G0ETP V2 SDR HF Receiver Assembly Guide

Doc Version 0.2 (updated 1.7.2021)

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 the RF attenuator which is in a QFN-20 package; this can 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. Flux will also be required.

Radio PCB Assembly Notes

I suggest two main methods for assembling the SMT parts onto the radio PCB; one is to hand solder each part (as I did first) and the other is to use solder paste and heat the board. 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.

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 you older builders who have previous experience 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. (You really should REPEAT this whole process with a fresh printout after assembly is complete before you first apply 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 as they hold the part better). 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 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 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 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.

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 when sliding the board off the iron when it is hot.

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 just a hot plate.

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:

- Do NOT use too much solder paste: Put a TINY amount onto each pad and try not to rely on the solder resist too much. (Alternatively, tin the PCB pads with an equal amount of solder.)
- Coat the pad side of the chip with a thin layer of flux
- Drop the chip as accurately as you can onto the board
- Flow the board on a hot plate

The 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 the part until such time that you think it is not working.

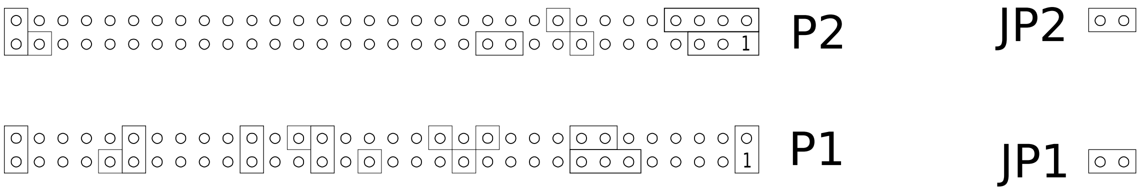
Header Pins

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 pins (for JP1 and JP2) are much more difficult to cut as the plastic is glass loaded and is quite brittle. Snapping or cutting with wire cutters is generally unsuccessful. Sawing with a fine, toothed blade works, as does a Stanley knife blade heated in a gas flame. The connector strip can be cut into the following lengths:

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

Once cut they can be placed on the underside of the ST board as follows (as seen looking at the underside of the ST board):



Place the assembled radio board over these pins and solder.

Finally, the male header strip can be easily broken into the required lengths for the various peripheral interconnects. Note that the header strip 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:

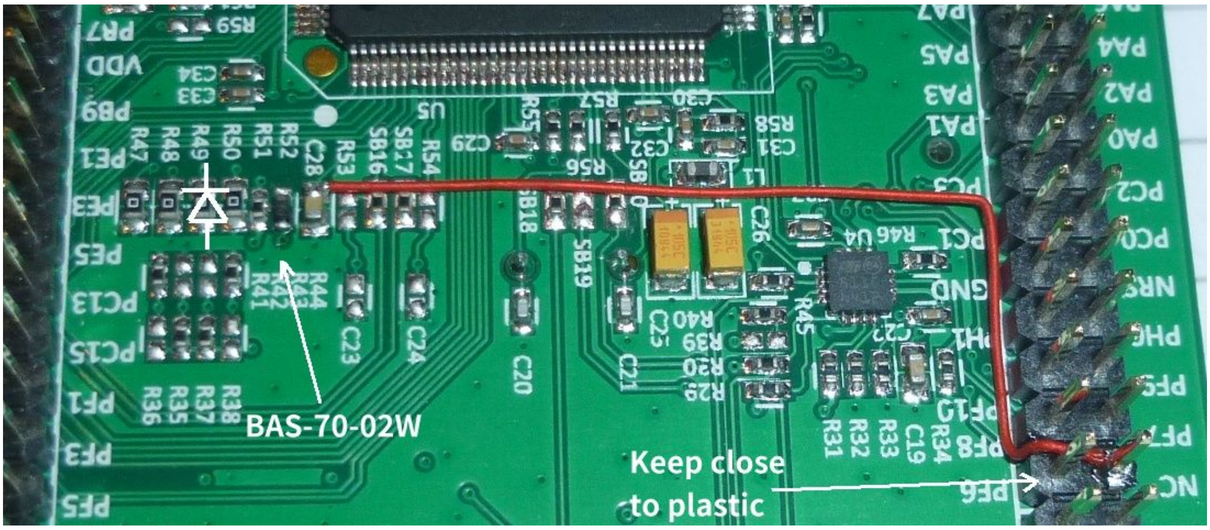
External Vbat Supply

I have chosen to send Vbat from an external battery in on 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 diode supplied in the kit. The cathode (the end with the stripe) goes on the end of R52 nearest the ARM.

Solder a fine fine wire to P2 pin 4. This can be done to the pin above the plastic but make sure the wire is as close to the plastic as possible. Shave off any excess solder with a razor blade so that the female header can fit all the way down the pin.

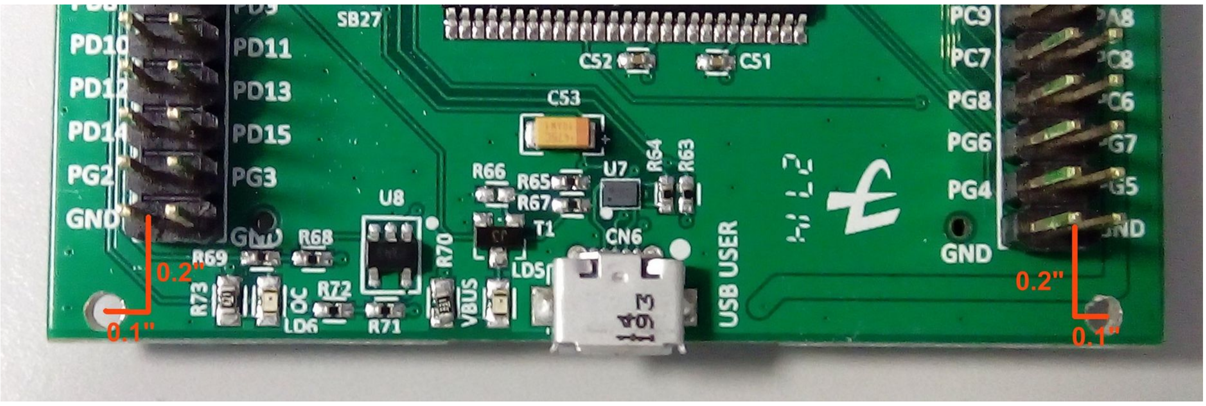
Route this wire to C28 next to the new diode. Solder it to the +ve end of C28 (end nearest the ARM):



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:

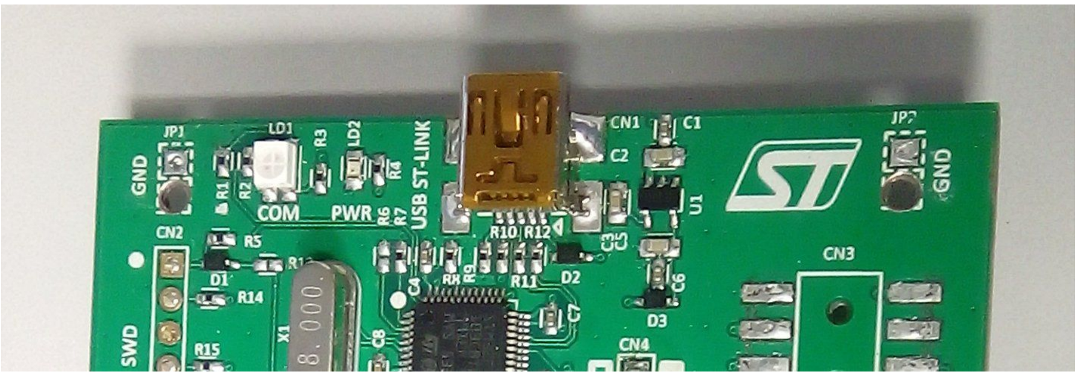
Place a piece of Vero board over the discovery pins of P1 and P2 to act as a 0.1” 'guidance grid' for drilling. Make 2 x 1mm pilot-holes at the display end of the board, 0.2" (2 Vero holes) towards the board **end** from the corner pins and 0.1" towards the board **edge**:



These holes can then be opened up to 2mm.

At the STLINK end of the board, 2 more holes can be drilled through the pin of JP1 and JP2 farthest from the edge of the board. It is best to suck all the solder from the holes before drilling:

Since the pins of JP1 and JP2 are now used to connect to the V2 SDR PCB, I have moved these 2 mounting holes to be 0.1” out from the end pin of JP1 and JP2. Need to add a new picture.



These holes should not cause the mounting screws to come into contact with any signals besides GND. To be extra safe, use M2 nylon screws.

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

Discovery AVDD Decoupling - TBD

Some audio noise from the ARM 3V VDD is getting onto the DAC output. This can be heard when the AGC gain is wound all the way back to 0dB and the audio volume increased. This is the same in the V1.2 design and I have lived with it for years, as the HF band noise is generally way above this noise. If this is causing you a problem then the AVDD feed to the ARM chip will require additional decoupling.

I have not defined a method for this yet. VDDA is filtered with L1 (ferrite bead) C26 and C25, ADC/DAC VREF+ is filtered with R58 (0R) C25 and C30. This filtering may be OK at HF but is inadequate for audio frequencies. So far, replacing both L1 and R58 with 100R has not degraded DAC performance, nor has it solved the audio noise. Much larger decoupling capacitors should next be tried.

Radio Construction

12V Power Input

A 2.1mm coaxial power socket is mounted on the rear panel of the radio. The +12V feeds through a forward-biassed 1N4001 diode and then through the on/off switch, which in my case is also on the rear panel but could be built into the volume POT if requires. The diode is to protect against reverse-polarity power; you can use a P-channel FET if you are keen to avoid the voltage drop of the diode.

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 warm. To lower the dissipation in the PCB 5V regulator you can feed radio board power **pin 3** from +12V via a 27R 3W dropper resistor. Alternatively you can feed it from an 8V regulator (7808) mounted on the rear panel.

STM32 5V Regulator

A TO-220 7805 voltage regulator should be mounted on the rear to provide the main power to the Discovery board. This will need a 1uF film capacitor on its input and a 0.1uF film capacitor on its output.

This regulated 5V feeds the Discovery board via **pin 4** of the power connector.

Main Tuning Shaft Encoder

I have 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 will 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 adding a little friction. I achieved this by folding a tissue to ~3mm thick and then cut it down to a size that would fit behind the main tuning knob (4 x 2cm in my case). I used an A4 paper hole punch to make a 1/4” hole in the centre of the tissue rectangle. This soft pad was then threaded over the tuning shaft and trapped between the tuning knob and the front panel. The knob can be positioned on the shaft to adjust the level of friction. (I prefer a light friction so the knob can still be ‘spun’ but does not drift 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 steps. (Note that use of the wrong encoder type can be corrected for in the software 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.

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 the audio level delivered to the headphones is potted down. Due to the switching arrangement, 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 required 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.

STM32F429I Discovery Programming

Programming options include OpenOCD or the STM32 ST-LINK utility from ST.

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"
```

STM32 ST-LINK utility

This can be downloaded from the ST website. Look for STSW-LINK004. Instructions are on the web also. (Note that this will also require the Windows USB driver for STLINK to be installed.)

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

Debugging Guide

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

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 (2.5V measured across the 27R dropper feeding the board).
- 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 TCXO can be calibrated using any accurate off-air reference signal, such as RWM on 9996kHz[1]. Be aware that many AM broadcast stations have an accurate carrier frequency but *by no means all* do. 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
- 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 get 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[2]. The Abracon ASVTX-09-26.000MHZ-T VCTCXO supplied in the previous kit is much better but I was unable to find stock *anywhere*. 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.

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 10mm 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. (If you are going to raise the LCD display with a thicker foam pad

1 Also available on 4996kHz and 14996kHz.

2 FT8 mode on 14074kHz works fine for me.

as described below, you might like to do this first in case the display alignment changes slightly.)

- 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.
- This window can be held in place by the LCD display when the Discovery board is mounted, or glued in place if you do not plan to raise the LCD display on the board. To get the LCD display closer to the display window (and to make enough room to squeeze push-buttons between the Discovery and the front panel) you will have to raise the LCD display on the Discovery board:
- To do this, CAREFULLY separate the display from the board by prizing it off its foam pads with a plastic tool (something that wont scratch). You'll need to be careful of the flexi cable that feeds the LCD, however there is nothing DIRECTLY behind the display except the foam. (The flexi cable solder line is ~2 1/4 inches in from the edge of the board.) Also, avoid the temptation to prize the display off the foam pads by levering – the display is thin and can be damaged from behind.
- Use new foam pads and some plastic to make a spacer so that the display will sit ~8mm clear of the PCB. (Get this right **before** you take the backing paper off the new foam pads!)

HA8LFK Filter Interfacing

Needs new text.

[EOF]