Four State QRP

T41 – EP

Manual

Getting Started

This is an ambitious kit provided by the 4 State QRP Group. Before you begin assembly there are several things that you must know.

1. This radio was designed by Albert Peter, AC8GY and Dr. Jack Purdum, W8TEE.

2. In the time this radio was kitted, the T41 project has continued. It has not been stagnant and improvements have been made both in hardware and software. This is **not** to say that this radio requires anything, it is a very functional and high performing radio as is, however, should you wish to upgrade and update the capabilities within this radio, you will be able to do so after it is completed. It is an Experimenter's Platform after all.
Note: Build the kit in its stock configuration and confirm its operation BEFORE attempting to modify it.

3. Where do you find out about these updates? Well first, on the 4SQRP groups.io channel, particularly the T41 sub-group. Second, on the SoftwareControlledHamRadio group, which is also on groups.io. The Software Controlled Ham Radio group is where most of the updates, enhancements, and modifications are discussed. It is also where Al and Jack can be found.

4. As of right now, the 4SQRP T41 consists of Version 10 / 11 boards with some modifications.

Board Name	Version	Version Date	Notes
Switch Matrix	V009a	6-6-22	
Filter Board	V009a	6-6-22	
Exciter Board	V009b	2-11-23	May have a cap taped to the bag.
T41 Filters Board	V009b	1-30-23	This is not the same as the Filter Board
QSD Board	V009b	2-12-23	
20w PA Board	V009b	3-9-23	
Main Board	V009b	2-28-23	
Power Supply	V009b	1-30-23	
PCM1808 ADC	cjmcu-1808	No date	Has 5 cans labeled 10 100v VT
PCM5102 DAC	PCM 5102	No date	Has TRS jack mounted to board
Boost Converter	No marking	No date	Two large capacitors and a toroid

5. The current software version for the 4SQRP kit is 49.2K

Things you will need to buy other than the kit:

Order the following:

1. Book - Digital Signal Processing and Software Defined Radio - Theory and Construction of the T41-EP Software Defined Transceiver by Albert F. Peter, AC8GY and Dr. Jack Purdum, W8TEE.

2. Teensy 4.1 - Do not buy the version with pins pre-soldered
   - 600Mhz ARM Cortex M7 processor
   - 7936K Flash Memory
   - 1024K RAM Memory (512K tightly coupled)
   - 4K EEPROM (emulated)
   - 32 and 64 bit floating math co-processor
   - 55 digital input/output pins, 35 PWM output pins
   - 18 analog input pins
   - RTC for date/time
   - 8 serial, 3 SPI, 3 I2C ports
   - USB device 480 Mbit/sec; USB host 480 Mbit/sec

3. Teensy Audio Adapter Board (Audio Hat)
   - Make sure to get the Teensy Audio Shield for Teensy 4.0/4.1. This is not the same as the audio shield for earlier Teensy boards.

4. Micro SD Card - for the Teensy
   Either
   - 32GB SDHC Micro SD Card - formatted with FAT32 file system

   Or
   - 64GB SDXC Card - formatted with exFAT file system

   These should be:
   - Class 10 or SD10 but the speed is not critical

   Lesser options may work, consult Al and Jack’s book for more details

5. 5” RA8875 SPI Display
   - 800 x 480
   - TFT
   - Color
   - LCD
   - I2C
   - Serial
   - SPI

   Display Options
   - Interface - 4 wire SPI pin header
   - Power Supply 3.3v
   - No touch panel
   - No Micro SD Card Interface
   - No Font Chip

While you are waiting on parts, get started by reading the next part of this section, the **T41-EP Assembly Guide** and spend some time reading the book to get a feel for how the T41 works and how it is put together.

Spend some time on the 4SQRP and SoftwareControlledHamRadio groups, just don't let the talk of newer versions and enhancements confuse you. Concentrate on 4SQRP T41 posts. The other posts will make better sense once you've built the 4SQRP T41. Then you can decide if you want to upgrade or enhance the radio. Or just sit back and operate it bone stock. It'll be up to you.

Once you get started building the radio, read the sections of the book applicable to the section you are about to build. This will give you a deeper technical understanding of how the radio is built and prepare you to modify or repair the radio in the future.

T41-EP Assembly Guide
by
Jack Purdum, W8TEE and Al Peter, AC8GY

This is a supplemental document to the *Software Defined Radio Transceiver: Theory and Construction of the T41-EP Amateur Radio SDT* book (aka the "book") as sold on Amazon. The primary purpose of the book is to present an understandable overview of Software Defined Radio (SDR) receivers/transceivers and how Digital Signal Processing (DSP) is used in an SDR radio. The framework for the SDR/DSP discussion is the actual implementation of those circuits and building the T41-EP (T41 from now on) Software Defined Transceiver (SDT). The manual is designed to assist the T41 builder in constructing a working version of the T41-EP SDT as presented in the book. The book is *not* required to assemble the T41.

The purpose of this assembly guide is to help you build the T41.

Several assumptions are made about the build process. We assume:

- The builder has read the Software Defined Radio Transceiver book in sufficient detail to understand the operation of the T41.The builder is familiar with and comfortable translating schematic diagrams into actual circuits.
- The builder is employing the latest set of PCBs, which at the time of writing is V011.
- The builder is adept at soldering SMD parts, most of which are 1206 size. (Prefabricated boards are available from http://www.4sqrp.com/T41main.php.)
- The builder has access to adequate test equipment suitable for checking out the completed modules. • The builder is reasonably familiar with RF test procedures.
- The T41 is being constructed in a case like the one described in the book.
- Finally, that the builder/user has experience constructing complex RF projects.

We would like to emphasize that building the T41 is an advanced construction project. Beginners are encouraged to build other simpler kits and projects before attempting to build the T41. If that is not possible, then find someone (i.e., an Elmer) who can help you during the construction process. A good place to start your search is your local radio club.

We have successfully built and used several of these radios. More recently, Bill Schmidt, K9HZ, has had board sets fabricated to make it easier for interested hams to construct the T41. To date, Bill has distributed about 250 sets of PCBs. 4SQRP.com is producing a "semi-kit" that has all of the SMD components on the board and includes the parts necessary to complete the T41, less the Teensy and display. This guide should prove helpful to both groups.

While it takes time to build, in our opinion the effort is well worthwhile. We think the T41 has features not found in rigs costing 10x as much.

The best way to approach building the T41 is to start with the DC power supply and progress through the various modules. At each stage, the builder should read the book chapter(s) associated with that module and study the schematic circuit diagram before proceeding to build that board.

T41 Modules

The T41 Software Defined Transceiver consists of six main modules and other components attached to those modules. The six modules are each implemented on a printed circuit board, all of which are interconnected by means of ribbon cables, 50-ohm mini coax, or audio cables. The PCBs have provision for ribbon, coax and audio jacks, or the user may elect to direct wire the connections. The Modules consist of the following:

Main or Digital board

Contains the Teensy Microcontroller, ADCs, DACs, and Clock Generator, as well as connections and interfaces for the display, encoders, front panel switch matrix, and audio preamplifiers and audio power amplifier.

Receive Board

Consists of the RF preamp, Quadrature Sampling Detector (QSD), and anti-aliasing filter, as well as a 4 to 1 frequency divider.

SSB Exciter Board

Contains a quadrature modulator (QSE), an RF summing amplifier, and another 4 to 1 frequency divider.

Filter Board

Contains six multipole high-pass and low-pass filters:

a. 2.8MHz high pass filter always in the circuit
b. 30MHz low-pass filter, always in the circuit
c. Switched low-pass filters for 80M, 40M, 20M, and 15M.
d. Switching is done using RF relays.
e. Also has the transmit/receive switching.

DC Power Supply Board

Provides 5V DC, 3.3V DC, and 12V DC to the other modules.

Outboard 12V to 25V Boost converter for RF power amp

RF Power Amp Board

a. 20W RF power amplifier.
b. Input approximately 40mV from Exciter Module
c. Requires 25V @ 4A max DC.
d. Must have a heatsink with adequate cooling surface. Approximately 150mm x 50mm x20mm as a minimum.

Tools and Assembly Materials

You must have certain tools and test instruments to complete this project. In addition, there are other tools you need to either have or have access to them. If you do not have these tools or instruments, see if a member of your club has one they will let you use. Also, junior colleges/colleges/universities normally have labs with such tools in them. Check the EE or Physics department. Many high schools will also likely have such test equipment. Explain that you are a licensed ham radio operator "building a sophisticated SDT transceiver". That helps to identify your credentials. Also, be prepared to take the T41 to their lab. It might be convenient to take a battery pack with you, too.

You should read the section in our book titled *General Construction Considerations* found in Chapter 17, too.

Must have:

1. A clutter-free, open, workspace that can be left undisturbed by cats and family members. We found a large white towel covering the workspace; good for helping to locate dropped SMDs, and an ESD mat.
2. Good lighting or a directed reading light with accessible AC power for gun/iron and test equipment.
3. Good eyes, or a magnifying visor.
4. A good digital voltmeter/multimeter (DVM). A good option is the VC97 for about $30. It is useful that your DVM have an audible continuity feature, as you will use it often to check for shorts between IC pins.
5. A good soldering iron (around 60-80W). If you don't have one, consider buying a solder station with an air gun. A good station will also include a soldering iron with controls to control both the iron and gun. Expect to pay around $75. Don't forget solder paste for the air gun and a tip cleaning pad for the iron.
6. Solder. Small soldering irons work best with small solders. Optional - Radio Shack still sells solder online which has 2% silver and is 0.022" in diameter. We think the 2% silver makes it easier to solder things, but that may be just in our heads. (Jack had a BestBuy salesman try to talk him into gold plated wires for a printer cable ($49.95 vs $7.95). When asked why, the salesman said it will make the printer print things faster.) Anything much larger than 0.300" is not recommended.
7. Wire strippers. The 5" adjustable strippers work well and usually cost less than $10.
8. Small needle nose pliers. We find small pliers made for working with jewelry a good choice and they are under $5.
9. Small screwdriver set. The set we got was from Harbor Freight and named "Precision Electrical Screwdriver Set" with a cost of less than $10.

10. Tweezers. We find a 5-6 piece set which includes a hooked pair ideal. Look under "tweezers for electronics" and the set cost should be under $10.
11. Drill and bits for panel controls.

Fine-tooth file for panel edges. Can also be used to strip enamel off toroid wire. (You can also burn off the enamel, but it smells like feet when you do it.)

Nice to Have or Have Access To

1. Oscilloscope. Must be able to sweep at RF speeds (e.g., 100MHz or more). If it's under $100, it's probably not fast enough to be useful at RF.
2. Signal generator capable of sine wave output frequencies for all HF ham bands. Expect to pay $100-150. Some include a frequency counter, too.
3. Spectrum analyzer capable of scanning through the HF bands. Expensive and probably not worth buying for just one or two projects. Look for a friend.
4. Dummy load capable of sinking at least 25W of power.
5. Cable crimper used to attach 10- and 16-conductor ribbon to IDC connectors.
6. Digital camera. Kinda nice to document your progress and to take close-up of toroids when you've failed to get turn-counts to agree after multiple attempts.

General Construction Process for All Boards

For each module, the general process is as follows:

1. Following the BOM and schematic for the module to be built, gather all of the parts.
2. Print off a copy of the circuit diagram.
   Arrange the parts on a piece of cardboard and using double-sided sticky tape attach and label each non-SMD part. See Figure AM-7 below.
3. SMD parts should be left in the manufacturer's packaging until it is time to place and solder the part onto the board.
4. Start assembly with any SMD ICs that are on that module board. IC's like the Si5351 are pretty small and it helps if nothing is nearby as you position the chip for soldering. Especially nettlesome are those pesky skyscraper electrolytics that are on some boards.
5. Place each IC on the PCB and solder in place using solder paste and a heat gun. Use only the minimum amount of solder paste – see Chapter 17 for more details. It is possible to solder ICs with a small soldering iron, but the heat gun using solder paste is much easier. Watch a few online videos that have a title like "solder SMD components tutorial". You will see how the paste/heat gun combo makes things much easier and how the solder mask on the PCB works to your advantage with a heat gun.
6. Test for continuity of the IC leads and check for shorts between pins. This is one of the most common points of failure in constructing the T41. If there is a problem, remove the part, clean off excess solder using solder wick and resolder. Sometimes you can just use the solder wick without removing the IC from the board. However, do not overheat the pads as it is possible to lift them from the board with too much heat. Don't trust your eyes...recheck it with your DVM.
7. After all ICs are mounted, begin to add the 1206 SMDs.

8. We recommend starting at one side of the circuit diagram and mounting and soldering parts following the schematic. After each part is soldered in place, mark that part off on the schematic with a yellow highlighter. We do not recommend doing batches of similar parts. This tends to lead to missed parts and missed solder joints.
9. After each part is attached, take the tip of the soldering iron and touch it to one side of the SMD briefly and then the other, pausing a couple of seconds between sides. (See Figures 17-12 to 17- 14.) If the part has been properly attached, it will not move. We find that another common point-of-failure with SMD-type construction arise from partially soldered 1206 SMDs.
10. Once all of the SMDs are in place, begin the assembly of the other components, following any specific instructions given below for that module. Generally, you work from components that lay flat on the board (e.g., ICs, caps, resistors) to electrolytics, toroids, trim pots, and other parts that “stick up” from the board.
11. After the module is complete, follow the test procedure(s) for that specific module. • If all is well, move on to the next module. Do not assume everything is fine and move on to the next board. Test it first.

Detailed PC board Surface Mount Component Installation

All of the resistors and capacitors (except electrolytics) are relatively large (i.e., 1206). The kit sold by 4SQRP is a “semi-kit” where the SMDs are already mounted on the board. The reader may wish to review the section Soldering SMD Components in Chapter 17 of the book which discusses using either an air gun or soldering iron for mounting SMD components. We present a short discussion about SMDs here in the event an SMD might need to be replaced.

Install ICs

As mentioned above, start your soldering with the IC surface mount components first. Do this to avoid dislodging any small SMD with the heat gun and for you to have enough room to maneuver the IC into place.

There are two ways the ICs may be soldered in place, either pin by pin or using solder paste and a heat gun. We prefer the solder paste approach.

To use the solder-paste approach, you need to have a hot air gun. Set the air temp to about 400°F and then follow these instructions:

1. Put the IC in place on the PCB pads using tweezers in one hand. Observing the # 1 pin locator, place the IC on the solder pads with the tweezers. Carefully align the IC pins and solder pads.
2. With the other hand place the tip of a solder aid or small screwdriver on top of the IC to firmly hold it in place. Then using the tip of another small tool place a very thin line of solder paste on the outside of the PPCB pads just next to the IC feet. Don’t use too much paste! Doing it this way reduces the chance of excess solder under the IC legs, causing shorts.

3. Use a small orifice on the heat gun and alternately heat each set of mounting pins. If your solder station has an airflow control, use the lowest setting that melts the paste, but doesn't blow nearby components away. When the solder paste melts and the solder flows around the IC pins, the heat gun should be removed. Note how the solder mask on the PCB causes the solder to flow to the proper position on the board. **DO NOT** release the pressure until the solder is cooled. This may take 15-20 seconds.
4. We generally use a small sharp instrument or the needle that comes with the paste as an applicator for the paste because the syringe either applies too much paste or clogs after a few uses. Dip the sharp tip into the tube and draw out a drop onto a plastic surface. Then pick up a bit of paste with the sharp tip and spread it on the IC pads. Jack uses a scrounged dental pick instead of the syringe.

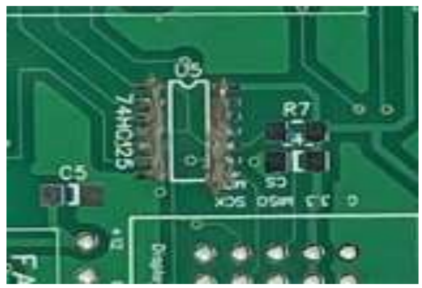


Figure Section00-1. Painting IC pins with solder paste.

Before moving on, carefully test each of the IC pin connections for both continuity and shorts with adjacent pins. Now, do it again.

If problems occur, do the following:

- Hold the IC down as before and re-heat the solder joints.
- If that does not work, remove the IC using the heat gun, completely clean
- the solder pads using solder wick and re-do the steps above to attach the IC.

Sockets and pin headers

Install Pin header sockets and pins for test points:

1. Select the correct number of socket pins.
2. Solder one pin in the row of pins on the bottom side of the PCB.
3. Make sure the connector is perpendicular and flush with the PCB and straighten if necessary.
4. Solder the remaining pins.

For single test point pins, place a small piece of masking tape over the pins to hold in place, solder and straighten if necessary. (Your work looks more professional if you now remove the masking tape!)

Ribbon Cable and other connectors

Install IDC connectors:

Most of the boards have IDC connectors to simplify inter-board connects. Observe the location of the index on the IDC connector. These connectors are used to interface with the 10-conductor cable. Figure AM-2 shows the female/male IDC connectors. Towards the bottom right is the male connector, shown with its “key” opening on top. Above the male connector is a female connector. Note that the plastic key shown on the bottom of the female connector must align with the key on the male connector.

The top left is another view of the female connector, showing the “teeth” that bite into the 10-conductor cable used with the IDC. The part below the female connector on the left is used to align the conductor cable. If you look closely, you can see the ridges that help align the cable with the teeth on the female connector.

The last part is a strap that secures the pieces, so they don't come loose. Pay attention to the plastic “keyhole” molded into the male IDC connector and match it to the silk screen outline on the PCB. (You can see an example of the silk screen for an IDC connector near the lower-left side of Figure AM-3. The indent in the silk screen in Figure AM-3 should align with the male IDC key slot shown in Figure AM-2. The key ensures that the connection is made properly.*)*

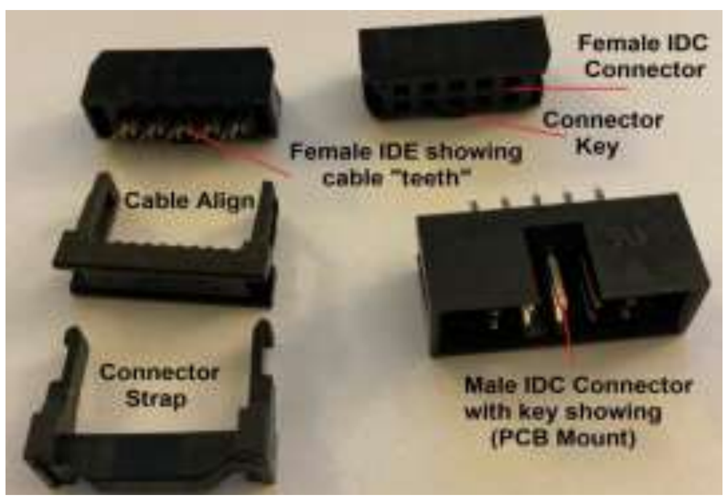


Figure Section00-2. The IDC connector.

Later when you make the cables that plug into these connectors, you should make sure you understand how the connector and its socket mate. Take a bare wire and stick it into a pin hole on the socket to see how each lines up with the “teeth” that end up penetrating the cable.

- Solder pins on the underside of the PCB
- Ensure the pins are straight

Conclusion

The sections that follow present specific instructions for building each board. We encourage to read each section completely before your start assembling that board.

Section 1
Building the Power Supply

Introduction

First, because of Covid 19 and the ensuing supply chain problems, 4SQRP had to make substitutions at various places as the kit was being developed. As a result, there are some modifications that are made to conform to the PCBs that were already made. Several such changes were made in the Power Supply relative to its PCB. For example, the LM1117 3.3V regulator was replaced with OKI-78SR-3.3/1.5-W36-C buck regulator. This requires some modifications to the regulator and the way it is mounted on the board.

The Power Supply PCB

First, Figure 1-1 shows how the bare PCB looks. Near the center of Figure 1-1, you can see the silk screen for the LM1117 3.3V regulator. That regulator has been replaced with the OKI-78SR mentioned earlier. Because the pinouts are different for the two regulators, the OKI has to be modified according to these instructions:

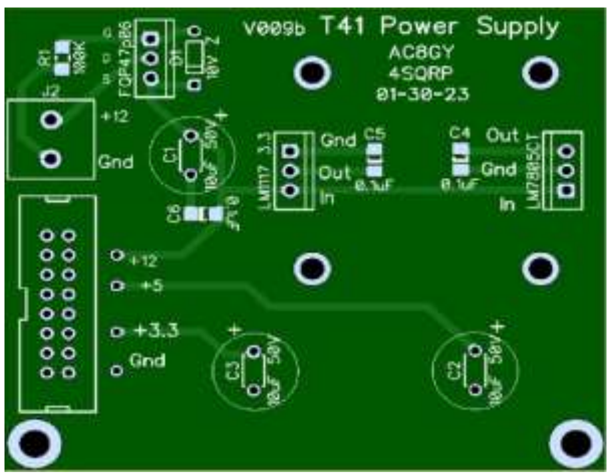


Figure 1-1. The Power Supply PCB.

The LM1117 Replacement Voltage Regulator

In the Rev A kit, the PCB is marked V009b, and U1, the LM1117, was replaced by a switching power converter module. The order of pins in the PCB footprint for this module is incorrect, so the U1 module must be modified.

This method is a quote from Tom Sevart N2UHC.

"The correct way to do it is to lift pin 1, solder a wire to it, *flip the regulator module around*, then put pin 2 in the ground hole, pin 3 in the output hole, and wire from pin 1 to the input hole.


Figure 1-2. Modified OKI voltage Regulator.

Below is a chart of the pin outs for the regulator.

INPUT/OUTPUT CONNECTIONS OKI-78SR	
Pin	Function
1	Positive Input
2	Common (Ground)
3	Positive Output

Replacing an Electrolytic Capacitor with a Tantalum Capacitor

The next deviation is the mounting of the tantalum 10µF capacitor on the PCB. Once again, a substitution was made using the tantalum cap instead of a normal electrolytic cap as shown as C3 on the PCB. Figure 1-4 shows that the longer lead is the positive lead, and it should be inserted in the positive (+) lead hole for C3 on the circuit board. Solder the leads on the back side of the board. First inspect the solder connections on both sides of the PCB. They should be smooth, round, and conical going up the wire a bit. Also, if you take your thumbnail and “strum” the soldered lead, it should give off an almost musical note. If you hear a thud instead, touch it again with the soldering iron. It might need a tad more solder, too.

Figure 1-4. Identifying the tantalum positive lead.

The Other Components

First, locate the D1 Zener diode. It has a dark band on the cathode end similar to that shown in Figure

Figure 1-5. Zener diode.

1-5. If you look closely in Figure 1.1, you can see a painted line on the silk screen for D1. Bend the leads so they fit the spacing for the diode on the PCB and insert it, making sure the dark band aligns with the silk screen band on the PCB. Solder in place. It only takes a few seconds to strum the leads here, too. If you hear the dreaded thunk, touch again with the soldering iron. Don't hold the iron on the lead too long as diodes are a tad frail. Trim the leads close to the PCB.

Mount the pale green, two conductor, screw terminal at J2 on the board. Make sure the two openings on the side of the connector face away from the PCB. (See Figure 1-6.) Solder in place.

Now mount the 16 pin IDC connector onto the PCB. Note that the connector has an opening, or key, cut into one side. Make sure that key lines up with the small cutout you see on the silk screen on the PCB. Hold the connector in place with tape while you flip it over to the back side of the PCB. Solder one of the end pins. Now flip it back over again and make sure the connector sits firmly on the PCB. If not, reheat the pin and try again until it's tight to the top side of the PCB. Now, flip the board over and solder the rest of the pins.

Mount the IRF5305 MOSFET at the location silk screened FQP47p06, just to the left of D1. In Figure 1-1, you can see two closely spaced lines silk screen on to the component's space. This is supposed to represent the metal tab that you see on the back of the MOSFET. Position the metal tab accordingly and solder into place. Trim the leads.

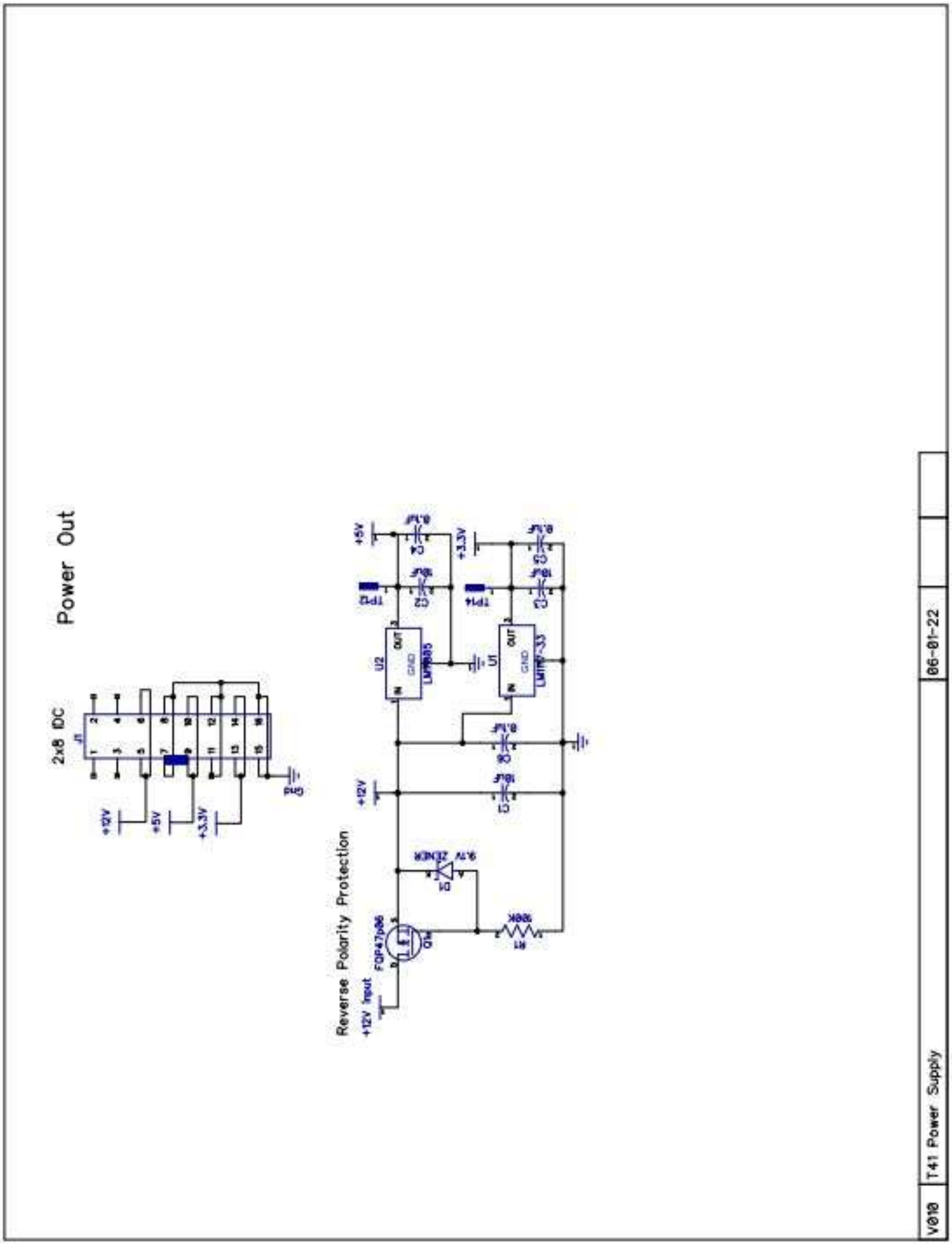
Now take the finned heat sink, the gray mounting pad, the LM7805 voltage regulator, and screw and mount the voltage regulator to the heat sink. Perhaps the easiest way to do this is to place the screw through the hole in the metal tab on the LM7805, then place the mounting pad on the screw, and then thread the screw into the op-most hole on the heat sink. (Some builders have reported that the hole in the LM7805 is too tight. Either thread the bolt through the tab or gently ream the hole.) Now thread on the nut on the other side of the heat sink. Insert the three leads of the LM7805 into the matching holes on the PCB. There are two mounting stubs on the bottom of the heat sink that match up with large holes in the PCB. When in place, flip to the bottom side of the PCB and solder the LM7805 leads in place and trim. You can also put two small blobs of solder on the two mounting stubs of the heat sink for added rigidity.

Testing

With a Digital Volt-Ohm Meter (DVM), measure the voltages at the test points shown in Figure 1-6. While there may be slight differences, the voltages should be very close to 12V, 5V, and 3.3V. If the voltages do not appear correct, check your soldering for cold or missing solder joints.
Attached below is a pdf of a Ver 10 power supply board that is very similar.

Figure 1-6. The completed board

Finished picture compliments of Jim Jones W0NKN



Power Supply Schematic and Bom

Quantity	RefDes	Value	Name
3	C1, C2, C3	10uF 50V	CAP Thru hole
3	C4, C5, C6	0.1uF 50V	CAP SMD 1206
1	D1		10V ZENER
1	J1		2x8 IDC
1	Q1	FQP47p06	MOSFET_ENH_P
1	R1	100K	RES_1206
1	U1		LM1117-33
1	U2		LM7805
1	Power Connector	See Notes	
2	Heat Sinks	See Notes	

Notes

1. Electronics-Salon 4X TO-220 Heatsink, Small Power Aluminum Heat-Sink, SKU: HS-001-4
   PCB Mounting hole spacing 25.4mm

2. Power connector
   OIKI 2-Pin 5.08mm Pitch PCB Mount Screw Terminal Block, Straight Plug-in
   2-Pin (2 Pole) Screw Terminal Block Connector, Pluggable Male Female Phoenix Type Connector for Arduino PCB Shield

2. IDC Connector
   16 Pin 2x8 2.54mm Pitch

V001

This completes the power supply assembly.

Section 2: Assembly Instructions for Main Board

Inventory the components in the bag. It should contain the following Main PCB plus the BOM items in Table 2-1.

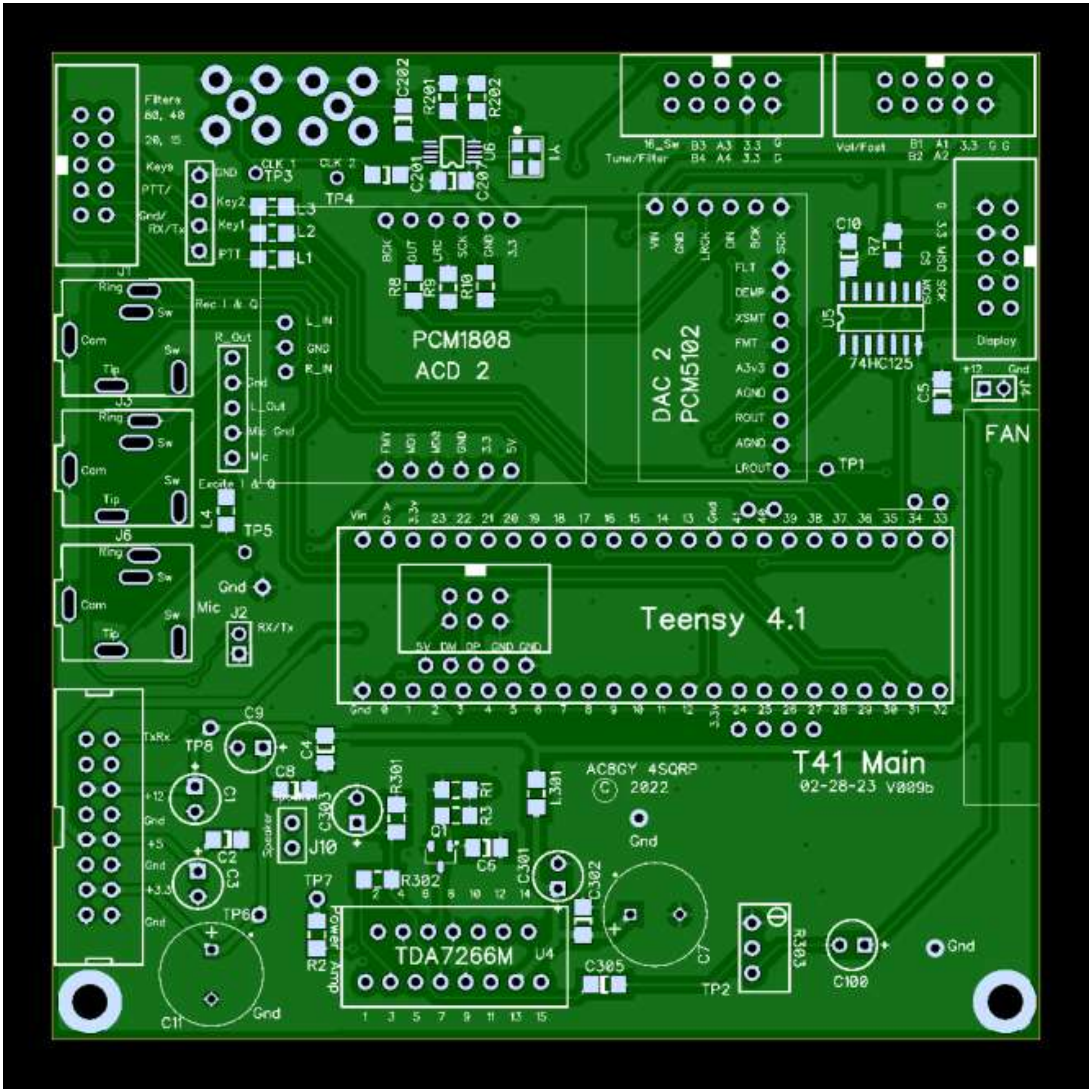


Figure 2-1. The Main PCB.

Table 2-1. BOM for Main Board

	Qty	Reference	Description	Details	√
1	6	C1, C3, C9, C100, C301, C303	10uF electrolytic		
2	2	C7, C11	470uF electrolytic		
3	1	J1	2x8 Box Header		
4	4		2x5 box header		
5	2	CLK1, CLK2	SMA		
6	3	J4, J5, J6	3.5mm audio		
7	1	R303	10k pot		
8	1	U3	TDA7266M		
9	1	U5	PCM1808		
10	1	U7	PCM5102		
11	1		3 pin female	Under the PCM1808	
12	3		6 pin female	Under the PCM1808 and PCM5102	
13	2		25 pin female	24 pin, cut one off - Under the Teensy	
14	1		10 pin female	9 pin, cut one off - Under the PCM5102	
15	3		40 pin male	Break off as needed	
16	2		15 pin long pin female	Teensy header, 14 pin, cut one off	
17	3	J2,J4	2-pin 2.0mm	for fans and J2	
18	1		5 pin female cable	40cm, 5 pin	
19	1		2 pin 2.54mm cable	speaker	
20	4		10 pin cable	To encoders, relay board, display	
21	1		Teensy fan		
22	1		USB Host Cable		
23	1		4 pin female cable		

Inventory the items, checking off each as you identify them. If you are missing any parts, please contact us at http://www.4sqrp.com/index.php, and the item will be promptly sent to you.
There are components on this board that are polarized, meaning they must be inserted properly in the board to function. In every case, there is a marking on the PCB that indicates the proper orientation of the component. Double check each part before inserting and soldering.

Placing Board Parts

Go down the BOM list above, install components beginning with C1, through U3. Solder, and trim the leads.

☐ Note: *Please wait to install the SMA connectors labeled Clk1 and Clk2 until the boards have been mounted on the chassis. This will allow you to customize your cable routing.*

☐ Pay attention that the key cutouts on the IDC connectors matches the key indicators on the silk screen.

☐ The electrolytic capacitors have a stripe painted on the case to indicate the negative lead.

☐ All the caps and connectors should fit snugly on the PCB, with little to no space between the component and the PCB.

☐ Depending on how you lay out your boards in the enclosure, it may be easier to mount the SMA connector for Clock 1 on the bottom side of the board and Clock 2 SMA on the top of the board. (SEE "NOTE" ABOVE)

☐ Mount the three audio jacks located near the middle-left on the Main PCB. Make sure the connectors fit snugly to the PCB. Solder in place.

☐ There are 3 keyed, 2 pin, 2.0mm connectors, but only two are now used:

Figure 2-2. Two pin connectors

☐ Place one connector at J4, just above the fan on the right edge of the PCB. Make sure the connector key cutout faces towards the fan and away from the IDC connector. Solder in place.

☐ Place the second connector at location J2 (near the Mic audio connector), with the cutout key facing towards the Teensy. See Figure 2-1. Solder in place. Save the third connector for use later in the assembly.

☐ Place R 303 near the bottom right-hand corner of the board. Before installing please check the resistance as described below.
NOTE: Connect an ohm meter to the Center pin and either of the ends pins and read the resistance. Adjust the screw for approximately 50% resistance or 5k ohms.
Solder and clip the leads.

☐ Break off two header pins and place the short ends into the holes marked “Speaker” at position J10, immediately below C8. (See Figure 2-1.) Use a piece of tape to hold them in place as you flip the board over and solder them into place from the back side.

Preparing for the PMC1808

Locate the PMC1808, the 3 pin female header, and two, 6 pin female headers.

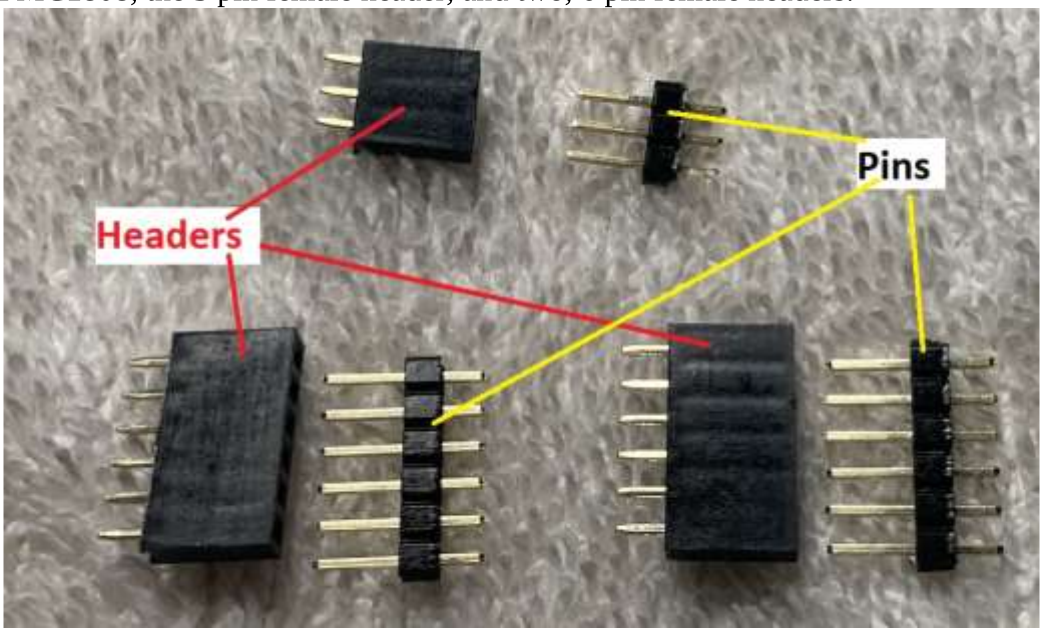


Figure 2-3. Preparing the headers and pins.

☐ Take one of the 40-pin male header strips, and snap off one 3-pin section, and two six pin sections. The components look like those shown in Figure 2-3.

☐ Insert the longer pins of each male header into the corresponding female header.

☐ Place the mated headers into the PCB in the positions under the PCM1808 marking, with female headers oriented facing down. See Figure 2-4.

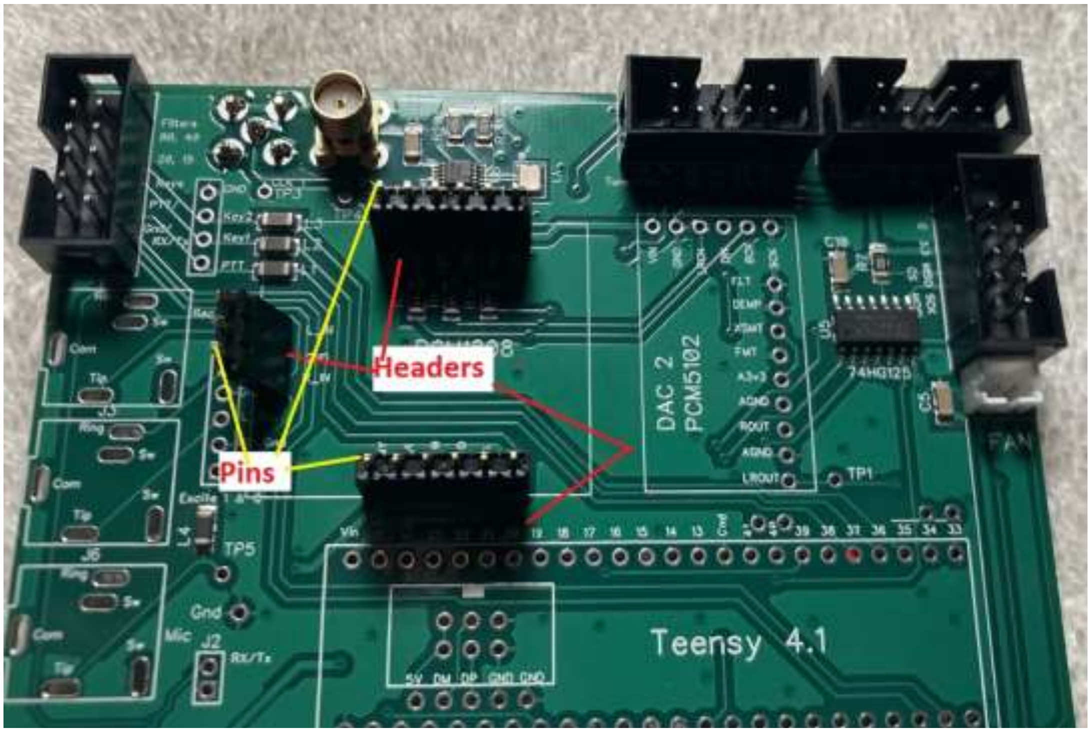


Figure 2-4. Dry fitting the combined headers and pins on the PCB.

☐ Place the PCM1808 module onto the exposed male header pins, and solder one pin on each header, then check the alignment of all of the parts, if they are all good solder all of the pins on top.

☐ Invert the PCB, and, while supporting the PCM1808 from below, solder the pins of the female headers in place.

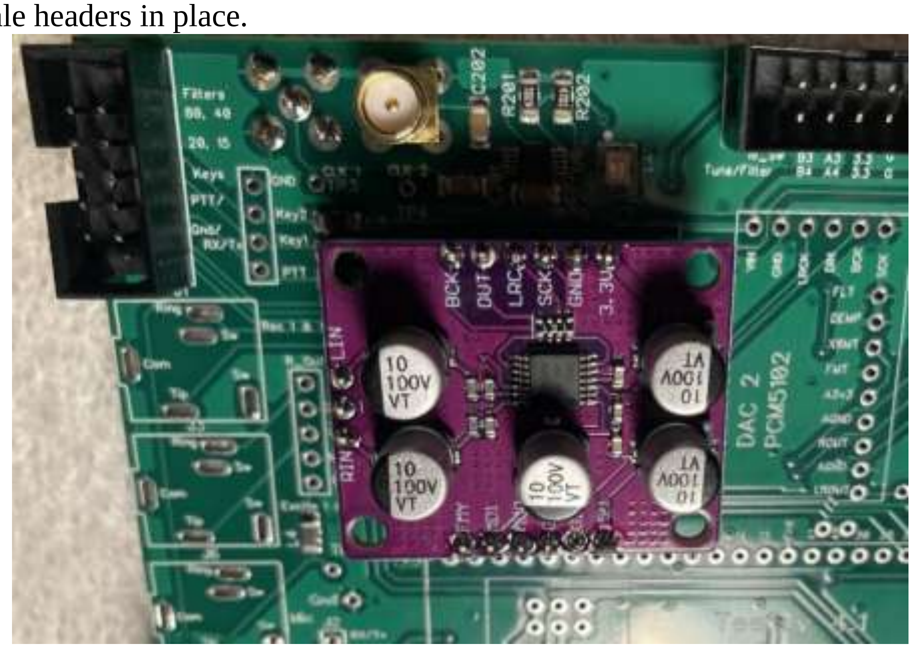


Figure 2-5. Placing the PMC1808 on the pins and solder into place.

Preparing for the PCM5102

☐ Locate the PCM5102, the 6-pin female header, and the 10-pin female header.

☐ One of the positions of the 10-pin female header must be removed to make it a 9-pin header. Perform this operation by crushing the end-most cavity of the header with a pair of diagonal cutters. (see Figure 2-6) The metal contact and unwanted plastic may then be removed.

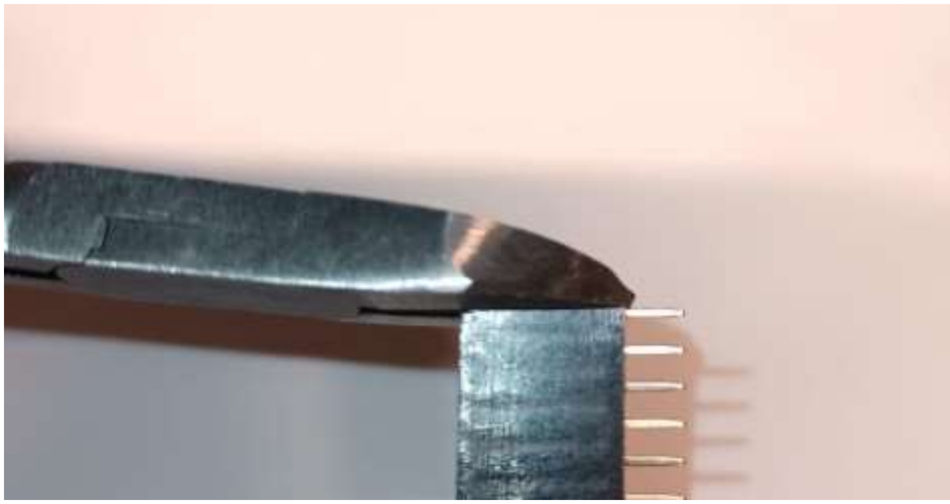


Figure 2-6: Removal of female header contact

☐ As an alternative to crushing the 10th pin, you can simply snip of the 10th pin at the base of the header and mount it so the "snipped" pin almost touches the 6-pin header, as seen in Figure 2-7. We've had a few situations where crushing the header loosened the adjacent pin, so this seemed a little safer.

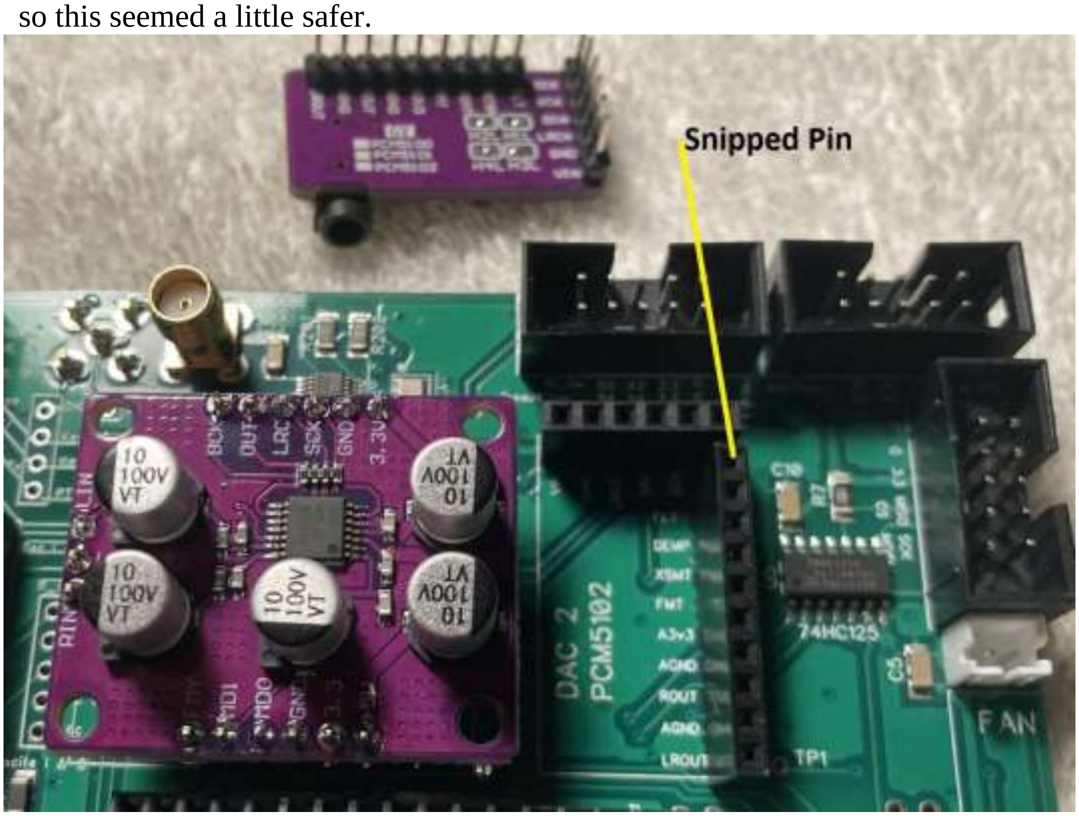


Figure 2-7. Header without crushing plastic shell.

☐ Use some tape to hold the two headers in place, flip the PCB to the back side, and solder just one end pin first and then flip the board over and make sure the header sits tightly on the PCB. Now solder the remaining pins.

☐ Snap a 6-pin and 9-pin section from a male header strip. Mate these to the female headers from the previous step, inserting the long pins into the headers, leaving the short pins exposed.

☐ Place these mated header sections into their corresponding positions on the PCB in the area for the PCM5102 and place the PCM5102 module onto the exposed short header pins.

☐ Solder each of the header pins at the upper surface of the PCB5102 module.

Mounting the Teensy 4.1 and Audio Board

☐ Locate the two, 25 pin, long-tailed Female headers. Cut one pin from each, making them into 24-pin headers. Here, again, you can leave the headers as 25 pin headers, but snip off the 25th pin on each header close to the bottom of the header, as seen in Figure 2-8. (Note there are still 25 sockets, but #25's pin has been cut way.) While the header is a little longer than necessary and "hangs over" the silk screen outline, the longer header does not interfere with any other components on the board and removes the risk of loosening the 24th pin. However, you *must* remember to insert the pins starting on the righthand end of the header strip. Otherwise, all of the Teensy pins will be "one off".

Figure 2-8. Modified Teensy socket header.

☐ **NOTE:** Locate your Teensy 4.1 board. It is necessary to modify the board by cutting a hair-like trace on the bottom of the board. See Figure 2-9. Check the two pads with a DVM after cutting the trace to make sure they are separated and the DVM shows an open circuit between the two pads.

Figure 2-9. Cutting the Teensy pads.

☐ Take the Teensy board (user supplied part) and two 15 pin long pin stackable header and fit them onto the Teensy towards the end with the USB connector. See Figure 2-10. (In Figure 2-10, we came up short of 15 pin stackable headers, so we made our own using a 10 pin and

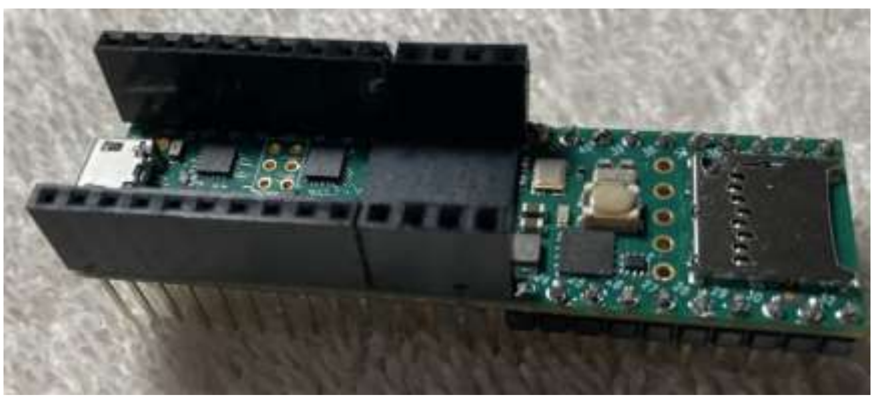


Figure 2-10. Long pin stackable headers.

4-pin long stackable headers.) You only need 14 of the 15 pins, so you can either remove he 15th pin and its plastic, or you can snip off 15 pin and let the extra length hang out over the left edge of the Teensy. Once the long pin stackable headers are in place, flip the PCB over and solder one end pin of each header. If the header is sitting tight on the PCB, solder the rest of the pins on the bottom side of the Teensy.

☐ Break off 2 10-pin pieces of header pins and insert them in the remaining holes in the Teensy 4.1 board with the long pins extending from the bottom. Solder the short pins to the Teensy 4.1 board. See Figure 2.10.

☐ Break off two 14 pin headers and insert them into the Teensy Audio Adapter (user supplied part). Solder them in place as shown in Figure 2-11.

Figure 2-11. Audio Adapter Board with 2 14-pin headers solder in place.

☐ Take the Teensy Audio Adapter and the Teensy 4.1 board and fit the audio board into the long pin header sockets. **NOTE**: if you left the 15th pin in place on the long pin stackable header, make sure you insert the Audio Adapter starting with pin #2, *not* pin #1. Align the boards as shown in Figure 2-12.

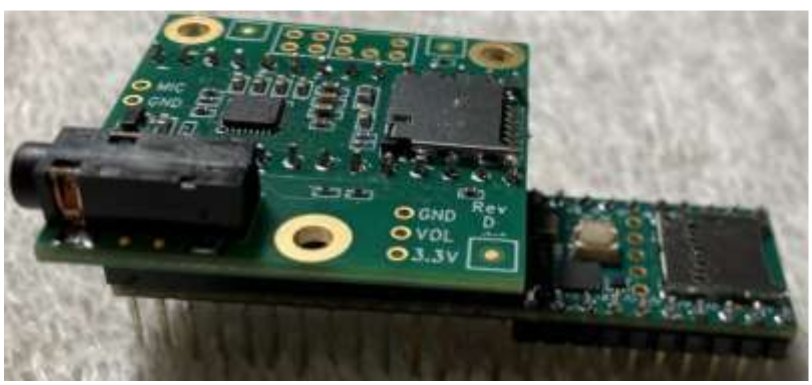


Figure 2-12. Audio board plugged into Teensy.

☐ Carefully unplug the Teensy Audio Adapter board from the Teensy.

☐ Locate one of the 40-pin male header strips. Break a 5-pin section from it.

☐ Insert this five-pin header into the location directly to the left of the PCM1808 and solder it into place from the bottom side.

☐ Locate the 5-pin cable assembly. Cut the cable approximately 3” (8 cm) from one end. Separate the conductors, and strip approximately ¼” (6 mm) insulation from the cut ends.

☐ Locate pin 1 on the connector, it will be marked with a triangle or arrow. Solder the cable conductors to the input pads of the audio adapter board per the table below, or as shown in the picture below. (Some of the labels are on the bottom of the board.)

Table 2-1: Audio Adapter Board Cable Connections

Pin	Connection
1	LINE OUT R
2	LINE OUT G
3	LINE OUT L
4	GND
5	MIC

Note: *The original Manual has Connection 4 (GND) and 5 (Mic) reversed on the audio board. Please confirm this before soldering.*

Figure 2-13. Audio board mounted on Teensy with cable.

☐ Plug the Audio Adapter board into the female header on the Teensy, making sure that the SD card socket faces to the right. Plug the 5-pin cable into the 5-pin header on the PCB, with pin 1 of the connector oriented toward the top of the board.

☐ Carefully insert the Teensy into the sockets on the main PCB. **NOTE**: It is very easy to bend the header pins resulting in one or more pins not being properly seated into its connector.

From the side of the board, check under the Teensy to make sure all of the pins are properly seated in their sockets. See Figure 2-11.

Miscellaneous

☐ Break off a 4-pin section of the male header strip.

☐ Insert this four-pin header into the position at the upper left corner of the PCB, to the left of PCM1808. Hold the pins in place with tape, flip to the back of the PCB, and solder into place.

☐ Mount the TDA7266M audio amplifier (the centipede-looking thing) near the bottom-left of the board. (See Figure 2-1.) Flip the board over and solder in place.

☐ Locate the small fan, and a white 2.0mm 2-pin male header Mate the fan power cable connector to the 2-pin header.

☐ Insert the connector into the J2 'RxTx' position, with the red wire in the position marked '+' on the PCB silk screen. Solder the connector into place.

Cooling Fan

☐ Disconnect the fan cable connector from the male header at J4, and mate it to the second 2.0mm male header.

Alternative Fan Mounting

Hot gluing the fan in place makes it difficult to access the Teensy SD card holder. Indeed, it is impossible to remove the micro-SD card without reheating the glue holding the fan in place. It's almost as difficult to unplug the Teensy to access the micro–SD Card. You might want to prepare the SD card *before* you mount the fan. Instructions for preparing the micro-SD card are found in the Software section of the manual.

The data on the micro-SD card allows you to enter a DX call and the software uses your QTH to draw a vector from your QTH to the foreign station's QTH. (Figure 2-14 shows the bearing display.) The angle displayed shows the (azimuthal) bearing to that station. Even if you don't have a rotatable beam, this bearing feature is useful for entering a call prefix and determining the country associated with the call. The software has over 400 entries in its lookup table, including all DXCC countries. Using the bearing feature requires that the micro-SD card holds your QTH map data. The code supports up to 10 alternative QTH locations in case you use the T41 at different places.

Note that the vectors emanating from the map are the actual longitude/latitude coordinates of the user. As mentioned above, you can have up to 10 different QTH locations. After selecting the location that you wish to use, the bearing calculations are performed using that location. The first location in the list of 10 is the default location.

Other Cooling Fan Mounting Options

One option is to hot glue the fan in place just behind the Teensy Board.
This may cause issues if you need to remove the SD card.

I glued a small strip of Velcro to the fan and the main board. This seems to work quite well.

An alternative to permanently gluing the fan in place is to hot glue a small (mini) magnet (e.g., Amazon B0936M3WPK) to the fan's location on the PCB and hot glue a second magnet to the bottom of the fan. The magnets hold the fan in place, but it can be moved out of the way should you need to access the micro-SD card. See Figure 2-15. (It may look a little precarious, but the magnets are strong enough to hold the fan in place.)

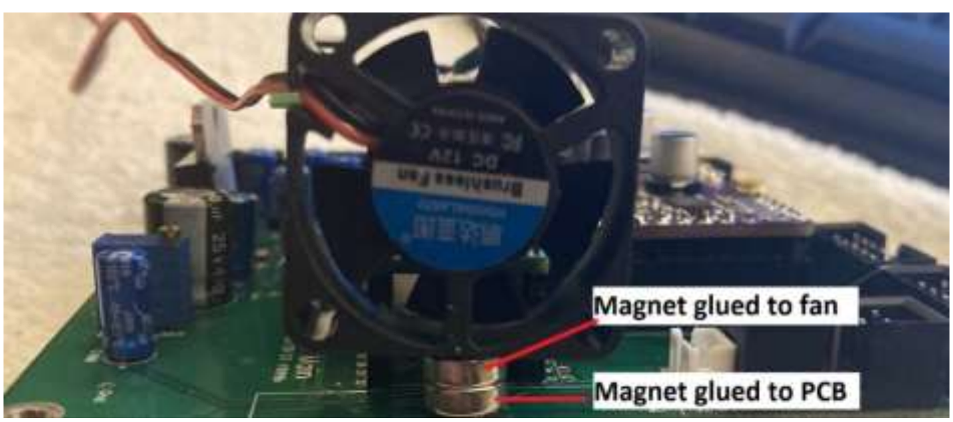
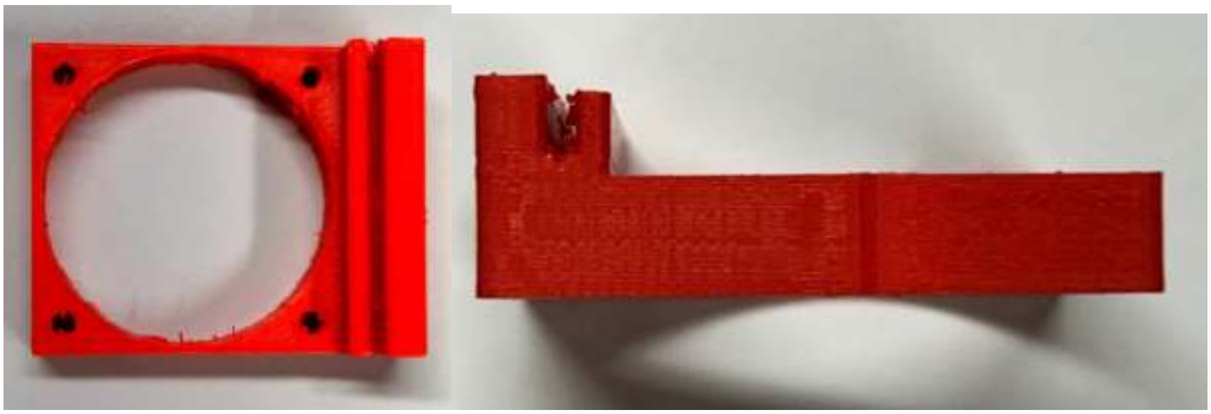


Figure 2-15. Hot glued magnets to hold fan.

Another option was shared with us by a builder in Europe. He designed and 3D printed a mounting bracket for the fan. The SDT file is available here: https://4sqrp.groups.io/g/T41/files

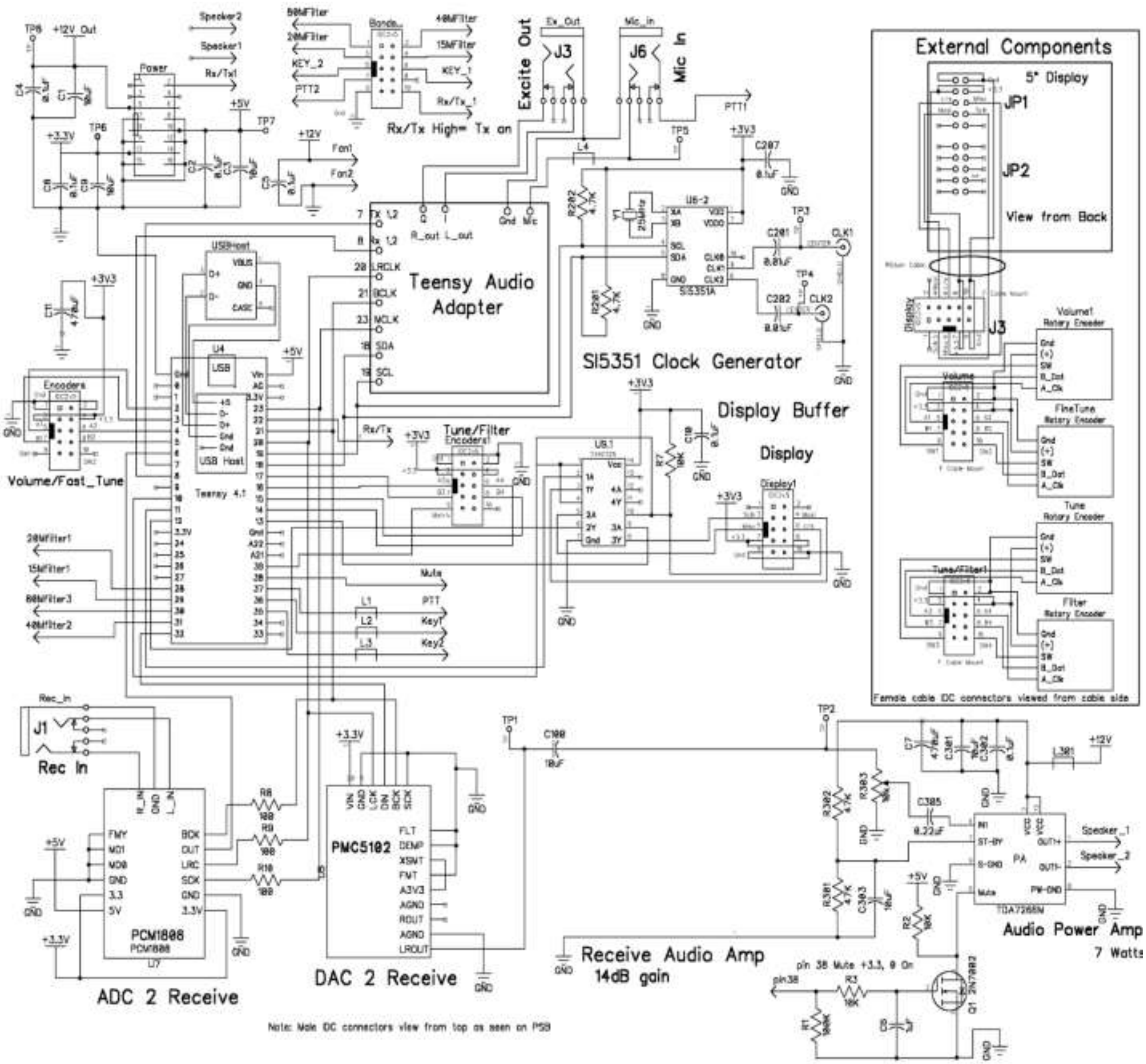
Picture of the 3d printed fan mount.
Compliments of Roger Trett



☐ Solder the 2-pin fan power connector to the PCB in the J4 location immediately above the fan on the PCB. Be certain that the red wire of the power cable goes to the +12v connection, and the black wire goes to Gnd.

☐ Connect a two wire (red/black) cable assembly to the speaker connection at J10. Leave the other end unconnected for now.

This completes the Main board construction.



External Components
5" Display
JP1
JP2
View from Back
Volume1 Rotary Encoder
FineTune Rotary Encoder
Tune Rotary Encoder
Filter Rotary Encoder
Female cable IDC connectors viewed from cable side.
Excite Out
J3
Mic In
J6
Rx/Tx High= Tx on
Teensy Audio Adapter
SI5351 Clock Generator
Display Buffer
Display
Tune/Filter
Volume/Fast_Tune
Teensy 4.1
USB Host
J1
Rec In
PCM1808
ADC 2 Receive
PMC5102
DAC 2 Receive
Receive Audio Amp
14dB gain
Audio Power Amp
7 Watts
TDA7266M
Note: Male IDC connectors view from top as seen on PSB

Section 3: Encoder Boards and Switch Matrix Assembly

☐ Inventory the components in the bag. It should contain:

	Qty	Description	√
1	4	Rotary Encoder Assy	
2	2	2x5 male box header	
3	1	5 pin male header strips	

☐ Inventory the items, checking off each as you identify them. If you are missing any parts, please contact the kitter listed at http://www.4sqrp.com/T41main.phpx, and the item will be promptly sent to you.

☐ Some of the components on this board are polarized, meaning they must be inserted properly in the board to function. In every case, there is a marking on the PCB that indicates the proper orientation of the component. Double check each part before inserting and soldering.

☐ There are two encoder PCBs and each one has a top and bottom side. Figure 3-1 shows the top side of a PCB. The encoders are mounted on the top side of the board so their shafts can extend through the front chassis panel.

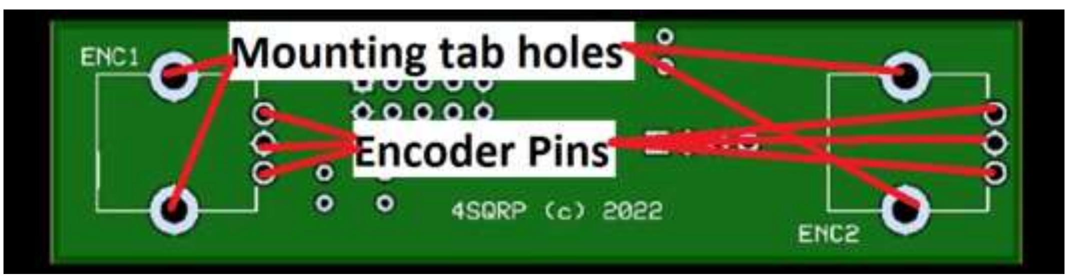


Figure 3-1. Top side of encoder PCB.

NOTE: Because of supply chain issues, not all encoders fit into the two large mounting tab holes in the PCB. This is not a problem and is easily fixed in the next step. For now, simply snip the mounting tabs off.

☐ Install the rotary encoder pins in the encoder pin holes and solder the rotary encoders on the top side of the PCBs. If your encoders have the mounting tabs snipped off, just place a small amount of solder on the mounting tab holes and the metal encoder body.

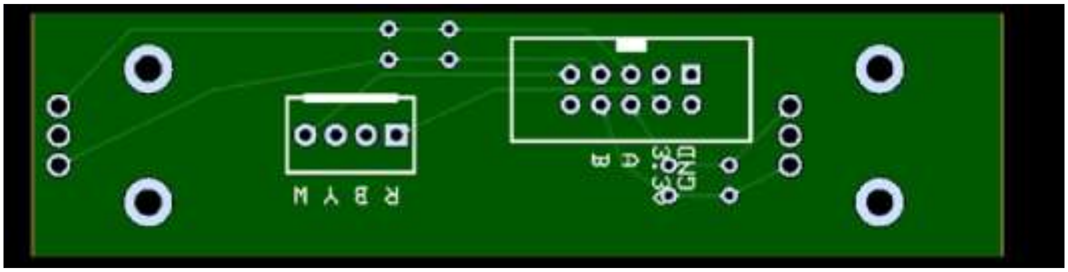


Figure 3-2. The bottom side of the encoder PCBs.

☐ Figure 3-2 shows the bottom side of the encoder boards. Solder a 2x5 IDC connector in the space shown by the silk screen. Note that the connector key faces the outside edge of the PCB. Install and solder one pin of the IDC connector. Flip to the other side and check that the header is flush with the PCB. If not, reheat the connection and push the IDC connector until it sits firmly on the PCB. Solder ea the remaining IDC pins.

☐ There is a 5-pin male header strip included in the bag of parts. Snap off one of the pins and install the remaining 4 pin headers on the bottom side of the encoder PCB. (The silk screen reads RBYW in Figure 3-2.) Note that the short pins are soldered to the PCB and the long pins protrude away from the PCB. The finished board should look like Figure 3-3.

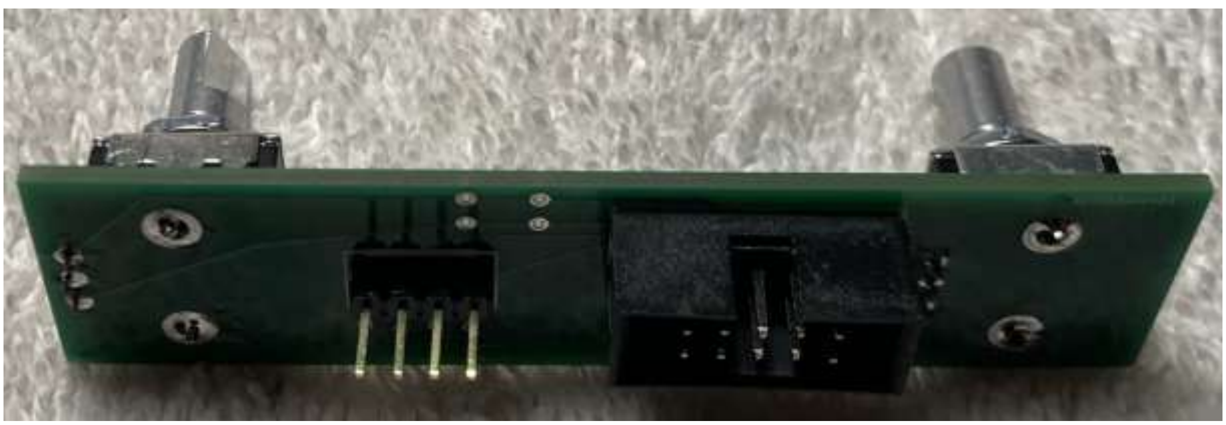


Figure 3-3. Completed encoder board.

Switch Matrix

The switch matrix is used to alter the state of the T41 without having to wade through a menu system. Jack and Al have selected the features that are most often changed during operation and tied them to one of the switches. There is also a menu system for the T41, but these are for less often changed items.

There are 18, PCB mount momentary ON switches used in the matrix. See Figure 3-4 (taken before resistors and capacitor soldered in place). Also included are 18 caps for each of the switches.

Figure 3-4. Pre-SMD switch matrix.

☐ Mount each of the switches in the provided mounting holes. The tabs are spaced such that they snap into place when mounted correctly.

☐ Check that the switches are level and align with each other. If they don't align properly, chances are one of the switch tabs is not seated in its mounting holes correctly. Adjust until they align properly. This alignment is more important than you think. Each of the 18 switches will be fitted later with a plastic cover before it's inserted into the front panel. If the switches are not seated properly (i.e., flush) to the PCB, the plastic switch cover will likely bind when pushed making that switch always ON. Not good. Take your time and make sure the switches are mounted flush with the PCB.

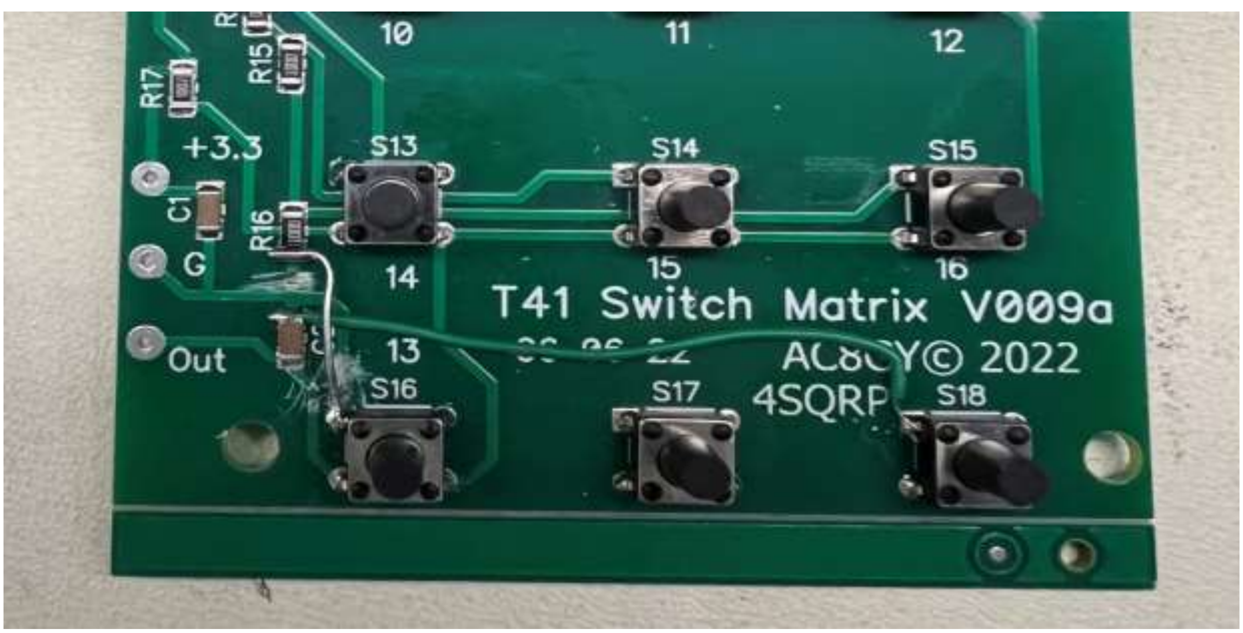
☐ Flip the board over and solder each switch to the PCB.

☐ *The original Switch Matrix board only has 16 active switches.*
It is necessary to modify the switch connections to enable the 17th and 18th switch.
These Mods and pictures are courtesy of Wayne McFee NB6M.

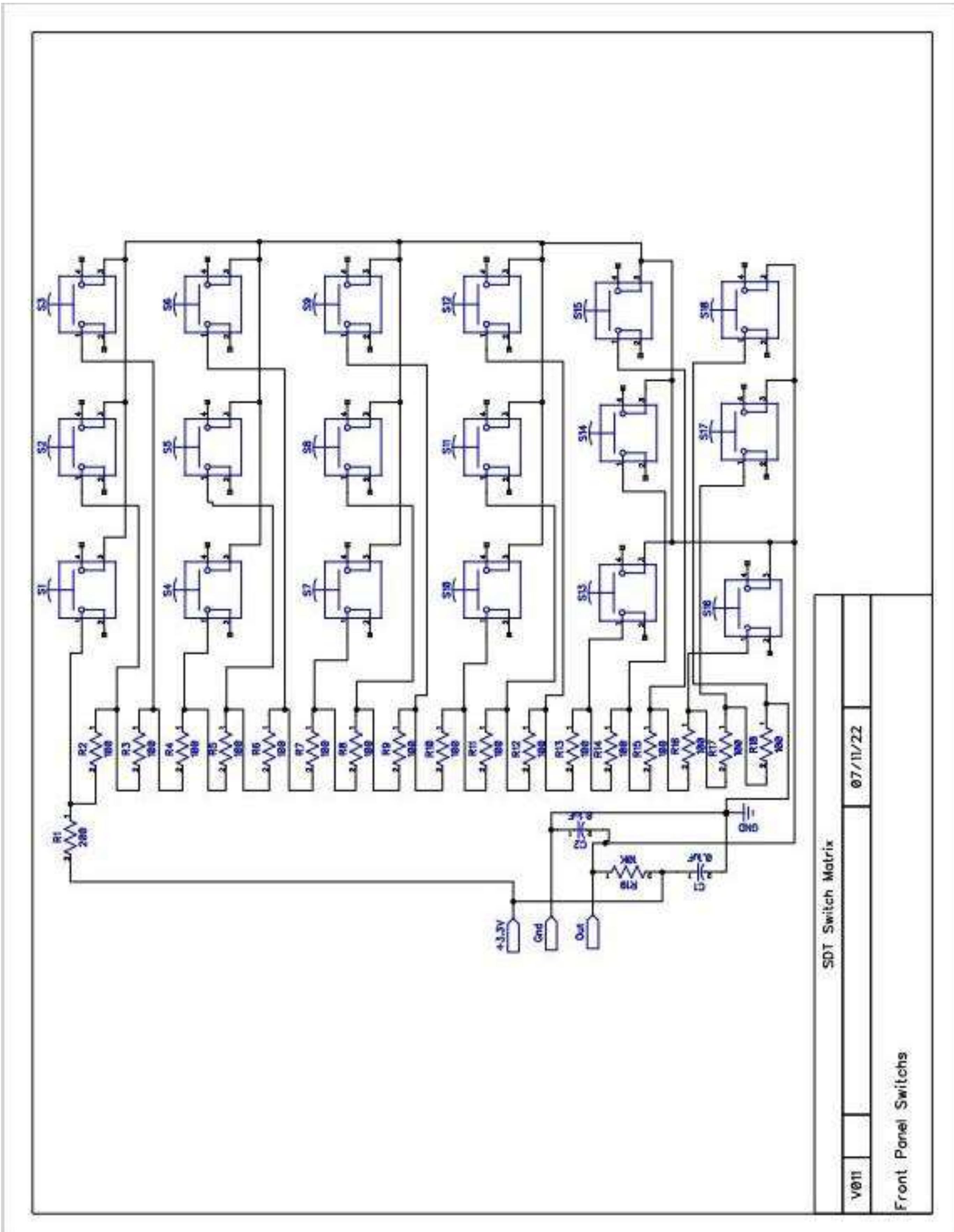
In order to modify the switch matrix like the board version 11 schematic shows (attached below), the following modifications were made:

With the board oriented as shown,

Cut the traces between the bottom of R16 and ground, and between the top of S16 and ground.
Install a jumper between the bottom of R16 and the top of S16.
Install a jumper between ground (top of C2) and the top of S18.
Install a 100 Ohm resistor between the top contacts of S16 and S17, and another between the top contacts of S17 and S18.
Install jumpers from the lower (Voltage Out line) contacts on S16 to the lower contacts on S17 and from the lower contacts on S17 to the lower contacts on S18. See the attached photos for clarification.




+3.3
G
Out


SDT Switch Matrix
07/11/22
V011
Front Panel Switchs
+3.3V
Gnd
Out
GND

Quantity	RefDes	Value	Name
2	C1, C2	0.1uF	CAP_1206
1	R1	200	RES_1206
15	R2, R3, R4, R5, R6, R7, R8, R9, R10, R11, R12, R13, R14, R15, R16	100	RES_1206
1	R17	10K	RES_1206
16	S1, S2, S3, S4, S5, S6, S7, S8, S9, S10, S11, S12, S13, S14, S15, S16		B3F-1000
35			

SDT Switch Matrix				
V001			09/04/21	
Front Panel Switchs				

This completes the encoder and switch Matrix PCB construction.

Section 04:
QSD Receive Board, Rev A

☐ Inventory the components in the bag. It should contain the items shown in Table 4-1:

Table 4-1. QSD Board BOM

	Qty	Ref	Description	√
1	11	C5, C7, C9, C204, C205, C206, C208, C305, C307, C310, C311	10uF electrolytic	
2	1	C11	470uF electrolytic	
3	2	J1, J2	SMA RF connector	
4	1	J3	2x8 Male box header	
5	1	J4	3.5mm audio	
6	1	T1	FT50-43 toroid	
7	1	W	23 inches AWG 26 wire	

Inventory the items, checking off each as you identify them. If you are short any parts, please contact the kitter listed at http://www.4sqrp.com/index.php, and the item will be promptly sent to you.

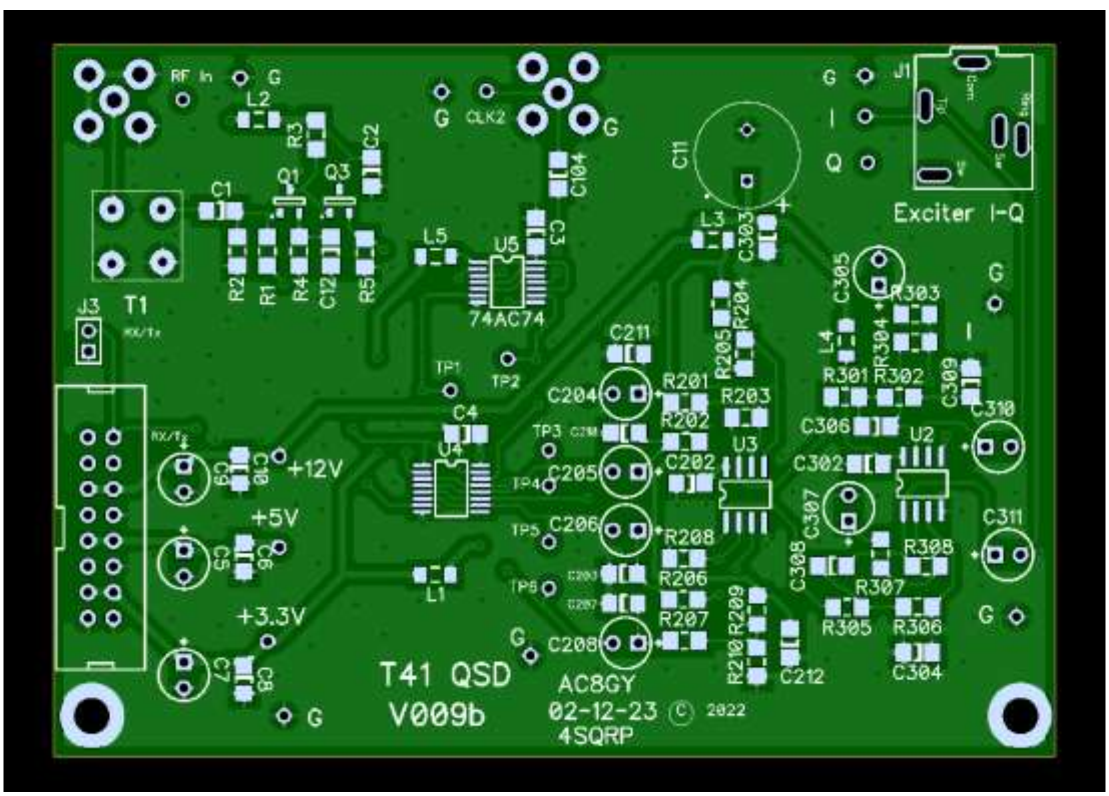


Figure 4-1. The QSD PCB.

☐ NOTE: *Please wait to install the SMA connectors until the boards have been mounted on the chassis. This will allow you to customize your cable routing.*

☐ All of the components on this board are polarized, meaning they must be inserted properly in the board to function. In every case, there is a marking on the PCB that indicates the proper orientation of the component. Double check each part before inserting and soldering.

☐ Solder the 11 10µF electrolytic capacitors at their respective silk screen locations on the board. Pay close attention to the polarity of the caps (gray stripe on the side is the negative lead). After all of the caps are in place, flip the board over and solder in place. It doesn't hurt to "strum" the leads before you trim them. If they go "thud" rather than producing a music-like note, touch the lead with the soldering iron again.

☐ Mount the 470µF electrolytic cap onto the board, again paying attention to its polarity. Flip over, solder, strum, and trim.

☐ Mount the 2x8 IDC connector in the place provided near the lower-left edge of the board. Flip to the back and solder one pin. Flip to the front and make sure it is properly seated and tight to the board with the key facing outward. Flip back over and solder the rest of the pins in place.

☐ Solder the 3.5mm audio connector in the space provided on the PCB. Flip over and solder one pin. Flip back to the front side and make sure it is sitting flush on the PCB and its properly aligned. Flip back to the other side and solder the remaining 4 pins to the board.

Toroid Winding and Mounting

☐ Carefully unwind the coil of AWG 26 magnet wire and gently unravel and smooth out any kinks that might be in the wire. With two sets of pliers, firmly grab each end of the uncoiled wire and gently stretch the wire to remove any "memory" the wire may have from being coiled. You should end up with a straight piece of wire with no kinks or bends in it.

☐ Using the numbers from the Kits and Parts web site (https://kitsandparts.com/xtoroids.html), or measure about 12" of wire and cut. With fine grit sandpaper or a box cutter, gently remove about 0.75" of the enamel coating from one end of the 12" wire. Take care not to nick the wire if using a box cutter. (These ends eventually are soldered to the board and it's necessary to remove the insulating enamel for a good connection.)

☐ First, wind the 15 turns on the secondary. Holding the 0.75" stripped end in your fingers, pass the other end through the center of the toroid. Each pass through the center of the toroid counts as one turn. Grab the free end and pass it through the center of the toroid again. Pull the wire tight so each turn fit snugly on the core, but not so tight that you can't move it a little. You now have 2 turns on the core. Repeat until you have 15 turns on the core. When you are finished, spread the turns evenly around the core.

☐ As you can see in Figure 4-2, the "unscraped end" is longer than we need. Scrape about 0.75" of the enamel insulation off the long end, starting at the bottom of the coil. If you are using a box cutter, lay the wire on a flat surface and gently scrape off the enamel. Now loosely twist these two ends together. We do this so we can distinguish these turns from the new coil and turns added in the next step.

Figure 4-2. 15 turns on TF50-43 core.

☐ Cut the remaining wire to about 8.25” and use pliers to gently stretch the wire. Scrape about 0.75” of enamel off one end. Grasp the scraped end and feed the other end through the center of the core. Loop the free end around and through the center of the core again and snug down gently so the turn sits firmly on the core. Repeat for 10 turns, wrapping over the first 15 turns that are already on the core. Try to space the turns evenly over the entire core as you wrap them, as it is much harder to slide them now that the other 15 turns are on the core. The finished toroid should look similar to Figure 4-3.

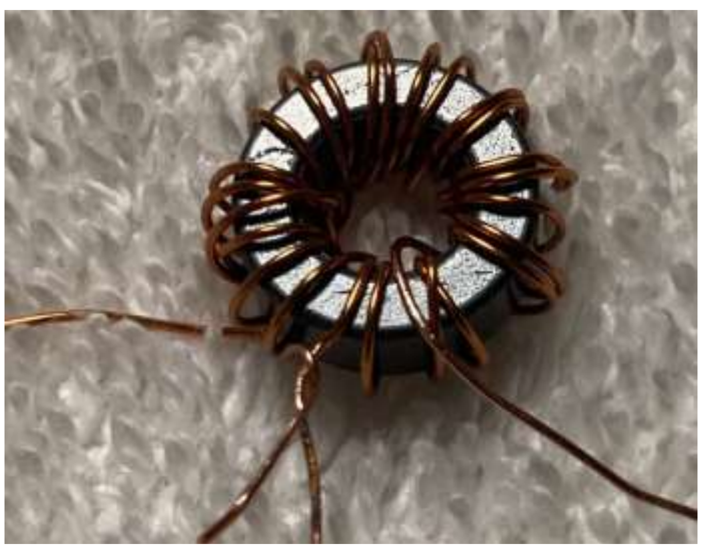


Figure 4-3. Finished toroid.

Notice that the ends of the 15 turns are still twisted together in Figure 4-3. This is to ensure that the proper leads are placed in the proper holes on the PCB. Figure 4-4 shows where those leads should be placed.

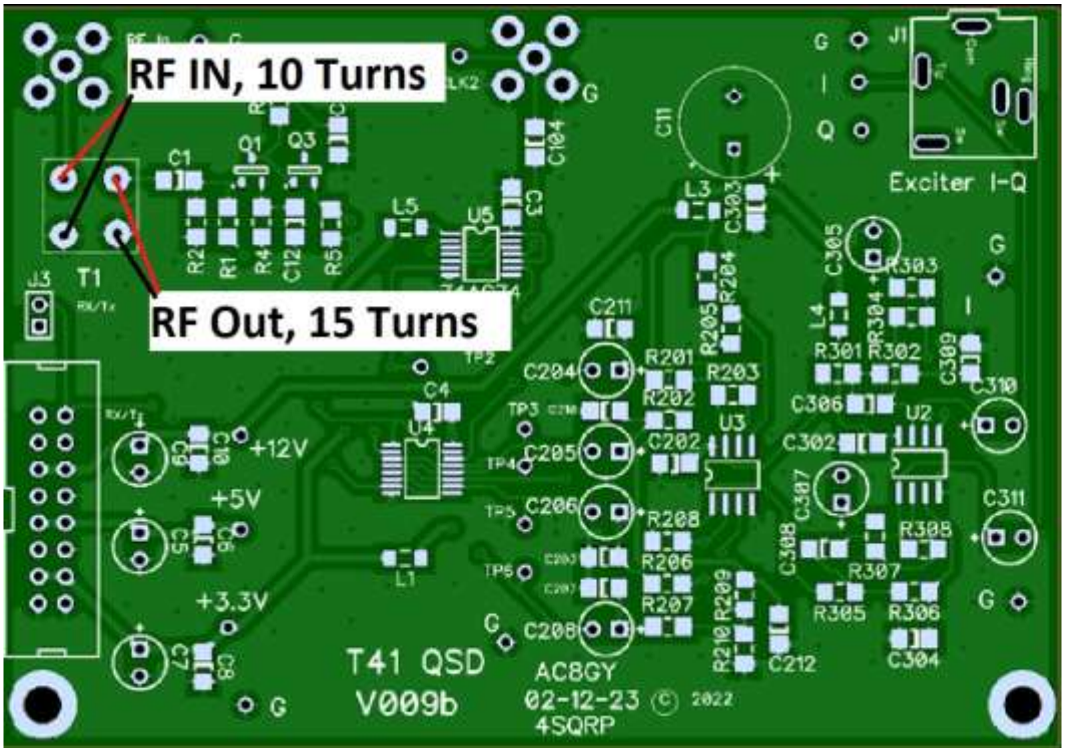


Figure 4-4. Mounting the toroid.

It has been brought to our attention that some builders have had issues with 10 Meters.
Some of the QSD boards will work fine, while others won't receive or calibrate.
The suggested fix is to change U5 to the part number listed in the link below.
They are available from several sources, but the one I used is listed below.

https://www.digikey.com/en/products/detail/nexperia-usa-inc/74LVC74APW-118/946696

Modification of the QSD Board for Calibration

The version V010/V011 QSD boards require a simple modification for the calibration process to work. This modification changes the enable signal to the Tayloe demodulator chip to be always ON. Version SDTVer050 software (and higher) control receiver enable via the Si5351 PLL IC (receiver LO is disabled during transmit).

A trace on the QSD PCB is cut, and a 0Ω jumper is added to the circuit. See Figure 3.

The modification allows the Receive circuits to operate at the same time as the Transmit circuits. This requires breaking a trace on the receive board and adding a shunt 0Ω resistor as shown in Figure 10-3.

NOTE:

Some builders choose to install two male connectors across these cuts so that the modifications can be restored after the calibration is finished. If you choose to do this just short or open the correct trace to Calibrate and replace the shorting pin caps when finished.

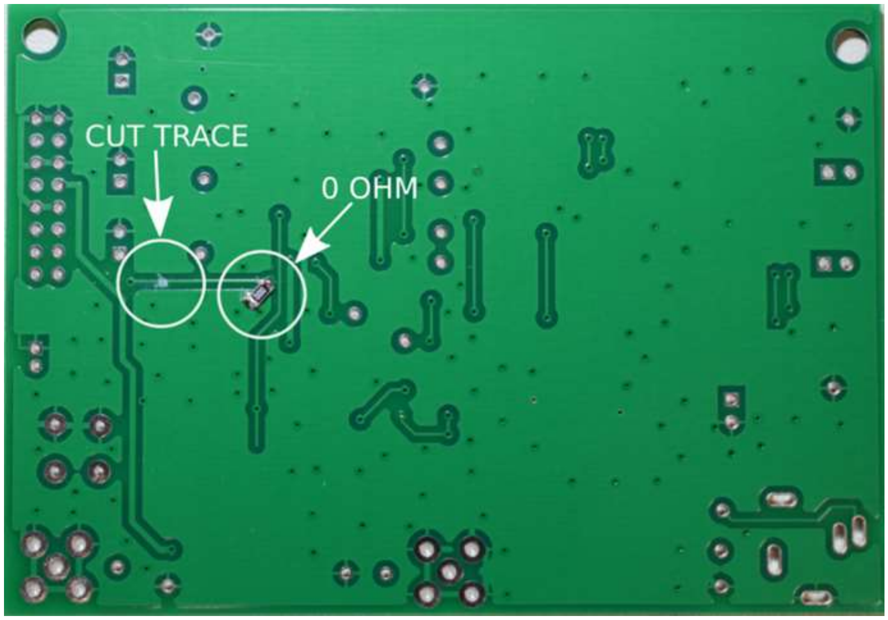
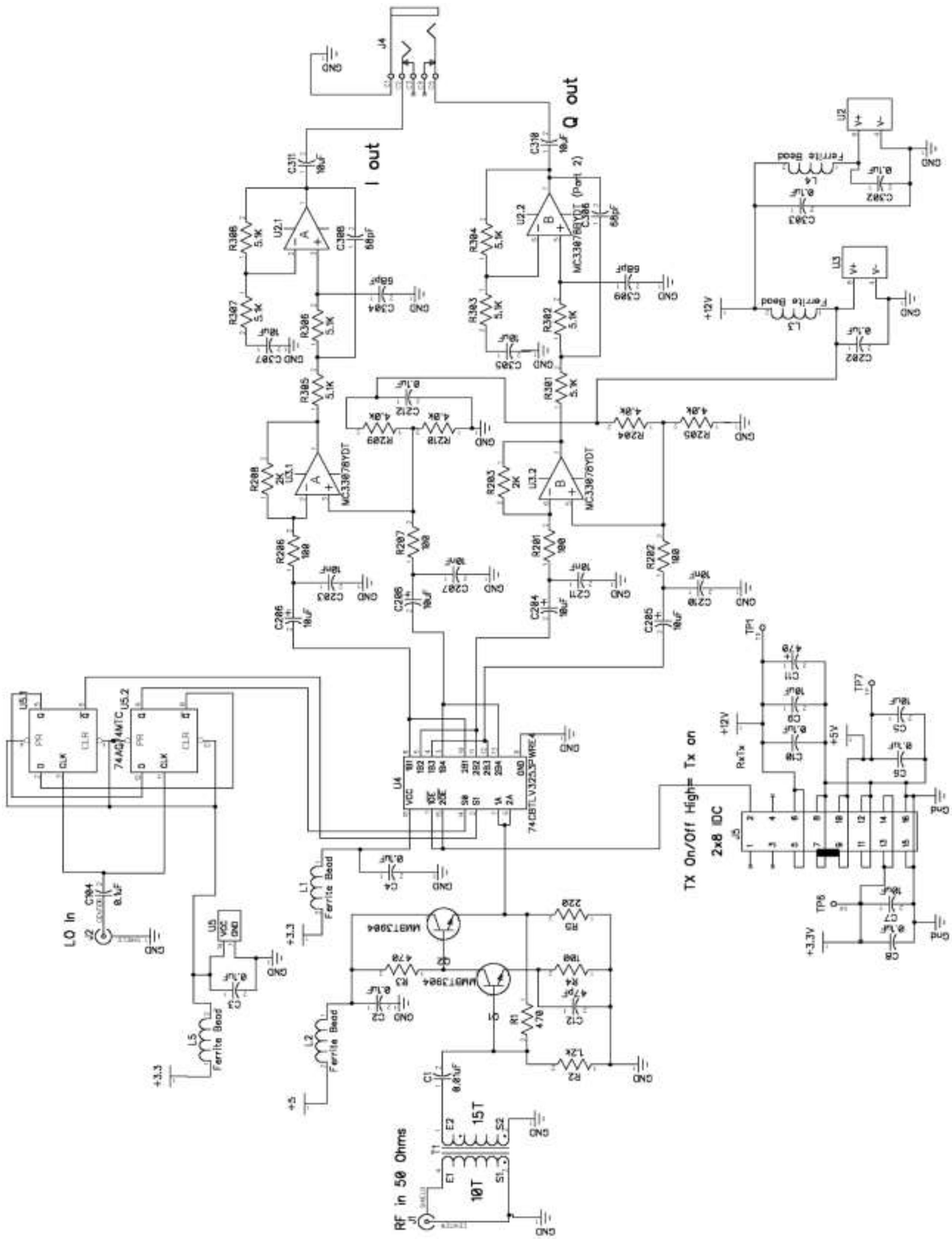


*Figure 10-3. QSD board modifications (V10 and V11 boards)***.**

This completes the QSD board.


I out
Q out
LO In
RF in 50 Ohms
TX On/Off High= Tx on
2x8 IDC
+12V
+5V
+3.3V

Section 5 Exciter Board

☐ Inventory the components in the bag. It should contain:

Table 5-1. Exciter board BOM.

	Qty	Ref	Description	√
1	6	C2, C6, C14, C17, C22, C29	10uF Electrolytic capacitor	
2	4	C4, C7, C12, C16	100uF Electrolytic Capacitor	
3	1	J1	2x8 Male Box Header	
4	1	J2	3.5mm audio Jack	
5	2	J3, J4	SMA	
6	1	J5	2 pin pin header	
7	1	T1	0553-0013-BC-F	

NOTE: Please wait to install the SMA connectors J1 & J2 until the boards have been mounted on the chassis. This will allow you to customize your cable routing.

Inventory the items, checking off each as you identify them. If you are missing any parts, please contact the kitter listed at http://www.4sqrp.com/T41main.php, and the item will be promptly sent to you.

☐ Many of the components on this board are polarized, meaning they must be inserted properly in the board to function. In every case, there is a marking on the PCB that indicates the proper orientation of the component. Double check each part before inserting and soldering.

☐ Starting at the top of the BOM list, go down the list, installing components into the PCB the capacitors through connectors. Pay attention to the polarity of the capacitors and the position of the key for the IDC connector.

☐ Locate the transformer. Depending on the manufacturer it may or may not have pins 4 present. There is not a corresponding hole in the PCB for the pin, so if the transformer has pin 4, snip it off flush to the body of the transformer.

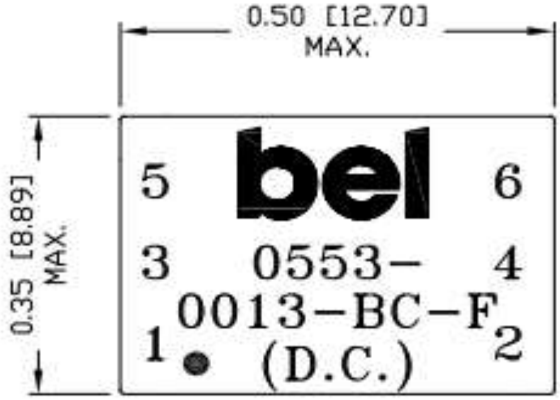


Figure 5-1. Transformer connections.

☐ Go down the rest of the list of components, installing them one at a time.

Note: *The labels for J3 and J4 are reversed on the schematic.*
Clk_1 is at the top of the board and RF out is at the bottom of the board.

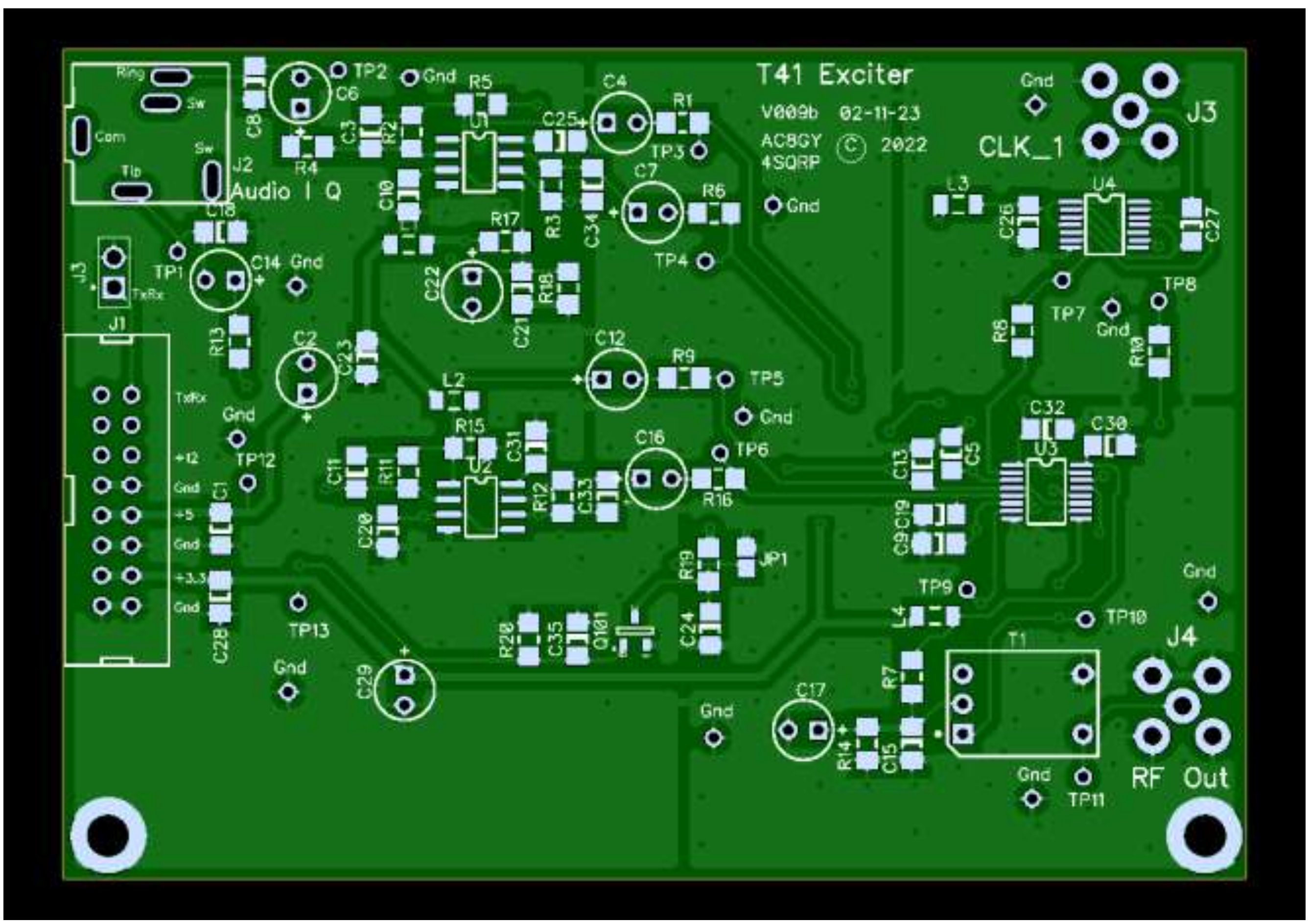
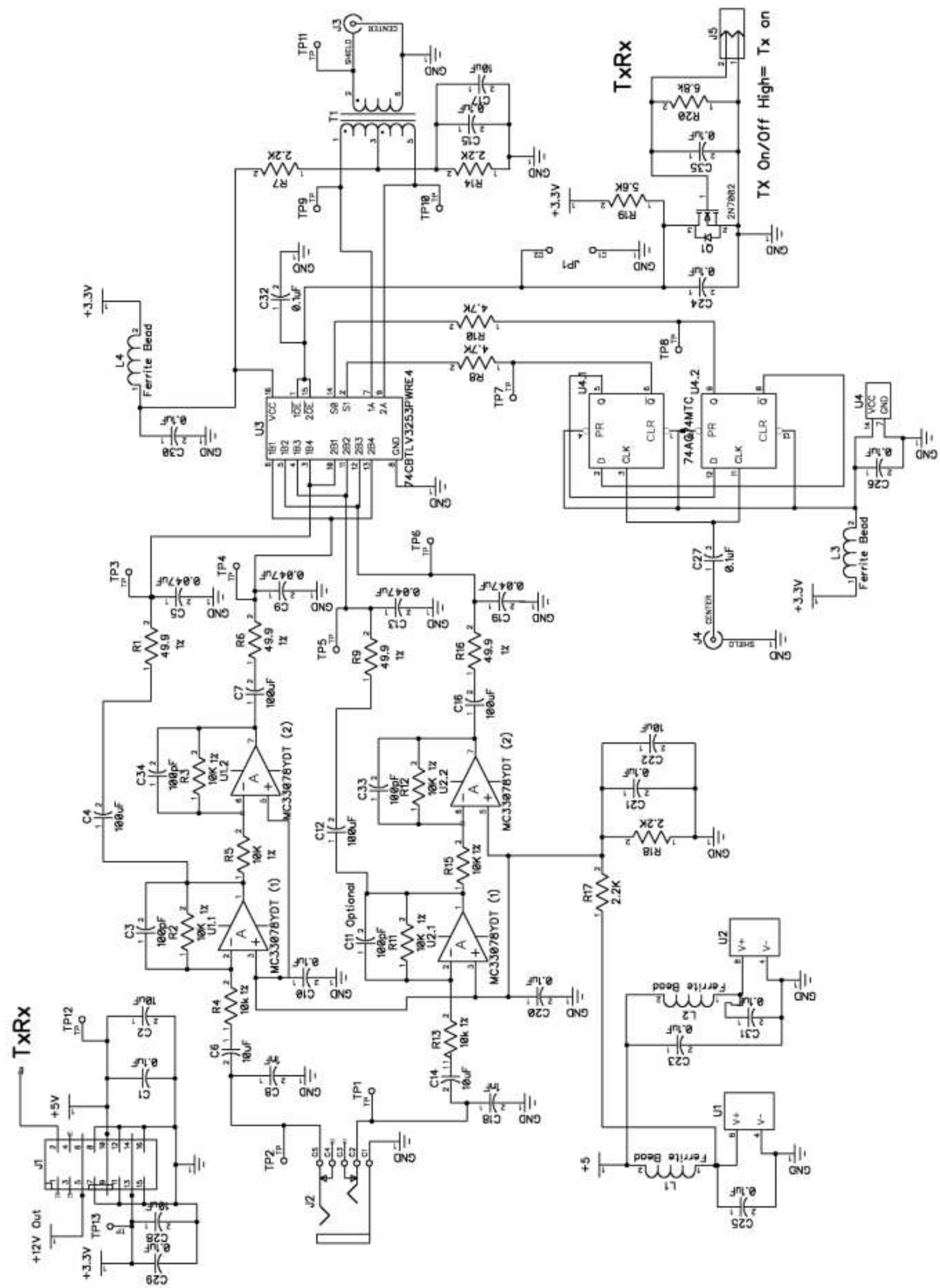


Figure 5-2. Component Placement


TxRx
TX On/Off High= Tx on
74CBTLV3253PWRE4
74AC74MTC
MC33078YDT
2N7002
Ferrite Bead

Section 6: Power Amplifier

☐ Inventory the components in the bag. It should contain:

Table 6-1. BOM for PA board.

		Qty	Reference	Description	√
1		1	C11	10uF electrolytic	
2		1	C301	100uF electrolytic	
3		2	J1, J2	SMA connector	
4		1	J8	DC terminal block	
5		2	J3, J7	2.0mm 2-pin header	
6		1	JP1	2.54mm 2-pin header	
7		3	Q1, Q4, Q302	2N3906	
8		2	Q2, Q3	2N3904	
9		4	Q101, Q102, Q201, Q202	IRF510PBF	
10		1	Q301	TIP127	
11		1	Q303	IRF5305	
12		4	R100, R102, R200, R203	10k pot	
13		2	R202, R205	240, 1W	
14		6	R300a-f	1 ohm, 1W	
15		1	T1	BN43-202	
16		1	T2	FT50-43	
17		1	T3	BN43-202	
18		1	T4	FT50-43A	
19		1	T5	BN43-3312	
20		1	U1	LM7815CT	
21		1	U2	LM317T	
		Hardware Components			
		Qty	Description		√

22		4	heat sink pad	
23		4	shoulder washer	
24		4	M3.5x12 screw	
25		1	heat sink	
26		1	heat sink fan assy	
27		4	M3.5x12	
28		2	Nylon M4x10 Screw	
29		4	6-32 x 1/2 screw	
30		1	6/32 x 5/16 screw	
31		5	6-32 nut	
32		4	6 - 1/8" spacer	
33		1	Pin Header Jumper	
34		8'	AWG 26 magnet wire	
35		3'	AWG 22 magnet wire	
36		1	1" square closed cell foam	

Inventory the items, checking off each as you identify them. If you are missing any parts, please contact the kitter listed at http://www.4sqrp.com/T41main.php, and the item will be promptly sent to you.

☐ Many of the components on this board are polarized, meaning they must be inserted properly in the board to function. For these parts, there is a marking on the PCB that indicates the proper orientation of the component. Double check each part before inserting and soldering.

Components

☐ Install electrolytic capacitors C11 and C301 at their respective positions on the board, paying attention to their polarity. Flip the PCB and solder, strum, and trim the leads.

☐ NOTE: *The SMA connectors can be soldered on this board. They will be mounted on the same side as the components.*

☐ Solder the J8 terminal block at its location with the openings facing away from the PCB. Flip the PCB and solder in place.

☐ Locate the two conductor 2.0mm cable, and mate it with the two white 2.00mm, 2-pin headers.

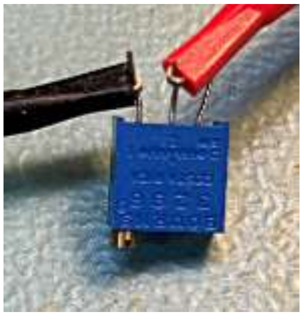
☐ Paying attention to the position of the red wires of the cable, solder the headers to the board in the J3 and J7 locations, with the red wire corresponding to the ‘+’ marking of the silk screen. Remove the cable when done.

☐ Install Q1, Q2, Q3, and Q4 at their locations on the board making sure the flat side of the transistors match the silk screen flat side. Flip to the backside of the PCB, solder, strum, and trim.

☐ Skip the installation of Q101, Q102, Q201 and Q202 for now.

☐ Install the trim pots (blue boxes with brass screw) R100, R102, R200, R203 at their locations. Make sure the brass screw matches the screw's location on the silk screen.

NOTE: *before installing these trim pots, connect an ohm meter to the middle and left leg of the pot, with the adjustment screw on the left. (see Picture below) Turn the pot to the left, or counterclockwise till the resistance is neat zero. This will preset them for Amp adjustments later in the instructions. Flip the PCB and solder.*



10 k Pot connections

☐ The 1 watt power resistor leads must be folded to be installed vertically. The six 1 ohm resistors (brown, black, gold) are inserted between adjacent buses in the R300 region. Figure 6-1 shows the resistor grouping. We prefer to mount them with the color bands near the top of the resistor. This has *no* effect on the circuit. We just think it looks neater if all bands are oriented the same. (It's an OCD thing…) Note how they connect to two separate busses on the board. Of the 4 buss traces, two are connected to each other which forms a "rail" for the resistors. In other words, we are connecting the six 1Ω resistors in parallel. Ohms law tells us that six 1Ω resistors in parallel have a total resistance of 0.167Ω. If you measure the resistance between the two test points shown in Figure 6-1, you should see about 0.167Ω.

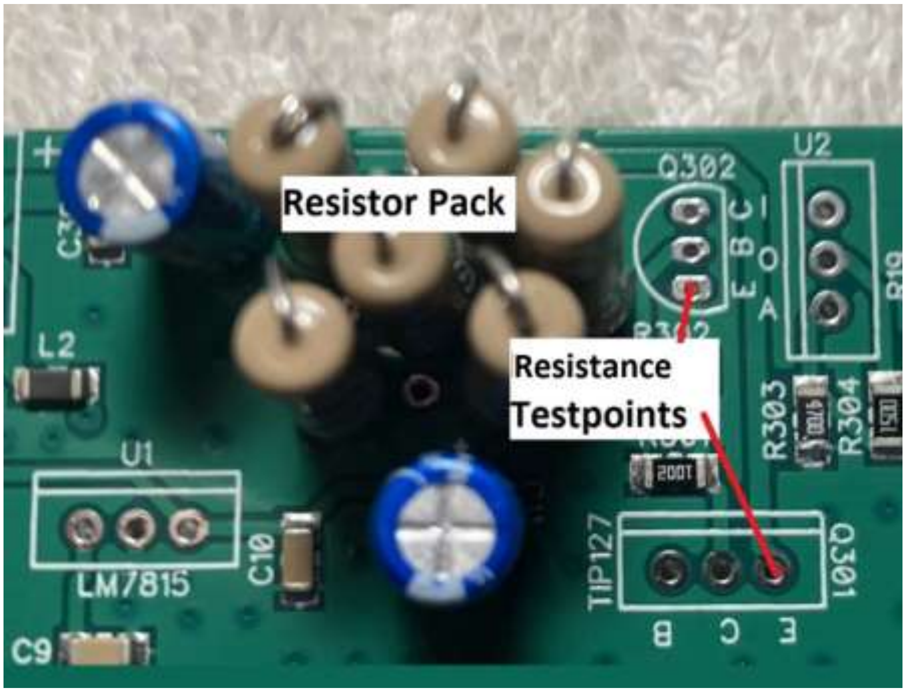
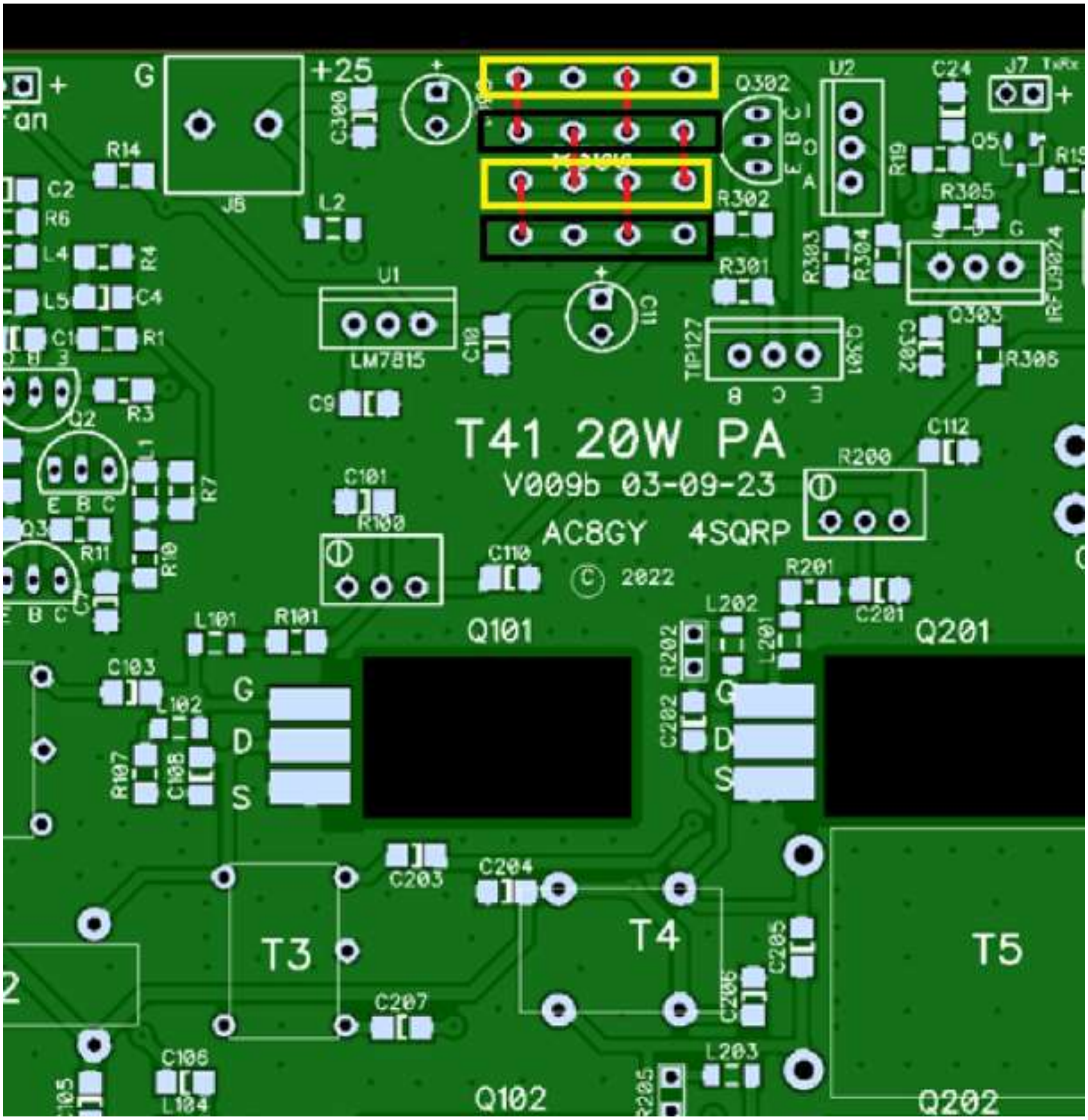


Figure 6-1. The 1Ω resistor group.



☐ Install the two 220Ω (red, red, brown) resistors R202, R205 at their locations. These must also be vertically oriented. A few boards seemed a bit “clogged”. You can clear a clog with a sharp metal tool (e.g., dental pic, darning needle, solder pic, etc.) and gently ream the hole. Another option is secure the board in a bench vice, heat the clogged connection from the backside, and push the leads through from the front side. Once mounted, flip to the backside, solder, strum, and trim.

☐ Attach the heat sink to U1, the LM7815, using a 6-32 x 5/16” screw and nut. In Figure 6-1, you can see the two parallel lines just under the U1 silk screen. These two closely-spaced lines represent the tab on the regulator. Make sure you orient the regulator to match the silk screen.

☐ Install U1 and U2 on the PCB. Again, pay attention to the tab silk screen marking on U2. Flip to the backside of the PCB and solder, strum, and trim the leads for both regulators.

Transformers

Table 6-2: Transformer Windings

Ref	Core	Primary	Secondary	AWG22	AWG26
T1	BN43-202	4T 22 AWG	3T 26 AWG, bifilar	8"	2x 6.5"
T2	FT50-43	10T 26 AWG, bifilar			2x 8.5"
T3	BN43-202	4T 22 AWG	3T 26 AWG, bifilar	8"	2x 6.5"
T4	FT50A-43	10T 26 AWG, bifilar			2x 12.5"
T5	BN43-3312	2T 22AWG	3T 22 AWG	7.5", 9.9"	

☐ Locate the BN43-202 cores. These are the two smaller binocular cores. Cut two lengths of 26AWG wire, each 6 ½” long.

☐ Place two lengths of the wire parallel, and manually twist the lengths together approximately one full twist per inch. We find it easier to twist the two wires if you can secure one end in a small, hobby-type, bench vice. (Useful for holding PCBs, too.) See Figure 6-2. If you don’t have a small bench vice, vice grips also work. Neither tool is required; it just makes it easier.

Another way to twist the wires is to use a variable speed electric drill. Clamp one end of the two wires in the drill chuck. Using a pair of plyers hold the other end of the wires tightly and use the drill to twist the wires. Use a very low speed to do the twisting.

Figure 6-2. Hobby bench vice.

☐ Pass this wire through one of the binocular cores and wind it through, passing through each aperture three times. Both ends should begin and end at the same side of the core, with roughly the same length extending at beginning and end of the winding.

☐ Cut a length of 22AWG wire 8” long. Wind it through the core four times.

☐ Unwind the twisted pair conductors extending from the side of the core and spread them per the illustration below. Strip the enamel insulation from the wire to about ¼” from the body of the core.

☐ Using a multimeter to check the continuity of the secondary windings. There should be continuity between A and C, and between B and D. (See Figure 6-3.) There should be no continuity between A and B or between C and D. If this is not the case, rotate wires C and D so that they exchange position.

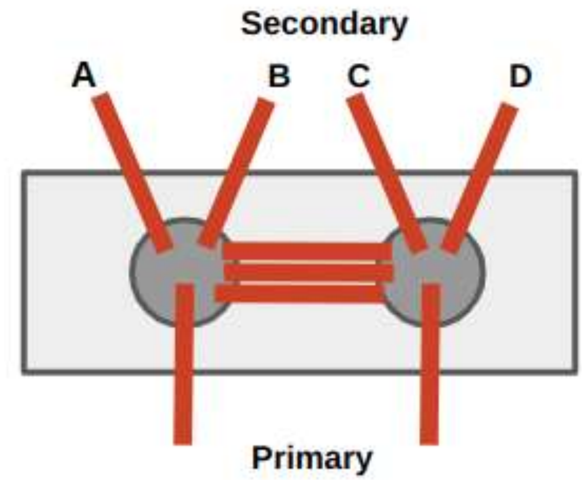


Figure 6-3: T1 and T3 construction detail.

☐ Place the wires through the holes on the PCB for T1 so that the body of the core extends vertically from the surface of the board. Wires B and C pass through the same hole on the middle of the Secondary side connections. Pull the wires snug, and solder into place. Trim excess wire length flush with the PCB.

☐ Repeat this process for T3.

☐ Locate the toroidal FT50-43 core. It is the smaller of the two toroid's. Cut two, 8 ½" lengths of 26AWG wire. Place the two lengths together, and twist together roughly two full turns per inch.

☐ Wind the twisted pair of wires through the toroid, passing through the center of the core ten times. Pull each turn snug against the core so that there is no gap between the core and wire.

☐ Unwind the twisted pair back to the body of the toroid and strip the enamel roughly ¼" back from the ends. Spread the ends per the diagram below.

☐ Check the continuity of the windings. There should be continuity between A and C and between B and D. (See Figure 6-4.) Verify there is no continuity between A and D or between B and C.

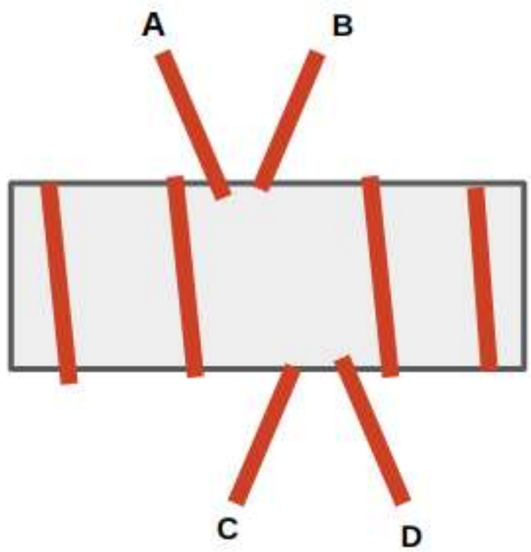


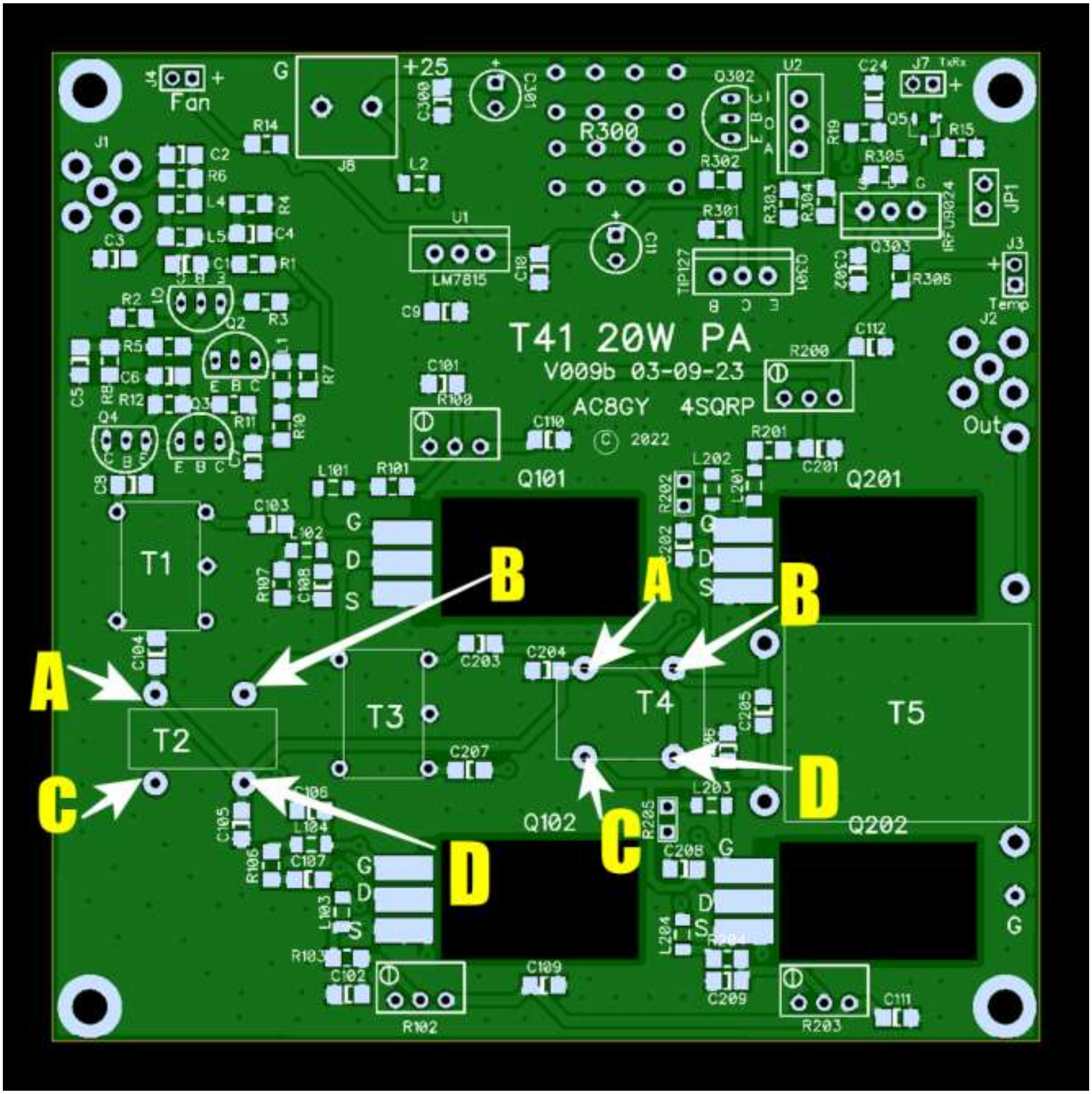
Figure 6-4: Toroid winding diagram

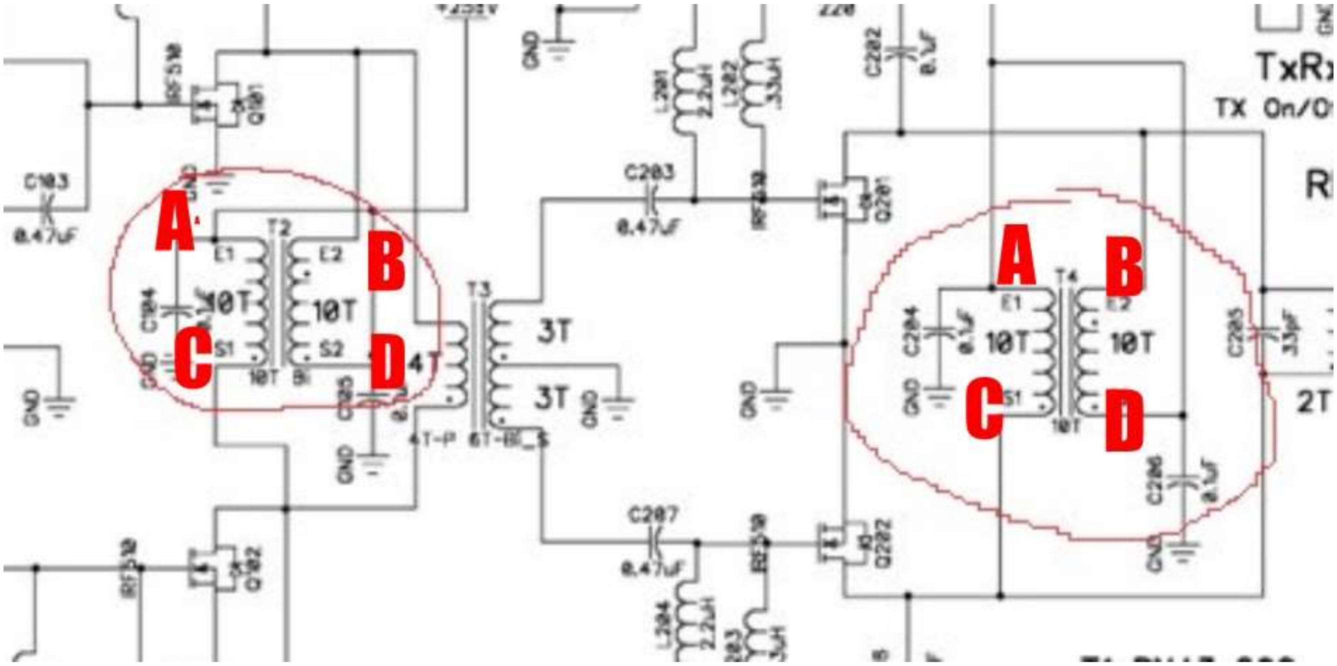
☐ Insert the stripped wires through the holes for T2 in the PCB, making sure that A and C are on the Left side and B and D are on the right. If either pair is reversed, you will not be able to set the idle current on the output transistors. Pull the wires snug, and solder into place.
Lead A and C must be landed on the two upper holes, and Lead C and D must be landed on the two bottom holes. This APPLIES to both T2 And T4.

☐ Locate the larger FT50A-43 toroid used for T4. Cut two lengths of 26AWG wire 12.5" long.

☐ Repeat the fabrication steps used for T2.

☐ Locate the large binocular core used for T5. Using the 22 AWG wire, cut one 7.5" and one 9" piece of wire. Wind 2 turns on the core for the transformer primary using the 7.5" wire. Starting from the other side of the core, wind 3 turns on the core. When you are done winding the transformer, you should have two leads protruding from either side of the T5 core.


T41 20W PA
V009b 03-09-23
AC8GY 4SQRP
© 2022
T1
T2
T3
T4
T5
Q101
Q102
Q201
Q202
A
B
C
D
A
B
C
D
Fan
+25
Out
Temp
TxRx
LM7815
TIP127
IRFU9024
R300



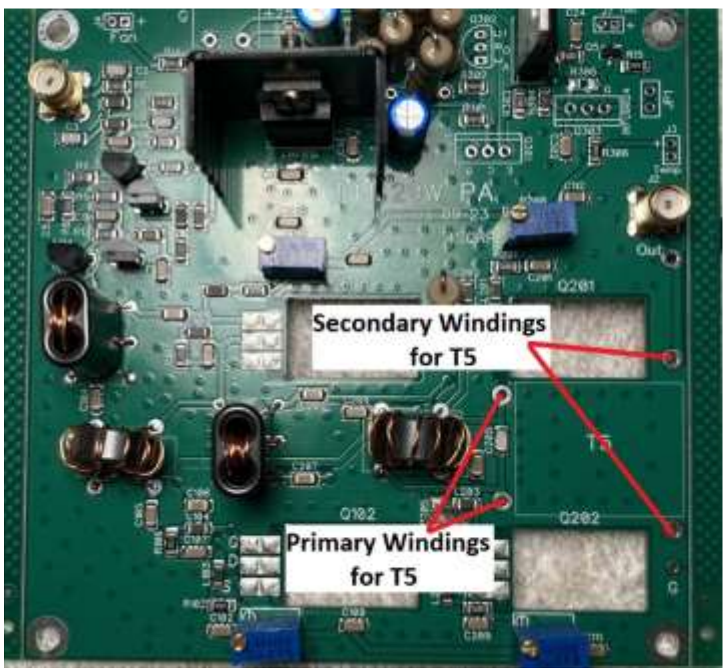


Figure 6-5. Partially completed PA PCB.

☐ When finished, remove ¾” of enamel insulation from the primary and secondary leads.

☐ Thread the two leads for the primary winding of T5 through the two holes nearest the center of the PA PCB. Thread the other two leads from the secondary winding of T5 through the two holes closest to the edge of the PCB. See Figure 6-5.

Other Components

☐ Mount the Q302 transistor (2N3906) at its location to the right of the resistor pack. Note its orientation. Solder, strum, and trim.

☐ Mount the U1 and U2 (LM7815CT, LM317T) voltage regulators at their locations as silk screened on the PCB. Make sure the tab is aligned with the tab outlined by the silk screened on the PCB. Solder, strum, and trim.

☐ Mount Q303 (IRF5305) on the PCB, again paying attention to the position of its grounding tab. *NOTE*: Some boards have this transistor silk screened on the board as IRFU9024. Due to supply chain problems, the IRFU9024 became hard to locate, so the IRF5305 is used in its place. Solder, strum, and trim.

☐ Mount the two plastic cable connectors at locations J4 and J7. The leys should face away from the PCB. Solder in place.

☐ Mount connector J8 at its location and solder in place.

☐ Solder two header pins at location JP1 and solder in place. Take the 2-pin plastic jumper and temporarily place it on 1 pin of JP1. This does not jumper the connection but keeps the plastic jumper at a known location. This jumper is used during the calibration of the T41.

☐ All components are now in place on the PA board. Set the board aside, along with the unused hardware, until the chassis assembly is started.

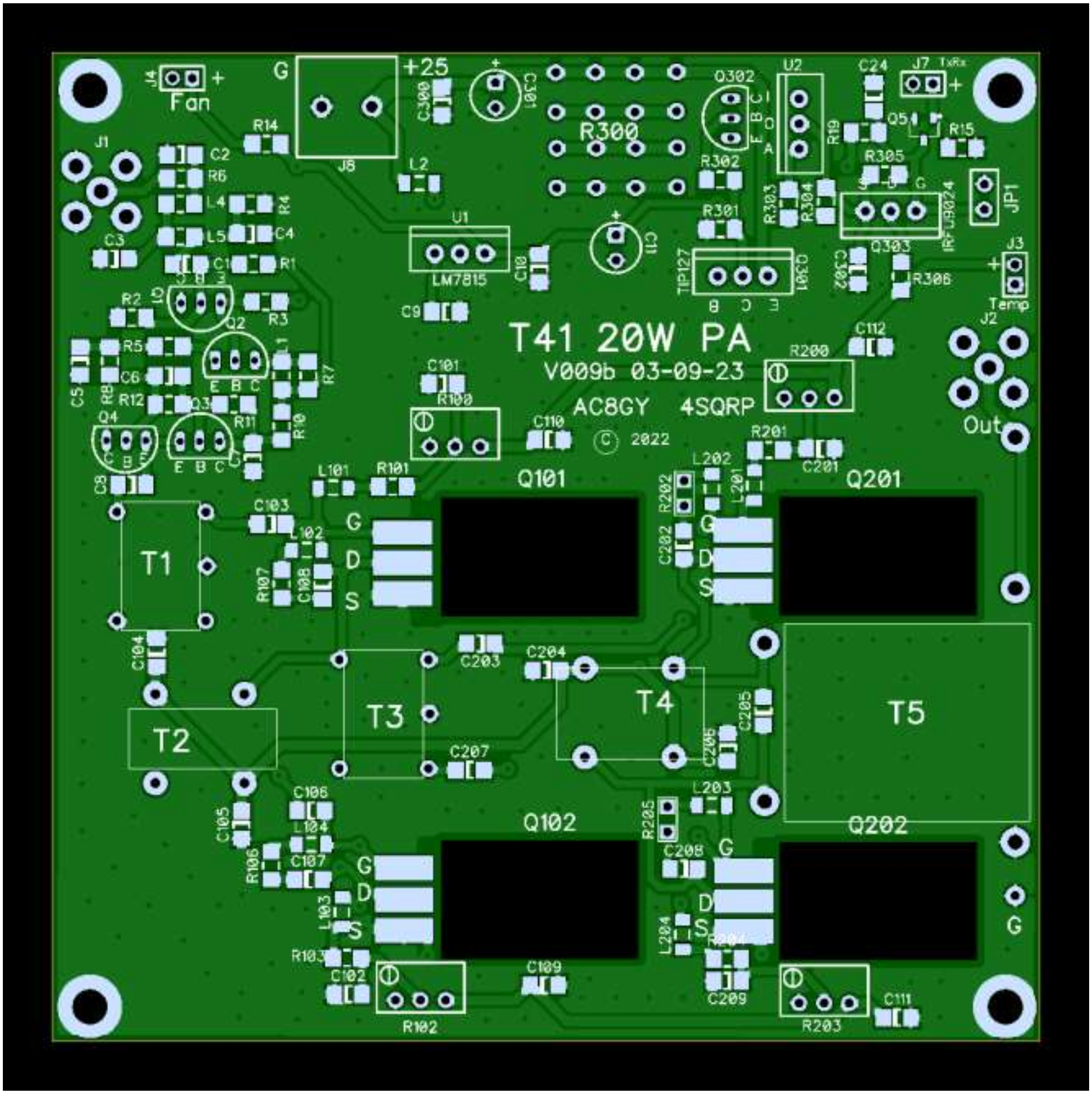
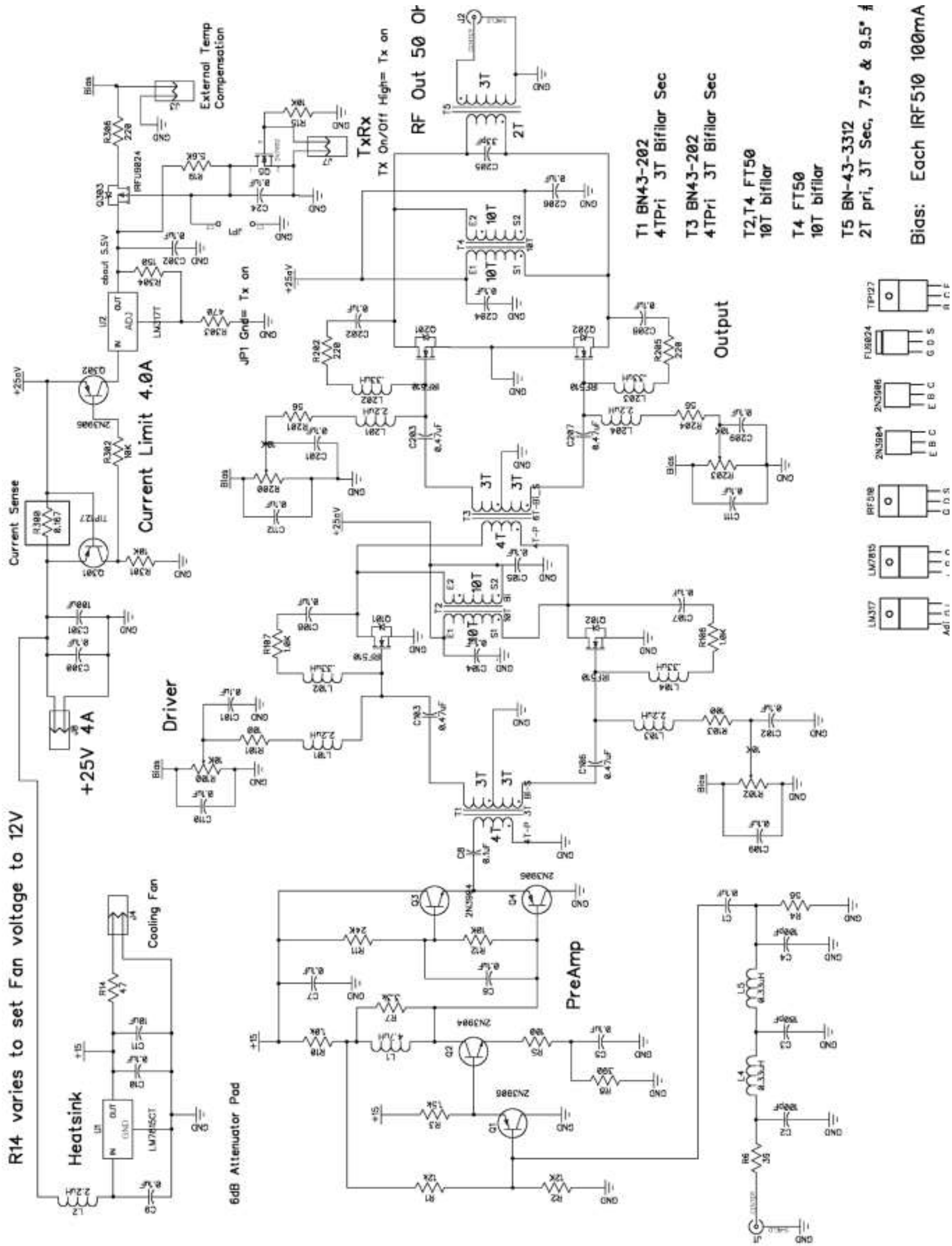


Figure 6-6. The PCB.


R14 varies to set Fan voltage to 12V
Heatsink
Cooling Fan
6dB Attenuator Pad
PreAmp
+25V 4A
Driver
Current Sense
Current Limit 4.0A
External Temp Compensation
JP1 Gnd= Tx on
TxRx
TX On/Off High= Tx on
RF Out 50 Oh
Output
T1 BN43-202
4TPri 3T Bifilar Sec
T3 BN43-202
4TPri 3T Bifilar Sec
T2,T4 FT50
10T bifilar
T4 FT50
10T bifilar
T5 BN-43-3312
2T pri, 3T Sec, 7.5" & 9.5"
Bias: Each IRF510 100mA

Section 7: Filter and Relay Boards/Daughter Board

☐ Inventory the components in the bag. It should contain:

Table 7-1. The BOM for the Filter and Relay PCBs.

	Qty	Ref	Description	√
1	5	D1, D2, D3, D4, D5	1N4148	
2	1	J1	2x8 Male box header	
3	3	J2, J6, J9	SMA	
4	1	J3	2x5 Male Box Header	
5	5	K1, K2, K3, K4, K5	Relay	
6	12	L1, L2, L3, L4, L5, L6, L7, L8, L9, L11, L12, L13	T37-6 (yellow core)	
7	5	L14, L15, L16, L24, L25	T68-2 (red core)	
8	5	Q1, Q2, Q3, Q4, Q5	BS170 transistor	
9	10		4 pin female header	
10	1		40 pin header	
11	20 feet		AWG 26 wire	

NOTE: It's a good idea to install the SMA connectors during the final assembly.
It will make the cable routing easier.

It has been suggested that the order of mounting the components on the filter board is easier if you install them in this order:

1. Diodes
2. Transistors
3. IDC headers
4. Relays
5. Inductors

The small to large order makes it easier to install the parts since the board can lay flat during assembly.

Inventory the items, checking off each as you identify them. If you are missing any parts, please contact us at the address posted in this document, http://www.4sqrp.com/T41main.php, and the item will be promptly sent to you. As will see in the figures below, there are two PCB boards for this part of the assembly. The first board is the relay, or main filter board, and is the larger of the two boards. It is easily identified by the three SMA and five relay silk screens in Figure 7-9. The second smaller toroid board is where most of the toroid's are mounted.

Winding the Toroid's

☐ The first thing you need to do is wind the toroid's for the filters. Locate the toroid cores and magnet wire. We suggest you start with L1 in Table 7-1 and proceed straight down the list. Note that the first 13 cores are all the smaller (yellow) cores. We also suggest you scrape ½" of the enamel off one end of the wire and use this as the starting end of the toroid. (It's easier to scrape enamel off before you start winding the toroid.) Use fine grit sandpaper or the edge of a box cutter to remove the enamel. If using a box cutter, take care not to nick the wire.

☐ Before winding each core, cut the length of wire listed in Table 7-2 for that part. Start winding the toroid with about 3/4" of wire free before passing the other end through the center of the toroid core. (This 3/4" length of free wire includes the ½" of enamel-free wire.) Count each turn as the wire passes through the center of the core. Pull the wire snug. You want the wire tight enough that there is very little gap between the wire and the core. However, you want it loose enough that you can slide the windings on the core. That is, you need to be able to distribute the windings on the core evenly around the core. See Figure 7-1.

Table 7-2. Winding data for the toroid's.

Reference	#turns, core	Length of wire, inches	√
L1	11t T37-6	7.5	
L2	13t T37-6	8.5	
L3	10t T37-6	7	
L4	12t T37-6	8	
L5	14t T37-6	9	
L6	12t T37-6	8	
L7	16t T37-6	10	
L8	17t T37-6	10.5	
L9	16t T37-6	10	
L11	22t T37-6	13	
L12	24t T37-6	14	
L13	22t T37-6	13	
L14	21t T68-2	19	
L15	23t T68-2	20.5	
L16	21t T68-2	19	
L24	25t T68-2	22	
L25	25t T68-2	22	

☐ When finished winding the required turns on the core, count the turns again…just to make sure. When you are satisfied that the turn count is correct, leave another 3/4" or so extra wire and trim off any excess. Now carefully remove the enameled insulation from the other end of the wire using fine-grit sandpaper or scrape the insulation off with an Exacto or box cutter. Use the silk screens on the PCBs to find the location of each toroid. (L24 and L25 are on the main filter board with the relays.)

☐ Holding the two trimmed and scraped ends between your fingers, use a small tool (e.g., toothpick, awl, fingernail?) to spread the turns out so they are evenly distributed on the core. Your toroid should look similar to the toroid in Figure7-1

Figure 7-1. Sample toroid.

☐ Insert the wires for each inductor through the appropriate holes in the board. Flip the board to the backside, then snug them to the PCB and then solder. Closely inspect the solder joints afterward to be certain the enamel wire has wetted with solder completely. We always check with an ohmmeter which should show zero resistance between the two toroid ends. Reheat and apply more solder if necessary.

When finished, your filter board should look similar to Figure 7-2. Although not required, we usually.

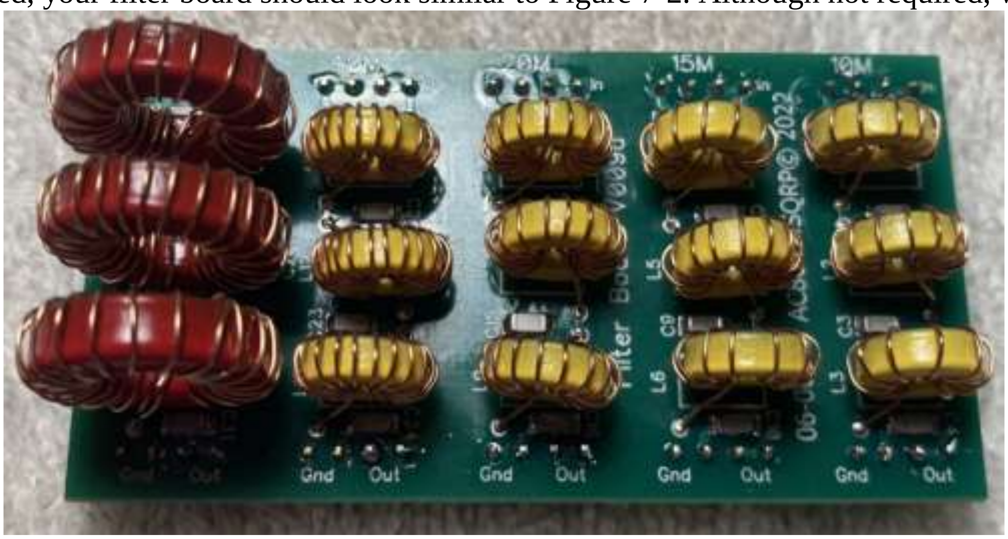


Figure 7-2. Filter daughter board ready to insert into relay board.

secure each coil with a little dab of hot glue to prevent stressing the solder connections.

Main Filter/Relay Board

Relays

☐ Locate the 5 relays and insert them into their positions as indicated on the silk screen of the main filter board. Using a stiff piece of cardboard, cover the relays and flip the PCB to its backside. (The cardboard makes this task easier.) Solder a corner pin on each relay.

☐ Flip the board over and check to make sure the relays are seated flush with the PCB surface. If not, reheat the pin holding the offending relay and push the relay flush to the board. Once all the relays are properly seated, flip to the backside of the PCB and solder all of the relays in place. When finished, your board should look similar to Figure 7-3.

Figure 7-3. Relays mounted on main filter board.

Diodes

☐ Some of the components on the main filter board are polarized, meaning they must be inserted properly in the board to function. In every case, there is a marking on the PCB that indicates the proper orientation of the component. Double check each part before inserting and soldering.

☐ Mount the 1N4148 diodes at their respective places as indicated by their silk screens on the relay PCB. Note that each diode is marked with a band on one end. (See Figure 7-4.) That band should align with the line in the diode’s silk screen on the PCB. When soldering diodes and transistors, don’t dawdle with the soldering iron on the leads as these components are a little fussy about excessive heat. Heat the wire and pad for 2-3 seconds, apply the solder, and after another second, withdraw the soldering iron from the connection. Flip to the backside of the PCB, solder, inspect, strum, and trim the leads.

Figure 7-4. Diode with cathode marking band

Miscellaneous Components

*NOTE:*It is important to place the following components on the board at this time because some solder leads could become inaccessible after the toroid board is mounted.

☐ Locate the 3 SMA connectors and mount the on the main filter board. ***NOTE***: It is easier to access the J2 SMA connector later in the assembly (located on the lower-right edge of the PCB) if it is mounted on the *backside* of the PCB. Solder all 3 connectors in place. However, if you elect to mount the toroid board on the backside of the main filter board, you *must* mount J2 on the topside of the main filter board. Otherwise, the toroid board “covers up” the J2 SMA connector. You may wish to read the section titled *Mounting the Toroid Board* below before deciding.

☐ Locate the 2 IDC connectors and mount them at their silk-screened locations. Make sure their keys face the outer edge of the PCB. Flip to the backside and solder just 1 pin on each connector. Inspect to see that both IDC connectors sit flush on the PCB. If not, reheat the offending pin and push the connector into place. Flip to the backside and solder the remaining pins.

☐ Locate the 5 BS170 transistors and mount them at their locations. Make sure you orient them as indicated by the transistor’s “flat” side as indicated by the silk screen. Flair the two outside legs on each transistor as you mount them so they won’t fall out when you flip the board to the backside. Solder, strum, and trim the leads.

☐ Locate the 2 large toroid cores (T68-2, red) used for L24 and L25. (See Table 7-2.) Wind each core with 25 turns of #26 magnet wire and trim the leads to about 1” in length. Remove the insulation from the ends with a fine-grit sandpaper or scrape the insulation off with a box cutter or Xacto knife. Measure the resistance between the two bare leads. It should show zero resistance. Thread their leads through the mounting holes seen in the silk screen position, Flip the board over and solder both into place. Your completed board should look similar to Figure 6-5. (Note we mounted J2 on the backside of the PCB for our installation.)

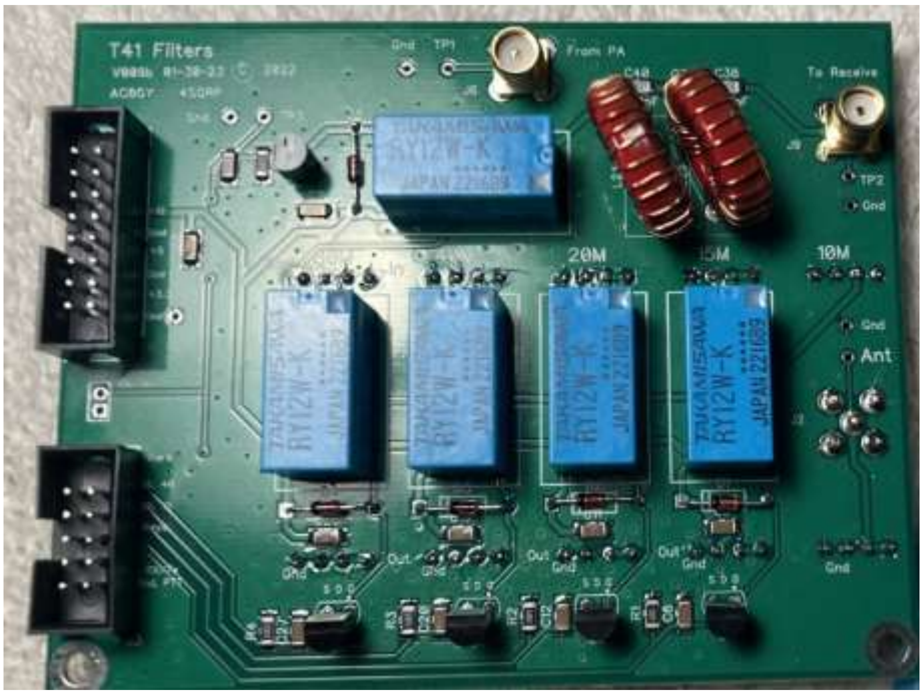


Figure 7-5. Main filter board with components soldered in place.

You may wish to put a blob of hot glue on L24 and L25 to hold them in place.

Mounting the Toroid Board

☐ Snap off 10, four-pin segments from the male header strip. Insert these into the female headers seen in Figure 7-6..

☐ Place the filter relay board face down onto your work surface. Insert the mated header assemblies into their positions on the back of the relay board, female side down, EXCEPT for the pair in the 10M position which should have the male side down.

☐ Place the filter daughter board onto the headers, being careful to assure that the 10M filter section aligns with the 10M headers.

☐ Solder the header pins of the filter daughter card.

Alternative Toroid board mounting

☐ Some builders may wish to experiment with the filters. When we started building the filters, we were pretty certain that the toroid board would change during development. For that reason, we elected to use 4-pin headers (not supplied with the kit) on the main filter board and solder the 4 pin segments to the toroid board. This would allow us to easily separate the toroid board from the main filter board. Figure 7-6 shows the 4-pin headers soldered to the backside of the main filter board.

Using the 4-pin headers makes it much easier to experiment with each band's filter. Indeed, the spacing of the headers is such that QRP Labs' single-band filters can be plugged directly into the headers if you have those filter boards. (The QRP Labs filters work fine, but the T41 filters are designed to work with our filter board.) If you do elect to mount the filter board on the backside of the main filter board, SMA connector J2 *must* be mounted on the topside of the board. Otherwise, the toroid board would cover the SMA connector. It seems that most builders choose to mount the filter board on the back side of the Pc board to allow better access while troubleshooting.

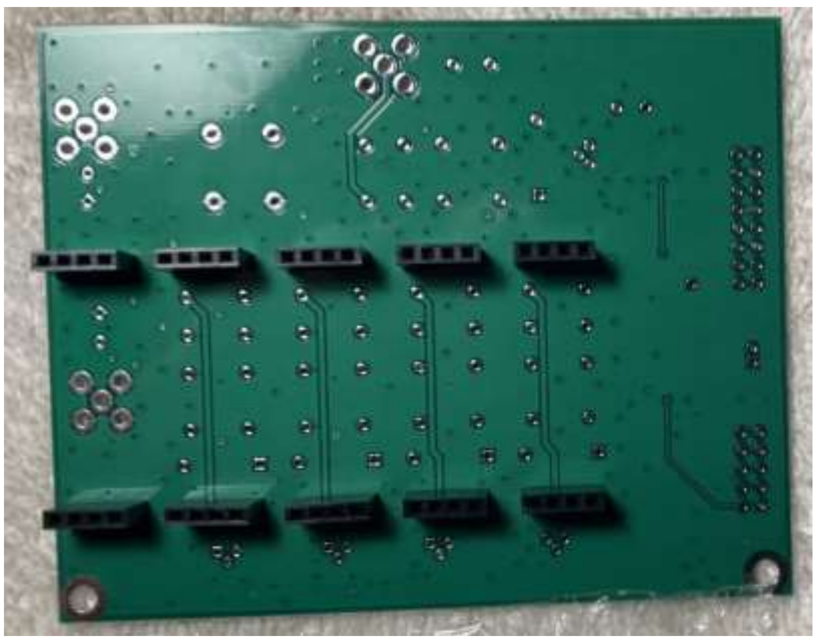


Figure 7-6. 4-pin headers soldered to the main filter board.

These headers could be mounted on the topside of the main filter board, but that would require larger spacing between it and the board mounted next to it in the chassis assembly. We did a "top mount" with an earlier version of the filter board. (See Figure 7-7.) Either mounting method works fine.

Figure 7-7. Early version of filter board.

☐ You should now solder the pins that will hold the toroid board in place. (Either with or without the optional 4 pin headers.)

☐ Inspect each soldered header pin to be certain that the solder quality is satisfactory, and retouch as necessary.

PA Fan

☐ Although originally it was intended to power the PA cooling fan from the regulated 15v on the PA board, a better design solution is to run the PC cooling fan from the system 12v bus. We will pick this up by modifying the Filter Relay board by tacking on the PA fan 2-pin connector onto the back of the Filter Relay PCB, across the power connector +12v and Ground pins.

The final board will look like Figure 7-8 if you mount the toroid board on the backside of the main filter board.

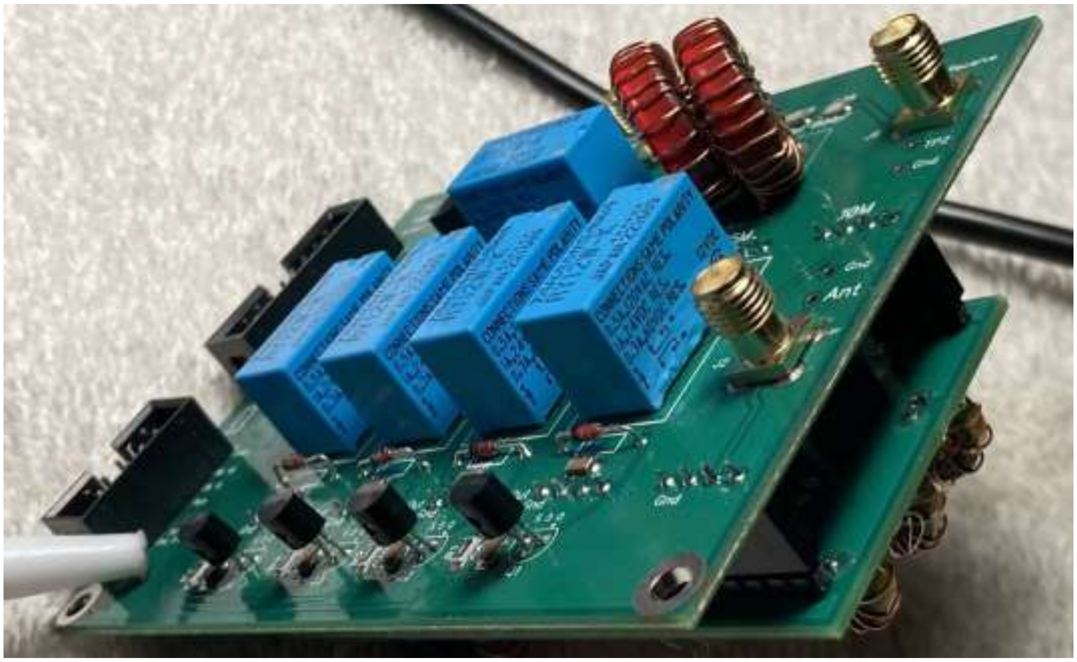


Figure 7-8. Finished board with backside mounted toroid board.

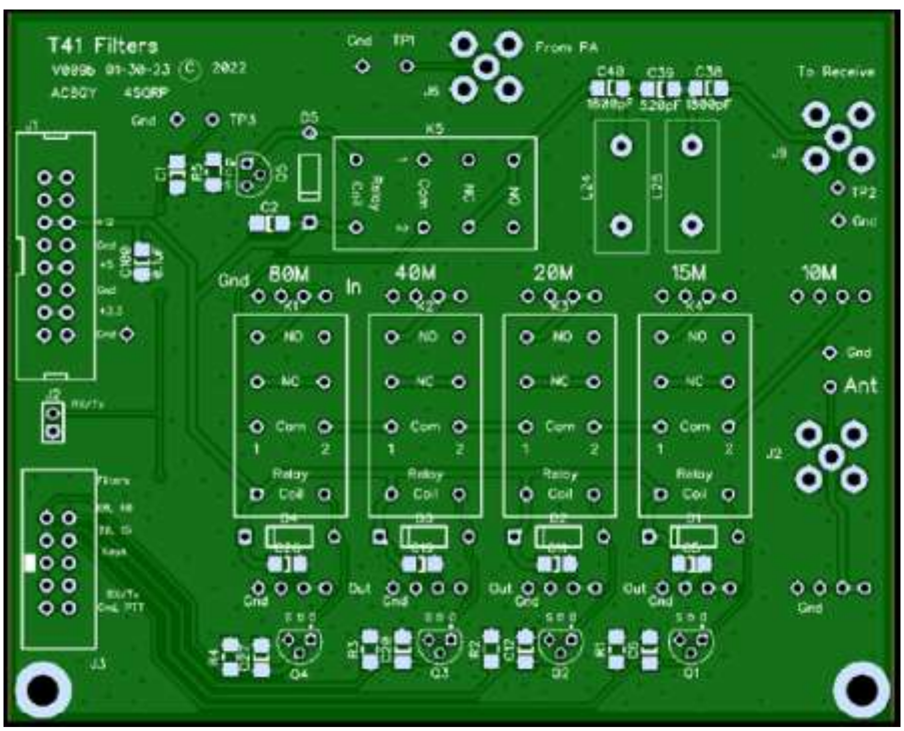


Figure 7-9. Relay filter board.

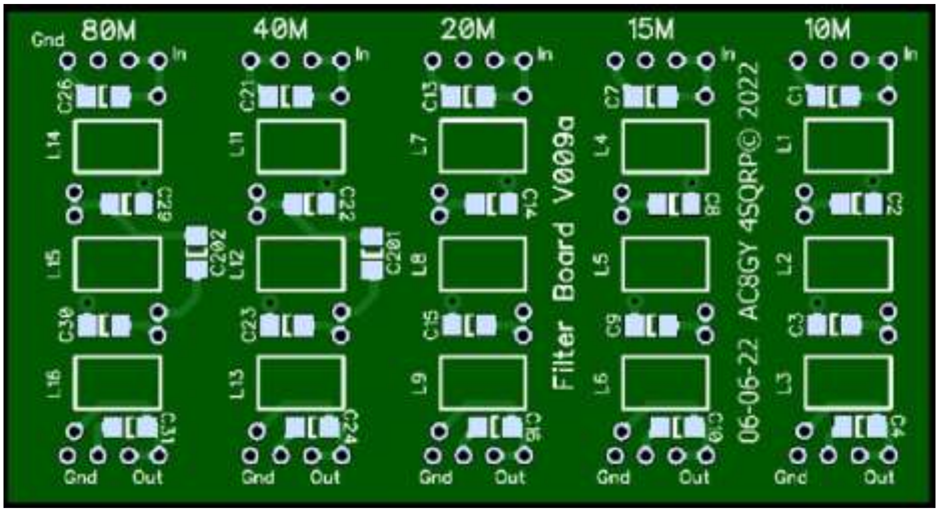
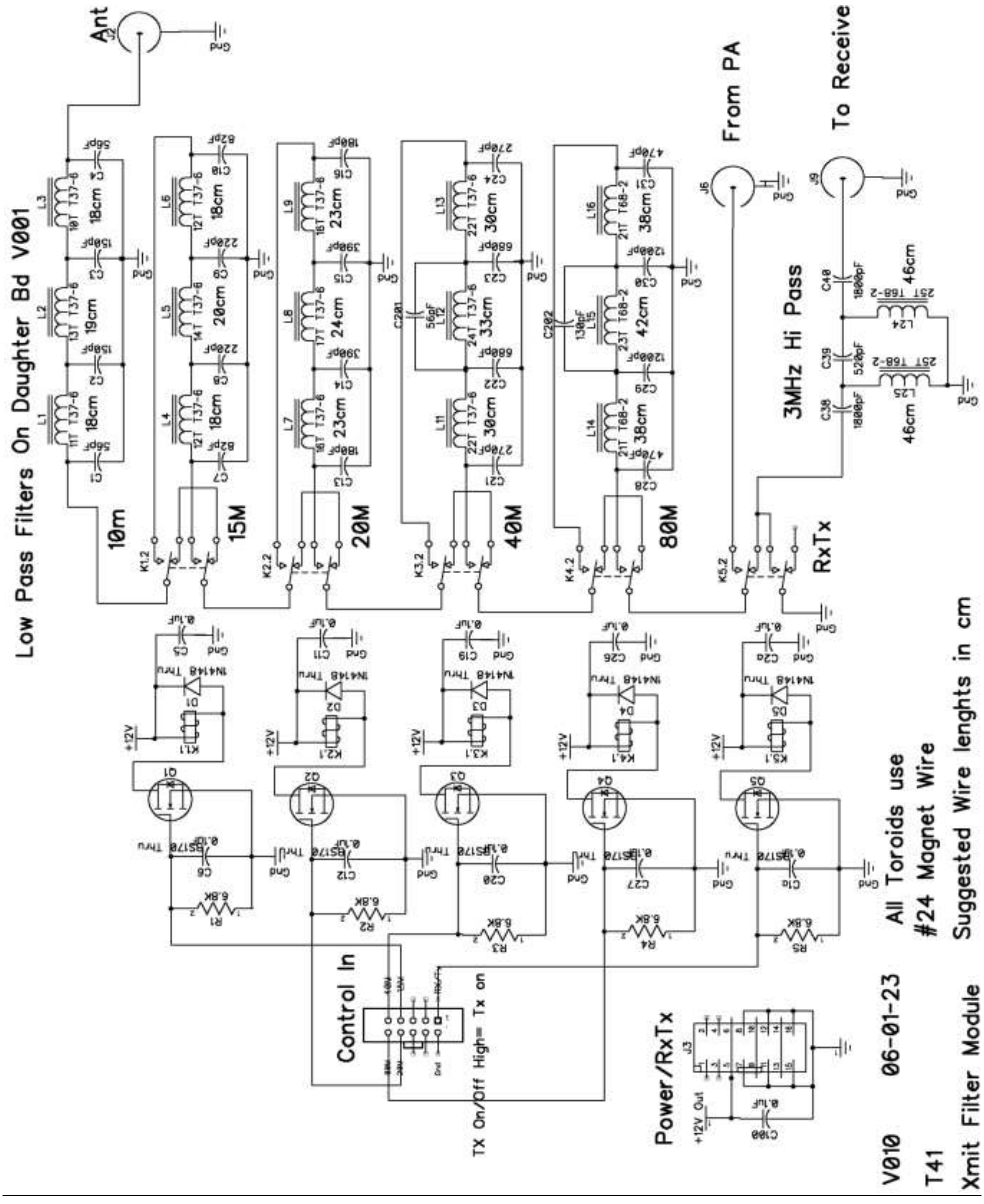


Figure 7-10. Daughter filter board.


Low Pass Filters On Daughter Bd V001
Ant
From PA
To Receive
3MHz Hi Pass
10m
15M
20M
40M
80M
RxTx
Control In
TX On/Off High= Tx on
Power/RxTx
All Toroids use
#24 Magnet Wire
Suggested Wire lenghts in cm
V010 06-01-23
T41
Xmit Filter Module

Section 8: Chassis

☐ Inventory the components in the bag. It should contain:

Table 8-1. Chassis BOM

	Qty	Description	√
1	1	Boost Converter	
2	6	M3x6 screw	
3	4	3.5mm audio patch cable, 15 cm	
4	4	15 cm SMA cable	
5	2	Right Angle SMA cables, 30 cm	
6	2	Right Angle SMA cables, 15 cm	
7	22	Mounting Blocks	
8	58	6-32 x 5/16" machine screw	
9	8	6-32 x 1/2" machine screw	
10	8	6-32 nut	
11	4	4-40 x ½" screw	
12	4	4-40 nut	
13	4	1/4" #6 spacer	
14	4	1/8" #4 spacer	
15	4	1/8" #6 spacer	
22	2	Tilt feet sets	
23	4	#3x 1/2" self tapping screw	
24	1	SMA to BNC Bulkhead 30cm	
26	1	Male-Female SMA elbow	
27	1	Enclosure Board Set	
28	1	Power Switch	
29	1	Inline fuse holder	
30	1	8A fast blow fuse	
31	1	DC Power 2.1 x 5.5mm	
32	1	DC plug 2.1 x 5.5mm	
33	2	Big knobs	
35	2	medium knobs	
36	1	speaker	
37	1	Ribbon Cable	
38	1	Teensy Audio Board	
39	1	USB Host Cable	

40	1	2-pin Dupont cable, 30 cm	
41	2	2-pin, 2mm cable	
42	5	2x8 IDC connector	
43	5	1/8 stereo jack	
44	1	1/8 stereo jack, switched	
45	2	Rubber feet	
46	2'	red hookup wire	
47	2'	black hookup wire	

Inventory the items, checking off each as you identify them. If you are short any parts, please contact the kitter listed at http://www.4sqrp.com/T41main.php, and the item will be promptly sent to you.

REAR PANEL ASSEMBLY

Boost Converter

☐ Locate the rear panel, the Boost Converter module, and four M3-6 screws. (These are the screws that fit into the standoffs on the back of the rear of the module.)

The bottom two holes don't seem to align with the holes with the holes in the case. Either drill two new holes or glue the two bottom stand offs to the case with epoxy.

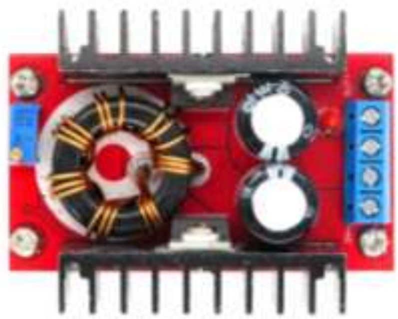


Figure 8-1. The Boost converter module.

☐ Mount the Boost Converter to the inside surface of the rear panel, with the blue screw terminals at the bottom.

Figure 8-2. Nylon mounting hardware.

NOTE: Some kits are supplied with a different boost converter and the bottom 2 panel holes are slightly above the holes on either side of the blue screw terminal. Those kits are supplied with special nylon mounting hardware as shown in Figure 8-2. Take 4 nylon spacer screws (circled in red in Figure 8-2) and insert them into the 4 corner mounting holes on the boost converter module. Secure each spacer screw to the converter board with a nylon nut. Your converter board should look similar to Figure 8-3.

Figure 8-3. Spacer screws mounter to converter board.

☐ Take the converter board and align it with the top 2 back panel mounting holes. Take 2 nylon mounting screws (circled in green in Figure 8-2) and feed them from the backside of the panel into the top 2 spacer screws on the converter. Tighten the screws, but don't overtighten…you don't want to strip the threads of the hardware.

☐ The boost converter is now mounted to the back panel with the top mounting hardware, but the bottom 2 spacer screws are secured only to the convert board. This mounting method will hold the boost converter in place without securing the bottom two spacers, yet still prevent the boost converter from shorting out against the back panel. However, if you wish, place two blobs of hot glue at the points where the bottom spacer screws meet the back panel. The hot glue will mount the converter more securely to the back panel. This is what we did and figure 8-5 shows the boost converter in place.

Heat Sink

☐ Locate the large heat sink/fan assembly, and the two nylon M4x10 screws from the Power Amplifier parts kit.

Figure 8-4. Heat sink

☐ Place the heat sink behind the enclosure Rear Panel so that the larger diameter pair of the six drilled and tapped holes showing through the square hole cutout in the rear panel is slightly offset towards the right. See Figure 8-5. Mount the heat sink to the enclosure Rear Panel using the two nylon M4x10 screws.

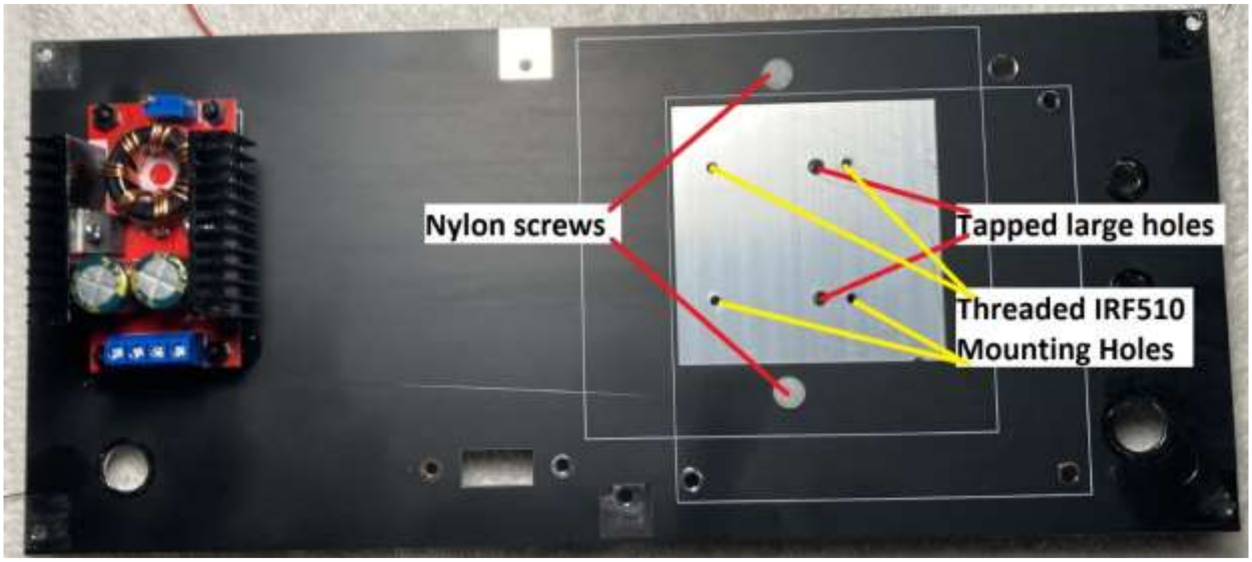


Figure 8-5. Heat sink mounted in place.

Diode Board

☐ Locate a two conductor cable with a 2.0mm female connector. Locate one of the white, 2.0mm male connectors from the Power Amplifier parts kit. Mate the two.

☐ Solder the connector to the Power Amplifier board in the J3 'Temp' position oriented so that the red wire side of the connector aligns with the '+' marking on the connector position.

☐ Trim back the wires from this connector to 3 inches in length. Strip back ⅛" of the insulation from both wires.

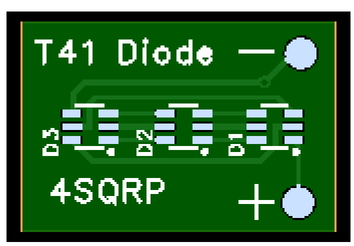


Figure 8-6. Diode Board

☐ Solder these wires to the two round pads on the Diode board, the red wire to the '+' and the black wire to the '-' pad. (Note these are pads, not thru-holes and this image is shown without the ICs.)

Mounting BNC connector

☐ Locate the BNC antenna connector and cable assembly. Mount the connector in the hole seen in the lower-right corner of Figure 8-5. Secure into position with the supplied lock washer and nut. *NOTE*: This must be done at this point in the assembly because the mounting flange of the connector fits *below* the PA PCB. If you don't mount this connector now, you will have to file away part of the edge of the PCB to get it to fit in place. As it is, the BNC connector is a tight fit. You may have to take a few strokes on the inside of the hole with a metal file before the connector will fit.

Mounting the IRF510s

☐ In the bag of power amplifier parts, locate the four IRF510 transistors, four grey heat sink insulator pads, four plastic shoulder washers, and four M3.5x12 screws. (These screws are for mounting the PA transistors, verify that they fit into the tapped holes in the heat sink.)

☐ Mount the IRF510 transistors to the heat sink by first laying down the grey heat sink pad, the transistor, and place the small, clear plastic shoulder in the hole in the transistor tab, and thread the mounti9ng screw into the heat sink. Tighten the screw, but don't over-tighten to the point of stripping the threads. Handle the shoulder washer carefully as they are easy to lose. Note: The body of the shoulder washer may not fit readily into the hole in the transistor tab, but if the screw is installed and tightened, it will draw the shoulder washer down into the hole. See Figure 8-7.

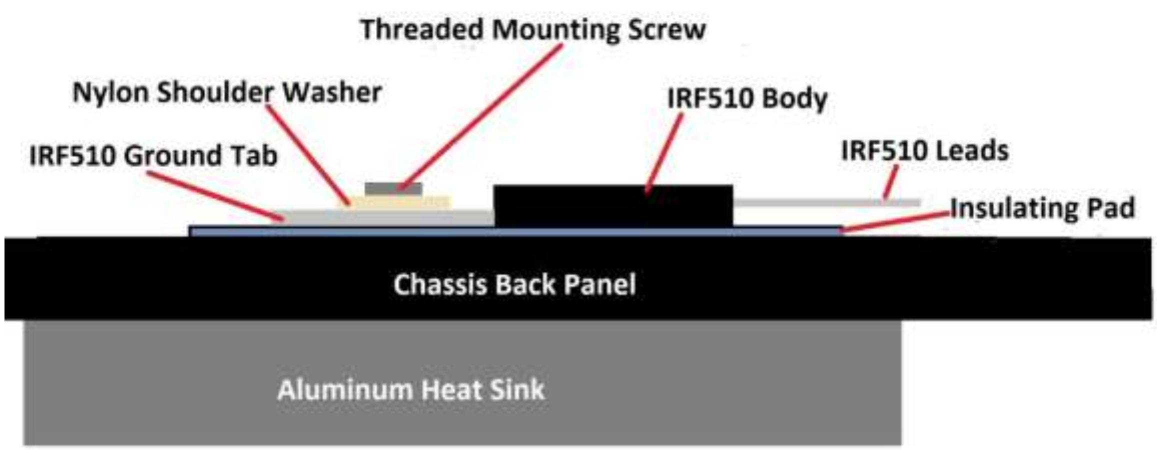


Figure 8-7. Mounting the IRF510 to the heat sink (side view).

☐ Repeat for each of the other IRF510 transistors.

☐ Using a multimeter continuity checker or DVM, verify that there is no conductivity between tabs of the transistors and the heat sink. Touch the threaded mounting screw and the heat sink and it should show a dead short (0Ω). However, touching the IRF510 ground tab and the heat sink should show an open circuit (infinite resistance). If either of these conditions is not true, re-mount the IRF510. Your back panel should look similar to Figure 8-8.

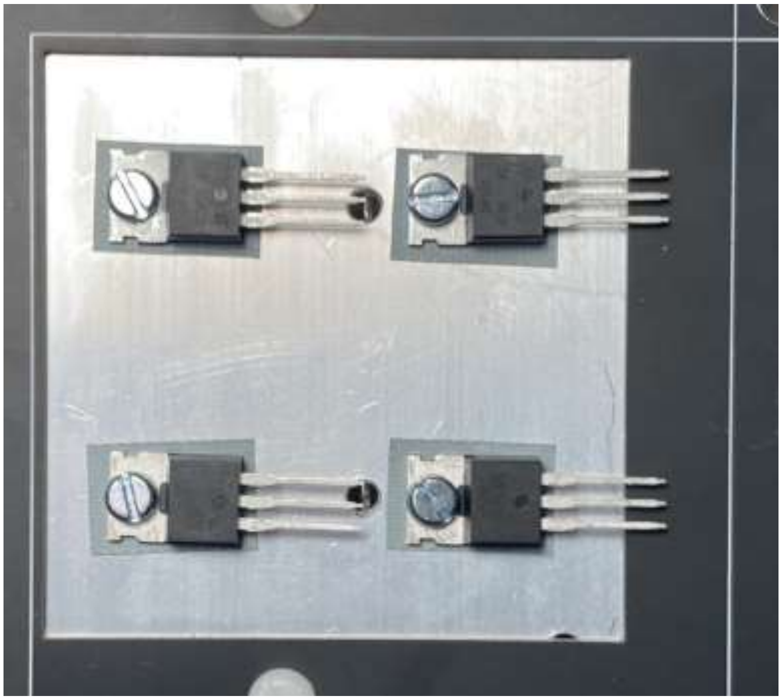


Figure 8-8. The back panel with IRF510s mounted to the heat sink.

☐ Bend the legs of the IRF510s upward at a right angle. A good way to do this is with a pair of needle nosed pliers. With the jaws of the pliers in line with the lead to be bent, grasp the lead about 1/16" from the body of the IRF510 and bend up 90°. Repeat for all other IRF510 leads. When finished, your transistor leads will look similar to those seen in Figure 8-9.

Diode Board Placement

☐ Before securing the PA board in place, place the Diode Board *component side down* against the heat sink, directly under where T5 is located when the PCB is in position. Place a square of closed-cell foam between the PA board and the Diode board. This applies pressure to hold the Diode board in place against the heat sink. You can hold the diode board in place by taping its leads to the back panel and placing the foam pad on top of the board. See Figure 8-9. (The red and black leads go to the diode board under the foam cell.)

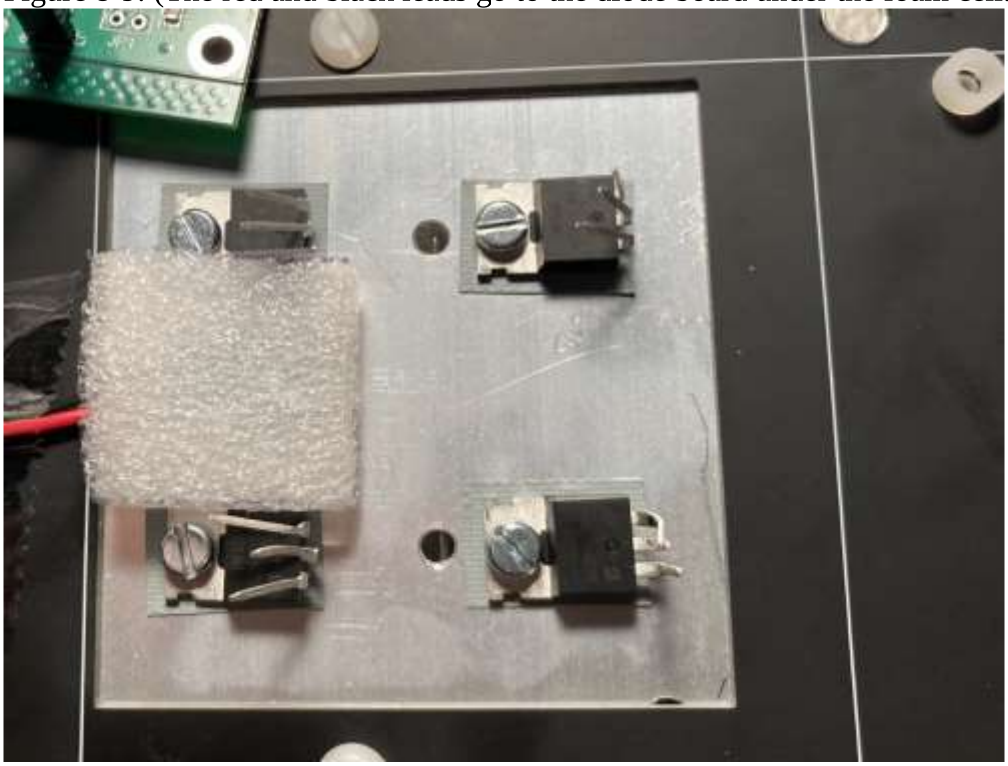


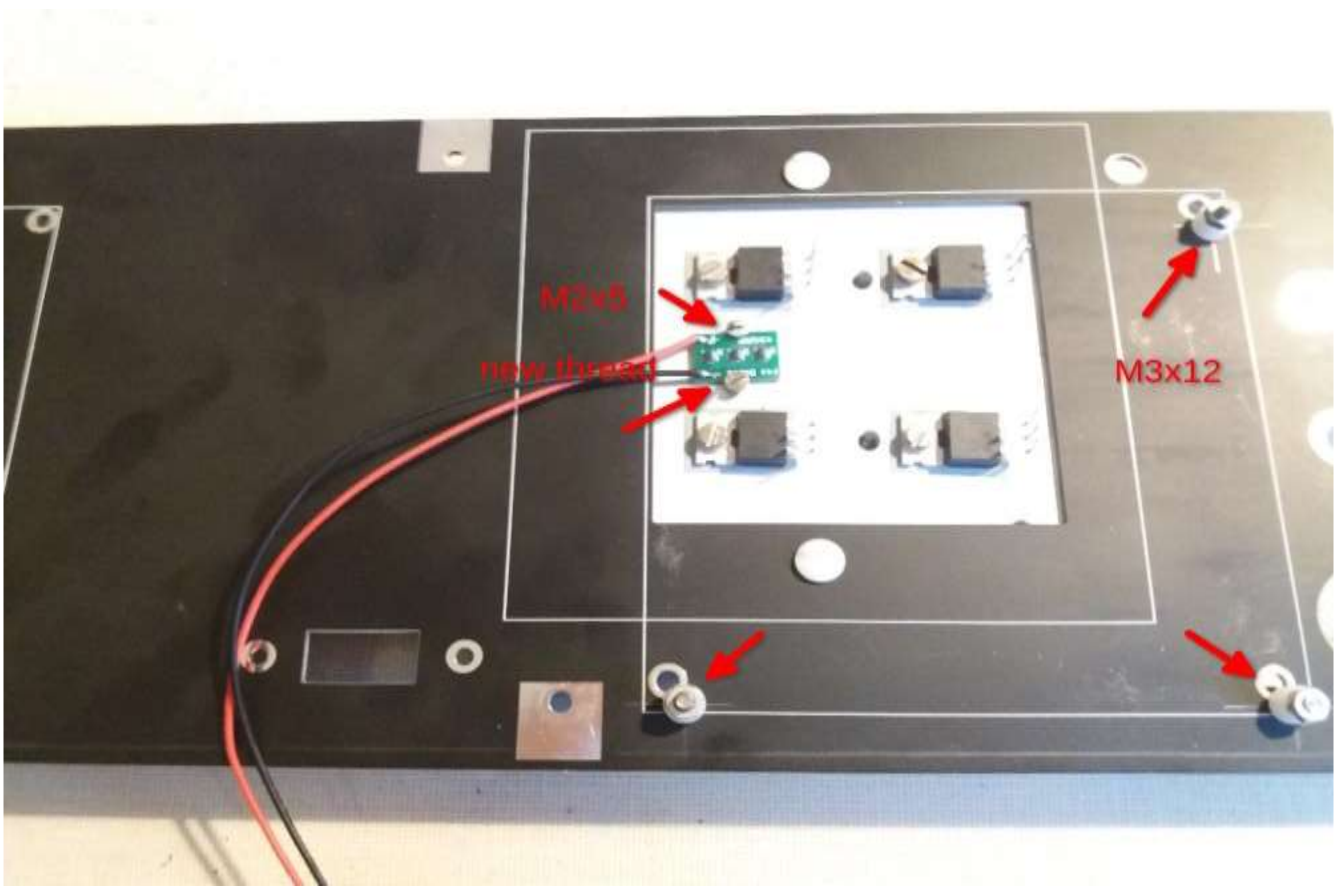
Figure 8-9. Placing the diode board on heatsink.

Below is a optional way to mount the diode board.

The Picture above is a suggested mounting modification of the amp board to the heat sink, by Roger from the Czech Republic. He has relocated his mounting holes down and to the right which allows the power transistors leads to land straight on the solder pads.
He also chose to drill and tap two m2-5 mounting holes for the diode board.
You will have to determine the location of the new hole locations.

Mount the Power Amplifier PCB to the enclosure rear using three, 6-32 x ½" screws in the corners of the Power Amplifier board. Pass these screws through the rear of the enclosure panel, and place 1/8" spacers over the screws before placing the Power Amplifier board over them, and 6-32 nuts over this. An easy way to place the spacer is to slide the spacer between the panel and the PCB and push the shaft of a small screwdriver through the PCB, the spacer, and then into the mounting hole. With your fingers, hold the spacer in place by pinching the back panel and PCB. Withdraw the small screwdriver and pass the threaded screw into the hole from the backside. Push the screw through the back panel, the nylon spacer, and the PCB. Loosely thread the nut onto the screw in order to leave enough room to repeat the process with the other 2 screws. (Only 3 screws hold the PA board in place.)

☐ When all three nuts are in place, tighten securely. (This is a lot simpler if you have a small socket set.)



☐ Bend the legs of the IRF510 transistors over to their corresponding solder pads. Trim them to length so they do not extend beyond the ends of the solder pads. Solder into place. (The leads have to be bent slightly “sideways” as they do not align perfectly with the solder pads. You can see this in Figure 8-10. This isn’t a serious problem, just make sure there are no shorts after you solder the leads.)

☐ Plug the Diode board cable into the J3 Temp header. When you are finished, your back panel should look similar to Figure 8-10.

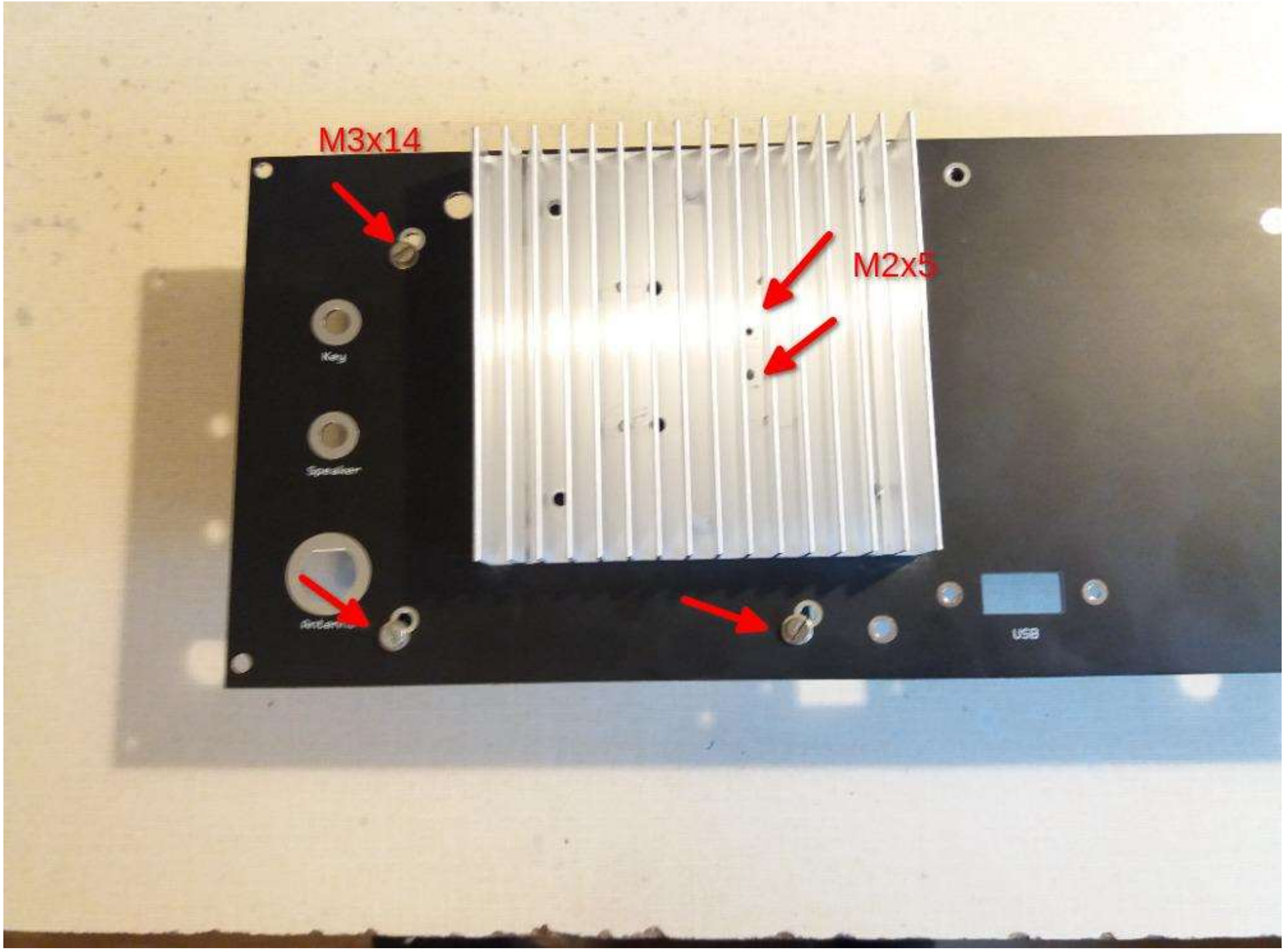


Figure 8-10. PA board in place.

This picture is a view of the back side of the modified mount.

Power Amplifier Bias Adjustment

☐ ***NOTE***: Before starting the bias adjustment, set the four blue trim pots to their minimum resistance.(Turning the set screw *counter clockwise* reduces the resistance. Measure it to be sure) The minimum is probably between 100-200Ω. We assumed they were already set to zero, in section 6. Failure to do this may result in a failure like Figure 8-11. The ensuing explosion was spectacular! The corner piece of plastic hit the ceiling with such force that it left a small nick in the drywall! Moral: don't assume the trim pots are set to their minimum values and don't work without safety glasses!

See details on this adjustment in the power board assembly guide section 6.

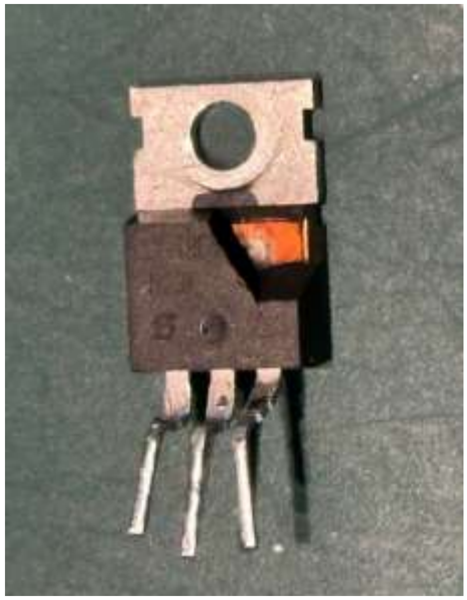


Figure 8-11. An abused IRF510.

☐ Cut a piece of heavier black hookup wire of sufficient length to run between the screw terminal blocks on the Boost Converter module and the Power Amplifier board. Strip about ¼" of the insulation from both ends and tin the leads.

☐ Connect the black hookup wire between the terminal of the screw terminal block on the Power Amplifier board marked 'G', and the terminal on the Boost Converter marked '-Out'.

☐ Cut a piece of red hookup wire the same length and strip ¼" of the insulation from each end and tin the leads.

☐ Connect the heavier red hookup wire between the terminal of the screw terminal block on the Power Amplifier board marked '+25', and the terminal on the Boost Converter marked '+Out'.

☐ Temporarily connect the SMA-to-BNC cable to J2, the 'Out' SMA terminal on the PA board. Connect a 50 ohm dummy load to the BNC.

☐ Bridge the 2-pin header JP1 with the plastic jumper (2.54mm 2-pin) provided in the power amplifier parts bag.

☐ Temporarily connect a 12v power supply to the '+In' and '-In' terminals on the boost converter.

☐ Apply 12v DC to the Boost Converter. Monitor its output voltage with a multimeter. Turn the blue potentiometer on the Boost Converter module until the voltage on the output terminals reads +25 volts. Figure 8-12 shows a typical test setup. Notice the clip leads from the boost converter. The green clip goes to the +25V on the J8 terminal block and its black lead goes straight into the GND screw terminal of the boost converter. The other end of the green clip lead completes the connection through the DVM (not visible in photo).

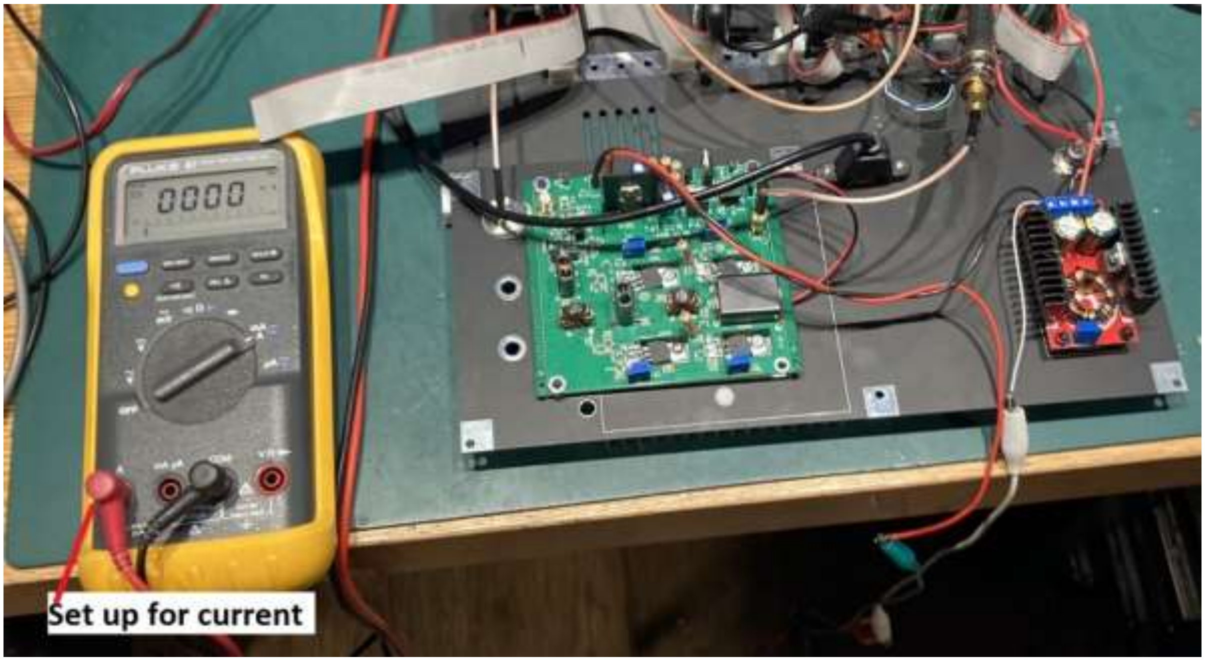


Figure 8-12. Set up for setting bias.

☐ Measure the current from the +25 volt power supply. You can measure this by connecting the meter between the +25V connection on the boost converter and the +25V input connection on the PA board at connector J8. (Note that most DVMs require you move the red input lead from the voltage to the Ammeter input jack,) When you first observe the current, you'll likely see around 50ma of current after allowing for a short warm-up period. This 50ma of current is the *baseline current*.

☐ While monitoring the current from the +25v power supply, turn the set screw on the (blue) trim pot R100 on the Power Amplifier board clockwise until the current increases by 200 mA above the baseline. That is, when you first observe the current, you saw around 50ma of current. Turn R100 clockwise until you read 250ma on the meter.

☐ Turn the trim potentiometer R102 clockwise until the +25v power supply current increases by another 200 mA. You should now see a total of about 450ma of current.

☐ Turn the trim potentiometer R200 clockwise until the +25v power supply current increases by another 200 mA. You should now see a total of about 650ma of current.

☐ Turn the trim potentiometer R203 clockwise until the +25v power supply current increases by yet another 200 mA. You should now see a total of about 850ma of current.

☐ Turn off the power supply and remove its cables from the boost supply module.

☐ Remove the jumper from JP1.

☐ Remove the SMA to BNC RF cable from the Power Amplifier board.

☐ Mount the SMA to BNC RF cable assembly into the rear panel in the Antenna position.

☐ Mount the USB Host cable into the rear panel using two, M3x6 screws.

Top Panel Assembly

☐ Locate the enclosure Top Panel, the speaker, four 6-32x5/16" screws, and four 6-32 nuts.

☐ Mount the speaker into the panel using the hardware. Do not over-tighten the screws as the frame of the speaker may become warped.

Front Panel Assembly

Encoder Board

☐ Locate the two completed encoder board assemblies. Remove the hardware from the encoders.

☐ Mount the encoder boards with the 4-pin header strip at the position adjacent to that of the switch matrix board of the enclosure Front panel, with the 2x5 pin box header above the 4-pin header. Tighten the shaft hardware of the encoders to hold the board in place.

☐ Locate the 4-pin cable assembly. Cut the cable approximately 3 inches from the connector. Strip back the insulation approximately ¼" from the end of the wires. (Save the remaining part of the cable.)

☐ Solder these wires into the labeled pads on the Switch Matrix board. Note that the center two wires are twisted together.

<table>
<tr><td>Red</td><td>3.3v</td></tr>
<tr><td>Black</td><td rowspan="2">G</td></tr>
<tr><td>Yellow</td></tr>
<tr><td>White</td><td>Out</td></tr>
</table>

Switch Matrix

☐ Place the 18 clear plastic switch covers on each of the 18 switches of the switch matrix. Take one of the side panels and place it over the switch assembly. Flip the switch board and panel over so the switch panel is on top. Now, firmly press on the switch board. The purpose of this is to insure that the plastic switch covers are firmly seated on the switches. The side panel just makes it easier to flip the switch board without the covers falling off.
Some builders have glued these caps on to keep them in place.

☐ Mount the Switch Matrix board to the front panel using four, 6-32 x ½" screws, ¼" spacers, and 6-32 nuts. Finger-tighten the screws. Make sure the switch labeled '1' passes through the "Select" switch opening as labeled on the front side. Now, test all 18 switches to make sure they move freely with their covers in place. If you mounted a switch crooked (i.e., not sitting parallel on the PCB), it is possible for the switch cover to bind on the edge of the PCB when pressed. If this happens, remove the switch PCB from the screws, reheat the connection, and reseat the switch so it sits straight on the switch PCB.

☐ After testing the free movement of the switches, tighten the screws and recheck for free switch movement. Repeat these last two steps until the switches move freely when mounted on the panel.

☐ Plug the 4-pin cable from the Switch Matrix board into the 4-pin header on the Encoder board so that the color of the cable conductors matches the labels on the board. (We used an older board in the photo, which is why you see the small wires at the bottom of the figure. Older boards had all 18 switches, but only 15 were part of the matrix. As the feature set increased, all 18 switches were used by the software.)

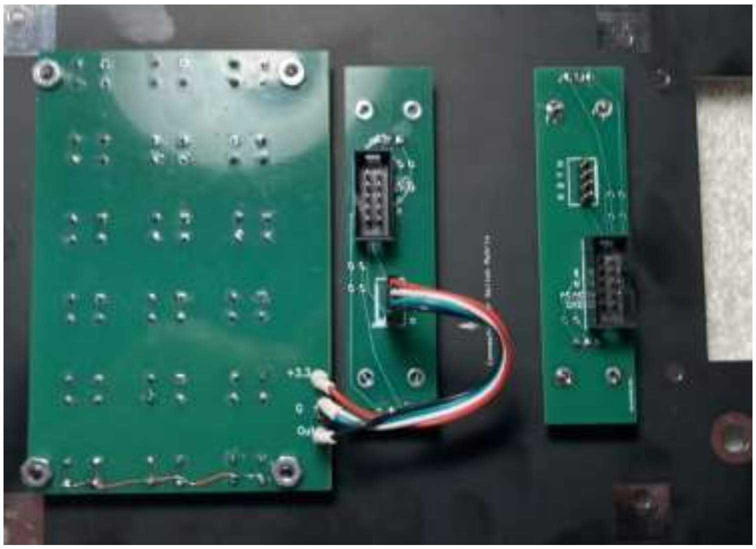


Figure 8-13. Encoder boards and switch assembly mounted on back panel.

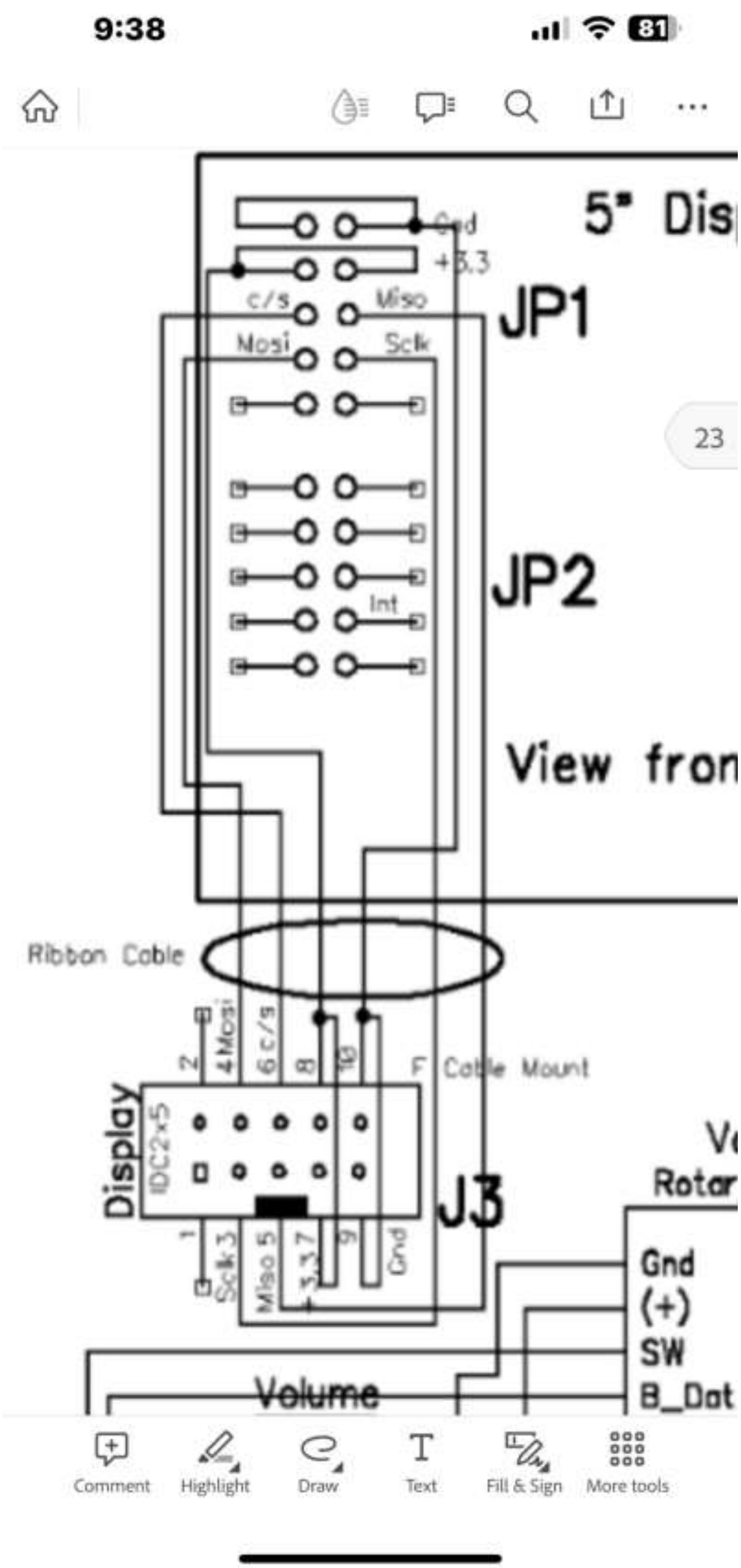
NOTE: The right-hand encoder board is reversed, please rotate it and reinstall.

Display Installation and Connection

☐ Take the display panel and remove the clear protective film from the front. Mount to the enclosure Front Panel using four 4-40 screws and nuts, oriented with the pin header at the top left corner. Space the display from the front panel using the ⅛" thick #4 spacers.

☐ Find a 10-pin pre-made ribbon cable about 6 inches long. It will connect the power and data for the Main board to the display. Plug one end into the connector just above the CPU fan plug on the edge of main board. It will only plug in one way since it's keyed. It was marked display till the Connector was installed and the label was covered up. The other end plugs into the connector at the top left corner of the display (looking from the rear) labeled JP1. The key on the connector will be facing toward the JP1 label.
Note: The bottom two pins will be hanging off the connector, since it only has eight pins. Make sure your display's jumper setting is set for 3.3 volts.

Next Page is a diagram of this circuit.



DISPLAY CONNECTIONS.

☐ Mount the toggle power switch into its position in the front panel.

Bottom Panel Assembly

☐ Locate the four plastic enclosure feet pieces, and the four #3 self-tapping screws.

☐ Mount the folding tilt feet within their holders using the self-tapping screws. See Figure 8-14.

Figure 8-14. Tilt feet pieces in place.

☐ Mount the enclosure feet to the bottom of the enclosure Bottom Panel using 5/16” 6-32 screws and nuts.

Mounting blocks

☐ Locate 10 of the white cubic mounting blocks and 20 of the 5/16” long 6-32 screws.

☐ Attach two mounting blocks lower edge of the Power Supply, QSD, Main Board, Exciter, and Relay boards. Orient the blocks as shown in Figure 8-15.

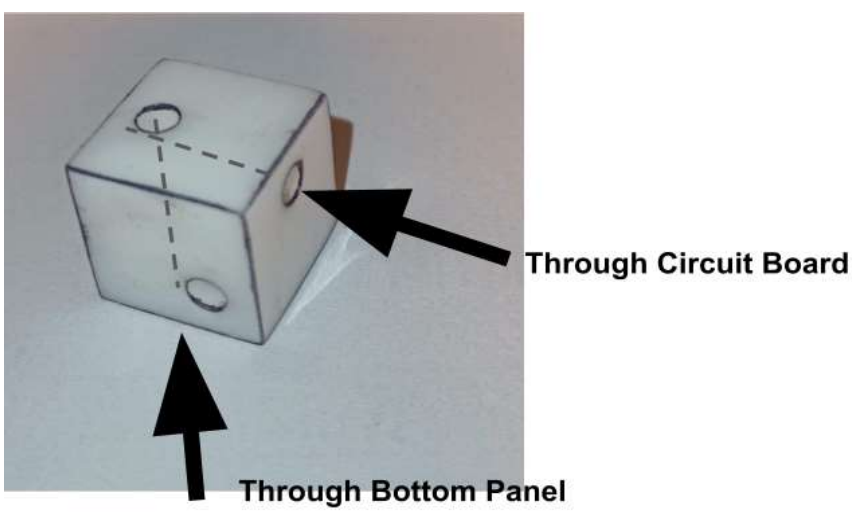


Figure 8-15. Mounting blocks

☐ Mount the boards to the enclosure bottom panel using 5/16" screws up through the enclosure bottom panel. The component side of the board should face the display side (right) of the panel. Space the boards evenly across the bottom panel. A suggested mounting order listing the board and its mounting hole position is as follows, which spaces the boards roughly evenly:

NOTE: These are suggested spacing for the boards.

Please adjust the spacings between the QSD, Main and Exciter boards to that the 3.5 mm audio cables will reach.

Table 8-2. Mounting positions

Board	Hole Position
Power Supply	3
QSD	6
Main	10
Exciter	14
Relay	18

☐ Locate six mounting blocks and six 5/16" 6-32 screws. Attach mounting blocks to the four corners and center of the front and rear edges of the enclosure Bottom Panel board. You can see some of the mounting blocks in Figure 8-16

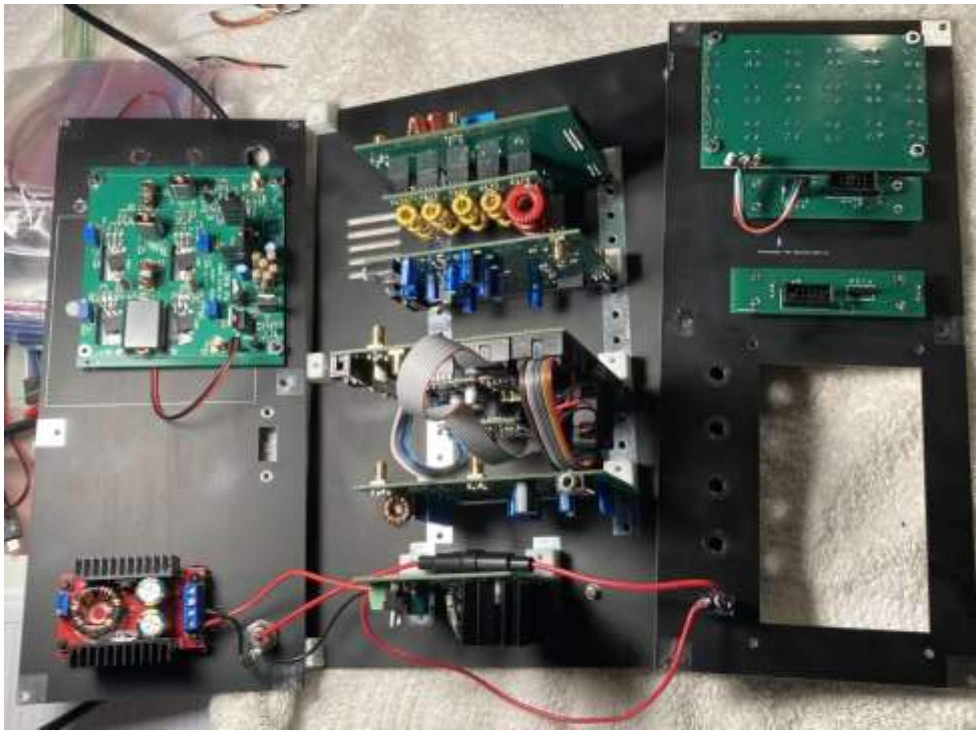


Figure 8-16. Front, bottom, and back panels with mounting blocks.

Chassis Wiring

Power Supply and Fuse

- ☐ Lay the front and rear panels on the work surface face down, in their respective positions adjacent to the bottom panel.
- ☐ Locate the inline fuse holder and the fuse. Install the fuse into the fuse holder.
- ☐ Cut the wire connecting the sections of the fuse holder in half.
- ☐ Strip back the insulation on the fuse holder wire approximately ¼” on each end.
- ☐ Solder the fuse holder between the center terminal of the DC power jack on the rear panel and one of the terminals of the front panel power switch.
- ☐ Cut a length of red hookup wire of sufficient length to connect between the front panel power switch and the screw terminal block on the Power Supply board. Strip back the insulation about ¼” from each end.
- ☐ Solder one end of this wire to the unused terminal of the Front Panel power switch
- ☐ Cut a length of red hookup wire of sufficient length to connect between the Power Supply board and the screw terminal block on the Boost Converter module on the Rear Panel. Strip back the insulation about ¼” from each end.
- ☐ Twist the bare copper ends of these two pieces of wire and insert them into the ‘+12v’ terminal of the screw terminal block on the DC Power Supply board. Tighten the set screw firmly.
- ☐ Insert the other end of the second wire into the ‘+ Input’ terminal of the Boost Converter module, and tighten the set screw firmly.
- ☐ Cut a length of black hookup wire of sufficient length to run between the grounding tab of the DC power jack in the Rear Panel, and the screw terminal block on the Power Supply board. Strip the insulation back from the ends about ¼”.
- ☐ Cut a second length of black hookup wire of sufficient length to run from the grounding tab of the DC power jack on the enclosure Rear Panel to the Boost Converter module. Strip the insulation back from the ends about ¼”.

☐ Solder one end of each of these wires to the ground tab. Attach the other end of the first wire to the 'Gnd' terminal of the terminal block on the DC Power Supply PCB. Mount the free end of the second wire to the '- Input' of the screw terminal block on the Boost Converter module. When you have finished wiring the power supply, it should look similar to Figure 8-17.

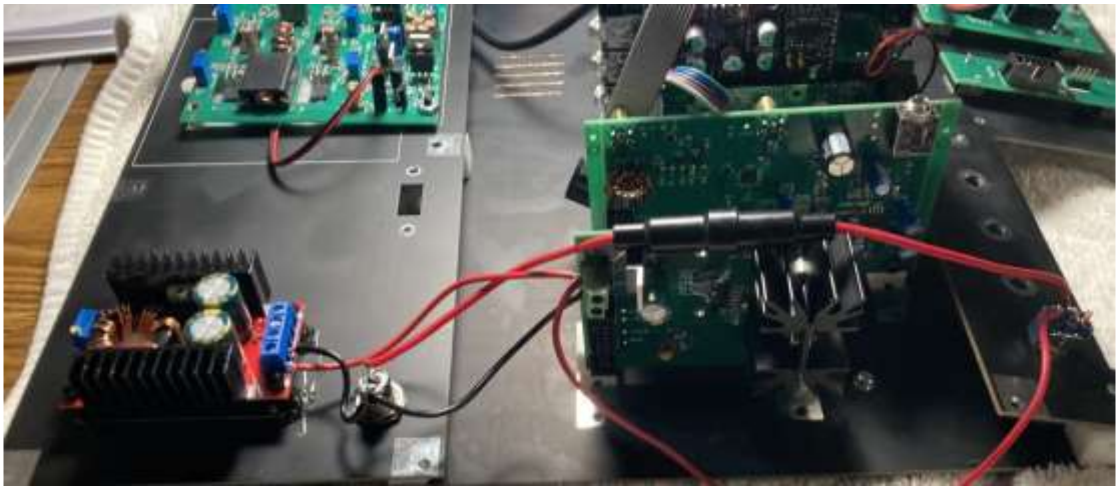


Figure 8-17. Power supply and fuse wiring.

Signal Cabling

☐ Locate two of the 3.5mm audio plug cables.

With the component side of the PCBs facing you, the Main PCB has three 3.5mm audio jacks at the upper left rear edge. Plug one of the cables into the upper jack marked rec. I-Q. Plug the other end of this cable into the audio jack at the upper right corner of the QSD board the Jack is marked Exciter I-Q.

☐ Plug one end of the second cable into the middle jack on the rear corner of the Main PCB, plug is marked Exciter I-Q Plug the other end of this cable into the audio connector of the Exciter Board marked Audio I-Q.

NOTE: Please adjust the spacings between the QSD, Main and Exciter boards to that the 3.5 mm audio cables will reach.

As you begin connecting the SMA connectors between the boards, you can choose which side of the boards would be best suited for routing the cables. Choose your route, tack the SMA connectors in place, then remove the card and solder them.
You may need to use some of the SMA 90-degree connectors on the cables to prevent interference.

☐ Locate the two 15cm SMA coaxial cables.

☐ Attach one end of the first cable to the upper SMA jack on the QSD PCB marked CLK-2. Connect the other end to the SMA jack on the upper edge of the Main board Marked CLK-2

☐ Attach one end of the second 15cm SMA cable to the left-hand SMA jack at the upper edge of the Main PCB marked CLK-1. Connect the other end of this cable to SMA jack at the upper right corner of the Exciter PCB marked CLK-1.

☐ Locate the two 30 CM SMA coaxial cables.

☐ Attach one end of the first cable to the SMA jack at the upper left corner of the QSD board marked RF in. Attach the other end to the SMA jack at the upper right corner of the Filter Relay PCB Marked “to Receive.

☐ Attach the second cable to the SMA jack at the lower right corner of the Exciter PCB marked RF out. Connect the other end of this cable to RF input jack J1 at the bottom right corner of the Power Amplifier board.

☐ Attach a cable from the upper center connector of the filter board marked “from PA” to the lower left corner of the PA board marked “out”.

☐ This should leave only one Cable that goes from the bottom connector of the Filter board To the BNC antenna connector on the rear of the radio.

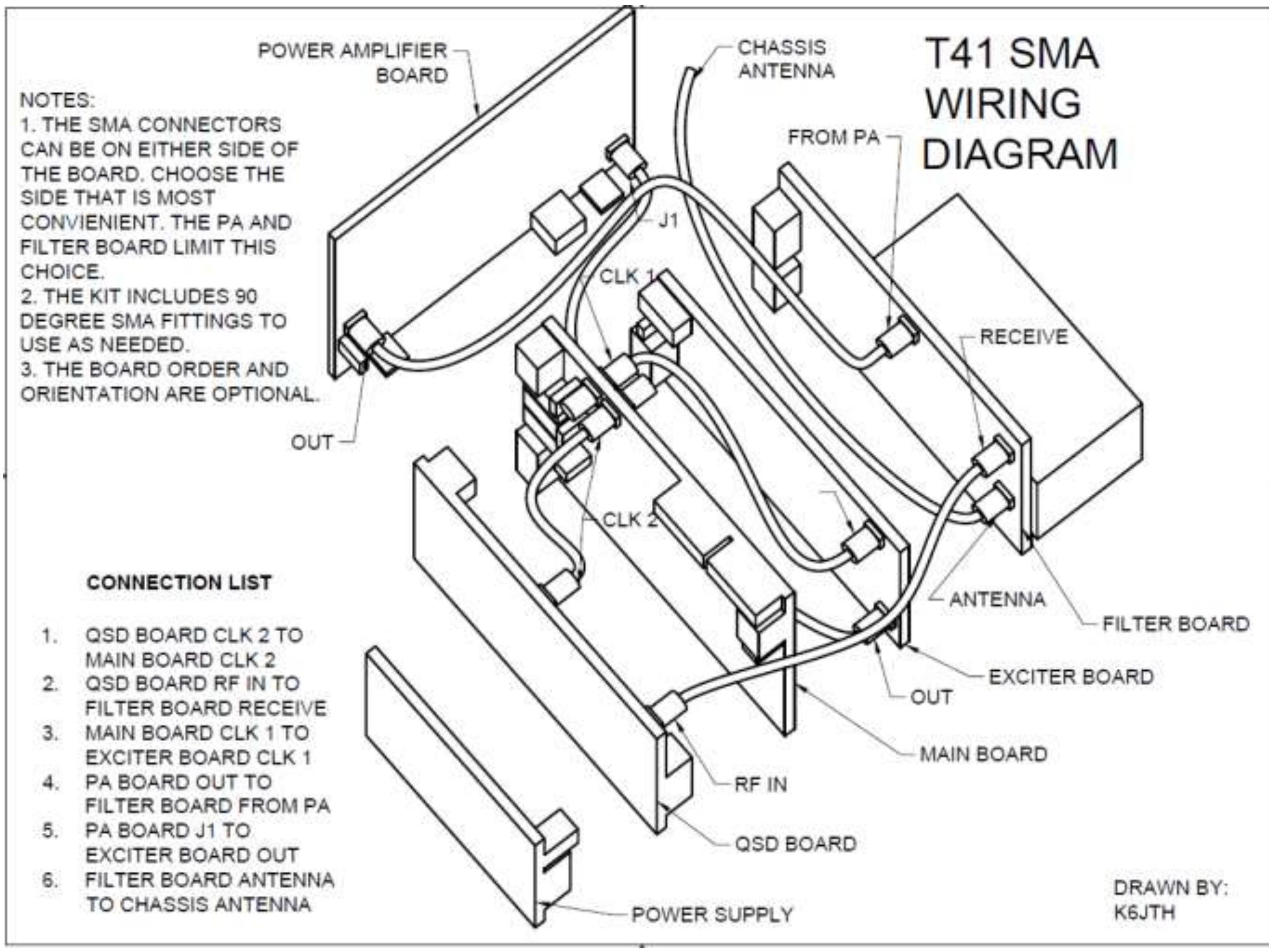


Figure 8-18. SMA Wiring Diagram.

Ribbon Cables

☐ We now assemble the DC distribution cable. Locate the section of ribbon cable and the five insulation-displacement connector assemblies.

☐ Take one of the connectors, and insert the cable through it, left-to-right, with the red conductor in the pin-1 position. Leave about ½” extending through the other side.

☐ Seat the cable within the connector by compressing the connector sections within a hobby vise, or by gently tapping it with a soft mallet until the sections are well mated.

☐ Fold the short extending section over the body of the connector and fix it into place with the latching clip.

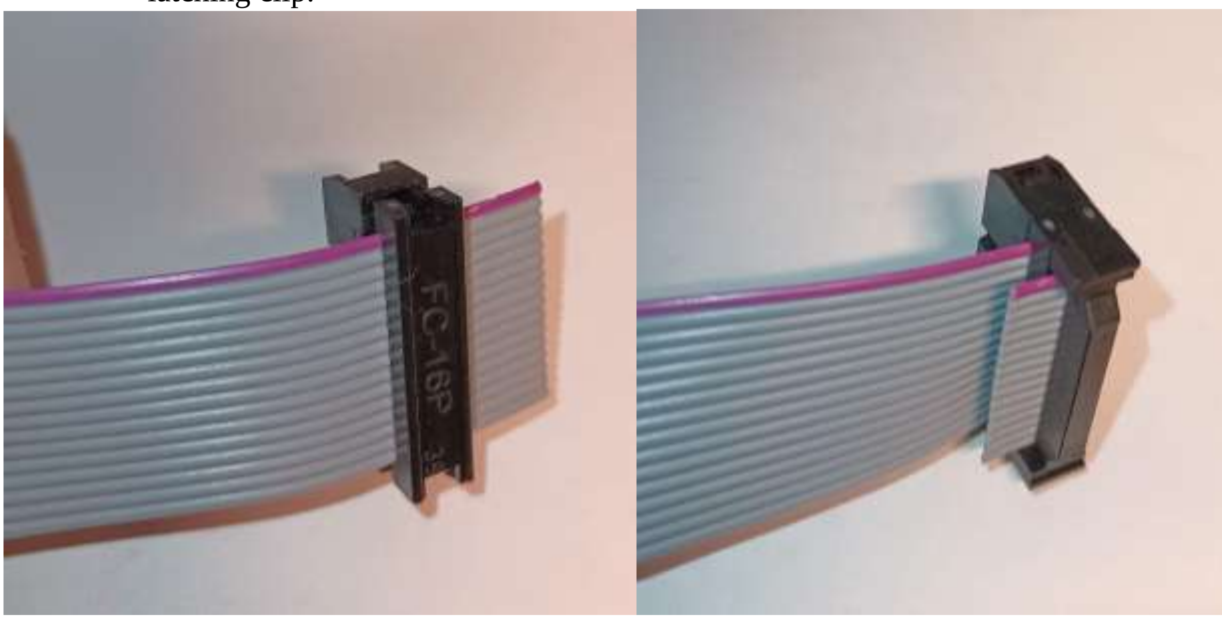


Figure 8-19. Securing cable connector to cable.

☐ Insert a second connector onto the ribbon cable and mount it about 3.5 inches from the first connector. Compress the connector sections to mate the connect sections. Fold the cable with the previously mounted connector left over the top of the newly mounted connector and fix it into place with a latching clip.

☐ Repeat this process with the remaining three connectors, but space them four inches from each previously mounted connector.

☐ Cut off the remaining ribbon cable.

☐ Install this ribbon cable assembly into the power connectors at the left edges of the PCBs.

☐ Locate the four, 10-conductor ribbon cable assemblies.

☐ Install one cable between the header at the upper left corner of the Main PCB and the lower left corner of the Filter Relay PCB.

☐ On the upper edge of the Main PCB, there are two header connectors on the right side of the board. Install one cable between the left-most of these connectors and the Encoder board closest to the Switch Matrix PCB on the enclosure Front Panel.

☐ Install a cable between the right-most connector on the upper edge of the Main PCB and the Encoder PCB furthest from the Switch matrix PCB.

☐ Install a cable between the header on the right side of the Main PCB and the pin header at the upper left corner of the Display module. This connector is unpolarized, but pin 1 is at the top of the header. Install the ribbon cable to this header with the red-marked pin 1 conductor at the top.

☐ Locate the remaining section of the cut-off 4-pin cable. Verify that it has enough length to run from the left edge of the Main PCB to the front panel location. If not, use some of the cut-off ribbon cable to extend its length.

☐ Strip the insulation ¼" back from each of the four conductors.

☐ Pull off a three-conductor strand from the 12" long section of ribbon cable, including the strand with red-colored insulation. Separate the conductors to about an inch from each end, and then strip back the insulation about ¼" from each end.

Audio Jack Connections

☐ Locate the five unswitched 3.5mm panel-mount audio jacks.

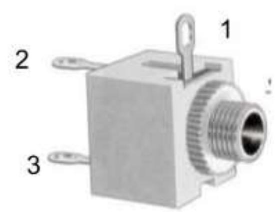


Figure 8-20 Audio Jack Pinout

☐ Twist together conductors 1 through 3 of the 12" section of 3-conductor ribbon cable to conductors 1 through 3 of 4-pin cable. Figure 8-21 shows the completed cable. Study it before reading the following instructions. The figure should help to make the directions clearer.

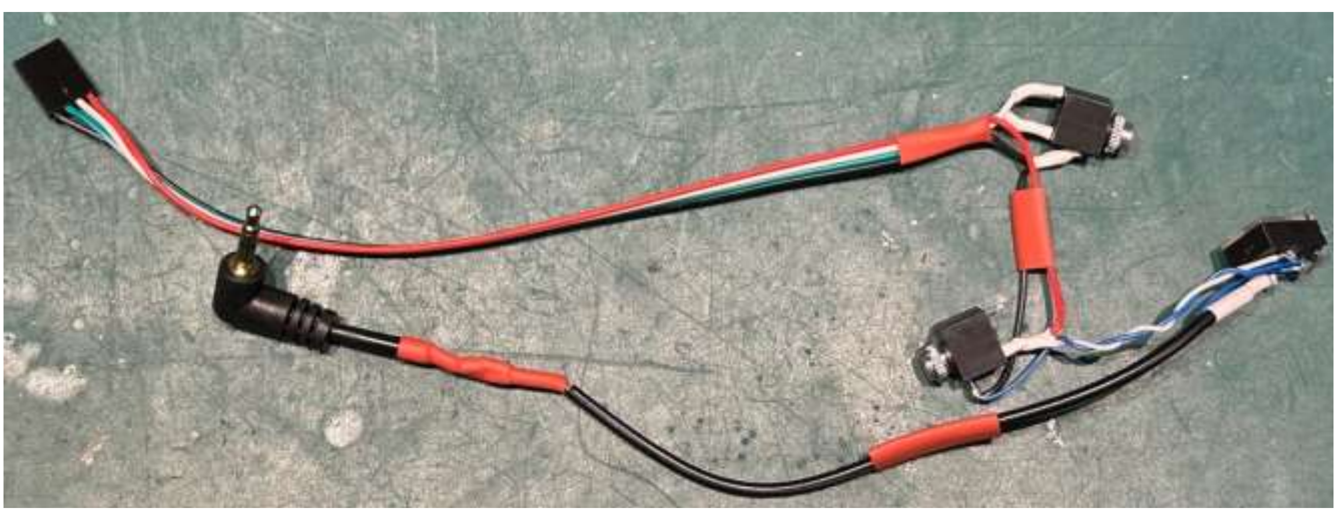


Figure 8-21. Front panel connector cabling.

Table 8-3. Audio Jack Connections

Pin Function	Pin Number	Key	PTT	
Gnd	1	1		
Key 1	2	2		
Key 2	3	3		
PTT	4		2	

☐ Solder the twisted-together conductors 1 through 3 of these two cables to terminals 1 through 3 of the first jack.

☐ Mount this in the ‘Key’ position in the enclosure Front Panel

☐ Solder the other end of the 12” 3-conductor ribbon cable to another audio jack, conductors 1 through 3 to be soldered to terminals 1 through 3 of the jack.

☐ Mount this jack in the ‘Key’ position in the enclosure Rear Panel

☐ Solder conductor 4 to pin 2 of another audio jack.

☐ Mount this jack in the ‘PTT’ position in the enclosure Front panel.

☐ Plug the 4-pin connector onto the 4-pin header on the Main PCB, taking care to be certain that pin-1 of the connector lines up with the uppermost pin on the header, which is marked ‘Gnd’.

☐ Locate a 3.5mm audio patch cable. Cut the plug off of one end. Strip back the outer jacket insulation about 1”. Trip the insulation back about ¼” from the ends of the individual conductors.

☐ Separate a three-conductor section of the spare ribbon cable, and using a marker, color one of the conductors to designate conductor 1. Separate the conductors at one end. Strip back the insulation about ¼” from each of the three conductors.

☐ Using a multimeter, determine which conductor in the audio patch cable connects to the shell of the plug. Solder this one to conductor number 1 of the 3-conductor ribbon cable.

☐ Similarly, identify the conductor in the audio patch cable that connects to the ring of the audio plug. Solder this to the second conductor of the ribbon cable.

☐ Finally, the last conductor, which connects to the top of the audio plug, is to be soldered to the third conductor of the ribbon cable.

☐ Cut this cable to length so that it can reach from the audio jack on the left of the Main PCB, and the front panel.

Table 8-4. Connection for MIC jack

Contact	Conductor Number	Jack Contact	Function
Shell	1	1	Gnd
Ring	2	2	PTT
Tip	3	3	Mic

☐ Solder the ribbon cable to one of the 3.5mm audio jacks per the table above. Mount the jack in the front panel ‘MIC’ position.

☐ Locate one of the 0.1”-spaced, 2-conductor cables. Cut to a length that will connect it between the SPEAKER header on the Main PCB and the front panel. Strip back the insulation on the conductors at the cut end ¼” from the end.

Figure 8-22 shows the switched audio jack and Figure 8-23 shows it’s internal connections. Use a continuity checker to determine the pin assignments. It’s different from the other front panel jacks because we want the audio to the T41’s internal speaker to turn off when a headsets is plugged into the headphones jack. Removing the headphones turns on the internal speaker.

Figure 8-22. The switched Audio Jack/

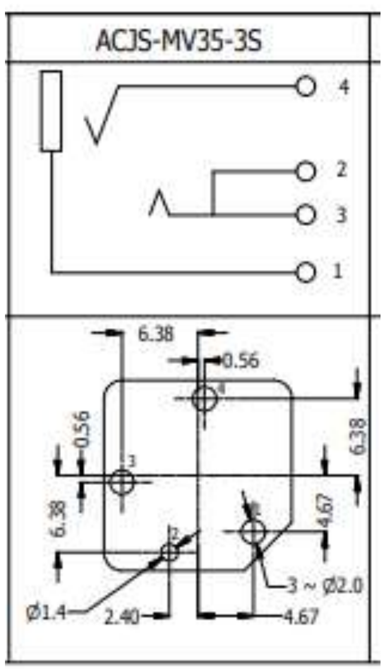


Figure 8-23. Switched Audio Jack Pinout

☐ Solder the end of the 2-conductor cable to pins 3 and 4 of the switched audio jack.

☐ Separate a 12” length of two conductors of the ribbon cable, and strip back ¼” of the insulation of both conductors on both ends.

☐ Solder one end of the cable to pins 2 and 4 of the switched audio jack.

☐ Mount the jack into the Front Panel in the ‘Phones’ position. See Figure 8-21.

☐ Solder the other end of the 12” cable to the speaker terminals

☐ Separate another 12 inch long 2-conductor cable from the ribbon cable. Separate the conductors by one inch at each end, and strip the insulation ¼” on both ends.

☐ Solder one end of this cable to pins 2 and 3 of the last audio jack. Mount this jack in the rear panel in the ‘Speaker’ position.

☐ Solder the other end of this cable to the speaker terminals.

☐ Separate another 12 inch long, 2-conductor section from the ribbon cable. Separate the ends about an inch, and strip the insulation from the ends about ¼”

☐ Solder one end of this cable to the speaker terminals, and the other end to terminals 2 and 3 of an audio jack.

☐ Mount this jack in the Rear panel in the ‘Speaker’ location.

Enclosure Assembly

☐ Using 5/16” 6-32 screws, attach the Front and Rear panels to the Bottom panel.

☐ Locate six of the mounting blocks. Using 5/16” 6-32 screws, mount these to the corners and centers of the upper edge of the Front and Rear enclosure panels.

☐ Using 5/16”, 6-32 screws, mount the Left and Right side enclosure panels.

☐ Connect the SMA end of the SMA to BNC cable to the SMA elbow connector. Then connect this to the SMA connector at the lower right corner of the Filter Relay board.

☐ Locate a 2.0mm two-pin cable. Connect it between the J2 at the left side of the Main board and J7 on the PA board.

☐ Plug the end of the rear-panel mounted USB cable into the USB-mini connector at the end of the Teensy 4.1.

☐ Plug the PA fan cable into the J4, noting the red wire and the + pin.

☐ Attach the top cover to the enclosure using four 5/16”, 6-32 screws.

☐ Affix the two adhesive rubber feet to the rear of the bottom panel of the enclosure.

Microphone Connections

☐ A microphone may be configured for the T41 by connecting to a 3.5mm stereo jack. The connections should be verified with your mic and a DVM using the information in Table 8-5.

Table 8-5. MiC connections

Contact	Conductor Number	Jack Contact	Function
Shell	1	1	Gnd
Ring	2	2	PTT
Tip	3	3	Mic

Congratulations, your T41-EP is assembled and ready to align!!

Section 9: Installing the Arduino IDE, the Teensy Software Patch, and Non-Standard Library Files

By
Jack Purdum, W8TEE

These directions are given so that, if you need compiler/library support, we are working from the same installation parameters.

1. Download the IDE from:

   https://www.arduino.cc/en/software

2. Create a new directory for the IDE on your hard drive. We suggest you use the current IDE release number in the name:

   C:\Arduino2.2.1

3. Select the download that matches your needs from the Download Options. I selected the

   Windows Zip file, as seen in Figure 9-1:

Figure 9-1. The Arduino IDE download screen.

4. The site now asks you for a donation to support the Arduino IDE. Personally, we donate each time we download a new release. we hope you will do the same so they will continue to improve the IDE. Click one of the two options ("Just Download", "Contribute & Download"). If you click Contribute, you are given payment options. If not, it proceeds to Step 5.
5. What happens next can depend on your computer. After the Download/Contribute option, my computer immediately started copying the IDE Zip file into the Downloads directory of my Surface. If that happens on your system, after the download completes, copy the Zip file to the directory you created in Step 2 (C:\Arduino2.2.1). On my main computer, the Arduino site prompted me to enter the directory where I want to install the IDE. In this case, I entered the directory from Step 2 (C:\Arduino2.2.1)

6. At this point, you should have the Zip file located in your IDE directory (C:\Arduino2.2.1).
7. Using the utility that allows you to manage your files (e.g., Windows Explore if you're using Windows), move to your IDE directory. Click on the Zip file you just downloaded. At the top of the page of Windows Explore, you will see an “Extract All”. Click on this to expand the contents of the Zip file. It will ask you to “Select a Destination” for the extracted files, and will have already filled in the C:\Arduino2.2.1) directory. Click “Extract” at the bottom of the dialog box to expand into the Arduino1.8.19 directory.
8. Go get a cup of coffee and a sandwich, as the extraction can take a while.
9. When you are done, the directory will have an entry that is the same as the Zip file name you just extracted (e.g., arduino-2.2.1)). Double-click on that directory and you will see something similar to Figure 9-2:

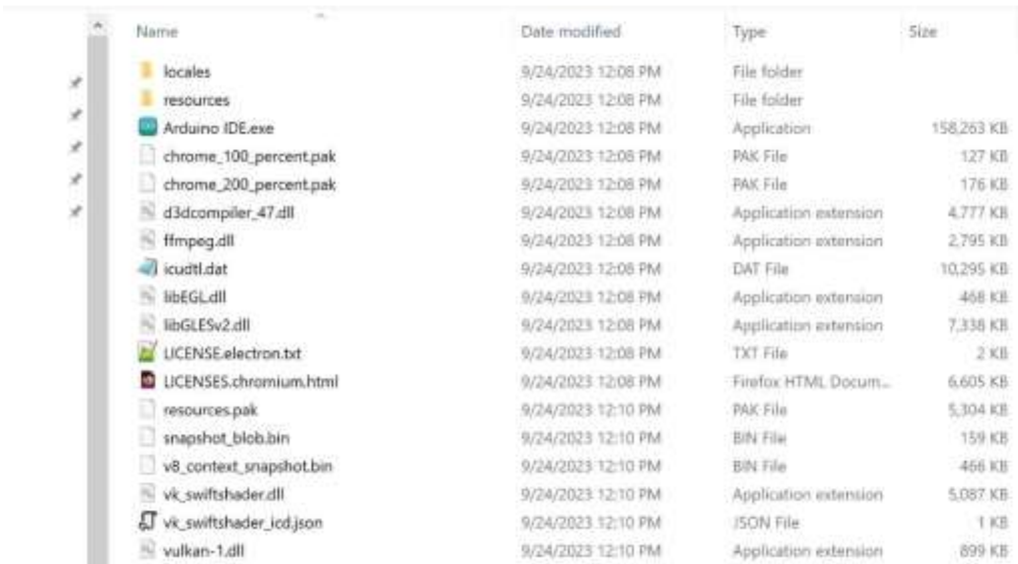


Figure 9-2. The Arduino IDE installation subdirectory.

10. Now press the Ctrl-A keys at the same time. This highlights all of the files you see above. Now press the Ctrl-C keys at the same time. This copies all the highlighted files into the Window Copy buffer.
11. Now move “up” on directory so you are in your root IDE directory (i.e., C:\Arduino2.2.1)). Now press Ctrl-V, which will copy the previously-highlighted files into the root directory. If done correctly, you should see something similar to the image below. Note that files are now in your Arduino2.2.1 root directory on drive C
12. The IDE is now installed.

Installing the Teensy Software Patch

The T41 uses the Teensy 4.1 microprocessor from PJRC rather than one of the Arduino family of microprocessors. Because the two microprocessors respond to different instruction sets, we need to use core and supplemental libraries designed for the Teensy 4.1 microprocessor. This requires you to install a “software patch” supplied by PJRC. This is a simple process and only takes about a minute. Just follow the following instructions:

1. In the figure above, the third file down is name Arduino IDE.exe. That is the file that holds the Arduino IDE program. Double-click on that file. This will load and execute the Arduino IDE. Be patient, the IDE is a big program and it does a lot of "set up stuff" when it starts. It's worth the wait.
2. The image below is the upper-left corner of the Arduino IDE. I have already clicked on the Boards Manager icon, which I have circled in yellow in Figure 9-3:

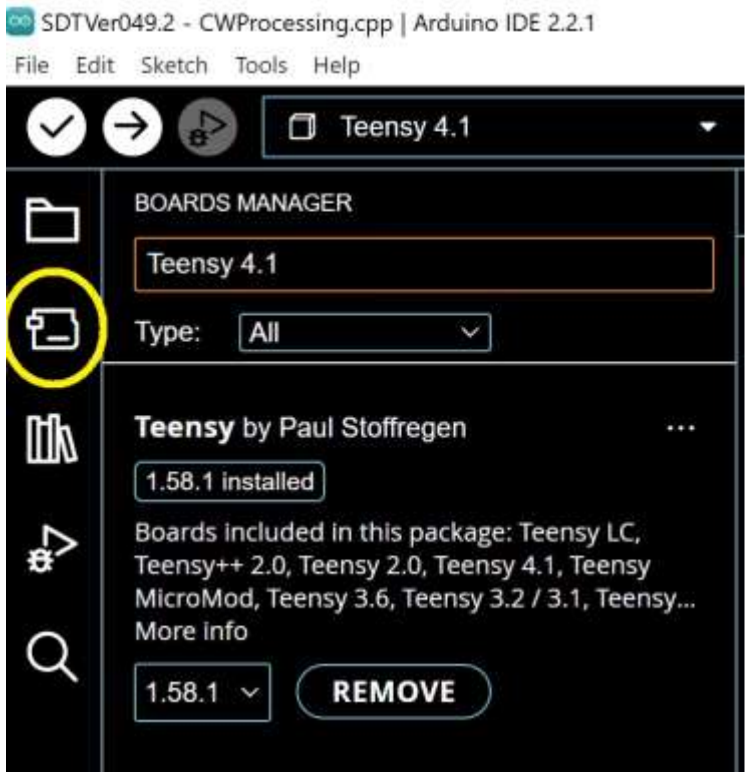


Figure 9-3. Upper-left corner of Arduino IDE (with Preference set for Dark High Contrast).

3. In the input field (outlined in red) of the Board Manager, I have typed in the name "Teensy 4.1". The IDE quickly located the patch for the Teensy. This version of the patch is 1.58.1. Because I have already installed the patch on my system, it says "installed" after the patch name. Because you have not yet installed the patch, you are given the option to install or remove the patch. Select the install option. BTW, Paul Stoffregen is the principal and brains behind the Teensy family of products. You should learn his name as you never know when someone will ask you who he is at some social event.
4. When the Board Manager finishes, the Teensy core functions and a number of supporting libraries will have been automatically installed for you. However, you will see momentarily that the T41 requires some additional libraries that are not found in the standard Arduino or Teensy installation.

Test the Install

To see if your install work, load the Arduino IDE.exe program file. Click the *File → Examples → 04. Communication → ASCII Tabl*e example program, as shown in Figure 9-4:

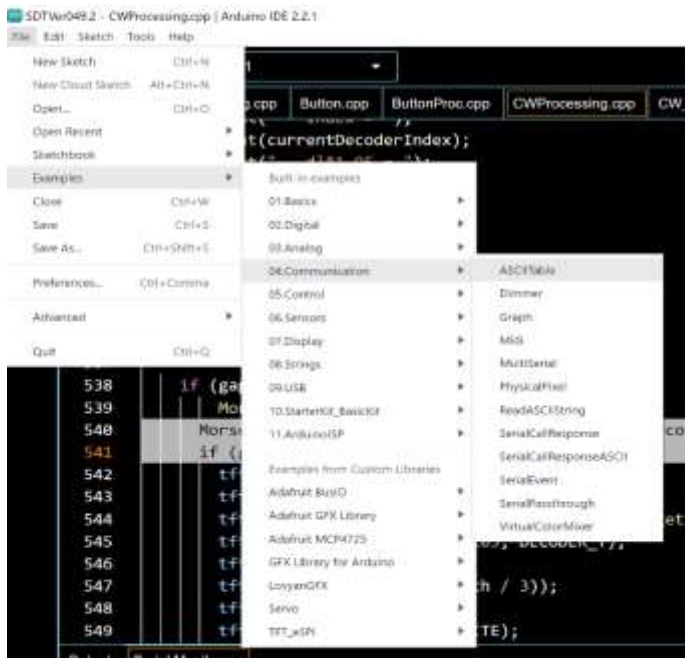


Figure 9-4. Selecting the ASCIITable example program.

Next, you need to tell the IDE which microprocessor board is being used as well as the port to use to upload the code from your PC to the Teensy. First, attach a USB extension cable to the back of the T41 and attach the other end to one of your PC's USB ports. Now touch the *Tools → Port* menu selection and select the port that is connected to the T41 and the Teensy as shown in Figure 9-5. I use COM4 for the *port*.

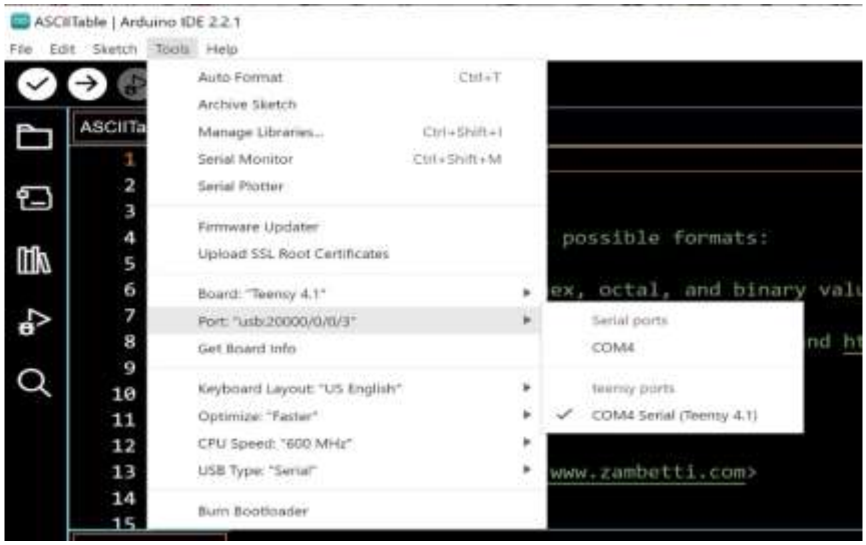


Figure 9-5. Selecting the PC post for software downloading to Teensy.

Right above the Port selection is the Board selection option. I have already set this to Teensy 4.1.

Activating the Serial Object

Next, click on the Serial object icon found in the upper-right corner of the IDE. See Figure 9-6.

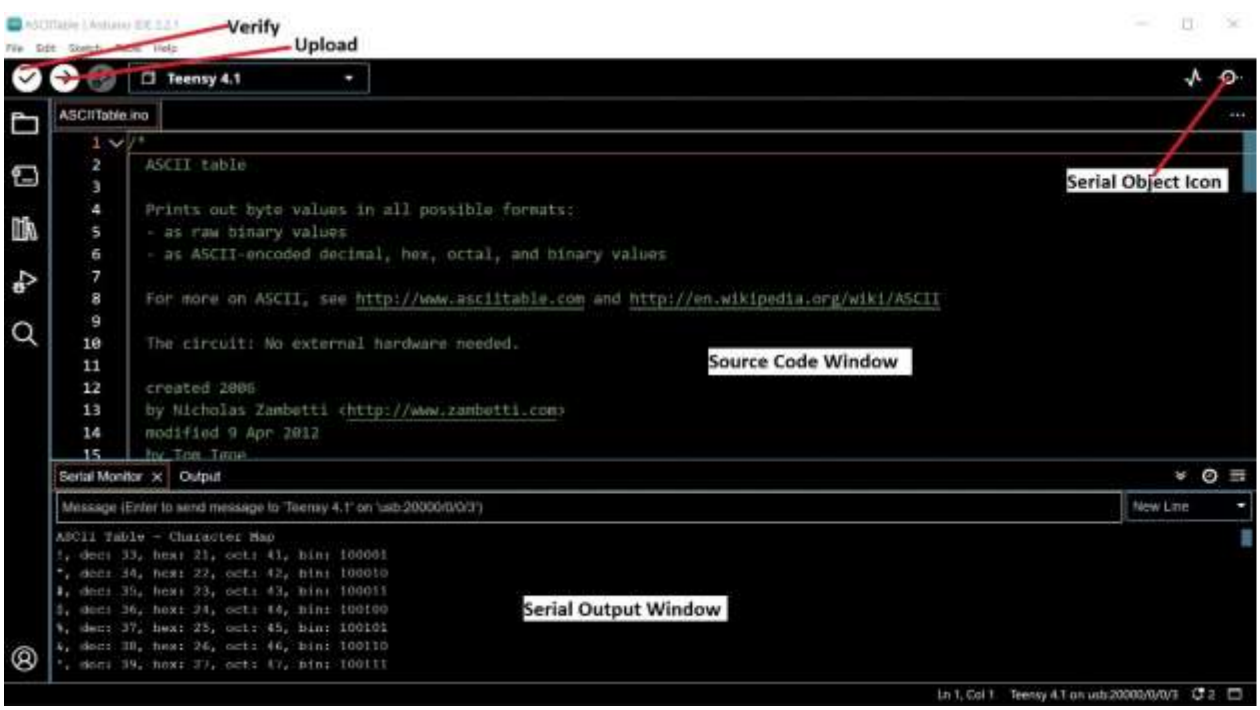


Figure 9-6. Activating Serial object with Serial Object icon.

You are now ready to compile and upload the ASCII Table program to the T41. Click the Compile/Upload icon (labeled Upload above), which is found in the upper-left corner of the IDE. IN a few moments, the program output appears in the Serial Output Window. (The Verify icon only compiles the program but does not upload it from your PC to the T41.) Congratulations! You just compiled and ran your first Teensy program (...and you thought you couldn't program!)

Non-Standard Library Installations

Most of the library files needed by the T41 are included with the standard Arduino IDE and Teensy software patch installations. However, some additional files are not part of the “standard” installation processes. We call these “non-standard” libraries. Non-Standard libraries are identified in the SDT.h header file in the T41 project's source code. For example, line 32 in that header file has the line:

#include <si5351.h> // https://github.com/etherkit/Si5351Arduino

The comment at the end of that line is the URL where you can go to download the missing library file.

The Library Manager starting with IDE 2.0 has significantly simplified the task of adding a non-standard library. In the figure below, I have circled the Library Manager icon in yellow. Click on the Library Manager icon to activate that feature.

Now type in the name of the library you are trying to install. You can determine the library name by the primary file name of the library to load (e.g., si5351). Type that primary file name into the Library Manager's input field (i.e., the red box) and press Enter. The IDE will search for the file and display all matches it finds. In the example below, it finds three entries, of which the third one matches what we see on line 32 in the SDT.h header file. Click the install button and the Library Manager will install the missing library.

Brackets versus Quotes on #include Preprocessor Directives

There are two flavors of *#include* preprocessor directives: 1) *#include <libraryX.h>* or 2) *#include “libraryX.h”*. These directives behave a little differently after the Teensyduino software patch is installed. These differences are explained below:

#include “libraryX.h”-- This causes the compiler to first look in the T41 project directory for the libraryX.h file. If it does not find the file in the T41's project directory, it then searches the Teensy's library directory
C:\Arduino1.8.19\hardware\teensy\avr\libraries

#include <libraryX.h>--This causes the compiler to first look in the Teensy's library directory and if it doesn't find the file, it looks in the Arduino library directory:
C:\Arduino1.8.19\libraries

If the compiler ever says “File not found”, it may be you have not yet installed one of the non-standard libraries.

Here is an optional method for Installing the Ardunio IDE, patches and Libraries.

Making a Second Arduino IDE Installation

Colin Kaminski
Jun 27 #16373
Changed by Dave Hampton G3UHU 28 June 2022
(Formatted for WORD and spell check vk3pe. July 2022)

Here are some Windows 10 instructions for isolating the libraries between Arduino projects. It should be noted that there are better ways, like having libraries attached to the sketch but this is more foolproof for people who don't use the Arduino IDE every day. I personally have to do this because updating the Arduino IDE breaks some work code I maintain.

1 CREATE A FOLDER FOR THE IDE :

1a Open File Explorer, select This PC > Local Disk(C) and create a New Folder

1b Rename the folder arduino-T41-EP

2 CREATE A FOLDER FOR THE T41 CODE:

2a Open File Explorer, select This PC > Local Disk(C) and create a New Folder

2b Rename the folder T41-EP.

3 DOWNLOAD THE LATEST ARDUINO IDE:

3a At your web browser, go to:
https://www.arduino.cc/en/software
3b Under DOWNLOAD OPTIONS, click on Download Arduino IDE in zip format

3c Check the download status on your browser, and wait until the zip file has completed downloading:

4 EXTRACT THE ARDUINO IDE TO THE DESTINATION FOLDER (arduino-T41-EP)

4a Go back to File Explorer and select the downloaded arduino-1-8-19-windows.zip
4b Click on Extract All.
4c This opens a dialog window where you have to browse to the destination folder
which you created in step 1 i.e. C:\arduino-T41-EP,
4d Click on Extract Wait....

5 MAKE A DESKTOP SHORTCUT FOR the ARDUINO-T41-EP IDE

5a If mot still there, in File Explorer, go to Local Disk(C;) > arduino-T41-EP folder and click to open it.
5b Right click on arduino.exe and then click on Create shortcut. That creates a shortcut into the same arduino.exe folder,
5c but we want it on the desktop, so click and drag arduino.exe-Shortcut on to the desktop. Then right click and Rename it to arduino-T41-EP.

6 CHANGE SKETCH location to C:\T41-EP

6a double click on the arduino-T41-EP and decide what you want to do about any Windows Defender warnings,

6b In the IDE : File > Preferences, change Sketchbook location to C:\T41-EP
6c Click on Ok and Close the IDE
6d Re-open the IDE, (there might be some old sketch showing) but
6e File>New to create a blank sketch:
6f File>Save this should open the T41-EP folder showing the libraries and sketch folders

7 DOWNLOAD 7 zip
The T41-EP code is available as 7Z compressed files. To use those files you need to install 7-Zip
7a Use web browser to navigate to
https://www.7-Zip.org
7b Near the top is a short list of variants for different platforms. Click on Download for the exe appropriate to your platform
7c when downloaded, click on Open file
7d When the User Account Control window appears with "Do you want.....to make changes to your computer"'
7e Follow the installation instructions

8 DOWNLOAD THE LATEST T41 CODE

8a At the time of writing this is available from the files area at the SoftwareControlledHamRadio at groups.io in V010 Files.7z Use your browser to navigate there
8b Right click on the file name and mouse over 7-Zip, in the sub-menu, click on Extract
8c Under Extract to:
Browse to, or enter, C:\T41-EPvV010Files\ and click on OK.
8d In the C:\T41-EP folder, click on V010 Files which you will see contains another folder also called V010 Files.
Open this folder, you will see 5 folders, double click on V010 Source Code and you can see it contains the folder SDTVer10

8e Double click on that file name and you will see a heap of files, one of which SDTVer010.ino

9 DOWNLOAD AND INSTALL TEENSYDUINO
If open, Close Arduino-T41-EP

9a use your browser to navigate to https://www.pjrc.com/teensy/teensyduino.html
89b On the left under Teensy, click on Download+Install. On the next page, click on Windows XP.....11 Installer.
TeensyduinoInstall.exe will be in the downloads folder. double click on the file name
and follow the instructions until Choose Arduino location to install Teensyduino
where you should browse to C:/arduino-T41-EP/arduino-1.8.19/ then click next until Finished - click Done

10 DOWNLOAD AND INSTALL si5351 library

10a Start arduino-T41-EP
10b Navigate to https://github.com/etherkit/Si5351Arduino
10c Follow the instructions under Library Installation, second paragraph
10d Check that C:\T41-EP\libraries includes Etherkit_Si5351 folder and rename that as Si5351

11 DOWNLOAD AND INSTALL Rotary library

11a First, use File Explorer to check in C:\arduino-T41-EP\arduino 1.8.19\libraries
and in C:\T41-EP\libraries that there is no existing Rotary library.
11b Navigate to https://github.com/brianlow/Rotary
11c Click on the Code button and then Download ZIP
11d When the browser download status is complete, click on the file icon and
Rotary-master.zip will now be visible in File Explorer
11e Click to highlight the filename and then cick Extract and then extract all
In the pop-up widow , browse to C:\T41-EP\libraries and click Extract.
11f Rename the Rotary-master folder to Rotary

12 DOWNLOAD OTHER LIBRARIES

12a Start arduino-T41-EP

12b Sketch > Include library . Manage Libraries
12c When Library Manager is displayed type Adafruit_GFX into the entry box at the top. Adafuit GFX Library will appear in a panel lower down.
12c move the mouse to that panel Click Install. A dependencies panel will appear. Click Install all.
The Adafruit GFX library will be installed into C:\T-41EP folder

12d Close the IDE
12e In File Manager, go to C:\arduino-T41-EP\arduino-1.8.19\hardware\teensy\avr\libraries folder and open it.
Select the following folders:
- Audio
- Bounce
- Encoder
- Metro
- SD
- SPI
- Time
- Wire

and copy/paste them to C:\T41-EP\libraries
12f In File Manager, go to C:\arduino-T41-EP\arduino-1.8.19\hardware\teensy\avr\cores\teensy4 folder and open it.
Select the following files:
- arm_const_structs.h
- arm_math.h

and copy/paste them to C:\T41-EP\libraries
12g In File Manager, go to C:\arduino-T41-EP\arduino-1.8.19\hardware\teensy\avr\cores\teensy4\avr folder and open it.
Select the following file eeprom.h
and copy/paste it to C:\T41-EP\libraries

*** Still need to find:
- stdio.h
- stdlib.h
- string.h
- util/crc16.h
- utility/imxrt_hw.h

13 MAKE THE STATISTICS PANE OF THE IDE A BIT EASIER TO READ

13a Using File Explorer, get to:
C:\T41-EP\arduno 1.8.19\lib\theme\theme.txt and open it in Notepad.
Save a copy like theme-original.txt

copy a white foreround number .....

14 FINALLY - LOAD THE T41 SKETCH INTO THE IDE

14a Double click on the arduino-T41-EP shortcut icon on the desktop The IDE is displayed
14b File > Sketchbook > V010 Files > V010 Files > V010Files > V010SourceCode >SDTVer010
That REALLY ought to be a shorter path !!!
14c The first tab shows the code for SDTVer010, i can’t read the titles of the other tabs, got to mess with theme. txt some more, I guess.

15 COMPILE THE CODE

Custom Header File: MyConfigureFile.h

The SDT.h file contains all of the overhead information that is needed by the T41. The MyConfigurationFile.h is a custom header file that stores information that is unique to your particular T41. The contents of that file is shown in Figure 9-7:

```
//====================== User Specific Preferences =============

#define DEBUG      // Uncommented for debugging, comment out for normal use
#define DECODER_STATE              0         // 0 = off, 1 = on
#define DEFAULT_KEYER_WPM          15        // Startup value for keyer wpm
#define FREQ_SEP_CHARACTER         '.'       // Period, space, or combo
//                    Name you gave to BMP map file. Max is 50 chars
#define MAP_FILE_NAME              "Cincinnati.bmp
#define MY_LAT                     39.07466   // Coordinates for QTH
#define MY_LON                     -84.42677
#define MY_CALL                    "W8TEE"    // Default max is 10 chars
#define MY_TIMEZONE                "EST: "    // Default max is 10 chars
#define PADDLE_FLIP                0          // 0=right paddle=DAH, 1 = DIT
#define STRAIGHT_KEY_OR_PADDLES    0          // 0 = straight, 1 = paddles
//          The number of milliseconds to leave error message on screen
#define SDCARD_MESSAGE_LENGTH      3000L

//#define SD_CARD_PRESENT          0          // 0 = NO CARD, I = CARD

//====================== System specific ===============
#define CURRENT_FREQ_A             7200000     // VFO_A
#define CURRENT_FREQ_B             7030000     // VFO_B
#define DEFAULT_POWER_LEVEL        20 // Default power level
#define DEFAULTFREQINCREMENT       4  // (10, 50, 100, 250, 1000, 10000Hz)
#define FAST_TUNE_INCREMENT        1  // Default from above for fine tune
#define SPLASH_DELAY               4000L   // Splash screen
//                    Default 40M band. see around line 570 in SDT.h
#define STARTUP_BAND               1
#define CENTER_SCREEN_X            400
```

```
#define CENTER_SCREEN_Y           245
//                   Use in drawing QTH map
#define IMAGE_CORNER_X
#define IMAGE_CORNER_Y
#define RAY_LENGTH                190
// ==== Pick one of the following encoder configurations
//#define                           NORM_ENCODER
#define                           FOURSQRP
```

Figure 9-7. The MyConfigurationFile.h.

You can fairly easily determine what new values must be substituted for things to look right (your call and longitude/latitude).

We have assumed that no micro SD card is present in your T41. However, you should consider the investment in a micro SD card (< $5.00) because we do save the EEPROM information on the SD card for backup, plus you can use the Bearing feature of the T41. Finally, it's easiest to put in before the case is completely assembled.
Once you have customized the *MyConfigurationFile.h* to your satisfaction, compile and upload the T41 source code and see how you like your changes. You are how ready to align the transceiver.

If the compiler cannot find the file in either of these two places, it sets your computer on fire. Naw, just kidding. However, it is important to note that the presence of the Teensy software patch does impact the way the compiler looks for *#include* files.

This completes the software installation.

Section 10: T41 Calibration Features

The T41 Calibration comprises 5 major steps:

1. Frequency Calibration.
2. Transmit IQ optimization.
3. Receive IQ optimization.
4. CW Power Level.
5. SSB Power Level.

For the best results, these steps should be performed in the order shown above, starting with frequency calibration using WWV or other signals.

T41 Frequency Calibration

The frequency calibration routine sets the T41 internal reference clock to within a few Hz of the US NIST broadcast at 2.5 MHz, 5 MHz, 10 MHz, 15MHz and 20 MHz under the call sign of WWV. To simplify the calibration process, two new features are used; SAM AM detection and Direct Frequency input to set the T41 receive frequency. More about that after we describe the overall frequency calibration process.

T41 Frequency Calibration Process

The T41 clock generator depends on a 25MHz crystal to create accurate and stable frequency readout and transmission. One must calibrate the T41 clock using a standard frequency source as a comparison in order to achieve accurate tuning. It is normally desirable to have a receiver frequency readout accurate to a few Hz. Even a 50 Hz inaccuracy is objectionable in SSB transmissions. To put this in perspective, a 0.1% difference in a 14MHZ signal is 14 KHz. 10 Hz accuracy is within 0.00007%! There are two parts to achieving good frequency accuracy:

1. Access to an accurate frequency reference source
2. The ability to actually measure the difference between the reference and the T41 displayed frequency

Accurate frequency sources are readily available in the form of modern digital signal generators, standard sources such as Rubidium Frequency Standards, or on-the-air sources such as WWV in the USA and similar radio time/frequency services around the globe.

WWV broadcasts a variety of frequency and time signals on 5MHz, 10MHz and 15MHz, as well as others. WWV’s carrier is accurate to one part in 10^{14}, which is many orders of magnitude more accurate than amateur radio requirements, or for that matter, it is much better than reasonably priced signal generators or affordable frequency standards. Needless to say, WWV or similar broadcasts are our preferred signal references.

That brings us to the second part of the frequency calibration process; how to measure the difference. There are many ways to accomplish this, such as zero beating the T41 output with the standard, using the SSB detector to create an off-tuned sidetone and measuring its frequency, or using the SAM PLL error signal as a guide.

Synchronous Amplitude Modulation Detection (SAM)

SAM is one of the best methods of achieving AM detection. The approach uses a Phase Locked Loop (PLL) to detect and replace the AM carrier with a steady carrier of constant amplitude. This eliminates one of the main causes of fading and garbled detection of AM signals. While most T41 users won't be interested in shortwave DX, SAM has a major feature that makes it a very useful tool for accurate frequency calibration: a readout of the deviation in Hz of the tuning frequency of the T41 from the received WWV carrier signal. Basically, one adjusts the T41 clock until the deviation is as small as possible.

As part of the SAM detection process, the difference between the received signal from a source, such as WWV, and the internally generated frequency derived from the T41 clock is measured. The difference between the signal source (e.g., WWV) and the generated frequency from the T41's clock is called the *error signal. The error signal* is calibrated in Hz and displayed on the screen. As the T41 clock is adjusted, the error signal is updated. For normal AM detection using SAM, the tuning can be a couple of kHz away from the received signal and SAM achieves good signal detection.

For calibration purposes, you tune in WWV or other known reference at its correct displayed frequency, and then use SAM to observe the computed PLL error. The T41 frequency calibration factor is then adjusted in real time to achieve minimum frequency error. It is possible to achieve an accuracy of several Hz in this manner. The long-term stability of the Si5351 crystal is reasonably good but is somewhat temperature dependent. We have found that, if the T41 is allowed to warm up for 10 to 15 minutes prior to calibration, the frequency tuning accuracy stays to within a couple of Hz or better over many hours, as long as the ambient temperature is reasonably constant.

Frequency Calibration Steps

To perform the T41 Frequency Calibration using WWV, follow these steps:

1. Select the Direct Frequency Input routine using the "user1" button (#17).
2. Listen to the detected station signal to verify that WWV is actually tuned.
3. Now set the demod mode to SAM using the Demod pushbutton (#8).
4. Go to menu item ***Calibrate*** and select ***Freq Cal.*** Figure 2 shows the Frequency Calibrate screen.
5. Input the frequency of one of the WWV stations and press the "<" key as shown on the screen. For instance, set the Band to 40M and enter 5 for the 5MHZ WWV station. (Band selection must be high enough to allow the signal to pass through the low pass filter.) Figure 1 shows the Direct Frequency input screen. Press the Select pushbutton.
6. Above the spectrum display note the indicator (SAM) and the deviation readout. (See Figure 2.)
7. Using the Filter/Menu encoder, adjust the frequency calibration factor (displayed in the header) to achieve a minimum value of the SAM frequency error. A value of 1 or lower should be achievable.
8. Note the Freq Calibration value for future reference.

9. Press Select to store this value in EEPROM.

The T41 is now calibrated to within a few Hz and the unit is ready for the next step.

In the event that WWV is not available, any known accurate station can be used, or an accurate signal reference source, such as a Rubidium Frequency standard can be substituted for WWV. Just use a sufficiently low-level signal of no more than 1mV as input.

I/Q Receive and Transmit Calibration

The T41 uses quadrature detection and modulation to achieve demodulation of SSB and AM signals and to create SSB and CW transmission signals. It is necessary to have accurate phase shifts and equal I and Q signal levels over the frequency range of 3 MHz to 30 MHz to achieve minimum alias or "ghost" signals. These ghosts appear either as mirror images of the signal around the central DC display in the case of Receive, or as adjacent sideband signals for SSB and CW transmission. A one-time IQ calibration procedure is necessary to account for gain differences and phase shifts in the I and Q analog circuits as a function of frequency. The calibration is performed in software for each band and receive/transmit mode. This results in a matrix of 28 parameters (four for each of 7 bands).

For Receive calibration, a steady signal within the band of interest is required. One way of doing this is to use a signal generator at the T41 antenna input. Another is to use the T41 Exciter to

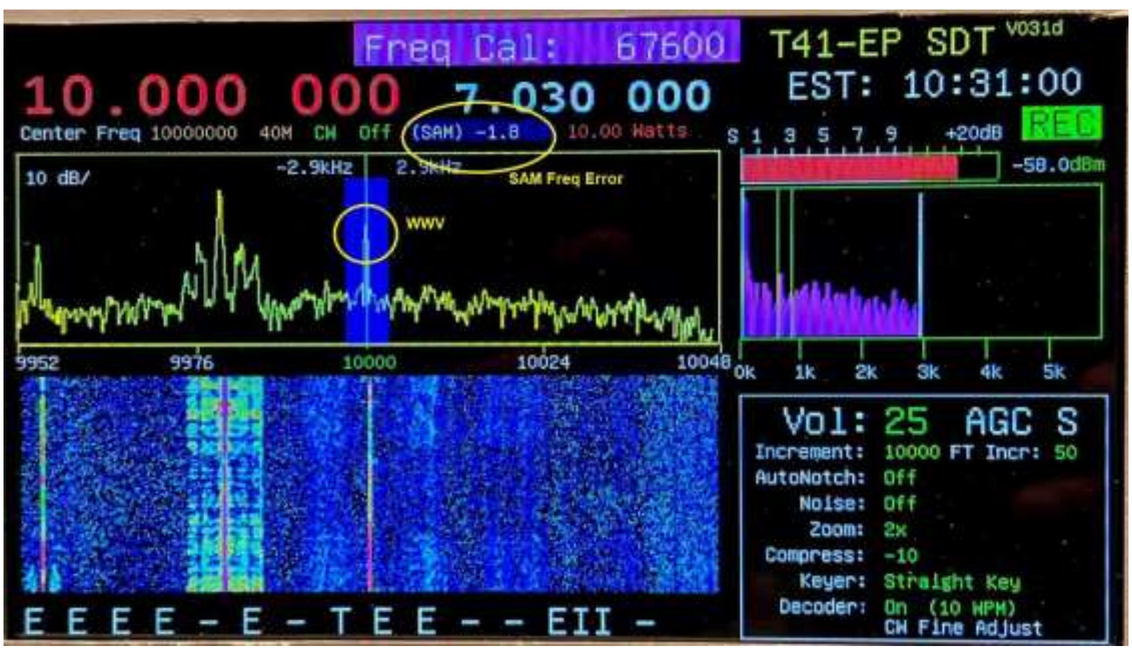


Figure 110-1: Frequency Calibration Screen

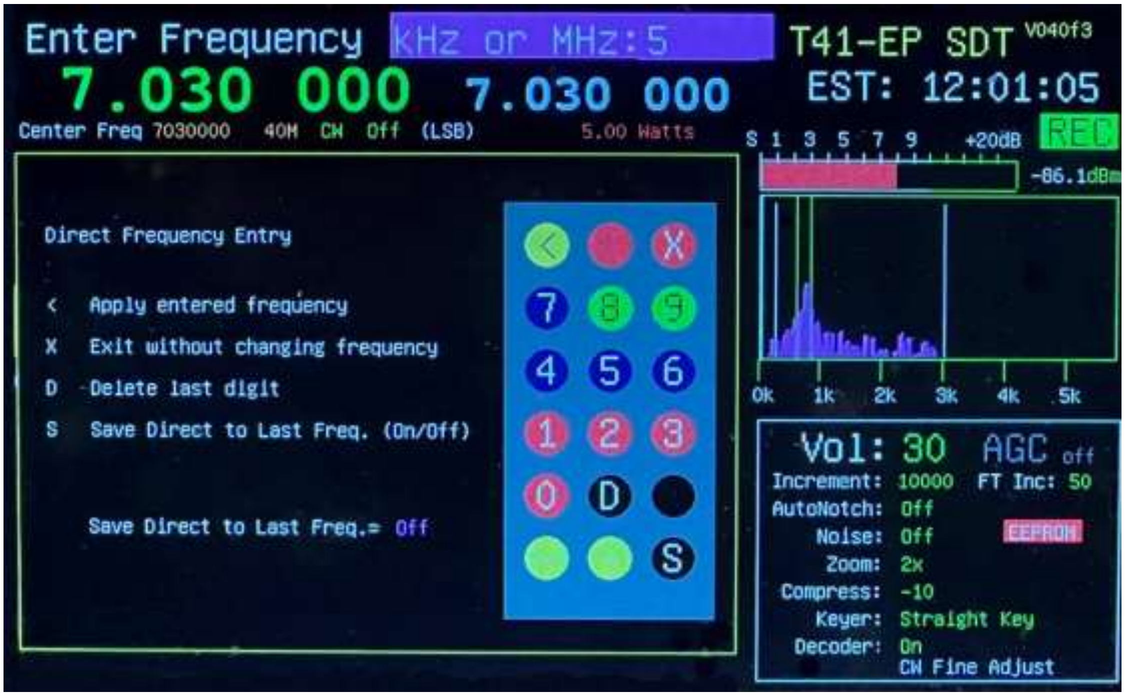


Figure 10-2. Direct Frequency Input

generate that signal. We have provided a means for T41 internal signals to be used, eliminating the need for an external signal generator.

For Transmit calibration, the required signals can be generated in software and output to allow observation of the signal spectrum. This, of course, requires access to either a spectrum analyzer or a receiver with a band scope frequency display. Alternatively, the T41 spectrum display can be used to observe the sideband created in the quadrature modulation process. This requires a special connection arrangement and a software routine that processes Receive and Excite functions simultaneously.

Modification of the QSD Board

The version V010/V011 QSD boards require a simple modification in order for the calibration process to work. This modification changes the enable signal to the Tayloe demodulator chip to be ON at all times. Version SDTVer050 software (and higher) control receiver enable via the Si5351 PLL IC (receiver LO is disabled during transmit).

A trace on the QSD PCB is cut, and a 0Ω jumper is added to the circuit. See Figure 3.

The modification allows the Receive circuits to operate at the same time as the Transmit circuits. This requires breaking a trace on the receive board and adding a shunt 0Ω resistor as shown in Figure 10-3.

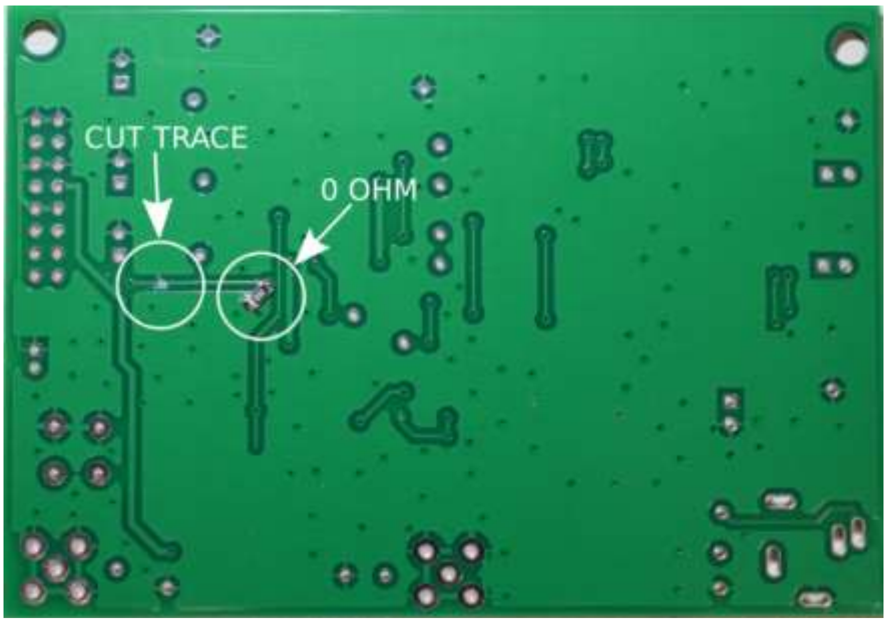


Figure 10-3. QSD board modifications (V10 and V11 boards).

Coaxial Configuration for Calibration

A temporary routing of the coaxial cables is required during calibration. Also, a 20dB to 30dB SMA attenuator is required.

The re-routing of the coaxial cables is performed as follows. Remove the cable from the input of the Power Amplifier; this is the RF output of the QSE. Remove the cable from the receiver connector on the Filter board; this is the RF input of the QSD.

Insert a 20 dB attenuator between the QSE output cable and the QSD input cable. Refer to the image below which shows the temporary cable routing. (See Figure 10-4.)

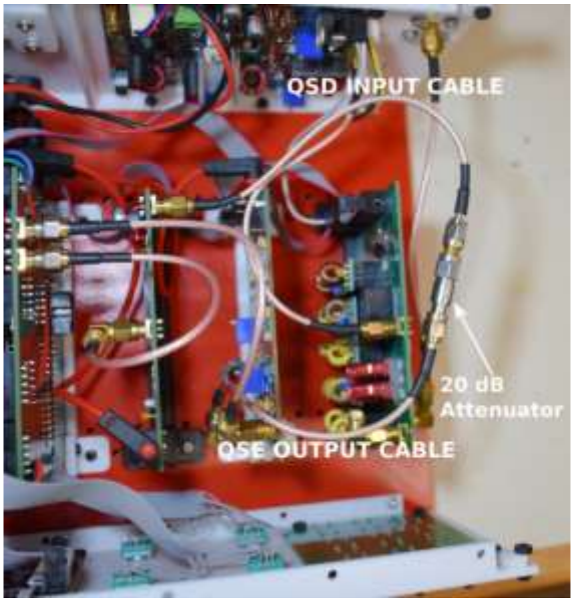


Figure 10-4. Insertion of attenuator.

Transmit Calibration

To perform T41 Transmit Calibration follow these steps:

1. Route the output QSE and input QSD cables together with a 20dB to 30dB attenuator inline.
2. Select the band to calibrate.
3. Set the T41 mode to CW. (The CW Mode indicator can be seen in the Band Info, just below the last three zeros in the VFO A frequency.)
4. Select ***Calibrate*** from the menu and select ***Xmit Cal.***
5. The display uses the 4X zoom, as shown in the Information Window in Figure 10-5. The main signal component is now 3 kHz away from the center, in the demod window. The adjacent sideband component is on the other side of center, also 3 kHz away from center. Figure 10-6 shows the display in USB mode.

Figure 10-5 Transmit IQ Calibration Screen (LSB)

6. Use the Filter/Menu encoder to minimize the alias amplitude (i.e., the number seen in the Amplitude circle in Figure 10-5.)
7. Once the alias is as low as possible using the Gain adjustment, switch to the Phase adjustment by pressing the "User1" button (#17). Again, adjust for the lowest alias amplitude. The increment value (i.e., the number just above the Amplitude value in Figure 10-5) can be toggled between 0.001 and 0.010 using the "User2" button (#18) to give you more control over the rate of change in the Amplitude number.
8. It is necessary to go back and forth between Gain and Phase to achieve the best minimum. Once the best minimum is achieved, press the Select button to save the values to EEPROM
9. Repeat for all bands.

Receive Calibration

The cable connections are the same as before, but the display is different.

1. Using the same settings as for transmit, select ***Rec Cal*** from the ***Calibrate*** menu.
2. The display uses the 1X Zoom and has several readouts and a main frequency component 24KHz above the center for LSB. There is another component at 24KHz above the carrier in a red window. The circled Alias in the red window in Figure 6 is the alias we are attempting to minimize.
3. 

Figure 10-6: Transmit IQ Calibration Screen (USB)

Figure 10-7. LSB IQ Receive Cal screen

4. The display uses the 1X Zoom and has several readouts and a main frequency component 24KHz above the center for LSB. There is another component at 24KHz above the carrier in a red window. The circled Alias in the red window in Figure 10-6 is the alias we are attempting to minimize.
5. For USB, the positions are different, as shown in Figures 10-6 and 10-7.
6. To perform the Receive IQ calibration, follow similar steps as in the Transmit case.
7. Once the best minimum is achieved, press Select to store this value in EEPROM.

Figure 10-8. USB IQ Receive Calibration screen

8. Repeat for the other bands.

When all bands have been successfully calibrated, move the QSD and QSE cables back to their normal positions.

Power Output Calibration

Because the T41 RF power output varies by band and to account for differences between individual unit performance, a provision to calibrate the power output by band in CW and SSB modes is provided. The T41 should be connected to a dummy load and an accurate RF power meter should be used to measure the output power level. If you do not own a power meter, ask at your club meeting if someone has one you can borrow. If that fails, high school, junior college, and college/university Physics labs or EE departments may have one. You should make it clear you are a licensed amateur radio operator. You may have to take your T41 to their lab, but that's better than buying a power meter.

Power calibration is done in two parts – CW and SSB for each band.

To use the Power Output Calibration feature, first go to *RF Set* menu option and select *Power Level*. Set the *Power:* to *5* or *10*. This is the power output setting in watts.

CW Power Calibration

1. Set the Mode to CW using the Mode pushbutton (#9) and select the band of interest. In *CW Options*, set the *Key Type* to *Straight Key.*
2. In the Calibrate menu option, select *CW PA Cal*. Plug in your Straight Key. Key the transmitter and write down the average power out.
3. Release the Key and change the setting to increase or decrease the power level setting to give a 5 or 10W output depending on the Power setting (referenced above). Note that you should read the power output from the power meter connected to the T41, *not* the CW PA Cal number shown in Figure 10-9. Repeat until the output is 5 or 10W. Figure 10-9 shows the CW PA output Cal screen.

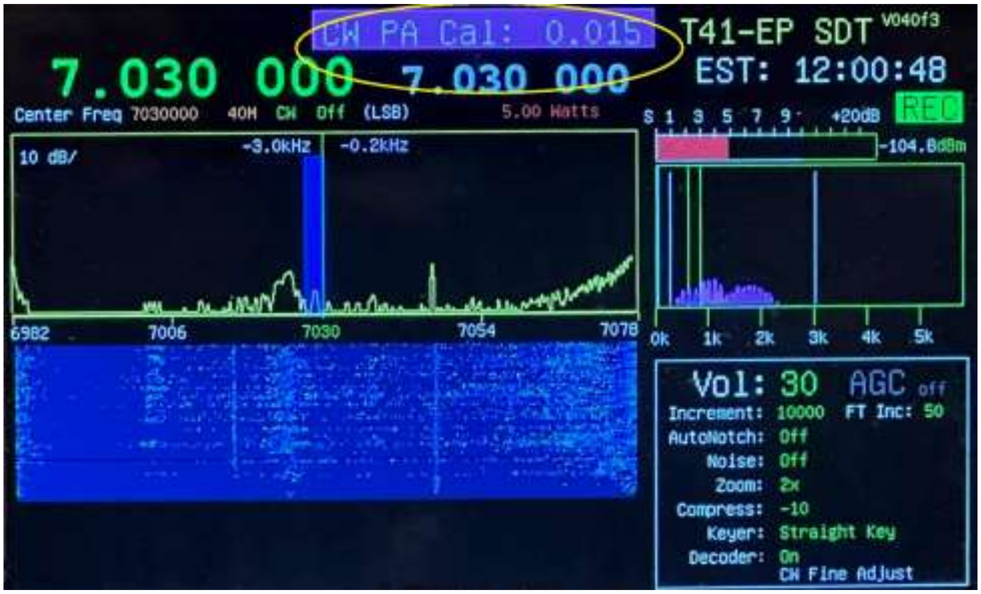


Figure 10-9: CW Power Amplifier Power Calibration

Repeat for the other bands.

SSB Power Calibration

The SSB power calibration can be done here or at the end of the calibration process. Again, note that the steps must be done for each band.

1. Next set the Mode (#9) to SSB and select the band of interest.
2. In the *Calibrate* menu option, select *SSB PA Cal*. Plug in your mic and PTT switch. Key the mic and write down the average power out with a normal speaking voice and mic position.
3. Release the PTT

4. Note that there is a separate *Mic Level* menu setting. If your output is considerably different from the CW output, you may wish to change the Mic gain before setting the SSB Power Cal.
5. Change the calibration setting to increase or decrease the power level.
6. Continue this process until a satisfactory power level for a normal speaking voice is achieved.
7. If possible, monitor the power output waveform to ensure that clipping is not occurring. If that happens, reduce the mic gain, and recalibrate the SSB power out. Note that a rig with a panadapter/waterfall display can be used as a rough indicator of signal quality.
8. Repeat for the other bands.

This completes the T41 power output calibration process.

T41 Frequency Stability

A 12-hour frequency stability test was performed using the SAM output as a gauge of stability. The T41 was first warmed up for a couple of hours and then the Si5351 clock was calibrated to better than 1 Hz compared to the 5MHz WWV signal. Figure 10-10 shows the results; plotting the SAM frequency error signal versus time at 5-minute intervals. Obvious outliers caused by signal loss were excluded. Over the 12-hour test period, the frequency was maintained to within about +/- 2Hz at 5MHz. Good enough for government work.

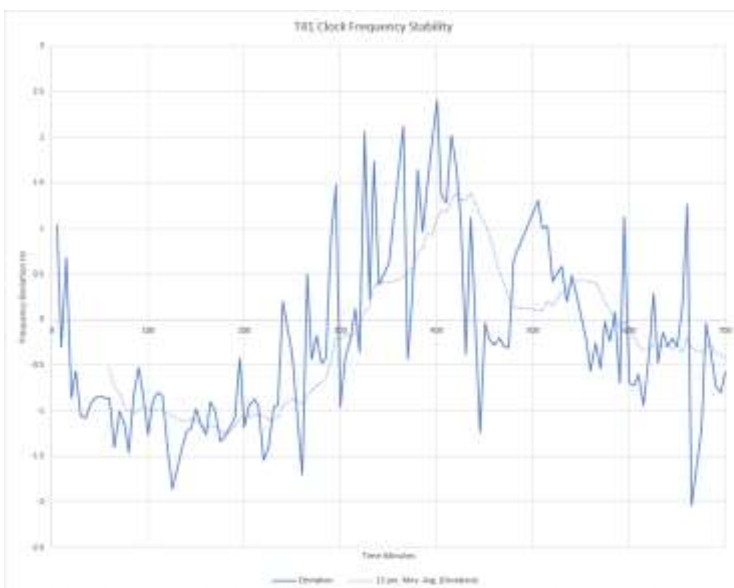


Figure 10-10. Frequency Stability

After Calibration is Finished

Let's be honest: Calibration is something you only want to do once. Alas, things can and do go wrong so we want to preserve your calibration work as best we can. There are two things we would suggest you do once you complete the calibration of your T41.

Backup EEPROM Data to SD Card

One of the Main Menu options is EEPROM. A secondary EEPROM menu option is Copy EEPROM → SD. This option simply copies the EEPROM data, which now includes your unique calibration data, to your SD card. While we haven't "required" T41 owners to install a micro SD card in the Teensy SD cardholder, it could be the best $5 insurance policy you can buy. We encourage you to install an SD card and immediately use the EEPROM copy option to move that EEPROM data to the SD card.

However, things can still go wrong. Mr Murphy could decide to arrange a power failure right in the middle of that EEPROM save and corrupt the SD data. If you're really unlucky, it might blow the Teensy at the same time! Therefore, we also encourage you to do a small modification to the EEPROM backup code.

Modify EEPROM Defaults Code

The is a function named *EEPROMSaveDefaults2()* in the EEPROM.cpp source code file. The purpose of the function it to store default data values in the EEPROMData object. These default values were selected because they contain values that are "close enough" to allow the T41 to function. They are not ideal values for your system because all of the components in your T41 is quite likely different than the ideal values for my system.

Because your system is now calibrated, it's pretty much set up using values that enables it to perform at its best. What you should do is substitute your post-calibration EEPROM data for the current default values. This means it performs at its best using *your* data, not ours.

Altering the default data is easy. However, before doing anything, copy the exiting T41 source code project files to a backup place, just in case Murphy's hiding somewhere close by. Once that's done, just follow these steps:

1. Using the Arduino IDE, load the SDT project source code into the IDE.
2. Click on the MyConfigurationFile.h tab to load the header file into the IDE's Source Code Window. If you don't see the file listed in a tab at the top of the Source Code window, click the ellipsis (…) at the end of the tabs at the top-right and click the file from the dropdown list.
3. Line #3 should be uncommented so it looks like this:

   ```
   #define DEBUG
   ```

   This turns on some debug information after the next compile.
4. Click the Serial Monitor Icon at the top right of the IDE, just above the ellipsis.
5. Compile and upload the code. The T41 software is now running in debug mode.
6. The program now automatically display the contents of the *EEPROMData* object just below the Source Code window. You may have to click on the Serial Monitor link to see the output.
7. Scroll to the start of the EEPROM data dump using the scroll bar at the right edge of the Serial window.
8. Click on the EEPROM.cpp source code tab. This moves the EERPOM source code into the Source Code window.

9. Look for the *EEPROMStoreDefaults2()* function. It should be somewhere around line 1056. (You can also type Ctrl-F and type in the function name to find it.)
10. Note how the EEPROM code in the Source Code window shows the default values *we* choose, but the EEPROM data in the Serial Output window is *your* data. Change the values in the Source Code window to your values from the Serial Output window. Take your time.
11. When you've finished, scroll through the list to check your work. When you're satisfied that the Source Code window now contains your values, compile/upload the code to your T41.
12. That's it. Now if you ever have to restore your T41 to its default values, you're using values that define *your* system, not ours.

Conclusion

This completes the calibration and backup process. Now enjoy the fruits of your labor!

Spine

4SQRP T41-EP Manual