T41-EP SDT v12 RF Module Assembly Manual

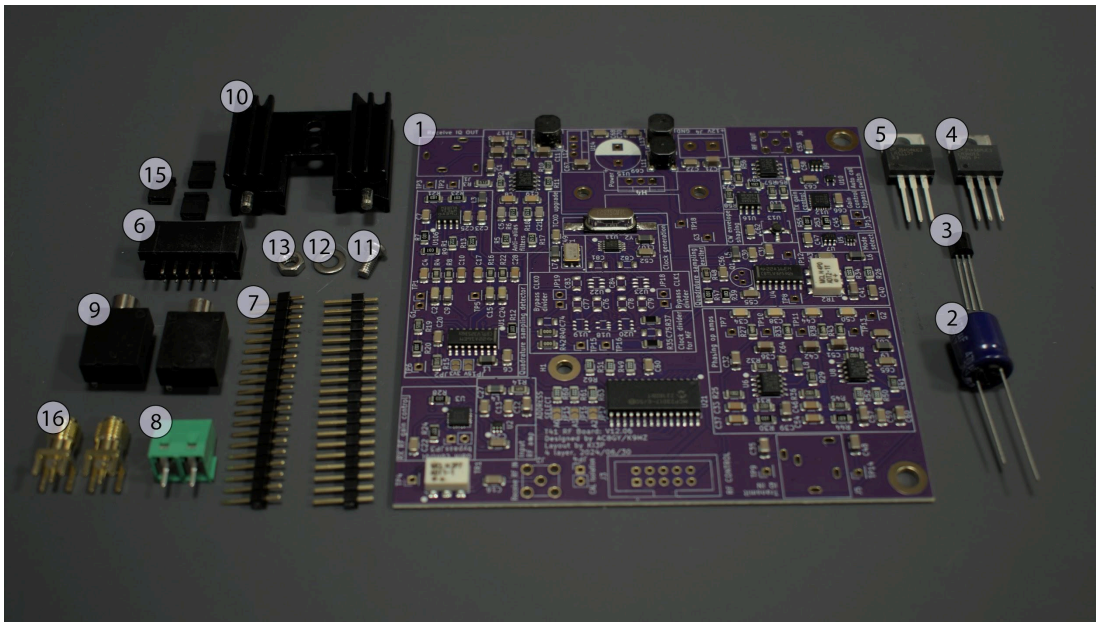


Figure 1. RF board and components.
by Justin Giorgi (AI6YM) and Dr. Jack Purdum (W8TEE); edited 2024-12-20

The T41 V12 RF board is a combined QSD (Quadrature Sampling Detector) and QSE (Quadrature Sampling Detector) with some added circuitry for gain control and CW envelope shaping. The result is a high-performance baseband transceiver for 160M-6M.

What's Included (Parts List)

1. (1) RF Board with SMD components pre-installed.
2. (1) 470uF Electrolytic Capacitor
3. (1) BS170 Transistor
4. (1) LM7805 5V Voltage Regulator
5. (1) LM1117T-3.3 3.3V Voltage Regulator
6. (1) 2×5 IDC Box Header
7. (1) 40pin 2.54mm Pin Headers
8. (1) 2pin Molex Terminal Block
9. (2) ⅛" Audio Jacks
10. (1) TO-220 Heatsink
11. (1) 6mm M3 Stainless Steel Screw

12. (1) M3 Stainless Steel Washer
13. (1) M3 Stainless Steel Nut
14. (1) 0.5g Thermal Paste Packet (not pictured)
15. (3) 2.54mm Jumpers
16. (2) SMA PCB Connectors

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

You Will Need

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

Safety Matters!

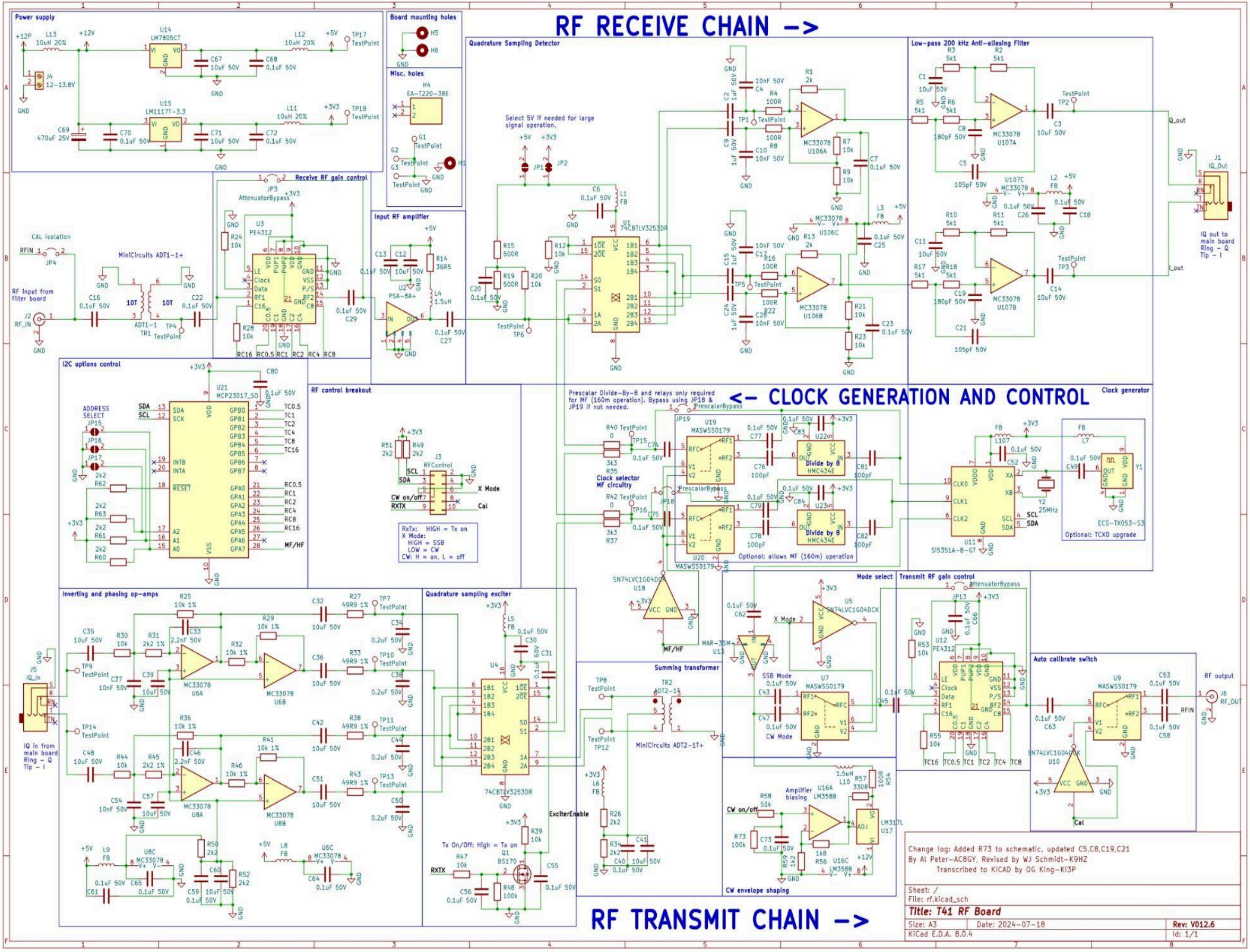
Soldering irons are hot. Everything they touch gets hot.

Have a fire extinguisher nearby!

Solder splatters. Your eyes are not easily replaceable.

Wear your PPE!

Schematic


RF RECEIVE CHAIN ->
<- CLOCK GENERATION AND CONTROL
RF TRANSMIT CHAIN ->
Power supply
Board mounting holes
Misc. holes
Quadrature Sampling Detector
Low-pass 200 kHz Anti-aliasing Filter
Receive RF gain control
Input RF amplifier
I2C options control
RF control breakout
Clock generator
Inverting and phasing op-amps
Quadrature sampling exciter
Summing transformer
Mode select
Transmit RF gain control
Auto callbrate switch
CW envelope shaping
Change log: Added R73 to schematic, updated C5,C8,C19,C21
By Al Peter-AC8GY, Revised by WJ Schmidt-K9HZ
Transcribed to KiCAD by OG King-KI3P
Sheet: /
File: rf.kicad_sch
Title: T41 RF Board
Size: A3
Date: 2024-07-18
Rev: V012.6
KiCad E.D.A. 8.0.4
Id: 1/1

Jumpers

Start by installing the five (*JP3, JP4, JP13, JP18, JP19*) 2-pin jumper positions on the board. (See Figure 3 below.) For each one, break off two pins from one of the 2.54mm pin headers provided. Place the header on the board, longer pins on the top of the board, secure with tape, and solder one pin from the back side of the board. Check and correct the alignment of the header before soldering the second pin.

Pro-Tip: Only *JP4* (which enables the calibration loopback) and *JP18–JP19* (which bypass the clock divider circuit) are required for this build. The remaining jumper positions are optional, but may be helpful for debugging or modification of the board.

Place a jumper on *JP18* and *JP19* to bypass the clock divider circuit, which is not populated on my boards. DO NOT place a jumper on *JP4* except while calibrating the radio.

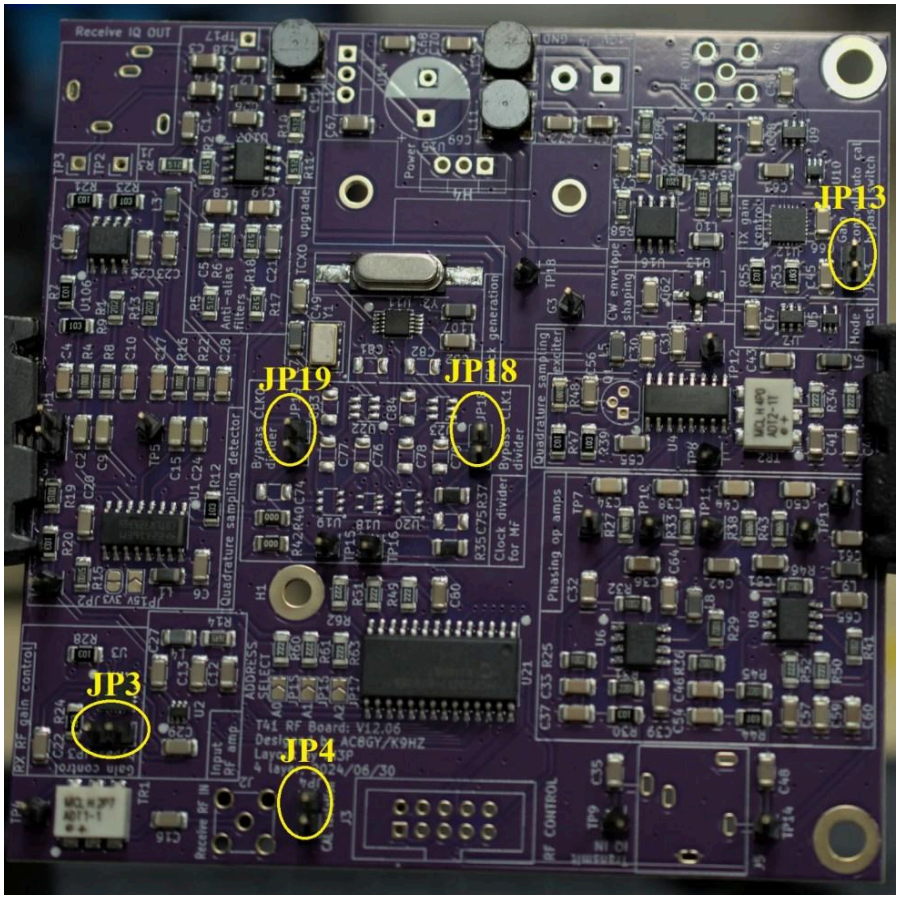


Figure 3. Jumper pin locations

Test Points

Next install the 18 (i.e., *TP1-TP18*) 1-pin test points on the board. For each test point, break off a single pin from one of the 2.54mm pin headers provided, place the pin on the board, secure it

with tape, and solder it in place from the back side of the board. Correct any misalignment by reheating the pin and adjusting it with a pair of tweezers or pliers.

Pro Tip: Locating the test points is a bit of a needle-in-a-haystack problem. It is easier to locate their approximate location by holding the board up to a light and viewing the board from the back side. Most of the test points are small, isolated, holes. (We used yellow header pins to make them stand out a bit, as seen in Figure 5.)

All of the test points are optional, feel free to leave them off the board if you prefer. More than enough pins are provided to populate all of the jumpers and test points. A ground point (*G1*) is available next to *TP1*, adding a pin here is also optional but may be useful. (Note: Figure 5 shows most, but not all of the test points populated.)

Transistor Q1

Transistor *Q1* is located near the center of the board, towards the right side in Figure 4. (See Figure 4.)

Bend the center lead of the provided BS170 transistor forward (towards the curved part of its package) slightly. This is done to accommodate the footprint of the transistor on the PCB.

Insert the leads into the pcb and press the part into place. Make sure you orient the flat side to match its silk screen on the board. (The flat side should face the nearby IC towards the right edge of the board.) The transistor will not sit flush with the board, but should be placed less than 1cm (½”) off the board. Solder the device in place from the back side of the board.

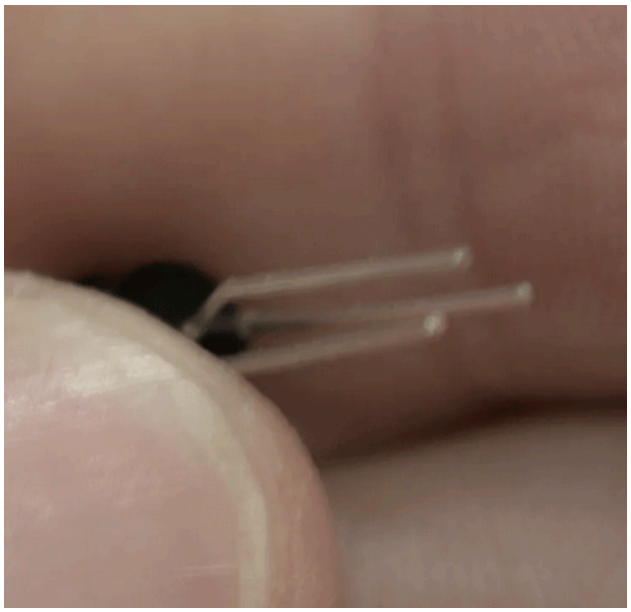


Figure 4. Bending the center lead of Q1.

Electrolytic Capacitor and Voltage Regulators

Locate the large electrolytic capacitor. The capacitor (*C69*) is positioned near the top if the board in Figure 5 and is marked with a white half-circle. The gray stripe on the capacitor defines the negative lead and should be fed through the white half-circle on the board. Once the capacitor is in place and flush to the PCB, secure it with tape, solder one pin, check/fix alignment, and then solder the remaining pin. Trim excess length from the leads of the capacitor.

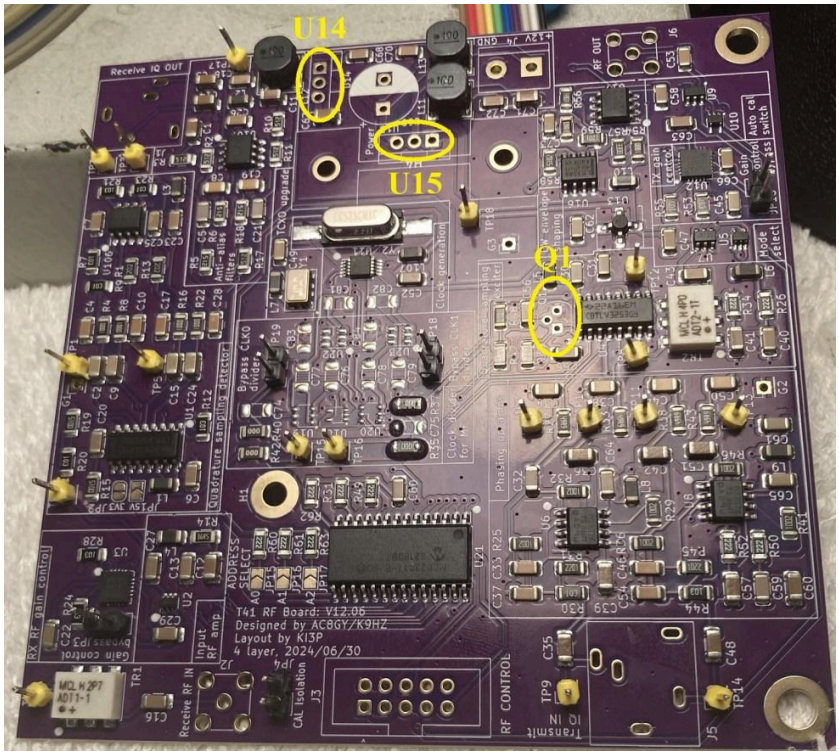


Figure 5. RF board test points, transistor Q1, and voltage regulators U14, U15.

Place the LM7805 voltage regulator at *U14* on the board. (See the top edge of Figure 5.) Be sure to align the metal tab (back) of the component with the marking on the silkscreen. (The metal tab should be close to *C69*.) Secure the component with tape, solder one pin from the back of the board, check alignment of the device, and then solder the remaining pins. Use heat and flux as necessary for any difficult pins.

Place the TO-220 heatsink in the position behind *U15*. DO NOT INSTALL *U15* YET! Secure the heatsink with tape and solder both mounting pins to the board from the back side. This may require extra heat from a heat gun and a good bit of flux but these pins will eventually solder.

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

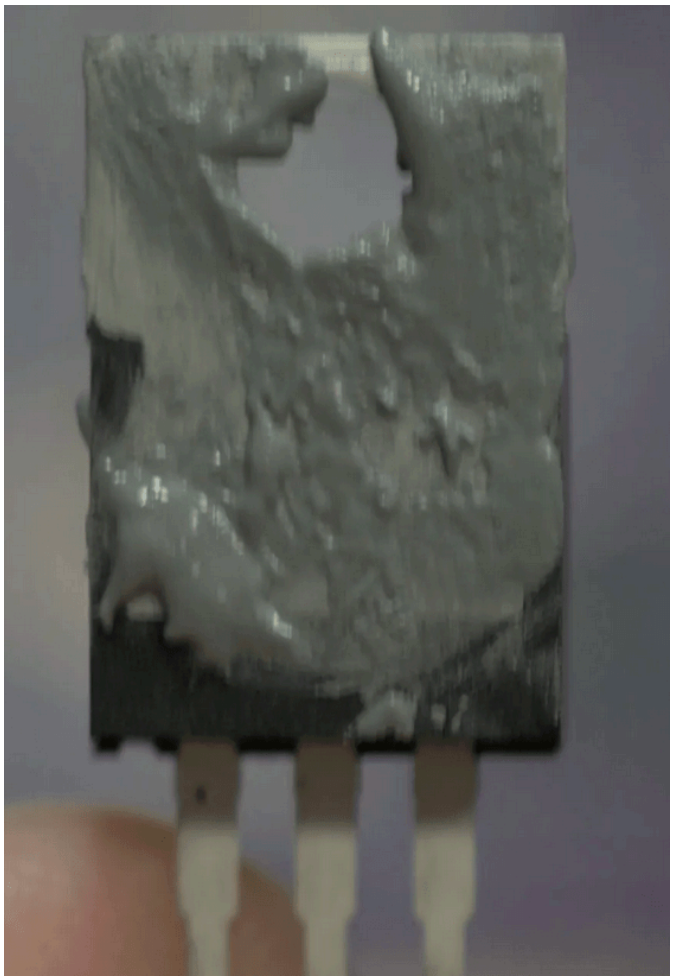


Figure 6. Applying thermal past to the metal tab of U15.

Locate the large TO-220 heat sink and *U15*, the LM1117T-3.3 3.3V voltage regulator. Take the small packet of thermal paste, snip the corner off, and apply a drop of the paste onto the metal tab of *U15*. Spread the paste to cover the back of the metal tab, as seen in Figure 6.

Insert the voltage regulator into the PCB with the metal tab against the heatsink. Take the small mounting bolt, add the washer, and thread it through the hole in the *U15* tab. Thread the bolt through the top-most hole of the heat sink and thread on the nut. Tighten securely. Using the top mounting hole supplies the greatest surface area of U15 to be in contact with the heat sink.

Solder the leads of the voltage regulator using heat and flux as necessary.

Connectors

Place one of the SMA connectors on the board, secure it with tape, and solder one pin from the back side of the board. Check and fix the alignment of the connector before soldering the remaining pins. Repeat for the second SMA connector. If you try to hold the SMA connectors in place while you solder them, you will only do that once. They get very hot.

Pro-tip: The SMA connectors may be soldered on either side of the PCB. Think about how you plan to arrange the boards in the finished radio before you solder on the SMA connectors.

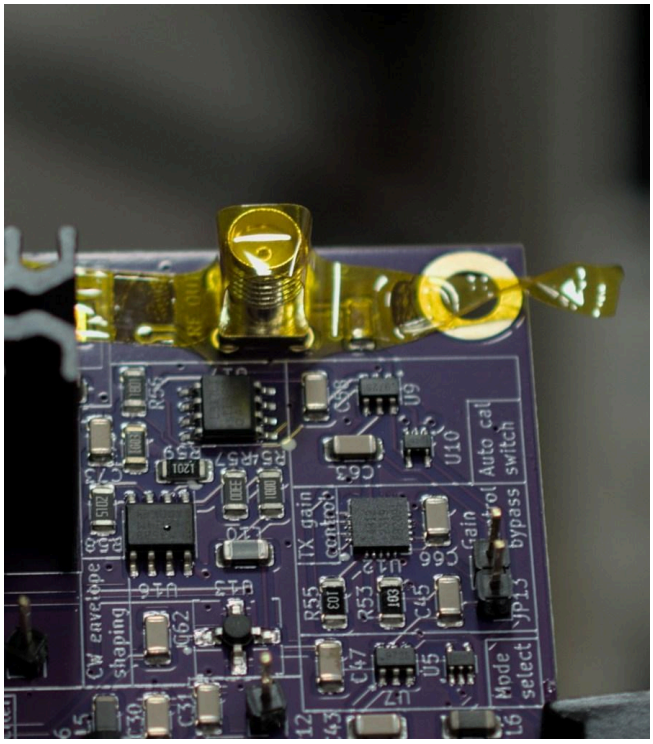


Figure 7. SMA connector on RF board.

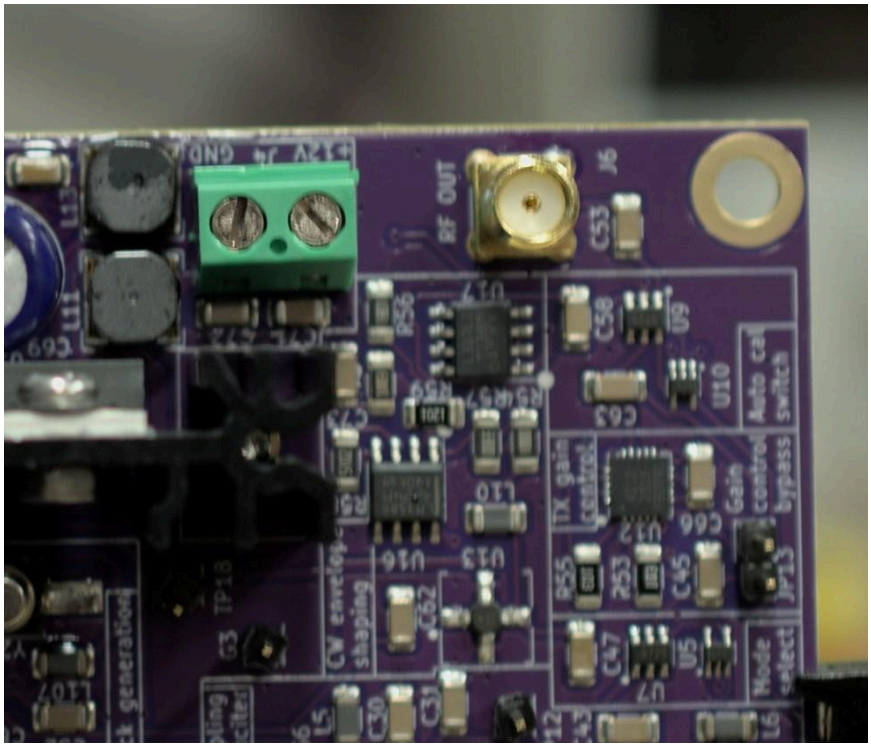


Figure 8. Molex connector on RF board.

Place the 2pin Molex connector on the board with the wire terminals facing out, secure with a bit of tape, solder one pin on the back side of the board, check alignment, and then solder the remaining pin.

Place the IDC box header on the front side of the board, making sure its header key faces the edge of the board. Secure the box header with tape, solder one pin on the back side of the board, check alignment of the header, making sure its snug to the PCB, and solder a pin opposite of the first. Check alignment again, and then solder the remaining pins.

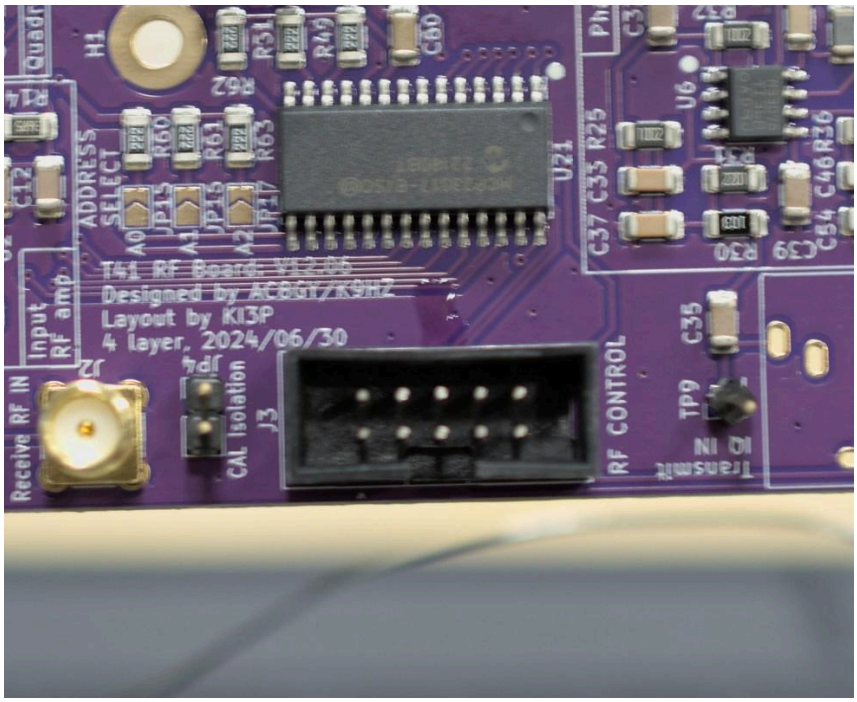


Figure 9: IDC box header on RF board.

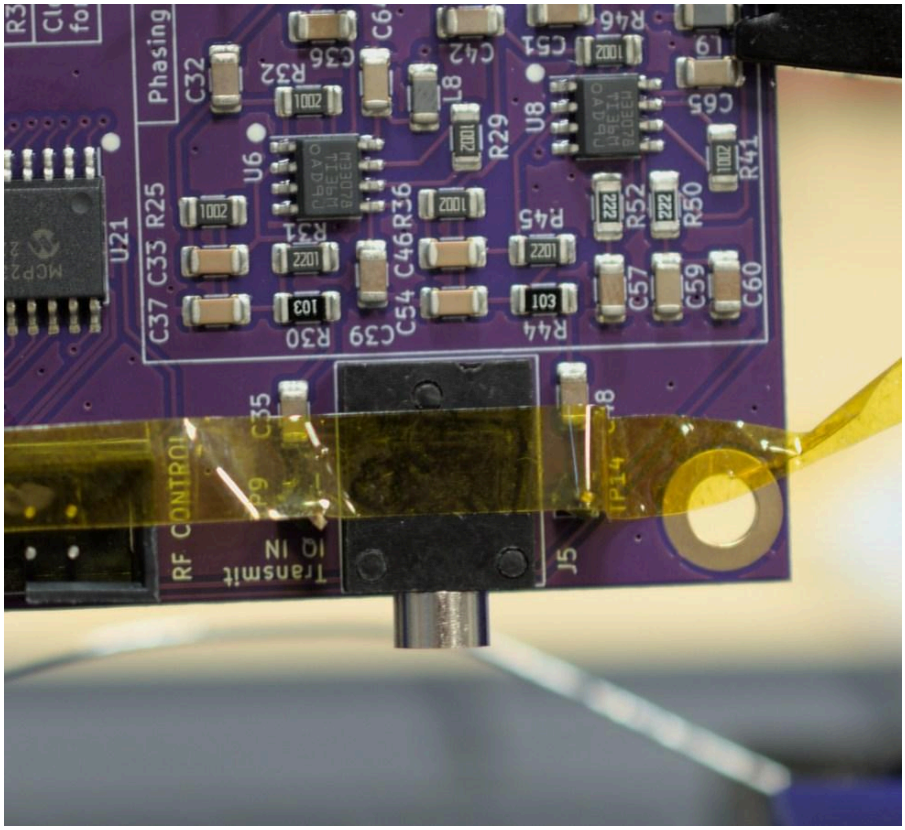


Figure 10. Stereo 1/8″ audio jack on RF board.

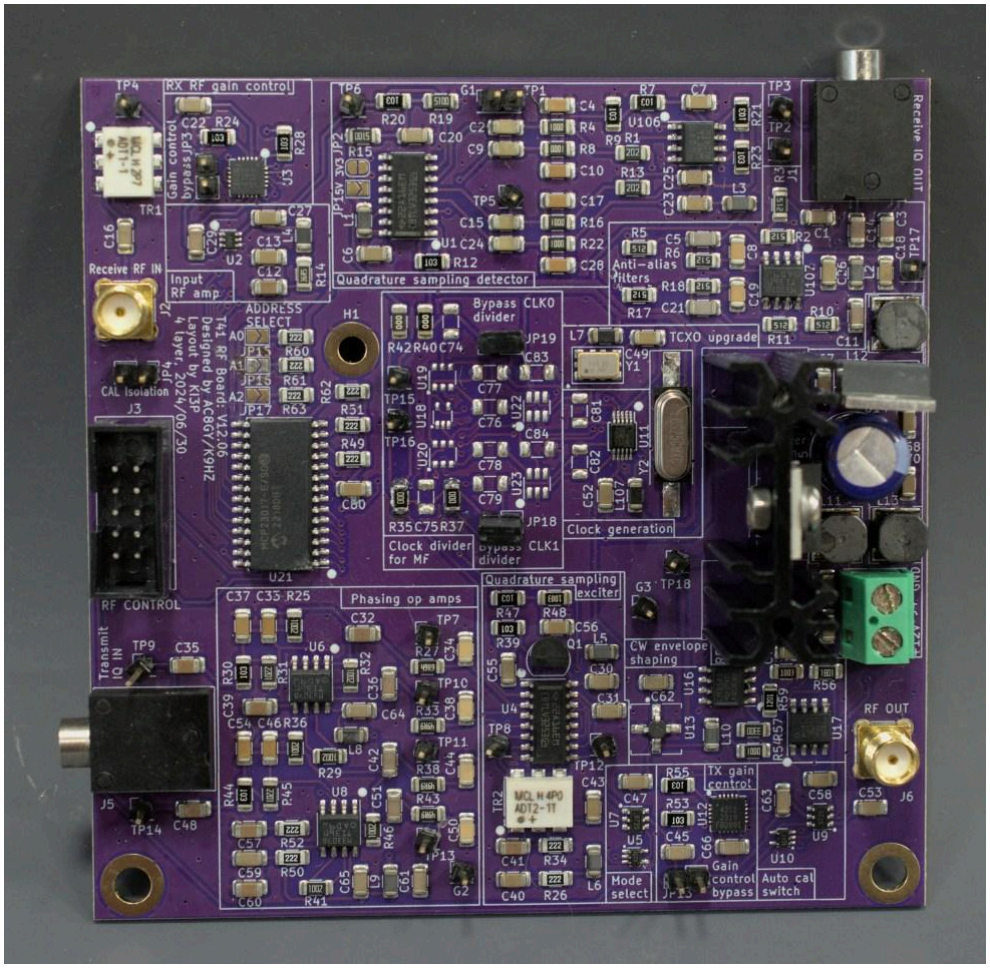
Place one of the ⅛” audio jacks on the front side of the board, secure with a bit of tape, solder one pin on the back side of the board, check alignment of the part, solder a second pin, double check alignment, and then solder the remaining pins. Repeat for the second audio jack.

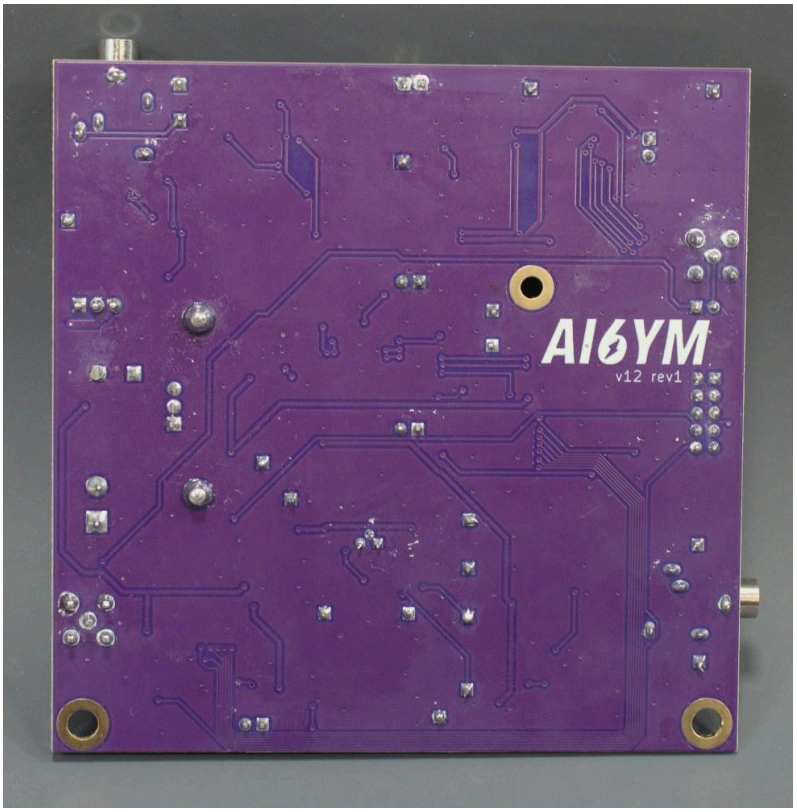
Finishing Up

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

After cleaning the board inspect all of your solder joints. Use a jeweler’s loupe to get a good view. Touch up any joints that are cold or incomplete. It may be necessary to clean, dry, and inspect your boards again. That’s fine. Make sure the joints are good before continuing.

Once you have a clean, dry, and thoroughly inspected board covered with high-quality solder joints, proceed to testing.





Testing

Bare Board Power Up Test

Set your bench power supply to 12V and set the current limit to ZERO. Connect power to your board.

SLOWLY increase the current limit on the supply. The supply voltage should be reached with minimal (<50mA) current draw.

Check for regulated voltages of 5V on *TP17* and 3.3V on *TP18* (+/- 0.2 is fine).

Calibration & Functional Testing

Before calibrating the RF board ensure that you are running the latest software release. A copy is kept on my GitHub repository which has been tested against my kits, newer versions which I have not yet tested may be available on Oliver's (KI3P) Github.

The book *Digital Signal Processing And Software Defined Radio* by Jack Purdum (W8TEE) and Al Peter (AC8GY) provides a detailed discussion of DSP and how it is implemented in the T41-EP. The book also contains a detailed description of the calibration process for older versions of the hardware. The process is also described on the SCHR site. Most of the Calibration process is unchanged for this version (V12) of the T41 hardware.

The RF boards provided with my kits include the TCXO (temperature compensated crystal oscillator) which eliminates the need for frequency calibration. By connecting the RF board to the Main board and a suitable antenna you should have a functioning receiver. Try tuning in high power transmitters such as W1AW or WWV.