

G4DDK 23cm and 13cm VLNA2 (Very Low Noise Amplifier)

Sam Jewell, G4DDK



Introduction

This paper describes further development of the 23cm and 13cm VLNA designs first published on my web page in 2007 and subsequently published in CQ VHF Summer 2008 and the Proceedings of the 13th International EME Conference, Florence 2008. The improvements are largely due to the efforts of Sergie, RW3BP, [1] who has managed to achieve noise figures below 0.2dB together with good input return loss and high stability. Inspired by Sergie's work I have made a number of changes to the original 23cm VLNA kit. Whilst the kit version of the modified 23cm VLNA does not claim to reach the full performance claimed by RW3BP, it shows considerable performance improvement over the original design and with a little extra effort from the kit-builder may be able to approach the performance seen by Sergie.

A noise figure of around 0.2dB is now readily achieved by following the construction details in this article without the critical alignment that was required for the original VLNA. The pre-amplifier is now stable even with the input open circuit. Input return loss is 4dB at 1296MHz; and whilst not outstanding, this is much better than has previously been achieved and leads to much less uncertainty in the measured noise figure. Gain remains around 35 to 37dB, depending on final alignment and is a trade-off against noise figure.

Recent work has been directed towards improving the noise figure of the 13cm VLNA. By incorporating several of Sergie's modifications I have been able to achieve a worthwhile improvement in the performance of the 13cm VLNA. A noise figure 0.25dB

(or better) at around 27dB gain has been obtained, with careful alignment. The 13cm VLNA modifications and adjustment are described in Appendix 2.

Circuit description

A low noise Mitsubishi MGF4919G HEMT replaces the original NEC NE32584C HEMT in the critical low noise front end. An Avago ATF54143 is used in the second stage because it is capable of simultaneously providing low noise and a very high dynamic range.

The circuit diagram of the modified VLNA is shown in Appendix 1.

The self supporting input components are a feature adopted from the low noise pre-amplifier design by WD5AGO [2] and others. The input noise match uses a C/L/L circuit where the input capacitor both matches and provides a DC block for the bias on the gate of TR1. A series low-loss inductor (L1) provides another part of the match such that TR1 'sees' the optimum noise match when 'looking' out towards the 50 ohm source. The third part of the noise match is provided by a shunt inductor (L2) from the input capacitor (C1) to the bias decoupling point. All three noise match components are air supported rather than mounting onto pads on the PCB. This reduces losses in the matching circuit and allows easy matching adjustment by 'bending' the series input inductor. The use of a thick silver plated wire for L1 and L2 does not reduced losses enough to warrant its use in this stage.

Input return loss is improved by the use of series negative feedback. In this case by using long HEMT source leads. This technique has been used for many years, but its use has been tempered by the possibility of introducing instability at higher frequencies. Counter intuitively RW3BP has used longer than normal source leads in his improvement to the design. These are in the form of thin copper wires (L3 and L4) between the source leads and the source grounding pads on the PCB. He has also eliminated the lossy (and hence noise inducing) drain resistor (R3 in the original design) that is normally used to ensure stability. He also used a further short length of copper wire to connect TR1 drain to the top of matching inductor L5. When copying Sergie's modifications I noticed a tendency for the amplifier first stage to oscillate at about 8-9GHz. This is different to the oscillation at 16GHz that had previously been observed. Curing the oscillation is simple and consists of placing a small piece of absorber material close to C7 and L5, as shown in Fig 16, below. This is the same material as used for the end-wall absorber (to eliminate any 16GHz oscillation) and inside the lid (to make the metal lid 'invisible' to the high gain pre-amplifier stages!). Don't attempt to operate the pre-amplifier without these very necessary pieces of absorber material.

Source feedback is also used in the second stage, but is in this case, printed on the PCB (Shown as L7 but is actually two printed inductors) and cannot be adjusted. It was designed for optimum performance when the PCB was laid out.

A 5 volt 500mA surface mount voltage regulator (IC1) is used to provide power for both pre-amplifier stages. In the case of the first stage the 5V is connected to R4 (220R) before

continuing on to the drain of the MGF4919G via the various decoupling and matching components. When the bias for TR1 is correctly adjusted the drain current will be 16mA and this will cause the voltage across R4 to be 3.5V, giving a TR1 drain voltage of 1.5V. Slight variations on this voltage are to be expected when finally adjusting for optimum performance. The MGF4919G is a depletion mode device, requiring a small negative voltage (Gate to Source) to control the drain current. The negative voltage is generated by a surface mount ICL7660 DC-DC inverter chip (IC2). It produces -5V output for +5V input. This voltage is 'potted down' by the resistor chain consisting of R12, R13 and R14. R14 allows a small range of adjustment of the bias voltage such that TR1 cannot draw too much current. If the -ve bias voltage should fail TR1 would draw maximum current, limited to 22mA by R4. If the bias voltage is set too high, such that TR1 drain current is pinched off, the drain voltage could rise to 5V. Whilst I have never seen any degradation due to this effect the maximum drain voltage can be limited to 3V, as a precaution, by connecting a 3.0V zener diode across C5, with the cathode (bar end) to the junction of C5 and R4. It should not be necessary to connect a 10uF noise decoupling capacitor across the zener diode as the diode would only be effective under fault conditions and the pre-amplifier would not normally be expected to provide low noise operation under these conditions.

Active transistor bias is used to hold the operation of the ATF54143 enhancement mode FET (TR2) stable over a wide range of temperature. A BC807 PNP transistor (TR3) is the active bias device. TR2 operates at 64mA and can get rather warm. R9 sets the drain current for TR2. Without the active bias the noise and gain performance of the FET can change noticeably between switch on and operation after several hours in front of a dish antenna. The second stage operates with approximately +0.54v on the gate and +2.95V on the drain at 64mA drain current.

SMA female connectors are recommended for use with the VLNA. SMA male connectors can be used if required. N connectors are not recommended for two reasons.

- 1) Difficulty of ensuring a good electrical contact around the input, where losses must be minimized and impedance maintained right up to the input noise match components.
- 2) Most EME systems now use a septum polarizer. The inherent isolation between the transmit and receive ports of the feed is high enough that only a small SMA connectorised relay is required to protect the pre-amplifier when transmitting. Relays with N connectors are usually larger than small SMA types and hence the pre-amplifier is likely to be further from the input connector with attendant increased loss and hence increased system noise figure. Terrestrial systems are not so noise-critical so it doesn't matter so much what relay and connectors are used.

Construction

Whether you choose to buy the kit or just the PCB it is highly recommended that you build the pre-amplifier into the recommended size of box and place the input connectors

where shown. The same also applies if you decide to make your own PCBs from one of the published masks [3] [4] [5].

Not only is the box size important for stability, but also the recommended absorber material **MUST** be used.

One of the major changes from the original VLNA designs is the use of the Mitsubishi MGF4919G in place of the NE32584C.

The PCB is shown in fig1 and the current designation is G4DDK VLNA-IssB

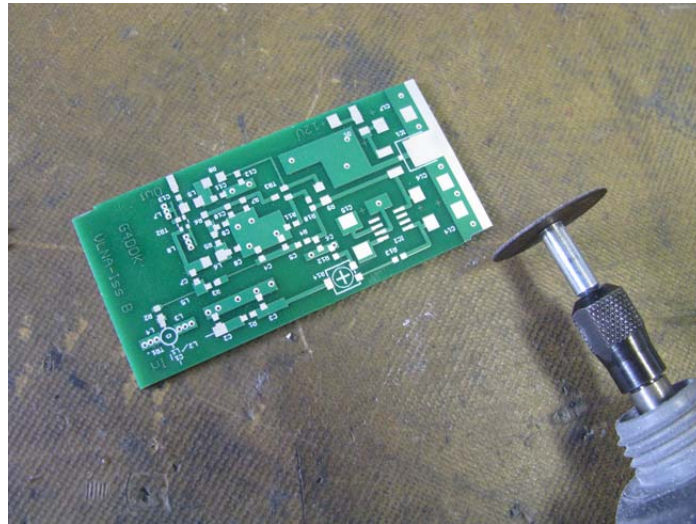


Figure 1

The Dremel abrasive/slitting wheel is used to grind two small notches in the corners of the PCB. These are required to allow the tin plate box to fit together properly. Ensure the notches are in the correct corners, as shown. The notches should be 1mm deep and 5mm long.

After grinding the PCB notches assemble the box (without soldering it together) and ensure that the PCB fits properly. As long as the notches are the correct size, it will fit perfectly into the box.

Clean the PCB with Isopropyl Alcohol (IPA) or Universal solvent (Rapid Electronics 34-0778) [6].

Optionally you can spray the component side of the PCB with spray solder flux to aid soldering. The small container (Rapid Electronics 87-0720) is highly recommended. It is cheap and long lasting.

Populate the PCB before soldering it into the tinplate box. Believe me, this is the **BEST** way!

Place the PCB on a white kitchen towel, under a good light, and with a small, fine-tipped soldering iron and miniature 0.3mm diameter fluxed solder, suitable for SMD assembly, populate the board with all parts except TR1, L1, L2 and C1. You will need approximately 25cm of the fine solder to complete the PCB.

DO NOT attempt to use normal 0.7mm diameter solder. It WILL make a mess of the PCB!

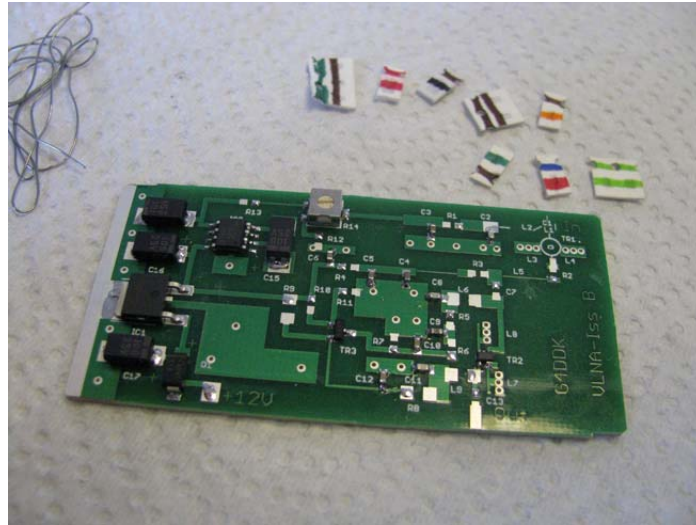


Figure 2

Start with the Tantalum capacitors, C14, 15, 16, and 17. Observe the correct polarity for all four and especially the orientation for C14.

Follow with IC1 regulator, first soldering the two (or 3) legs and then solder the tab down flat and firmly to the PCB ground plane.

The resistors, capacitors and inductors are next, followed by TR3, IC2, and TR2. Note that R3 is now replaced with 3n3H SMD inductor L10.

I find it best to apply a small 'spot' of solder to one SMD pad. Place the component, using a small pair of tweezers, (preferably stainless steel). Quickly solder the end with the solder spot, ensuring the component is absolutely flat to the PCB pads at both ends. Solder the other end and then go back and re-solder the first end.

Figure 3 shows the component overlay for the 23cm VLNA.

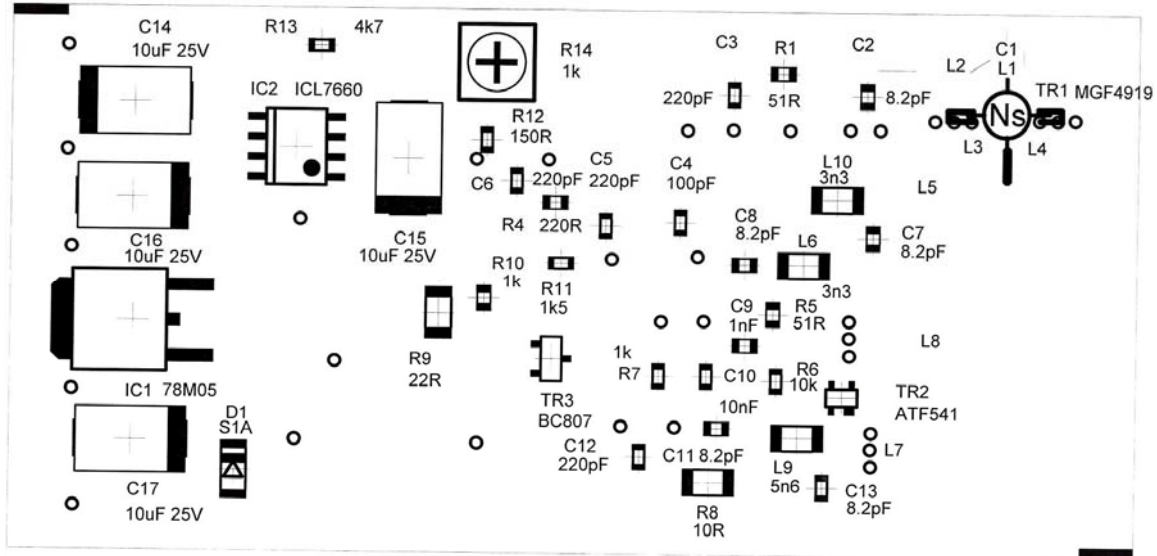


Figure 3

When all the parts have been soldered to the PCB cut two 5mm lengths and one 6mm length of the supplied 0.315mm diameter enamelled copper wire as shown in figure 4



Figure 4

Tin the ends of the three wires with a hot soldering iron tip (400C).

Insert the two 5mm wires in the two source pad vias nearest TR1 as shown in figure 5. The longer wire should be carefully soldered to the pad at the end of L5 and bent as shown in figure 4. The two shorter wires should extend 3mm above the PCB. Carefully bend them into hook shapes, also as shown.

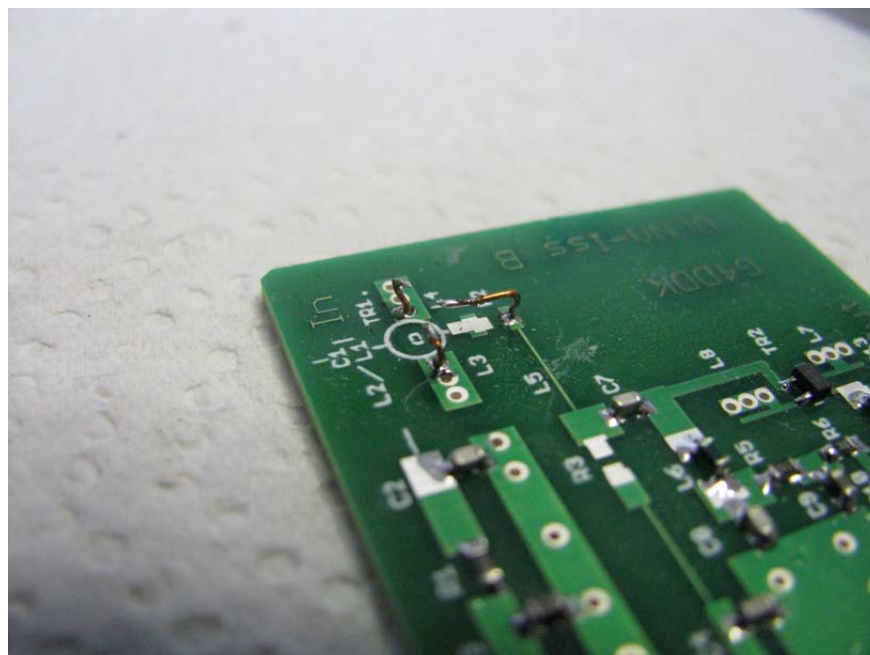


Fig 5

Now VERY CAREFULLY solder one of the wider TR1 source leads to one of the source wires with the wire ABOVE the source lead. Solder the second source wire to the other source lead and finally solder the drain lead to the end of the wire going to L5 pad.

Figure 6 shows this arrangement. However note that this photo shows the FET attached to some thinner leads that were used for the 13cm VLNA, described later. The general idea can be seen from this photo. These thinner wires were found to be too long, so don't copy them! The FET will be approximately 2mm above the PCB with the leads as specified.

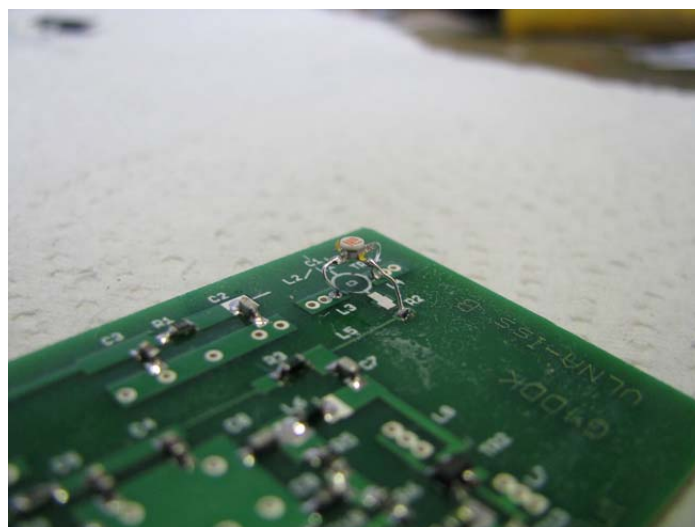


Figure 6

It is recommended that a temporary thin wire is soldered between the pad at the top C2 and the gate of TR1 to protect the FET from accidental static damage when fitting the PCB into the box.

Inspect the populated PCB and ensure you have soldered every component to its pad and that the Tantalum capacitors are the correct way round.

Put the PCB on one side.

Please note that the edges of the tin plate box can inflict serious cuts to the fingers if handled carelessly. Handle with care. The author accepts no responsibility for any cuts you inflict whilst handling the box.

Select the two sides of the box. With a Vernier calliper set to 10mm, scribe a line inside the box side on both the long sides and the short sides. The two sides are scribed identically as shown in figure 7, where the lines have been emphasised with a marker pen.

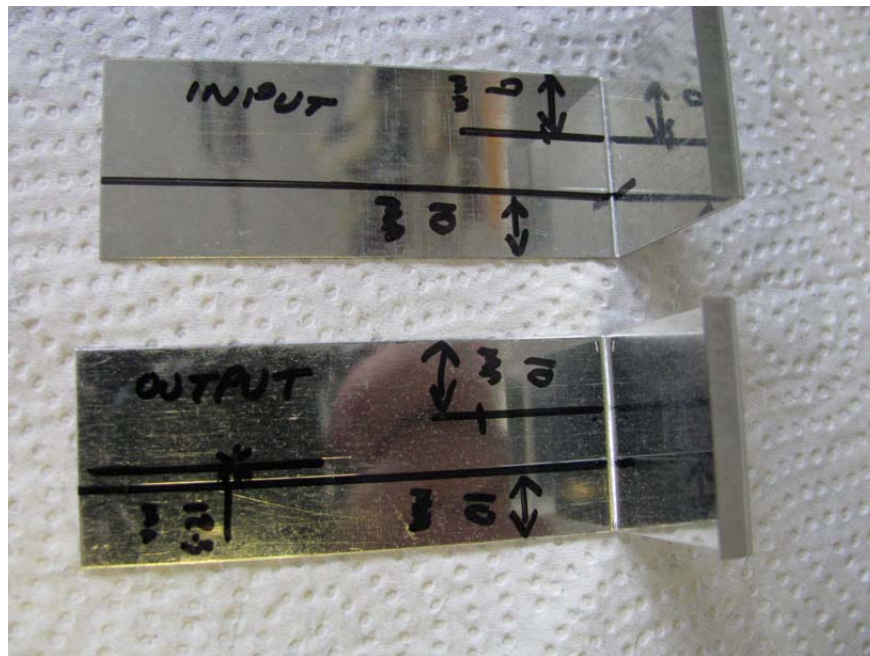


Fig 7

On the side selected to be the input side, scribe a second line 9mm down from the other edge and extending about 10mm from the fold between the long and short sides.

On the other side selected as the output side, scribe another line 10mm down from the other edge and extending about 20mm from the fold.

Re-set the calliper to 12.3mm and FROM THE SAME EDGE AS THE FIRST 10mm LINE mark up 12.3mm and extending about 25mm from the end of the side.

Hold the PCB up to the output side and mark on the shorter 10mm line a point in line with the +12v input pad.

Mark a similar point in line with the output track on the 12.3mm line.

Drill a 4mm diameter hole for the input connector and the output connector, using a sharp drill bit. Drill from the inside of the box. Clean the hole on both sides with a small file or better still, a sharp countersink bit.

Drill a 2mm diameter hole for the feedthrough capacitor, C18. Again countersink the hole to ensure a clean hole.

Preparing the SMA connectors

Prepare the two SMA connectors as shown in figure 8.



Fig 8

These are the recommended 2 hole female SMA connectors, made from brass and gold plated. The diameter of the TEFLON™ should be 4mm. Use a sharp blade to cut the insulation 0.5mm from the back of the connector. Slide the insulation off the connector spill. Cut the spill to an overall length of 1.5mm using the miniature Dremel slitting tool. Carefully face off the spill with the same tool.

Now to fit the PCB into the box, as described next.

Offer the PCB up to the output side of the box as shown in figure 9, carefully aligning the ground-plane side to the scribed line. Push the PCB well into the short edge. Tack-solder the PCB on both the long and short edges.

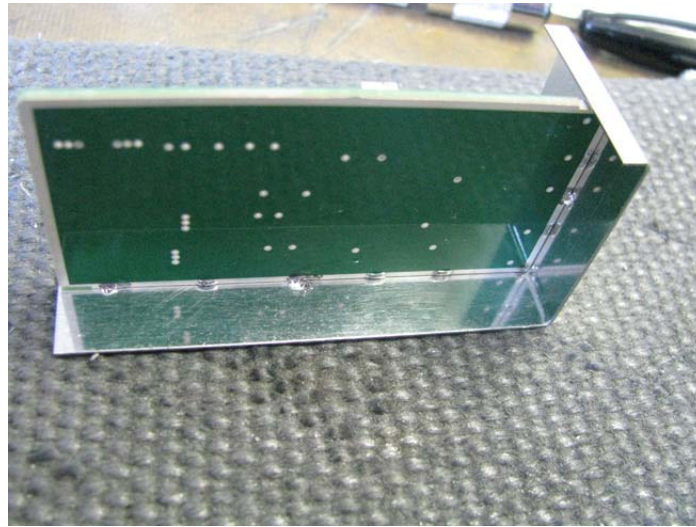


Fig 9

Once you are sure it is square to the box, seam-solder the PCB to the box. This will require a 50w soldering iron and 0.7mm diameter fluxed solder.

Now place the side of the box, with the PCB, into the bottom of the box. Fit the second edge and ensure a good fit. Tack-solder the overlap at each of the two corners as shown in figure 10.

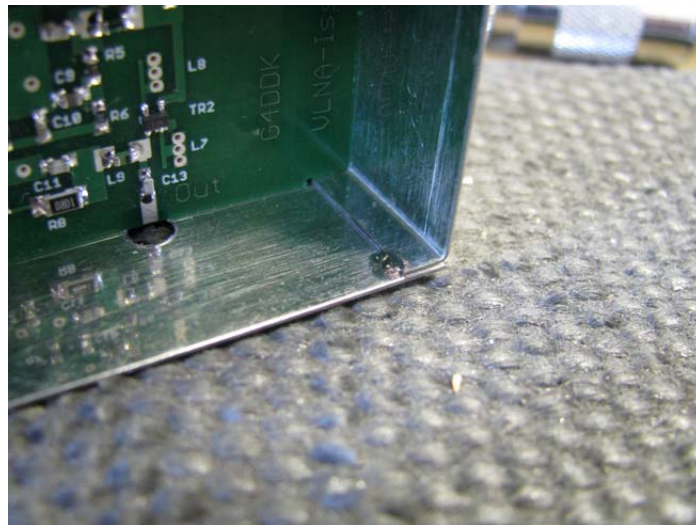


Fig 10

Now fit the lid to check for a good fit. If it is satisfactory remove the bottom cover and tack-solder the remaining two corners to form a rigid box section.

Carefully align the unsoldered edges of the PCB on the ground-plane side with the 10mm scribed line on the input side of the box and solder. When you are happy it is all square, seam solder all round the ground plane to attach it to the box. Also seam solder the component side copper along the short edge next to the regulator IC1. The seam soldering is shown in figure 11.

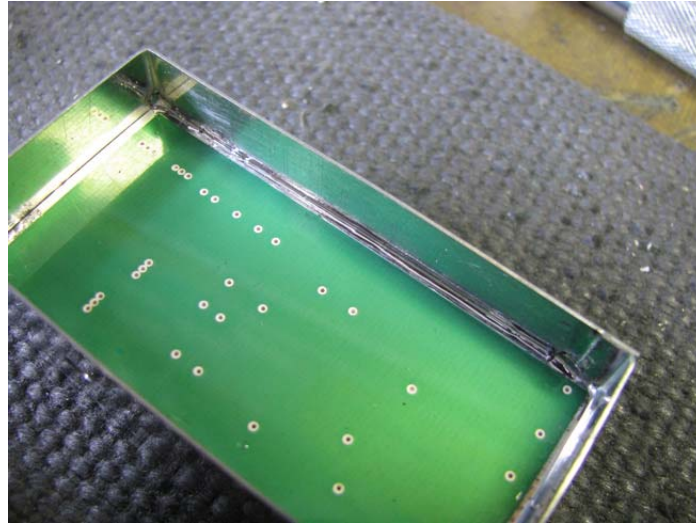


Fig 11

Once the PCB is soldered into the box, the connectors can be fitted by seam soldering their flanges to the tin plate box as described next.

The connectors and feedthrough capacitor

Solder the input connector first. When soldering, apply pressure to the end of the SMA so that it is pressed down firmly onto the tin plate box. Use a wood or aluminium handle so as not to damage the connector. There should be no gap showing. Apply the solder into the mounting holes until it is seen to seep out under the flange of the connector. The soldered connector is shown in figure 12. This is critical for lowest noise performance.

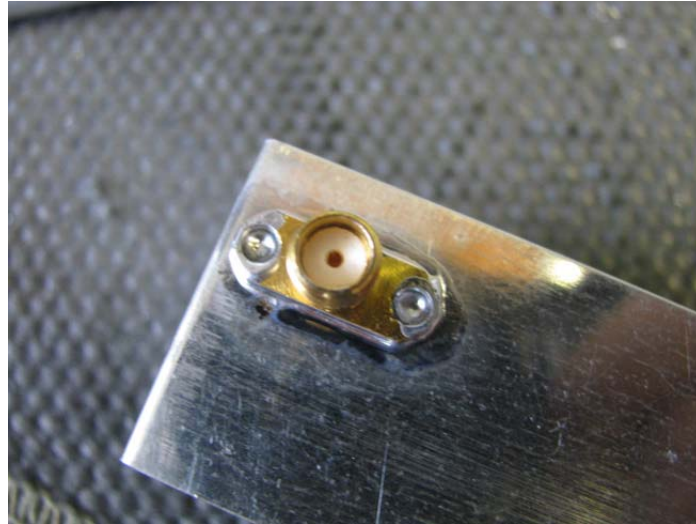


Fig 12

Turn the box over, supporting the input connector against damage. Solder the output connector in the same way as for the input connector. When you are happy with the two connectors, solder the 1500pF Tusonix feedthrough capacitor into the 2mm diameter hole. Make sure you thread it through the 2mm ground solder tag first, as shown in figure 13.



Fig 13

Use a short length of insulated wire to connect the +12v supply pad to the end of the feedthrough capacitor inside the box.

After assembly and fitting the connectors and feedthrough capacitor, clean off any flux residue with IPA or Universal solvent.

The input matching components

Wind the two input coils, L1 and L2 as shown. L1 should be $3 \frac{3}{4}$ turns of enamelled copper wire of 0.315mm diameter. This coil is close-wound on a 2.5mm diameter former (use a 2.5mm diameter drill shank). The top turn is then opened out as shown in figure 14.

L2 is wound with $2 \frac{1}{2}$ turns of the same 0.315mm wire, close-wound and on a 2.5mm diameter former. Note that the connecting tails should be at 180 degrees to each other, as shown.

The ends of both coils must be tinned.



Figure 14

Fitting the input noise matching components

Carefully tin the end of the input SMA connector spill. Holding the input capacitor C1 with fine tweezers, solder it in place as an extension of the input spill. It should not be at an angle as shown in figure 15!

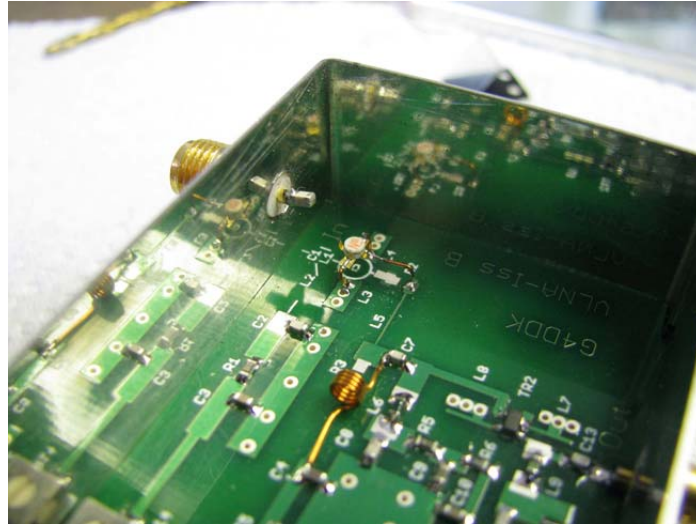


Fig 15

Also note that the hand wound choke shown in place of L10 in figure 15 is now replaced by an SMD 3n3H inductor for repeatability.

Remove the shorting wire between TR1 gate and C2.

Solder L2 between the top of C2 and the end of C1. C1 is now supported so it will not now move. If necessary, go back and re-solder C1 accurately in line with the input connector spill.

Solder the opened-out end of L1 to the junction of L2 and C1. Solder the other end to the gate of TR1. It should go without saying that the soldering should be completed quickly and cleanly. The input to a low noise pre-amplifier is no place for sloppy workmanship. The completed input is shown in figure 16



Fig 16

The coils should be orientated as shown. The two pieces of absorber material (T2 and T3) should be placed as shown. The smaller piece (T3) should be placed (straight) to cover L5 and parallel to L8. The long strip (T2) should be stuck to the end wall of the box, close to L1 and pressed down onto the PCB. The larger tile (T1) should be stuck inside the lid of the box and towards one end so that when the lid is in place the absorber is over the RF stages.

This completes the VLNA build.

Testing and adjusting the 23cm VLNA

Voltage checks

Connect +12 to +20V to the feedthrough capacitor C18 and ground to the solder tag.

Connect good quality 50R SMA terminations to the input and output connectors.
Adjust R14 fully anti-clockwise and then turn clockwise about 30 degrees (not critical).

Switch on the supply and check that the total current taken is approximately 80mA.

With a digital voltmeter, check that the output from IC1 is +5.0V +/-0.1V

Check that the output from IC2 is $-5.0V \pm 0.2V$. Test at IC2, pin 5.

Check the DC conditions around the circuit.

A suitable probe for checking voltages in RF 'hot' areas is shown in figure 17

Attach a 10K resistor to the end of a meter red (+) lead. Wind the lead of the resistor around the probe lead. Cut the other resistor lead to about 3mm long.



Meter Probe

Fig 17

Measure the voltage on the drain of TR1. It should be $1.5V \pm 0.1V$. Use R14 to adjust it to 1.5V initially. It will probably need to be changed later as you tune for best noise performance. If you don't have terminations on the input and output of the pre-amplifier it might self-oscillate. If that occurs the two drain voltages may read incorrect values due to rectified RF.

Measure the voltage on the drain of TR2. It should be $2.95V \pm 0.1V$. You cannot adjust this voltage. It will either be right or wrong. If it's wrong you have a fault to find!

If these are correct, you can proceed to adjust the pre-amplifier.

Adjusting the pre-amplifier for lowest noise performance.

If you have an HP8970A/B or Eaton/Ailtech 75 series meter with a 5dB ENR noise head then it is a simple matter of connecting the noise source to the pre-amplifier input and the output to the NF meter input, and adjusting L1 for lowest noise figure, consistent with a gain of about 35 – 37dB. If you have a 15dB ENR head then you **MUST** use it with a 10dB attenuator or the results may be considerably in error.

The attenuator's value **MUST** be known to 0.01dB at 1296MHz for accurate results. That is a tall order.

Depending on the nf meter, you may need to use an intermediate frequency converter to convert 1296MHz down to, say, 144 or 30MHz.

If you are able to measure noise figure and gain between 1000MHz and 1500MHz it is worth checking that you have adjusted L1 for lowest noise figure at 1296MHz (or wherever you need lowest). The gain maximum will be found to be around 1100MHz \pm 50MHz when the noise is at a minimum at 1296MHz. Peak gain is over 40dB. This is the

normal characteristic of the pre-amplifier. The noise minimum is located HF of the gain peak.

Once you have adjusted L1 for lowest noise figure, re-adjust R14 to give highest gain (probably only 0.5dB or so increase) consistent with lowest noise figure.

Adjusting for lowest noise figure without a NF meter

If you do not have, or are not able to gain access to, a suitable NF meter but have a spectrum analyser that covers up to 2GHz then you can use that noise peak as a means of adjusting L1 for minimum noise figure. Figure 18 is a screen shot of the output of the pre-amplifier from 250MHz to 2GHz. The broad noise is clearly seen, peaking at about 1075MHz at a level of about -55dBm in 3MHz bandwidth.

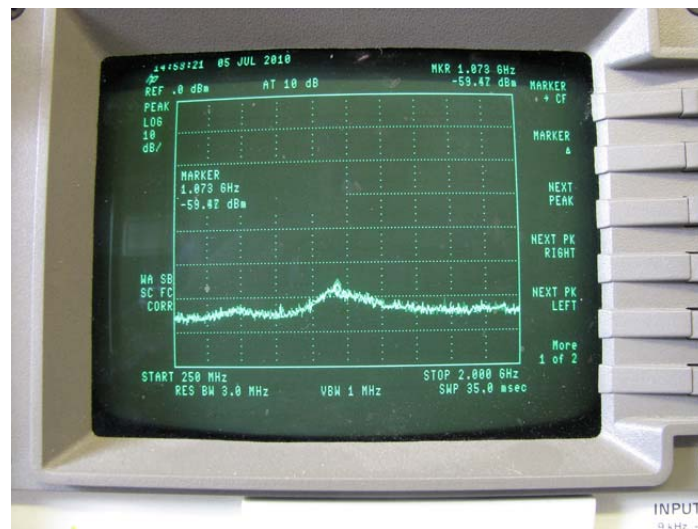


Fig 18

The pre-amplifier input is terminated in a 50R SMA and the output is connected to the spectrum analyser through a suitable low loss coaxial cable. Adjust L1 for a noise peak at approximately the frequency shown. This adjustment is critical for lowest noise figure.

If you do not have either a noise figure meter or a suitable spectrum analyser, then the pre-amplifier will have to be adjusted using a weak off-air signal, such as a beacon. This is best done using FM demodulation where the beacon signal is just strong enough to cause some quieting of the receiver noise output. In this condition the audio signal to noise ratio is strongly dependant on the carrier to noise (which you are trying to minimize) ratio over a range of just a few dB.

Once the pre-amplifier has been tuned for lowest noise figure the lid can be fitted. The absorber material (T1) will ensure that the pre-amplifier remains stable. The lid essentially becomes 'invisible' to the pre-amplifier stages below.

Operating the pre-amplifier

The pre-amplifier can be operated with a supply of between 10V and 24V. The input Tantalum capacitor is rated at 25V in all the latest kits (post serial 1100).

It is possible to power feed the pre-amplifier over the output coaxial cable by adding a 10 turn choke between the spill of the output capacitor, C13, and the end of the feedthrough capacitor C18, inside the box. Do be careful when doing this that this is the **ONLY** power supply in use.

I usually operate my masthead (terrestrial and EME) pre-amplifiers in parallel with the antenna changeover relay or protection (EME) relay. The relay can simply be power fed from the end of C18 outside the box. If a 28V relay is used, it will be necessary to change C17 from the 25V working Tantalum to a 35V version or you will end up letting smoke out of the pre-amplifier box! It is wise to connect a 1N4000 series diode across the relay coil to short any back EMF when the relay releases. I have operated my systems this way for many years without any pre-amplifier failures. When running the pre-amplifier from +28V in this way make sure the case is in contact with a large area of metal to act as a conduction cooling system for the regulator.

Pre-amplifier stability

With the modifications proposed by RW3BP the pre-amplifier should be completely stable even with the input open circuit. The stability is **NOT** guaranteed at all phase angles, however, so operating it open circuit is not recommended.

The pre-amplifier input return loss is approximately 4dB. Whilst this is not as good as RW3BP has measured, it is much better than the earlier VLNA design. Low input return loss will **NOT** improve the pre-amplifier performance, but it does mean that the uncertainty in NF measurements is much reduced, leading to more confidence that the measured noise figure is correct when carefully measured in a calibrated system.

The modified pre-amplifier is much easier to tune for low noise figure than the original design and it is often unnecessary to do more than adjust L1 as discussed to achieve better than 0.25dB NF.

That's it. You should now be ready to use your new very low noise 23cm VLNA2 pre-amplifier.

References

[1] RW3BP

http://www.vhfdx.ru/apparatura/rw3bp_1296mhz_lna_optimization

[2] WD5AGO
Low Noise 2-stage amplifier for 23cm
Proceedings of the Microwave Update 1999, Plano Texas.

[3] G4DDK
www.g4ddk.com

[4] Proceedings of the 13th International EME Conference Florence, August 2008

[5] 'low Noise Pre-amplifiers for 1.3, 2.3 and 3.4GHz Amateur Bands' CQ-VHF Summer 2008

[6] Rapid Electronics
www.rapidonline.com

Appendix 1

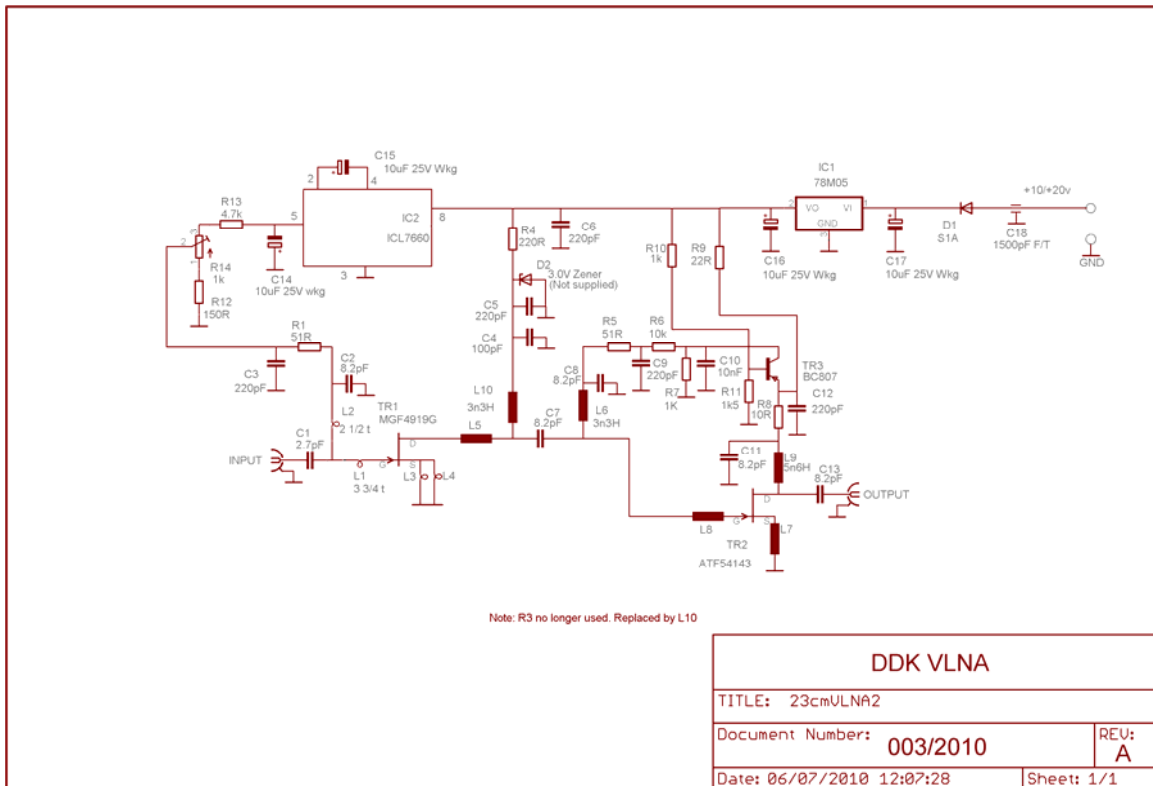


Fig 19 Circuit schematic of the 23cm VLNA2

Appendix 2

13cm VLNA2

Introduction

After much further building and testing the 13cm VLNA2 is ready for release.

The principal differences between the 23 and 13cm VLNAs are in the input match component values and in some of the interstage component values.

It is possible to noise match the 13cm VLNA anywhere between 2300 and 2400MHz. It should be possible to noise match outside this range but at this time testing has not been completed.

Fig 20 shows the component layout for the 13cm VLNA2

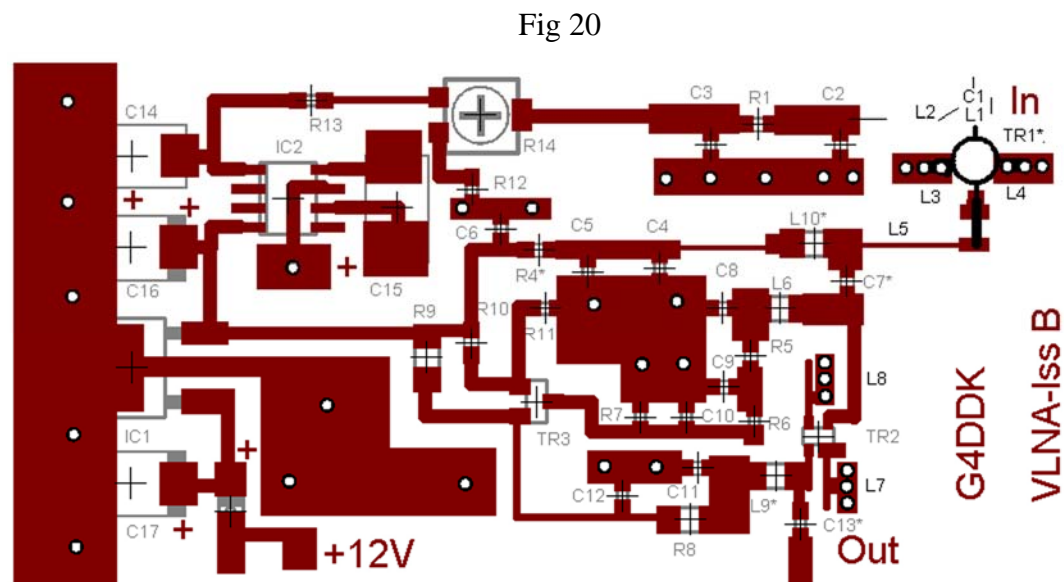


Fig 20 13cm VLNA2 PCB and component layout. 200% scaling

At first glance the inside of the 23 and 13cm VLNA2s look identical. Indeed, the same PCB, case, connectors and components are used in both designs. Look more carefully and you will notice that the input inductors, L1 and L2 are quite different.

It will not be possible to see that any other components are different as only two capacitors and one SMD inductor value has been changed elsewhere on the board, and these parts do not have their values marked on them. On the component overlay the changed components are marked with an *

You will also notice that there is only one piece of absorber on the PCB. The long end-wall piece is not required in the 13cm design. The larger absorber tile is retained in the box lid.

The circuit schematic is shown in figure 21

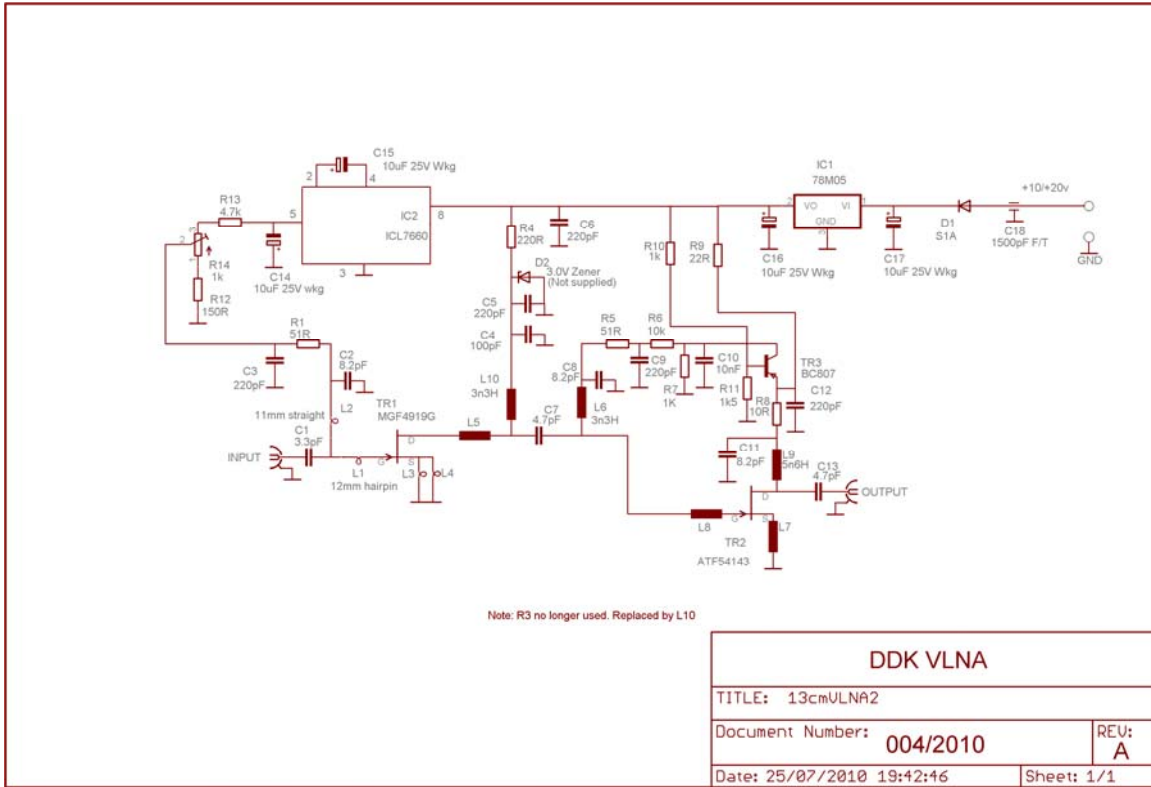


Fig 21 13cm VLNA2 Circuit schematic. Note:13cm component changes are shown with an ‘*’

Differences between the 23 and 13cm VLNA2

Component	23cm VLNA	Colour code	13cm VLNA	Colour code	Comments
C1	2.7pF	red	3.3pF	purple	
C7	8.2pF	green	4.7pF	black	
C13	8.2pF	green	4.7pF	black	
L1	3.75 turns		11mm straight length		
L2	2.5 turns		12mm length hairpin		
L9	5.6nH	Green/blue	5.6nH	Green/blue	Was originally 2.2nH in the 13cm VLNA
T2 Long tile	required		Not required		
T3 Small tile	required		required		

Table 1

Assembly

The assembly is identical to the 23cm VLNA2.

The source lead lengths for TR1 are the same for both VLNA versions.

Cut L1 to 12mm long and tin the ends of the wire.

Cut L2 to 11mm long and tin the ends of the wire.

Solder L2 between the end of C2 (there is a pad for this) and the end of C1. L2 must be straight.

Gently bend L1 into a hairpin so that it fits between C1 and the gate of TR1. IT MUST BE ORIENTATED as shown. VERY slight bending of L1 towards the input or output minimises the noise figure at 2320MHz. L1 will actually be orientated more vertically than is shown in figure 20, below. T2 end wall absorber must NOT be used on 13 or 9cm versions of the VLNA2.

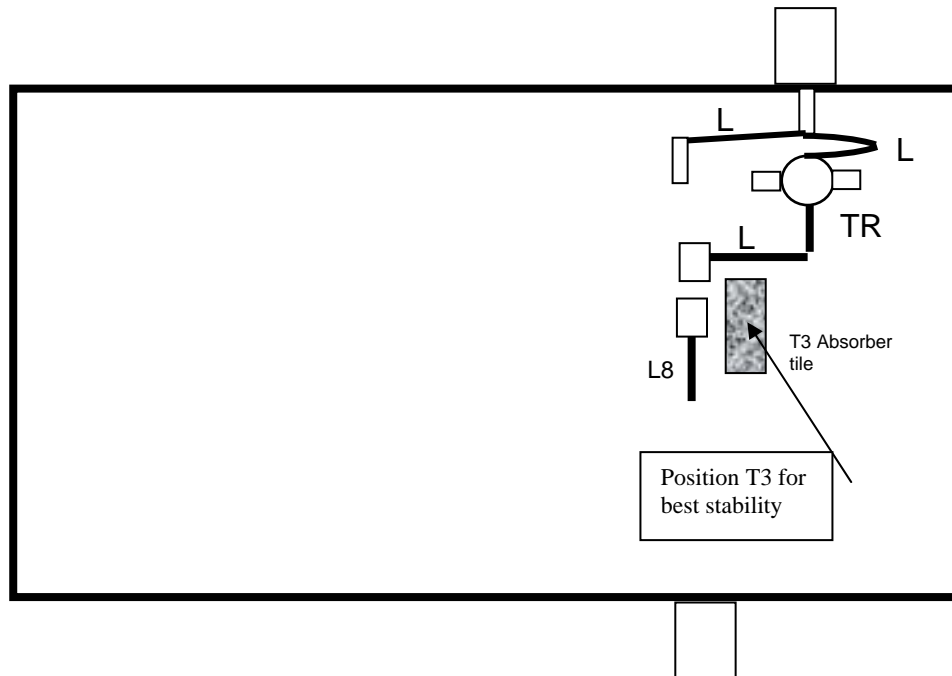


Fig 20 Layout of the 13cm VLNA2 input components and the position of absorber tile T3. T3 must NOT cover either L5 or L8 or noise figure will increase. Note that T2 is not used on 13 or 9cm.

Results

Prototype 13cm VLNA2 pre-amplifiers have a noise figure of 0.25dB, or better, with an associated gain of 26 - 27dB.

Document history:
Issue A released 07/07/2010
Issue B released 27/07/2010 – 13cm added
Issue B2 released 18/08/2010 – 13cm T3 info amended