BATC Kit MTK1 for MiniTiouner v1.9

Change Log

2015-08-12	V1.9	ZD2 BZW04-17 should be BZW04-17B Added advice for replacements
2015-08-04	V1.8	Correction to reception frequencies
2015-08-01	v1.7	First release

Outline

MiniTiouner is a device for receiving digital TV transmissions on the amateur radio or broadcast bands. It is designed by F6DZP and is driven by his MiniTioune software running on a PC. The PCB and some power supply modifications are by G4EWJ.

It can receive amateur TV transmissions with symbol rates (SR) down to 112k and also broadcast symbol rates up to 30M and beyond. It currently receives DVB-S QPSK transmissions only, but may receive other modes in the future with more PC software development.

Apart from an Eardatek (EDS-4B47FF1B+) or Sharp (BS2F7V0169) NIM receiver module and an FT2232H USB module, the kit contains the PCB and all the component to build a MiniTiouner. Components are mostly through-hole, apart from 4 surface mount capacitors and the sockets for the NIM. No component or pcb changes are needed to use either the Eardatek or Sharp NIM.

The PCB has been optimised for low SR reception with the Eardatek module and may require extra heatsinking for the Sharp module and for broadcast use at high symbol rates. Tests with the green pcb indicate that it is better at getting rid of heat than the prototype, so try experimenting.

These instructions relate to the NIM1473 version of the pcb and Build A components list.

Contents

The components have been sorted into bags. Check that all components are present before starting.

A:	PCB	1	
B:	NIM sockets		either 2 off 24 way or 1 off 50 way
C:	Surface mount caps	5	1 spare
D:	Resistors	6	
E:	Diodes	2	
F:	Zener diodes	2	
G:	74HC10 and socket	1 and 1	
H:	Regulators and fasteners	3	
I:	Switching PSU module	1	
J:	LED	1	
K:	Ceramic caps (leaded)	9	
L:	FT2232H sockets	2	
M:	Headers		either 2 pin and 6 pin or 2 off 2 pin
N:	Fuse	1	2 legged self-resetting type
O:	Electrolytics	5	1 spare
P:	Ferrite chokes	5	
Q:	Jumper	1	

LK1, LK2 and LK3 are wire links, some of which are fitted during testing,

Other positions on the PCB are for future development and apart from C17, not supplied.

C17:	220uF
J2:	This is an unconnected 16 pin header. It could be used to wire to points on the PCB to extract the transport stream data directly.
J3:	This connects to 5 spare I/O pins on the USB module.
C11, C12:	These are on the I2C bus clock and data lines.
H1:	This is an M3.5 hole to fit a support for the PCB.

Construction

The kit may have components separated into bags or all in one bag.

It is suggested that the components are fitted in the alphabetical order on the bags.

Apart from the NIM sockets, all components mount on the upper surface of the PCB.

There is an excellent construction video on YouTube by Colin G4KLB.

https://www.youtube.com/watch?v=vXUIQPeslwM

That was a test version of the kit and there have been a few minor changes since, so go with these instructions if there is a conflict.

E.g. Only fit the western 2 pins of J4.

Voltage testing on the NIM pads is done before fitting the FT2232H module.

The 74HC10 has a socket.

Fit LK2 instead of LK3 during testing.

The method of grounding the NIM.

A: PCB

The PCB is tinned with leaded solder and leaded solder is recommended.

Components on the PCB are numbered from left to right.

CN1 on the PCB is known as CN3 on the USB module and in some circuit diagrams.

The pcb copper coating is 2 oz, which is double the standard thickness. This should help to reduce voltage drops along high current tracks and maybe with heat conduction.

B: NIM Module Sockets

The NIM can be soldered directly to the PCB instead of using sockets. Sockets are recommended, so that other types of NIMs could be fitted in the future.

If a previously soldered NIM is being used, it is unlikely that its pins would fit into the sockets and direct soldering is the only option. If this is the case, do not fit the sockets and fit the NIM later.

The rows may have to be splayed apart slightly to fit onto the edge of the board.

If the kit has the 50 way socket:

The socket has more pin positions than are needed, so the northern most position goes opposite the NIM1 legend on the pcb and is unused..

Remove unused pins with pliers. Be guided by the pads on the PCB as to which pins to remove. The pins on the sockets are easily bent, so make sure they are all reasonably straight.

Solder the most northern pin on the upper surface and the most southern pin on the lower surface. Make sure that the socket pins are pressed down against the PCB. Melt the solder joints and make sure that the socket is square on to the pcb before soldering the rest of the pins. Go to step C.

If the kit has 2 24 way sockets:

On one socket, remove all 12 pins in one row and then 2 pins in the other row. Remove unused pins with pliers. Be guided by the pads on the PCB as to which pins to remove. The pins on the sockets are easily bent, so make sure they are all reasonably straight.

On the other socket, remove 4 pins on both rows at the same end.

Attach the northern socket (pins in both rows) by soldering a single pin at one end of the upper row and then a single pin at the other end on the lower row.

Make sure that the socket body is square on to the PCB and touching it. Melt the solder joints and adjust the socket position to get it exactly right.

On the southern socket, solder a single pin at the northern end of the socket. Make sure that the socket pin is pressed down against the PCB. The southern socket should butt up against the northern socket and be in line with it.

Support the PCB to keep it level and plug the NIM into the sockets to check that the alignment is good. Be careful not to pull the NIM away from the PCB until both ends of the southern socket have been soldered. Melt the solder joint and adjust the southern socket position to get it exactly right.

With the NIM still plugged in, solder a single pin at the southern end of the southern socket. Make sure that the socket pin is pressed down against the PCB. This pin is used for feeding 18v into an LNB, so the soldering iron is unlikely to harm the NIM..

Unplug the NIM carefully and solder all the pins on both sockets.

C: Surface Mount Capacitors

C14, C15, C16, C18.

There will be one spare (not marked on the PCB). Keep it for future use.

D: Resistors

Keep the resistor legs for later.

If you have the blue resistors, the colours are as below, but measure if in doubt.

R1, R2, R4, R6	3k3	orange, orange, black, brown, brown R6 varies the LED brightness.
R3	82k	grey, red, black, red, brown
R5	56k	green, blue, black, red, brown

E: Diodes

D1, D2 1N4007

Fit both with the band to the west.

F: Zener Diodes

ZD1	6.0v	1N5233BTR	Glass type. Fit with the band to the south.
ZD2	17v	BZW04-17B	Bi-directional. Fit either way

ZD1 catches any spikes from the output of the switcher. The prototype ran for weeks without this, so it is just a bit of insurance.

Note that there is a via hole underneath ZD1. Do not use this hole. The holes for ZD1 are just outside its pcb legend.

ZD2 is used to prevent any transients above 18v on the external supply from exceeding the input voltage rating of the switching psu module. If you plan on using an external psu of 18v - 24v, then this should be omitted.

If ZD2 is replaced by a unidirectional type, the band should go to the west.

G: 74HC10 and Socket

U1. Fit the socket with the semi circular notch to the south.

The chip itself will be fitted later.

If you are using a different 14 pin socket, or different 26 pin sockets for the FT2232H module , check that there is enough clearance below the FT2232H module first.

H: Regulators

If the regulators are going to have extra heatsinking, leave them until later.

Ensure that the 3-legged regulators are in the correct place.

Fit on the upper surface. Use heatsink paste if available. Also use heatsink paste under the screw head and nut and washer.

U2	1v adjustable	MCP1826-ADJEAT
U3	3.3v	MCP1826S-33EAB
U4	5v	MCP1826S-50EAB

It is suggested that the legs are not cut and that the minimum amount of leg protrudes below the PCB so that the regulators are lying in a 'knees up' position. This gives the option to lift the regulators in the future and fit a flat heatsink between them and the PCB.

The pins of U2 are very close together. Solder the middle pin first and then work outwards. Make sure that the back of the solder tip is clean.

I: Switching PSU Module

PSU1

Fit one resistor leg in each corner of the switcher on the PCB. There are two holes at each corner, but only one of the two needs to be wired.

The square inductor goes to the south-west.

Slide the switcher onto the legs and support it about 3mm above the PCB before soldering and cutting short the legs. Remove the support.

If you are using your own switcher, it is recommended that the pot on the switcher is removed and replaced by 62k in parallel with 220k to set the voltage to 5.5v nominal.

The maximum input voltage of the regulators is 6.0v, so it is vital that the switcher output is correct.

J: LED

The longer positive leg goes to the north, to coincide with the barred end of the PCB legend.

K: Ceramic Capacitors

C1, C4, C7, C8, C9, C10, C12, C19, C20 100nF

Fit flush with the PCB.

L: 26 Pin Sockets

CN1, CN2

Solder one pin at diagonally opposite corners and make sure the socket is flat against the PCB and parallel to the pcb legend before soldering the other pins.

M: Headers

J1	2 pin	Optionally connects LNBA and LNBB inputs (F sockets) together on the NIM.
J4	2 pin or 6 pin	Expansion for an LNBP21D LNB controller.

For J4, if the kit has a 2 pin header, fit at the western end.

If the kit has a 6 pin header**, break off 2 pins and fit at the western end.** Keep the other 4 pins for the future.

A jumper can be placed here to send the external psu voltage to LNBA, the left hand F socket. If the jumper was placed on the wrong pins, it could put 12v on the NIM and damage it.

N: Resettable Fuse

FS1 200mA

This is in the current path from the external supply to the F sockets and gives some protection against short circuits.

It may prevent tracks from melting, but it may not react fast enough to protect the diodes, depending on the external supply. An external 1A quick blow fuse is recommended.

See later for more information about powering preamps and converters.

O: Electrolytic Capacitors

C2, C3, C5, C6 220uF keep C17 for future use

These are shown on the PCB as rectangular, with a bar marking the positive end.

The capacitors themselves are marked with a dashed line at the negative end.

E.g. C2 is fitted with the dashed line to the east and C3 to the west.

If the NIM is going to be soldered, do not fit C2 until that has been done.

P: Ferrite Chokes

L1, L2, L3, L4, L5 Laird Technology 28C0236-0JW-10

Fit flush with the PCB.

Q: Jumper

Save this for later. It is used to feed voltage out of the LNB sockets for preamps.

Final Steps

If the regulators are going to have extra heatsinking, fit the regulators and do that now.

Attach psu wires to the VIN and GND pads in the south-east corner of the pcb. Connect from the underneath of the pcb if possible, to leave space for a board to fit onto the J4 header in the future.

The only components places on the pcb that should be empty are:

C11, C13, C17, J2, J3
LK1, LK2, LK3 some of which are fitted during testing
The NIM, if it is going to be soldered

C17 (220uF) and one surface mount capacitor should be left over.

Testing

The NIM, FT2232H module and 74HC10 should be unplugged.

Fix any problems before moving on to the next step.

The 5.5v switching power supply module PSU1 is still isolated from the regulators, so it can be tested in isolation.

Step 1:

Connect a 9v-15v external psu. Up to 24v should be possible, but has not been tested. Remove ZD2 for external psu voltages above 18v.

The LED should light.

If not, follow the input voltage northwards through L5, westwards through D2, eastwards through R6 and onto the north (positive) leg of the LED.

Check that D2 is fitted with the band to the west.

Remove the external psu and check the polarity of the LED with a meter. The positive end should be to the north.

Step 2:

Use the regulator screws or H1 in the north-east corner of the pcb as a ground point.

Check for 5.4v - 5.6v on the north-west pads of the switcher PSU1.

If not, check for supply voltage on the north-east pads of the switcher, which are also connected to the western pads of ZD2, D2 and R6.

Step 3:

Remove the external psu.

Fit a wire link for LK1. This connects the output of the switcher PSU1 to the 5v regulator U4. Make it a vertical loop so that it can be easily removed in the future if the switcher needs to be adjusted.

Connect the external psu and check for 5v at the test point marked TP5v towards the north-west corner of the PCB.

Put a clean resistor leg into pin 3 of CN1 (CN1 is equivalent to CN3 on the USB module pcb) and check for 5v. Pin 3 is one pin north of the south-east corner pin.

These two points are directly connected to the 5v regulator U4, so not much can go wrong. If not, check for 5.5v on pin 1 (northern) of the 5v regulator U4 and for 5v on pin 3 (southern). Check the joints on the switcher, U4 and CN1.

Step 4:

Remove the external psu.

Fit a wire loop for LK2 (this is different from the video). This connects the output of the 5v regulator U4 to the input of the 3.3v regulator U3. The input of the 1v regulator U2 is already connected to the output of the 3.3v regulator.

Check for 3.3v and 1v at the test points.

Check for 3.3v on pin 14 of U1 (southwest corner).

If any of the above is not correct, check the joints of U1, U2, U3, U4.

LK2 and LK3 form a triangular 3 pin link. LK3 can be fitted instead of LK2 to feed the 3.3v regulator directly from the 5.5v output of the switcher. This might be useful in the future to balance the current through the regulators. Only one of LK2 and LK3 may be fitted at the same time.

Step 5:

Connect the external psu to check voltages at the NIM pin pads.

Be careful not short adjacent pads together. Be extra careful if you need to do this later with the NIM connected, as supplies could be shorted together with disastrous results.

Check for 3.3v on NIM pad 11. This is the southern pin on the upper surface of the PCB.

Check for 1.0v on NIM pad 13. This is one pin north of the 3.3v pin above.

Turn the PCB over with the NIM sockets / pads to the right.

Check for 3.3v on NIM pads 3 and 4. These are either side of the gap in the southern socket

Check for 5.0v on NIM pad 5. This is the second pin north of the gap.

If any supplies are missing, check joints on L1, L2, L3, L4.

Step 6:

Remove the external psu.

At this point, the PC drivers for the USB module should have been installed. If not, see appendix 1.

The FT2232H USB module needs to be configured for MiniTioune. Modules from the BATC shop are supplied already configured. If the module has not been configured, see appendix 2.

If a socket for the 74HC10 U1 has been fitted, fit the chip now. The pins need to be perpendicular to the body. Hold the chip horizontally using both hands and press the row of pins against a hard surface starting near the chip body..

Step 7:

Remove the external psu.

Fit the USB module with the socket to the south and connect a USB cable to the PC.

Connect the external psu.

Start the MiniTioune software and message should appear saying "NIM not detected". This is correct and shows that the software can communicate with the USB module.

If a message appears saying "No USB device", there is a problem. Check all joints on CN1 and CN2.

Exit from the MiniTioune software.

Step 8:

Remove the external psu.

If the NIM sockets have been fitted, go to step 9.

Slide the PCB between the rows of pins on the NIM and support the PCB to keep it level.

On the northern (double row) set of pins, solder the northern pin on the upper surface.

Solder an end pin on the southern (single row) set of pins making sure that it is pressed against the PCB.

Check that the module is square on to the pcb and level and then solder all the pins.

Fit C2. It needs to be raised above the pcb slightly, to avoid touching the NIM.

Go to step 10.

Step 9:

Remove the external psu.

If the NIM sockets have been fitted, plug in the NIM now. If the 50 way socket has been fitted, the most northerly position opposite the NIM1 legend should remain unused.

The tabs on the NIM should be centred in the slots on the pcb.

Step 10:

Remove the external psu.

There are no ground connections on the NIM pins, so the ground tabs on the NIM must be soldered to the pcb. This applies if the NIM is soldered or plugged in.

Place a resistor leg through the holes in pads P1 and P2, bend the wires towards the top and bottom ground tabs on the NIM and solder the wires on the top and bottom of P1 and P2. Cut the wires to length leaving a little extra that will not be soldered to make it easy to detach the legs in the future.

Support the pcb to keep it level and solder the wires to the ground tabs. Keep the lower wires above the tabs so that the NIM can still be placed on a flat surface.

Step 11:

For brief operational details of the MiniTioune software, see appendix 3.

Connect the external psu, start the MiniTioune software and software should be showing activity with various readings changing.

If the software says "NIM not detected", check that the correct NIM is set in the **minitioune.ini** file in the same folder as the software. Check all joints.

Connect an antenna to the left hand F socket and start receiving. It is advisable to remove the external psu when connecting and disconnecting antennas.

If you do not have a local transmission to receive, it is possible to test the MiniTiouner using a satellite dish. See appendix 4.

How Hot Does It Get?

With the Low SR box ticked and with the Eardatek NIM, the regulators and pcb get slightly hotter than the heatsink on the Eardatek NIM. The switcher should be less hot, but it does pick up some heat from the pcb through its legs.

When finger testing the 1v regulator U2 (5 pin) or the switcher PS1, be careful not to touch any metal parts or pins, above or below the pcb, apart from ground tabs. The voltages are set by resistors ~100k, so skin conductivity could cause a change in the output voltage.

Mounting in a Box

Make sure that there is adequate ventilation.

The pcb is slightly proud of the NIM module, so use washers or another spacer inside the box, so that the PCB is not strained if nuts are used to attach the F sockets the box.

Powering Preamps and Converters

An external 1A quick blow fuse is recommended.

It is likely that a preamp or converter will be used with the MiniTiouner. It is possible to send a voltage out of the F sockets on the NIM, in the same way that a satellite receiver powers an LNB.

There are two ways to do this.

Method 1

By putting a jumper on pins 1 and 2 (western end) of J4, the MiniTiouner supply voltage will come out of the left hand F socket. This passes through D2, FS1 and D1, so the voltage will be less than the external psu voltage. It can be measured on the western leg of D1.

Short-circuiting the F socket could cause a large current to flow, depending on the external psu. The resettable fuse FS1 limits the current. It doesn't give complete protection, as it can take a second or two for the fuse to react and increase its resistance.

The fuse FS1 has a standoff current rating, which is the maximum before its resistance starts to increase. It also has a trip current rating, which is the limiting current. The second is about twice the first, so they are not a precise device. They also take some time to recover after tripping. It is best to use one with the lowest rating possible, so some experimentation may be necessary. FS1 has a standoff rating of 200mA. (This is my understanding of how they work - please advise if it is incorrect).

Some devices that were measured were a DG0VE converter at 170mA (the datasheet says 210mA), a SUP-2400 at 70mA and an LNB at 135mA.

A maximum standoff current rating of 250mA for FS1 is recommended. D1 and D2 are rated at 1A, so the current taken by the MiniTiouner itself (300-500mA) plus the current going to the F socket needs to be less than this.

Method 2

Remove any jumper on J4, and put a jumper on J1.

This connects the two F sockets together.

A voltage introduced into the right hand F socket will appear on the left hand F socket. This can be used for satellite operation by connecting a satellite receiver to the right hand F socket and an LNB to the left hand socket. The signal is available to both the NIM and the satellite receiver.

A preamp or converter could be powered this way by feeding a voltage into the right hand socket.

Caution must be taken to avoid short circuits as there is no protection and the NIM or the MiniTiouner could be damaged.

Appendix 1: Installing FT2232H USB Module Drivers

It is hoped to bring all the relevant information into these instructions in time, but in the meantime go to:

http://www.batc.org.uk/forum/viewtopic.php?f=15&t=4102&start=40#p9134

and scroll down for driver installation details by G8GTZ.

Appendix 2: Configuring the FT2232H USB Module

The ports on the module have to be configured to work in a certain way. This will have already been done on modules from the BATC shop.

http://vivadatv.org/viewtopic.php?f=80&t=363&sid=cf2b9e36d65c3416e5a94dd20542d379#p1061

Appendix 3: Basic MiniTioune Software Operation

In the **minitioune.ini** file:

make sure that the correct **NIMmodel** is selected.

set all the **Fec** options to 1

set **AddrUDP** to 127.0.0.1 and **Port** to 1234

Start the MiniTioune software.

Some possibilities for frequency settings are:

Symbol rate	Frequency	Offset	Use
04167	01319000	00000000	GB3UD repeater output
02000	01337000	00000000	437MHz DG0VE
00333	01963000	00000000	437MHz SUP-2400
00333	02135000	00000000	146.5MHz SUP-2281
22000	10729000	09750000	E4 and others on Astra 2

Symbol rates above ~6000 should have the Low SR box unticked, but try experimenting if the symbol rate is near that figure.

Press the UDP button.

Start VLC. Select Open Network Stream and enter udp://@127.0.0.1:1234

If the NIM has locked on the signal, there should be picture and sound.

If there are several programs in the transport stream, such as on satellite, these can be selected by clicking Playback and Programme.

Appendix 4: Testing with a Satellite Dish

A universal LNB (the most common type) can be set for vertical or horizontal polarisation by setting the voltage to 13v or 18v. It can also be switched to high band by sending a continuous 22kHz tone.

By using method 2 above and setting the satellite receiver to a channel that has the same polarisation and tone setting as the one to be received by the MiniTiouner, both the satellite receiver and the MiniTiouner will receive the signal.

The tests below use method 1, without a satellite receiver.

Astra 2 Test (Sky 28.2E)

- Remove the external psu.
- Remove any jumper on J1.
- Put a jumper on pins 1 and 2 (western end) of J4.
- Connect a satellite dish with a universal LNB (the most common type) to the left hand F socket.
- Connect an external psu to the MiniTiouner of 12-14v nominal. This will set the LNB for vertical polarisation and low band (< 11700MHz).

If the dish is pointing at Astra 2 at 28.2E (Sky):

- Untick the Low SR box
- Set the symbol rate to 22000
- Set the frequency to 10729000
- Set the offset to 09750000

The MiniTiouner software should lock on. Click the UDP output button.

- Start VLC and select the network stream.

There should be audio and video.

Other channels on the transponder can be viewed. On VLC, click Playback and then Programme to see the list of channels. They may appear as numeric until VLC has decoded the program list.

There are no low symbol rate transponders on Astra 2, so check the regulators for hotness during this test.

Close the MiniTiouner software.

Eutelsat 5 (France 5W) Test:

- Tick the Low SR box
- Set the symbol rate to 05968
- Set the frequency to 11609000
- Set the offset to 09750000

The MiniTiouner software should lock on. Click the UDP output button.

- Start VLC and select the network stream.

There are two channels on this transponder. VLC may select the encrypted one and not show anything. If so, click Playback and then Programme and then select **Offre BIS.TV** in the list.

Other Satellite Channels

There is a low symbol rate radio channel on 5W on 11458.2MHz, SR543. It is horizontally polarised, so it can only be received with method 2 above using a satellite receiver. My version of VLC will not decode the audio though.

There is another radio transponder at 12.5W, 12569.3MHz, SR890, horizontal. Again, it will need method 2. My VLC will decode this one.

There are a few vertical transponders at 3.1E in the range of SR1000-4000.

There are some sub SR1000 transponders on 16E.

An excellent resource for finding satellite transmissions is http://www.lyngsat.com/

G4EWJ 2015-08-15
g4ewj@yahoo.com