K9HZ T41 MAIN BOARD
BUILD INSTRUCTIONS for PCB V12.6
May 1, 2024

Background Information:

The K9HZ T41 MAIN board is the next generation of hardware for the T41 software-defined transceiver. This is essentially Al Peter's V011 board with the addition of some extra control lines for off-board accessories, front panel support, display voltage selection, new display driver section, and a soft-ON/OFF unit that provides for Teensy shutdown code. The only plug-in boards used in this version are the Teensy audio hat and the Teensy board itself. The rest of the plug-in board hardware has been moved to the MAIN board so that it can be customized to the needs of the T41. This gives lower noise and wider flat bandwidth signals from and to the DAC and ADC. Note that software exists for testing out your V12 MAIN board. The MAIN board should be the first board in your T41 that you build because the Teensy processor will be used for testing the functions of the other boards when you build them.

Four things are new:

1) The V012 MAIN board has its own power supply (and, in fact, all V012 boards have their own power supply), 12V power is switched on at the MAIN board and distributed to the rest of the radio boards.

2) You will notice there are now parts on the back side of the MAIN board. This was done to optimize space and reduce noise.

3) There is a "Teensy Audio Hat Adapter" (shipped with the MAIN board) that shifts the audio hat to one side of the Teensy so that it clears the on-board power supply heat sinks. When this adapter is used, you must use 90-degree header pins for connection of the Q-OUT/AGND/I-OUT signals adjacent to Teensy pins 17/18/19 on the top side of the MAIN board. More about this later.

4) You must select the correct display voltage via J2 (WARNING! get this right or you may destroy your display) based on the display voltage ordered (either 3.3V or 5V).

There is a legacy issue that remains as well:

1) To use the legacy Switch Matrix board and encoders (ala the V010/V011 boards), one jumper wire needs to be installed to connect PIN 9 of the front panel/ Encoder connector on the MAIN board to PIN 39 of the Teensy (do that on the bottom). If you are using the new K9HZ Front Panel boards and connections, this jumper is not needed.

Inventory and Prework

Before you begin, inventory your parts against the latest V012 BOM to make sure you have everything you need to complete the MAIN board, including a Teensy Audio Hat and the Teensy Audio Hat offset board (see Figure 1).   The complete BOM is available on the GITHUB at: T41/T41_V012_Files_01-15-24/T41_V012_BOMs at main · DRWJSCHMIDT/T41 · GitHub

Optional hardware to consider when assembling the Main board.

There are a set of options that can be selected and added to the Main board either during the initial build, or later as needed to provide enhanced hardware functionalities. In its simplest form, the Main board is 100% compatible with previous versions of the T41 hardware. However, with a few low-cost components, the hardware configuration can support an add-on "scanned" (i.e. using a MCP23017 MUX instead of an analog pin) switch matrix and encoder "front panel" module, an accessory connector for built in advanced diagnostics and testing, and a start-up/ shut-down module that can run user specific code in the Teensy during radio start-up and shut-down.

Please read the following paragraphs and select the options you want to customize your T41 in advance of warming up your soldering iron.

OPTION 1: A base built Main board supports the V010/V011 switch matrix and four encoders. If you will use this configuration, populate the "front panel" and "encoder" locations with 2x5 pin male IDC connectors for the encoders, and connect the analog switch matrix board to pins 1 (GND), 9 (SW), and 10 (3.3V) of the front panel connector as was done on the V011 main board. However, if you decide to add the scanned front panel module, leave the "encoder" 2x5 pin male header box connector off the board. The front panel module will plug into the 2x5 pin male header box connector labeled "front panel". No other hardware changes are necessary.

OPTION 2: To prepare the main board for advanced diagnostics and testing, populate the "ACC" connector, J3, on the back of the MAIN board with a 2x4 male header box connector, or at least a 2x4 row of male header pins. See Figure 2.

OPTION 3: The on-off switch module is a clever way to electronically turn the T41 radio on and off with a FET. Like the V010/V011 power supply board, the basic design also provides reverse polarity protection. It also incorporates a small ATTINY85 processor programmed to communicate with the Teensy such that when the off button is pressed, it tells the Teensy to execute a "shutdown" routine with user code (examples: save selected parameters, last band and mode, volume, etc., custom shut down screen, stay active with screen saver, etc.). When the Teensy has completed the shutdown routine, it communicates back to the ATTINY85 to complete the shut down and turn the radio off. See Figure 3.

If you want to build the on-off module hardware in the red box on the V012.6 schematic, populate the main board with the thirteen parts shown in red on the BOM. A pre-programmed

ATTINY85 was shipped with your V012 board sets. If you don't want to use this module, please populate the two parts shown in the blue box on the schematic and blue on the BOM to provide reverse polarity protection.

OPTION 4: The display voltage is now selectable between 3.3V or 5V by the placement of the jumper on J2. Make sure the voltage is selected properly before connecting the display or you may damage it. See Figure 4.

OPTION 5: Note that on the V012 main board, three I2C busses are brought out for use by add on modules. Buss "0" is available on pins 1 and 3 of the "RF Control" connector, buss "1" is available on pins 5 and 7 of the "Front Panel" connector, and buss "2" is available on pins 5 and 7 of the "Bands" connector. While some functions of the T41 are controlled with these I2C busses, there are plenty of unused addresses left for experimentation.

Two pins on the "Bands" connector, pins 6 and 8 have been brought out for reading FOR and REF power when connected to an inexpensive line section (directional coupler) that you provide. If the front panel module option is used, all pins on the "Encoders" connector can be repurposed for external modules and experimentation.

A list of the I2C addresses used by the V012 boards can be found here:

 T41/T41_V012_Files_01-15-24/T41_V012_Design_Documents/T41_V12.6_I2C_Assignments.xlsx at main · DRWJSCHMIDT/T41 · GitHub

Building the Boards

With a medium heat, fine-tipped soldering iron (30-40W) assemble the V012.6 Main board in the following sequence for best results. Print out the BOM and check the parts off the list as you mount them. It essentially amounts to mounting the smallest parts close to the board first. Leave the larger parts like Q1 (on the bottom side of the board, or Q4 which is really just Q1 on the top side of the board), U1, and U3 until the end. Use the schematic and the part designators on the board for part location.

1. Mount two **SMD** parts (D5 and R16) on the backside of the board first **ONLY IF** you have opted to use the reverse voltage protection instead of the clever soft on-off routines. Both are clearly marked as D5 and R16. Remember… mount EITHER the **BLUE** parts or the **RED** parts but not both (**BLUE** part designators here are for just reverse polarity protection on the bottom side of the board, while **RED** are just for the on-off power control on the top side of the board, and **BLACK** are for either option). See Figure 3.
2. Mount U5 and U6, the SOIC packages. There are several good ways to do this. You can put a drop of super-glue gel on the board and put the part in place with a tweezers… and then solder the part. You can also put a spot of solder on one corner pin on the board… solder the part down at that point, and complete the soldering. You can also

use paste solder and a heat gun… whatever technique works best for you. There are plenty of YouTube videos on soldering SOIC packages onto circuit boards that can be reviewed before you start. See Figure 5.

3. Mount the SMD parts on the top side of the board. Mind the polarity of the three LEDs (D2, D3, D4) and the diode **D1**. The white bar on one side the footprint on the board should match the white bar drawn on the diode part. Remember to only mount **D1** if you are building the on-off power control option. See Figure 6.

4. Next, mount the test point header pins and connections (not the boxed headers). Note that the "exciter out" I/Q output jack has pins next to the Teensy for connection to the teensy audio hat. You should use 90-degree header pins to clear the audio hat above them properly. See Figure 7.

5. Mount the thru-hole parts except for the voltage regulators (U1 and U3) including the female headers for the Teensy and for **U4**. Mount **U2**, **Q2**, and **Q3** if used. See Figure 3.

6. Mount the thru-hole capacitors and IDC test pins. See Figure 8.

7. Mount the connectors. This includes the 2x5 IDC male pin connectors, and the audio jacks. **NOTE** that the "Acc" connector, the "Bands" connector, optionally the ethernet connection, and the second Teensy USB connection go on the back side of the board. Mount the 12V power connectors J1 and J4, and use a red and black sharpie to code the "+" and "-" terminals on the connectors.

8. Mount IDC pins at J2 and jumper the proper position for the voltage of your display. When in doubt, set it to the 3.3V position to start. See Figure 4.

9. Finally, mount **Q1** (or **Q4**… same part number) on the board (**Front** or **Back**) and voltage regulators U1 and U3. When mounting the voltage regulators, start by mounting U1 and U3 securely to their heat sinks using a small dab of heat transfer compound and the appropriate screw. Then solder the heatsink and regulator to the board. There are holes to mount the heatsink and regulator assembly to either side of the board for your convenience. **If you will not use the offset board for the Teensy Audio Hat, the heatsinks and regulators should be mounted on the back side of the board.** Note the pin orientation of the regulators if you mount them on the back side.

10. Solder an 8-pin DIP socket or socket pins at **U4** on the board and mount **U4**. See Figure 3.

11. You can now add a fan to the edge of the board. Glue the fan down using hot glue and solder the fan power wires to the holes provided. The fan is 12 volts and connects to J2.

12. Solder a set of stacking pins on the Teensy 4.1 as shown in Figure 9. If you use multiple sets of pins in each side to complete the side, you may need to file the side of the pin set plastic a little to get the sets to fit side by side (a Dremel works perfectly for this).

13. Solder the female pin sockets to the MAIN board for the Teensy 4.1. See Figure 10.

14. Cut the trace between the pads on the bottom of the Teensy 4.1 shown in Figure 11

15. Carefully insert the Teensy into the socket on the MAIN board. Take your time to make sure all of the pins line up, particularly those on the inside that are hard to see.

16. Press a set of short IDC pins into the top of the stacking pins on the Teensy 4.1 as shown in Figure 12.
17. Place the offset adapter board on top of the IDC pins added above as shown in Figure 13 and solder those pins into place.
18. Mount the IDC pins on the Teensy Audio Hat as shown in Figure 14. This is a 2x5 IDC pin set with one of the pins removed with a needle nose plyers.
19. Remove the offset adapter board with pins soldered from the Teensy 4.1. mount the Teensy Audio hat on the offset adapter with a set of short IDC pins as shown in Figure 15. Solder both sides of the pins to the boards. Then plug the offset board with the Teensy Audio Hat into the Teensy. Again make sure all of the pins align properly.
20. For those wanting to mount the Ethernet Connection to the Teensy, please check this out: PJRC Store
21. Make a cable to connect the "R_OUT" (or Q) and "L_OUT" (or I) and Ground (GND) on the Teensy Audio Hat pins with the 90-degree pins next to the Teensy board (pins 17, 18, 19). The best way to do this is to use IDC jumper wires with female header socket. Twist the wires together for best noise immunity.

Testing The V012.6 Main Board.

1. Remove the Teensy with offset board and audio hat from the MAIN board.
2. Connect the main board to 12V.
3. Short the pins at SW1 together momentarily with an IDC jumper or screw driver (be careful not to contact anything else!). The power LEDs D2 (for 3.3V) and D3 (for 5V) should come on. Measure the 3.3V power at TP2 and the 5V power at TP1 to make sure it's the expected value.
4. Remove power from the board, wait a few seconds, and then replace the Teensy and audio hat on the MAIN board.
5. Reconnect the 12V and short the pins at SW1 together again.
6. Connect a USB cable to the Teensy and you computer, and load the I2C address finder sketch from the Arduino IDC. The sketch can be found here: https://github.com/DRWJSCHMIDT/T41/tree/main/T41_V012_Files_01-15-24/T41_V012_Software/T41_V12_Software_For_Board_Testing/V12_Main_Board_Test/i2cscan
7. Running this sketch from your computer with watching the serial monitor, you should see:

```
Scanning Wire ...
I2C device found at address 0x0A  !  [Correct Address for Teensy Audio
Hat]done
```

8. Turn the power off. Connect the display to the "DISPLAY" connector on the MAIN board. Make SURE the cabling is correct!

9. Load the "basicTextFunctions" from the same directory sketch into the Teensy and observe the display. It should draw characters on the screen (see Figure 16).
10. Load the code to test the display located here: (NOT READY YET: fill this in when the code is done)
11. Reconnect the 12V and short the pins at SW1 together again.
12. After a few seconds you should see the display in Figure 26.
13. If you have made it this far the MAIN board is working. More if it will be tested when building the rest of the boards for the V12 T41 radio.
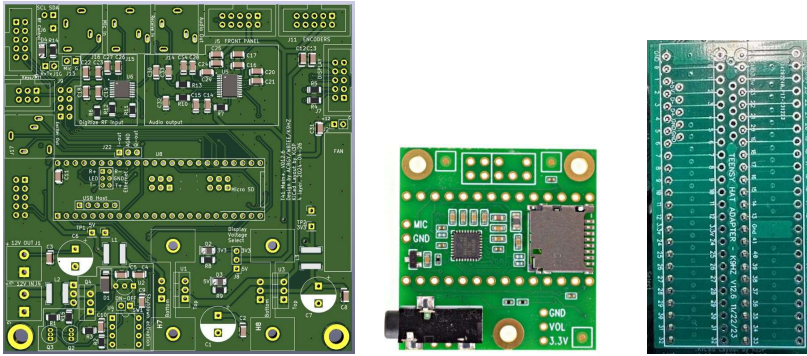
Figure 1. V12 Boards (Main, Teensy Audio Hat, and Hat Offset)
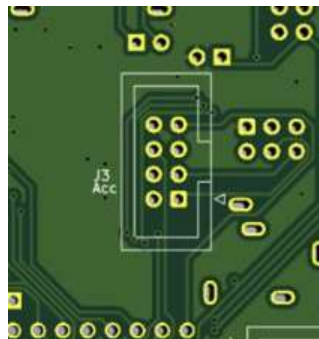
Figure 2. J3, the "ACC" Connector on the Back of the MAIN Board.
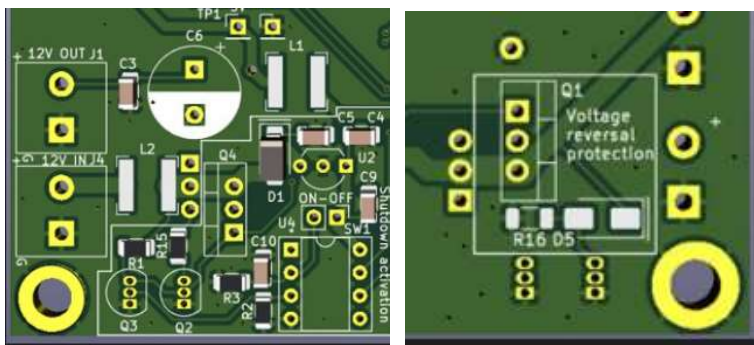
Figure 3. On-Off Switch and Reverse Voltage Protection Options.
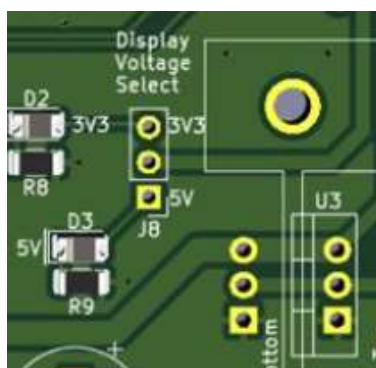
Figure 4. Display Voltage Selection Jumper.
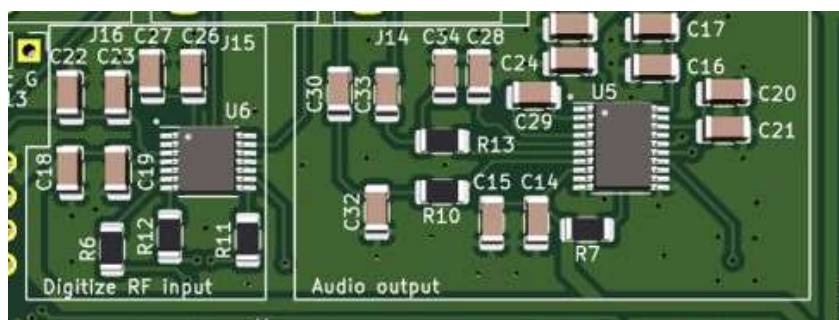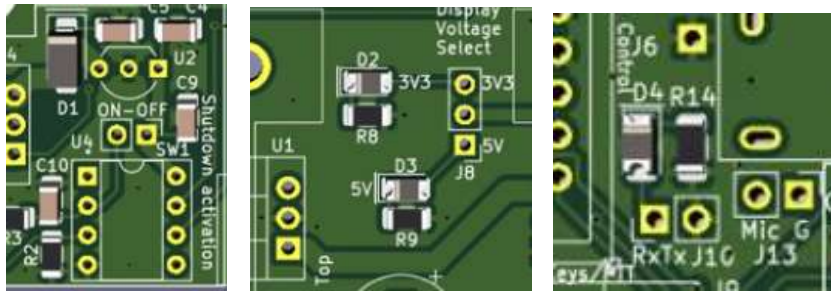

Figure 5. Mounting SOIC U5 and U6.

Figure 6. Mounting D1, D2, D3, and D4.

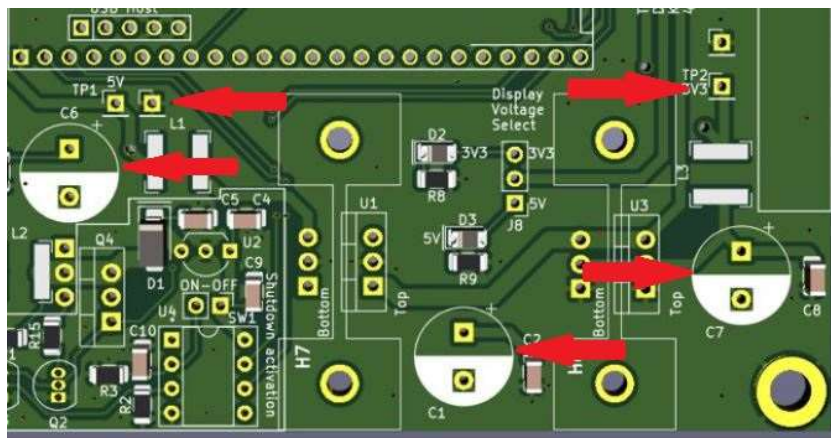**Figure 7. 90 Degree Pins Under Audio Hat Offset Board.**
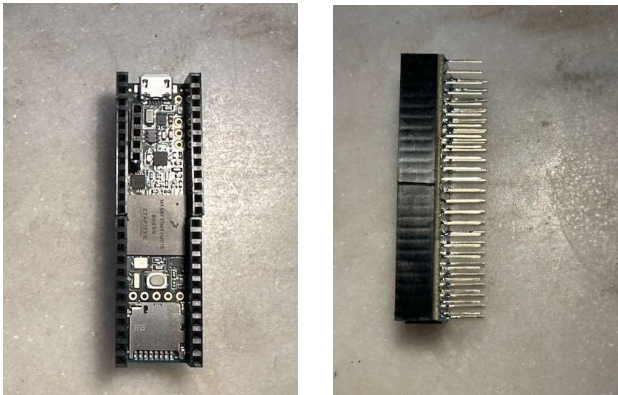
Figure 8. Location of Thru-hole Capacitors and Test Pins.

Figure 9. Adding Stacking Pins to the Teensy 4.1.

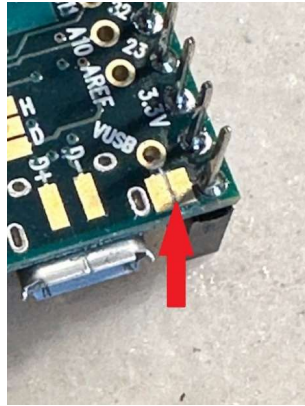**Figure 10. Placement of Pins on the Main Board for the Teensy.**

Figure 11. Cut Power Trace on the Botton of the Teensy 4.1.

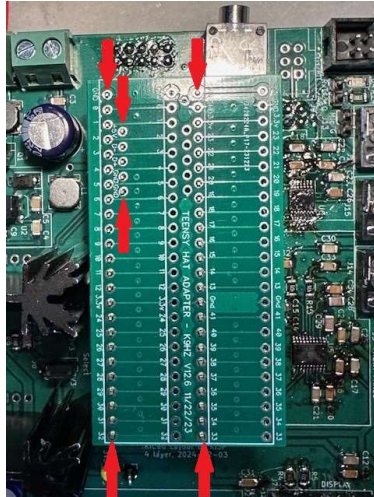**Figure 12. Teensy Mounted on the Main Board with IDC Pins Stacked on Top.**

Figure 13. Solder the Offset Adapter Board on the Pins Stacked Above.

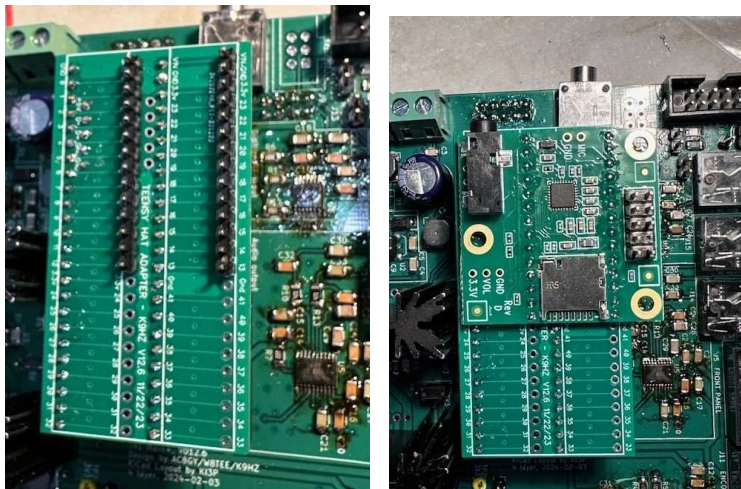**Figure 14. Mounting the Connection Pins on the Teensy Audio Hat.**

Figure 15. Mounting the Teensy Audio Hat on the Offset Adapter Board.

(Add photo once display test code has been written)

Figure 16. Display While Running Display Test Code.