modification appears to be ***completely*** effective in removing the “tick” noise from the higher bands and does not appear to compromise the operation of the I2C data stream in any way.

To verify the integrity of the I2C data stream – which is shared with the Si570 frequency synthesizer – I monitored the data stream with a digital storage oscilloscope and measured the worst-case slew rate – which was actually the risetime - the result being noted below.

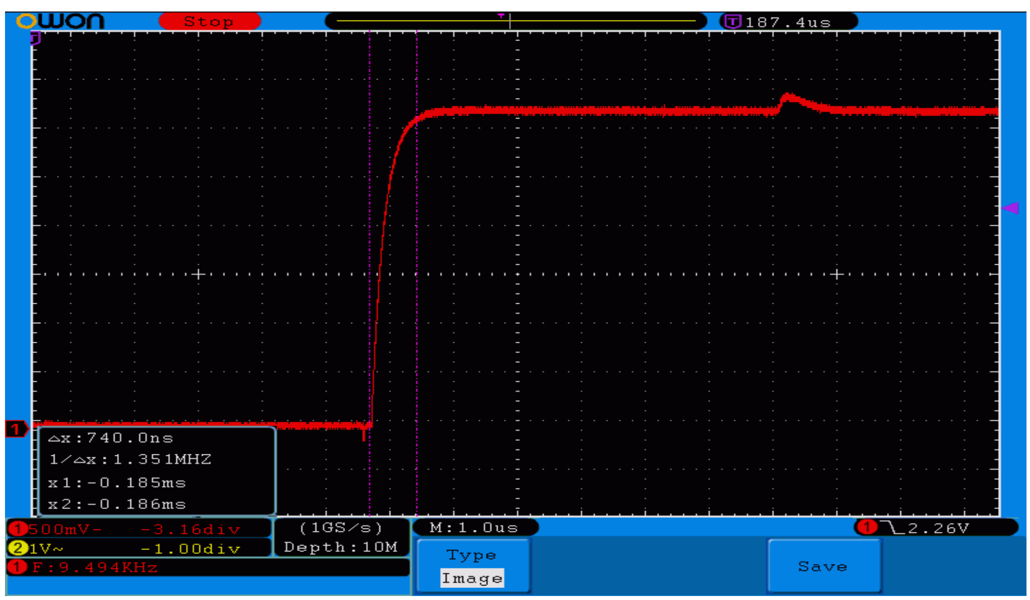
As can be seen, the risetime is less than 740 ns at the 10%-90% points which is a fraction of the I2C clocking rate, well within the time at which the edge of the clock into which the data is strobed in by the peripheral.

It should be noted that R16 and R17 are already present to moderate the slew rate of the data being sent ***from*** the Si570 and MCP9801 temperature sensor to prevent ***those*** devices from causing clicks!



The added 470 ohm resistors in series with the SCL and SDA lines. Note that this picture also shows the added 3.3 volt MCU regulator mentioned above. 1/8 watt, leaded resistors are used, but surface mount resistors could be used, if desired.

(END OF DOCUMENT)



Of the rise and fall times on the I2C, the slowest slew rate was the ***rise*** *time shown in this digital oscilloscope capture, which is well within the acceptable time for the clocking rate!*