**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](https://github.com/DRWJSCHMIDT/T41/tree/main/T41_V012_Files_01-15-24/T41_V012_BOMs)

**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 with a 2x4 male header box connector, or at least a 2x4 row of male header pins.

**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 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.

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.

**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 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.

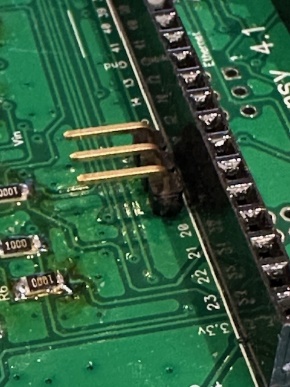
A list of the I2C addresses used by the V012 boards can be found here:

[T41/T41\_V012\_Files\_01-15-24/T41\_V012\_Design\_Documents/T41\_V12.6\_I2C\_Assignments.xlsx at main · DRWJSCHMIDT/T41 · GitHub](https://github.com/DRWJSCHMIDT/T41/blob/main/T41_V012_Files_01-15-24/T41_V012_Design_Documents/T41_V12.6_I2C_Assignments.xlsx)

**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. It essentially amounts to mounting the smallest parts close to the board first. Leave the larger parts like Q1, 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 **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, **RED** are just for the on-off power control, **BLACK** are for either option).
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.
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 bar on the footprint on the board should match the bar on the diode. Remember to only mount parts **D1** if you are building that option.
4. Next, mount the test point header pins and connections (not those in boxes). 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 here to clear the audio hat above them properly. See below.



1. 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.
2. Mount the thru-hole capacitors and IDC test pins.
3. Mount the connectors. This includes the 2x5 IDC male pin connectors, and the audio jacks. **NOTE** that the “Acc” connector, the “Bands” connector, the ethernet connection, and the second Teensy USB connection go on the back side of the board. Mount the 12V power connectors and use a red and black sharpie to code the “+” and “-“ terminals on the connectors.
4. Set the jumper at J2 in the proper position for the voltage of your display. When in doubt, set it to the 3.3V position.
5. Finally, mount Q1 on the board (**Front** or **Back**) and mount 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. Mount **U4**.
6. 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.
7. Put a set of stacking pins on the Teensy 4.1 as shown in Figure x.
8. If you will use the offset adapter, Add a set of short IDC pins to the offset adapter as shown in Figure x.
9. If using the offset adapter, mount the Teensy Audio hat on the offset adapter with a set of short IDC pins as shown in Figure x. Then plug the offset board into the end of the Teensy with the board mounted USB mini connector.
10. Make a cable to connect the “R\_OUT” (or Q) and “L\_OUT” (or I) with ground to the 90-degree pins next to the Teensy board (pins 17, 18, 19). The best way to do this is to use jumper wires with female 0.1 inch header socket ends (AKA DuPont connectors) cut to the needed length and soldered to the Audio Hat. 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



**FIGURES AND TABLES**

**Table 1. V12 Boards (Main, Teensy Audio Hat, and Hat Offset)**