

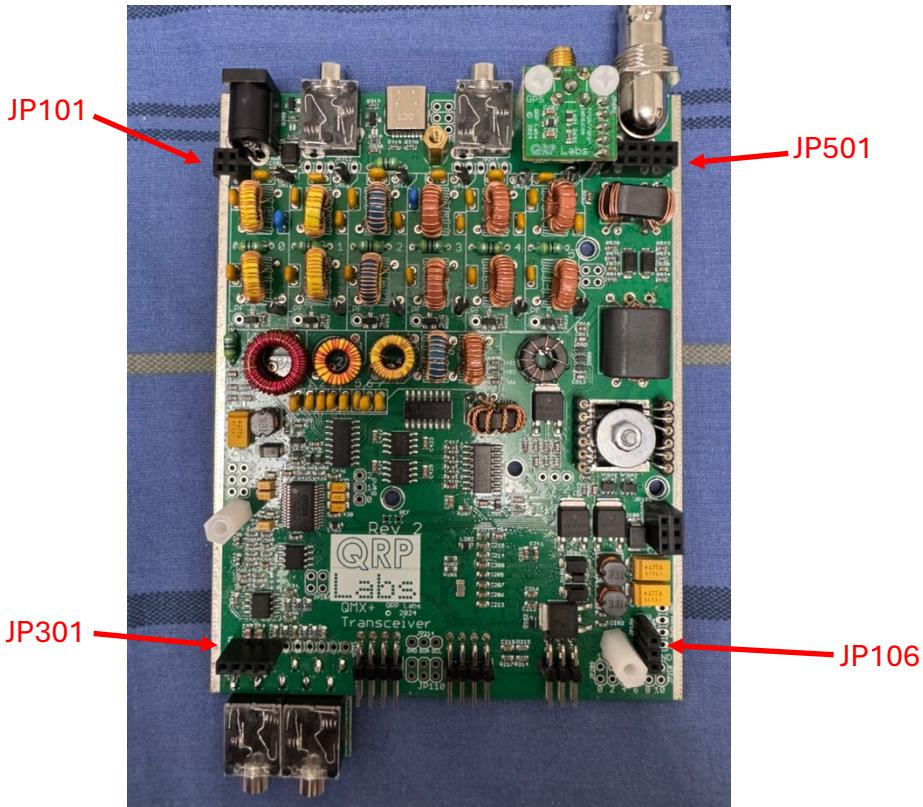
QMX+ ATU, Battery, Charger and speaker solution

Build instructions

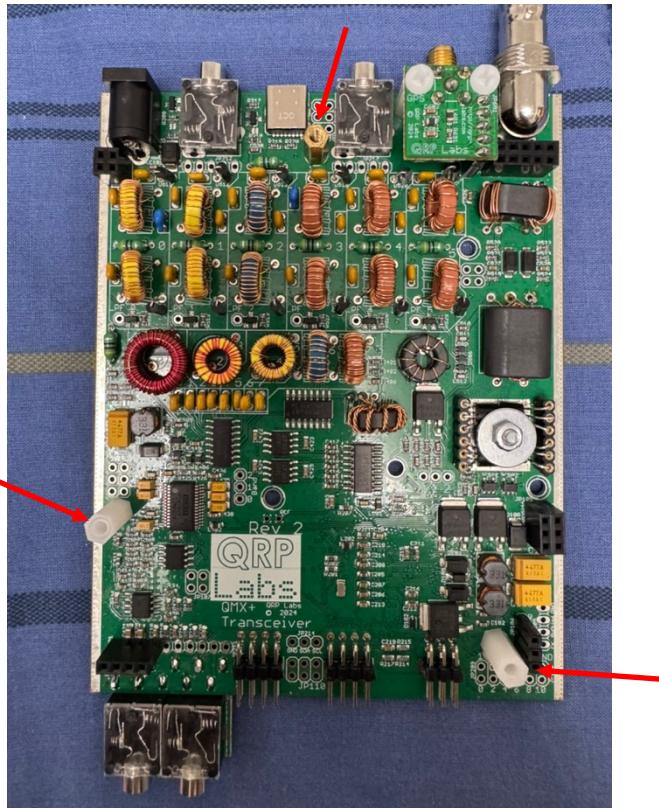
Ver 1.0 (DRAFT)

By AC8L

1. Install connectors to the QMX+ main board

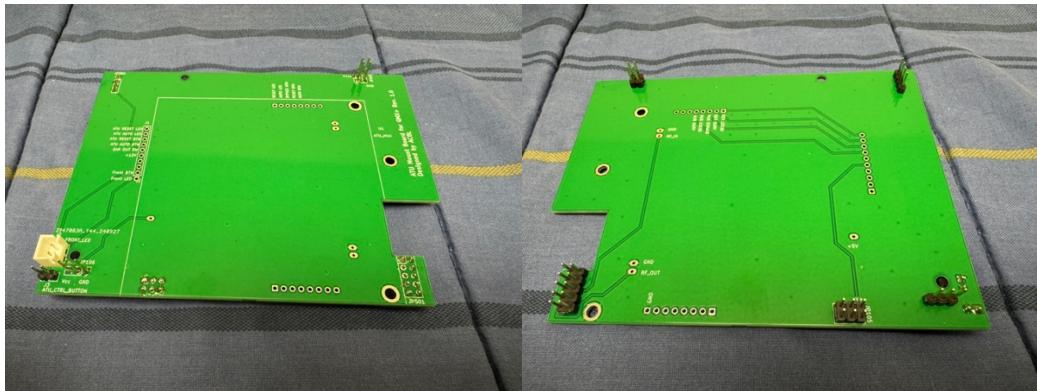


2. Install three 11mm nylon standoff spacers

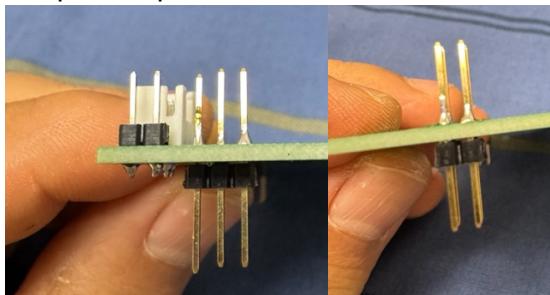


Use 6mm nylon screws to secure spacers from the bottom of the QMX+ main board.

3. Install pin headers to ATU Mounting PCB



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



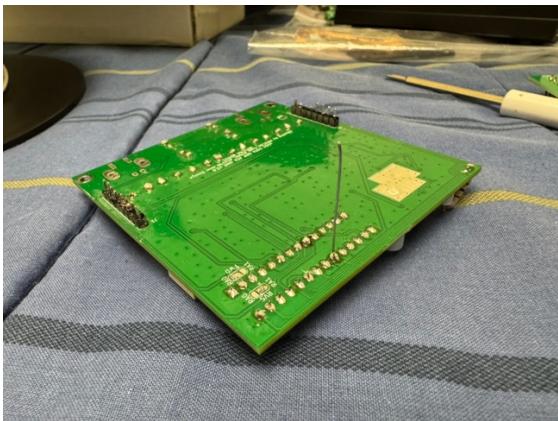
- Only J2 (front-panel LED) and J3 (front-panel button) connectors are soldered on the front of the PCB. All others – on the back.
- At this point we DO NOT attach J1 connector!

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+.

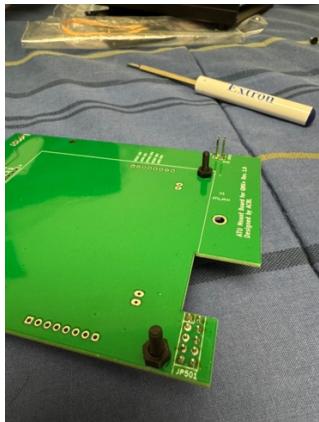
4. Prepare ATU to be soldered to the ATU Mount PCB

At this point ATU has to be fully assembled with all the toroids and DIP socket installed on it.



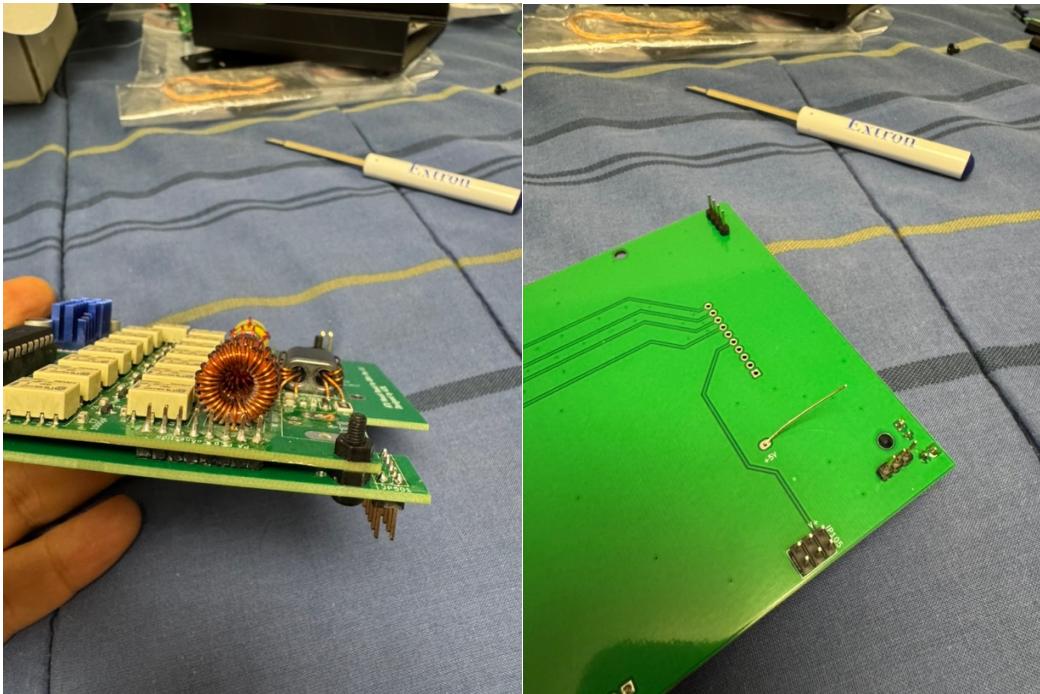
Solder both 8-pin male pin headers to the ATU from the bottom (see photo above). Also solder one of the left-over through-hole component pins to the PIN 20 of the pc microcontroller. On the photo above it is a pin 6 from above on the row close to the PCB edge. Make sure the soldered PIN is as vertical as possible.

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

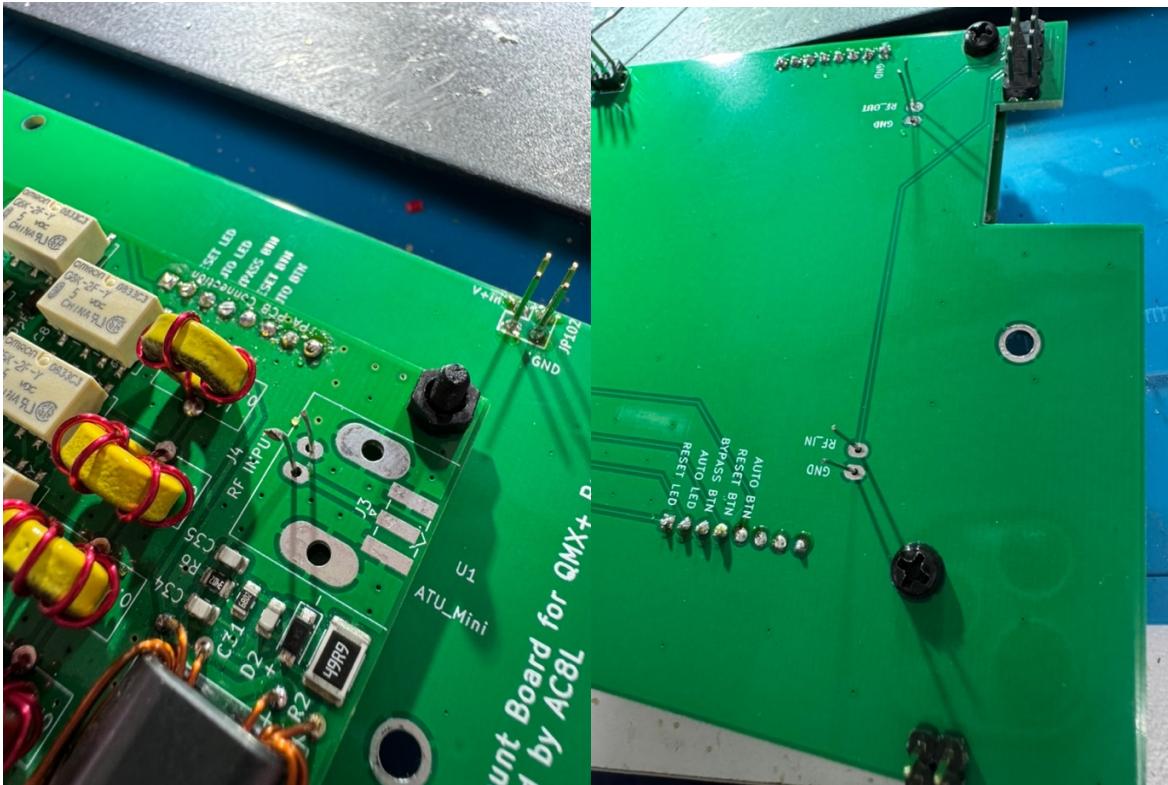


Attach two 12mm nylon screws from the bottom of the ATU mount PCB and secure them with hex nylon nuts from the top.

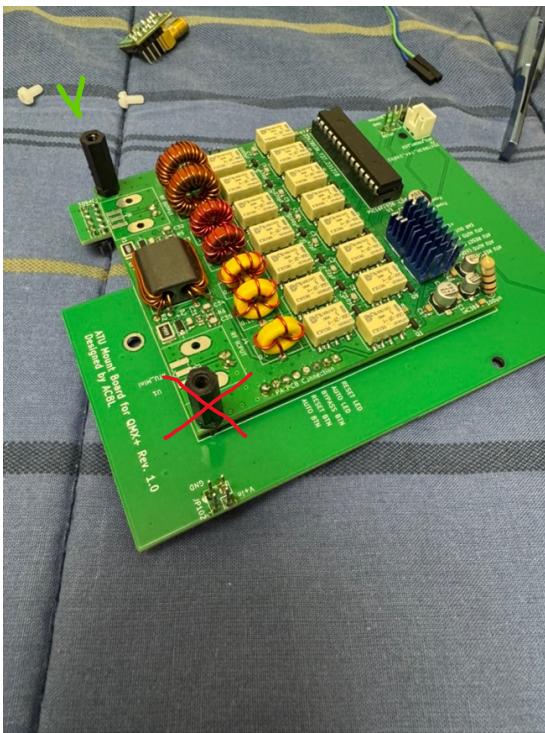
6. Attach ATU to the ATU Mount PCB



- 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 “+5V”.
- Secure the ATU to the ATU mounting PCB using 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), not interdependent, loosely coupled to each other 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:

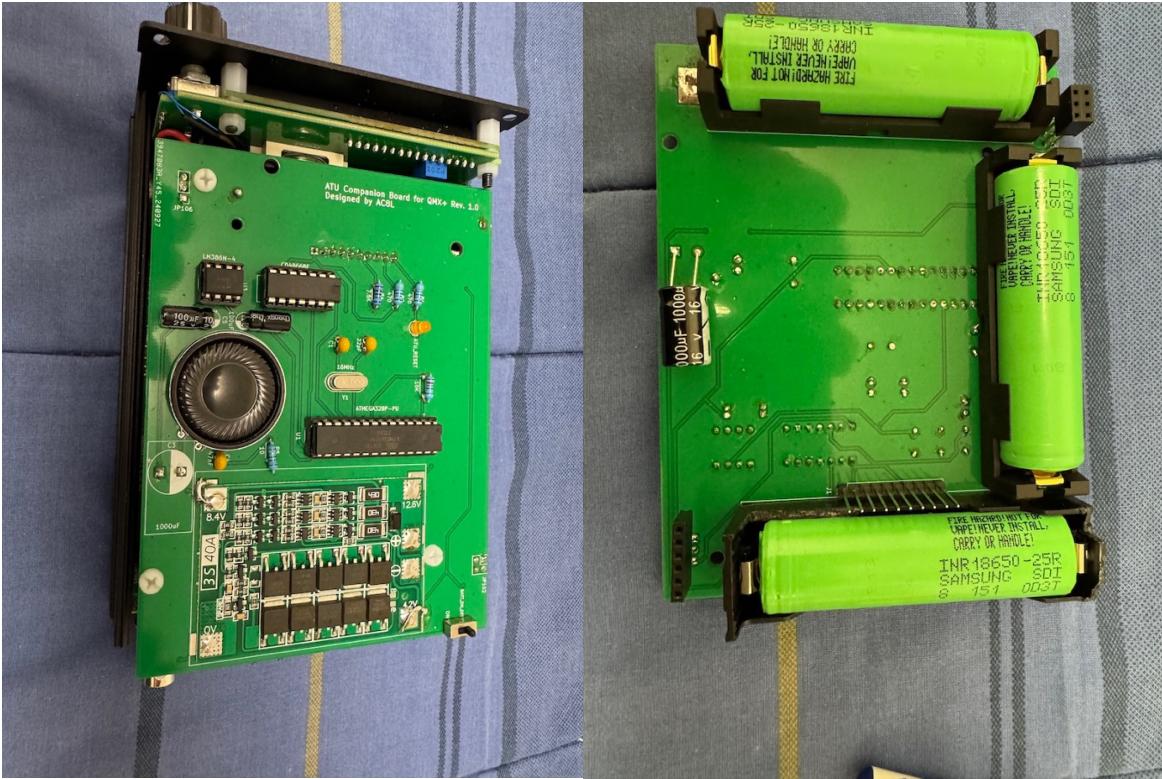
- All inclusive - ATU, battery, speaker.
- ATU only
- Battery only
- Speaker only
- ATU and battery
- ATU and speaker
- Battery and speaker

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

QMX+ Mod	Components
ATU	R1,R2,R3,C1,C2,D1,Y1,U1,U2,JP106
Battery + charger	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 risk 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.

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.

