All modifications listed here refer to the PCB Revision 0.4

Absolutely necessary modifications are red. These are either necessary because they allow the devices function only reliable or otherwise present on our list of materials is no longer correct due to component changes and one otherwise has parts on and what is missing, or the modifications were subsequently only possible with a lot of effort. To what extent the parts lists of M0NKA page are complete and correct, I sadly can not say because I do not ever watch this. The final list ("BOM0.4") contained in any case so many mistakes that I would prefer to use verified in our project group. Otherwise, it makes sense to build the unit, initially without the other modifications and then make gradually individual modifications. The danger itself incorporate error is so greatly reduced.

Legend:

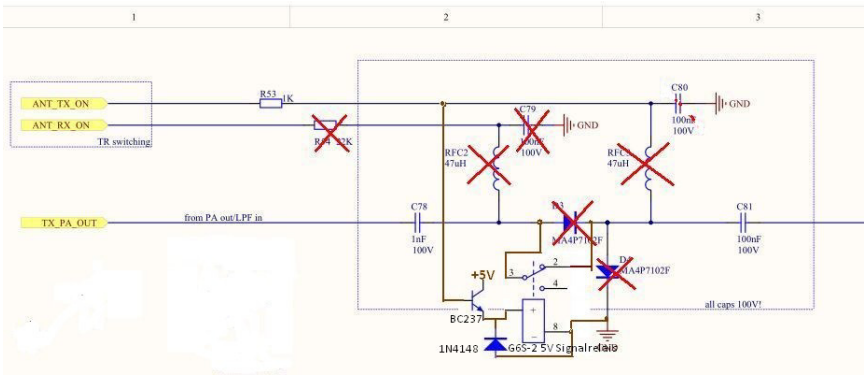
All modifications are named according to a scheme. Generally, it is useful to first modify as little as possible, because then you have an easier troubleshooting when something goes wrong. (WW-XX-Y-ZZZ)
WW: RF = RF PCB, UI = UI PCB

XX: Platinum Revision

Y: "N" = modification with normal priority, "H" = urgent anzuratende modification
ZZZ: sequential numbering of all modifications

- (RF-04-H-001) The final stage tended to "dynamic overshoot". Means: depending on drive level could in addition to the desired signal own impulses occur to 1MHz * * creepy. Remedy brings a 10uF tantalum (!!) capacitor in parallel with C106.
- (RF-04-H-002) The quiescent current through the two final-stage FET is turned off not sure if the device is in RX mode. The reason is that the output of the LM2931 a minimum load required to safely reach the "0V". A solution could be resistance 1K 2,2K ... parallel to C96. To minimize spurious noise that could spread to the transmission over the PA bias setting is changed from 2.2uF to 100uF tantalum (!!) C96.
- (UI-04-H-003) The screen is not directly soldered, but put two female connectors. The strips are not readily available because the grid size is unusual 2mm. They can be purchased at Farnell under the order 110-9733. Thus the height of the display is not substantially greater despite capping, the two black plastic strips of the two pin headers on the display panel pried gently and then the pins themselves about 3 ... 4mm cut off with scissors.
- (UI-04-H-004) R30, R31 and R32 (all 0R) do not mount on the UI circuit board, if one has the newer LCD display HY28B. These hinder otherwise the future use of the touch function of the display panel. If you have the old HY28A, one may only populate when operating the display in SPI Nodus, otherwise you disabled even there the touch function.
- (RF-04-H-005) The phasing of the I / Q-audio signals is changed by a series circuit of capacitors. To minimize, to be C71 and C73 replaced by 0R resistors.

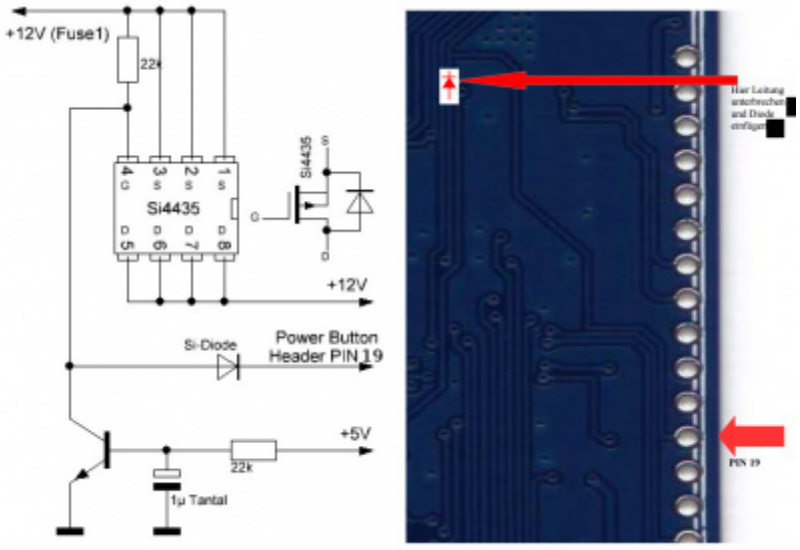
- (UI-04-H-006) The harmonics of the display PWM dimming schemes can cause interference. To minimize this, the electrolytic capacitor C74a is replaced with a tantalum capacitor of equal value.
- (RF-04-H-007) The PIN diodes for switching the antenna switch is not a low enough through, thereby the sensitivity of the receiver is reduced. As a remedy, is replaced by 220R and R54 22K by 3.9K R53 1K.
- (UI-04-H-008) Chokes RFC1-RFC3 on the ui board be changed from 4,7uH to 47uH.
- (RF-04-H-018) The transistor Q2 and the resistor R40 omitted. The entire receiver scheme is now "disappeared" in the firmware and the sound chip. An assembly of these two components would only weaken the sensitivity of the receiver.
- (UI-04-H-020) The two pins of P6 be extended wire with a few centimeters and mounted in the housing that later easily reprogram the bootloader is possible without having to disassemble half machine.
- (UI-04-H-021) is "moved" after (UI-04 -: - 025). Those who had already performed this mod, removed the wires please again.
- (RF-04-H-023) In the SWR measuring bridge is a design error. The end of the many windings of T3, which now goes to the RF side of the final stage, to be pulled out of his pad again. Next to it is the "correct" line: the MAIN PAGE of the bridge that goes to the BNC connector. You can scrape the conductor next to the toroids and solder the drawn end of the wire there again. In addition, the following modifications are necessary: R59 and R60 are changed by 1K to 0R, R58 and R62 from 100K to 2,2K, C82 and C83 of 100nF to 2.2uF. This SWR measuring bridge is reliable throughout the operating range of the MCHF.
- (AG-xx-H-024) makes it possible, with other querzutauschen complete boards, we should set us on a consistent use of the male and female bar. From "grown reasons" we propose the use of the female connector and the UI board using the plug connector for the RF board.
- (UI-04-H-027) damaging the STM32F4 to prevent static charges on the keyer lines and on the PTT line, are displayed on the UI board of the header pins 12 and 13 each have a Zener diode 3.3V down to earth (cathode to the pins).
- (RF-04-H-029) The RF switch with two PIN diodes is problematic. The pin diodes have at lower frequencies (that is starting from 30m going down) significantly diode characteristics, which leads to a distorted transmission signal and power loss. Therefore, it makes sense to remove the entire switch: RFC2, RFC3, R54, C79, D3 and D4. For a 5V relay is fitted with NC contact.



This bridged-energized the connections of former D3. The work coil is connected to one end connected to ground, the other goes to an emitter follower (NPN standard transistor, eg BC237) thereof to the emitter, the collector goes to + 5V, the basis of the common terminal of R53 and C80. During transmission, the relay switches and disconnects from the receiver clean and completely. Not the freewheeling diode (LL4148, etc.) in the work winding forget! All this fits easily in the place where once there was the diode switching. Side effects, there is no - self CW-BK is still possible. There are also photos of such a structure. The modification (RF-04-H-007) has thus become naturally fell.

- (RF-04-N-009) The two gate resistors R81 and R82 were of 220R reduced to 100R (only if the amplifier at high output current fluctuation has).
- (RF-04-N-010) At high speaker volume, the ripple of the 8V supply voltage coupled to the receiver mixer. Why does the LF amplifier attached to the stabilized voltage as well? I have it connected with a FET connected directly to the 12V input voltage. This current changes will affect at high volume no longer to receive. Additionally, I have connected with this switched 12V input of 8V voltage regulator and the two voltage divider elements directly to the 12V. As a result, the quiescent current consumption in "off" mode is now fallen to less than 1uA. The following changes are necessary:
  - On the ui board is the line "PC13" excoriated by the processor to the power switch and the header below the LCD and placed a diode in this line (anode to the processor out)
  - On the rf board Pin4 is from U3 highly placed as well as the sides of R9 and R13, which will go to 12V. These three connectors are now merged with wire and attached to SW_12V the new circuit.

- The rest will be seen from the unique labels of the new circuit is wired. On the right, the place can be seen where the additional diode is installed. The site is located on the ui board below the LCD. The cathode of the diode is then in the picture above and below the anode (for STM32F4 out).



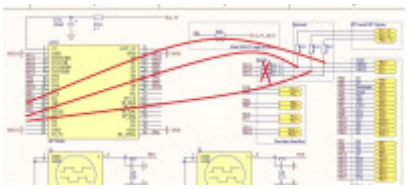
- (RF-04-N-011) The quiescent current through the transmit driver not only flows "also upon receipt" - it flows even when the device is switched off completely (!!!) because the nonsense is, the following changes are made:

The resistors R74 and R76 are equipped with 2.2K instead 470R

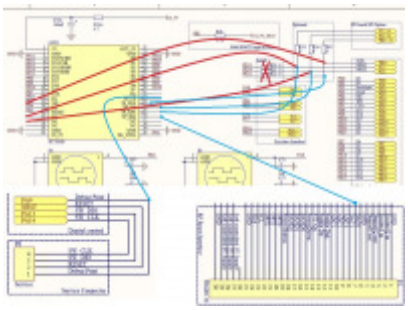
- - The resistors R73 and R75 are unsoldered
- - To the solder pads to bases out they are again soldered "perpendicular"
- - A capacitor 100nF is standing soldered from "ground end" of R79
- - The three "upstanding member ends" are now connected via a braided wire and the braided wire to Pin1 U2 soldered ("PTT_5V"). From now on, only quiescent current flowing through the driver when it is sent.
- (RF-04-N-012) Because firmware fully uncaught asymmetries in the ADC converters displaying the SWR will not work without changes. The feeding of the two voltages must be low impedance. For the following modifications are necessary: R59 and R60 are changed by 1K to 0R, R58 and R62 from 100K to 2,2K, C82 and C83 of 100nF to 2.2uF.
- (AG-xx-N-013) It is possible that the clock frequencies of the ui-board lead to disruptive spurious responses. To minimize this, one can attach a shield between the board-ui and the RF board. I have taken for this purpose on both sides a plastic-coated copper foil of a consumer electronics device. This can even "around turn around the board. The imagination knows no limits!
- (FW-xx-N-014) The Si570 is available in many different versions. The CMOS version alone in three versions ... The difference is in the startup frequency and address. the latest firmware 219.22 includes automatic detection of the start frequency. The firmware of my github also includes automatic address recognition. This can be used "in the drawer existing Si570", any.
- (RF-04-N-015) To get more transmit power, the small SMD transformers T5 is replaced by a small Binocular BN43-2402. The primary winding has received 3 turns 0,3mm enameled copper, the secondary winding is replaced by 4 turns 0,3mm enameled copper wire with a center tap.
- (RF-04-N-016) and RFC5 RFC6 be replaced by a respective binocular BN43-2402 4 turns 0,3mm enameled copper wire to get more transmission power.
- (UI-04-N-017) who will no longer want to perform with the Windows tool "mchf-manager" but via USB flash drive firmware update, the missing 5V supply of the USB port must be retrofitted. Given (the end of the board connector back) is placed a Schottky diode with the lowest possible voltage drop (anode to the resistor) to 5V pin of the USB OTG connector (which is the outermost pin for platinum under the edge) of R45. It makes sense also, a fuse (or even better Polyfuse) insert with 0.5A in the line to be in shorts to the USB socket on the safe side. Directly to the USB jack 100u / 6V tantalum capacitor is placed from 5V to ground to smooth yet. It is necessary then the USB Key bootloader of DF8OE from its Github . Since the firmware version 219.24 of the boot

loader in the master branch is included. Here is a link to guarantee functioning USB memory sticks . As luck would have it, have with me almost all that I have lying around (and there are many!) Run at all - but others had almost only "non-functioning sticks" at hand.

- (UI-04-N-019) If you want to save its configuration data and future advanced data in a serial EEPROM, equipped U7 on the UI board with a 24LC1025. This is a 128KB serial EEPROM whose memory is well equipped for future additional firmware features such as "station buttons". Before soldering placing a thin wire of Pin8 to Pin3. The EEPROM is used must be either new or if it has unsoldered somewhere, be described prior to installation with a programmer complete with 0xFF. The auto-recognition presupposes that the EEPROM has a stand 0xFF in all locations! Since the firmware 0.219.26.7 the serial EEPROM is supported by the firmware. **Note: the supplied of Chris 24LC01 can already more than a year not save the configuration data because it is hopelessly too small. Incorporate this EEPROM is counterproductive!**
- (UI-04-N-025) Who is the future and want to win a lot of additional GPIOs the MCU for other extensions, the newer LCD can operate (after 01/2015) in SPI mode. To do this, the resistors R30 ... R32-soldered on the ui board and three wires to be laid: the common end of R30 / R33 to LCD Pin 16 (SDO), the common end of R31 / R34 to LCD Pin 17 (SDI) and the common end R32 / R35 to LCD Pin 13 (SCK). Thus, the SPI bus is set to the LCD. Now, the jumpers must be set correctly on the back of the LCD: For SPI operation must be soldered IM0 and IM1, IM2 remains empty. Through major advances in firmware programming, there are no significant differences in speed between the more parallel and the SPI mode.



- (UI-04-N-026) If you want to activate the touch screen capabilities, has embarrassed five wires and R30 ... R32 unsolder (when the are loaded). The pin names are at the bottom of the LCD panel, I use these designations - the wires are laid but on the UI board itself. The wires do not interfere with the function of MCHF, even if the firmware does not yet support the touch screen! TP_IRQ hits the header Pin7 (== PA4 MCU), TP_SDO goes to the common terminal of R30 and R33, TP_SDI goes to the common terminal of R31 and R34, TP_SCK goes to the common terminal of R32 and R35 and TP_CS goes pin 1 of P8 (== PO9 the MCU). From the testing firmware 0.219.26.13 the touch screen is supported. Please note that the red wires are NOT necessary for the function of the touch screen. These wires are for the operation of the LCD in SPI mode required (UI-04-N-025). If one makes the touchscreen modification, it makes sense to do this modification with the same.



- (RF-04-N-028) The driver transistors used have a cutoff frequency of 300MHz. When operating in an emitter circuit the higher bands are already affected by a decrease in the gain. One can use the identical BFQ18A instead of DXT3150. The output power thereby increases by 3 ... 6dB. On the lower bands, it is (depending on the executed other modifications) now no longer possible to set a smaller output power than 15W Power menu. Who is affected, must activate the item "Reduce Power on Low Bands" Power menu. This gain will be reduced at all frequencies below 8MHz by 6dB.
- (RF-04-N-030) The small, hard to be soldered buffer ICs U9, U12 and U13 are unnecessary - no - they even worsen the phase behavior and thus the function of MCHF! Therefore, they are not equipped. R19 is replaced by a 0R resistance and the pins connected 2 and 4 of U9 with a braided wire. In U12 and U13 respectively pins 1 and 6, and 3 and 4 are bridged with a braided wire. This will "bypassed" the buffer.

DF8OE, 06/20/2016

Download as PDF