Building a W1GHZ-based 2.4GHz Transverter

by

Doug VK4ADC

20 May 2011, continuing.....

My reasons for following this style of transverter construction has been documented on my "**2.4 GHz & 3.4 GHz Transverter Ideas** (http://www.vk4adc.com/web/index.php/microwave-projects/63-transverters/141-24-ghz-a-34-ghz-transverter-ideas.html) " web page. This is my first attempt at getting equipment running on this band so please forgive any assumptions that turn out to be "false leads" along the way. What I will do here is to "tell the story" about what I achieve.

I should also add that I don't actually have any test equipment that will allow me to directly operate or measure above 1 GHz so I have had to contemplate & devise other ways of tuning up the device. My frequency counter has an upper limit of 1020MHz, my sig gen tops out at 1040 MHz, and (as I found out) my spectrum analyser doesn't quite make it to 1000 MHz...

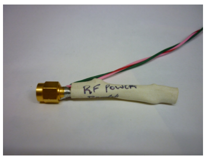
This project sounds like an almost impossible task without really suitable test equipment but if you don't try then you can't succeed... and I have built up equipment before now without access to test gear. If all else fails, there are plenty of microwave enthusiasts out there who will assist you in getting it going - and many do have the gear to make it easy.

The master plan :

a. a W1GHZ transverter PCB, fitted with 4 x 32mm OD copper pipecap filters
b. an X-Locker 3 synthesiser PCB at 752.6666 MHz
c. a 1-2 watt PA using an RFMD SPZ-2026Z MMIC
d. an antenna changeover PCB using an Omron G6Z relay
e. a sequencer PCB using a PICAXE

... and maybe a lower-noise receive preamplifier using a RFMD SPF-2086TK GaAs FET plus 2 more pipecaps for better front-end selectivity..

I couldn't locate my previous homebrew "UHF low power load/detector" so I built up another one consisting of 2 x 100 ohm 0805 SMD resistors in parallel, a HSMS-286F SMD diode as a peak detector and a 10nF 0805 SMD capacitor as an integrator. These parts were all mounted at the end of a short length (about 60mm) of UT141 semi-rigid coax with a SMA male plug on the far end. I then created a calibration table for RF powers from -10dBm to +13dBm against my sig gen at 100 MHz, with the hope that it will retain some semblance of accuracy up to at least 1000 MHz, and hopefully beyond that.



The RF Load/Detector is not much to look at as the SMD parts are all sheathed in heatshrink - but it does the job.

For info, the following voltages were tabulated : -10dBm = 0.056V, -5dBm = 0.122V, 0dBm = 0.249V, +1dBm = 0.284V, +2dBm = 0.325V, +3dBm = 0.370V, +4dBm = 0.420V, +5dBm = 0.479V, +6dBm = 0.542V, +7dBm = 0.615V, +8dBm = 0.693V, +9dBm = 0.787V, +10dBm = 0.887V, +11dBm = 1.003V, +12dBm = 1.133V and +13dBm = 1.278V, all measured on a DMM set to the 4V DC range. My generator only produces +13 so calibration beyond that was not possible.

Using the DMM was ok but a quick look in the parts boxes produced a 50uA meter and a quick test using the sig gen with it in lieu of the DMM indicated that it was useful to measure from -5dBm to +1dBm as was. I ran a few further tests and determined that a 22K resistor in series made the power range cover 0dBm to +13dBm. That meter and the load/detector -

plus it's calibration table for the two ranges - will remain as part of the test gear from now on.

Step A : The W1GHZ transverter PCB :

The 4 PCBs arrived in the mail from Paul W1GHZ in the form of 2 x 2.4/3.4 transverter boards plus 2 x local oscillator/multipliers. The plan is to build up the 2.4 GHz (based on 2403 MHz for operation here in Australia) transverter PCB first, then the X-Locker3 and get them going and follow that up with the remainder of the basic building blocks. Of course, the transverter construction is exactly the same as for 2.3 GHz (i.e. 2301 or 2304 MHz) operation and, physically, only the pipecap tuning positions are different.

The Minikits-supplied pipecap's OD was found to be exactly 32mm so the positioning on the PCB was based on that value. Paul has placed a "via" on the PCB at the centre points for each pipecap so it is an easy enough task to mark the top of the PCB at points 16mm from the via in 4 directions. I used a fine felt pen and just placed dot points at around 90 degree points from each central via. If your pipecaps are slightly different diameter, adjust that 16mm (i.e. 1/2 of the OD) value as required.

Marking out the centres with a 45 degree square

Gives a centre point cross

After drilling to 6mm and tinning with solder

The 5mm nut is soldered to the top.

Note that this process has been revised in view of the heat required to solder the pipecap to the PCB. The image has been left here to show what it might look like...

Inside view of the pipecap. This is what it will look like inside once assembled onto the PCB

This shows the position of the locknut on the screw thread. Note that a lock nut is essential.

I had pre-prepared the pipecaps by drilling the central hole in the top and soldering a 5mm nut (with the plating filed off on one broad-side (the under-side) plus around each of the the 6 sides) to the top but that came unstuck at the time when I was actually installing them on the PCB. I suggest that you drill the hole (mine were drilled 6mm) and tin the top around the hole but don't attempt to actually solder the nut in place - that comes later. I should also warn you that there are only two ways that you will be able to get the pipecap hot enough to "tin" - (1) a butane soldering torch (causes oxidisation/discolouring); or (2) a hot air soldering / desoldering tool. A normal workbench soldering iron will not provide anywhere near enough heat.

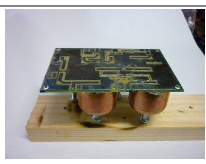
You will also need to "tin" around the outer (bottom) edge of the pipecap so that it makes it easier to solder it to the PCB and doing it while heating it up to tin around the tuning hole makes the job easier. I found that doing the heating phase on a scrap piece of timber was the best way to go, handling/rotating the hot pipecap with longnose pliers to save burning the fingers.

Soldering the pipecap to the transverter PCB is a similar story - you cannot get enough heat from the benchtop iron to solder the pipecap - it simply drains away all of the heat like the massive heatsink it is. I found that the way to do it successfully was to use the hot air tool (with the temperature set to around 430 degrees Centigrade) and work my way around the circumference of the pipecap, applying solder to the joint of the pipecap (already tinned) and the PCB (supplied tinned) as I went around. Then, while the pipecap is still hot, apply a little more heat on the top and attach the tuning screw's nut (with the screw protruding into and centred through the hole), applying some more solder around it.

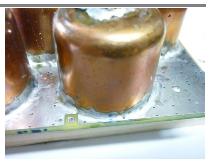
My final suggestion is that you mount the two middle pipecaps on the PCB first and the end ones last as that gives better access to solder in the central area.

With the 4 x pipecaps soldered to the PCB, then comes the task of adding the remainder of the components plus the probes into each pipecap. Following Paul's dimensions, I cut 8 pieces of tinned copper wire, bent then into a 90 degree angle, cut one to 11.6mm (= 0.4" + 0.060" = 0.46" = 11.68mm) then held it alongside the other 7 pre-bent pieces and cut all 7 to the same 11.6mm length. The other side of the "L" remained long until after the probe was soldered through the PCB. That ensured that all probes were the same length and were easy to install.

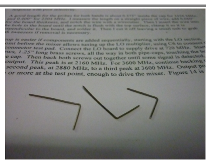
The 4 pipecaps soldered to the top of the PCB



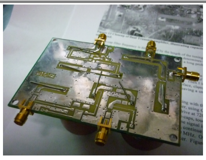
The underview shows nothing on the underside of the PCB - not even any scorching when a hot air gun is used for the soldering process.



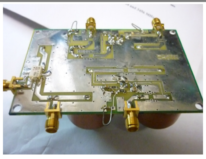
The pipecap is placed in the position according to the dots centred on the middle via and extra solder is added around the bottom circumference to provide a good mechanically-solid and electrically-sound joint.



This shows some of the 11.6mm long right-angle probes. A total of 8 are required.



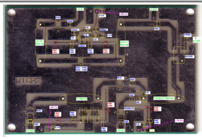
The probes are inserted correct way into the holes on the PCB and then soldered into place, ensuring that they are as vertical as possible. The excess is then cut off leaving a short horizontal piece soldered to the PCB land.



The remainder of the parts are added to the PCB after the above steps are taken...

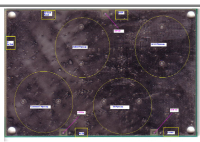
Note that the shorting straps/loops across each power supply point were installed before any of the semiconductors (as per the note below).

There is no actual parts layout provided in the original W1GHZ document but the positioning of the parts is fairly simple by following the schematic and placing parts as per that. I have scanned the PCB's themselves and have overlaid the parts positions on a copy - and am providing that here for others to follow :



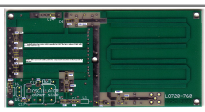
This is the parts layout of the bottom of the W1GHZ transverter PCB for 2.4/3.4GHz.

Full view link (/~vk4adc/web/images/UserFiles/Image/W1GHZ/Tvtr bot 2 lg.JPG)



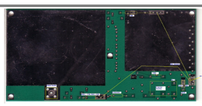
This is the parts layout of the top of the W1GHZ transverter PCB for 2.4/3.4GHz.

Full view link (/~vk4adc/web/images/UserFiles/Image/W1GHZ/Tvtr top 2 lg.JPG)



This is the parts layout of the bottom of the W1GHZ LO PCB for 2.4/3.4GHz.

Full view link (/~vk4adc/web/images/UserFiles/Image/W1GHZ/LO under 2 lg.JPG)



This is the parts layout of the top of the W1GHZ LO PCB for 2.4/3.4GHz.

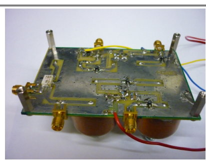
Full view link (/~vk4adc/web/images/UserFiles/Image/W1GHZ/LO top 2 lg.JPG)

Normal care should be taken when soldering in most of the semiconductors (ie. antistatic practices) but extreme care should be applied to the fitting of the receive preamplifier MGA86576 device. It is my habit to add a temporary shorting wire loop on the voltage feeds (to PCB ground) when adding MMICs and FETs on PCBs and I recommend you follow suit. It means that the output pin/drain is grounded via whatever supply resistance is in circuit and prevents a high static voltage from being developed - and it works for me every time, no damaged devices to date....

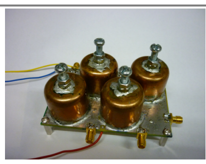
Once all parts were mounted on the PCB, I added 4 x 25mm long 3mm tapped metal standoffs to the underside of the PCB assembly (the opposite side to the pipecaps). That completed the transverter construction phase and it was nearly time to move on to the LO generation stage.

I powered up the LO multiplier stages, injected 752.666 MHz from my sig gen into the relevant port and used a Yaesu VR-5000 receiver tuned to 2258 MHz on the LO test port (adjacent to the mixer LO input point). The tuning screws on all pipecap filters are initially set all the way in - i.e. touching the PCB inside. I backed off each tuning screw about 2 turns then started to hear the signal on the receiver. I tuned the first pipecap for a peak then the second pipecap similarly. I went back to each one a few times then locked off the nuts. I now had LO injection functioning at 2258 MHz.

Operational note : I needed +8dBm from the generator at 753-odd MHz to make the first MMIC on the transverter board really start to generate harmonics... That is a little higher than Paul suggested in his notes - but be prepared for it.



This is the bottom view of the completed W1GHZ transverter PCB for 2.4/3.4GHz.



This is the top view of the completed W1GHZ transverter PCB for 2.4/3.4GHz.

Something to note :- the W1GHZ transverter is designed to run at the following voltages :

LO multiplier chain @ +8 volts

Transmit segment @ +8 volts

Receive preamp stage @ **+5 Volts**

Don't mistakenly put +12V on any of these, or even +8 volts on the receive preamp.

I cheated a little here - I later added an extra 78L05 under the PCB at the preamp power supply point so that I could feed it with +8V or +12V... without any over-voltage concerns.

Step B : The X-Locker3 synthesiser PCB :

The X-Locker 3 PCB was made up and initially a short hairpin inductor and a BB833 varicap diode was used for the VCO. When it was powered up and I looked for the frequency to make the VCO lock, using a variant of the normal X-Locker software, it was around 1000 MHz.. Whoops a bit too high, but better than being too low. I increased the size of the wire loop inductor by around 75% and then ended up with it locking from around 730 MHz to 860 MHz. I later changed the inductor to a 27nH SMD style and the varicap to a BB208-03 to obtain a suitable VCO frequency range. I required 2258.0 MHz injection to the mixer (for a 145 MHz IF) , and since the transverter PCB used frequency tripling, that meant that I needed 752.6666 MHz from the synthesiser. The magic numbers were N=6774 and R=90 (with a 10.000 MHz reference, thus a 111.1111 KHz PD frequency), with the second preset value at N=6777 to give 2259.0 MHz alternate injection, if required (=144 MHz IF).

My frequency counter showed the desired frequencies (752.666 & 753.0) were being generated but the noise obvious on a receiver tuned on-frequency was a major concern. Instead of a clean beat note on SSB, there was considerable "white noise" present too. That meant that the synthesizer would need more work - later though. My primary concern was to see if it would work with the transverter PCB for now.

The X-Locker 3 PCB produced a measured +9dBm at 752.666 MHz, so a slight margin on the noted +8dBm measured as being required for the transverter board LO requirements.

I connected the LO signal across via a short SMA male-male cable, powered up the multiplier stages and monitored the signal at the LO test point - again to note that at 2258 MHz, the white noise was going to be a problem. Even so, I connected the IF port to an Icom R7100 receiver at 145.000, the sig gen into the receiver port at 801.000 MHz (to use the 3rd harmonic) and the voltage onto the receiver front end. I had to put the generator level up to about -40dBm before I was able to find the initial signal peak on the "common" pipecap filter to the mixer. After that, I was able to reduce the generator output down and still hear a signal at around -70dBm out. My generator has 3rd harmonic attenuation of -43dB when used at 432 MHz (for tuning at 1296 MHz) but I have no idea what it is when used at 800 MHz, or even if the generator output level is correct up there. If it was in the same order, that makes it around -70dBm - 43dB = -113dBm. Maybe it's worse, but at this stage, I don't know.

I took the power supply off the front end, connected +8 volts to the transmit stages, 145 MHz at 0dBm into the IF port, and the transmitter output port via 3 x 20dB BNC cylindrical attenuators series-ed into the VR-5000 now tuned to 2403.0 MHz. The "common" pipecap was already close to being tuned through the previous operation so I unscrewed the transmit stage pipecap until it was physically out around the same amount and, yes, the signal at 2403 peaked. I went back and re-peaked all pipecaps, the LO chain ones included.

By this stage, I had "received and transmitted" at 2403, even if only within the workshop, and using my test equipment !!

My thoughts turned to the synthesiser-generated noise issue : what was I going to do about it ?? Time for some experimentation with the software settings within the chip ?? I will need to come back to experiment with this point later.

Step B(1) : The W1GHZ Oscillator/Multiplier PCB

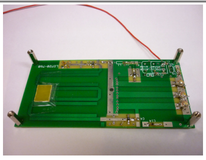
I thought "well, I have two W1GHZ LO PCBs - it might be time to build up one of those as a fallback..". I printed out just the LO pages from the original PDF and set about adding the components to the PCB.

I must point out that the inductor just after the resistor fed from the 80.000 MHz oscillator should be 0.22uH and not 0.22nH as shown on the schematic. With 0.22uH, that LC network resonates at 83 MHz so is close to being the series-resonant trap described in the text. At 0.22nH, it is effectively just a short circuit at 80 MHz.

I have a few different frequency TTL crystal oscillators in my junkbox so it was time to get the calculator working and see if I could use any of them. I had 50.000 MHz, 60.000 MHz, 64.000 MHz, 66.666 MHz, 80.0 MHz (the same as the original article) available. I decided that I would use a 14 pin DIL socket to mount the crystal oscillator so I could change it as required to try different oscillator configurations, and simply cut off all unused pins (2,3,5,6,9,10,12 &13). The 80.0 would provide 720 x 3 = 2160 giving a 243.0 MHz IF, and the 50.0 would give 750 MHz = 163 MHz IF - that is "if" the harmonic generation worked ok at a x5 value instead of a x3. I can get my hands on some 83.000 oscillators cheaply so I may yet use one of those later on, and that gives a 162.0 MHz IF for 2403. Not much different in IF but a lower frequency multiplication factor is involved.

I didn't have any MAR1 MMICs immediately available but an ERA3 has much the same gain so would fit the bill if the biasing resistor was changed. The rest of the components were on hand. Note that this LO board also runs at +8V, not +12V.

It didn't take long to build up the PCB more-or-less as described, and I decided to initially use the 80.000 MHz oscillator - however - I did make a change with the 240/250 MHz comb filter parts. Paul used 2 x 18pF 0805 SMD capacitors to tune his at 240 but I decided to use 2 x 15pF 0805 caps plus some small extra critical-tuning caps. At 240 MHz, these extras turned out as 4.7pF to give best output level (i.e. 34.7pF total {+/- tolerances}). The output level appeared to be around +10dBm at 720 MHz (actually 719.9961 MHz measured). The output was fed into the spectrum analyser and was clean - any products were certainly more than 30dB down.



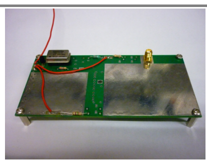
This is the bottom view of the completed W1GHZ LO PCB.

The right hand side shows the 3 5 - 30pF ceramic trimmers fitted.

Intrigued by the small rectangular piece of PCB at the LHS held down by sticky tape ???

I found that I could improve the output at 750 MHz by adding some "detuning effects" - and this is undoubtedly because this filter is optimised for 720 MHz.

I first noticed the effect when I had a finger on the end of the PCB and the output level rose.... I then looked for a way to do that more-or-less permanently.



This is the top view of the completed W1GHZ LO PCB.

The 50.000 MHz crystal oscillator is in use here although, with the DIL socket having been fitted, I can change that if/as other suitable crystal oscillators are located.

Knowing that the PCB was working, I fitted the 50.000 oscillator in the DIL socket and was greeted by the frequency counter showing close to 750.00 MHz and a single output frequency visible on the spectrum analyser, but with reduced level. I de-soldered the 4.7pF trimming capacity values (leaving just 2 x 15pF = 30pF) and the output level rose. I tried both 2.2pF and 3.3pF but it was difficult to say which was the better value - probably somewhere near or between both (operating frequency was now 250.0 rather than 240.0 as before, so less capacity is required to resonate there). I headed for my ceramic trimmer stocks, found some 5-30pF, removed the extras plus one 15pF cap from each filter section (so leaving 1 x 15pF fixed) and then added a 5-30pF across each section. That gave me a variable range from around 20pF to 45pF, more than enough to work around any future frequency shifts. I peaked each trimmer in turn, then repeated the process for maximum output on the analyser.

The RF output level looked higher on the analyser than it was at 240 MHz but actually measured lower on the RF power meter (& not sure why). Maybe the 720-760 MHz filter section is tuned well low (i.e. the 720 MHz area) in lieu of around 750 MHz (the test frequency at this point). The beat note when tuned across on the receiver was "clean and crisp" so at least that was an improvement. By the way, the maths work out such that with the LO at 2250.0 MHz, 2403.0 MHz = 163.0 IF - but that wasn't going to be a major problem during the testing phases. It would at least let me do "better" measurements on the receive and transmit sections.

As a matter of record, the homebrew RF power meter indicated a level of close to +11dBm at 750 MHz (the level at 720 MHz was not actually measured) from the W1GHZ LO PCB.

The frequency stability of the oscillator is simply terrible. From power on, it drifts enormously (when compared to SSB stability requirements) with a downward shift of 4-5KHz at 750 MHz happening in the first few minutes. It does slow it's drift rate after a while but is still affected by hand "presences" etc..

The next step was to interconnect the LO PCB and the transverter PCB and do some more tests..

Step A/B(1) : Testing the W1GHZ Transverter + LO PCBs

The tests were easy simply because I knew that both boards were working before I connected up the W1GHZ LO feed. Sure, the two pipecaps in the multiplier chain needed slight returning but that was all. The result was that with 153.000 MHz at 0dBm from the sig gen, the transverter board produced +12dBm at 2403 MHz. The most objectionable thing was the lack of stability of the 50.000 MHz crystal oscillator - it was drifting up, down, every which way. The heterodyne on 2403 would change if a hand was placed near the LO board, though this is something that wouldn't happen if it was all screwed in a metal box.

In receive mode, the sig gen at 801.000 (x 3 = 2403) was readily heard at 153 MHz on the R7100 receiver set up on the IF output port. Very clean in respect of on-frequency and near-frequency noises too. I wound the gen's attenuator down to around -80dBm (at 801) and could still hear the sig gen... Not quantative but still a good sign.

A suggestion was received today by email from Iain VK5ZD : Replace the ERA2 and ERA5 in the transmit stages with a GALI-39+ and GALI-84+ respectively for higher output. Iain recommended using a copper shim under the GALI-39+ and across the middle from ground plane "land to land" to maintain stability. I don't have these devices on hand but intend to purchase some...

16 June postscript : I built up the board as normal (ERA2/ERA5) and measured the output at about +13dBm with 0dBm IF input. I then powered it off, removed the ERA2, soldered in a tinplate bridging strip and then put the GALI-39+ in, powered on again and had just under +18dBm. Then I replaced the ERA5 with the GALI-84+ and the power level remained substantially unchanged. My suggestion is to change the ERA2 to a GALI-39+ and leave the ERA5 stage alone. Looking at the the specs alone indicates the GALI-39 has about 4dB more gain than the ERA2, the ERA5 & GALI-84 are about the same so theory became practice - just for a change.

Time for a slight sideways diversion at this point : two things came into play..

(1) The 83.000 crystal oscillators arrived. I ordered 4 of these and plugged them into the W1GHZ LO PCB - the output level was slightly higher but the 'cold' frequencies varied from -10KHz to +4.3KHz over the 4 items. The drift was a bit weird too - some drifted down in frequency as they warmed up, one drifted up. I picked the one that seemed stable-est and left it in the board. That frequency gives a 162.0 MHz IF for 2403.0 MHz, good for testing but also workable if your I.F. transceiver has been broadbanded to cover up to around 170 MHz.

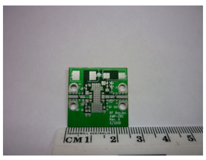
(2) I went ahead with building up a new synthesiser using a Silicon Laboratories' Si4133 chip : web page here (/~vk4adc/web/index.php/component/content/?task=view&id=125&Itemid=43).

This new synthesiser has to be properly tested with the W1GHZ transverter board but since it has plenty of output level, I don't really expect any issues.

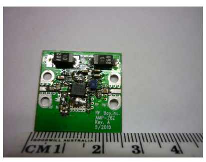
Yet to come......

Step C : a 1-2 watt PA using an RFMD SPB-2026Z MMIC

The small PCB and the device were ordered from eBay's RF Basic Store (http://stores.ebay.com.au/RF-Basic-Store) in the USA and took about 2-3 weeks to arrive by mail. The PCB comes without any documentation as such and the builder has to use the information from the device data sheet to ascertain values for the desired frequency of operation. The parts location is really a matter of transcribing the schematic into physical parts, with placement much the same as the schematic.



This PCB is small - just 25mm square - and about 0.5mm thick.
The layout is based on the use of 0603 size SMD passive parts.



This is the built-up view.

The board will be powered up shortly, once I have enough RF drive to make it function correctly.

Well it has been powered up and, to date, I have only managed to achieve an output level of +21dBm. This is a shortfall of about 10-12dB against the anticipated output so investigations will continue.

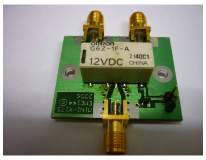
Step D : an antenna changeover PCB using an Omron G6Z relay

Well I was going to do my own PCB for this part of the project but since I had to order some G6Z relays from Minikits, I also ordered a couple of Mark's relay PCBs.

Assembly is quite straightforward even though the relay is a SMD-style product. Place it in position - and only one orientation gets the "legs" to line up, solder one connection pin, check that all other pins match up with the pads then solder the remainder. The "earthy" pins takes quite a bit of heat to produce a good joint. There is a place on the PCB for an SMD-format back-EMF diode and I used a BAT54 shottky diode in this position, it was the only one that I had here that had the correct lead orientation connections.

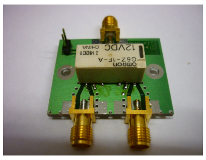
The SMA connectors are added next, the centre pin solders to one side while the "ground" pins are soldered on the other. Plenty of heat is required here too.

The final item is a 2-pin 0.1" header for easy connection of the relay power. Note that one side of the relay is permanently grounded so the incoming +12V must be switched on/off.



Diode and power connection header at RHS bottom corner, Common SMA connector at bottom centre.

The SMA connections are Common on the side with only one coax socket (top edge in the image below), and then on the side with 2 sockets, Normally Closed (N/.C) on the same side of the PCB(RHS below) as the Minikits logo and Normally Open (N/O) on the power connection header side (LHS below).



The LHS connector is N/O, the RHS is N/C.

Step E : a sequencer PCB using a PICAXE

I am currently translating my new sequencer design into artwork, this time with a PICAXE 18X SMD and designed around using all SMD parts. The new arrangement uses diode RF switching for the IF section, with different switches being activated to provide IF connections for two transverters/bands and two more again for the transmit/receive switching function. The transmit drive for each transverter board is adjustable separately.

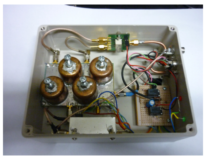
There are two input control functions : (1) PTT to change receive to transmit; and (20 the band select function. As such, only the two control signals plus supply voltage and the earth return are required to be sent to the remotely mounted transverter.

The board layout is almost complete and details will be added here when it is a physical device.

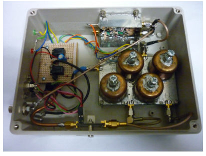
Step F : Making all of the above work together in a completed unit....

These are the inside views of the "temporary" 2.4 GHz transverter box assembled for the VK Winter Field Day 2011 (http://www.vk4adc.com/web/index.php/field-day-activities/31-2011-field-days/75-2011-winter-fd.html)... The external view is viewable on that web page.

The power amplifier section was left out of the assembly because the anticipated output level results weren't achieved in the lead-up to the assembly of the boards into the mounting box. I decided it was better to at least try it out and see if the basic parts worked.



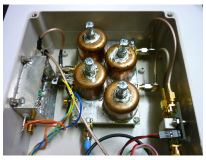
The antenna changeover relay is mounted on short standoffs on one vertical side



The synthesiser assembly is on the opposite vertical side, towards the far RHS.

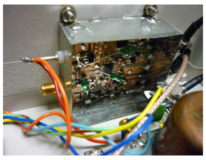
A quick "rough & ready" veroboard-style sequencer was quickly built up and uses two small 12V Omron G6H-2 DPDT changeover relays to do do the IF and DC power switching.

The coax running mid-air from bottom RHS to top LHS is the 10.000 reference lock signal input.



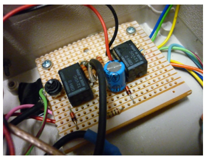
The main transverter board was mounted horizontally so that the tuning screws could easily be accessed.

The short loop at top left is the 752.666 LO signal across to the LO port on the transverter PCB.



A 'peek-in' view of the synthesiser in it's tinplate box. The top cover was left off somehow in the mounting process.

The RHS lower SMA on the synth is already outputting 1627.5 MHz at +10dBm to feed into the 3.4 Ghz transverter board when it is built up.



The sequencer veroboard up a bit closer. The electrolytic capacitor and a series resistor provides a "slow-down" function on both operate and release.

The black trimpot at left is the transmit drive level at the IF. The incoming transmit signal is terminated in 4 x 220 ohm resistors in parallel (about 55 ohms).