T41-EP SDT v12 Main Board Assembly Manual

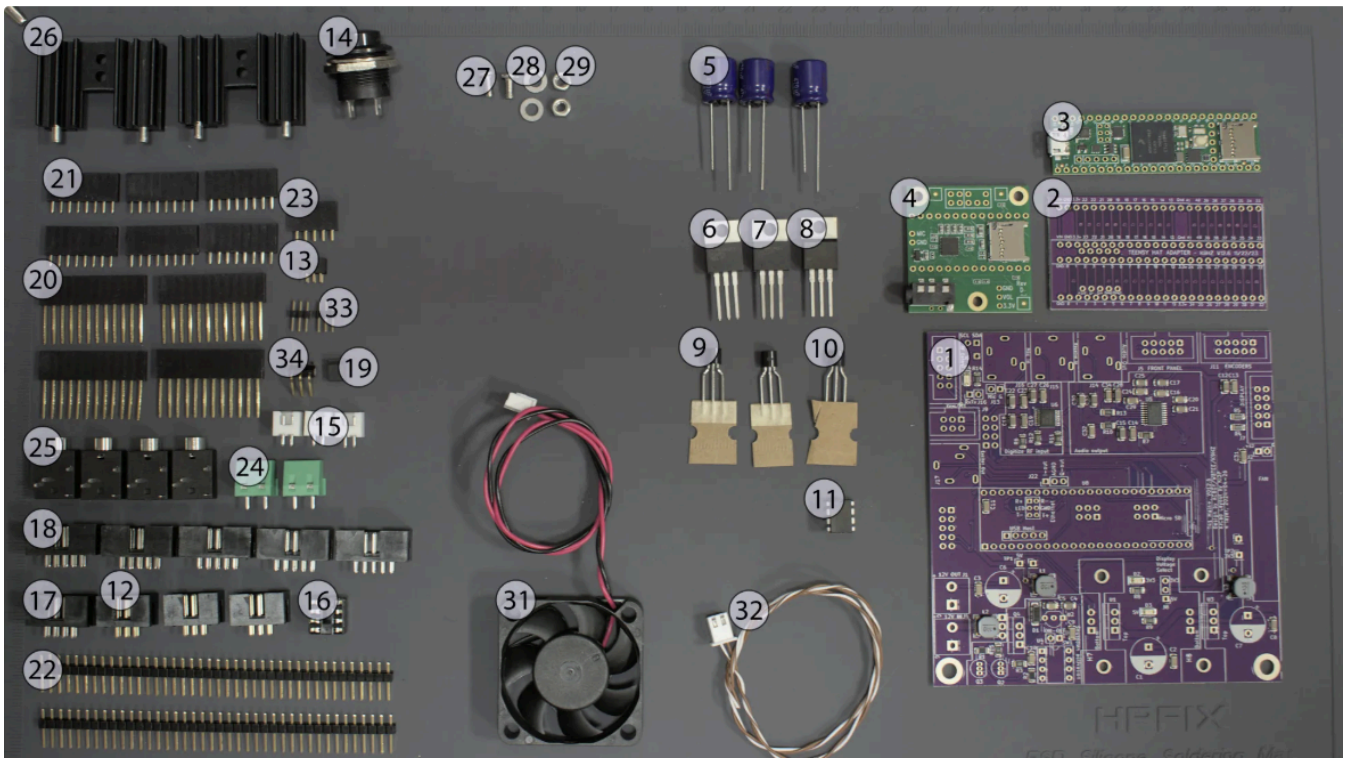


Figure 1. Parts included in Main Board Kit
by Justin Giorgi (AI6YM) and Dr. Jack Purdum (W8TEE)

The T41 V12 Main Board is the center of it all. A Teensy 4.1 provides the central processing, memory, and flash for the T41 SDT. On-board 192kHz ADCs and DACs digitize and un-digitize RF and audio signals. All of the other modules connect through this board to be controlled by the T41 software.

What's Included (Parts List)

1. (1) T41 V12 Main Board
2. (1) Audio Hat Extender Board
3. (1) Preprogrammed Teensy 4.1 MCU
4. (1) Teensy 4 Audio Hat
5. (3) 470uF Electrolytic Capacitors
6. (1) LM7805 5V Linear Voltage Regulator
7. (1) LM117T-3.3 3.3V Linear Voltage Regulator
8. (1) SUP90P06 Mosfet
9. (2) 2N7000 Mosfets

10. (1) AP7381 5V Linear Voltage Regulator
11. (1) ATTiny85 MCU
12. (3) 2x3 IDC Box Headers
13. (2) 2x3 2mm Female Headers (one pictured)
14. (1) SPST Momentary Pushbutton Switch
15. (3) XH 2pin Sockets
16. (1) DIP-8 Socket
17. (1) 2x4 IDC Box Header
18. (5) 2x5 IDC Box Headers
19. (1) 2.54mm Jumper
20. (4) 12pin 2.54mm Stackable Headers
21. (6) 8pin 2.54mm Female Headers
22. (3) 40pin 2.54mm Male Pin Headers (two pictured)
23. (1) 5pin 2.54mm Female Header
24. (2) 2pin Molex Connector
25. (4) 1/8" Stereo Audio Jacks
26. (2) TO-220 Heatsinks
27. (2) 6mm M3 Stainless Steel Screws
28. (2) M3 Stainless Steel Washers
29. (2) M3 Stainless Steel Nuts
30. (1) 0.5g Thermal Paste Packet (not pictured)
31. (1) 40mm 12V Fan w/pre-terminated power leads
32. (1) Pre-terminated power switch cable
33. (2) 3pin 2mm Male Pin Headers
34. (1) 3pin 2.54mm right-angle Male Pin Header

Missing a part? Send an email to justin@ai6ym.radio.

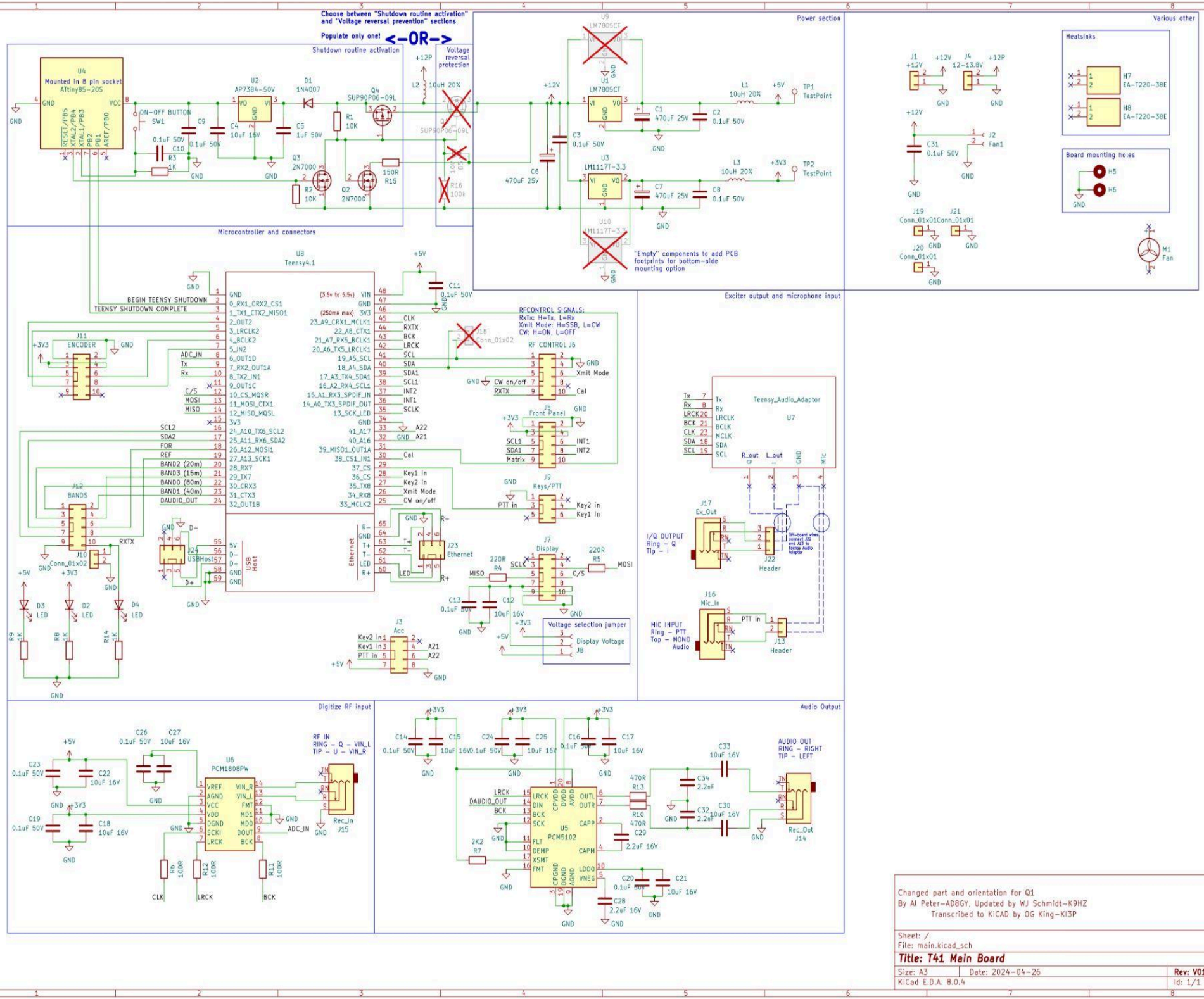
You Will Need

1. Soldering Station, hot air or separate heat gun optional but recommended.
2. Multimeter
3. PCB Vise or Helping Hands
4. Jewelers Loupe
5. Wire Cutters
5. Sandpaper
6. Solder of your preference, 60/40 tin/lead is recommended.
7. Flux Paste
8. Polyimide (Kapton) Tape
9. Solder Wick

Safety Matters!

Soldering irons are hot.
Everything they touch gets hot.
Have a fire extinguisher nearby!

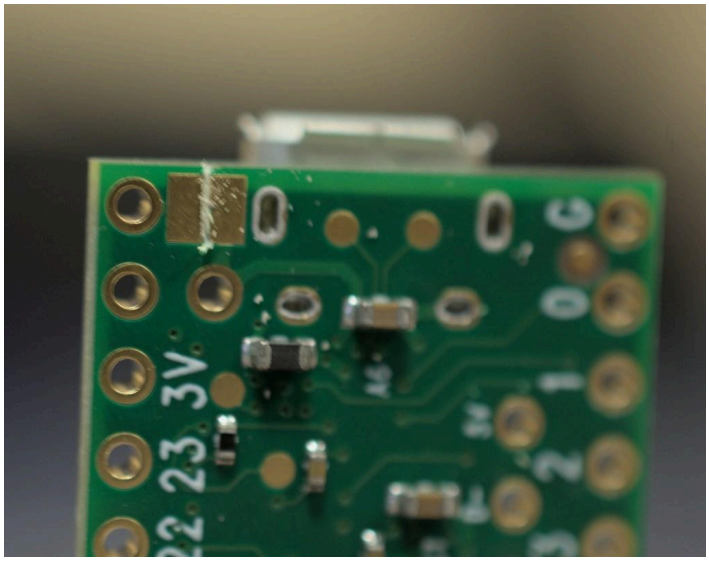
Solder splatters. Your eyes are not easily replaceable. Wear your PPE!


Choose between "Shutdown routine activation" and "Voltage reversal prevention" sections
Populate only one!
<-OR->
Shutdown routine activation
Voltage reversal protection
Power section
Various other
Heatsinks
Board mounting holes
Microcontroller and connectors
Exciter output and microphone input
Digitize RF input
Audio Output
"Empty" components to add PCB footprints for bottom-side mounting option
Changed part and orientation for Q1
By Al Peter-AD8GY, Updated by WJ Schmidt-K9HZ
Transcribed to KiCAD by OG King-KI3P
Sheet: /
File: main.kicad_sch
Title: T41 Main Board
Size: A3
Date: 2024-04-26
Rev: V012.6
KiCad E.D.A. 8.0.4
Id: 1/1

Schematic

Figure 2. Main board schematic

On The Teensy

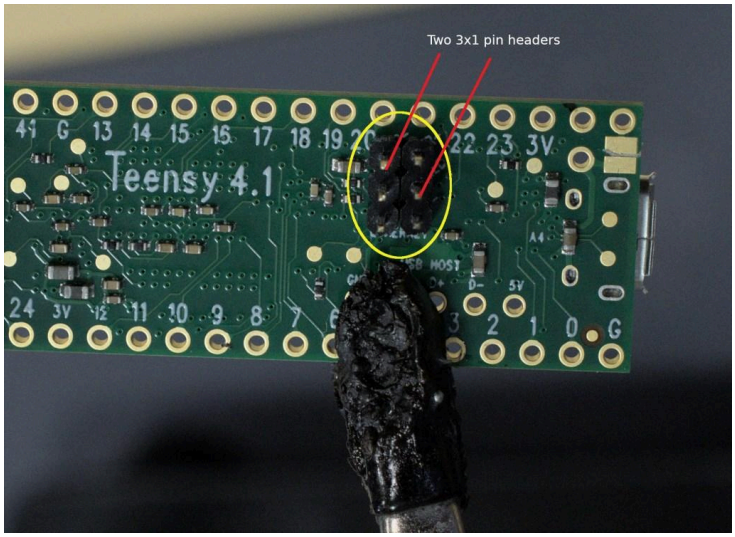


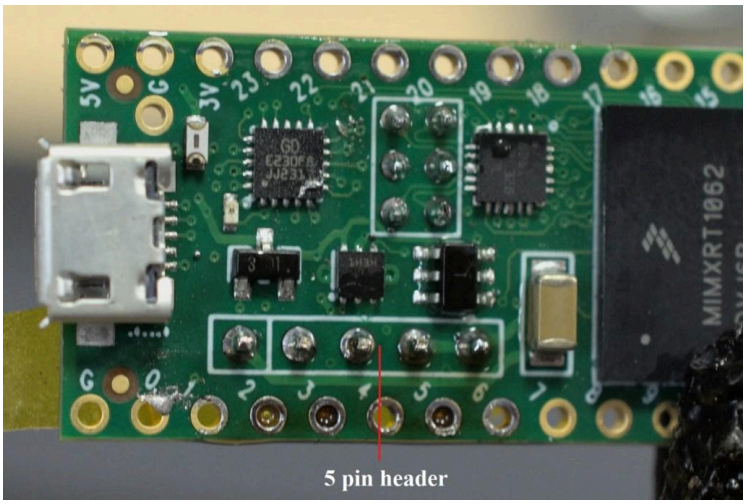
Cut the power trace that connects VIN and VUSB with a sharp hobby knife, utility knife, or razor blade. The cut is the light colored vertical line in the upper right of the board in the photo above. This prevents potential problems when the board is powered on and the Teensy is connected to USB. With an ohmmeter check that there is no conductivity between the two pads.

Note: You will need to bridge the two pads with solder again to program over USB or supply power on VIN during programming (e.g. by leaving the Teensy on the Main board).

Install the two 3-pin rows of 2mm pins (#33 in parts list) on the Teensy, these make up the ethernet header. This position will be very difficult to solder after installing other components.

Also, keep in mind that the pins for the headers are on the Teensy and their mating sockets are on the Main board.





Break off 5 pins from one of the 40pin 2.54mm pin headers and place them in the USBHost position on the Teensy. Note that this view is taken from the top side of the Teensy, which means the long pins are on the underside of the Teensy.

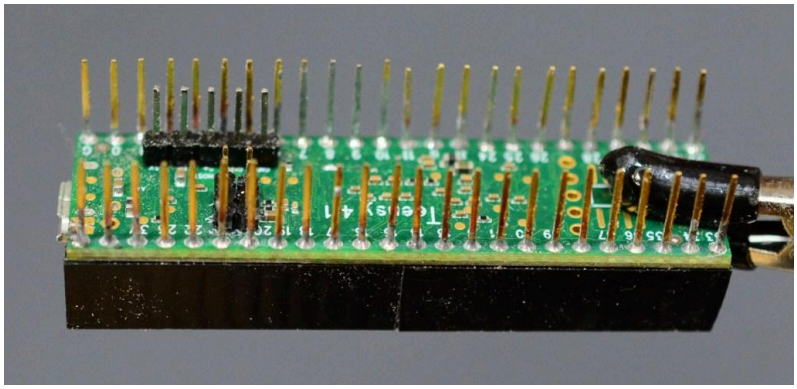
As always, secure the part with tape and start by soldering one pin, check the alignment of the header, solder the opposite pin, double check alignment, and then solder the remaining pins.

Note: The extender board has a position for these pins so a stackable header could be used here but it's entirely unnecessary. The main board exposes the USB host connection via an IDC box header on the bottom of the main board.

Next prepare the 12pin stackable headers. Stackable headers are easily identified by their pins which are about twice as long as standard header pins.

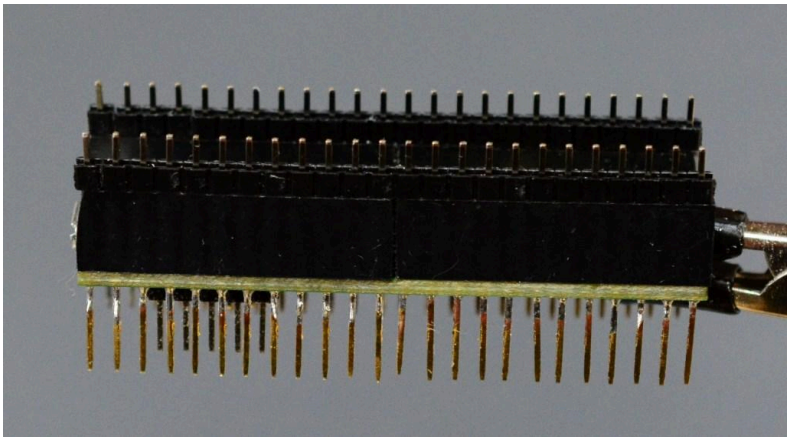
These headers are slightly too wide at each end to fit neatly next to each other. Lightly sand the edges of the headers where they will touch. Laying a bit of sandpaper or a flat-blade file on a table and rubbing the end of the header against it. Be careful not to sand all the way down to the pin or it will fall out and you will need to start again with a new header. Without this step, the headers will not align properly without excessive force. Forcing the headers into position will cause the Teensy board to bow.

Place the first header on the Teensy and secure it with tape. Solder one pin from the back of the board and check the header's alignment. Solder a second pin, double check alignment, and then solder the remaining pins. Repeat for each of the four headers.



Note: Some builders find pin alignment easier if they use a continuous 24 pin header rather than multiple headers lined up together.

The Extender Board & Audio Hat



Break off two rows of 24pin headers for the audio extender board. Press-fit these pins into the headers on the Teensy and place the audio extender board on top.

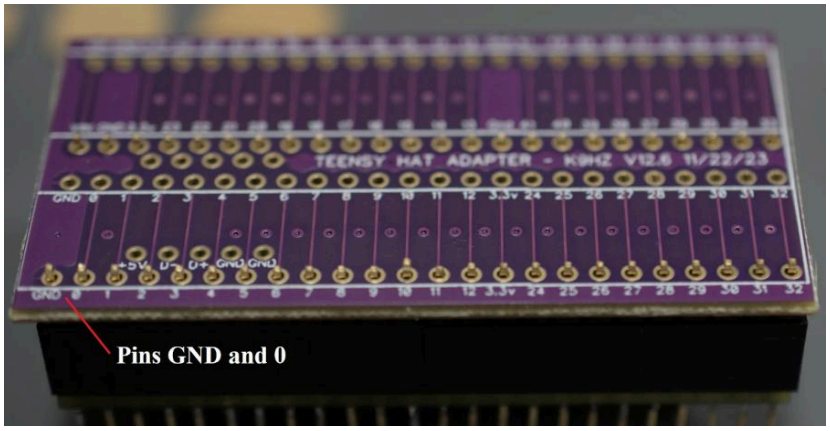
If necessary multiple shorter rows may be used but a single continuous row will help to align the stackable headers.

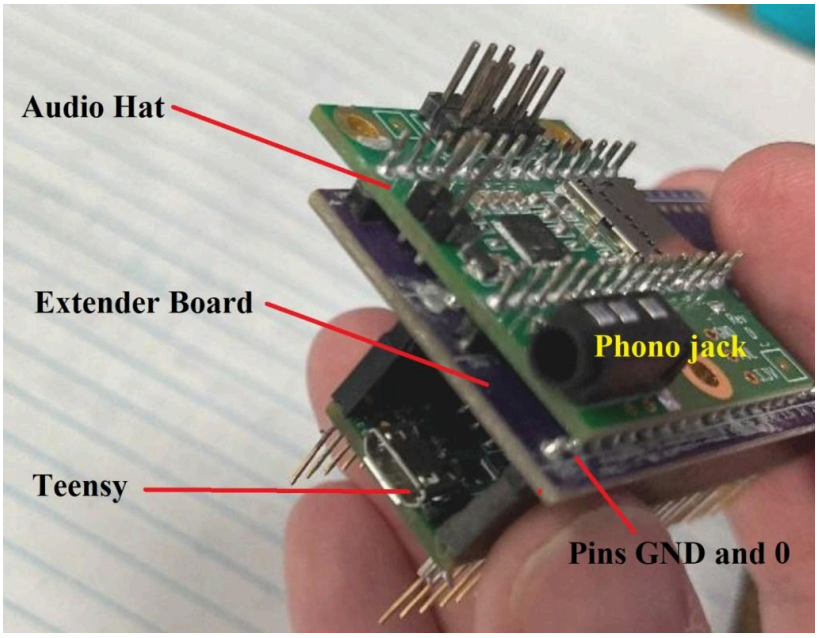
Fitting the parts together this way before soldering ensures good alignment of the pins and headers.

Now place the audio extender board on top of the pins. Notice that the first two pins in the lower-left corner are marked *GND* and *0*. This means the extender board "hangs over" the top edge of the Teensy.

Note: The extender board has four rows of 1x24 pin holes. It is important that you position the pins from the Teensy in the correct rows. Take a look at the pictures below to confirm you are placing the pins in the correct rows.

Solder each of the 1x24 pin strips to the audio extender board.



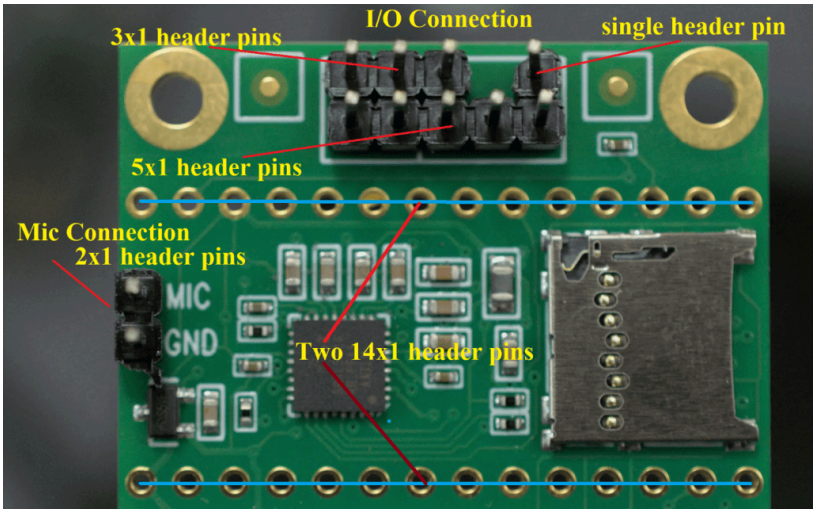


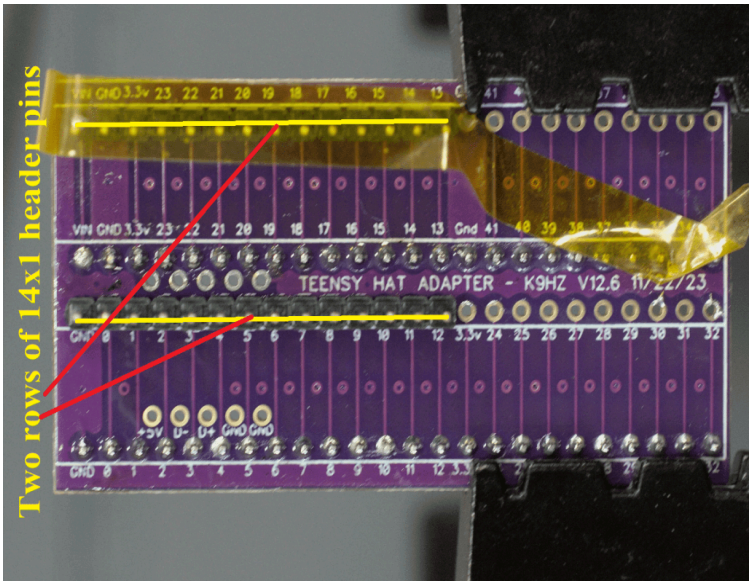
You can see the overhang better in this photo which shows the completed stack. Note the position of *GND* and *0* pins.

Remove the extender board from the Teensy and set both aside.

Break off additional pins to populate the IO section (top edge) and the mic connection (left edge) of the audio hat. One at a time secure your rows of pins with tape, solder a single pin, check alignment, and then solder the remaining pins.

Pro-Tip: You can use a single row of 5 pins for the top row of the I/O Connection. Just remove the fourth pin where the space is empty on the board.

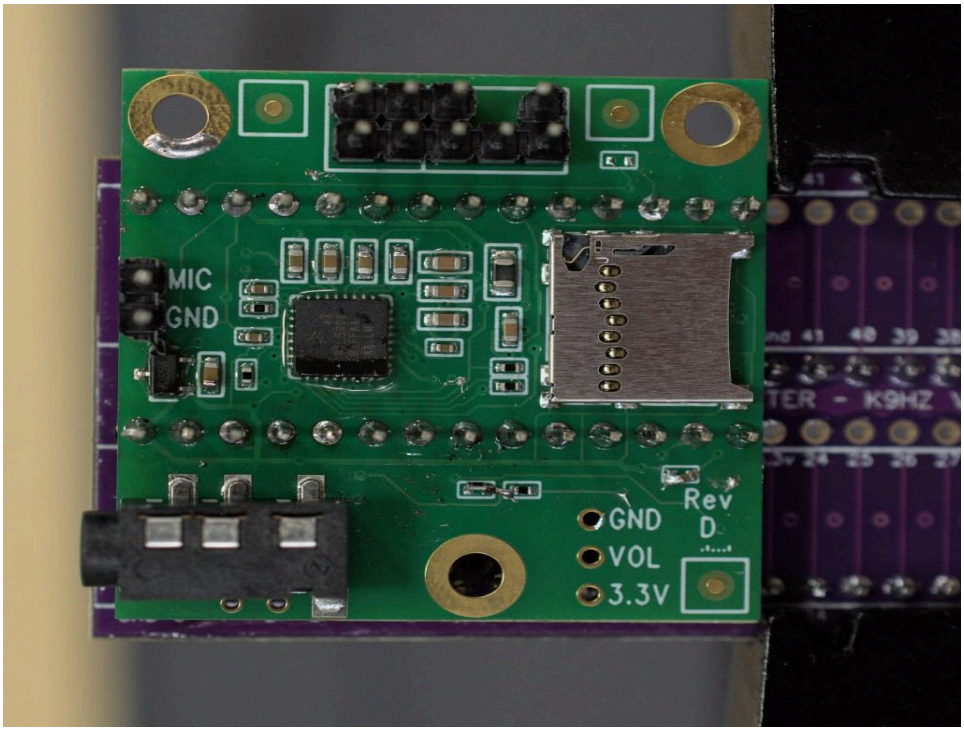




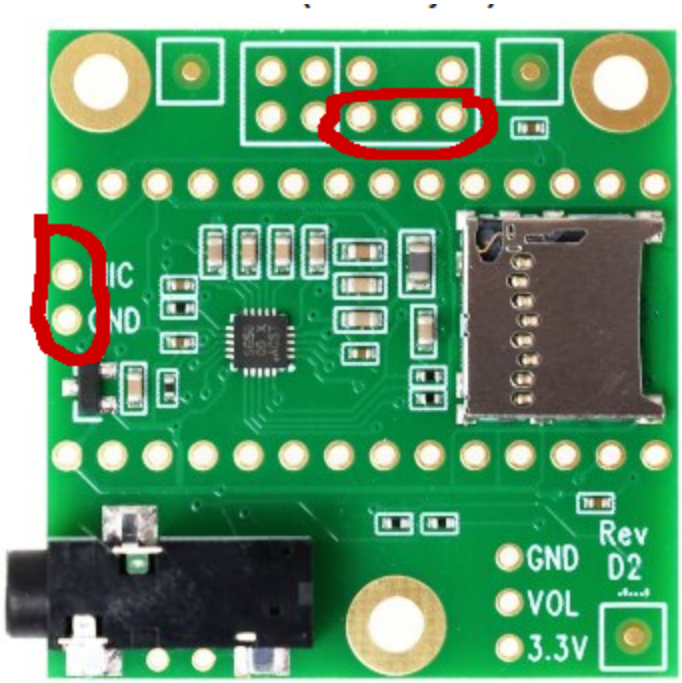
Break off two rows of 14pins for the connection between the audio hat and the offset board. Solder these rows onto the extender board with the pins facing upward (away from the extender board and Teensy). You can tell the proper orientation in the figure because the plastic header holding the pins is visible.

Fit the audio hat onto the 14pin rows on the Extender board. The blue lines in Figure 10 shows where the two rows of pins added in Figure 11 fit into the Audio Hat board. The phono jack on the audio hat should be aligned with the edge of the Extender board, as seen in Figure 9.

Solder the pins to the audio hat. As always check alignment after soldering a single pin so that you can correct any misalignment easily.

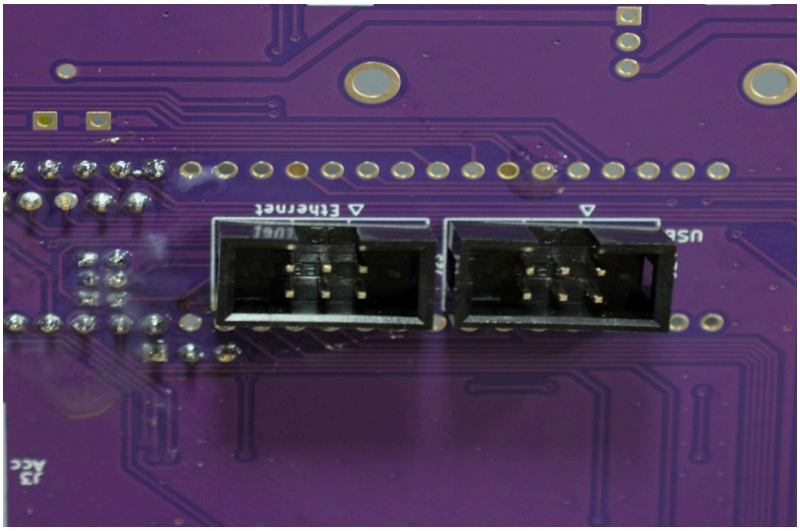


Clean the boards and stack the audio hat + extender board combination on top of the Teeny as shown in Figure 9. The audio hat will be above the side of the Teensy with the micro-USB port.



Now install the jumper wires that go between the audio hat to the Teensy. There are three right angle pins above the top row of Teensy headers on the main board. Those three pins go to the circled three pins on the audio hat shown above. Ground is in the middle of both so it's just a straight shot (no twists in the wires). These are the I/Q out signals for transmitting SSB. Below the mic jack on the main board are two pins, one is the mic and one is ground. They go to the two circled pins on the audio hat to handle the microphone input.

Teensy Headers on the Main Board



The box headers shown in Figure 13 are 3x2 IDC box headers and are soldered on the underside of the Main PCB. on the back, under the Teensy. These will be difficult to access once headers for the Teensy are soldered in place.

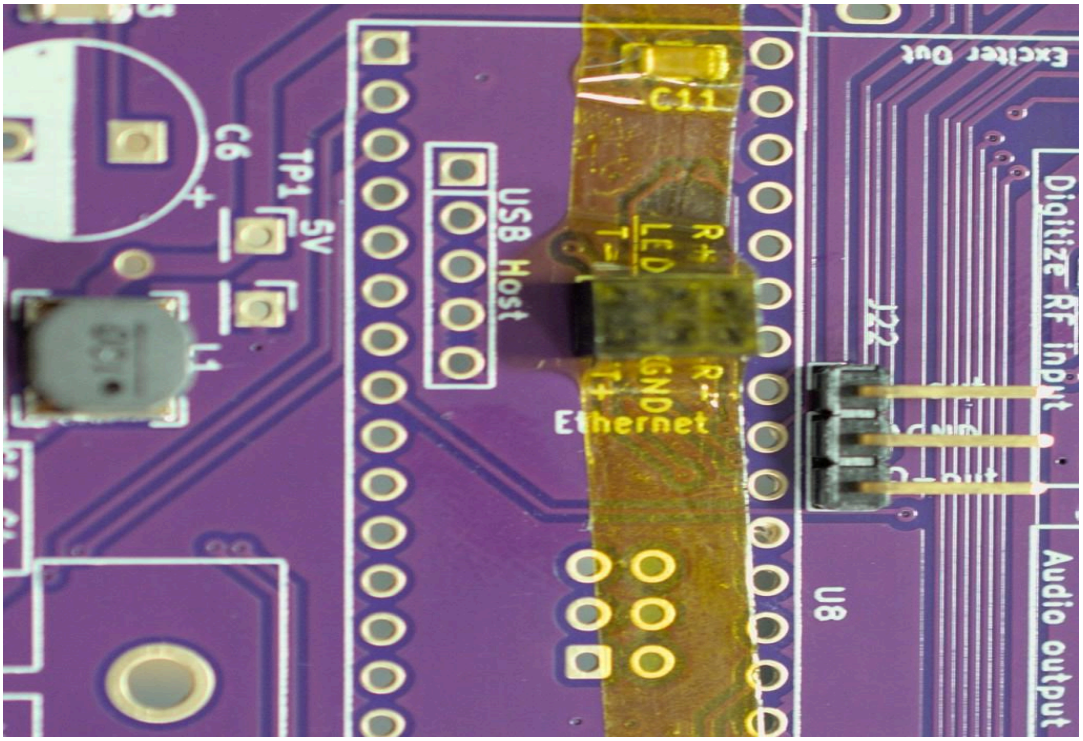
One at a time secure each header with tape, solder a single pin and check that the header sits flush with the PCB. Also make sure you have positioned it so the key on the socket matches the silkscreen. Now solder the opposite pin, check alignment, and then finish soldering the remaining pins. Repeat for the second header.

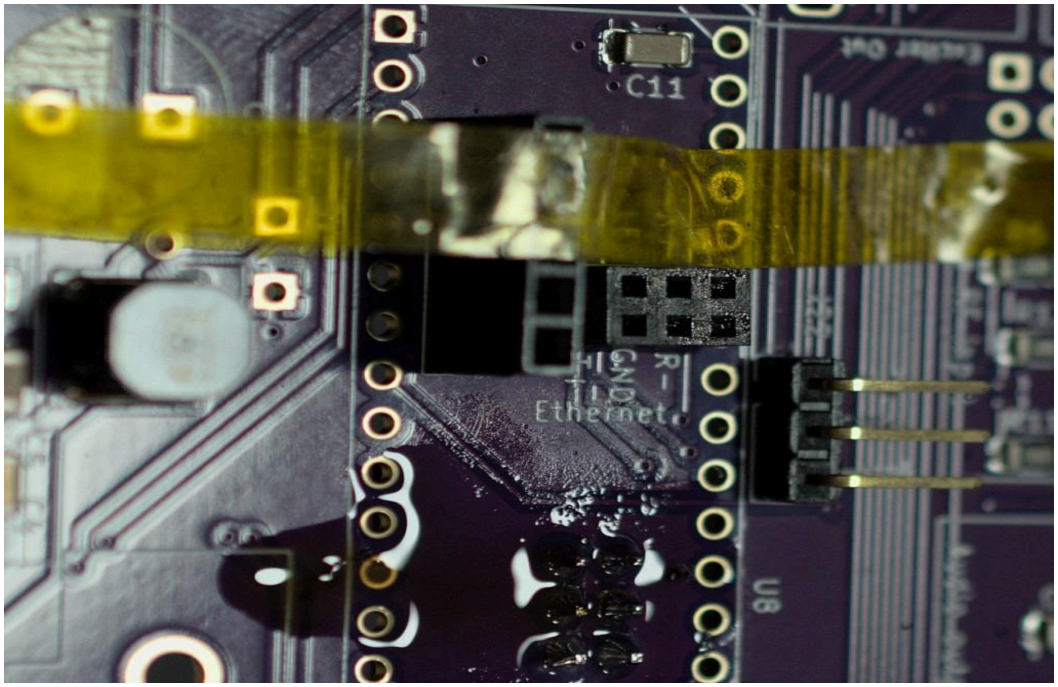
Note: These photos may show parts assembled in a different order, the assembly order in this manual is far easier than the order in which the test boards were assembled.

Place the 3pin right angle pin header at *J22*, secure it with tape, solder it in place.

Place the 2x3 2mm socket for the Teensy's ethernet connection on the front of the board and secure it with tape. From the back side, solder a single pin, check alignment, solder a second pin, check again, and then solder the rest of the pins.

Note: the second 2x3 2mm socket will be inserted into the one solder to the main board in order to bring the combined height to the level of the other headers



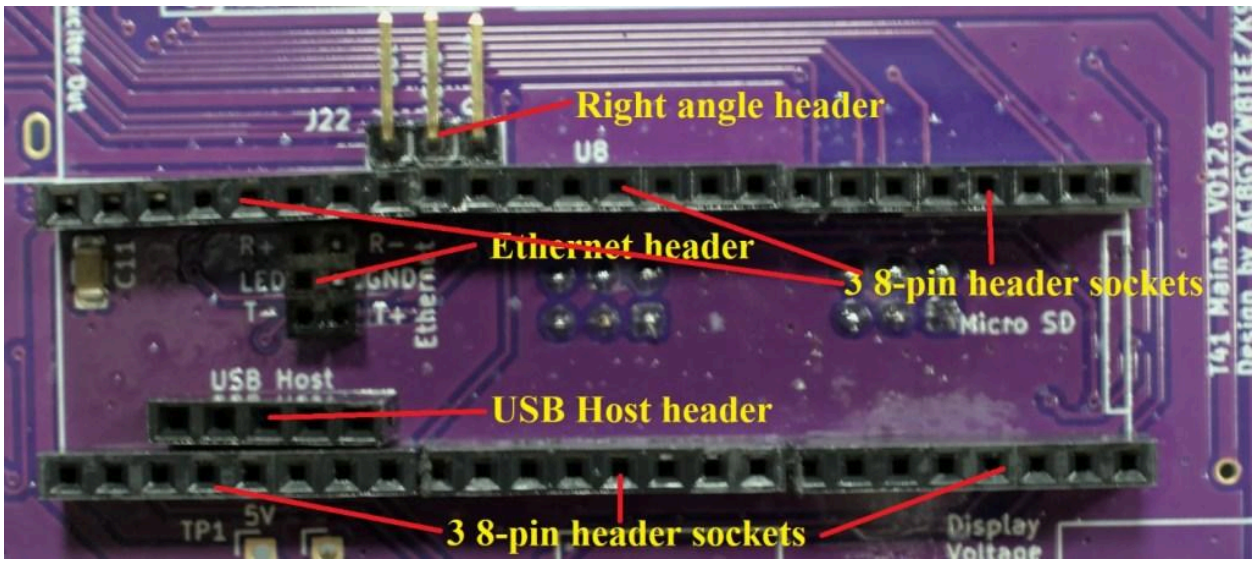


Place the 5 pin female header on the USB Host position (topside of Main PCB). Solder a single pin, check alignment, solder another pin, double check alignment, then solder the remaining pins.

Finally, place the Teensy headers. Six 8-pin 2.54mm female headers are provided which make up two rows of 24pin headers on the main board. These six header sockets hold the Teensy on the Main board. (These are in place in Figure 16.) Like the stackable headers on the Teensy the edges of these headers must be sanded lightly so that they fit next to each other without significant force (which may bow or damage the board).

After sanding (or filing), place each header one at a time, secure with tape, solder the pins from the back checking alignment. Make sure they are flush with the Main PCB. Repeat for each of the six headers. Slight imperfections in alignment are acceptable.

Note: Some builders find pin alignment easier if they use a continuous 24 pin header rather than multiple headers lined up together.



Pro-Tip: If you have a continuous strip of header pins handy, place each 8-pin header in its mounting holes and then fit the continuous strip of 24 pins into the 3 headers. This long strip of pins helps to align the 24 holes. This will help make it easier to mount the Teensy in these sockets later on. It is very easy to fold a pin under the Teensy when mounting it and it is almost zero fun locating such a mistake.

Use your loupe, or magnifying glass, to inspect your joints before continuing. *All of them!* There are a lot of solder joints making up these headers, it's easy to miss a couple of cold or incomplete joints. It may be very difficult to debug any bad connections to the Teensy.

Testpoints, Jumpers & Small Connectors

Break off pins as needed from the 2.54mm pin header strips for testpoints *TP1* and *TP2*, Display voltage Select (*J8)*, and Mic G (*J13)* on the front side of the board. Repeat for *J18* on the back side of the board. The testpoints (*TP*) and *J18* are optional. The testpoints adjacent to J6, TP1 and TP2 are optional ground connections (shown as J19, J20, and J21 on the schematic, but unlabeled on the Main PCB). The most accessible

of these is the one adjacent to TP2. This is a good place to install a ground pin for connecting test equipment.

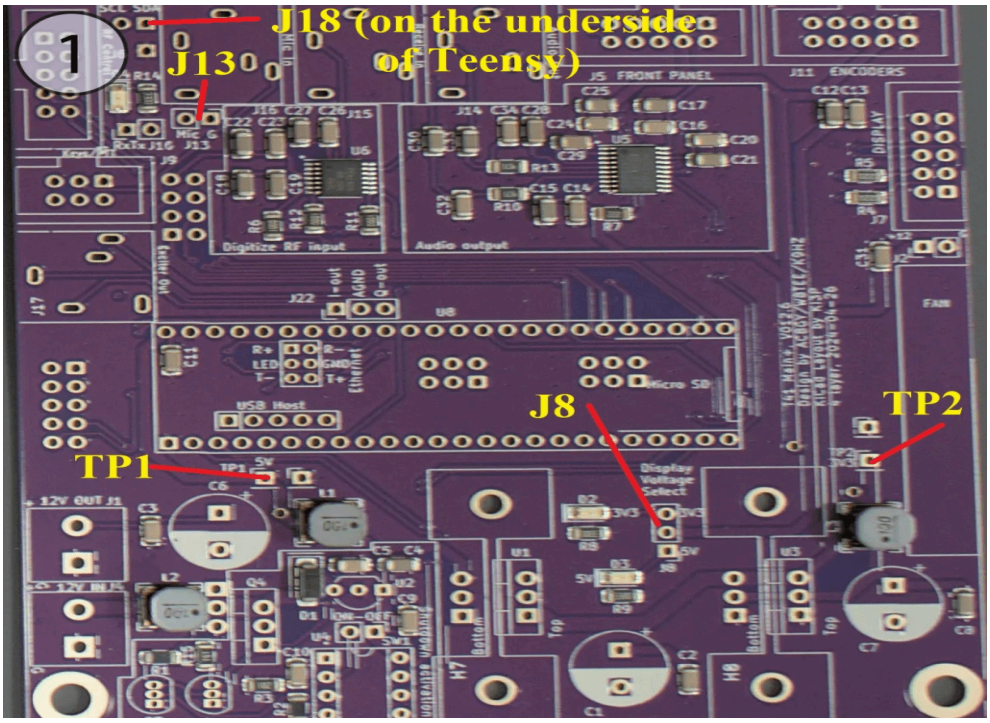
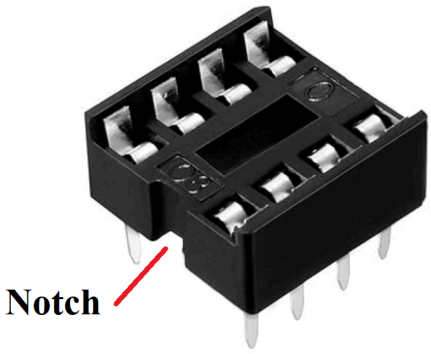


Figure 17. Test points and other pins.

For each of these sets of pins secure the pin row to the board with tape and solder from the opposite side of the board as usual.

Note: Once again these photos will often show parts assembled in a different order than is recommended in this manual. Following the order in the manual is strongly recommended!



Place the DIP (8 pin) socket on the front of the board at U4. Make sure to align the notch, or cutout called the “key”, in the top of the socket with the marking on the silkscreen (the notch faces toward the center of the board).

This is important so that, when it comes time to insert the ATTiny chip into this socket, you position it correctly. Inserting it incorrectly will not do you, the ATTiny, or the T41 any good.

Secure the socket with tape, solder one pin on the back, check alignment, solder the opposite pin, double check alignment, and then solder the remaining pins.

Place the XH sockets for PWR SW, TX/RX, and Fan. These sockets have a notch, also called the “key”, on one side. The connectors used with these sockets have a matching key that fits into the notch so it is important that they are oriented on the PCB correctly.

The socket (#1, Rx/Tx) in the upper-left edge of the board has its key facing the top edge of the PCB. On the center-right edge of the board, socket #2 (fan socket) has its key towards the top edge of the board, away from the fan. The last socket (#3, PWR) has its key facing the bottom-edge of the board, towards the ATTiny socket. The proper placement of the socket keys ensures the correct polarity (if any) is observed

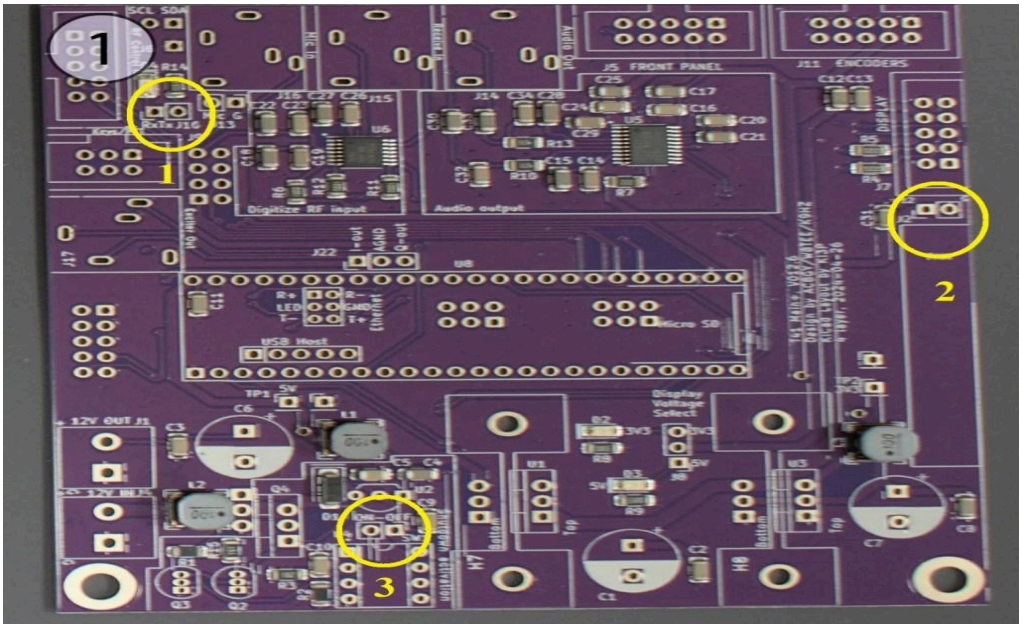
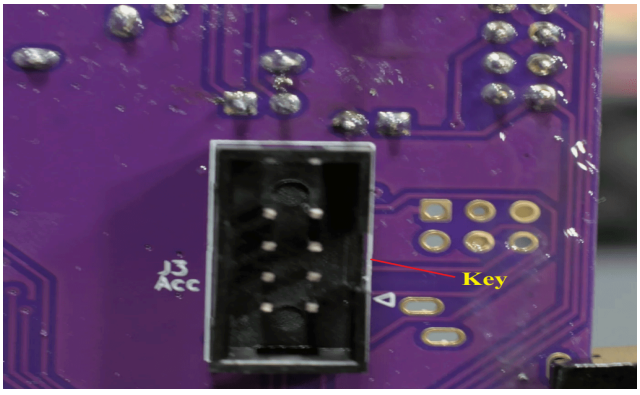


Figure 19. Placement of XH sockets.

One at a time place the socket, secure with tape, solder one pin, check alignment and fit to the board, and then solder the second pin.



Place the *J3 ACC* socket (a 2×4 IDC box header) on the underside of the board at its silkscreen position. Make sure the key is facing the edge of the PCB as shown by the silkscreen. Secure with tape and solder as usual from the front side of the board.

Repeat for the *J12 Bands* socket, a 2×5 IDC box header which is also placed on the back side of the board. Again, pay attention to the position of the key.

Capacitors & Semiconductors

Place the first electrolytic capacitor on the board, they are all the same value. The gray (negative) strip on the capacitor case aligns with the white (half-moon shaped) silkscreen mark on the board. Secure the capacitor by bending the leads away from each other on the underside of the board. Check that the cap sits flush to the top of the PCB. Because there are no “taller” components on the board, a gentle downward pressure with the soldering iron while soldering will assure it’s flush with the board. Trim excess length from the leads of the capacitor. Repeat for the remaining two capacitors.

Place the AP7381 voltage regulator (above the DIP socket) making sure the flat side of the case (it has a TO92 footprint and looks like a transistor) is oriented to match the outline on the board’s silkscreen. This is slightly above XH position 3 in Figure 19. Secure the part with tape and solder one pin from the back, check / fix alignment before soldering the remaining pins. Trim excess length from the leads of the component.

Repeat the process above for the two 2N7000 MOSFETs in positions Q2 and Q3. These components are in a standard TO-92 package so the leads are VERY close together. Inspect these joints carefully to ensure no solder bridges have been created.

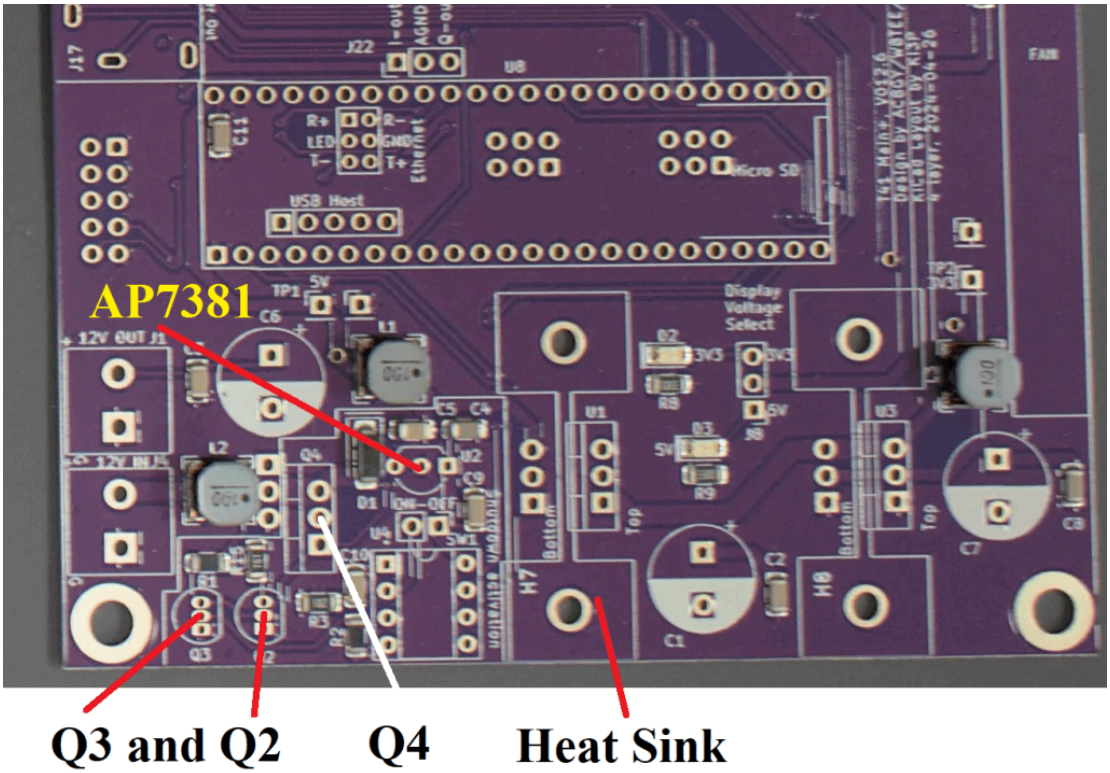


Figure 21. Transistors and heat sink placement.

Place the SUP90P06 MOSFET in position Q4, align the tab (located on the back of the part) with the outline on the silkscreen, secure with tape, solder one pin from the back, check alignment, and then solder the remaining pins.

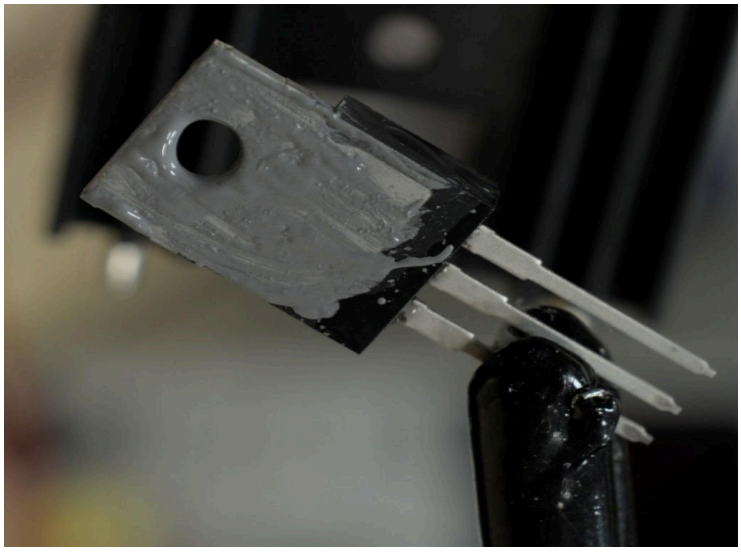
Note: Some builders felt that installing the heatsinks and their associated semiconductors would be easier if it were done after installing all of the connectors. There were no issues with this build using the described order, but you may wish to consider it.

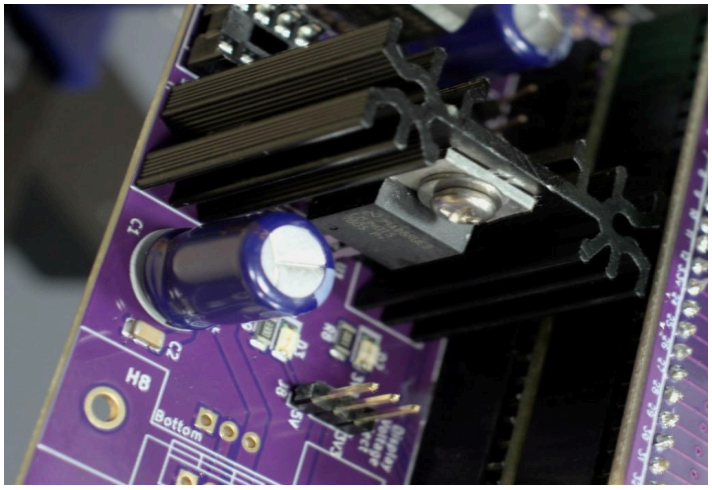
Place one TO-220 heatsink in the position behind U1. The larger flat area of the heatsink should face U1. DO NOT INSTALL U1 YET! Secure the heatsink with tape and solder both mounting pins to the board from the back side. This may require extra heat from a heat gun and a good bit of flux but these pins will eventually solder.

Note: Soldering the heatsinks ensures they are stable, doing this before installing the component on the heatsink reduces the risk of damaging the component with excessive heat.

Place a small drop of thermal paste on the back of the LM7805 voltage regulator and spread the paste evenly. Place this part on the heatsink at position *U1*. Secure the voltage regulator to the heatsink with the provided M3 bolt, washer, and nut. Tighten the bolt to secure the part and ensure solid contact with the heatsink.

Note: There are two mounting holes on the heat sink. If you secure the regulator using the top hole, it places more surface area in contact with the heat sink. It probably doesn’t make much difference, but the top hole is a little easier to use, too.





Solder the pins from the back side of the board. The center pin is connected to the tab and cooled by the heatsink, this pin may require extra heat and flux to solder successfully.

Repeat this process for the LM1117T-3.3 in position U3.

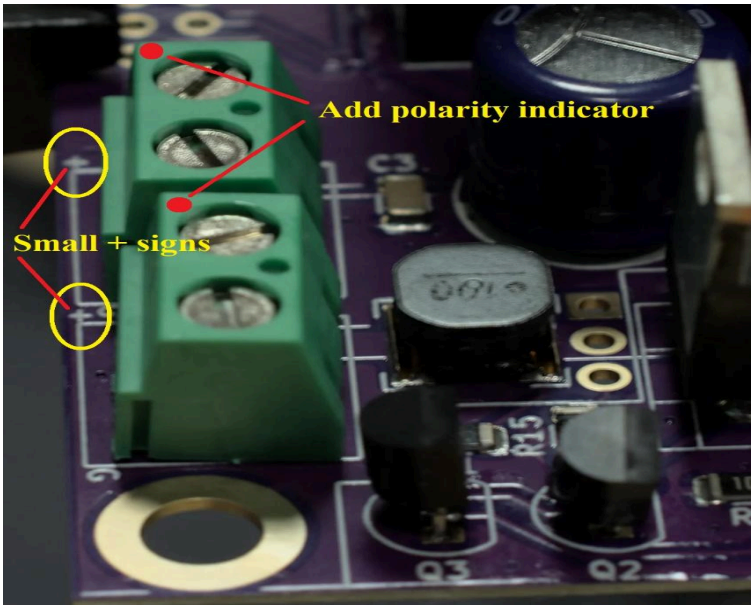
Connectors on the Main Board

Complete the main board by installing the remaining connectors on the board's edges.

Two 2 pin Molex blocks are provided for the 12V power input and 12V power output (to the other boards). For each of these, place the block with the wire terminals facing out, secure with a bit of tape, solder one pin on the back side of the board, check alignment, and then solder the remaining pin. The openings for the wires should face the left edge of the PCB.

The polarity for the wires feeding into the Molex connectors is indicated by a very small '+' on the PCB. If you have a marker pen (red is best), you might want to put a red dot in the top of the connector.

Pro-tip: Use a permanent marker to add a polarity indicator to the Molex blocks. If you've never made a polarity error, feel free to ignore this tip.



For each IDC box header, place the header on the board, secure it with a bit of tape, solder one pin on the back side of the board, check alignment of the header, solder a pin opposite of the first, check alignment again, and then solder the remaining pins.

For each phone jack, place the jack on the front side of the board, secure it with a bit of tape, solder one pin on the back side of the board, check alignment of the part, solder a second pin, double check alignment, and then solder the remaining pins.

Finishing Up

Before continuing to test your assembled board it should be cleaned. Use isopropanol (isopropyl alcohol), at least 70% concentration, with a soft brush to remove flux and solder residue. It's perfectly safe to soak the board as long as it is dried thoroughly before power is applied. Higher concentrations of isopropanol evaporate more quickly and so drying times are reduced. I frequently leave my boards overnight to dry but a reflow oven or plate set to a VERY LOW temperature can speed the process.

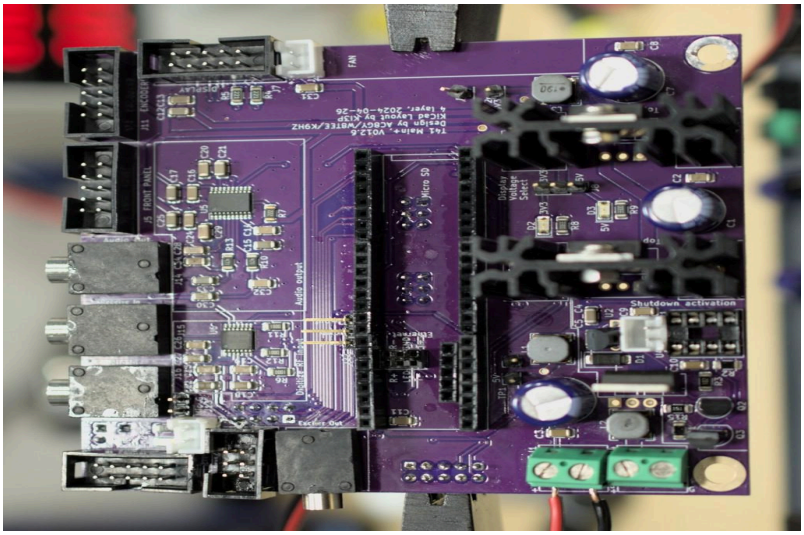
After cleaning the board inspect all of your solder joints. Use a jeweler's loupe to get a good view. Touch up any joints that are cold or incomplete. It may be necessary to

clean, dry, and inspect your boards again. That's fine. Make sure the joints are good before continuing.

Once you have a clean, dry, and thoroughly inspected board covered with high-quality solder joints, place the voltage selection jumper for the display. If you've purchased a display kit from me the display provided requires 5V. Do not install the Teensy or ATTiny yet!

Testing

Bare Board Power Up Test



Start with your main board and no installed components. Don't install the Teensy or the ATTiny MCU. Don't connect any other boards or the fan.

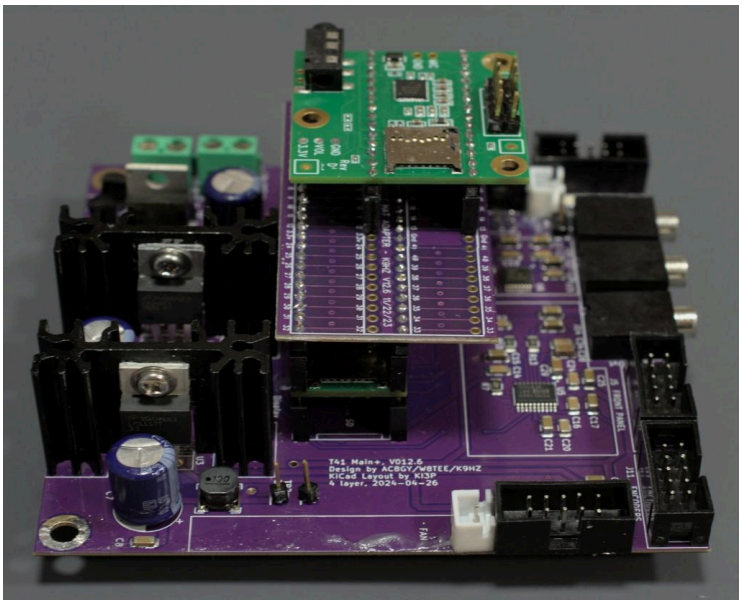
Set your bench power supply to 12V and set the current limit to ZERO. Connect power to your main board on the *12V OUT* connection, this bypasses the soft shutdown system and essentially powers the board directly.

SLOWLY increase the current limit on the supply. The board should power up (two LEDs will light) and the supply voltage should be reached with a current draw of 150mA

or less. If your board goes over 160mA before reaching the full voltage there is something wrong.

Check for regulated voltages of 5V on TP1 and 3.3V on TP2 (+/- 0.2 is fine).

Testing the ATTiny Power Control



After the bare board power up test, turn off power to the board and set the current limit of your power supply back to zero.

Install the ATTiny MCU in the DIP socket. Align the cutout in the top of the MCU with the cutout in the socket.

Warning: Installing the ATTiny MCU upside down will cause a short circuit and likely destroy the device! If powered on the device will become VERY hot, do not attempt to remove it without first disconnecting power and allowing the MCU to cool.

Connect the on/off switch to the XH socket above the DIP socket using the provided cable (the kit includes a pre-terminated cable for this purpose; see item #32 in the parts photo above). If the brass connectors do not easily slide onto the switch pins, gently file the edges to give the pins a little taper.

Connect a voltmeter to the 12V out connector. Connect power to the main board, this time to the 12V IN connection, and SLOWLY increase the current limit on your power supply. The target voltage should be reached with very little current load because the ATTiny starts up with the radio in the OFF condition. Any draw over 10mA is suspicious.

Check the voltage at the power output block, the voltage should be zero, since the radio is off (a few millivolts is fine).

Press and release the power button. The radio should start up in a couple of seconds with a load of about 30mA (more than 50mA is suspicious). Check the voltage at the output block; the voltage should be within 0.2V of the input voltage.

If you want to test the shutdown function now (optional), carefully connect one side of a 1K resistor to + 5 V by inserting one side of the resistor into a hook-up wire with double female connectors. Press the other end of the hookup wire onto TP1 (+5V). Push and release the power button to put the ATTiny MCU into its shutdown state. Now very carefully touch the other end of the 1K resistor to pin 7 of the ATTiny MCU. The LEDs should go off and the output voltage should drop to zero.

Turn off power to the board and set the current limit of your power supply back to zero.

Testing with the Teensy Installed

Place the second 2x3 2mm female header on top of the one soldered to the main board (the Ethernet connector). **Without this there will be no contact between the ethernet connector and the Teensy.** Install the Teensy (do not install the audio hat yet), being very careful when pressing the Teensy into the headers and ethernet connector on the main board. It is very easy to bend misaligned pins, especially the ethernet and USB host headers in the middle of the Teensy board. Indeed, it is easy to fold a pin under the Teensy and not even see it.

Connect a voltmeter to the 12V out connector. Connect power to the main board, this time to the 12V IN connection, and SLOWLY increase the current limit on your power

supply. The target voltage should be reached with very little current load because the ATTiny starts up with the radio in the OFF condition. Any draw over 10mA is suspicious.

Check the voltage at the power output block, the voltage should be zero, since the radio is off (a few millivolts is fine).

Press and release the power button. The radio should start up in a couple of seconds with a load of about 170mA (more than 200mA is suspicious). Check the voltage at the output block; the voltage should be within 0.2V of the input voltage.

Pressing and releasing the power button again should turn the radio off within a few seconds. There is no actual debouncing of the power switch, so sometimes it takes a couple of presses.

Turn off power to the board and set the current limit of your power supply back to zero.

Now install the Teensy Audio hat assembly, taking care to get the orientation correct (see pictures).

Connect a voltmeter to the 12V out connector. Connect power to the main board, this time to the 12V IN connection, and SLOWLY increase the current limit on your power supply. The target voltage should be reached with very little current load because the ATTiny starts up with the radio in the OFF condition. Any draw over 10mA is suspicious.

Check the voltage at the power output block, the voltage should be zero, since the radio is off (a few millivolts is fine).

Press and release the power button. The radio should start up in a couple of seconds with a load of about 190mA (more than 220mA is suspicious). Check the voltage at the output block; the voltage should be within 0.2V of the input voltage.

Pressing and releasing the power button again should turn the radio off within a few seconds. There is no actual debouncing of the power switch, so sometimes it takes a couple of presses.