# G0ETP V2 SDR HF Receiver Assembly Guide

Doc Version 0.3 (updated 14.1.2023)

## Introduction

This document contains notes and advice on constructing the V2 G0ETP SDR receiver. The V2 design is essentially a plug-on shield for an STM32F429I-DISC1 Discovery board and involves very little inter-board wiring compared to the V1.2 design. It still requires minor modification to the Discovery board and soldering SMT parts. None of the SMT soldering is particularly difficult except perhaps the RF attenuator which is in a QFN-20 package; this can only really be soldered by heating the board from underneath (with a hot plate or domestic iron) and for this it is better to use low-temperature solder paste. Liquid or gel flux will also be required.

## Radio PCB Assembly Notes

I suggest two possible methods for assembling the SMT parts onto the radio PCB; one is to hand solder each part with an iron (as I did first time around) and the other is to use solder paste and heat the board on a hot plate. I strongly recommend the use of a MICROSCOPE (USB ones are OK, or at the very least a decent head or desk-mounted magnifier). Even if your eyes are fantastic, your assembly will be worse if you do not use magnification.

Regardless of which soldering method you choose IT IS VERY IMPORTANT to put the bits on the right places first time around; if you start removing bits from the PCB you will be at risk of damaging the PCB tracks. I know this sounds obvious, especially to experienced builders but just take my word for it! I STRONGLY recommend printing out the placement drawing and mark off each component with a highlighter pen as it is fitted. Go through one component value at a time.

After you have put all the SMT parts on, I recommend printing out a fresh copy of the placement and going through with a highlighter pen again as you check the parts: It is better to catch an error now than after you have applied power to the board.

### Hand Soldering

Use a fine iron and fine solder. If your solder does not contain flux then you'll need a syringe of flux as well. I used leaded solder but modern lead free does actually work OK (its all in the flux really).

Tin one pad. Pick the part with tweezers (flat blade tweezers are better than pointed ones as they hold the part more securely). Melt the tinned pad and slide the component on, taking care that the end of the component does not tilt up before the solder has melted. Re-align the component if necessary. Finally solder the other end.

The SOIC ICs are easy to solder: Tack one corner, then the other, then do the remaining pins one at a time. The TSSOP packages are a bit harder: Again, tack 2 opposite pins to ensure the part is well aligned and secure. Soldering the remaining pins can be done a number of ways, one pin at a time, or drag-soldering using a broad iron tip and lots of flux (lay the solder across a bunch of pins, place the iron on and drag away from the chip). I do not recommend 'blobbing' all the pins and wicking-off the excess as this increases the amount of heat the IC is subjected to.

See https://www.youtube.com/watch?v=5uiroWBkdFY for inspiration!

## Solder Paste

Solder paste that melts at 138°C is easily available. The good thing about this is that a domestic iron secured face-up can be used to flow the solder from underneath. This low temperature subjects the components to less stress, particularly if you flow the board more than once during assembly (which I think will be inevitable).

Put a CONSISTENT amount of paste onto each pad. Because the paste DRIES OUT very quickly, you should only put paste on a few pads at a time then fit the parts. Once parts are on the board, BE VERY CAREFUL not to knock them while adding more parts. This is the main reason that I (re)flow the board after doing 10 – 20 parts.

My normal order of assembly would be:

- SMT passives (Rs and Cs) first, as these are probably more robust against multiple thermal cycles
- SMT ICs (but not the RF attenuator)
- RF attenuator
- Leaded components need to be done last so that the board will sit flat on the hot plate when doing the SMT parts. The leaded ones can be done with a conventional soldering iron.

Accurate placement of components is necessary but somewhat less important for the passives. The surface-tension of the solder should cause components to centre themselves up by when the board is flowed. (Gentle tapping of the board can help centre up the parts but don’t over do it.)

## Solder paste Flowing

This can be achieved using a hot plate and / or hot air gun/pencil. For those of us without a thermostatically controlled hot plate, a domestic clothes iron will work fine. This should be SECURELY mounted face-up. If the iron were to slip while your board is hot all the components are likely to fall off, meaning you’ll need access to more parts. Similarly, take great care not to knock or drop the hot board when sliding it off the iron.

Pre-heat the iron to ~150°C (probably near the lowest setting). To flow, slide the board onto the plate. Observe for solder flow and if necessary, nudge one or two parts with a spike (or gently tap the board) so that they align correctly on the pads using solder surface tension. Once flowed, CAREFULLY slide the PCB off onto a small board and let it cool down.

I did try setting my hot plate to just below melting point and using a hot air pencil to do the actual flow. I found that, even with the lowest airflow setting, it was very easy to blow the parts around. I would therefore suggest assembly with a hot plate only and no hot air.

## QFN-20 Attenuator Soldering

I left this part until last, mainly because I did not want to solder it then have it move by accident while soldering other parts. I discussed the technique for soldering these with one of my Technician colleagues and his advice is as follows:

- Tin the PCB pads with low-temp solder paste and an iron. Ensure there is a consistent amount of solder on each pad and do not use too much solder.
- Coat the pad side of the chip with a thin layer of gel or liquid flux
- Drop the chip onto the board and line it up the best you can
- Flow the board on a hot plate

The QFN chip should be seen to 'snap' down onto the PCB when the solder melts.

After the board has cooled you can do a walking pin-pin short test with a DVM. Other than that, just trust that it has gone down correctly until you have evidence later that it is not working.

## Header Pins

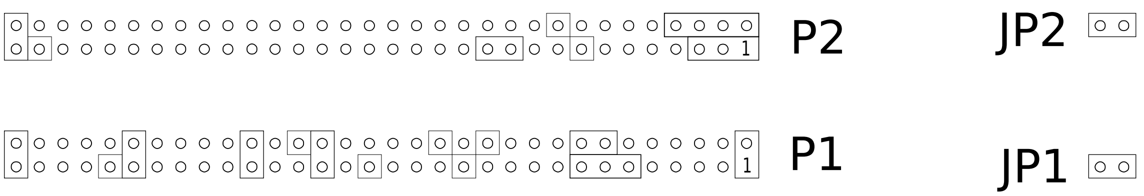
The through-hole components should only be added once the SMT part of the build is complete, as the board is unlikely to sit well on the hot plate with through-hole components poking through.

The female header-pin strips (for JP1 and JP2) are quite difficult to cut as the plastic is glass loaded and quite brittle. I have found snapping or cutting with wire cutters to be unsuccessful. I found a hot knife blade to be the best solution (e.g. a Stanley knife blade heated in a gas flame). Alternatively, sawing with a fine-toothed blade will work.

The connector strip should be cut into the following lengths:

- Single pin x 9
- 2 pin x 10
- 3 pin x 2
- 4 pin x1

Once cut, the pieces can be fitted onto the pins on the UNDERSIDE of the ST board as follows (picture looking at the underside of the ST board):




The assembled radio board can then be placed over these pins and soldered.

Finally, the male header strip for the front panel connections can be easily broken into the required lengths. Note that the header strips provided are Valcon THS-40-R-RA which have the plastic support on the **PCB side** of the pins – this is quite unusual but saves PCB space.

## Discovery Board Preparation

The STM32F429I-DISC1 Discovery board will need a few minor modifications:

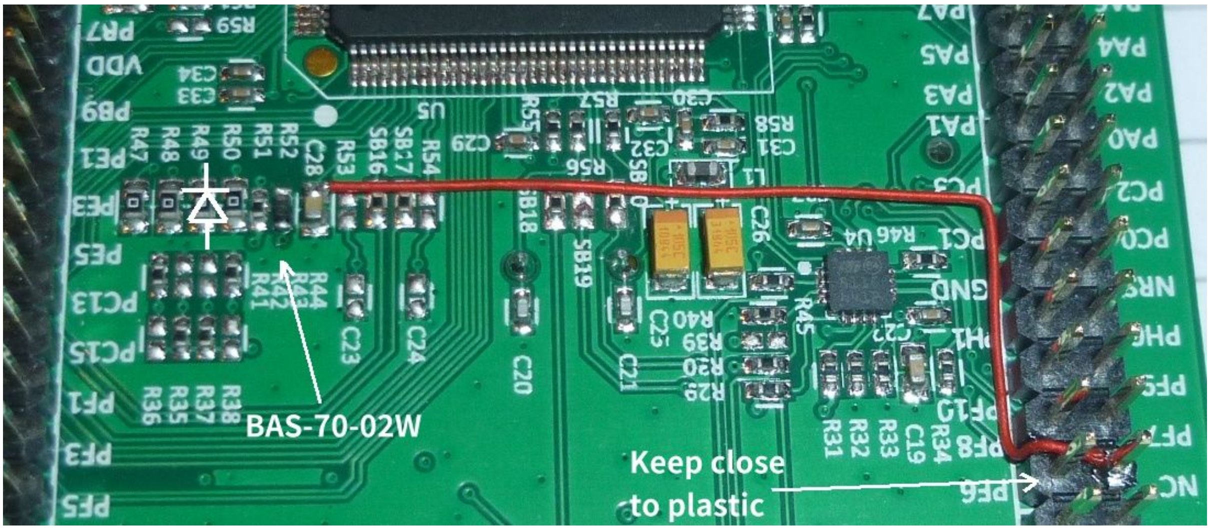
### External Vbat Supply

I have chosen to feed Vbat from the CR2032 battery into the Discovery board using connector P2 pin 4, which is marked as NC on the Discovery PCB.

Resistor R52 (on the underside near the white dot on the ARM chip) will need to be removed and replaced with the BAS-70-02W shottky diode supplied in the kit. The cathode (the end with the stripe) goes on the end of R52 nearest the ARM.

Next, solder a fine fine wire to P2 pin 4. This can be done to the pin itself (above the plastic) but make sure the wire is as close to the plastic as possible and shave off any excess solder with a razor blade so that the female header fits onto the pin correctly.

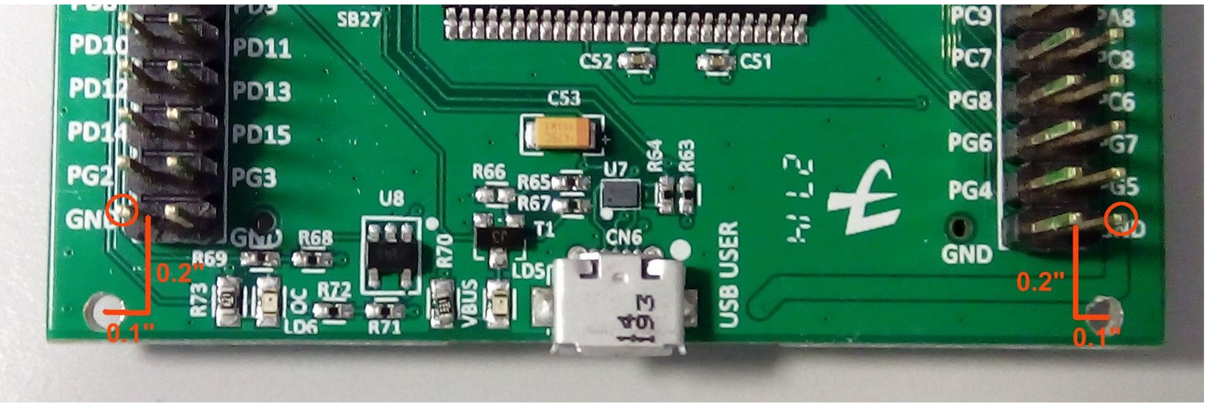
Route this wire to C28 next to the new diode. Solder it to the +ve end of C28 (end nearest the ARM):




## Discovery Mounting Holes

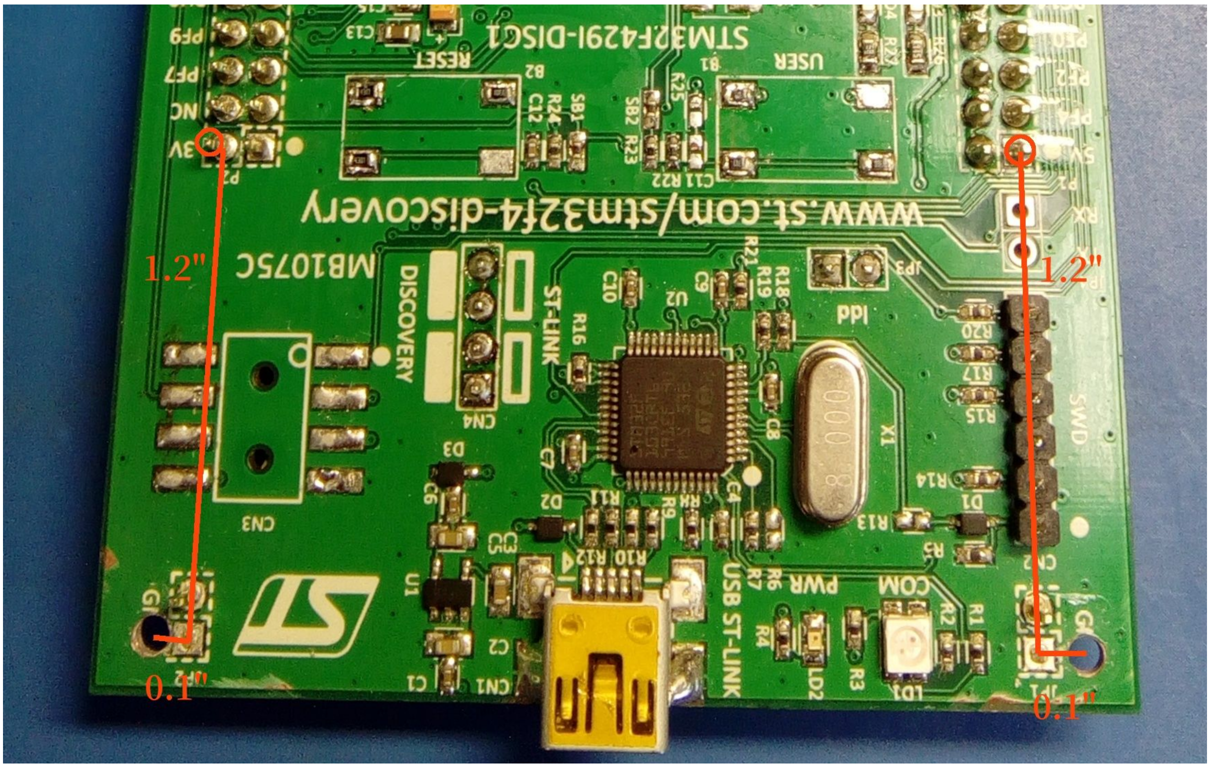
The Discovery board unfortunately has no mounting holes. I have found that the most secure way to mount the board is to drill 4 M2 holes directly through the PCB. I have checked the Gerber files for the rev B and rev C PCBs and it is safe to drill as follows:

Place a piece of 0.1” perforated prototype/Vero board over the discovery pins of P1 and P2 to act as a 0.1” grid drilling guide. Make 2 x 1mm pilot-holes at the display end of the board, 0.2" (2 holes) towards the board **end** from the corner pins and 0.1" (1 hole) towards the board **edge**:




These holes can then be opened up to 2mm.

At the STLINK end of the board, 2 more holes can be drilled next to JP1 and JP2. Again, using a piece of 0.1” perforated board located over both P2 and JP2, drill a 1mm pilot hole 0.1” towards the board edge from JP2 pin 1. Then do the same for JP1. These holes can then be opened up to 2mm.




These holes should not cause the mounting screws to come into contact with any signals besides GND. (To be extra safe you could use M2 nylon screws but I did not.)

If you are going to mount the Discovery board close behind the front panel (see building section at the end and the portable build page on the website) then you will probably need to remove the blue USER and black RESET buttons, connector CN3, as well as cropping the tall pins on the display side of the board. (The image above shows some of this clearing.)

# Radio Construction

## 12V Power Input

A 2.1mm coaxial power socket is mounted on the rear panel of the radio. The +12V can feed through a forward-biassed 1N4001 diode (for reverse polarity protection) and then through the on/off switch (which may be part of the volume control). You could use a P-channel FET instead of the 1N4001 diode if you want a lower forward volt drop.

**Pin 1** of the radio board power connector is GND. Raw +12V power goes to the audio amplifier via **pin 2** of this connector.

## Radio Board Supply (~8V)

The radio board has its own 5V and 3V3 regulators. These may be fed with 12V directly but the board will get slightly warm. To lower the dissipation in the radio

PCB's 5V regulator you can feed power **pin 3** with less than +12V. A 39R 3W dropper resistor can achieve this, alternatively you can use an 8V regulator (7808) mounted on the chassis. (My portable build regulates +12V → 8V and then the 8V → 5V with chassis-mounted 780x regulators.)

## STM32 5V Regulator

A TO-220 7805 voltage regulator should be mounted on the chassis to provide the main +5V rail to the Discovery board (about 200mA) via **pin 4** of the power connector.

Note all regulators should have input and output capacitors, such as a 1uF film capacitor on its input and a 0.1uF film capacitor on its output.

## Main Tuning Shaft Encoder

I have successfully used both a Bourns ENS1J-B28 L00256L optical shaft encoder and a Bourns EMS22Q51-B28-LS4 magnetic encoder. Both are 256 pulses/rev and work equally well. The magnetic encoder is now recommended as it is half the price of the optical one. Any incremental encoder with an output level between 3V and 5V should work here but try to keep the steps per revolution between 128 and 320.

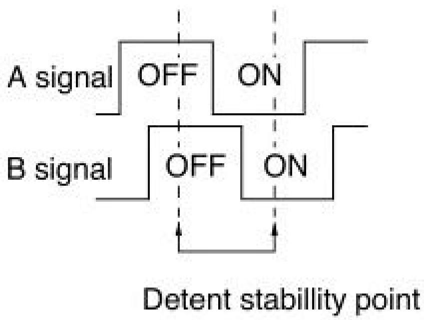
**Pin 1** of the tuning connector is GND. **Pin 2** is encoder A, **pin 3** is encoder B and **pin 4** is either +5V power (R68 fitted) or +3V3 power (R69 fitted). The PCB has 1k series resistors (R31, R58) to avoid over driving the Discovery input pins when a 5V encoder is used.

## Main Tuning Knob - Friction Lock

The shaft encoder is likely to spin too freely without providing additional friction. I achieved this by folding a tissue to ~3mm thick 'pad' and then cutting it down to a size that will fit behind the main tuning knob (4 x 2cm in my case). I then used an A4 paper hole punch to make a 1/4" hole in the centre of the tissue pad. This pad was then threaded over the shaft of the tuning encoder and trapped between the tuning knob and the front panel. The knob will need to be positioned on the shaft to achieve the desired amount of friction. (I prefer a light friction so the knob can still be 'spun' but does not drift round on its own.)

## Menu Shaft Encoder

Almost any cheap mechanical encoder with between 15 and 30 steps/rev with a momentary push button can be used, provided the encoder has 2x as many mechanical indents ('detents') as it has output pulses. For example, the Bourns PEC11R-42xxF-S0012 series which has 24 mechanical steps and 12 pulses per rotation:



(Update: The software can now accommodate use of an encoder that outputs the same number of pulses as it has mechanical detents by selecting the ‘menu step div 2’ option.)

The encoder GND connection goes to the menu connector **pin 1**. The quadrature output signals feed into **pin 2** and **pin 4**. The menu push button connects to **pin 3**.

Pull-up resistors and filtering capacitors for the menu encoder are included on the PCB. Note that on most mechanical encoders the push switch is completely floating, so this will need a ground connection as well (bridge a wire across to the GND connection on the rotary encoder).

## UI Control Buttons

These are all passive short-to-ground buttons. **Pin 1** of the buttons connector is GND. The VFO button (red) connects to **pin 2**, the VFO STEP button (black) to **pin 3**, the MODE button (green) to **pin 4** and the DISPLAY button (blue) to **pin 6**.

Pin 5 is currently a spare IO pin. Pull-ups are provided by the STM32 and filtering for all buttons is located on the PCB.

## Audio Output

### Speaker

I used a surplus but decent 3W 4” 4Ω speaker. I attached it behind the speaker grille using a hot-melt glue. (Note that when using hot melt glue to attach to **cold metal**, it helps to heat the metal up so that the glue bonds properly.)

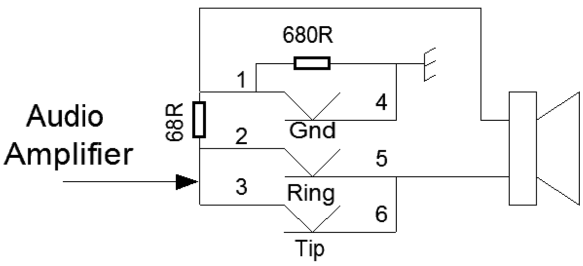
### Headphone Jack

The headphone jack is wired so that when headphones are plugged in, the audio level delivered to the headphones is potted down. Due to the switching arrangement I used to do this, is is necessary to use a 6-pin socket with 3 switches:


Pins 2-3 and 5-6 are joined so that it provides a mono signal to stereo headphones.

The amplifier GND goes to pin 4 (switched). The amplifier output goes to pins 2 and 3. The speaker ground comes from pin 1. The speaker signal comes from pins 5 and 6 (switched).

To pot-down the audio when headphones are inserted, resistors are added as follows: 68 ohms from pins 2-3 to 1; this is a shunt across the phones (and the speaker) that helps to reduce variation caused by differences in headphone impedance. A 680 ohm resistor goes between pins 1 and 4 and this provides a 'weak' ground to the headphones when they are plugged in. With no jack plug, the speaker has both a signal and a ‘strong’ ground connection. When headphones are connected they get a ‘weak’ ground through the resistor and the audio is disconnected from the speaker.



## Line Output Socket

It is useful to have a constant level output on the back for connecting to a PC (e.g. for recording or use with FLDIGI). I added a 3.5mm stereo jack on the rear panel. This requires a separate feed directly from the DAC on pin PA5 as follows:

```
PA5(DAC)-----|10k|---------------------------|1uF|--|----|1k|----L
                                                    |----|1k|----R
                                                    |----|1k|----GND
                                                    |----|33n|--GND
```

Unfortunately I forgot to add this to the V2.0 PCB layout so it will need to be built onto the jack socket.

## STM32F429I Discovery Programming

Programming options include OpenOCD or one of the STM32 ST-LINK utilities from ST, which are perhaps easier for Windows users. (See SDR website downloads area for details.)

### STM32CubeProg

This is now the preferred utility provided by ST and is supplied for Windows and Linux. Installation is straightforward. It is possible that the utility will want to update the ST-LINK firmware onboard he Discovery board the first time it is connected – this is expected.

Once the utility is connected to the Discovery board, the SDR firmware (.hex file) can easily be programmed into the board.

### STM32 ST-LINK utility

Look for ST product STSW-LINK004. This is the predecessor to the STM32CubeProg utility but installation and operation are similar.

### OpenOCD

I suggest installing OpenOCD V0.10.0 (or newer), as this is a currently supported tool for both Linux and Windows platforms. You will need a board config file for whichever revision of the Discovery board you have. The ‘DISCO’ board (PCB MB1075B) is now obsolete, so it is likely that you will have a ‘DISC1’ board (PCB MB1075C or newer).

- STM32F429I-DISCO (rev B PCB) Use stm32f429discovery.cfg
- STM32F429I-DISC1 (rev C or newer) Use stm32f429disc1.cfg

The following commands will flash the board

New revision of the Discovery board (MB1075C or newer):

openocd -f board/stm32f429disc1.cfg -c "program mxDiscoveryRadio.hex exit"

Old Discovery board (MB1075B):
openocd -f board/stm32f429discovery.cfg -c "program mxDiscoveryRadio.hex exit"

## Initial Power On

If you have not FULLY checked the radio PCB assembly go do this first.

When the Discovery board powers up, there is a 2-second delay (white screen) while the ST-LINK waits to see if it is connected to a USB host (PC). After this delay, the SDR software starts.

This software will detect when the contents of the battery backed up RAM is either un-initialised (first-time) or corrupt and will initialise it with default values. If for any reason you need to force the CMOS RAM settings back to default values, hold the VFO button down whilst powering on.

## Debugging Guide

If you have built everything correctly the 'radio will just work'. Unfortunately I have received a few emails where people have had problems, so here is a list of things to check first.

### V2.0 Radio PCB Voltages and Signals (rev 2.0 schematic)

(Voltages in brackets are as measured on my board.)

- The radio board draws about 92.5mA when everything is running. (If you are using a dropper resistor to feed the board’s 8V pin then you can measure this current as a volt drop.)
- The STM32 board draws ~200mA (at 5V)
- The audio power amplifier draws ~38mA at +12V (with no audio)
- Check the 5V regulator output (U11 pin 3) and 3.3V regulator ( U1 pin 3)
- Drop across R8 should be ~258mV
- Drop across R22 should be ~55mV and output of U5 ~3.53V
- Drop across R4 should be ~191mV and there should be 26MHz at ~0.8 V p-p on C6
- Drop across R9 should be ~197mV and an L.O. of ~3V p-p should be present at U3 pins 10 and 9 (9 delayed by 90 degrees)
- Drop across R21 should be ~23.7mV.

- Bias at T1 pin 5 (~2.5V) should propagate to C19, C26, C30 and C37.
- Drop across R29 and R41 should be ~6.2mV for 10R decouplers (~62mV if using 100R decouplers)
- Drop across R59 should be ~10mV and pin 4 of U9 should be ~2.5V
- U7 and U8 outputs should be ~2.5V
- Drop across R51 should be ~88mV
- Drop across R55 should be ~31mV
- U14 pin 4 should be ~6V (½ the 12V supply rail)
- Once this is all working, tune into a strong test signal and you should see sine waves THE SAME SIZE but 90 DEGREES OUT OF PHASE at C44 & C52 going into the ADC.

## TCXO Frequency Calibration

The SDR's reference TCXO can be calibrated using any accurate off-air reference signal, such as RWM on 9996kHz[1]. Be aware that not all shortwave AM broadcast stations have an accurate carrier frequency. RWM is a reliable source.

The calibration procedure is as follows:

- Select USB mode and 500Hz tuning step size
- Set IF shift to -700Hz (so we can hear 500Hz either side of 0)
- Set 'Freq quantise' to off in the menu
- Tune to 9995.5kHz, then set the tuning step size to 1kHz
- Flip between 9995.5 and 9996.5 and alter the 'Ref freq adjust' in the menu until you hear the same 500Hz tone either side of 9996kHz.

Note that the Fox-914B TCXO is a low cost part and appears to make adjustments to its output frequency in 3Hz steps, which is outrageous. If this is causing you a problem then try a better TCXO. The Abracon ASVTX-09-26.000MHZ-T VCTCXO supplied in the previous kit is much better but I was unable to find stock *anywhere* for the V2. Alternatively, a 26MHz 5 x 3.2mm XTAL may be used but some schematic changes will be needed, including placing a 0R link on R7.

1 Also available on 4996kHz and 14996kHz.

# Discovery Board Mounting Ideas

## Display Orientation

When looking at the display in landscape mode with the text the right way up, the 'bulk' of the Discovery PCB will be on the RIGHT hand side.

## Front Panel Display Aperture and Window

My recommendations for mounting the Discovery board behind a front panel are as follows:

- Drill holes in the front panel for the 4 M2 mounting holes that you have drilled in the Discovery board. Mount 4 6mm brass pillars behind the front panel using metal, countersunk screws.
- Cut a *rough* aperture in the front panel where the display will be (& not too big)
- Briefly mount the Discovery board on the pillars and make adjustments to the display aperture until it is the correct size (51mm x 38mm) and is well aligned.
- Make a plastic window that will sit in this aperture. I use 4mm perspex/plexiglass and cut a lip on all 4 sides so that the plastic sits in the aperture in the front panel. This lip can be cut by hand or with a router.

# HA8LFK Filter Interfacing

The use of a pre-selector filter is optional. Performance of the receiver without a pre-selector will depend on exactly what strong signals you have at your location. I live in an urban location reasonably far from any transmitter sites and with an HF wire antenna (nominally matched at 14MHz) connected to the receiver I have very few issues with spurious signals with **no** preselector filter.

The V2 SDR design includes an 8-bit latch that can directly drive the PIN diode select inputs of the HA8LFK filter. These outputs come out on the PRESEL header. Pin 1 is GND, pin 2 is unused on this filter, pin 3 selects 1.6-2.5MHz, … pin 9 selects 33-56MHz and pin 10 (+5V) is currently unused. For pictures if this wiring see the V2 SDR web page.

[EOF]