

**mcHF Hardware modifications as of September 28, 2015 as compiled
by KA7OEI for the Version 0.3/4 UI and Version 0.4 RF boards**

Comment to new builders – PLEASE READ:

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

ONLY 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. Note that most of the modifications to the Version 0.3 UI boards (and earlier) have been incorporated in some way into the 0.4 UI board and need not be done!*

- 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 prior 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 believed to have been correct for all versions.*
- 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 UI board. The silkscreen is correct on the Version 0.3 (and later) UI boards.
- **Bridge the tops of U8 and U10 with a small piece of aluminum or copper.** This is necessary to convey the temperature of the synthesizer (U8) to the thermometer (U10) so that the TCXO will function. Use epoxy for this rather than cyanoacrylate (“Super Glue”) for better heat transfer.
- **ALWAYS** use the “Frequency Translate” feature: The “RX LO LOW” setting is the preferred mode for a number of technical reasons.

The importance of placing a metal shield between the UI and RF boards:

- 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.

This shield should be covered with an insulating material (*self-adhesive paper, packing tape, plastic*) on **BOTH** sides to prevent it from shorting against any components/connections.

Experimentation has shown that this grounding this shield may or may not offer improvements so feel free to experiment, attaching a short ground lead to various locations and carefully comparing the generated noises to see if there is any improvement.

More recent modifications (RF board version 0.4 and earlier):

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.

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

With the original value of R54 set to 22k it had been observed that the current through the PIN diode 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 and a much lower insertion loss. 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, within the safe drive capability of RF board's U7's outputs.

Improvement in receiver low-frequency response and opposite sideband suppression:

Note: This modification is unnecessary if you use the (*highly recommended!*) “Frequency Translate” function available in recent firmware.

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 and improved stability for the PA bias regulator:

In certain cases it may be observed that there is a “squeal” on transmitted audio, particularly in the

TUNE mode when listening to the signal on another receiver. The problem turns out to be related to insufficient output capacitance on the LM2931 to maintain unconditional stability.

The data sheet for the LM2931 recommends a minimum load capacitance of at least 100uF to maintain stability and failure to observe this specification can cause this “squealing” and an unstable transmitter.

It was noted, both from testing and on the LM2931 data sheet that a 22uF tantalum unit such as that used elsewhere on the mcHF is seemingly enough to make it stable: Always be careful of polarity, noting which side of C96 is ground using an ohmmeter. **Comment:** The side of C96 closest to RF output transistors Q5/Q6 is ground.

If an electrolytic is all that is available, the **MINIMUM** recommended value is 100 uF and for this one may use any of the surface-mount 100uF capacitors (or 220uF units) used elsewhere in the mcHF, connecting it with short, flying leads, or one may use a standard through-hole capacitor. The voltage rating is unimportant as long as the capacitor that is chosen has a rating of 6.3 volts or higher.

When using a capacitor with leads, it is absolutely imperative that the leads be cut as short as possible to maximize the capacitor's effectiveness: Even a few centimeters will negate the capacitor's usefulness! Also, be sure to use a good-quality low ESR capacitor if one is using an electrolytic.

It was also 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 470 ohms to 10k: I used 2.2k since I had plenty of them left over from construction, but I'd recommend 470 ohms or 1k to more-quickly discharge the 100uF capacitor installed above.

* * * * *

Modification to improve performance and usability of the forward and reverse power detectors:

This modification is appropriate for firmware versions 0.0.219.3 and later.

It has been observed that with the original component values of R59/R60 and R58/R62 in the power detector, a static voltage of approximately 200 millivolts was present on the “PWR_FWD” and “PWR_RET” lines. Inspection with an oscilloscope revealed the presence of a very narrow pulse emanating from the MCU's A/D input lines of significant voltage, charging the capacitors (C82/C83) and this “bias” voltage was the result. Because the full-scale A/D voltage is just 3.3 volts, this 210 millivolt “bias” (approximately) represented a significant portion of the A/D range (approx. 6-7%).

Whatever the reason for this spurious signal, the easiest solution was to lower the source impedance of the voltage from the power detectors which was done by changing the values of the aforementioned resistors. With the change of the resistors, the capacitors were changed as well to minimize the possibility of “aliasing” occurring due to the fact that the forward and reverse power readings are not taken simultaneously, but via sequential readings: Slowing the rate-of-change minimizes this potential source of error.

Another potential problem that was observed was that the voltage being output by the power sensor exceeded 3.3 volts, a voltage above the A/D input range, at power levels below 10 watts – a power level attained on some bands by some users.

The modifications described below resolve these problems and are simple to perform and are as follows:

- Remove R59 and R60 and replace with zero-ohm resistors or jumpers.
- Change R58 and R62 to 2.2k resistors.
- Change C82 and C83 to 2.2uF capacitors.
- Replace R55 and R56 with 47uH inductors. If you do not have these, 4.7 uH inductors will work adequately.
- Additionally, it is recommended that one capacitor each of 1nF to 100nF be connected between pins 9 and 10 of the 30 pin edge connector on the RF board and ground (pin 6).

After the modification:

There are some steps required to calibrate for various offsets inevitably present in the circuitry after the modification that should be performed:

- Connect a dummy load to the transmitter
- Make certain that the MICROPHONE input is selected and turn the microphone gain setting to minimum.
- Set to USB or LSB.
- Set the band to 10 meters.
- Set the power to 0.5 watts.
- Go to the **CALIBRATION** menu and set the item “**Disp. Pwr (mw)**” to **ON**. This will enable the display of the detected power in milliwatts in a portion of the screen just below the “**RIT**” indicator with first number indicating forward power and the second indicating reverse power.
- With NO audio present (e.g. a quiet room or keying the transceiver with no microphone present) observe numbers in the upper-left corner of the screen, just below the “**RIT**” indicator, and adjust the calibration menu setting “**Pwr. Det. Null**” to zero these numbers.

- Once the **NULL** value has been determined, un-key the transmitter and then press-and-hold the **F1** button to save the setting to memory.
- It is possible that these two numbers will not exactly coincide at zero, but they should be within less than 5 units of each other. These numbers will also vary slightly in a random fashion: It is better that these numbers occasionally go slightly above zero than always stay below zero.
- If the forward and reverse power are more different than 5 or so points, check the following:
 - Make sure that there is no audio present – not even noise! There must be **NO** transmit power in this step. If necessary, set “**10m 5 W PWR Adjust**” setting for the current band to the lowest possible number, noting the original setting: You will have to momentarily set the power to 5 watts to allow this adjustment – but remember to set it back to 0.5 watts. Once you are done, restore the original “**10m 5 W PWR Adjust**” value and then press-and-hold **F1** when done to save the setting.
 - If the forward/reverse readings are still different, make certain that you have installed the proper-value components during the modification.
 - If resistors of lower precision than 1% were used (e.g. 5%) it is possible that normal tolerance variations are responsible for these differences, in which case it may be necessary to use a digital ohmmeter to match several pairs of resistors to within a fraction of a percent.
 - Check to see if your solder is based on “Water soluble” flux: If it is, be absolutely certain that this flux is removed as it can become conductive upon absorbing moisture from the air. This type of flux is more common with “Lead Free” solders.
 - If you still get radically different forward/reverse readings, check the voltages on pins 9 and 10 of the 30 pin connector: They should be equal, within a few millivolts. If not, check wiring.
- After functioning of the VSWR meter is verified and the **OTHER** VSWR/Power meter (e.g. T2/T3) related modifications discussed in this document have been performed, the wattmeter may be calibrated as follows.
 - ***DO NOT PERFORM THESE STEPS UNLESS YOU HAVE PROPERLY HEAT-SINKED THE FINAL TRANSISTORS!***
 - Connect a known-good 50 ohm dummy load and wattmeter to the output of the mcHF.
 - Set the transmit power to 5 watts.
 - Set the mode to USB, LSB or CW.
 - Go to the “**Pwr. Det. Null**” item and set it to **ON**. This will cause the forward and reverse power to be displayed just below the **RIT** indicator, in **milliwatts**.
 - Go to the “**80m Coupling Adj.**” item in the configuration menu.
 - Press the TUNE to activate the transmitter.
 - Quickly adjust the “**80m Coupling Adj.**” parameter so that the displayed forward power in milliwatts matches your wattmeter's power reading. (Remember: 5000 milliwatts = 5 watts.)
 - Repeat the above steps for 40 meters using the “**40m Coupling Adj.**” parameter.
 - Repeat the above steps for 20 meters using the the “**20m Coupling Adj.**” parameter.
 - Repeat the above steps using the “**15m Coupling Adj.**”, using a frequency on 12 meters.
 - Once you are done setting these values, press-and-hold the F1 button to save the settings to EEPROM.

Important comments:

- *If firmware version 0.0.219.3 or newer is used **without** the above modifications the power*

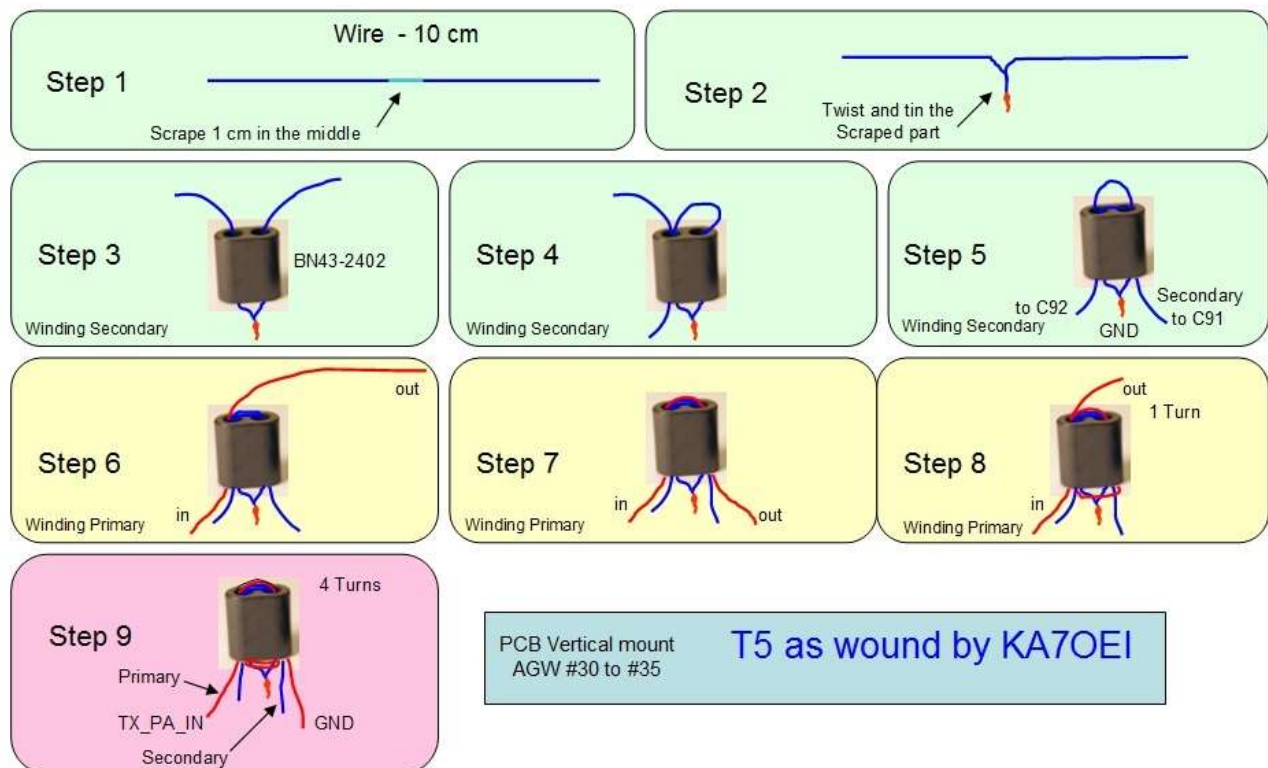
metering will incorrectly read power, even when there is no RF output power present, and it will read very high RF power during transmit. It will not correctly calculate VSWR.

- If the above modifications are done and a version of firmware **older** than 0.0.219.3 is used, the RF power metering will read very low. There was no VSWR metering on these earlier versions.
- As of firmware version 0.0.219.19 the start-up screen will give a warning if the above modification has not yet been done.

“T5” replacement:

The original part number for transformer T5 on the RF board called for a 1:4(CT) transformer, but a circuit analysis will indicate that this is inappropriate for the impedance of Q4/Q5, the RF Driver transistors. The current BOM calls for a 1:1(CT) transformer – a “less bad” match, but still not ideal in that there is a mismatch causing ripple/loss in the bandpass filters but it is less severe than with the original 1:4 impedance.

A better transformer would be a 4:1(CT) transformer. Unfortunately, one cannot simply turn the original transformer “backwards” since it lacks a centertap on the necessary side in order to properly drive Q4/Q5. Fortunately, it is easy to wind a custom transformer using a BN43-2402 core – the very same core used for RFC8 – and some small-diameter wire.



Step-by-step winding guide for constructing "T5"

Drawing kindly provided by Fabio, IZ0IBA

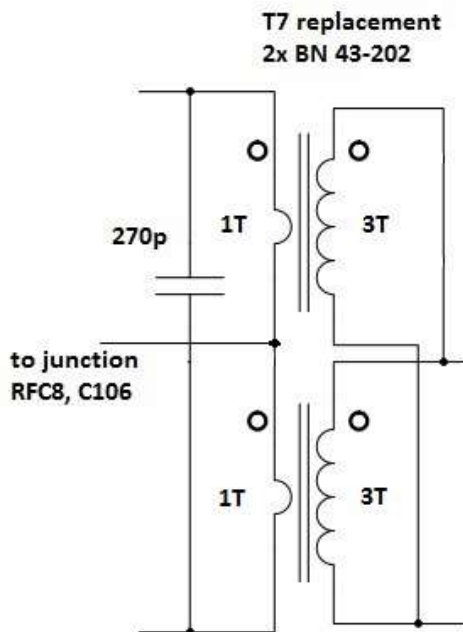
This reworked transformer is small enough to be soldered to the original pads occupied by T5: Just make the leads as short as practical.

Re-worked T7, the output transformer:

As it turns out there has been quite a bit of discussion related to the ability (or inability) experienced by some users of the mcHF to output a “solid” 5 watts on higher bands. Careful analysis by some of the folks on the Yahoo group has shown that the original transformer design suffers from a few design deficits, namely:

- Primary/Secondary coupling not as tight as it should be
- Excessive leakage inductance
- Transformation ratio not optimal for maximum power output at good efficiency
- Reactance associated with the transformer and other final circuitry is not compensated

As it turns out there are several possible solutions to this problem, all of them using the BN43-202 ferrite core and the solution depicted – one of several proposed – uses *two* of these cores.



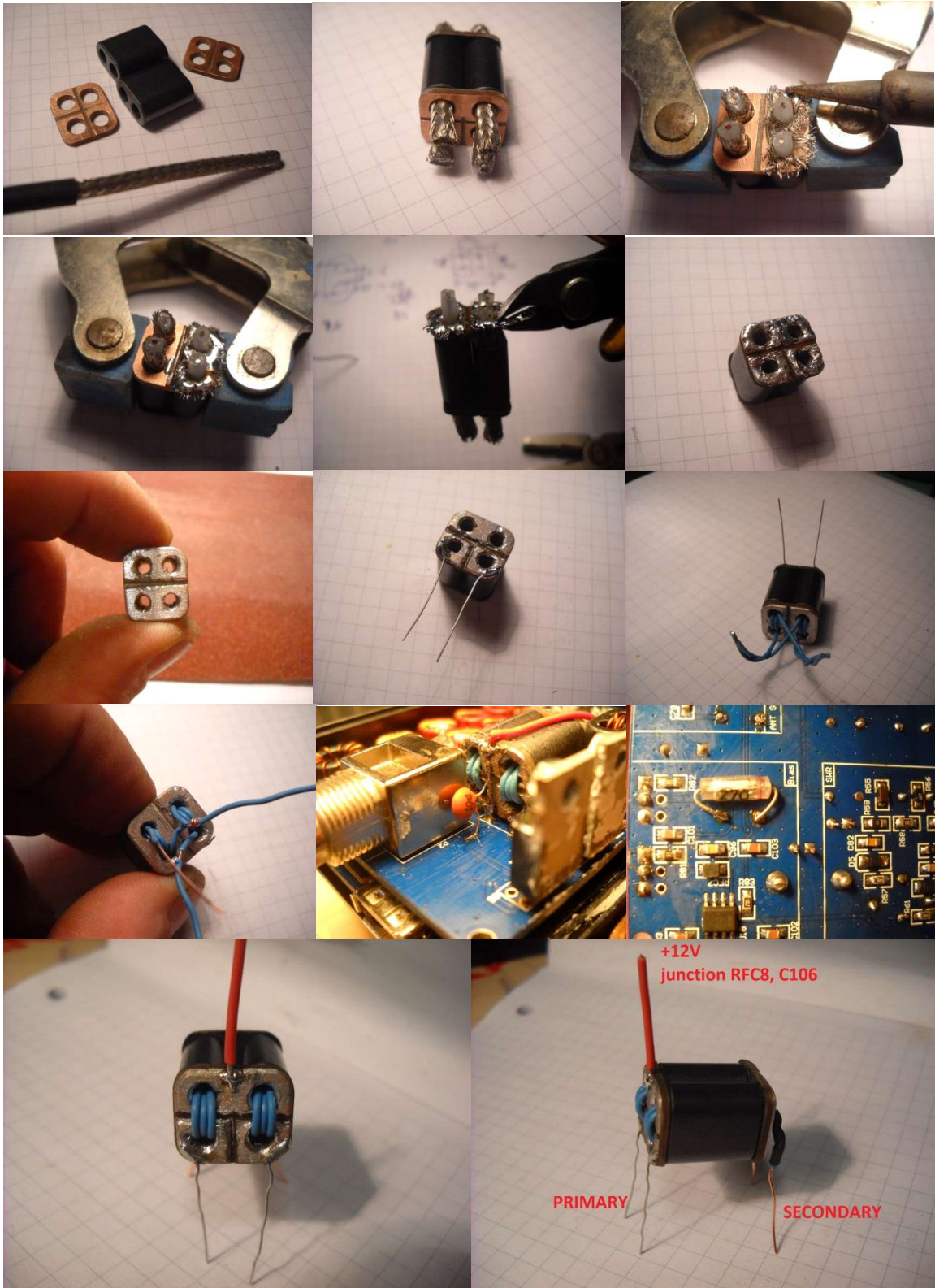
Wiring diagram of T7

Drawing provided by Marko, S54AM

It is recommended that the secondary be wound with “Teflon” (PTFE) covered wire if it can be obtained since it is capable of handling high temperatures and its low coefficient of friction makes it easier to wind the transformer.

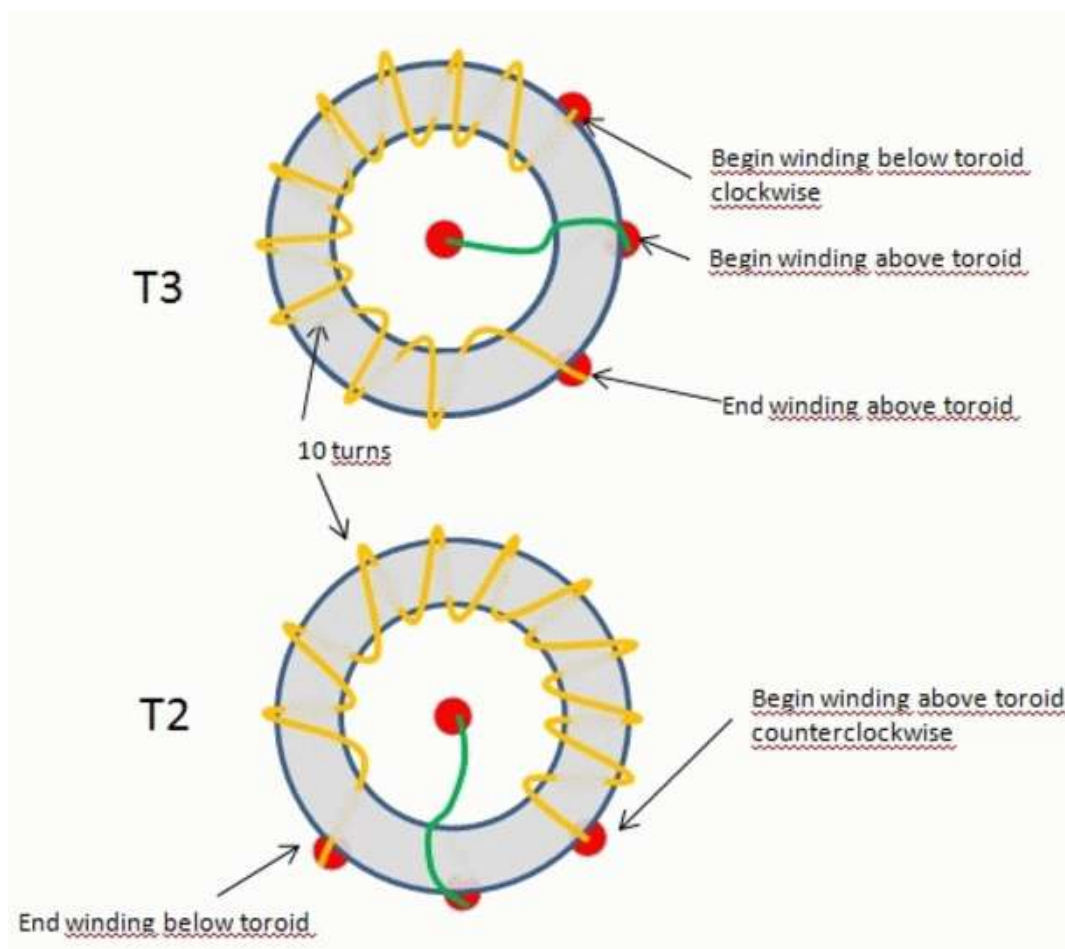
If one looks closely at the above pictures it can also be seen that on the “center-tapped” side of the primary (the left-hand photograph) a piece of copper-clad printed circuit board is used to connect the primary windings (the coax braid) inside each core together. While this makes for a very neat construction, it is not absolutely necessary, but if flat, copper material – such as PCB material or copper sheeting/flashing – is not used it is recommended that multiple strands of bare, copper wire be used to bond them together.

Finally, although not shown on the schematic – but visible in one of the photographs – it is recommended that a 0.1uF capacitor be connected between ground and where the V+ (red wire in the photograph) is connected on the two cores for optimal RF bypassing.



*Details of the constructing, winding and mounting a version of T7, the PA output transformer.
Thanks to Marko, S54AM for these pictures.*

Details of T2/T3 winding:



Winding "sense" details of T2 and T3

Drawing by Fabio, IZ0IBA

For T2/T3 note that the "sense" of the windings (*e.g. which one goes on top, the direction of winding etc.*) is **absolutely important** in order for the forward/reverse RF power metering to work!

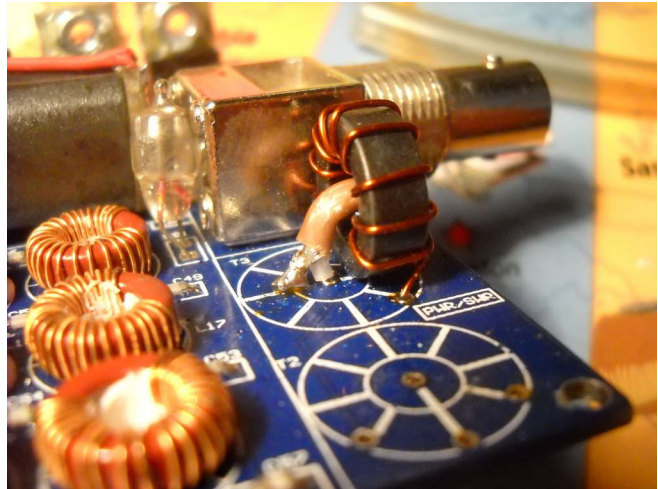
It has been determined that the one-turn "primary" winding should be **Faraday shielded** from the rest of the transformer in order for there to be adequate reverse isolation of the reflectometer to obtain useful readings on the higher bands.

Fortunately, this is quite easily done. Ideally one would use RG-178 coax which is Teflon™ (PTFE)-based, but this coax can be difficult to find, but if one is very careful in soldering the polyethylene-based RG-174 coaxial cable may be used, instead.

The first thing to do is to prepare two, short pieces of coaxial cable as depicted in the left-hand picture, below. Note that on each, the shield is cut off on only one end: The Faraday Shield works by blocking the "E" field radiated from the conductor, but not the "H" (magnetic) field – and it is the "E" field that can cause problems with anomalous, high VSWR readings on the higher bands, even if one is using a known-good dummy load!



Preparation of the two small pieces of coaxial cable for the Faraday-shielded "primary" windings.

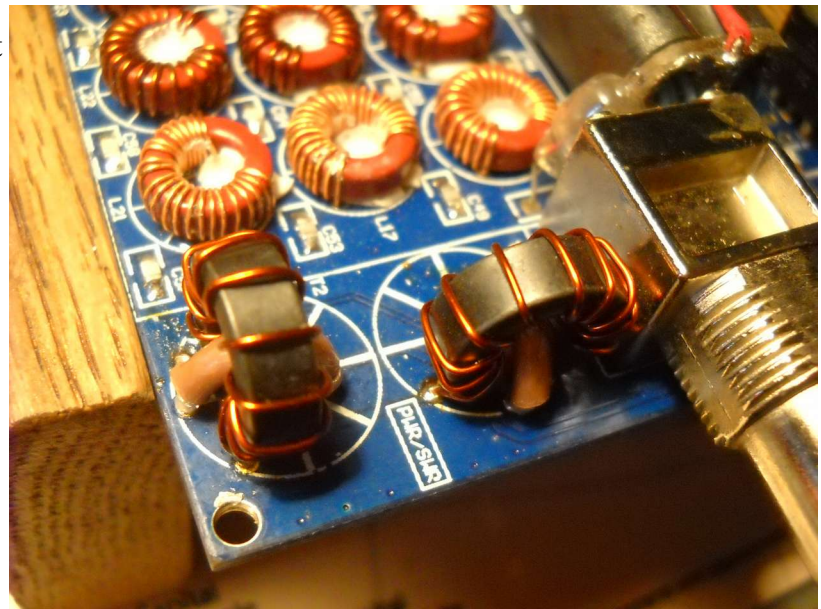


Connection and mounting of the transformer with the shielded primary.

*Details of winding of T2 and T3 with Faraday shielding using coaxial cable.
Pictures provided Marko, S54AM*

Again, it is important to note that the short piece of coaxial cable must have it shielded grounded at only **one** side: If both ends of the shield are grounded the "H" field will be suppressed and the transformer will not work at all!

If you use a polyethylene-based coaxial cable such as RG-174 care must be taken to avoid melting the dielectric when soldering the coax into place. A pair of needlenose pliers or tweezers may be used as a heat sink to minimize the amount of heat that is transferred to the plastic while soldering to prevent melting.



T2 and T3 installed with Faraday shield coaxial primary winding.

If you are retrofitting an existing T2/T3 installation you will want to clean the holes in the circuit board to be free of solder before installing T2/T3.

Photo by Marko, S54AM

For soldering the shield on one end of the coax, simply scrape the off a bit of solder mask in an area that is **known to be ground plane** and tin it before installing.

Reduction of standing current by the PA driver transistors:

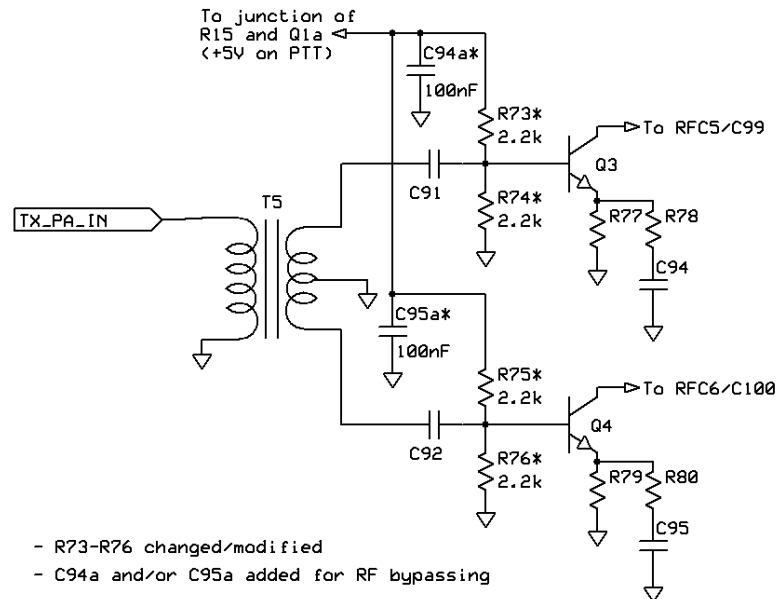
It may be noticed by those studying the diagram – or having happened to make current measurements of the transceiver – that PA driver transistors Q3 and Q4 are *always* biased on. Because of this, an extra 30-60 mA is consumed at *all* times when the power is applied to the transceiver: During receive, and even when it is turned off!

In studying the diagram you will note that bias resistors R73 and R75 are connected, via RFC5/RFC6, to the VCC_12 volt supply – which is always on. R73/R75 also provide a degree of RF degenerative feedback from the collector to the base of the transistors and reduce their impedance as well, possibly causing an increased mismatch to the output of T5, further reducing RF output, particularly at higher frequencies.

It is possible to derive the bias from a regulated source that is switched on when the transceiver is in transmit mode and one such supply of bias is the source of Q1a which provides a *regulated* +5 volts (*preferred over an unregulated bias voltage*) when the radio is keyed. Because the bias supply is now 5 volts rather than an unregulated 12 the values of R74/R76 must be changed as well to attain the desired currents.

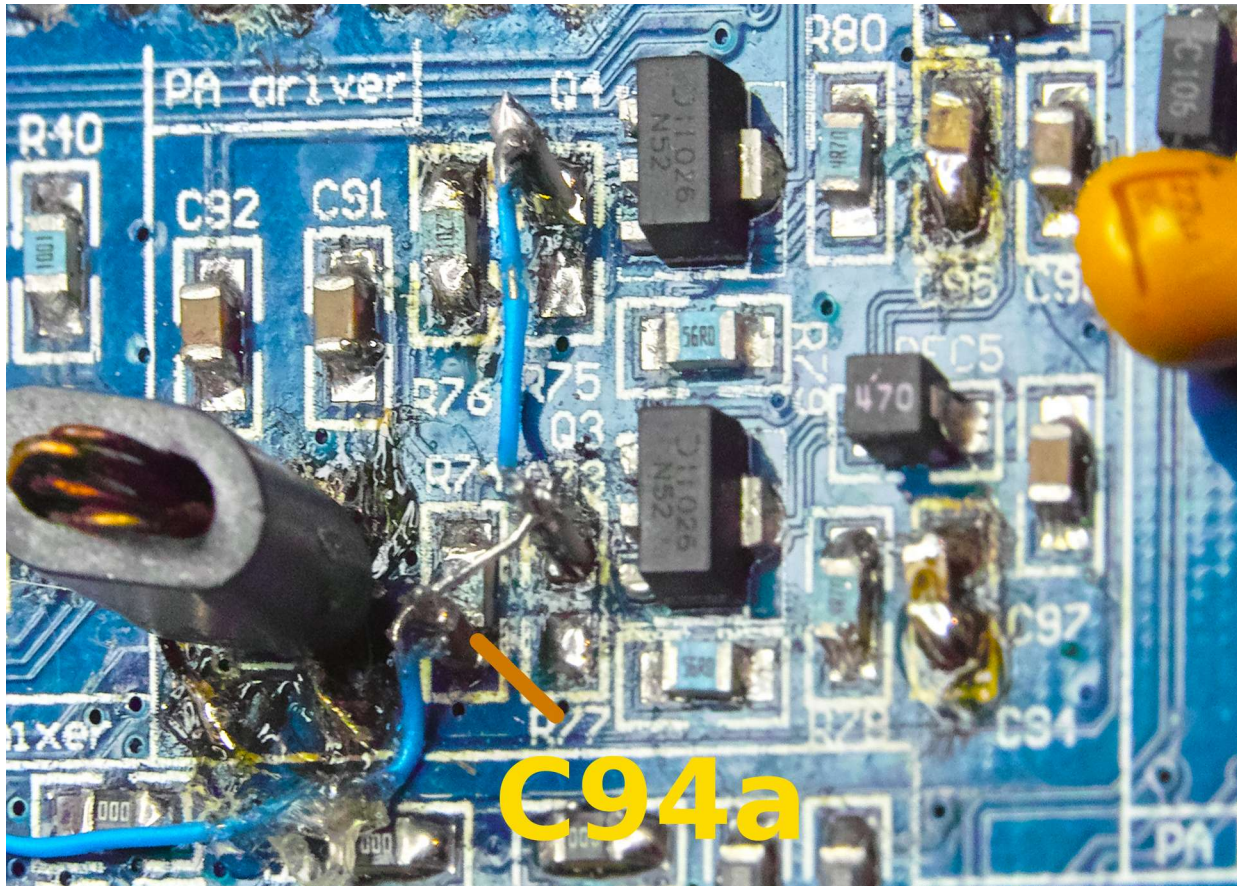
The modifications are as follows:

- Change R74, R76 to 2.2k
- Remove R73, R75
- Where R73, R75 would be, stand on end, in each location, a 2.2k resistor, the bottom end connecting to the base of transistors Q3, Q4.
- Solder, standing on end, 100nF capacitor(s) to the ground end of R74 and/or R76.
- Connect the "top" of these components (the two resistors and the capacitor(s)) together.
- Connect the tops of these components to the junction of R15 and Q1a (at R15) using a piece of very thin, insulated wire: I used #30 AWG wire-wrap wire, tacked down in several places with small dots of thermoset ("hot-melt") glue.



mcHF PA Driver Bias Switching Modification

If the above modifications are done, the "Off" current of the mcHF is reduced to 3-4 mA – the "off" current of U3 and the current through R9/R10 and R13/R14 and the low-volume receive current with the LCD at full brightness is reduced to approximately 360 mA.



An example of the modifications to add PA bias switching.

Note that R74 and R75 are standing on-end and that C94a is also on-end, soldered to the ground side of R74. #30 AWG "wire wrap" wire connects the circuits together and to the junction of Q1a and R15, the source of the switch +5V supply and that this wire is tacked down with dabs of thermoset glue.

In this photograph on may also see installed the "alternate T5" in the lower-left.

IMPORTANT NOTE REGARDING THE PA DRIVER (Q3/Q4) AND CAPACITORS C94, C95:

A **PREVIOUS** recommendation had been to replace C94 and C95 with 4.7nF capacitors to reduce the gain at lower frequencies (<10 MHz).

Unfortunately, reports and subsequent testing indicated that this can cause instability to some users so this recommendation is withdrawn: Continue using 47nF capacitors at these locations!

More recently it has been reported that with the "alternate" T5 design (*described above*) and/or the addition of the PA bias switch and its elimination of degenerative feedback via R73/R75 that the above modification is, again stable – but this has not been positively verified by a wide variety of users.

If you wish to make this modification **and** you have the ability to test the stability of the transceiver's transmit signal chain under a wide variety of conditions (different driver levels, supply voltages, frequency bands, load mismatches – both resistive and reactive, etc.) - particularly with a spectrum analyzer, please let us know what you find!

Modifications specific to UI board versions 0.3 and earlier

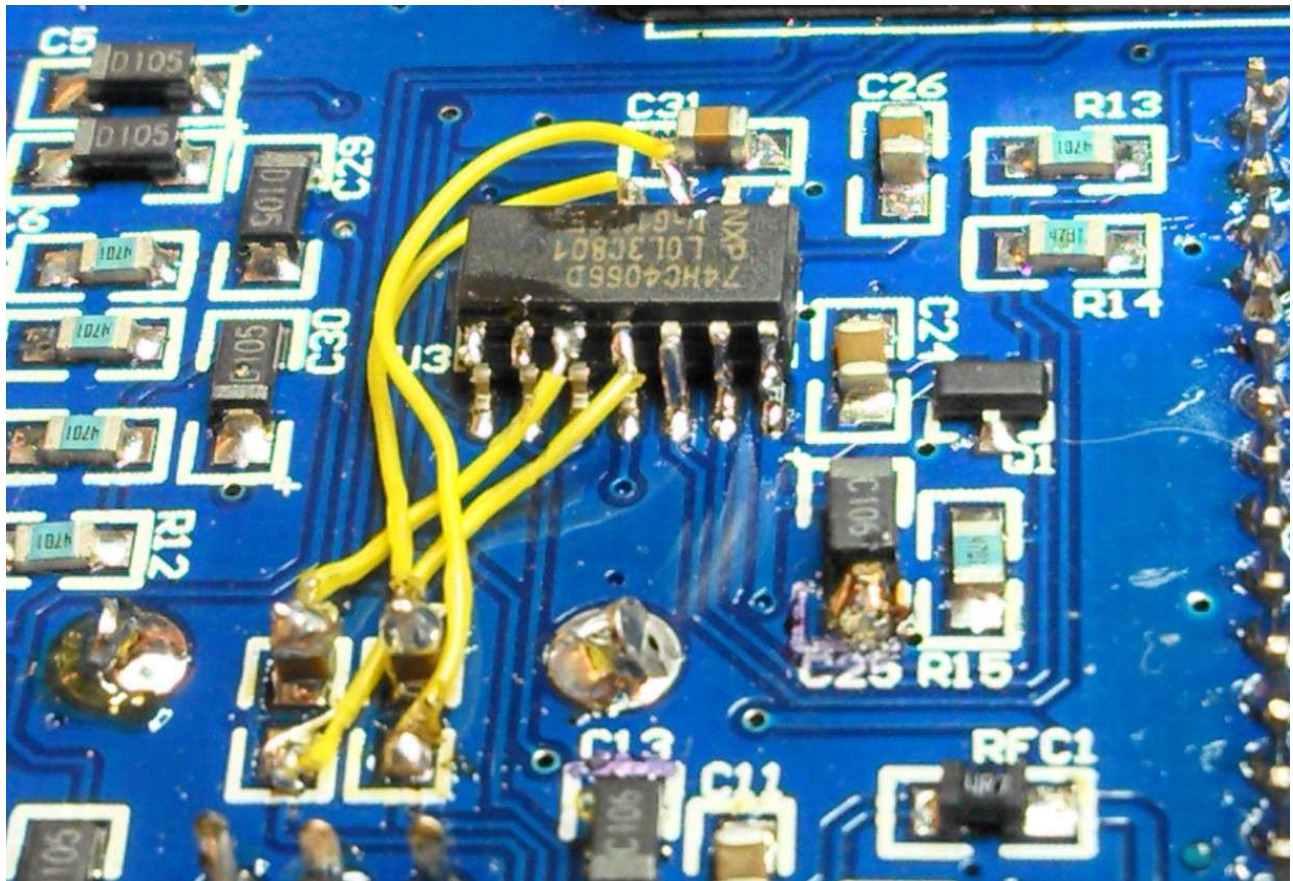
These modifications have been incorporated in UI board 0.4

If you have a version 0.4 UI board you may wish read these to understand the “how and why” related to these modifications and their understanding may prove useful for troubleshooting purposes.

Prevention of RX audio “breakthrough” on the TX mixer (Version 0.3 and older UI boards only):

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 using UI board version ≤ 0.3 should perform.**



The "U3a" modification, piggybacking 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.

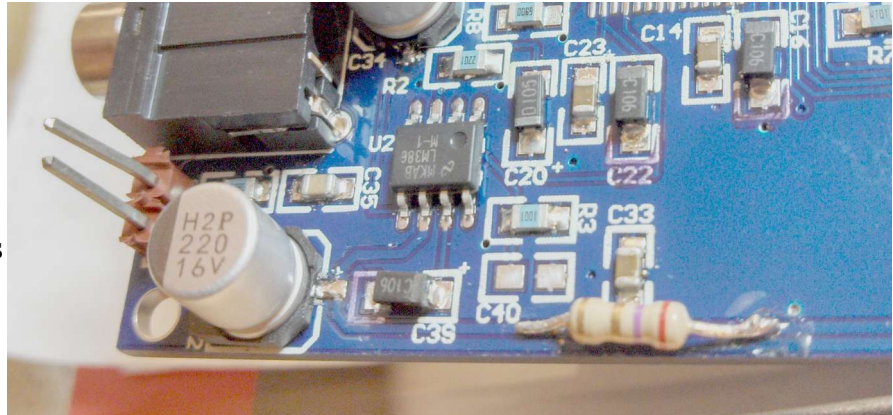
Reduction of conducted audio on the 8 volt line to prevent audio feedback (Version 0.3 and older UI boards only):

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.

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.



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

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.

Additional comments about this modification – how it is done in the newer board:

The newer version of the UI board (version 0.4) has *additional* filtering beyond that mentioned above in that it uses *two* resistors and *two* capacitors for filtering. In addition to the existing capacitor at C32 and the added resistor mentioned above, there is yet another capacitor to ground and series resistor: Both of these series resistors are 2.2-2.7 ohms and the capacitors are at least 220uF in capacitance.

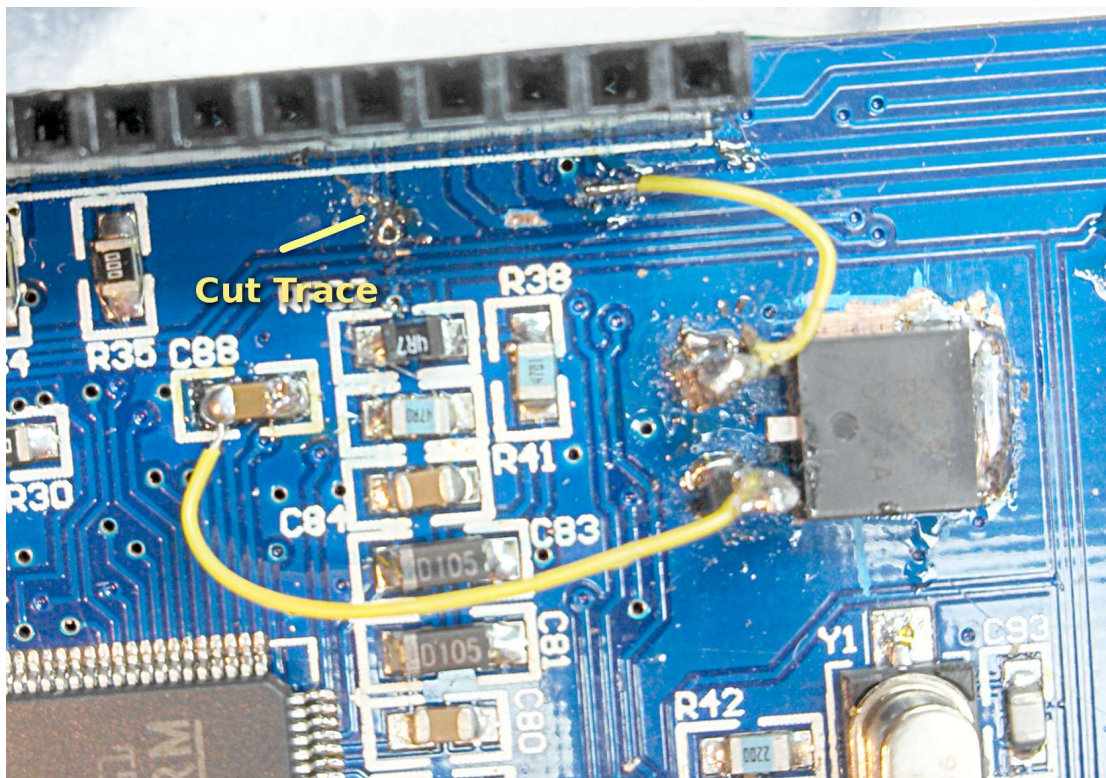
These two capacitors and resistor together provide somewhat better filtering than a single capacitor/resistor.

A separate 3.3 volt regulator for the MCU and additional LCD +5V filtering (Version 0.3 and older boards UI only):

It has been determined that the *most sensitive power supply is the main +5 volt supply*. This appears to be due to the fact that receive QSD mixer (*U15 and related components*) have minimal capability for power supply rejection. Because of this, even millivolt-level modulation of the +5 volt power supply with audio frequency signals will yield artifacts that are readily audible at low/no signal levels.

IMPORTANT: It is strongly recommended that known-good quality, **LOW ESR** electrolytics be used for the following modifications. If “generic” electrolytics or those of unknown quality are used it is possible that they will be unable to effectively suppress the power supply noises. Low ESR electrolytics will be rated for 105C – but not all capacitors rated for 105C will be low ESR.

It is *not* even necessary that this source of audio modulation be due to a device that operates directly from the +5 volt supply: Modulation of *any* of the power supplies can cause this, most likely due to ground loops causing slight voltage changes throughout the system – including that on the +5 volt supply – an example of this being the aforementioned needed isolation of the +8 volt supply line feeding the audio amplifier.



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.

This picture was taken before the addition of the 4.7 ohm resistors and additional capacitors described below and is included for clarity.

This leads to 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.

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:

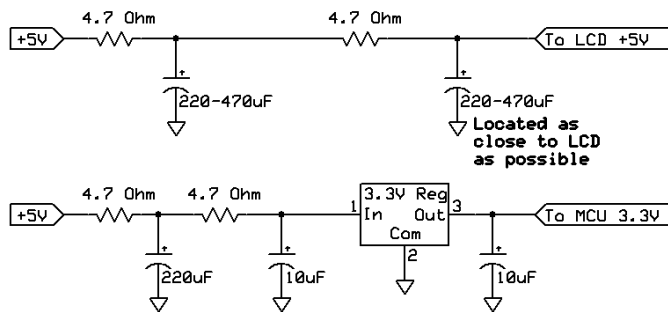
- The +5 volt supply for the LCD
- The transmission of noise *through* the 3.3 volt regulator.

When the “LCD Backlight Brightness” control function was added, the need for even better filtering of the LCD's power supply was made apparent as the modulation of the LCD's power supply current rippled (literally!) throughout the rest of the receiver!

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.)

The diagram 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 and load.

- The LCD has an internal 3.3 volt regulator that has a (worst-case) dropout voltage of 600mV – and less than 1 volt is dropped across the two 4.7 ohm resistors.
- 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.



Note: Additional isolation will be obtained if the LCD, MCU and USB +5 volt supplies were derived from a separate 5 volt regulator on the UI board, powered from the +8 volt rail as depicted below.

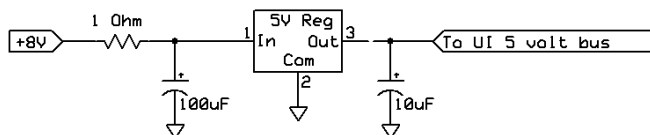


Diagram depicting the MCU's own, isolated 3.3 volt power supply. This also shows additional isolation of the LCD power supply as well as the suggested, separate 5 volt power supply for the UI board - see text.

For LCD filtering, place the noted capacitor as close to the LCD's pins as possible – preferably *at* the LCD itself! The 4.7 ohm resistors and the other capacitor may be placed farther away, but the second capacitor should be placed as close to the LCD's ground as well.

On the 3.3 volt MCU regulator, 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.

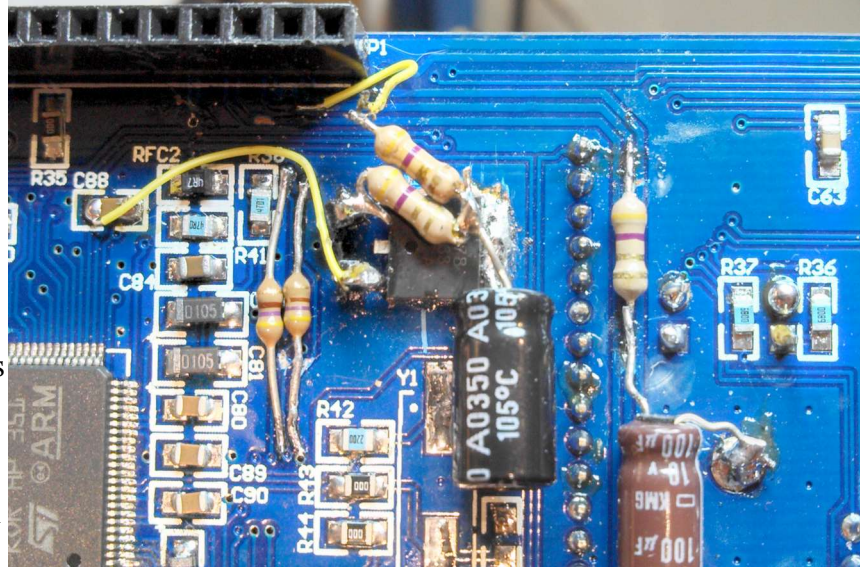
Finally, there is the depiction of a separate 5 volt regulator for the “UI 5 volt bus” that is *not included on the Version 0.4 UI board*. This regulator, powered by from the 8-volt bus, would feed the additional regulator. The rationale for including this regulator would be in the event that a device such as a memory stick, keyboard or network interface adaptor plugged into and powered by the USB interface (*the larger USB connector, J10*) might modulate the power supply, causing audible disruption to the receiver operation. In the aforementioned diagram it is possible that the suggested values of filtering components (1 ohm, 100 uF) may need to be increased to quash noise if it is found to be severe.

Isolation of the USB power supply on the UI board:

In the future, it is expected that USB devices may be plugged into J10 and/or J11, powered from the mcHF. Because these, too, use the +5 volt supply that is known to be very sensitive to noise, it would be prudent to isolate this supply as well as noted in the diagram, above.

Discussion:

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 increasing the capacitance of C32 to filtering and the additional filtering of the 5 volt LCD line (right). 1000uF can reduce this effect, as The resistor to filter the "tick" (see next section) are also visible. can the use of headphones or an external audio amplifier.

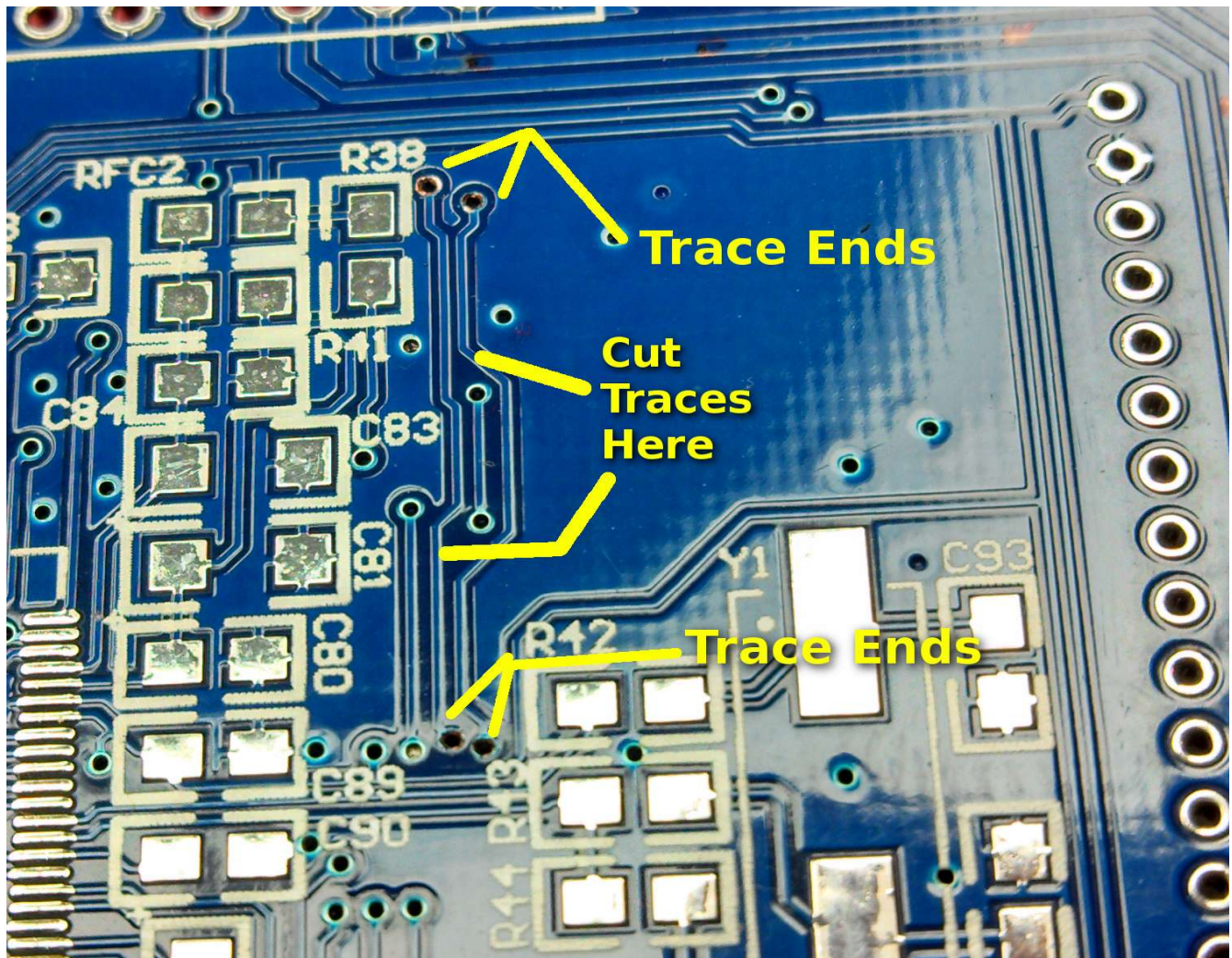
Further modifications to reduce this source of feedback are being investigated – but it's a good thing that the receiver's being improved to where this is a problem!

Elimination/reduction of the 1-second “tick” noise in the receiver – particularly on the higher bands

(Version 0.3 and older UI boards only):

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

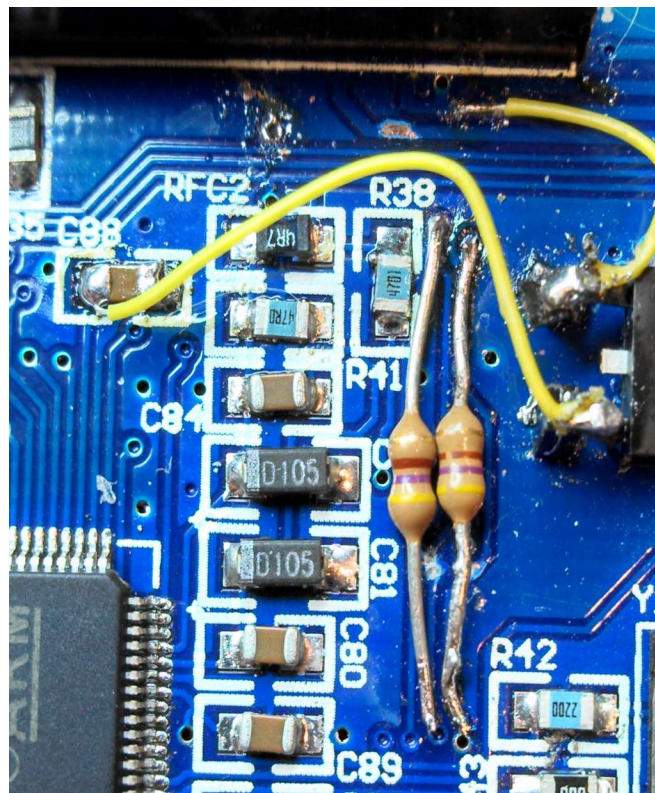
resistors, but surface mount resistors can be used, instead if that is your preference. (The 3.3 volt regulator used for the MCU – see the modification earlier in this document – can be partially seen.)

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!

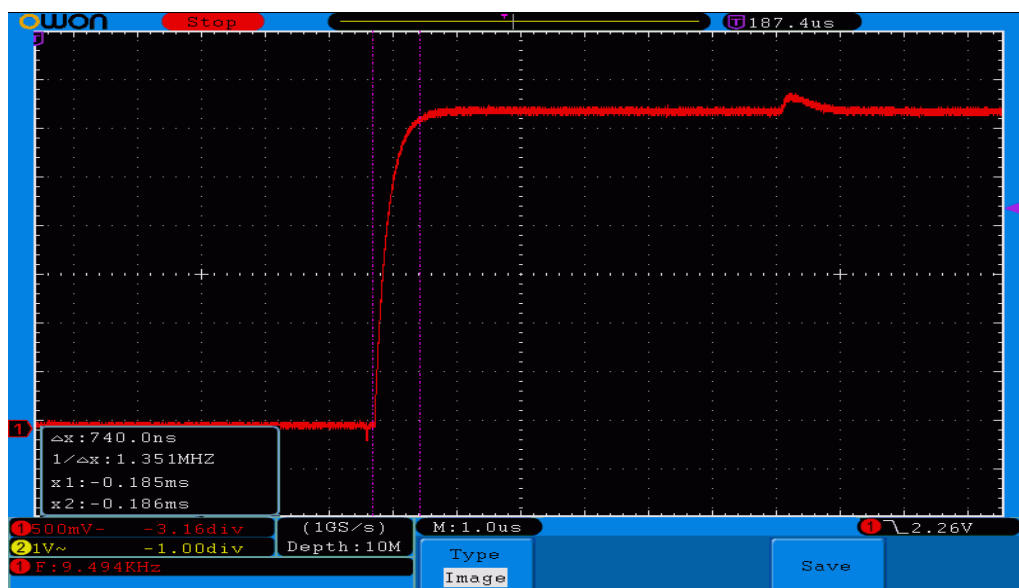


The added 470 ohm resistors in series with the SCL and SDA lines. Note that this picture also shows the added 3.3 volt MCU regulator mentioned above. 1/8 watt, leaded resistors are used, but surface mount resistors could be used, if

IMPORTANT:

Newer versions of firmware include the “Frequency Translate” function which should be turned **ON** and set to “RX LO LOW”.

(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!