Iceni 432MHz Transverter Technical Description

Introduction

Iceni is a 432MHz to 28MHz low power transverter module featuring a low noise local oscillator for excellent phase noise performance, overdriven double balanced ring mixer and high dynamic range HEMTS MMICs for low noise and excellent performance under strong out-of-band signal conditions.

The project was designed to use solid analogue design circuits to provide a frills-free, easy to use and low cost 70cm transverter module that could be used with an existing HF transceiver to produce a high quality signal on 432MHz as well as having excellent strong receive signal handling performance.

Surface mount components are used almost throughout. The PCB features a large number of plated through holes so that home production of the board is not recommended. Boards and short kits are available from the author.

Parameter	Measured	Comments
Rx noise figure	2.0dB	Usual uncertainties apply
Rx gain	21dB +/-1dB	Fixed gain
Rx maximum input level	0dBm	Max non-damage level
Rx IIP3	-10dBm +/-1dB	Measured at -37dBm/tone
RX Bandwidth	8.5MHz	3dB bandwidth
Tx gain	16.5dB +/-0.5dB	Tx attenuator at minimum
Tx IF Input level	0dBm	Max +6dBm at attenuator minimum
Tx output power	+20dBm	At +6dBm IF input. 0dBm max IF input recommended
Tx IMD	-50dBc/-38dBc	+7dBm/+10dBm per tone at output
Tx spurious	<-60dBc	Except second harmonic -53dBc at +16.5dBm output
Receive current	260mA at 12v	Unchanged from 9V to 14V
Transmit current	280mA at 12v	Unchanged from 9V to 14V

Table 1 measurements on unit serial number #4006

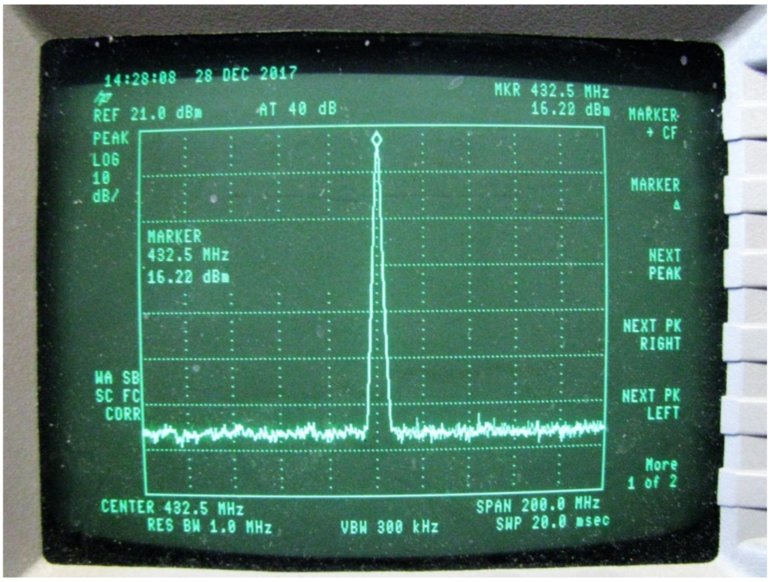


Photo 1 Iceni output spectrum at +16.5dBm output showing the clean spectrum, with LO and image over -60dBc

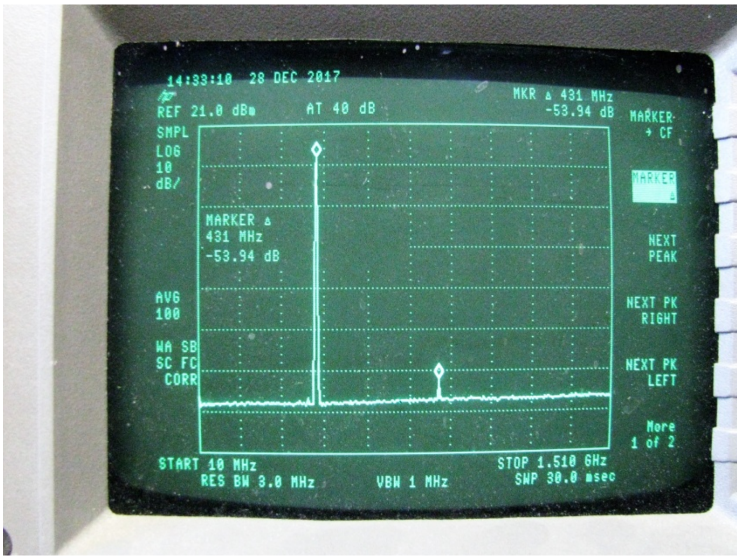


Photo 2 Iceni spectrum at +16.5dBm output and from 10MHz to 1510MHz. The only harmonic is the second at -53dBc

Circuit description

The Iceni uses similar circuitry to the 144MHz Anglian 3L transverter previously described. The original low noise crystal controlled local oscillator has been modified to produce a high level 404MHz output that is used to drive a 1.3GHz rated mixer. This mixer was chosen for low cost and good LO to RF isolation. Poor LO to RF isolation usually leads to higher than wanted LO appearing at the transverter transmit converter output. The Iceni meets CEPT recommendations in terms of unwanted spurious output signals. Six poles of helical filtering help to ensure the excellent spectral purity of the output signal.

On receive a low noise, high dynamic range MMIC input amplifier is preceded by a low loss 432MHz noise matching band pass filter. The MMIC is followed by a diode PIN switch, three pole helical filter and then the single double balanced mixer, also used on transmit. Following the mixer a high pass, low pass diplexer feeds a very high dynamic, low noise MMIC post mixer amplifier to provide a 14dB lift to the IF signal. This is followed by a two stage IF (29MHz CF) bandpass filter to ensure that the following HF transceiver is not subject to LO or other out of band mixer products.

On transmit a separate IF input is provided with level adjustment before a bandpass filter centred on 29MHz ensures that unwanted out of band IF signals are much reduced by this and the following diplexer before feeding the mixer. The optimum level of IF drive is -3dBm, although IF inputs up to about +20dBm (100mW) can be used if the transmit gain attenuator is used to reduce them to no more than 0dBm at the mixer input.

Two stages of transmit amplification are separated by a three pole helical filter. The first amplifier stage uses the same low noise, high dynamic range MMIC as the LO amplifier and receive converter RF stage. This provides 22dB of gain before the helical filter, which has approximately 4dB loss. Following the second filter a high dynamic range MMIC provides another 14dB of gain before the output low pass filter. The output power is 100mW (+20dBm). Maximum recommended output is +17dBm. Most small add-on 1-2w amplifier modules require no more than 10-20mW drive for full output. A range of these are available at low cost on EBay (why compete?)

It is possible to lock the Iceni local oscillator, for higher stability, by injecting a low level 101MHz signal to the LO input/output port.

+5V on transmit and a ground on transmit (EOT) output are provided for interfacing to any add-on amplifier and transmit is enabled with a press to talk (PTT) ground input.

The recommended power supply voltage is 10V, although up to 13.5V may be used. However the excess voltage will be lost as heat dissipation in the 5V regulator. This is not recommended, but if only a 13.5V supply must be used, then it is recommended that a 1.5W rated 10Ω resistor is connected in series with the input and located external to the Iceni.

The Iceni should be mounted into the recommended tin plate box. As an alternative, 4x 3mm holes are provided in the PCB to enable the board to be mounted direct to a box chassis, using suitable length threaded pillars and screws.

Kits (not built units) are available from G4DDK. See www.g4ddk.com for prices and full build information.

73 de Sam, G4DDK

DDKits

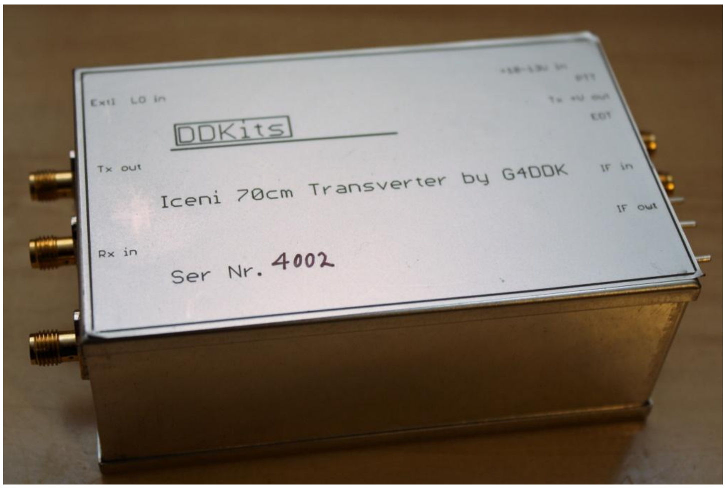
Iceni 432MHz Transverter Assembly Manual

For board version 1.0

The accompanying Technical Description introduces the design of the Iceni 432MHz transverter. This document details the recommended method of construction. You don't have to do it this way, but it is tried and proven and a kit of parts are available to save you from having to buy more SMD parts than you really need.

If you buy a kit and lose or damage components I can supply replacements. It is all-too-easy to 'ping' a small SMD part onto the shack carpet and lose it forever. However, I am not able to supply ad-hoc parts to those who choose to buy just the PCB.

This kit is not recommended for inexperienced SMD constructors. Many very small parts are used in its construction.



Let the building begin

The kit builder should ensure that they have a static-free place in which to work. Antistatic matting and wrist straps should always be used when working with small parts (and large ones too!).

A suitable insulated or grounded, temperature controlled, soldering iron should be used for the SMD work. A larger (>50W) iron will be needed to seam solder the PCB into the tinplate box. Do not underestimate the amount of heat needed to make a good seam soldered joint. Be careful as the box can get extremely warm when doing the seam soldering.

SMD board assembly

Small, stainless steel, tweezers are a must.

The order of work in building the kit is:

1. Inspect the PCB. Currently V1.0
2. Mark the position for the connectors and feed-through capacitor holes onto the box, using the PCB to locate the hole positions.
3. Drill and clear the holes
4. If the Iceni board is to be mounted into other than the recommended tinplate box, use the bare board as a drilling template.
5. Assemble the SMD board. **Do not fit crystal, heat-sink, L13 or the reversed supply protection diode at this stage. These all go on the top of the board once it is seam soldered into the box.**
6. Seam solder the assembled PCB into the box
7. Solder the SMA connector spills to the box and the PCB RF & IF tracks
8. Solder the feed-through capacitors to the box from the outside.
9. Wire the feed-through capacitors to the correct PCB pads using thin wire.
10. Stick the copper heat-sink to the PCB, as indicated later.
11. Solder the crystal onto the top side of PCB
12. Solder the supply diode between the appropriate feed-through capacitor and PCB pad
13. Solder L13 in place
14. Clean up any excess flux around the seam soldering, connectors and feed-through capacitors.
15. Test and alignment

Each of these steps is described in the following section.

Inspect the PCB

Use a magnifying glass to inspect the PCB for damage. If you spot any problems it will be necessary to return the board for replacement. Boards that have been soldered into the box and then reported as damaged will be considered on a case by case basis for obvious reasons!

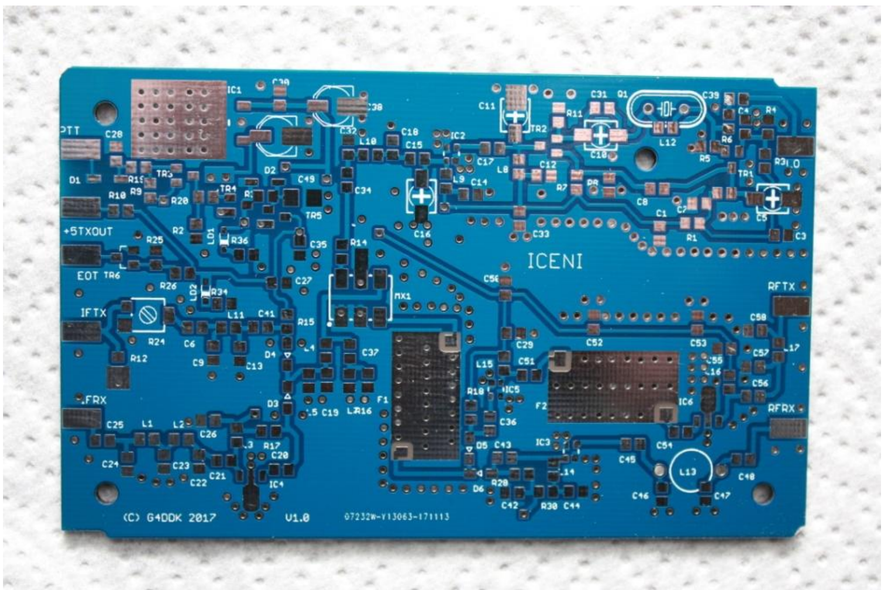


Photo 1. Iceni PCB

Note: On the V1.0 PCB there is a track break between R34 and LD2. You will need to bridge this with a blob of solder or LD2 will not illuminate on transmit. See later.

Marking the box

Disclaimer

Please be careful when handling the tinplate box. The edges can be sharp and can cut skin. The author takes no responsibility for any accidents you might have when handling the box. You have been warned!

Carefully examine the box. It has two overlapping edges. The PCB has two corresponding cut-outs that clear these overlapping edges. This shows which way up the PCB should be mounted into the box

Mark on the insides of the box, above the PCB, the word **'TOP'** with a marker pen so that these cannot be confused in the next step.

With the two sides as shown in Fig 1 , using, e.g a Vernier caliper, mark a line **all the way around** the inside of the box sides. The line should be 22mm below the rim of the box **TOP** side. This line is the plane for the PCB seam soldering. Everything to the **TOP** side of the box is regarded as the top (non component side) of the PCB

Hold the **IF input** end of the PCB up to one end wall and mark the location of the IF control connector pads and RF connector pads onto the box, using a scriber. The SMA connector marks should be **24.3mm** below the same rim as the 22mm lines. This is the centre line of the SMA spill. These are most easily marked by resetting the Vernier Caliper to 24.3mm and marking a second line inside the **END WALL ONLY, NOT on the sides**!

A third line should now be marked **AT THE IF END ONLY**. This should be **30mm** below the top rim. This line marks where the position for the **3.2mm** holes for the control feed-through capacitors should be located.

This is shown in Fig 1.

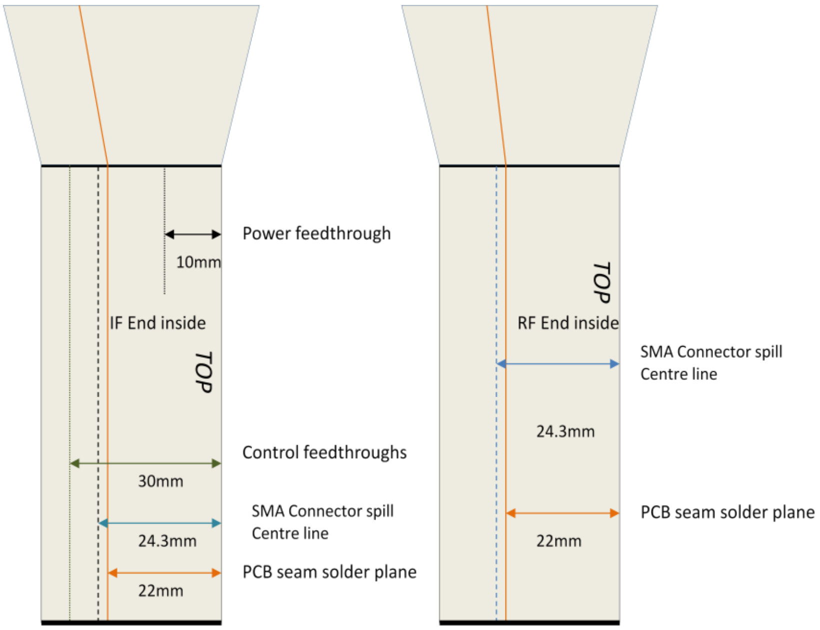


*Figure 1. the end walls of the box are marked with a third line **30mm** below the rim of the box. Mark the positions of the feed-through capacitor mounting holes directly above the PCB. The single hole, close to the rim of the box, is for the power feed-through capacitor. There is only one hole to be drilled and it should be 10mm below the rim. Also note that whereas the control feed-through capacitors are located BELOW the component side of the PCB, the power feed-through is located ABOVE the PCB. The cross coordinates for the holes are not given as it is more accurate to mark the positions of the holes using the PCB pads to ensure accurate location.*

Turn the PCB around and mark the positions of the two RF connectors and single LO connector on the RF End box end wall. The centre line of these connectors (assuming you use the supplied SMA connectors) is also **24.3mm** below the **'TOP'** rim of the box. There are no feed-through capacitors at this end of the box.

Hold the end wall of the box against a piece of wood as support and gently centre punch the positions of the holes ready for the next stage.

Drilling the holes

The five SMA connectors require **4mm** diameter holes. The four feed-through capacitors require **3.2mm** holes. All of these are, initially, best drilled with a 2mm pilot hole in a bench drill press with the end walls supported by the piece of wood. If the drill chuck does not clear the upright wall of the box, gently bend the wall slightly away from the drill. It should be gently bent back when you've completed drilling.

If necessary, clean up the holes with a countersink bit.

The punched and drilled box are shown in photos 2 and 3

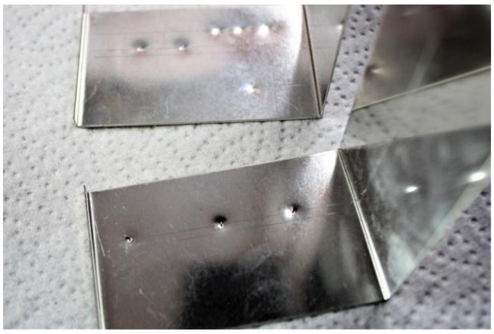


Photo 2 Box ends punched prior to drilling

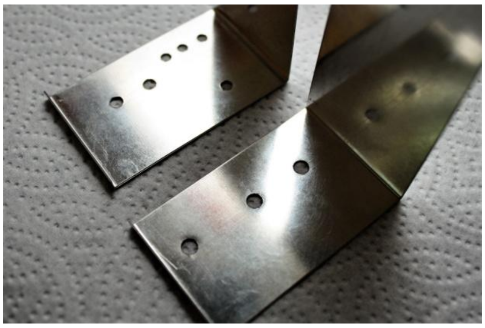


Photo 3. Box ends drilled as described

Assembling the SMD board

Start by identifying all of the parts and ticking them off on the Iceni component list.

You will need a small-pointed, temperature controlled, soldering iron suitable for SMD work. You will also require small diameter wire SOLDER of 0.25-0.3mm diameter. I prefer leaded solder.

Start by soldering all of the SMD resistors and capacitors to the board according to the component position overlay. Note the polarity of the tantalum and electrolytic capacitors.

Some SMD parts use a single colour identifier, others use two and three colours. This is because of a limited number of available colours of Sharpie pens! Don't mix Red and Orange. Place on a white sheet to see the correct colour.

Place the PCB, component side up, on a sheet of white kitchen towel. Any 'escaped' components are more easily spotted!

I recommend a small blob of solder on ONE of each of the many component pads (you can do the whole board at the same time, if you want). Photo 3 below shows my initial attempt at using solder paste and the supplied solder mask template, followed by reflow soldering. Only a few parts have been fitted to try out the technique for my first time. This board was then difficult to place and solder the remaining components due to solder on all pads. It was a good learning exercise.

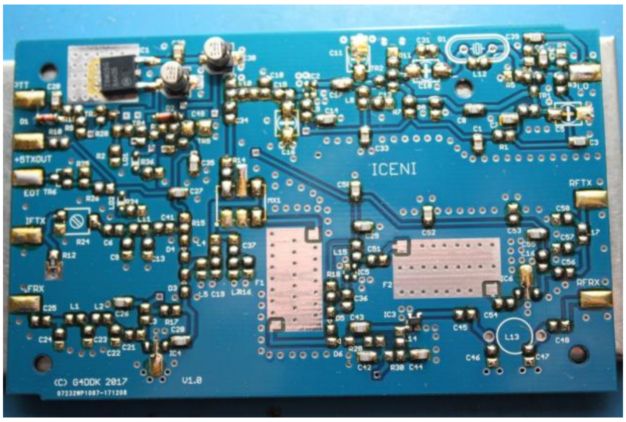


Photo 3. Partially populated test board using solder paste and template followed by reflow soldering. Shows promise for the future with assembled boards

Commence with the SMD resistors, double checking component values by both colour code and component code marking e.g. 100 = 10R, 471 = 470R etc.

Next, solder the SMD ceramic capacitors. Don't mix these up. Colour codes are similar on some parts. Measuring small value SMD capacitors can sometimes (often?) give misleading results. Use the colour codes. The single tantalum capacitor and the two electrolytics are polarity conscious. Check against the component connection sheet. Don't mix the 6 x 100pF capacitors marked with a SINGLE yellow line with the 47pF marked with two yellow lines.

The SMD inductors should be soldered by placing the pad beneath the component onto the solder blob on one pad. Heat the pad with the soldering iron. The inductor will drop down when the solder melts. Now solder the other end of the component. It may be best to go back and reheat the solder under the first join, just to be sure it is a good joint. Check each inductor for continuity using the Ohms range on your multimeter before moving on to the next inductor. Experience has shown that the inductors are responsible for most of the initial testing failures, due to a poor soldered joint. Take care to get these right.

Complete the SMD capacitor soldering with the trimmer capacitors. Note the grounded end for C5, 11 and 16. C11 should be orientated as shown by the silk screen on the board. i.e. the 'ground' end to R11.

Solder all of the SMD semiconductors observing the correct pin orientation. Make sure that the ground tab on IC4 and IC6 is flat to the solder pad as there is quite a bit of heat dissipated in these two devices. Similarly IC1, the voltage regulator. Make sure IC2, 3 and 5 (PSA4-5043) are soldered the correct way round. Start at the larger source pin and then the smaller, opposite, source pin finally soldering the remaining two pins.

The PSA4-5043 orientation can be confusing. The PCB is correct. Make sure you considering the device could be 90° out of position if you believe I've got it wrong!

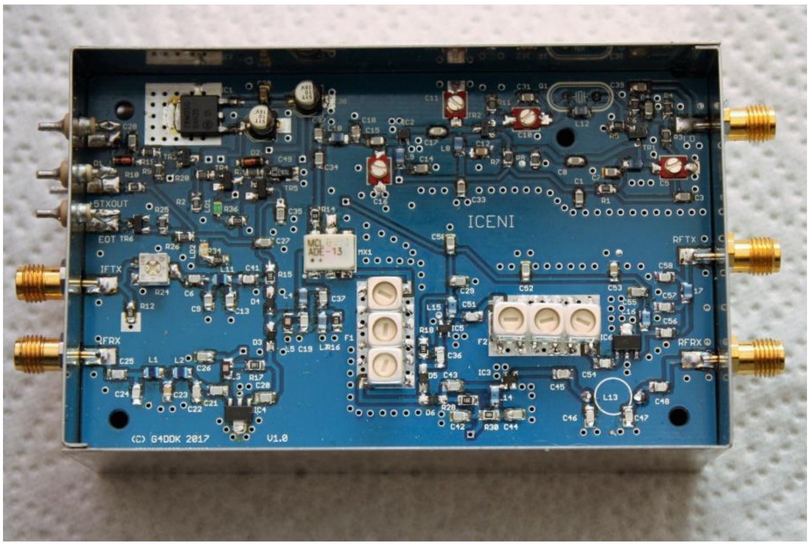


Photo 4. Assembled SMD board

Be careful handling the inductors. They are fragile and relatively expensive.

Solder the mixer, MX1 (observing the index dot), Filters F1 and F2, TX gain pot (R24) and trimmers onto the PCB.

The helical filter inputs and outputs are symmetrical. It doesn't matter which way round you solder it in. ALL the remaining pins must be soldered to the PCB. It is also important to solder the narrow ends of each filter to ground as shown in Photo 5.

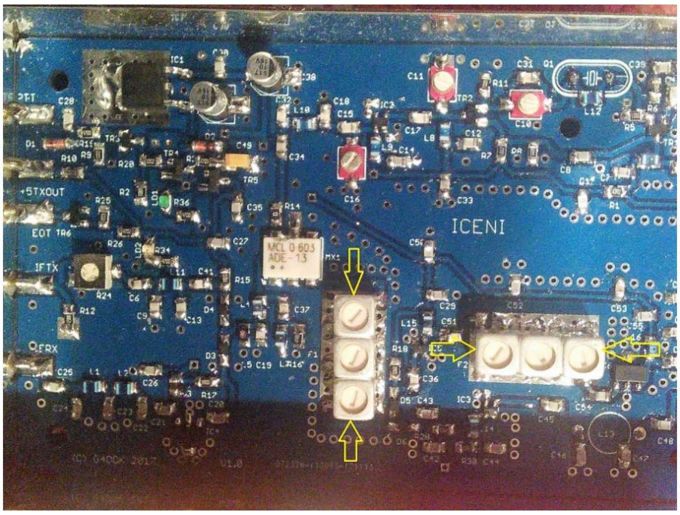


Photo 5 Solder the two helical filters along the edges shown by the arrows. The amount of bare board showing, to be solder to, is very limited on this board. Use a small pointed soldering iron and ensure a good solder filet along the length of the filter. Photo G4FRE.

There is one further thing to do. The connection between R34 and LD2 (Red transmit LED indicator) is broken on the version 1.0 PCB. A small solder blob, as shown in photo 6, solves this at low cost!

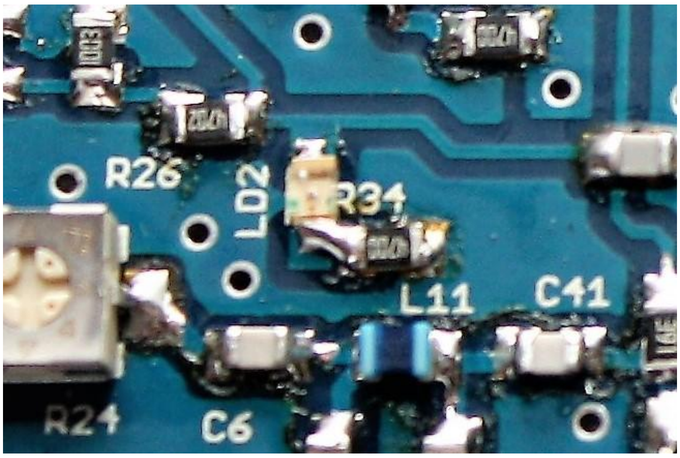


Photo 6 Solder blob between R34 and LD2

Do not solder the crystal or L13 onto the top of the board yet. Also, do not fit the copper heat-sink at this stage.

The SMD side of the board is now complete.

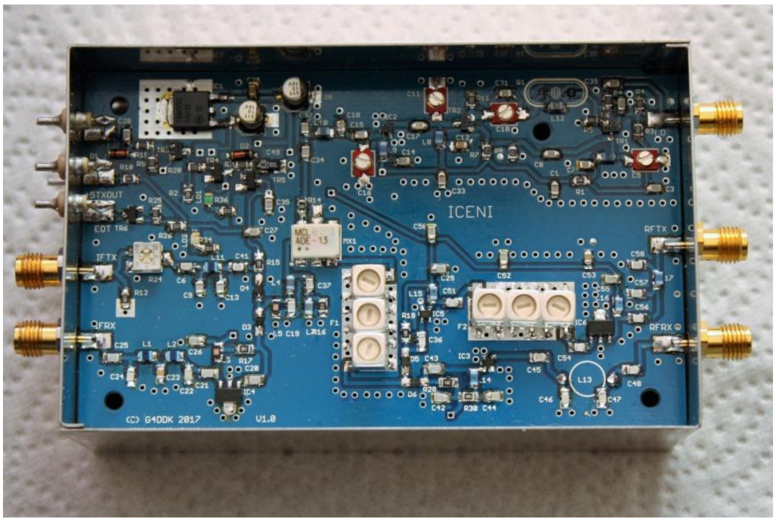
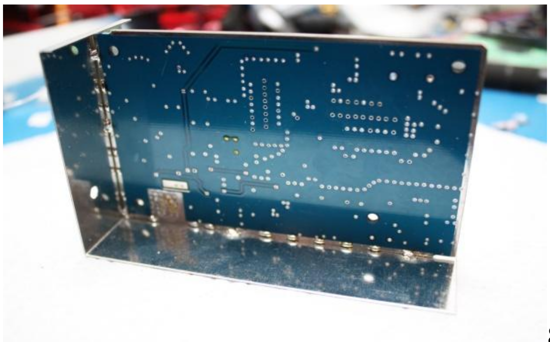


Photo 7. Complete, assembled board, in the recommended tinplate box housing.

Seam soldering

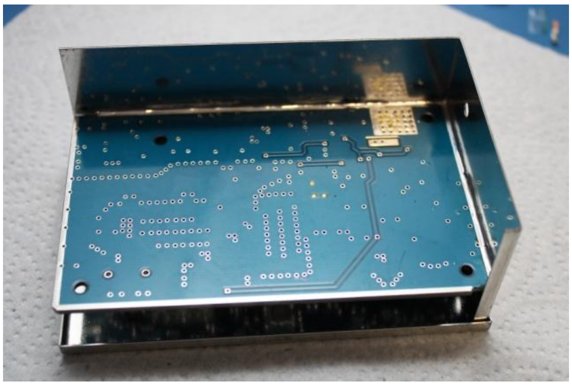
The stages involved in seam soldering the PCB into the box are shown below.

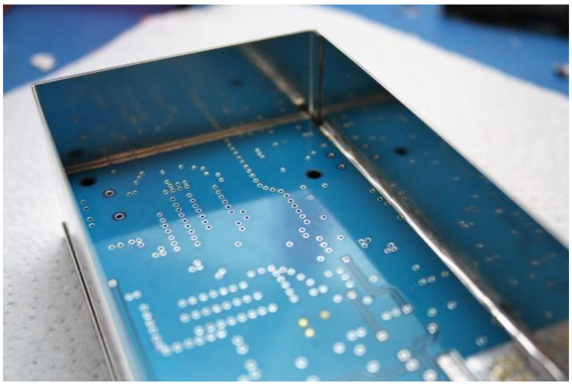
Mark the box as shown in Figure 1 near the beginning of this document.



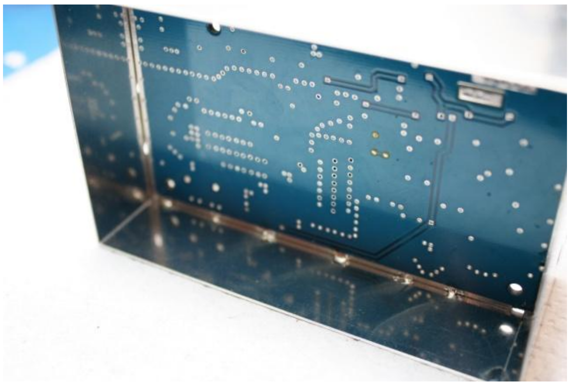
8a PCB offered up to first side of box (IF end) and tack soldered in 2 or 3 places along the 22mm line on both end wall and side wall.

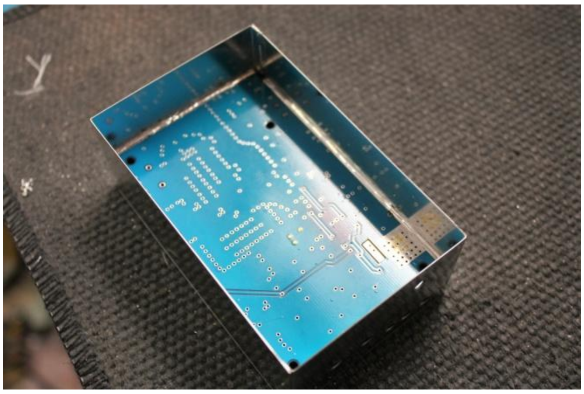
8b PCB seam soldered to the first side of the box along the 22mm line marked previously


8c PCB, with first side seam soldered and placed in lid of box to check fit


8d Second side of box carefully fitted in place in the lid. Not yet soldered

Tack solder the box corners to hold the assembly together


8e Second side tack soldered in place, taking care to keep the board level along the 22mm marked line.

 8f Second side seam soldered in place

Photos 8a - f. PCB seam soldering and box assembly stages

Check that the box top and bottom both fit correctly.

Soldering the connectors

Prepare the SMA connectors by cutting off the PTFE insulation from the rear of the connector. A Stanley knife is ideal for this, photo 9. Use a Dremel or similar tool, with cutting disc, to carefully shorten the SMA spills to about 5mm long. Side cutters may damage the connector by loosening the PTFE to spill fixing.

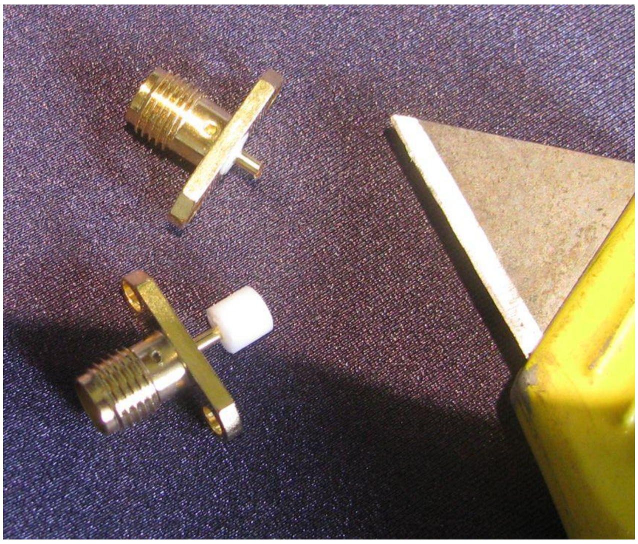


Photo 9. preparing the SMA connectors

When all five SMA connectors have been prepared they can be soldered to the outside of the box. The IF connections and the control feed-through end is shown in photo 10.

Photo 10. IF Connectors and feed-throughs soldered to the box end wall

I find the easiest way to fix the connectors prior to soldering the flange to the box is to tack solder the spill of the connector, through the **4mm** hole in the box end, to the pad on the PCB. Try to get the two hole connector flange vertical and the flange tight against the box and then tack the spill again. You can then **gently** rotate the connector body to bring it vertical. Make sure that the spill does not rotate and damage the PCB pad.

Repeat for the remaining four connectors.

Soldering the feed-through capacitors

Carefully slip one of the feed -through capacitors through the hole in the **3.2mm** solder tag. Push the feed-through capacitor through the **power** connection **3.2mm** hole in the box end. Bend the solder tag away from the box. Solder into place.

Repeat for the remaining feed-through capacitors. You shouldn't need solder tags on these feed-through capacitors. See photo 10

Soldering the crystal

Solder the crystal through the holes on the PCB. The crystal should be located on the TOP side of the PCB i.e. The side where the heat-sink and reverse polarity protection diode are fitted.
Avoid pushing the crystal case down onto the PCB. A small gap helps thermal stability by reducing heat conduction to the crystal. This is shown in Fig 2

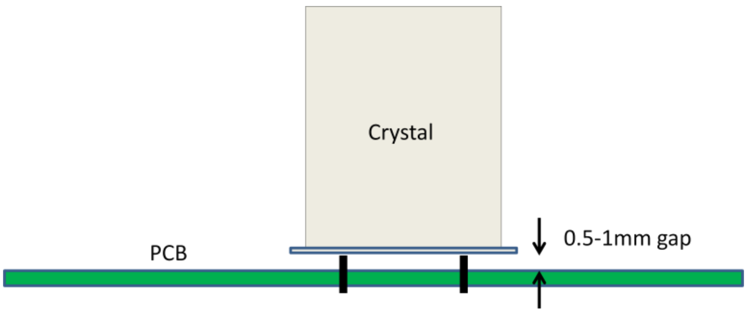


Fig 2 Soldering the crystal to the PCB pads

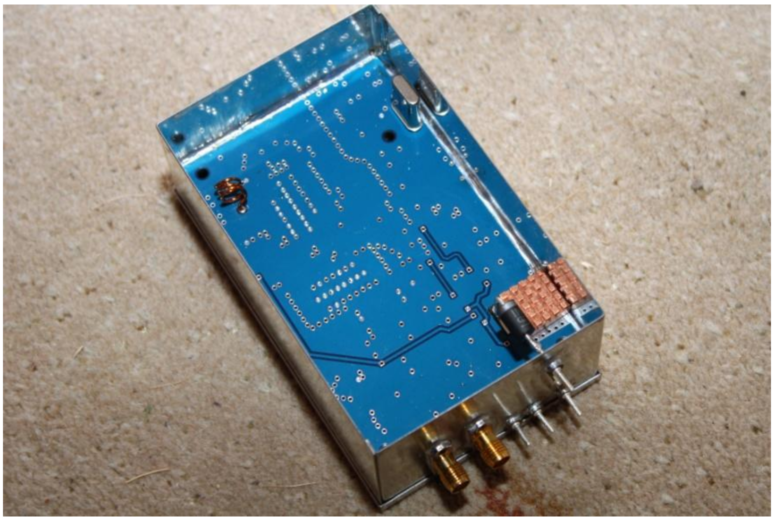


Photo 11 'Top' side of the PCB showing the Crystal, L13 (Alternative coil - see below), copper heat-sink and supply reversal protection diode

Details of L13

Two versions of the receive input coil are possible. The version using a pink TOKO 2.5turn S18 coil with Aluminium core is easiest to deal with, but since these coils are no longer made and only available on the surplus market, an alternative L13 coil could also be used. If you have a kit you might just have to accept the L13 that is included. The pin spacing on the PCB is slightly too narrow for the TOKO S18 coil. With care and some bending of the coil pins it can be made to fit.

The self-wound L13 consists of 3 turns of 22SWG enamel covered copper wire wound on a 5mm mandrel, with the turns spaced by one wire diameter. The short coil tails should be tinned with a hot soldering iron (the one used to seam solder the box is ideal) and the coil them sat down 2mm above the PCB, as shown in Photo 12 . The mandrel should be removed before fitting the coil.

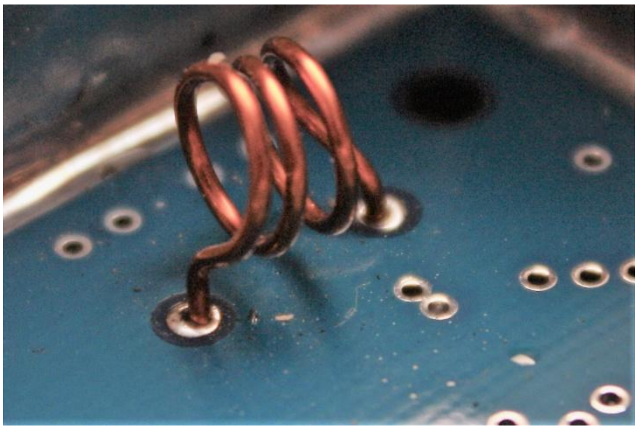


Photo 12. Alternative L13 3 turn coil, adjusted for lowest noise figure on receive.

Heatsink

Peel the backing from the underside of the small copper heat-sink and stick the heat-sink ON THE CRYSTAL SIDE of the PCB (designated bottom on this PCB) immediately over the voltage regulator. This is the on the opposite side of the board to the regulator! Be careful not to short the copper onto the adjacent power input pad. The heat-sink should be pushed firmly up against the inside of the box.

Connecting the feed-through capacitors to the board pads

On the 'top' side of the PCB connect the 1N5401 (or similar 3A silicon diode) to the power feed-through capacitor and the power supply input pad. In this case the cathode or banded end of the diode should connect to the Vcc pad on the top of the PCB as shown in photo 13. Note that you should carefully dress the diode leads to allow the diode to fit between the power feed-through capacitor and the Vcc pad.

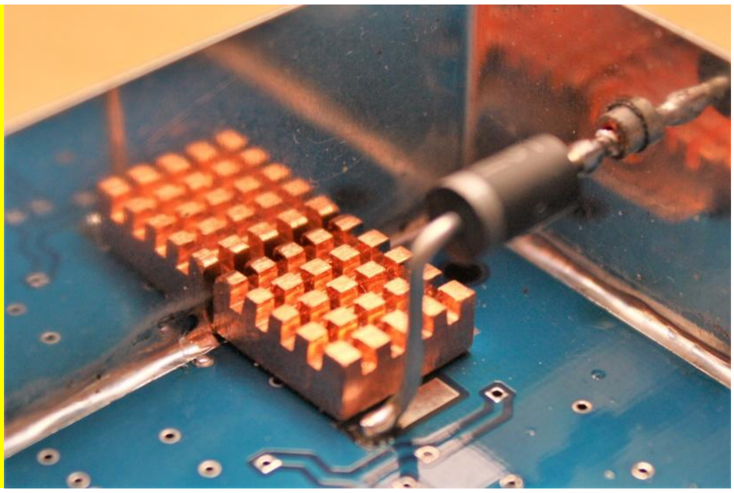


Photo 13. 1N5401 diode connected from the power feed-through capacitor to the Vcc pad and the copper heat-sink.

Connect the other feed-through capacitors to the adjacent pads (EOT, TX 5V and PTT) on the PCB with suitable, short, lengths of wire.

Crystal stability

Cut a small foam (plastic or rubber) 'box' to fit over the crystal. As the crystal is not subject to moving air in the screened box version, this may not be necessary. A better solution for either version is a crystal heater, such as a 38°-40°C Posistor (if you can still find one). The best solution is a DB6NT QH40 heater or the similar unit from RF Microwave (Franco Rota).

An even better solution is to frequency injection lock the internal oscillator by injecting a stable (by a disciplined GPS or OCXO) 101MHz signal into the LO port at a level of between -3 and +3dBm (not too critical).

Injection locking usually provides a really good LO source. The close-in noise will be determined by the PLL loop bandwidth, whilst far-out noise will be determined by the crystal oscillator in the Iceni.

Test and Alignment

Refer to the schematic circuit diagram.

Connect a 50Ω termination to the receiver (432MHz) input and IF (28MHz) output

Connect +10v to +12v to the power input feed-through capacitor. The supply should be set to current limit at no more than 300mA if possible.

Check that the current taken is about 250mA. If significantly more or less you probably have a fault and this must be fixed before proceeding.

Check that there is +5V present at the junction of R1 and C1.

Check that you have +5V at the junction of R30 and C42.

The green RX LED, LD1, should be illuminated

Ground the PTT feed-through

LD1 should extinguish immediately and then after a few tens of ms the red TX LED, LD2, should illuminate. If it doesn't illuminate, check that you have bridged the R34/LD2 track break with a small blob of solder. From V1.1 onwards the fault will be fixed.

Check that there is +5V at the junction of L16 and C55. The total current taken will briefly dip before the TX LED illuminates. Remove the PTT ground.

Switch the power off.

The 0Ω bridge, R14, will need to be temporarily removed in order to allow the local oscillator power to be measured. This resistor, or a short wire link, must be in place after the measurement in order for the transverter to work.

Ideally a spectrum analyser set to 404MHz centre frequency with 10MHz span and +30dBm amplitude should be used. However it is possible to do the alignment with a power meter and frequency counter or accurate frequency readout receiver.

Using a short coaxial lead, connect the lead between the pad (junction of R14 and C34) and the bare ground plane pad adjacent to R14.

Connect a spectrum analyser (preferably) to the other end of the lead. Set the input amplitude to +30dBm, span to 1MHz and centre frequency to 404MHz. Double check that you will NOT damage the spectrum analyser. If necessary, use additional attenuators between the test lead and the spectrum analyser input.

Switch the power back on.

Adjust C5 trimmer until you see a strong signal at 404MHz. Adjust C11 and then C16 to peak the 404MHz output. This should be close to +12dBm (although Beta builders have reported as much as +15dBm). Now adjust C10 to bring accurately onto frequency. It may be necessary to go back and forth, re-adjusting C5 and C10, as there is some interaction between these three trimmers. Set the frequency as accurately as you can to 404.000.000MHz. An error of 200Hz maximum should be allowed.

Switch the power off

Remove the spectrum analyser connection.

Replace R14 or wire link.

Connect a 28MHz-30MHz receiver to the RX IF output in place of the 50Ω termination.

Tune the IF receiver to 29MHz.

Switch the power back on.

The noise in the receiver should increase. It may still be slightly low at this stage.

Using a ceramic (preferably) core trimming tool, turn the cores of F1 clockwise (cores inwards) in each section of the filter. The noise should increase. Adjust the core of L13 Toko S18 (if used) or the spread of the three turn coil to maximise the noise and then adjust again to obtain the best SNR on a weak signal.

That is the receive converter aligned.

Switch off the Iceni power.

Connect the spectrum analyser, set to 433MHz centre frequency, 200MHz span, +30dBm amplitude reference level, to the RF TX (433MHz) output socket. Do not connect any 28MHz input signal at this stage.

Switch the power on.

Connect a ground to the PTT feed-through capacitor.

The spectrum analyser should show no output. There might be a very small, residual, 404MHz output if you check that frequency.

Adjust the TX gain potentiometer, R24, fully anticlockwise to give maximum attenuation.

Connect a 0dBm 29.00MHz input source to the 28MHz IF input. There might be a slight 433MHz output due to IF signal leakage across the attenuator.

Slowly turn the TX gain control clockwise 1/4 turn and as you do so the 433MHz output should increase slightly. Adjust the cores of F2 to peak the output. Again the cores should be turned clockwise (into) the filter body. There will be a broad, but distinct, peak.

Once the output is peaked, continue to adjust the TX gain pot up to a maximum of about +18dBm output. The output increase should be steady without any sign of jumps in level. Widening the spectrum analyser span to cover to 1.5GHz (starting at ~10MHz) should reveal just the 433MHz output plus a small amount of the second and third harmonic, but at over 40dB below the 433MHz output, at maximum output.

It might be necessary to go back and re-adjust C11 and C16 to further reduce the already-low 404MHz LO appearing at the output. Do not adjust far and in any case the 433MHz output should not reduce more than about 1dB.

Remove the PTT ground

Check that the local oscillator is accurately on 404.000.000MHz by monitoring with an accurately controlled time-base (clock) frequency counter. Readjust C5 and C10 if necessary to bring the LO accurately onto 404MHz.

Switch off the power.

The transverter is now aligned. It is possible to go back over the these adjustments (what radio amateur wouldn't!) to improve performance slightly. If you don't have access to a spectrum analyser it is HIGHLY recommended that you do arrange access to one as it would be irresponsible to assume everything is working as it should without properly checking it on the appropriate test equipment. Analysers are often available for this purpose at the regular UK Microwave Round Table events.

Interfacing the Iceni transverter to the HF transceiver

Drive level

Ideally the Iceni transverter mixer should be driven with a 28MHz level of 0dBm. Full transverter output will be achieved at this level. Back off the output to a maximum output of +18dBm

Single coax IF

It is possible to simply combine the two IF ports on the Iceni transverter using a 3dB splitter/combiner, such as one of those from Mini Circuits Labs, to bring the two ports to a single HF radio IF port such as the ICOM PRO range. It should also be possible to re-combine the 28MHz IF input and output from an Elecraft K3 with similar splitter/combiners so that the Iceni can be located remotely from the K3, with just a single coax connection

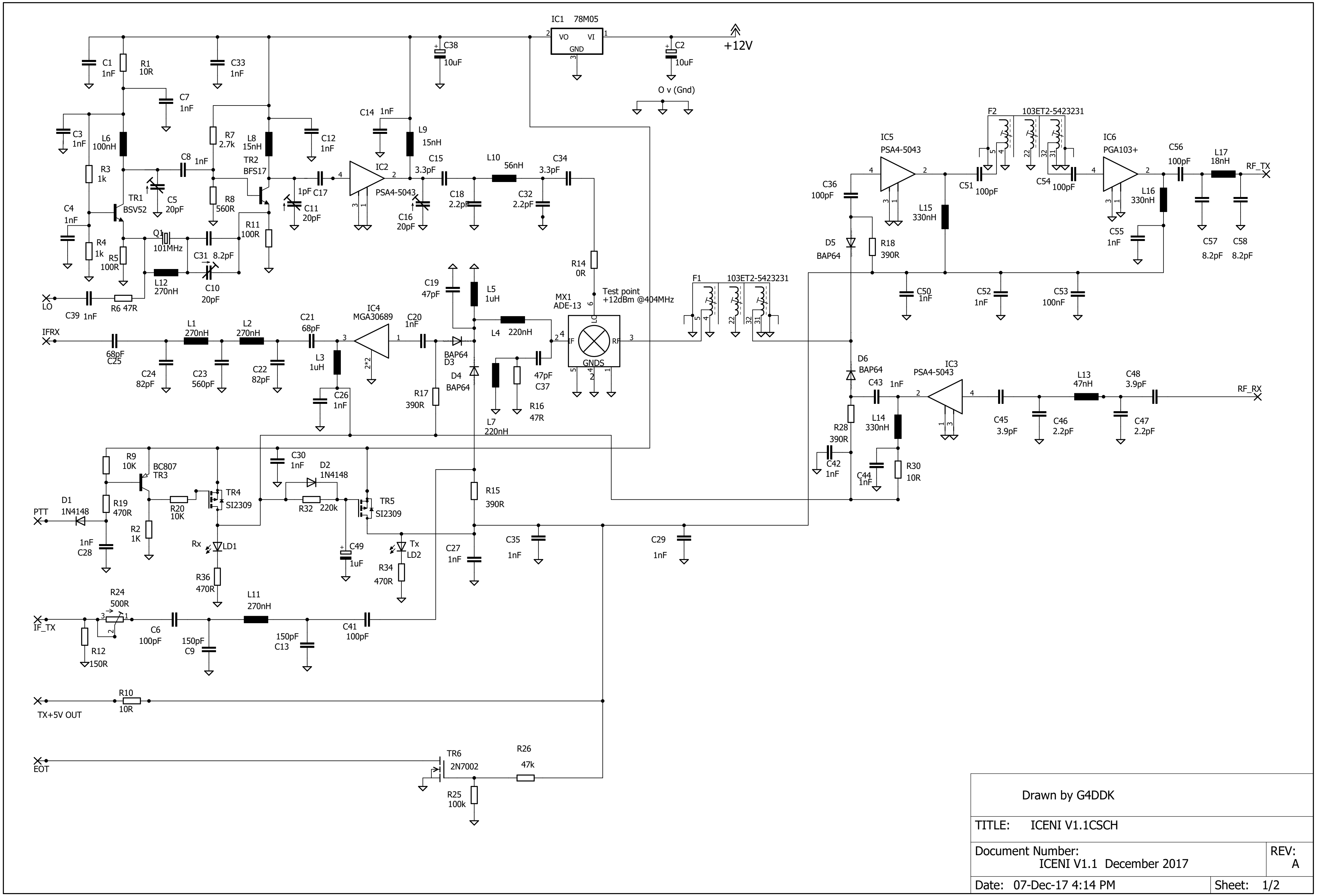
Higher power rigs

In the case of HF radios such as the IC7300, with no low level transverter output, it will be necessary to use an external power attenuator to dissipate some of the higher power output, or use one of the low level 'interceptor' boards . I personally use a 25W, 20dB attenuator with my IC7300, with the IC7300 output turned down to 10W. This has two advantages. If the power accidently gets turned up to full then the power attenuator will ensure that no more than 1W is applied to the transverter. The power attenuator will not burn out immediately, so you will have won some time to realise something is not right and turn the power down. Transverters are, generally, more tolerant to overdrive damage survival than modern RF MOS power amplifiers. Power attenuators are cheaper than a new transverter. Suitable 10m band power attenuators are simple and cheap to build.

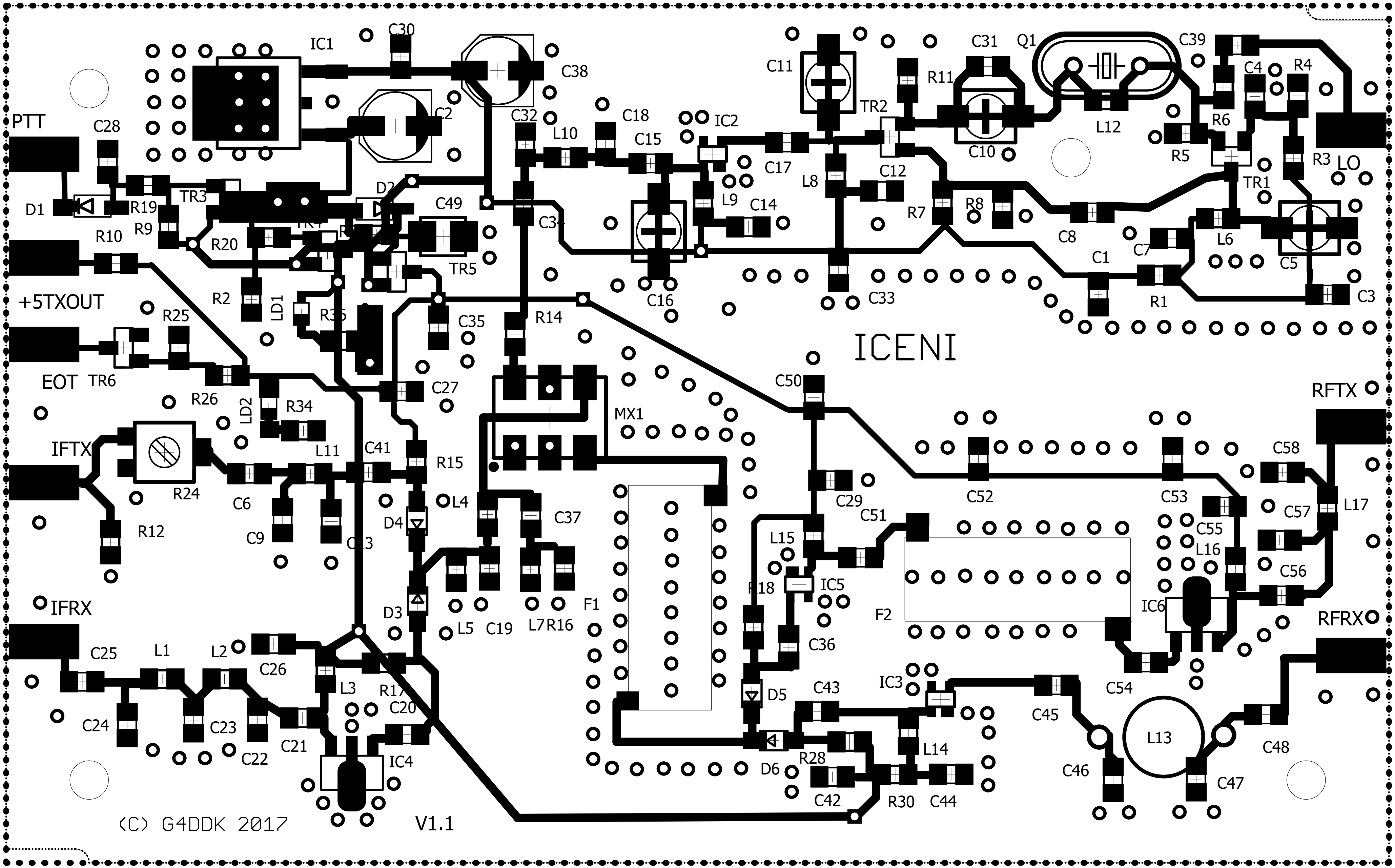
Secondly, the 20dB provides a good way to lose some of that unnecessary gain ahead of the HF rig. You may not want the HF rig's 20dB preamp on normally, but now it will come into its own in overcoming the loss of the attenuator.

Document History

Date	Issue	Changes
28/11/2017	Draft 1	Initial version
9/12/2017	Draft 2	Added text and some new photos
23/12/17	Draft 3	More photos and text
27/12/17	Draft 4	More photos and text
30/12/17	Draft 5	Filter photo and more text
5/1/18	Issue 1.0	Reviewed document



IC1 78M05
VO VI GND
+12V
C38 10uF
C2 10uF
O v (Gnd)
C1 1nF
R1 10R
C33 1nF
C7 1nF
C3 1nF
L6 100nH
R7 2.7k
L8 15nH
C12 1nF
C14 1nF
L9 15nH
C8 1nF
R3 1k
TR1 BSV52
C5 20pF
TR2 BFS17
R8 560R
C11 20pF
1pF C17
IC2 PSA4-5043
C15 3.3pF
L10 56nH
C34 3.3pF
C16 20pF
C18 2.2pF
C32 2.2pF
C4 1nF
R4 1k
R5 100R
Q1 101MHz
C31 8.2pF
R11 100R
C10 20pF
L12 270nH
R14 0R
LO
C39 1nF
R6 47R
C19 47pF
L5 1uH
Test point +12dBm @404MHz
MX1 ADE-13
IFRX
C25 68pF
L1 270nH
L2 270nH
C21 68pF
IC4 MGA30689
C20 1nF
L4 220nH
IF LO RF GNDS
C24 82pF
C23 560pF
C22 82pF
L3 1uH
BAP64 D3
D4 BAP64
C37 47pF
C26 1nF
R17 390R
L7 220nH
R16 47R
R9 10K
BC807 TR3
C30 1nF
D2 1N4148
TR4 SI2309
TR5 SI2309
R20 10K
R32 220k
D1 1N4148
PTT
R19 470R
R2 1K
C28 1nF
R15 390R
Rx LD1
C49 1uF
Tx LD2
R36 470R
R34 470R
C27 1nF
C35 1nF
C29 1nF
R24 500R
IF_TX
C6 100pF
L11 270nH
C41 100pF
R12 150R
C9 150pF
C13 150pF
TX+5V OUT
R10 10R
TR6 2N7002
R26 47k
EOT
R25 100k
F2 103ET2-5423231
IC5 PSA4-5043
IC6 PGA103+
C56 100pF
L17 18nH
RF_TX
C36 100pF
C51 100pF
C54 100pF
L16 330nH
L15 330nH
C55 1nF
D5 BAP64
R18 390R
C57 8.2pF
C58 8.2pF
C50 1nF
C52 1nF
C53 100nF
F1 103ET2-5423231
D6 BAP64
IC3 PSA4-5043
C43 1nF
L13 47nH
C48 3.9pF
RF_RX
L14 330nH
R28 390R
C45 3.9pF
C46 2.2pF
C47 2.2pF
C42 1nF
C44 1nF
R30 10R
Drawn by G4DDK
TITLE: ICENI V1.1CSCH
Document Number: ICENI V1.1 December 2017
REV: A
Date: 07-Dec-17 4:14 PM
Sheet: 1/2


ICENI
PTT
+5TXOUT
EOT
IFTX
IFRX
LO
RFTX
RFRX
(C) G4DDK 2017
V1.1