

# QMX+ ATU, Battery, Charger and speaker solution

Build instructions

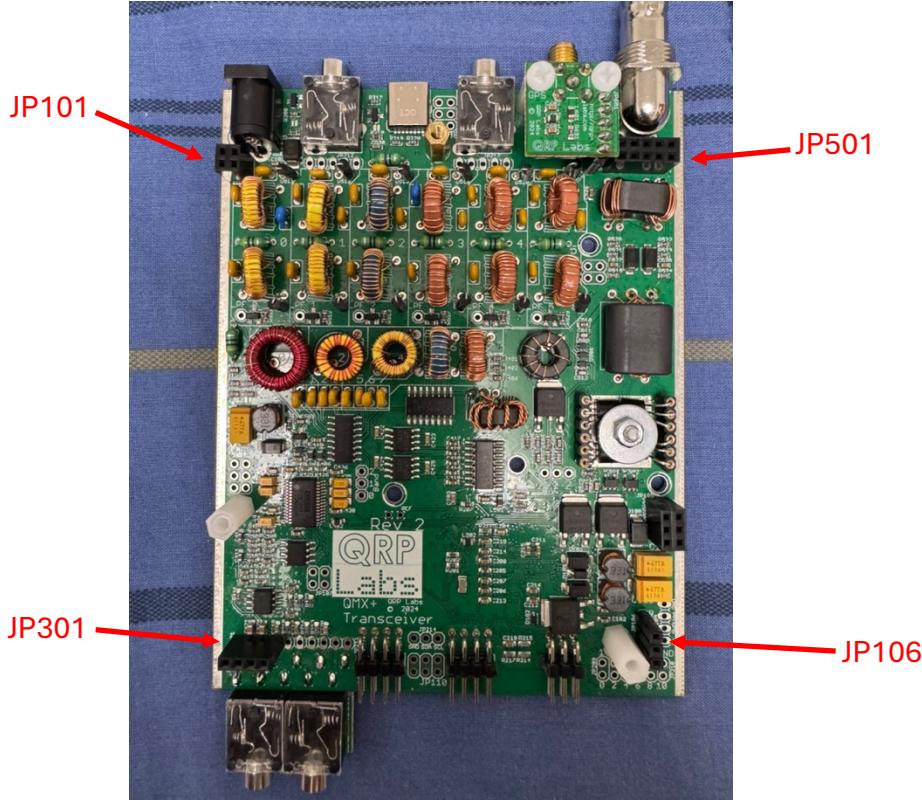
Ver 1.0

By AC8L

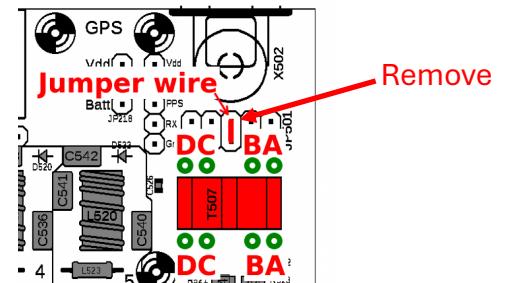
**WARNING!!! The battery portion of a solution is for the QMX+ 12V build only!!! It will damage the 9V QMX+ !!!**

Please read instructions to the end before starting the assembly. Then go back and start the assembly by following instructions step by step from the beginning.

## 1. Install connectors to the QMX+ main board

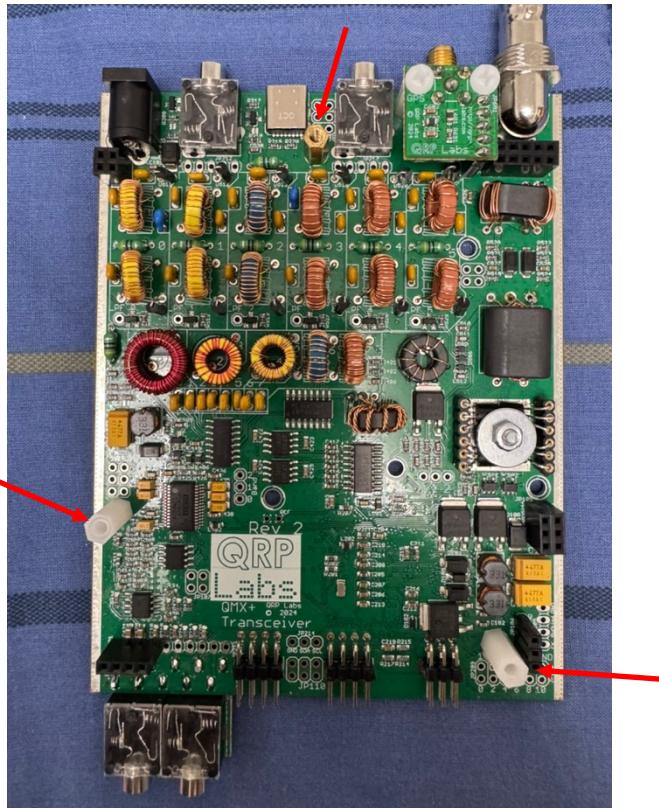


If you have QMX+ pre-built and functioning – prior to soldering JP501 you MUST remove the jumper wire soldered on it (QMX+ assembly instruction REV2.0 page 37).



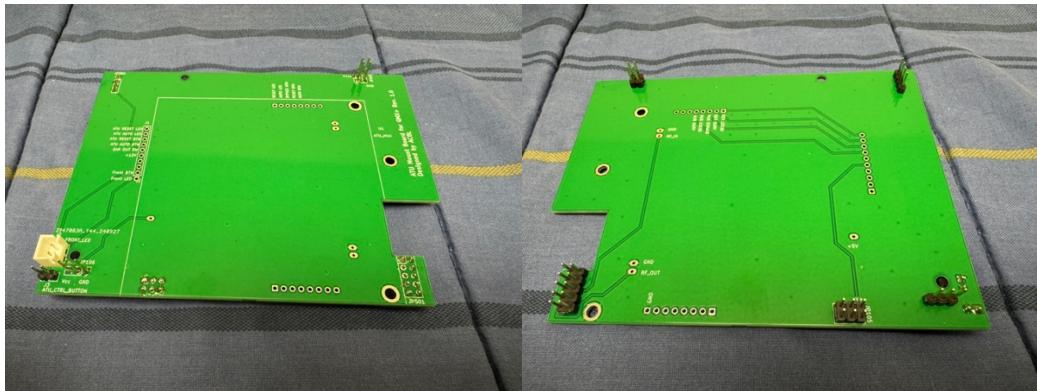
Do this carefully not to damage the nearby elements and soldering pads! E.g. completely remove GPS module if installed and unscrew its nylon spacers. After jumper wire removed – use desoldering suction tool to remove excess solder and cleanup holes preparing them for soldering the JP501 female socket.

## 2. Install three 11mm nylon standoff spacers

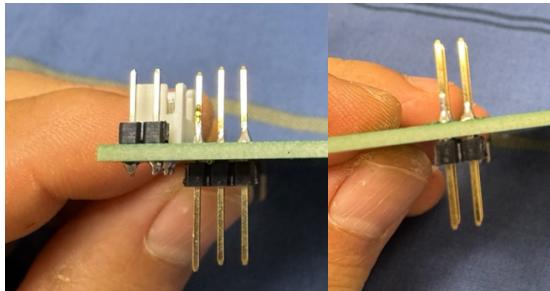


Use 6mm nylon screws to secure spacers from the bottom of the QMX+ main board.

## 3. Install pin headers to ATU Mounting PCB



Remember, JP106 connector (left) and JP102 connector are long-tail! Make sure the plastic part is flush at the bottom of the PCB.



- Only J2 (front-panel LED) and J3 (front-panel button) connectors are soldered on the front of the PCB. All others – on the back.
- At this point we DO NOT attach J1 connector!

The best tactic is to make sure pin headers are properly aligned with the female connectors on QMX+:

- Insert them into the female connectors on QMX+ main board
- Place ATU mount PCB on top, make sure pin headers are all inserted into their places on the ATU mount PCB.
- Temporarily secure the ATU Mount PCB to the QMX+ using three 6mm nylon screws. Screw them into the nylon spacers that are already attached to the QMX+ main board.
- Carefully solder pin headers of all connectors from the top of the ATU mount PCB.
- Unscrew 6mm nylon screws and detach the ATU mount PCB from QMX+.

## 4. Prepare ATU to be soldered to the ATU Mount PCB

At this point ATU must be fully assembled with all the toroids and DIP socket installed on it. Build ATU with [instructions at the WB2CBA's page](#) with following mods:

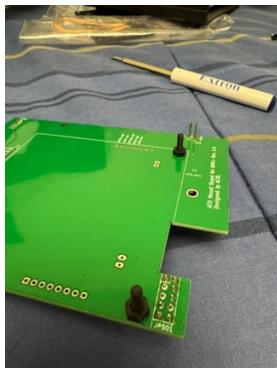
1. Do not source and solder two BNC connectors.
2. Do not source and solder male pin connectors to the ATU board. We will use different ones for our integration.
3. Use only the ATU section of the page, ignore the rest for the purpose of this project.

Do not insert the PIC16F1938 microcontroller into the ATU yet. We will need to program it at later step.



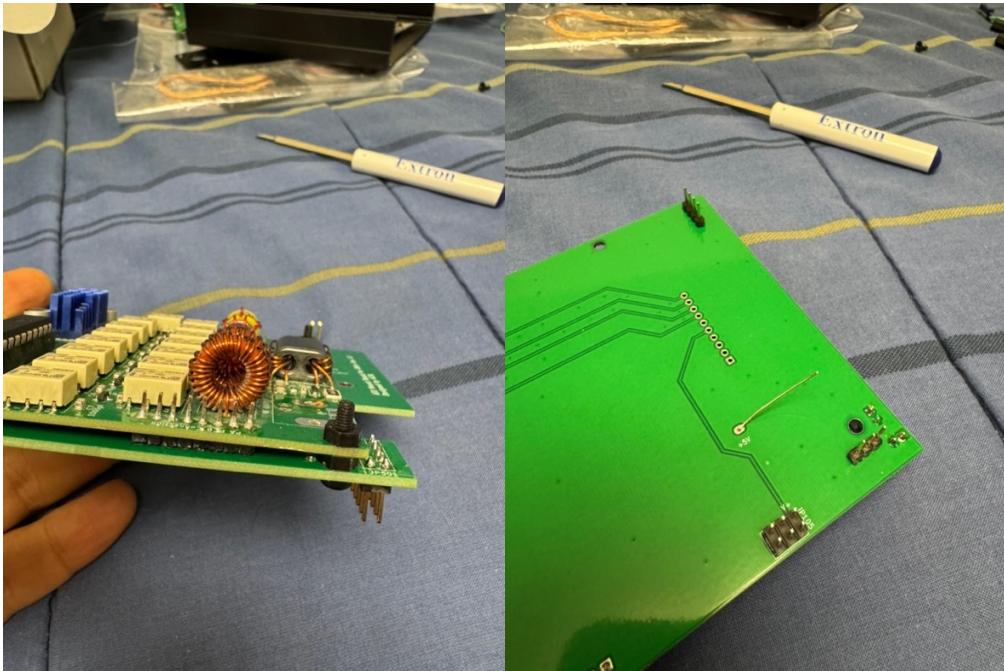
Solder both 8-pin male pin headers to the ATU from the bottom (see photo above). Also solder one of the left-over through-hole component pins to the PIN 20 of the pc microcontroller. On the photo above it is a pin 6 from above on the row close to the PCB edge. Make sure the soldered PIN is as vertical as possible and not bended.

## 5. Prepare the ATU Mount PCB for the ATU to be mounted on it

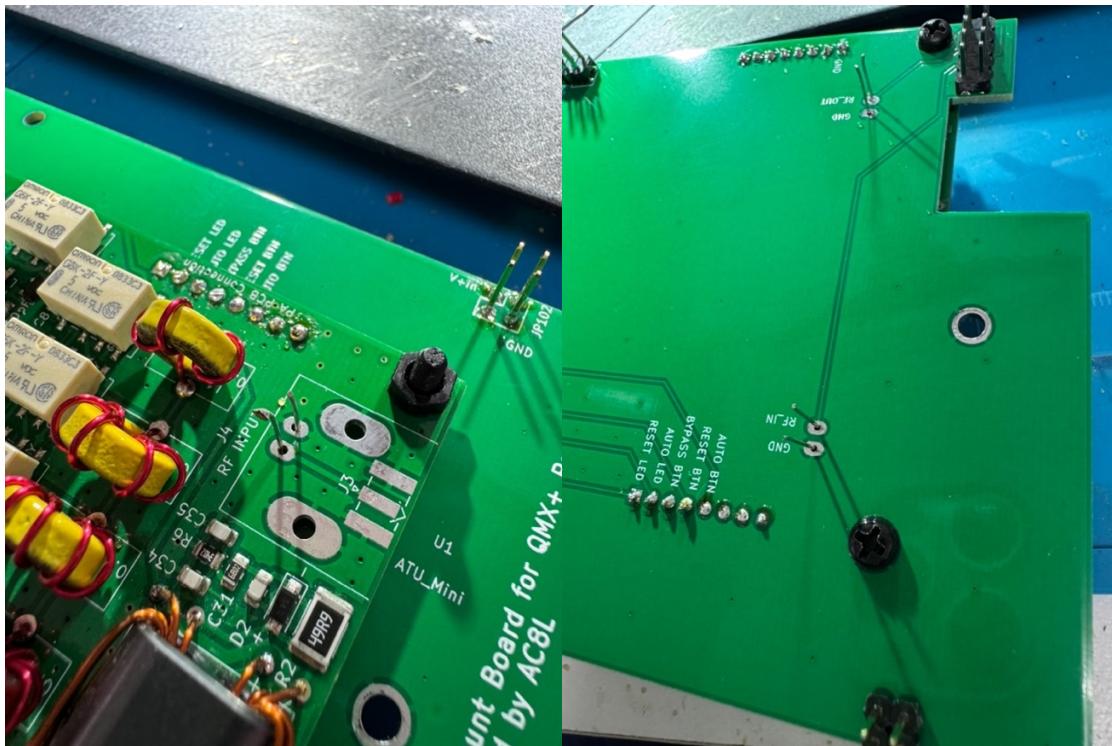


Attach two 12mm nylon screws from the bottom of the ATU mount PCB and secure them with hex nylon nuts from the top.

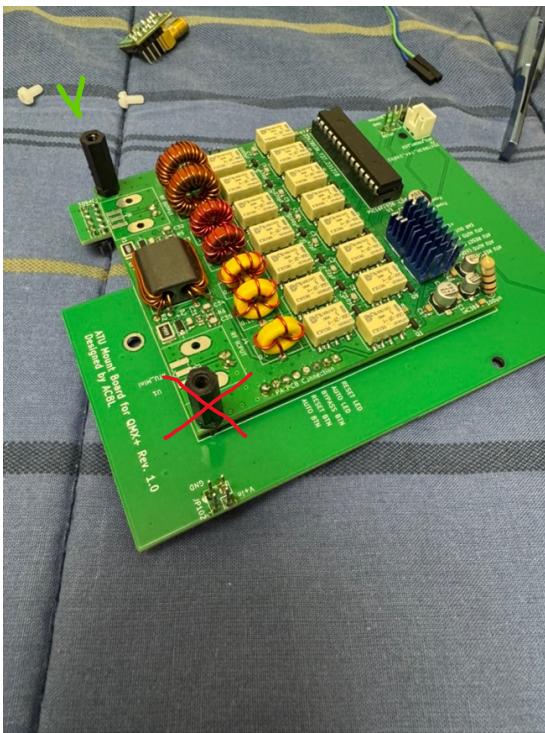
## 6. Attach ATU to the ATU Mount PCB



- Place the ATU on the top of the ATU mount PCB, make sure pin headers go through their holes on the ATU mount PCB and 12mm nylon screws go through their mounting holes on the ATU itself (left photo).
- Also, make sure the vertical pin soldered to the PIC microcontroller pin is also passed through its place on ATU mount PCB (right photo). The soldering pad it goes through marked as “+5V”.
- Secure the ATU to the ATU mounting PCB using a pair of M3 hex nuts (left photo).
- Solder pin headers from the bottom of the ATU mount PCB.
- Solder +5V pin to its soldering pad (left photo).



- Insert 2 left-over pins from through-hole components into RF-IN and ground pin next to it (left photo).
  - Make sure other side of pins are protruded through their corresponding soldering pads from the ATU Mount PCB bottom (right photo).
  - Do the same for RF-OUT connectors.
  - Solder top and the bottom of left-over pins.
  - Cut all excessive pin headers from the bottom of the ATU mount PCB and from the top of the ATU.



Attach ONE M3 15mm Nylon female-female spacer on top of the hex nylon nut securing the ATU to the ATU Mount board. Do it only for the 12mm screw close to the JP501 connector.

This concludes the ATU mount PCB assembly.

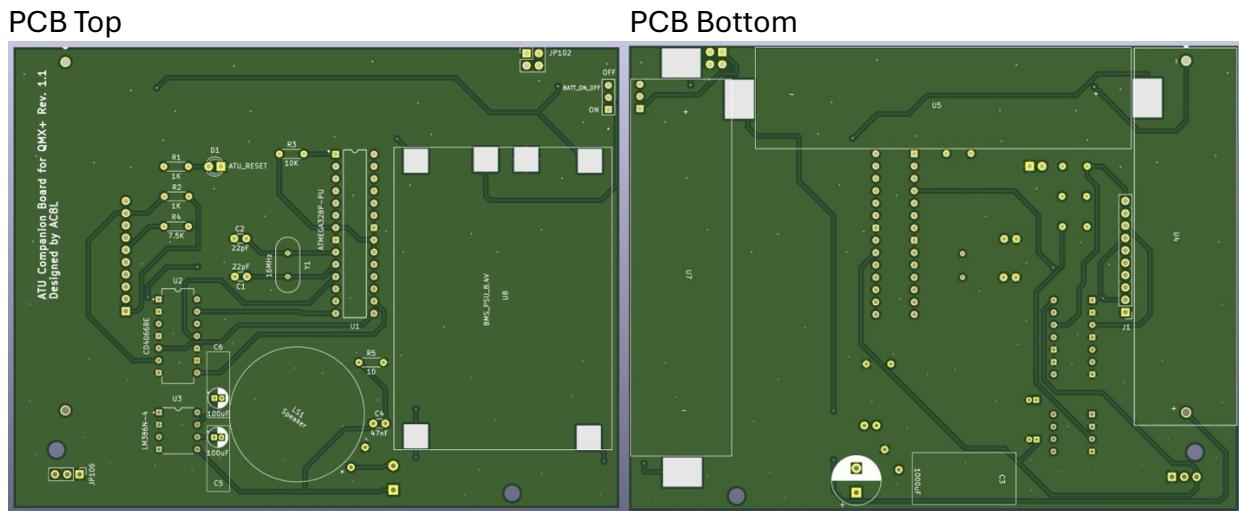
## 7. Assemble the ATU Companion PCB

The solution essentially consists of THREE distinct mods (ATU, battery and speaker), independent from each other, loosely coupled only through the QMX+ itself. You can pick and choose which one(s) you are going to build and add others later. Please consider following possible build configurations for the solution:

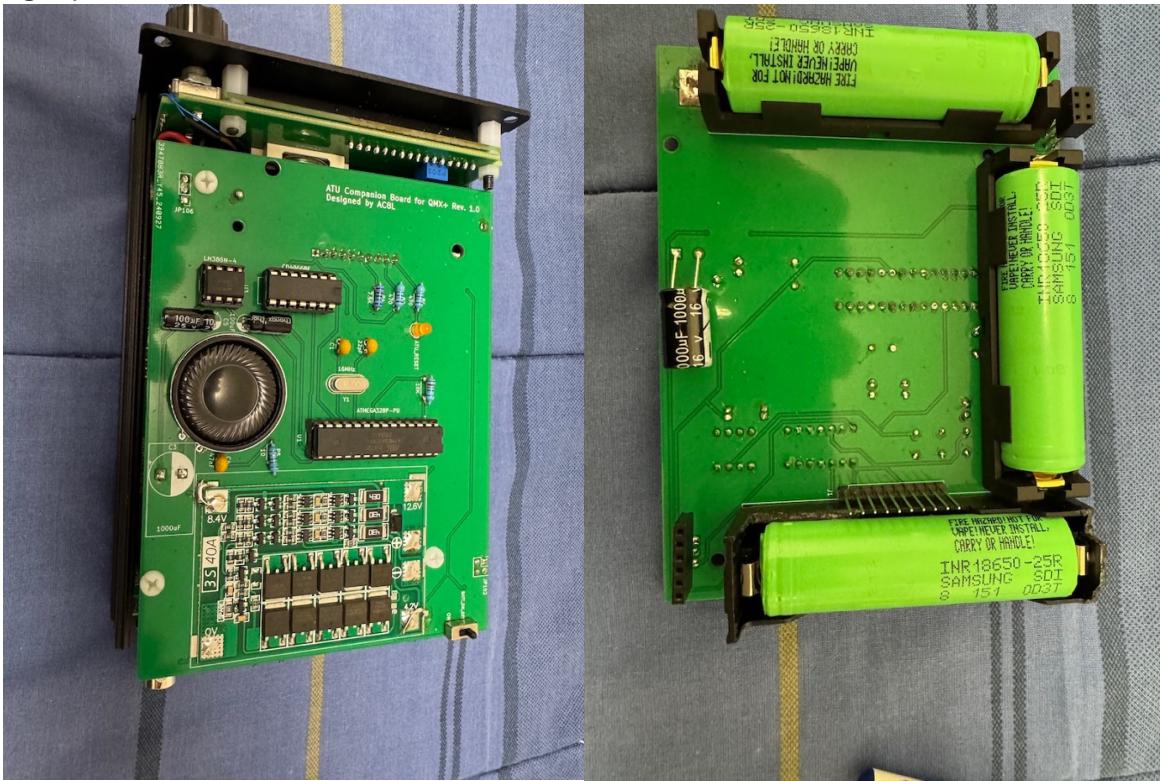
- All inclusive - ATU, battery, speaker.
  - ATU only
  - Battery only
  - Speaker only
  - ATU and battery
  - ATU and speaker
  - Battery and speaker

Depending on the build configuration you chose – populate components according to the following table.

| <b>QMX+ Mod</b>                            | <b>Components</b>                |
|--|----------------------------------|
| ATU  | R1,R2,R3,C1,C2,D1,Y1,U1,U2,JP106 |
| Battery + charger (QMX+ 12V build only!!!) | U4,U5,U7,U8,JP102,BAT_ON_OFF     |
| Speaker and audio amp                      | R4,R5,C3,C4,C5,C6,U3,LS1,JP106   |



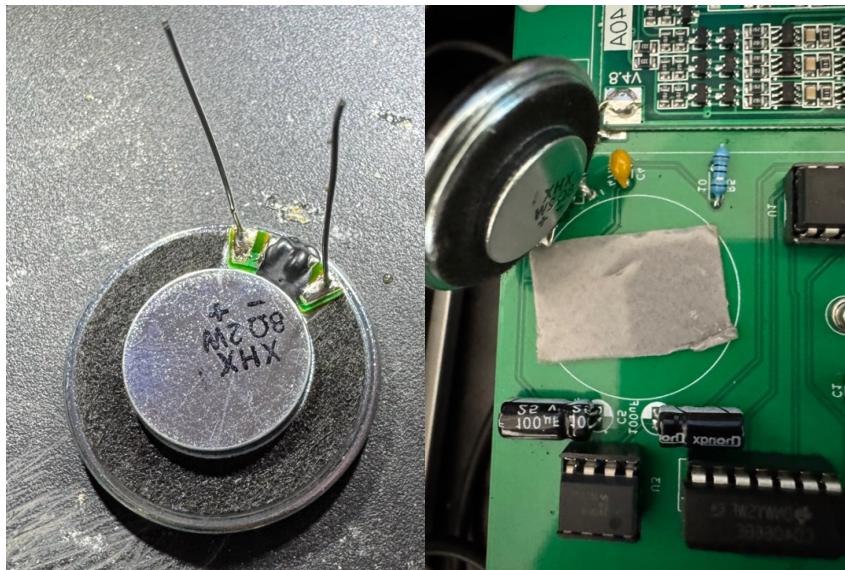
The front of the fully populated Companion PCB is shown on the left photo, the back on the right photo below.



- The C3 capacitor is soldered to the bottom of the PCB, it has a dedicated footprint on the bottom silkscreen for its placement.
- Battery holders' step on each other, so they must be soldered in the following order:
  - U5
  - U7
  - U4
- **It is mandatory to make sure that battery holder's polarity is matched with +/- signs printed on the silkscreen. Otherwise, you are risking losing your QMX+!**
- The U4, U5 and U7 footprints are highlighted with a white border line on the silkscreen. Make sure battery holders are precisely inside their footprints. Otherwise, you will have difficulty sandwiching the companion board on the top of the ATU and closing the QMX+ enclosure top cover.
- Make sure, batteries are not inserted into battery holders during the entire assembly process! They will be inserted during final checkups and assembly step.
- Insert U2 and U3 chips into their respective sockets, but not the U1. We will need to upload the Arduino sketch onto it and burn the Arduino bootloader if necessary. We will do this at the later step.

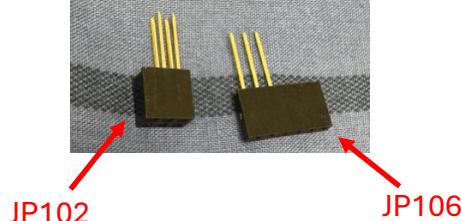
## 7.1. Speaker

- Speaker also can benefit from the unlimited stock of left-over through hole component pins you are expected to have after the QMX+ assembly, the left photo below shows speaker with pins soldered on it.
- Just solder pins to the speaker connector pads, insert it to the PCB and solder pins from the PCB bottom.
- Use 3M double-sided tape to secure the speaker to the PCB. This will also suppress the rattling sound.



## 7.2. Attaching JP102 and JP106 connectors

Because of the space between ATU Mount PCB and ATU Companion PCB we need long-tail female connectors for both – JP102 and JP106 connectors on ATU Companion board. However, I had difficulty sourcing 2 row 2x2 Long tail (JP102) and single row 3-pin female (JP106) connectors. Instead, I ended up using 2x3 long tail for JP102 and single row 6 ping connector for JP106, cutting unused pins as shown in the photo below:



To make sure, that proper spacing is achieved with proper electrical connectivity, it both PCB's needs to be sandwiched together before connectors are soldered to the ATU Companion PCB.

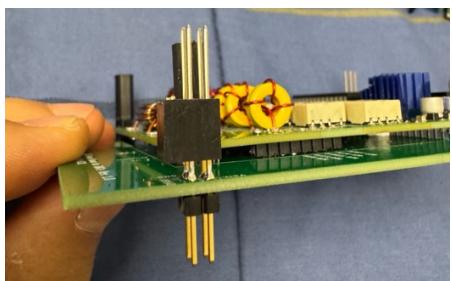
We will need 15mm nylon standoff male-female spacer and a hex nut, screwed onto it:



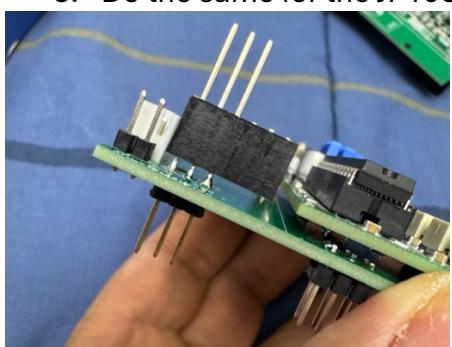
1. Attach spacer between ATU Mount board and ATU Companion board as shown in the photo below. ATU Mount board does not have to be attached to the QMX+ at this step.



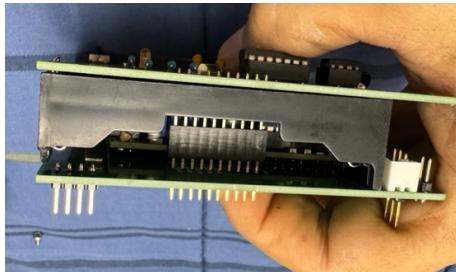
2. Insert 2x2 JP102 connector on **TOP!** of the ATU Mount PCB as shown below:



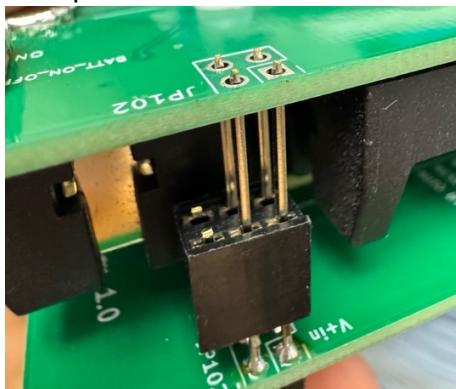
3. Do the same for the JP106 connector:



4. Place ATU Companion board on top of the ATU Mount board, forming a sandwich, like on the photo below:



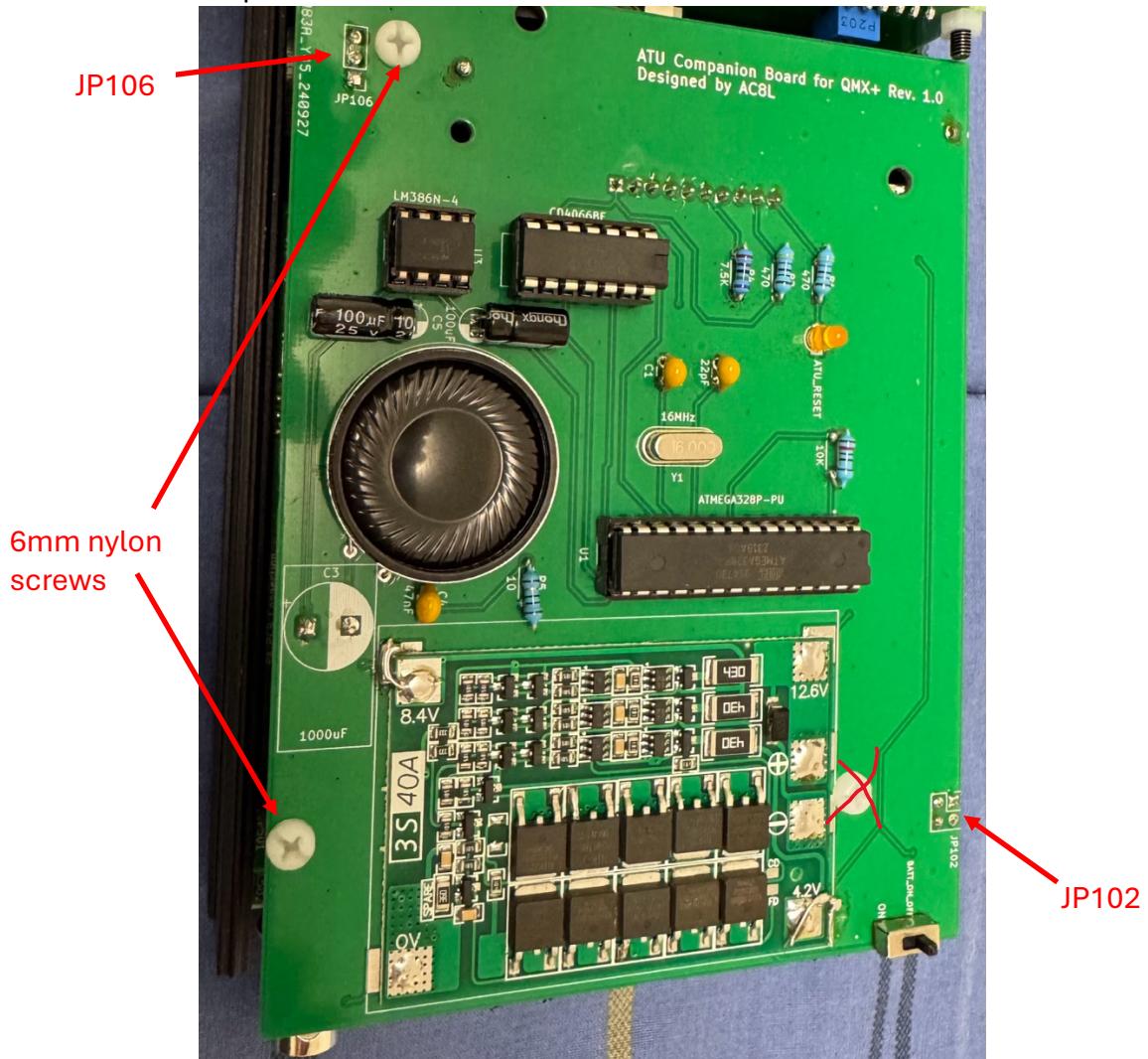
Make sure, JP102 connector pins are slightly protruded through the top of the ATU Companion PCB:



Same should be for the JP106 connector:

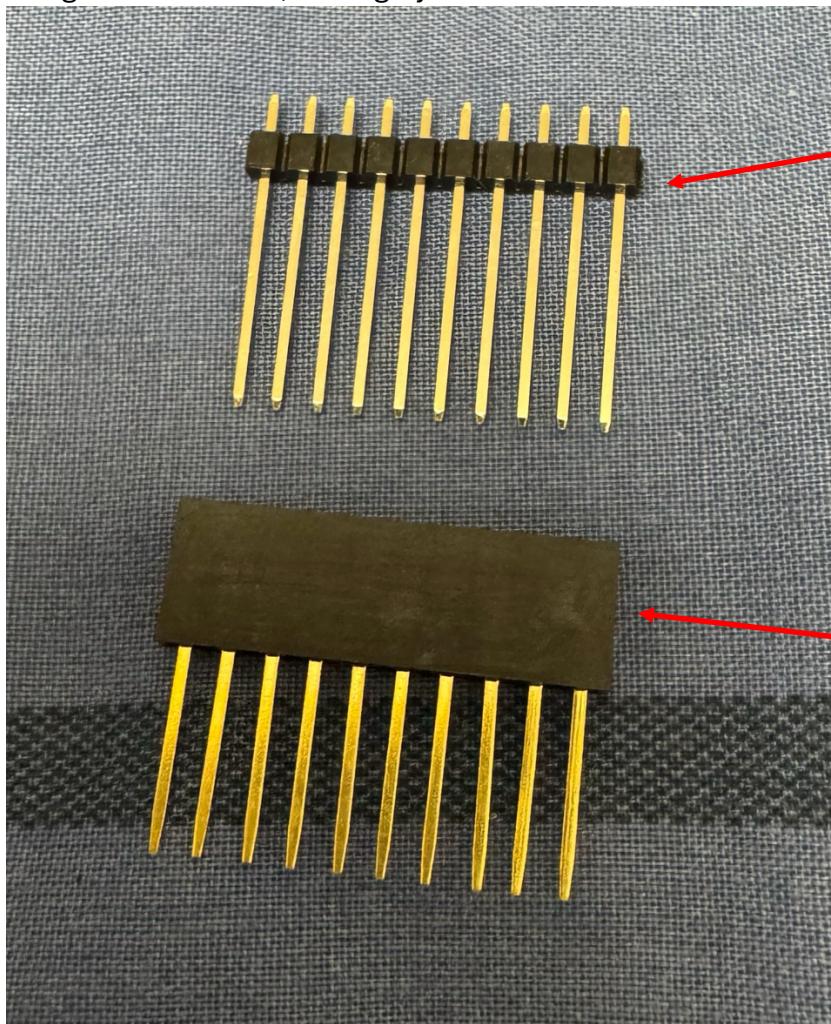


After PCB boards are sandwiched, secure them to each other using two 6mm nylon screws. When PCB's are secured, go ahead and solder both JP102 and JP106 from the top of the ATU Companion PCB:



### 7.3. Attaching J1 10-pin connector pairs between ATU Mount and Companion boards

Both PCB boards for the solution and interconnected through a pair of male and female long tail single row 10 pin connectors. Electrically it is not required if only battery and charger are installed, but highly recommended for mechanical tightness anyway.

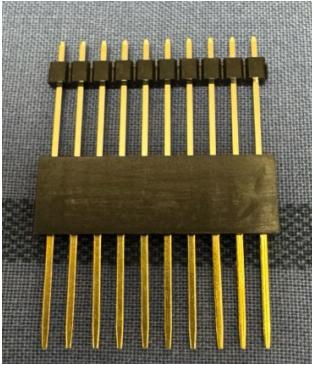


10-pin single row male long tail pin header – attached to the bottom of the ATU Companion PCB

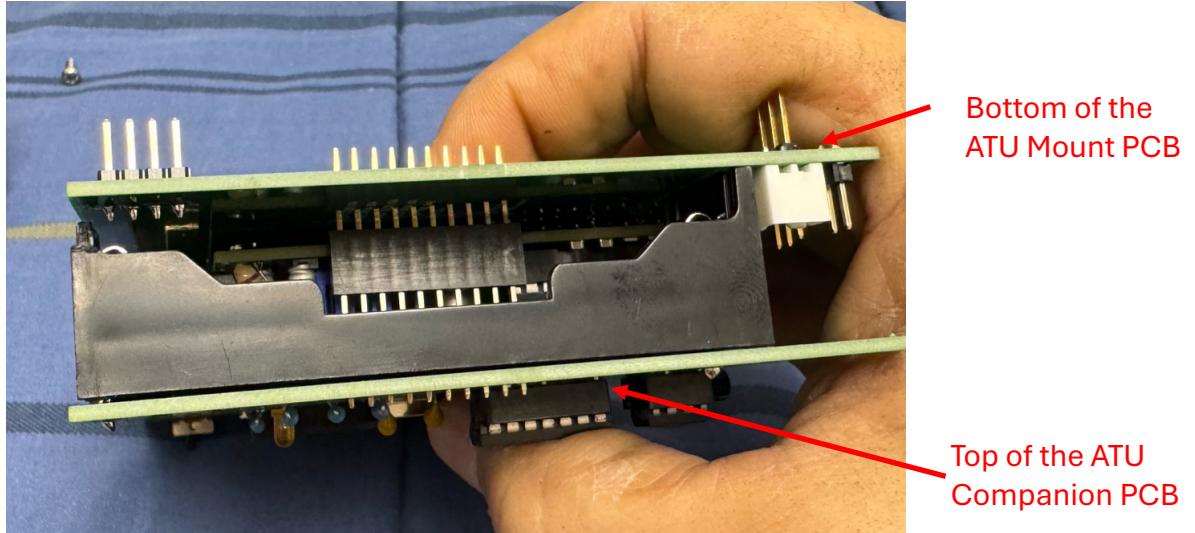
10-pin single row female long tail pin header – attached to the top of the ATU Mount PCB

To ensure the best placement and electrical connectivity, both PCB's will need to be sandwiched as in previous step. At this point, the ATU companion PCB should have both JP102 and JP106 already soldered, so this will add additional mechanical sturdiness and ease of assembly step.

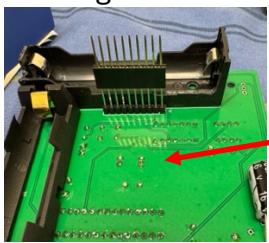
1. Insert both pin headers to each other, slightly push, but not too much, the male connector might damage the female counterpart if too much force is used:



2. Sandwich both – ATU Mount PCB and ATU Companion PCB together, while making sure the J1 connector pair is inserted in its place. To assure that male pin headers plastic part is flush against ATU Companion board, keep sandwich upside down:



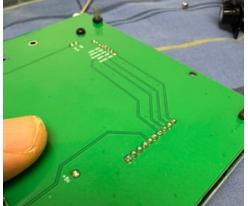
3. Squeeze both PCB's together and secure them to each other using two 6mm nylon screws as in previous step 7.2. While securing PCB's, keep them upside down.
4. Look from the sides to make sure that the plastic part of the male pin headers is flush against ATU Companion PCB's bottom:



5. Solder the female pin header from the BOTTOM of the ATU Mount PCB.



6. Cut out the excessive pin length of the female pin header:



7. Then turn sandwich around and solder the male pin headers from the top of the ATU Companion PCB:



Go ahead and detached PCB's from each other – the top of ATU Mount PCB should look like this:



**\*\*\* Congratulations! If you reached this place, it means you have successfully assembled both PCB's! \*\*\***

## 8. Modifications of the front panel of the QMX+ enclosure

I have tried my best to avoid the mutilation the QMX+ aesthetics as much as possible. However, current firmware features are not allowing yet assigning custom actions to the existing front panel controls of the transceiver. So, I have committed these modifications with great hesitation. Hopefully, as QMX+ firmware evolves, this feature of the build process might change accordingly in the future.

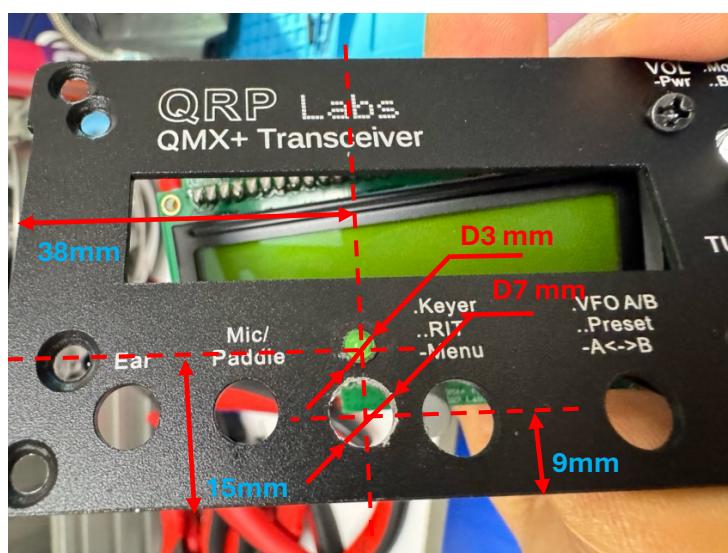
We have three options for the front panel modification:

1. Drilling the factory front panel (less desirable option)
2. Printing the replacement front panel on a 3D printer (budget option). STL file can be downloaded from [here](#). PETG filament is recommended due to the outdoor exposure.
3. Ordering the front panel from JLCPCB (most aesthetically appealing option).

*This step requires working with the 30 AWG Kynar wire. The only challenge with it - is that many cheap wire stripping tools are not stripping its insulation properly. The one of few that 100% is guaranteed to do the job – is Hakko CHP CSP-30-1 wire stripper: <https://a.co/d/90oB19d>.*

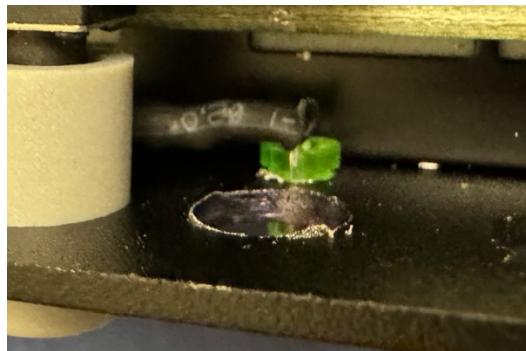
### **OPTION 1- Drilling the factory front panel**

1. Disassemble the QMX+ front panel assembly.
2. Drill two holes on it:

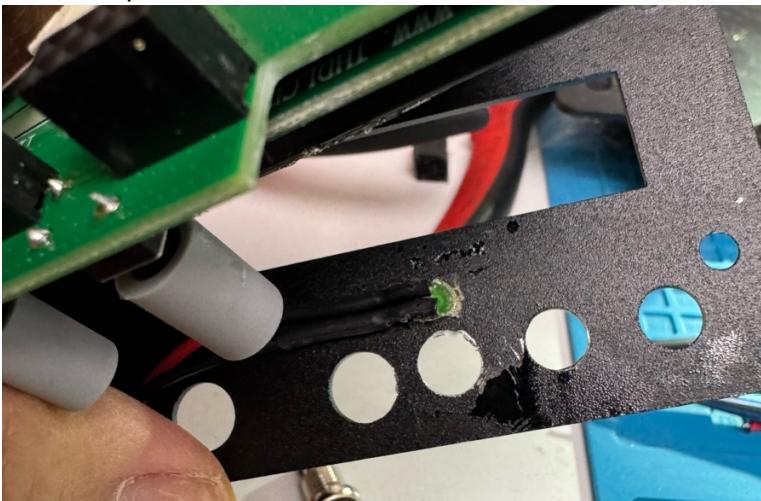


The 3mm hole will be used for the Autotune status LED, the 7mm diameter hole – for the ATU control button.

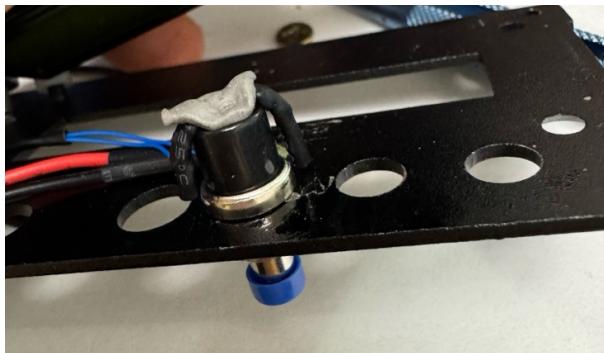
3. Solder the JST XH pigtail to the LED, red wire to the anode. Use shrink tubing for the each electrode. The use the one more shrink tubing on top for both electrodes.



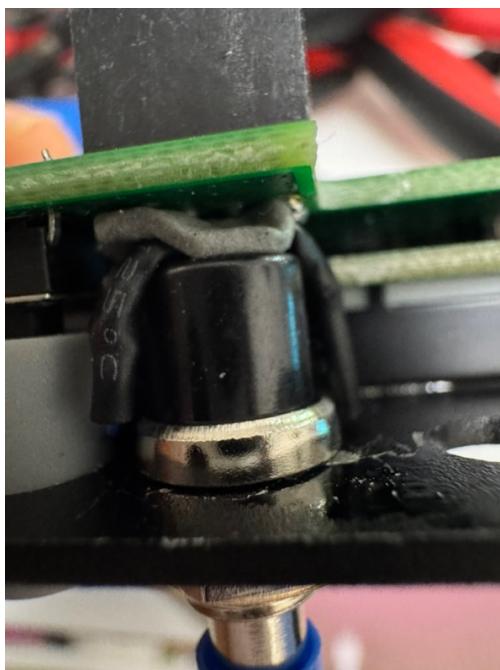
4. Bend LED electrodes by 90-degree angle, insert LED through the 3mm diameter drilled hole from the inside of the front panel. Make sure when front panel is assembled – wires are passed between front panel buttons. Use very small amount (half a drop of Gorilla glue, the gel version is the best) to secure the led inside the hole. Wait 24h to make sure glue is dry. Position of the LED from inside the front panel should look like this:



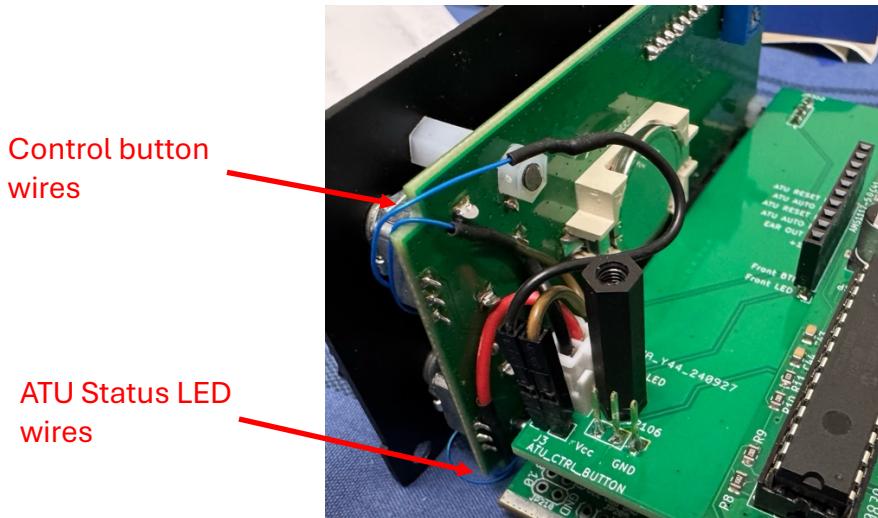
5. Solder about 15 to 20cm of Kynar wire into contacts of the control button. Use shrink tubing. Gently bend contacts after tubes are shrank:



5. Use small (approx. 10x10mm piece of the M3 double sided tape to close the uninsulated portions of button's contacts. Insert button from the inside of the front panel to the 7mm diameter hole. Secure it from outside using washer and a nut that comes with the button. Do not use instruments, just your fingers to tighten the nut!
6. Assemble the front panel. The position of the ATU control button against front panel PCB should look like this:

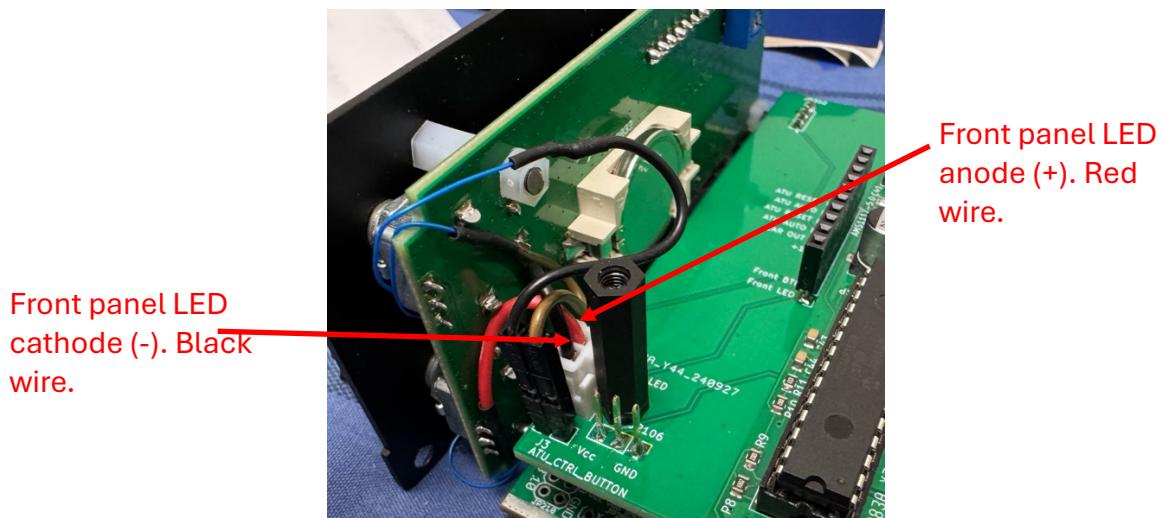


The only “safe” place to pass wires from the LED and button to the PCB compartment – is through corners of the front panel:



Now it probably makes sense why Kynar wire is used – it is to achieve this pass-through without risk of metal enclosure tearing down the thicker wire insulation.

7. Attach front panel to the QMX+ main PCB.
8. Attach the ATU mount PCB to the QMX+ main PCB.
9. Pass the control button Kynar wires from the top corner of the front panel as shown in the photo above.
10. Cut out the excess Kynar wires at approx. 3-4 cm from the corner. Cut and solder jumper wires to both ends of the Kynar wire. Use shrink tubing. The length of jumper wires should be around 7-8cm and jumpers should be able freely seated into their respective pin headers. The polarity of control button wires are not important.
11. Repeat the same procedure for LED wires. Except that LED already has a JST connection at the end of its pigtail. So, gently bend pigtail around bottom corner of the front panel. Find out the 7-8 cm long segment that bends through the front panel PCB edge. Cut out both wires and replace them using Kynar wire. Again, use shrink tubing. The polarity of LED wires DOES matter:



If wire positions do not match the photo (which is a usual case with Chinese pigtails from Amazon) – use tweezers to remove them from the JST connector and swap. Or you can just swap them while cutting and adding Kynar between them. Make sure that polarity is observed using multimeter's diode mode.

12. Disconnect the front panel ATU Status LED and control button from ATU Mount PCB. Detach the front panel from QMX+ main PCB. The front panel with modifications should look like this:



## **OPTION 2 – 3D printed front panel**



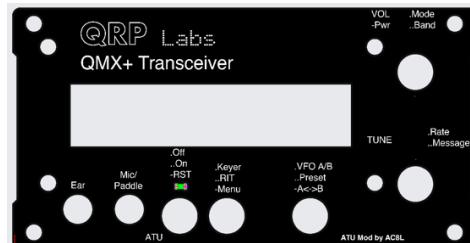
- Since mounting holes are already in place – just insert and screw in the control button.
- For LED – no need to glue it in. LED hole tolerance is enough for just squeezing the LED in. It will securely sit in its place.

## **OPTION 3 – PCB Front panel**

### **Go ahead and order one of two versions of the PCB front panel:**



- with LED manually soldered from the back.  
Fabrication files are [here](#).



- with SMD LED factory-soldered on the front.  
Fabrication files are [here](#).

Assembly process for both versions are identical except that for the second version we do not solder the LED – it is already there for us.

1. Bend and cut LED pins as shown in the picture below:



2. Insert the LED to the panel hole and solder it paying attention to the polarity:



3. Bend contacts of a 6x6 tactile button (6.5mm height):



4. Solder the button from the back of the front panel:



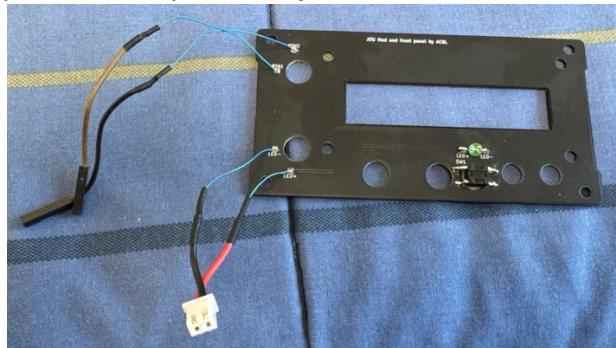
5. Make sure that button is flush against the surface and centered:



6. Print the button cap in 3D printer with gray filament and attach to the button's knob. [STL file can be downloaded from here.](#)



7. Solder LED JST connector and button wires with Kynar ends to the appropriate pads on the panel. Pay attention to the JST connector polarity:



8. Assemble the front panel module of QMX+.

#### ***Optional steps for PCB front panel***

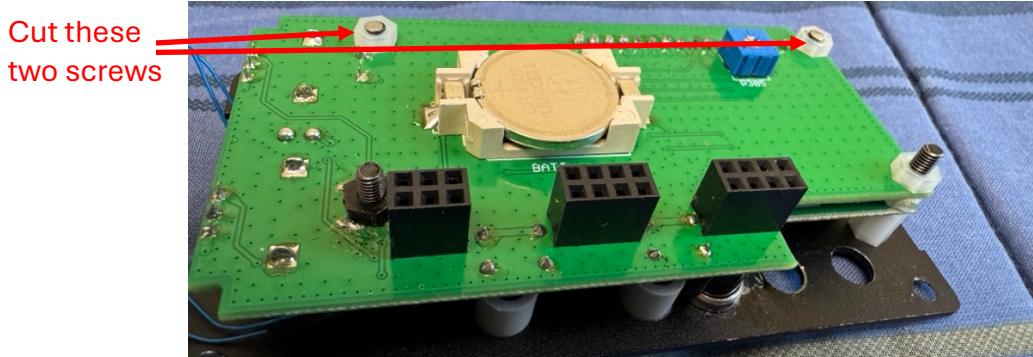
The mounting holes of the PCB front panel has diameter that allows flat head screws used in QMX+ front panel and enclosure fit such a way that only about .5 mm of their head is over the surface of the panel. However, if you want a perfect fit – you can use a Countersink tool on screw holes. To blacken the white PCB opening – use a Sharpie. The same sharpie can be used on PCB white edges and edges of panel's opening for the LCD:



[Link to the Countersink tool](#)

### **Last step – common for ALL 3 front panel options!**

Turn the front panel around. Use dremmel tool to shorten the top two screws. Make sure they are about 0.5 - 1mm from the nylon hex nut:



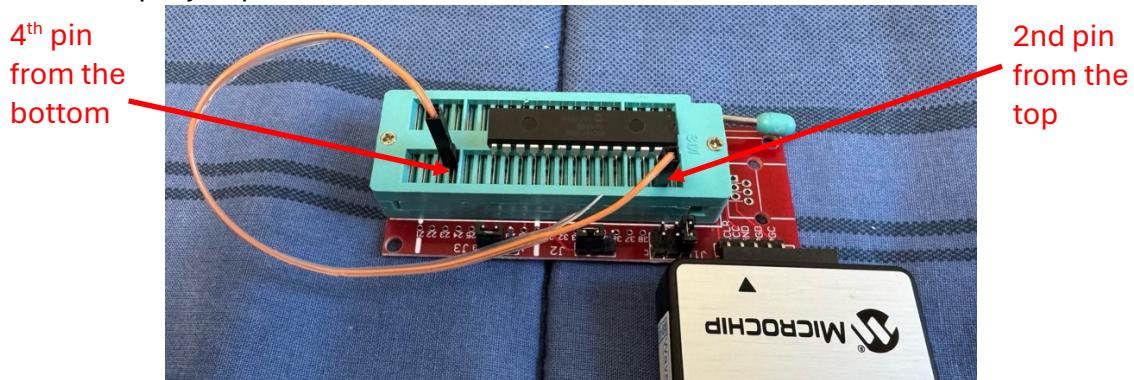
**\*\*\* Congratulations! If you have completed building  
the hardware portion of the solution! \*\*\***

## 9. Firmware upload

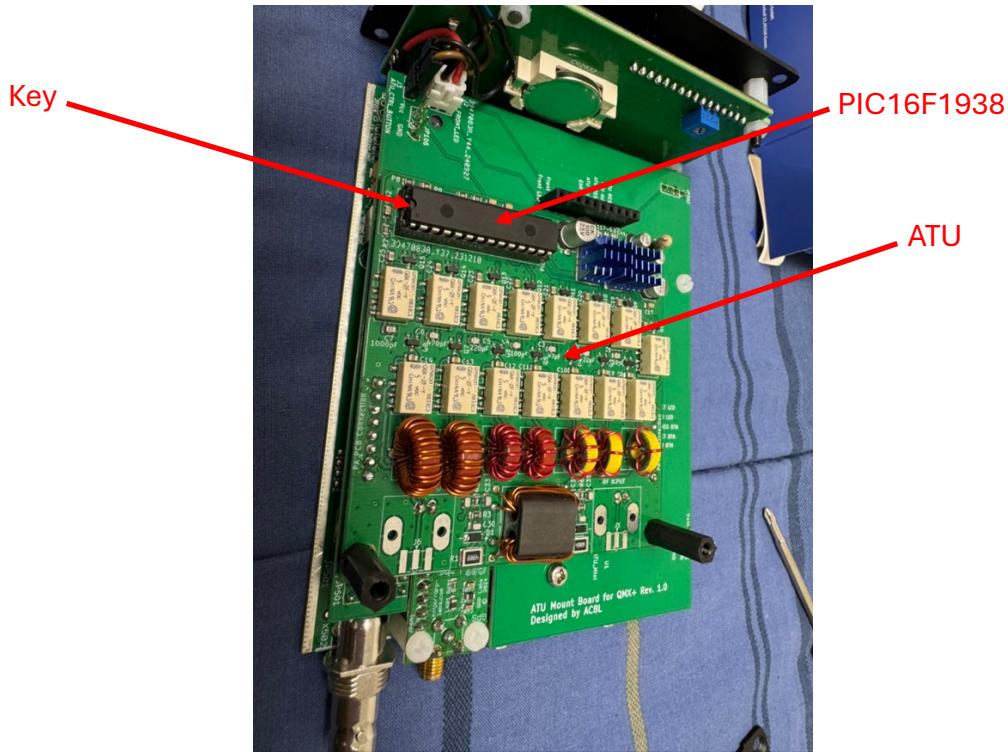
### 9.1. Upload firmware to PIC16F1938

The ATU for QMX+ integration needs a custom firmware that can be downloaded from [here](#). Full instructions of how to perform the PIC microcontroller programming can be found on ATU build page at Barb's site [here](#).

During programming I have encountered repeated error from PIC programming software that device ID could be found. It turned out to be that shim socket for the programmer was not routing the PGC signal to the appropriate microcontroller pin. To solve this problem, I have used a simple jumper wire:



After firmware is successfully uploaded – insert the microcontroller into its 28pin DIP socket on ATU board. Make sure the proper key orientation:



## 9.2. Burn the Arduino bootloader into ATMEGA328P if necessary

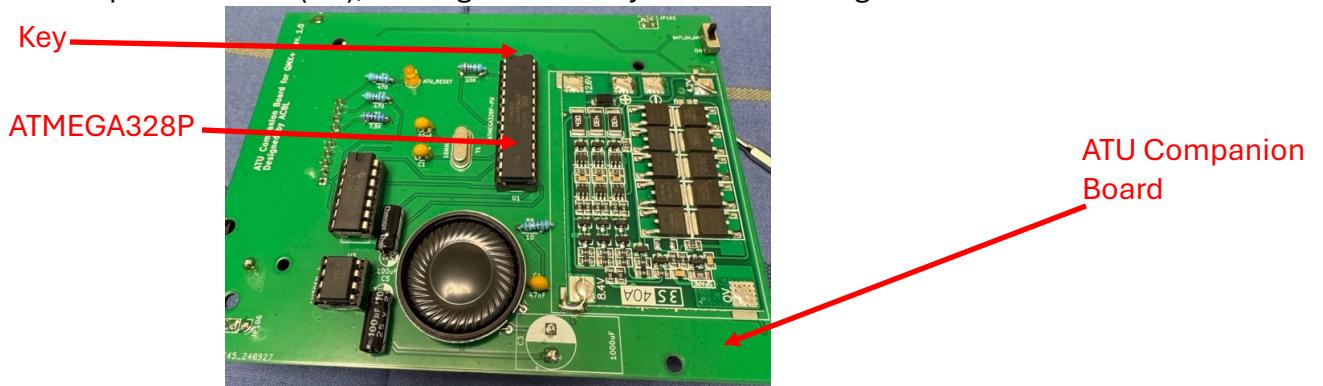
If ATMEGA328P was purchased from Digikey or Mouser – most likely it will not have the Arduino bootloader that will prevent using it with Arduino IDE.

In this case follow instructions [here](#) to burn the bootloader into it. You will need two Arduino Uno R3 boards for this procedure.

## 9.3. Upload firmware to ATMEGA328P

1. We will use Arduino Uno R3 as a programmer. Only one board is needed for this step. Remove factory installed Arduino chip from the Uno, plug in the chip you are going to use with ATU Companion PCB.
2. Download the firmware sketch from [here](#). Open it in Arduino IDE, connect the UNO board to your computer and select the proper port and device (UNO).
3. Firmware depends on OneButton library. Install it using Arduino IDE Library Manager.
4. Build and upload the firmware into the ATMEGA328P chip.

- Verify that firmware is loaded properly by opening Arduino IDE Serial Monitor tool. You should see a debug message “**QMX+ ATU companion board initialized.**” There might be some garbage characters around that text. Just ignore.
- Power off the UNO board. Remove chip from the UNO board and install into the ATU Companion board (U1), making sure the key orientation is right:



**\*\*\* Congratulations! If you have reached the last milestone before the final assembly and just a step away from enjoying the All-In-One QMX+ solution! \*\*\***

## 10. Final tests and assembly into the enclosure

- Make sure ATU Mounting board is detached from QMX+ main board and ATU Companion board is detached from ATU Mounting board. QMX+ assembly should be removed from the enclosure.
- Make sure switch on ATU Companion board is on OFF position:

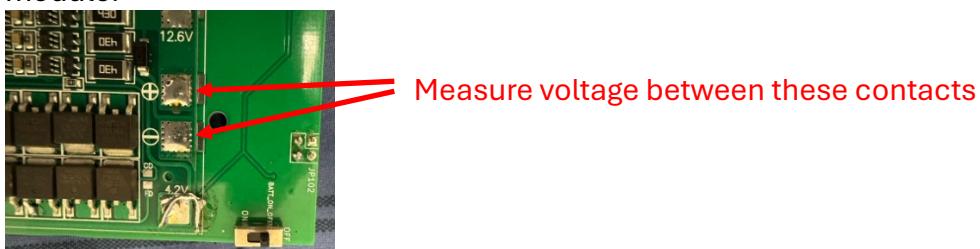


- Using multimeter, measure the voltage between contacts of each of the 18650 batteries individually. Batteries shipped partially charged, so the voltage could be somewhere around 2.5-3V.

- Insert batteries into battery holders, paying attention to the proper polarity:

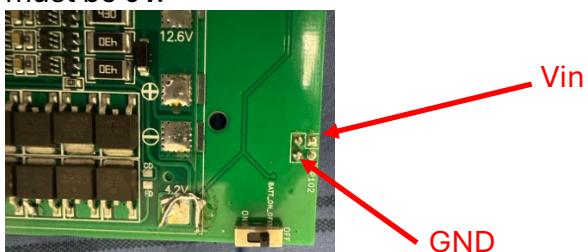


- Using multimeter, measure the voltage between (+) and (-) contacts of the BMS module:



Voltage must be a sum of the three individual 18650 batteries. For example, if each battery had voltage 3v, the sum should be 9V. If voltage significantly differs (more than 0.3V) from the sum, this might mean either improper polarity, or bad soldering of battery holders, or bad soldering of the BMS module. If this is the case, stop, remove batteries from battery holders, check contacts and fix the problem before proceeding further. **Improper battery placement or bad contacts between BMS module and PCB soldering pads might cause serious damage to the QMX+, thermal issues or even a fire if external charger is connected to the battery pack.**

- Measure the voltage between Vin and GND contacts of the JP102 connector. Voltage must be 0V.



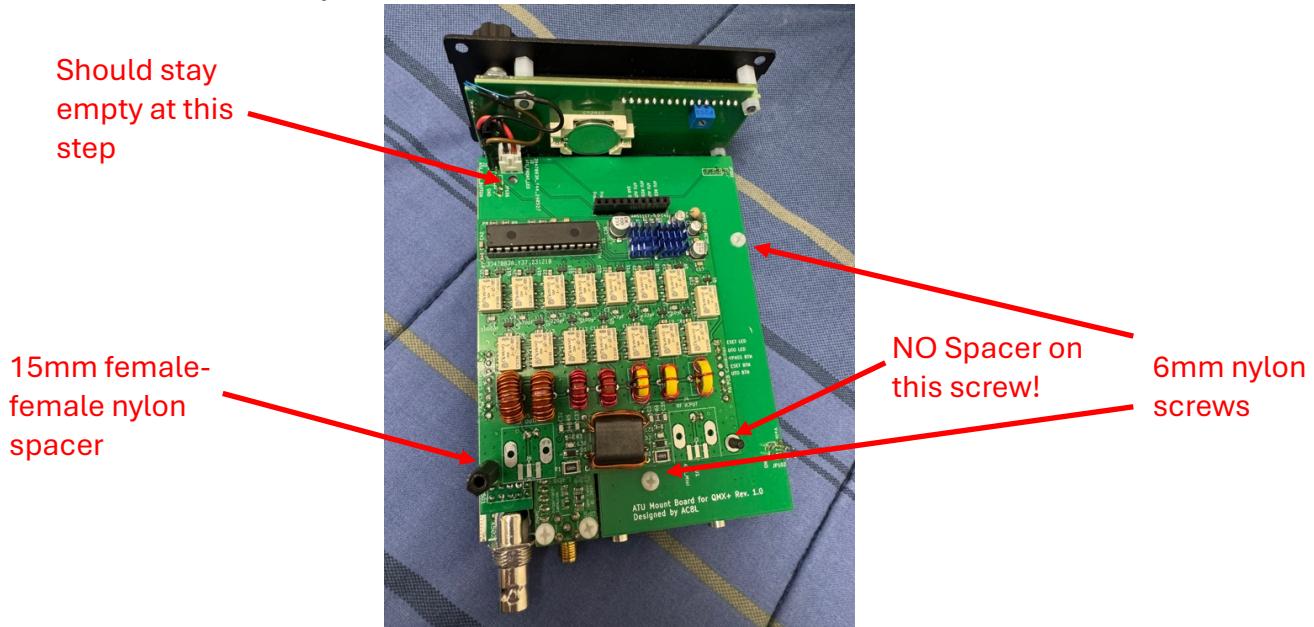
- Turn switch to the ON position.

- Measure the voltage between Vin and GND contacts of the JP102 again. Voltage must be equal to the voltage measured between (+) and (-) contacts of the BMS module.

**9. Turn switch to the OFF position!!!**

10. Attach QMX+ front panel into QMX+ main board.

11. Attach the ATU Mounting PCB to the top of the QMX+ main PCB and secure using two 6mm nylon screws:



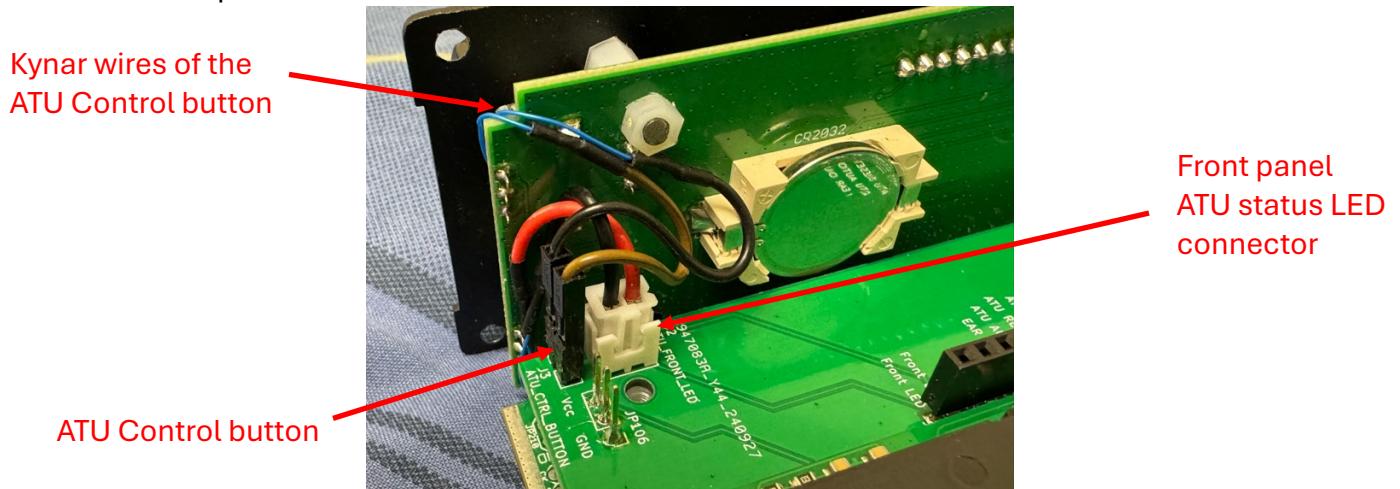
Do not attach anything to the mounting hole at the front left corner of the ATU Mounting PCB. After assembling the ATU Mounting PCB, there should be a 15mm female-female nylon spacer attached to the mounting hole on the ATU itself (lower right corner of the photo above). And there should be NOTHING attached on top of the nylon screw on the opposite side of the ATU. Make sure that male connectors JP106, JP501, JP302 and JP102 are properly inserted and secure within their female counterparts on the QMX+ main PCB.

12. Attach the JST connector of the front panel LED into its socket (J2) on the ATU Mounting PCB.

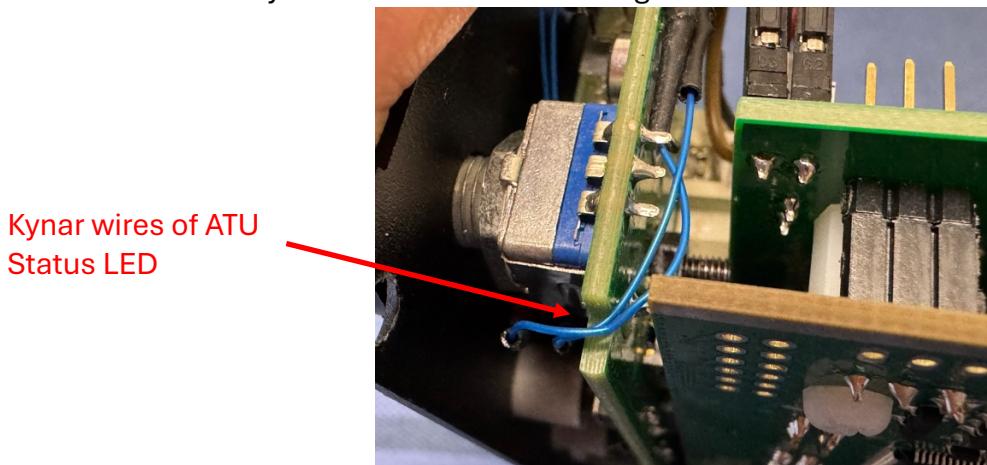
13. Insert the female pin connectors of the ATU control button into its connector (J3) on the ATU Mounting PCB. **If PCB front panel is used – make sure that wire coming from BTN1 soldering pad of the front panel is inserted to the male pin furthest from the Mount PCB front (closest to the JP106 connector):**



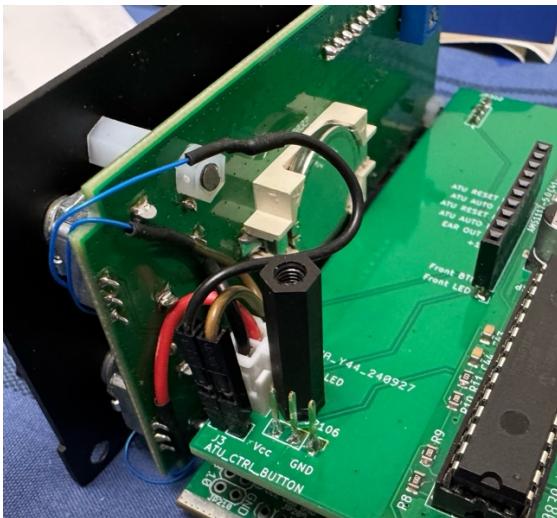
14. Route Kynar wires of the ATU control button through top left corner of the QMX+ front panel:



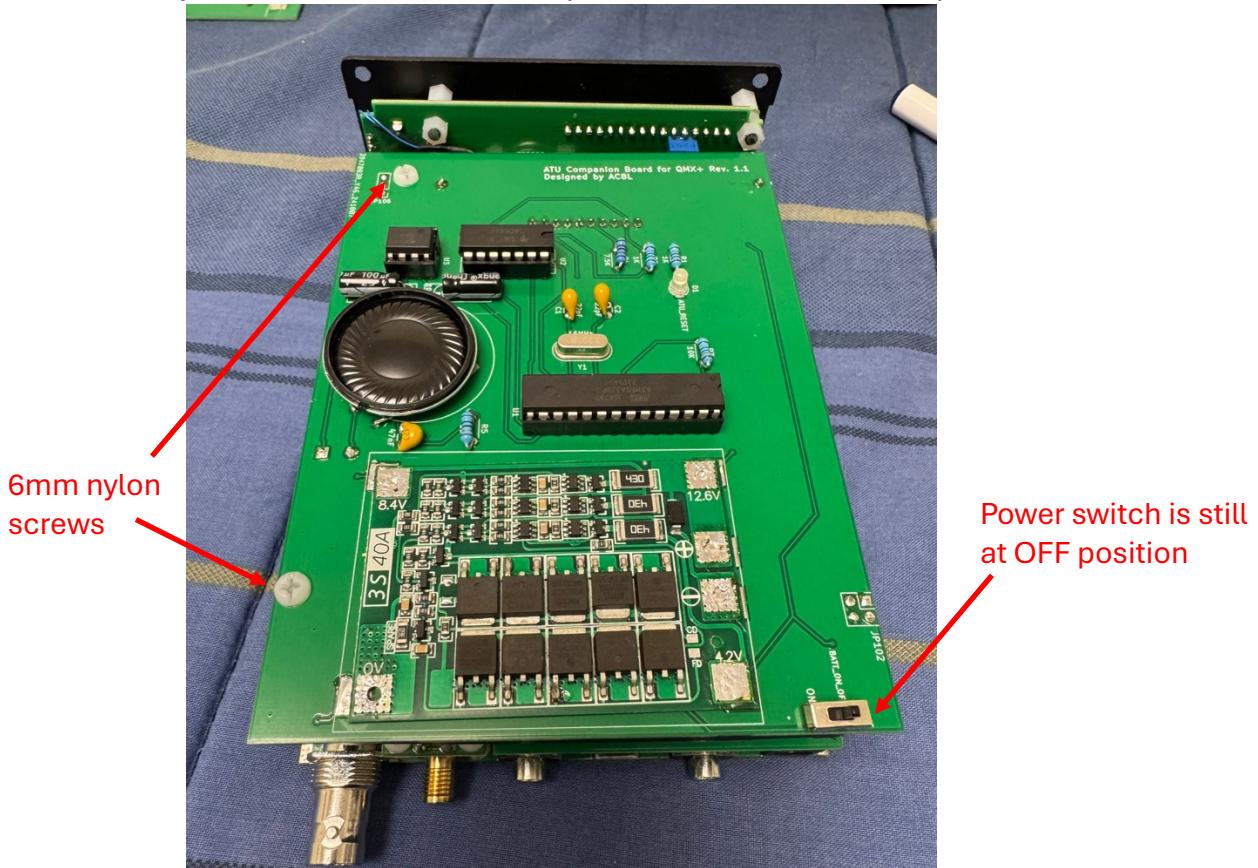
15. Route Kynar wires of the LED through bottom left corner of the QMX+ front panel:



16. Screw in 20mm male-female nylon spacer with nylon hex attached to its thread to the mounting hole next to the LED JST connector:

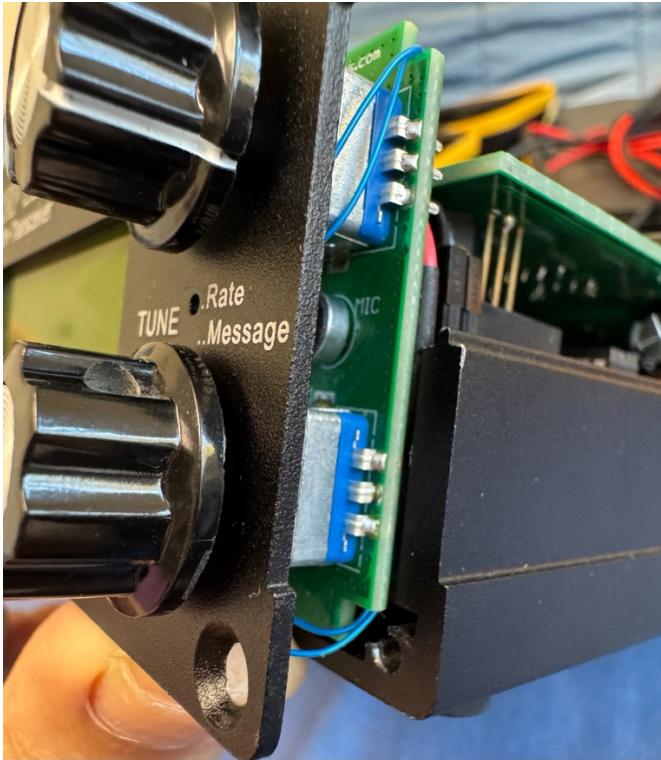


17. Attach ATU Companion PCB on top of the ATU Mounting PCB and secure using two 6mm nylon screws. Make sure that power switch is still in OFF position!



Make sure that connectors JP102, JP106 and J1 between ATU Companion PCB and ATU Mounting PCBs are properly inserted into each other.

18. Insert QMX+ assembly into the enclosure's lower half and secure by screwing in two lower front panel screws. While doing it – make sure that Kynar wires from the bottom corner of the front panel securely pass between front panel PCB and the edge of the enclosure. There should be a plenty of the clearance space:



19. Attach the back panel of the bottom half of the enclosure and secure using two lower screws.

20. Using a tip of a screwdriver move the slider of power switch on ATU Companion PCB to ON position.

21. Gently close the enclosure by its top and secure it using two front and two back screws. While doing this try not to touch front panel's control elements.

**\*\*\* Congratulations! We are done! \*\*\***

# 11. Operations

## Power On Behavior

1. When battery charge level is above the minimum threshold specified in power protection settings of the QMX+ (default is 7V), radio should turn on by pressing radio's power button.
2. If radio does not turn on, attach external 12V power supply and try to charge the battery pack above minimum protection threshold.
3. It is important to enable the numerical voltage display in QMX+ settings to monitor both – internal battery pack voltage and the voltage of an external power supply.
4. Usually with healthy batteries they should charge above the 7V level within less than an hour. After one hour you can test the battery behavior by:
  - Turn radio off
  - Disconnect external power supply
  - Turn radio on
5. After a successful test - continue charging battery pack to its full capacity of 12V for another hour or two. With 13.8V external power supply the battery pack will charge to the 12.4V capacity.
6. Because it is not recommended to power the QMX+ with higher than 12V, while charging with 13.8V the radio should be turned off.
7. With 12V power supply you can continue charging the battery pack while radio is ON. However, if battery is heavily discharged, the recommendation is not to do a TX while battery is charging, and radio is ON.
8. When ATU is present, and radio is being turned on - you should hear a click of ATU's relay within 1 sec after LCD screen is ON. This should be accompanied with a short 1 sec blink of ATU's status LED.
9. When speaker is present, depending on a volume level and band with antenna disconnected - you can hear a gentle hiss. You can adjust the volume level to observe the hiss behavior.

## ATU

- To enable the autotune – the SWR protection must be turned OFF with QMX+ firmware version 27. With version 23 it could work with SWR protection level 4.
- Double click – enable autotune
- Single click – disable ATU
- Long press – ATU reset
- The RF power higher than 2.9 Watt is enough to trigger the ATU tuning.
- The ATU remembers the autotune settings in EEPROM between power cycles.

## Battery

- Enable voltage display from QMX+ settings. The LCD will display current battery voltage.
- When external power supply is plugged – it will charge the battery and QMX+ display will show the voltage of an external power supply.
- One of interesting side effects of how battery pack is connected to the QMX+ - the external barrel connector is ALWAYS HOT. This could be used either for powering a small flashlight from QMX+ itself, or even for emergency charging of a cell phone using an adapter like this: <https://a.co/d/1OmckWj>



You will need to attach a barrel connector to it.

**But nothing that draws more current, like a bigger tablet, computer, PA, etc.**

## Internal speaker

- Just use volume level, internal speaker has enough volume for the sound to come out of the enclosure. No drilling was necessary.