mcHF Hardware modifications as of October 28, 2014
as compiled by KA7OEI for Revision 0.3 boards

- 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.

- Q1's footprint is backwards on board rev. 0.3 with a BFR93 or BFR93A, requiring that its leads be bent backwards over the body of the device before soldering. If a "BFR93R" is used, the footprint is 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 the Rev. 0.3 UI board. Again, not a "mod" - just something to know!

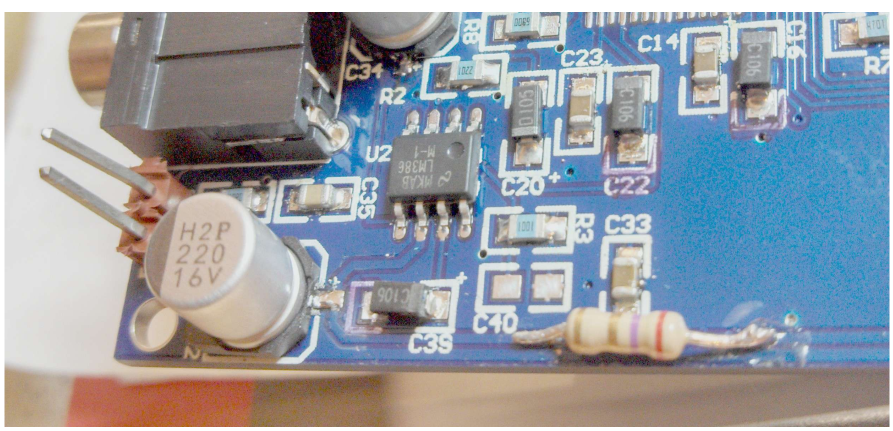
More recent mods:

Reduction audio on the 8 volt line:

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.

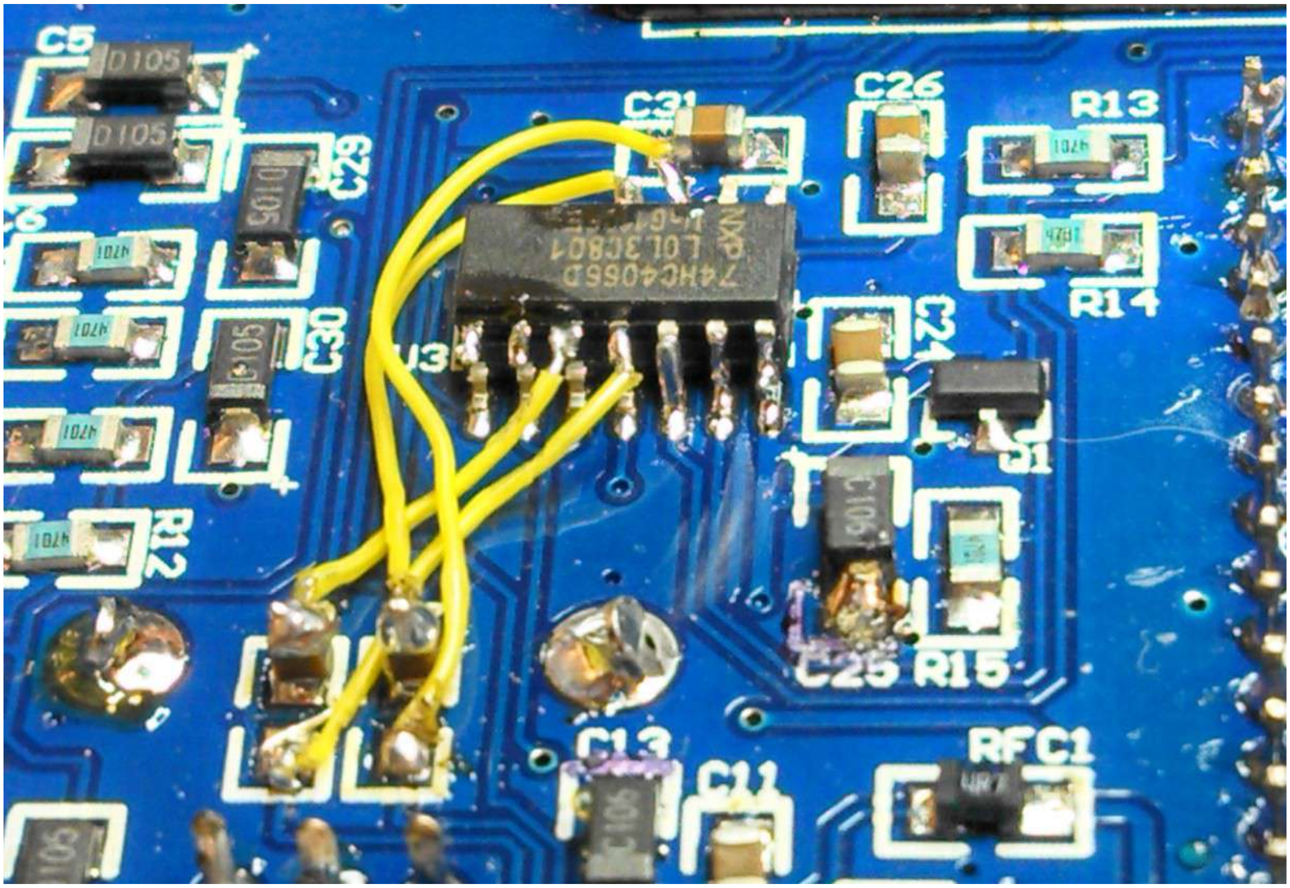


The 4.7 ohm resistor added to the UI_8V line feeding U2, the audio amplifier.

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.

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

The installation of "U3a", a 74HC4066 switch, piggybacked atop U3 - a pretty easy mod, once 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. The chip for "U3a" is the same type as U3.



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 can turn C7 and C8 around, the "open" end facing U3 rather than mounting 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.

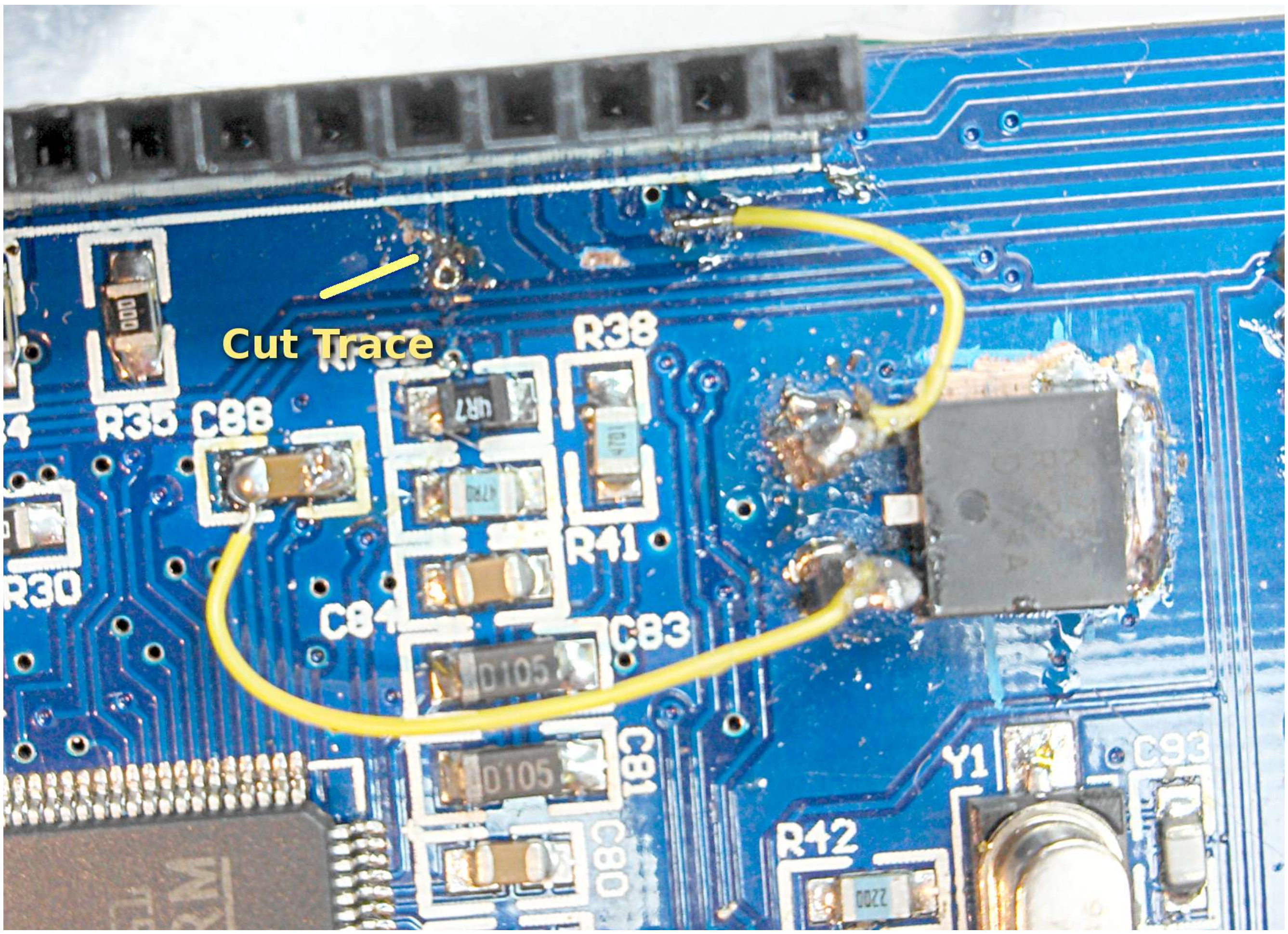
Note that the other two sections of "U3a" (2 and 3) are connected to the inversion logic (Q1 on the UI board) and are active when in receive mode and the corresponding I/O pins (pins 3, 4 and 8,9) are left floating (which will cause no problems on a '4066) - but these may be useful for something else...

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:

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 getting into the receiver due to the MCU's modulation of current on the 3.3 volt line shared with low-level audio circuitry.

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.



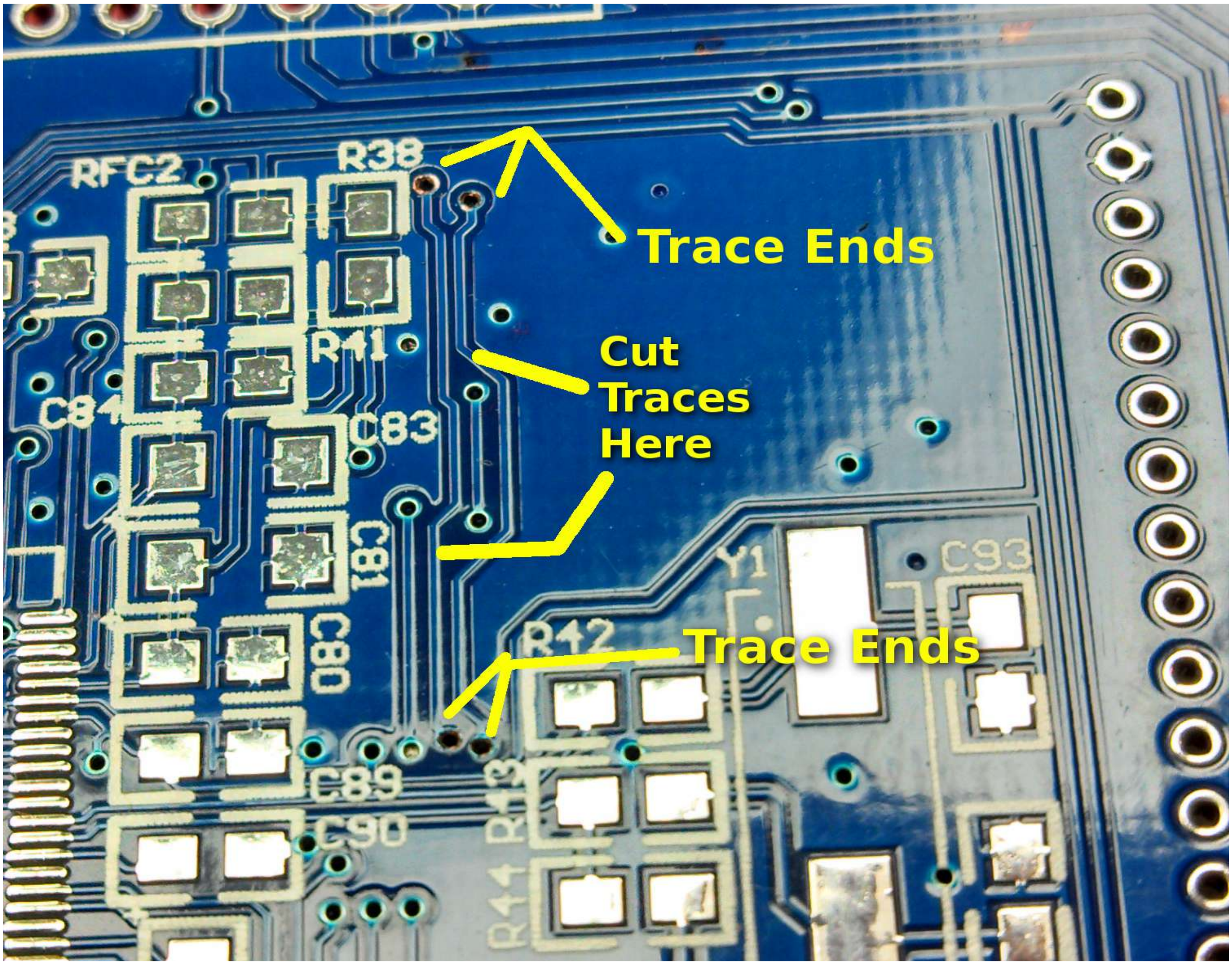
The added 3.3 volt regulator to deliver an isolated power supply to U4a, the MCU. This reduces a noise source that can get into low-level RF and audio stages. The trace is cut just ***above*** *the via.*

There are still some MCU-related noises on the 8, 5 and 3.3 volt power supply lines that need to be addressed, but this reduces the lions share of it.

Prevention 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.

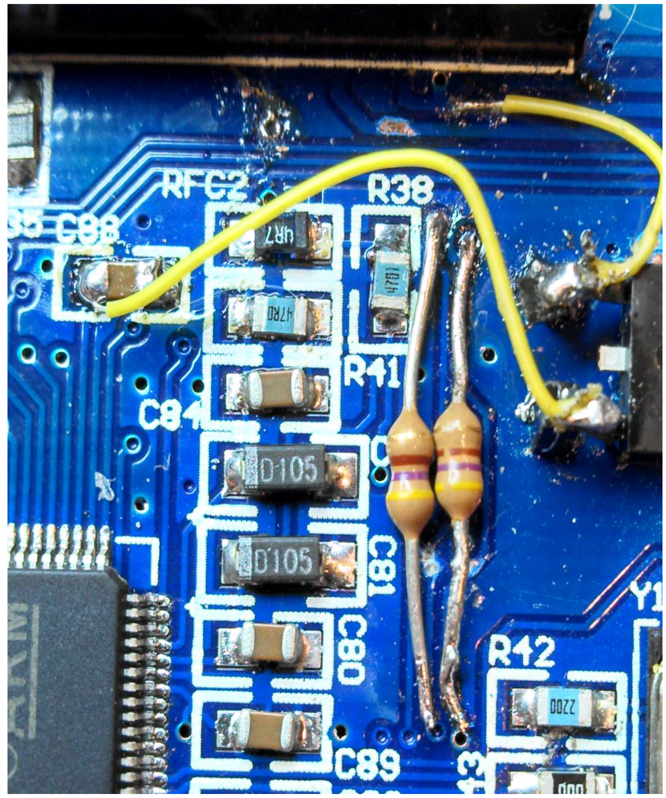
In the picture can be seen the two added 470 ohm resistors. As you can see, I used leaded 1/8 watt resistors, but surface mount resistors can be used, instead if that is your preference. (The 3.3 volt regulator used for the MCU – see the modification earlier in this document – can be partially seen.)

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.

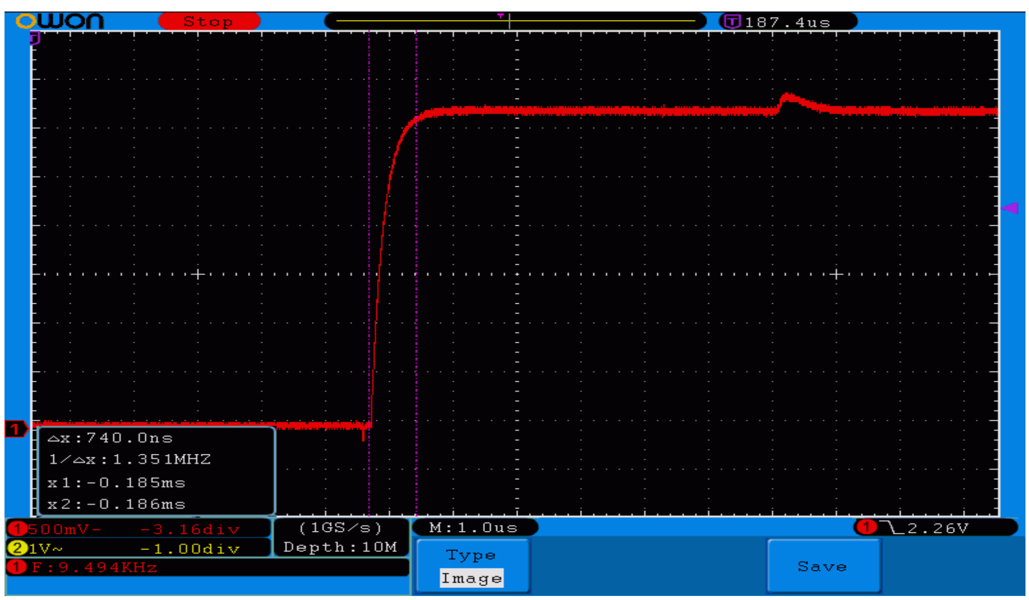
As can be seen, the risetime is less than 740 ns at the 10%-90% points which is a fraction of the I2C clocking rate, well within the time at which the edge of the clock into which the data is strobed in by the peripheral.

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 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!